ENGINEERING DYNAMICS COMPANY

HVE User's Manual

HVE Human Vehicle Environment

Markdown Edition — revised to describe the current version of the software

Contents

1. HVE User's Manual
2. HVE User's Manual — Section One: Overview
3. Using HVE — High-Level Overview
4. Chapter 1 — What Is HVE?
5. Chapter 2 — How To Use HVE
6. Chapter 2: How To Use HVE — Part B
7. Chapter 2: How To Use HVE — Part C
8. HVE User's Manual — Section Two: Menu Reference
9. Chapter 3: File Menu
10. Chapter 4: Set-up Menu (Part A — Position/Velocity, Driver Controls)
11. Chapter 4: Set-up Menu (Part B — Damage Profiles through Notes)
12. Chapter 5: View Menu
13. Chapter 6: Options Menu
14. Chapter 7: Help Menu
15. Section Three: Human Editor
16. Chapter 8 — Creating and Editing Humans
17. Chapter 9 — Human Model Definition
18. Section Four: Vehicle Editor
19. Chapter 10 — Creating & Editing Vehicles
20. Chapter 11 — Vehicle Model Definition (Part A: Sprung Mass)
21. Chapter 11 — Vehicle Model Definition (Part B: Exterior)
22. Chapter 11 — Vehicle Model Definition (Part C: Suspension)
23. Chapter 11 — Vehicle Model Definition (Part D: Brakes, Tires and Wheels)
24. Chapter 11 — Vehicle Model Definition (Part E: Drivetrain, Steering and Brake System)
25. Section Five — Environment Editor
26. Chapter 12 — Creating & Editing Environments
27. Chapter 13 — Environment Model Definition
28. Section Six — Event Editor
29. Chapter 15 — Creating & Editing Events
30. Chapter 16 — Event Model Definition
31. Section Seven: Playback Editor
32. Chapter 17 — Creating Report and Playback Windows

33. Chapter 18 — Editing the Event Sequence
34. Section Eight: 3-D Editor
35. Chapter 19 — Using the HVE 3-D Editor
36. Chapter 20 — 3-D Editor Object Tools
37. Chapter 21 — Manipulators
38. Chapter 23 — Basic 3-D Editor Tutorial
39. Section Nine: Video Output
40. Chapter 26 — Video Basics
41. Chapter 27 — Using The HVE Video Interface
42. Section Eleven: HVE Brake Designer
43. Chapter 29 — HVE Brake Designer
44. Chapter 30 — Brake Temperature Model
45. Chapter 31 — Antilock Braking Systems
46. Back Matter: Tutorial, Appendices and Index
47. Chapter 32 — HVE Tutorial
48. Appendix I — Installation & Setup
49. Appendix II — Messages
50. Appendix III — Coordinate Systems
51. Appendix IV — Units
52. Appendix V — Collision Deformation Classification (CDC)
53. Appendix VI — References
54. Appendices VII, VIII and IX — Reserved
55. Appendix X — Index (Topic Locator)

HVE User's Manual

Updated Markdown edition of the HVE User's Manual (Version 5, Seventh Edition), verified against the current application source. Part of the HVE Manual.

Sections

- Overview — Using HVE, What is HVE?, How to use HVE
- Menu Reference — Files, Set-up, View, Options, Help menus
- Human Editor — Human Editor
- Vehicle Editor — Vehicle Editor
- Environment Editor — Environment Editor
- Event Editor — Event Editor
- Playback Editor — Playback Editor
- 3-D Editor — 3-D Editor
- Video Output — Video Output
- HVE Brake Designer — HVE Brake Designer
- Appendices — Appendices

HVE User's Manual — Section One: Overview

Updated Markdown edition of the legacy HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), "USING HVE" overview and Section One, revised for the current HVE release.

Contents

Original pages	Contents
O-1 .. O-5	Using HVE: assumptions, how to use this manual, step-by-step procedure, conventions, problem determination, window manager basics
1-1 .. 1-6	Chapter 1, What Is HVE?: purpose, simulation environment concept, HVE-compatible calculation models, editors, databases
2-1 .. 2-39	Chapter 2, How To Use HVE (Part A): getting started, menu bar, modes, creating humans, vehicles and environments, the 3-D Editor, creating, setting up and executing events, the Event Controller, choosing options
2-40 .. 2-69	Chapter 2 (Part B): creating Report and Playback windows, output report types, Variable Output Table, Playback Controller, combining and editing multiple events, Audit Trail, creating case files, human/vehicle/tire databases, printing results, setting the view, overlays
2-70 .. 2-104	Chapter 2 (Part C): selecting user options (Key Results, coordinate axes, contacts, velocity vectors, skidmarks, targets, grid, units, render options, simulation controls, playback options, calculation options, DyMESH, Get Surface Information, preferences), getting help, video interface

Related references

Dialog-level field descriptions are maintained in the dialog reference:

- User Interface (menus, viewers, playback controls)
- Vehicles • Suspension/Steering • Brakes/Powertrain • Tires/Wheels • Safety Systems
- Humans • Environment
- Events and Driver Controls • Calculation Options • Reports and Output

Using HVE — High-Level Overview

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), "USING HVE — High-level Overview" (pages Overview-1 through Overview-10). Revised for the current HVE release.

Assumptions

HVE was developed for use by engineers and scientists who need a powerful simulation tool to study vehicle and occupant dynamics. HVE was also developed for motor vehicle safety researchers and crash reconstructionists who need to analyze motor vehicle dynamics, including collisions, and show or explain to others how accidents occur by analyzing and visualizing the sequence of events surrounding the accident.

An individual using HVE should possess the following background and experience:

- basic knowledge about computer systems (system maintenance, using a file system, installing peripherals, and using a window manager)
- technical knowledge in the area of engineering mechanics (the laws of motion) and mathematics (coordinate systems, algebra and trigonometry)
- knowledge about vehicles and how they work (general terminology)
- knowledge about human anatomy (general terminology)
- accident reconstruction experience (gathering and interpreting accident site evidence and drawing conclusions about accident causation)

Prior to using HVE, it is assumed the researcher has obtained all the required human, vehicle and environment data, either by direct inspection or by indirect means (built-in databases, literature searches, published data or, in some cases, witness information).

How To Use This Manual

This is a User's Manual. It explains how to use HVE. It does not directly teach the subject of human and vehicle dynamics, or explain how to reconstruct motor vehicle crashes. Further, this manual does not attempt to explain how specific reconstruction and simulation models work. For these details, consult the Technical Reference or Physics Manual for the particular reconstruction or simulation model (see the program manuals).

Every new user should read the following chapters:

Topic	Reference
High-level Overview	(this section)
What Is HVE	Chapter 1
How To Use HVE	Chapter 2
Human Editor	Humans
Vehicle Editor	Vehicles
Environment Editor	Environment
Event Editor	Events & Driver Controls
Playback Editor	Reports & Output
Video Basics	Video Output
Tutorial	Tutorial

This information will allow you to begin using HVE right away. However, you will also need specific information from time to time. In that case, you'll want to use the other chapters in this manual as a reference.

If you hate reading manuals and you want to get right to work, the following section may be of some help. However, you should understand that HVE is a very powerful and complex program, and to get the most out of it, you will need to refer to this manual.

Step-by-step Procedure

The following procedure outlines the basic steps for using HVE, along with an initial chapter reference:

Step	Reference
Select Active Humans	Ch. 8
Edit Humans	Ch. 9
Select Active Vehicles	Ch. 10
Edit Vehicles	Ch. 11
Create Environment	Ch. 12
Edit Environment	Ch. 13
Create Events (Select Humans, Vehicles, Calculation Model)	Ch. 15
Set up Events (Assign Position/Velocity, Driver Controls, Damage Profiles, Payload, Collision Pulse, Vehicle Mesh, Wheels, Accelerometers, Contacts, Restraint Systems)	Ch. 16
Execute Events	Ch. 15
Create Report Windows	Ch. 17
Edit Report Windows (Event Sequence)	Ch. 18
Create Playback Window	Ch. 18
Create Numeric and Graphic Reports	Ch. 17
Create a Videotape	Ch. 27 *(updated: current HVE builds create digital video files via File, Video Creator rather than videotape)*
Save your work	Ch. 32

Conventions

The following typographical conventions are used in this manual:

- *Italicized Words* — Used for button names or items in a list of items available for selection or choosing. Also used in the traditional sense for emphasis.
- **'Select'** — To click on an item using the mouse. No action follows a selection, but the item is highlighted, indicating it has been selected (selection is normally followed by another action).
- **'Choose'** — To click on an item using the mouse. An action, such as accepting a current selection or changing an item's attributes, normally follows a selection.
- **'Click On'** — Same as 'Select' or 'Choose' (see above).
- **'Double-Click On'** — To click on an item twice in rapid succession. Double-clicking on an item causes a selection, followed by the default action. For example, double-clicking on a filename in a file selection dialog causes the file to be selected and opened or saved (the first click selects the item, the second click is the same as pressing *OK*).
- **Bullet Item** — Used for emphasis or a list of important and related items.
- Step (➢ in the printed manual) — Used for describing step-by-step procedures.

> **NOTE:** Useful or important information — often a tip that may lead to improved results.

- **[1.1, 1.2]** — Numbers in brackets designate references found in Appendix VI. The first number is the material category (e.g., *Humans*), the second number is the reference number.
- `Filename` — Courier (monospace) font is used to designate user-entered alphanumeric strings (numbers and text).
- \<*Enter*> — Press the keyboard key labeled "Enter"; causes user-entered alphanumeric information to be read and stored by the program.

Problem Determination

Engineering Dynamics Company, LLC has support services available to you in case you encounter a problem while using HVE. The following are the preliminary steps you should perform to confirm your problem and assist EDC's technicians in providing the most efficient support possible:

- **Review The Manual** — Most issues related to usage are covered in the manual. If you encounter a problem that is not covered in the manual, continue with the following steps.
- **Duplicate The Problem** — If at all possible, try to duplicate the problem. This includes carefully retracing your steps to document how the problem occurs. It is *very* difficult for EDC's technicians to solve a problem unless they can duplicate it for testing on their computer system.
- **Create A Least Bombable Case** — This is EDC's in-house term, meaning "strip out all the superfluous details" before you begin testing. It is much easier for EDC (and you!) to solve a problem if you first remove irrelevant issues. Because it is *your* file, you are much more familiar with unrelated issues; otherwise, EDC's technicians will probably end up performing a lot of needless testing.
- **Contact EDC Technical Support** — Use e-mail, fax or telephone. Be prepared to provide the technical support representative the following information:
- Your user ID number
- Your System Hardware Profile report (available from EDC)
- HVE Version Information (available from the HVE Help system)
- Problem Description
- **Send The Case To EDC For Evaluation** — If the problem cannot be resolved by e-mail, fax or telephone, send a case file (see *Least Bombable Case*, above) to EDC for evaluation. The turn-around time varies, so discuss your requirements with the EDC support technician.

Window Manager Basics

If you have never used a computer with a window manager (i.e., X Windows or Microsoft Windows), you should learn some basic skills, such as using the mouse and choosing commands, before you start using HVE. This section of the HVE User's Manual provides some basic, yet important, concepts used by window manager systems.

Components

A window manager typically includes certain components that perform consistent and predictable functions routinely required by the user. The consistent nature of these components is fundamental to good user interface design, and helps the user learn how to use the application quickly.

HVE includes the following components:

- **Menu Bar** — The Menu Bar provides basic positioning and visibility control over an application:
- Click on the various Menu Bar options (*File, Edit, View, Mode, Set-up, 3-D Edit, Options* and *Help*) to display HVE's menus.
- Click on the Menu Bar to display HVE's main level viewers and dialogs (useful if they are hidden behind other windows or dialogs).
- Click on the Dialog Control button to display the Dialog Control menu. The selections on this menu allow the user to control the position and visibility of the window.
- Click on the Minimize button to iconify the window.
- Click on the Maximize button to cause the window to fill the screen.

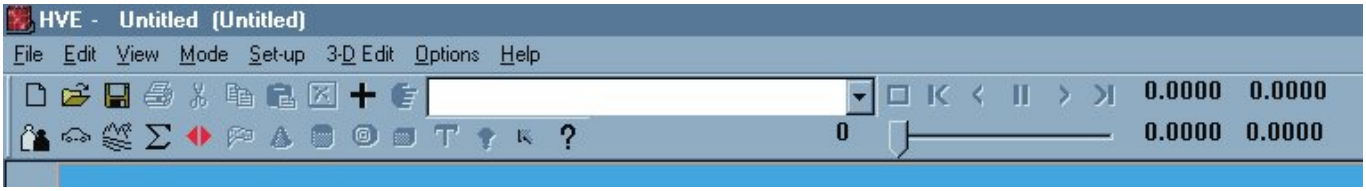


Figure 1: HVE's Menu Bar, with File, Edit, View, Mode, Set-up, 3-D Edit, Options and Help menu options.

- **Viewers** — Viewers are used to display and manipulate 3-D objects (humans, vehicles and environments). Viewers are used extensively by HVE, and understanding their use is important.

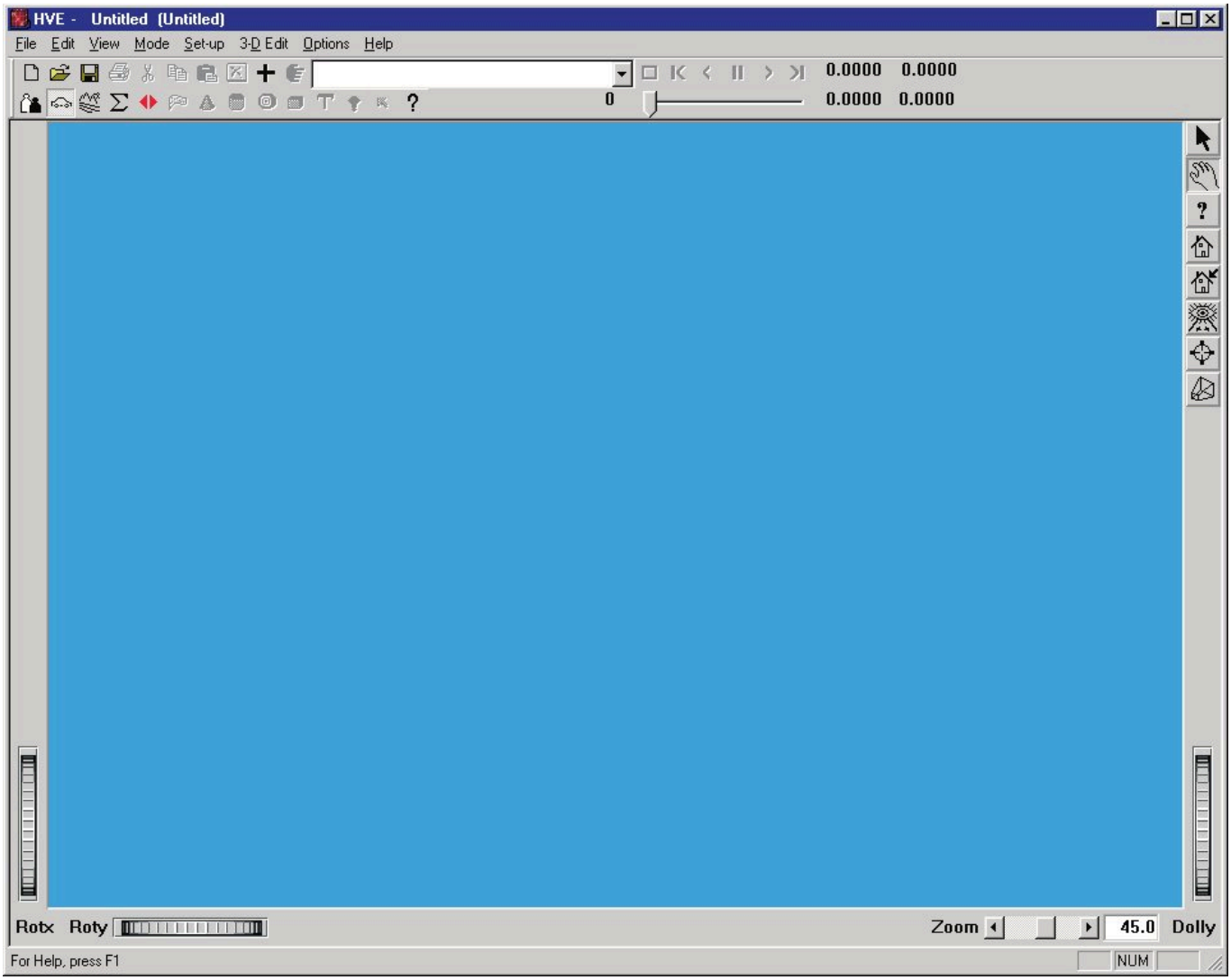


Figure 2: Typical Viewer. RotX and RotY thumb wheels rotate the object in the viewer about the viewer's X and Y axes, respectively. The Dolly thumb wheel increases or decreases the size of the object by moving the camera away from or towards the object. The Zoom slider increases or decreases the size of the object by reducing or increasing the viewer's included angle. Pick Mode or Manipulate Mode is selected by choosing the arrow or hand, respectively, near the upper right edge of the viewer.

All Viewers have at least two modes: *Pick Mode* and *Manipulate Mode*. The current mode is displayed along the right edge of the viewer. Choose *Pick Mode* to click on icons in the viewer (for example, click on the engine in the Vehicle Viewer to display the Drivetrain dialog). Choose *Manipulate Mode* to rotate and pan the object in the viewer.

- To Rotate, click and drag in the viewer using the left mouse button.
- To Pan, click and drag in the viewer using the middle mouse button (on a 3-button mouse). You may also drag by clicking the left mouse button while holding down the \<Shift> key.
- To Dolly the view, click and drag using the right mouse button. You may also dolly by clicking the left mouse button while holding down \<Shift> and \<Ctrl>.

> **NOTE:** These actions take a little practice, but, once mastered, are extremely helpful.

Mouse

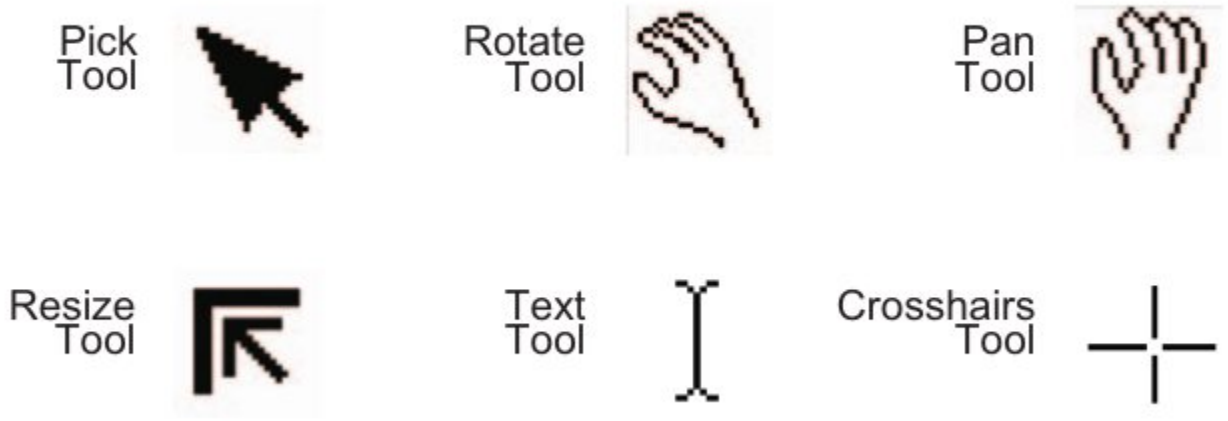


Figure 3: Mouse Pointer Tools (Pick, Rotate, Pan, Resize, Text, Crosshairs).

If you've never used a window-based application before, you'll need to know about using the mouse.

HVE normally uses mouse button 1 (this is normally the left mouse button, but can be reassigned using the Windows Control Panel). The middle and right mouse buttons are also used for specific tasks, as described below.

The mouse is used for performing a variety of tasks, such as selecting objects and entering text. The mouse pointer changes shapes, depending on the type of action it performs:

- **Pick** — Performs object selection in dialogs and viewers. In dialogs, it is used for choosing buttons and list selections. For viewers, it is used for choosing icons that display more information about the selected item.
- **Pan** — Used only in viewers, and only when *Manipulate Mode* is selected (see Viewers, above). Pan is performed using the middle mouse button, and drags the scene across the viewer in the direction of the mouse movement. (Left mouse button + \<Shift> may also be used.)
- **Rotate** — Used only in viewers, and only when *Manipulate Mode* is selected (see Viewers, above). Rotate is performed using mouse button 1, and causes the scene to be rotated about an axis normal to the center of the viewer in the direction of the mouse movement.
- **Resize** — Used for resizing viewers and modeless dialogs (modal dialogs cannot be resized; see below). To resize a dialog or viewer, click on the corner or along the edge and drag the mouse.
- **Text Cursor** — Used for entering text. After positioning the Text Cursor, typed data will appear to the left of the cursor. The Text Cursor also responds to the typical editing keys (Backspace, Delete and Insert).
- **Crosshairs** — Used by the 3-D Editor for entering X,Y,Z coordinates.

Dialogs and Other Interface Components

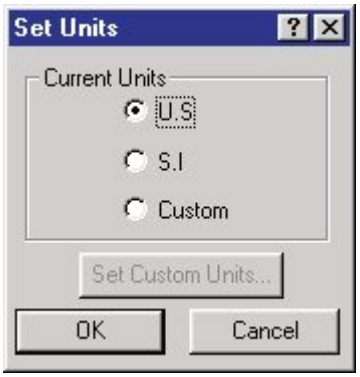


Figure 4: Modal Dialog (Set Units) and common dialog components.

Other components found in the HVE user interface include:

- **Modeless Dialogs** — A modeless dialog is used whenever an action need not be completed by pressing *OK*. For example, all Viewers (described earlier) are modeless dialogs. A particularly important issue concerning modeless dialogs is that items in the Menu Bar, such as *Open File* and *Set Camera*, are selectable when a modeless dialog is displayed.
- **Dialog Boxes (Modal Dialogs)** — A modal dialog is used whenever HVE requires information to carry out an operation. Modal dialogs always have *OK* and *Cancel* pushbuttons, and no other parts of the user interface (other than the modal dialog) may be accessed until one of these buttons is pressed (hence, the term *modal*). For example, if you try to choose the *File* menu while a modal dialog is displayed, HVE will either beep or ignore you.
- **OK Pushbuttons** — Click the *OK* pushbutton in any modal dialog to accept the entered data and continue with other operations.
- **Cancel Pushbuttons** — Click the *Cancel* pushbutton in any modal dialog to ignore any entered data and continue with other operations.
- **Help Pushbuttons** — Click the *Help (?)* pushbutton in any modal dialog to display help for the current dialog.
- **List Boxes** — List boxes allow you to select from among two or more options. Scroll through the list using the scroll bar, and then choose the option by double-clicking it, or by selecting it and pressing *OK*. For example, list boxes are used by file selection dialogs to choose the current filename.
- **Radio Buttons** — Radio buttons are used to present two or more mutually exclusive choices (compare with *Check Boxes*, below). Radio buttons are typically diamond-shaped (round on current Windows systems). The current choice is darkened and appears to be pushed in (like a radio pushbutton).
- **Check Boxes** — Check boxes are used to enable or disable an option. Check boxes are typically square-shaped. Unlike radio buttons, a series of check boxes is not mutually exclusive. An option is enabled when it is darkened and appears to be pushed in.
- **Command Buttons (Pushbuttons)** — Command buttons cause an action to occur, such as resetting values or displaying another dialog box.

> **NOTE:** A Command Pushbutton with an ellipsis (...) always causes another dialog box to be displayed.

- **Text Entry Boxes** — Some options require typed information. If the box is empty, simply place the mouse cursor in the box (the cursor will turn into a text tool) and start typing. If the box already contains information, click on the

information using the text tool to edit the field. To select the information, double-click on the field. Once selected, the information you enter will replace the existing information. You may also use the Backspace and Delete keys to edit the information.

- **Combo Boxes** — These are sometimes called *Drop-down List boxes*. A Combo Box displays a choice of options when you click the arrow next to the box. You can also edit the text in the box. Editing causes a new entry to be added to the list. Double-click on an entry to delete it from the list.
- **Option Lists** — An option list presents a choice of options when you click on the bar near the right end of the button. An option list is very similar to a Combo Box, with one major exception: the list cannot be edited.
- **Scroll Bars** — Scroll bars are used to navigate through list boxes. Click on the arrow at the bottom to move down the list, one item at a time. Click on the arrow at the top to move up the list. Click and hold the arrow down to automatically scroll through the list. You can also use the thumb slider to scroll more rapidly through the list, and click in the thumb slider trough to jump to a location in the list approximated by where you clicked.
- **Thumb Wheels** — Thumb wheels are used in 3-D viewers to pan, dolly in and out, and rotate the current scene (see Figure 2).

Chapter 1 — What Is HVE?

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 1, pages 1-1 through 1-7. Revised for the current HVE release.

Overview

HVE is a computer environment for studying interactions between humans, vehicles and their environment. HVE stands for

Human-Vehicle-Environment

, the three factors which influence a crash. HVE allows the user to create detailed, 3-dimensional models of humans, vehicles and environments, study their interaction using HVE-compatible reconstruction and simulation models, and to combine the results of several interactions involving multiple humans and vehicles into a single coherent sequence.

HVE is essentially five editors:

- **Human Editor** — Used for creating and editing 3-D models of humans
- **Vehicle Editor** — Used for creating and editing 3-D models of vehicles
- **Environment Editor** — Used for creating and editing 3-D models of the environment
- **Event Editor** — Used for setting up and executing reconstruction and simulation models which study the interactions between the selected humans, vehicles and environment
- **Playback Editor** — Used for combining the results of several events into a single, coherent sequence which may include multiple humans and vehicles

These five editors are the natural extension of what HVE does. Each editor is described briefly here in this section. For a complete description, refer to the individual editor's section in this manual.

Human Editor

The HVE Human Editor, like each of the five editors, has an information dialog and a viewer.

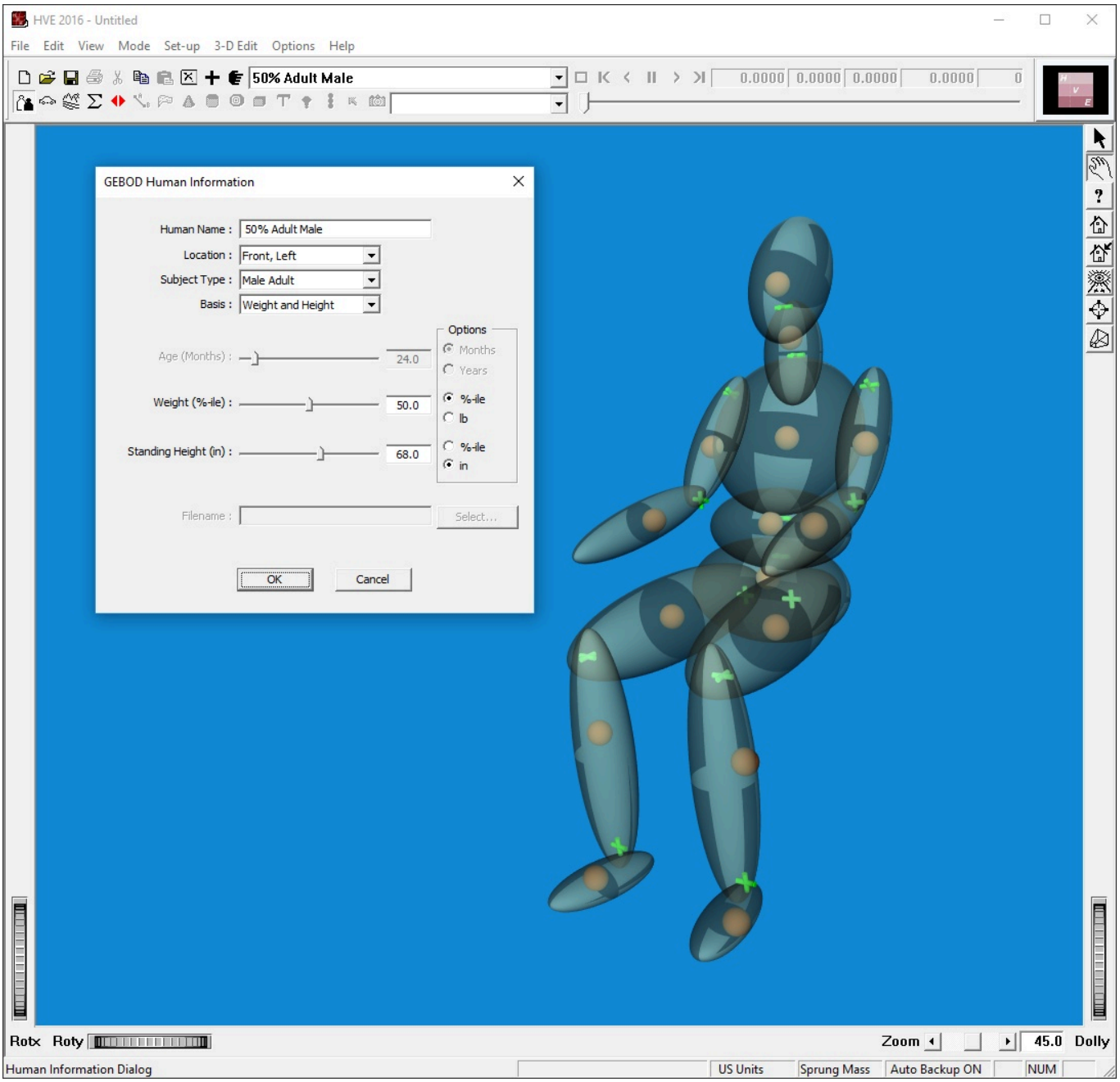


Figure 1-1: HVE Human Editor.

The HVE Human Editor contains several components common to the other four editors:

- Add Human pushbutton (*Add New Object* button on the toolbar)
- Active Humans List
- Current Human label
- Delete Human pushbutton (*Delete* button on the toolbar)

The Add Human pushbutton provides access to the HVE Human Databases and allows the user to add new humans to the current case. Any number of humans may be created, and, once created, are added to the Active Humans List. At any given time, the name of one of these humans is displayed in the Current Human label. This human is currently loaded in the editor and displayed in the Human Viewer. The current human may be modified using the tools available in the

Human Editor. If the current human is no longer required for study, it may be deleted using the Delete Human pushbutton.

See also the dialog reference: Humans, in particular Human Information Dialog.

Vehicle Editor

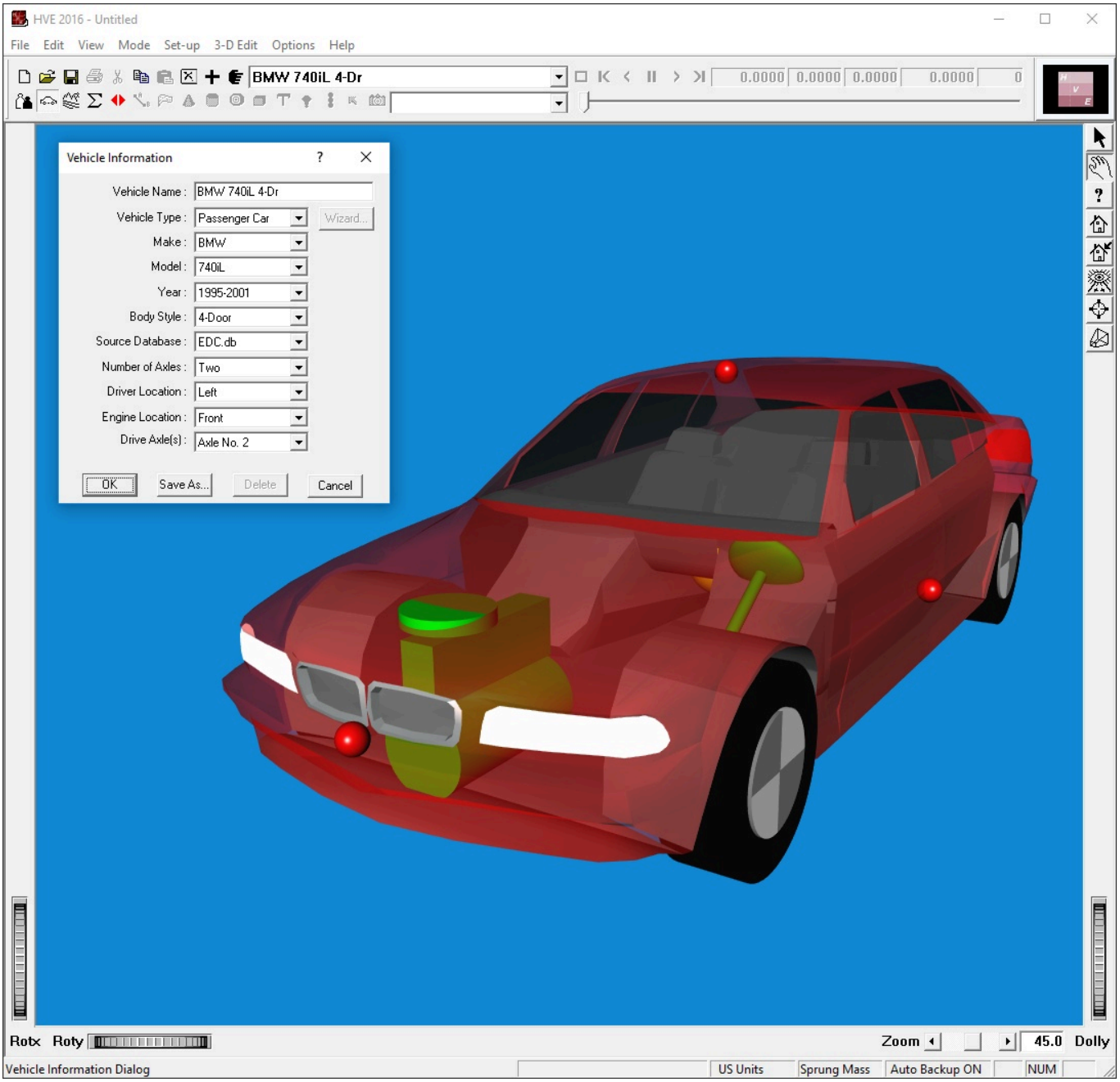


Figure 1-2: HVE Vehicle Editor.

This editor contains the following components:

- Add Vehicle pushbutton (*Add New Object* button on the toolbar)
- Active Vehicle List

- Current Vehicle label
- Delete Vehicle pushbutton (*Delete* button on the toolbar)

These components work for vehicles exactly like their Human Editor counterparts.

See also: Vehicles, in particular Vehicle Information Dialog.

Environment Editor

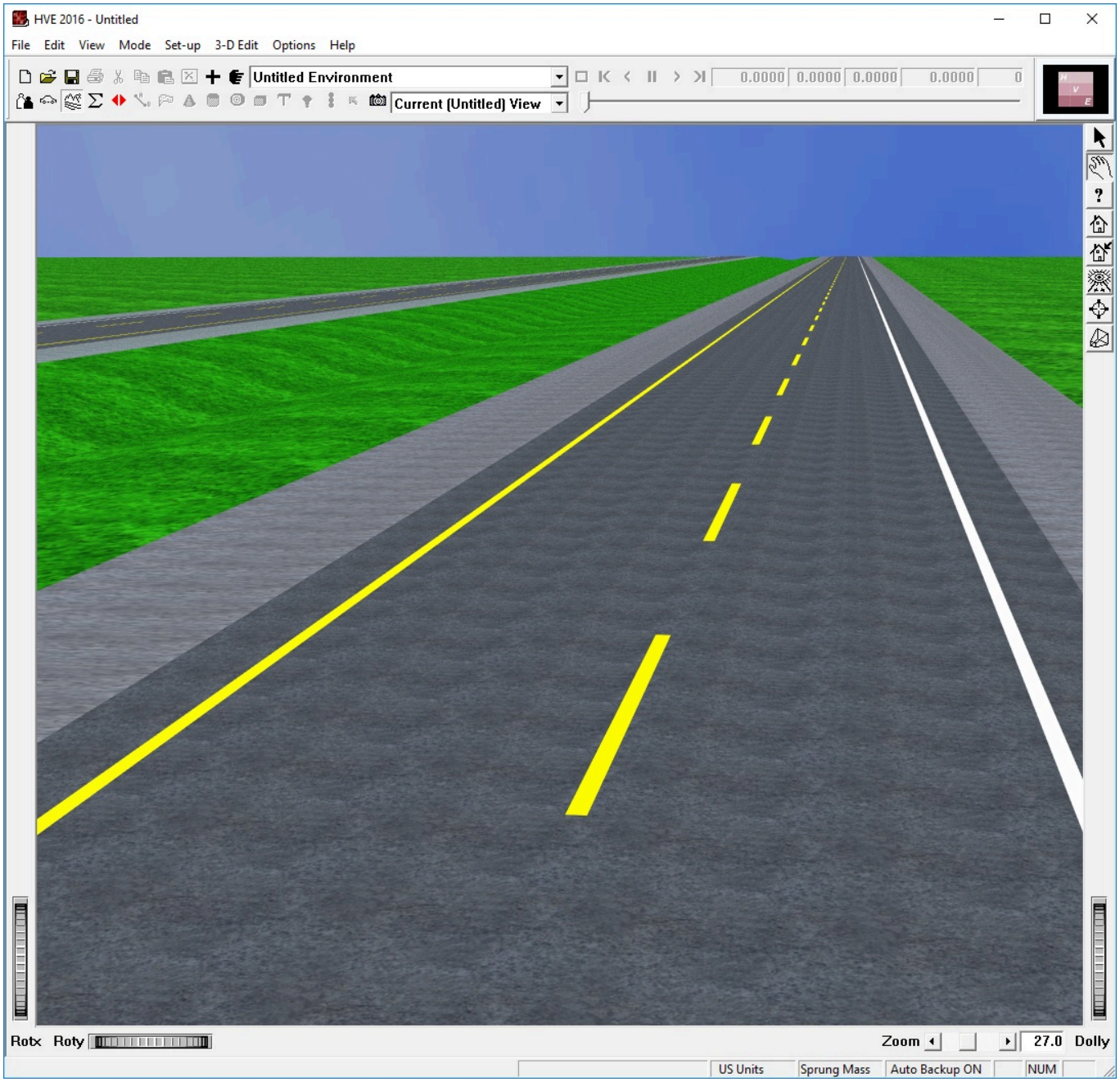


Figure 1-3: HVE Environment Editor.

The Environment Editor contains the following components:

- Add Environment pushbutton (*Add New Object* button on the toolbar)

- Current Environment label
- 3-D Editor (select *3-D Edit, Launch 3-D Editor* from the main menu)

These components work for environments exactly like the previously described editors.

See also: Environment, in particular Environment Information Dialog, and the 3-D Editor reference.

Event Editor

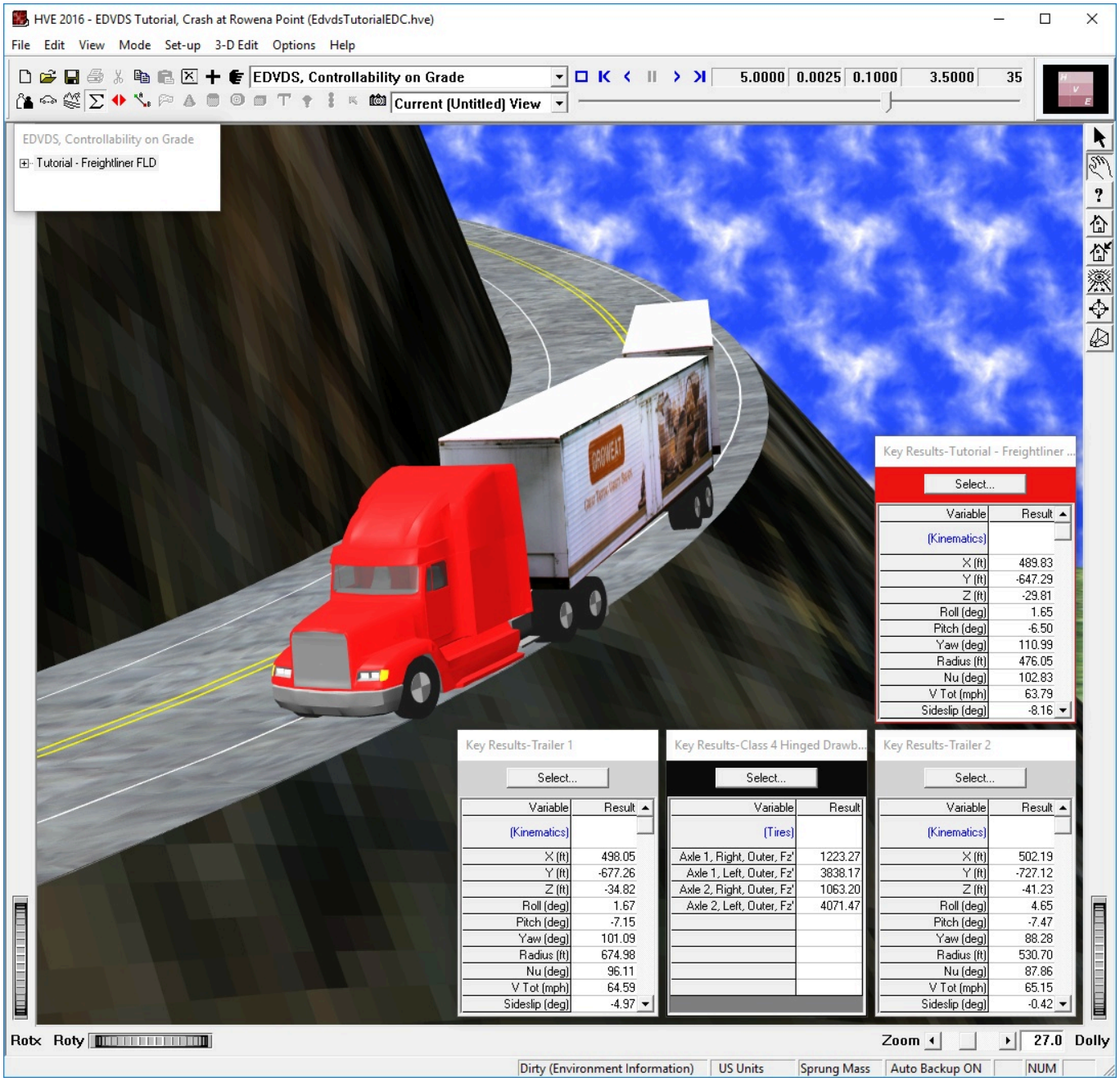


Figure 1-4: HVE Event Editor.

The Event Editor is used to study (i.e., simulate or reconstruct) the interactions between any of the active humans, vehicles and environment. An HVE Event is defined by selecting one or more humans and vehicles and a reconstruction

or simulation model.

The Event Editor contains the following components:

- Add Event pushbutton (*Add New Object* button on the toolbar)
- Active Event List
- Current Event label
- Copy Event pushbutton (*Copy* button on the toolbar)
- Delete Event pushbutton (*Delete* button on the toolbar)

These components work for events exactly like the other editors. The Event Editor also has an Event Controller that allows the user to execute, pause and review the current event.

See also: Events & Driver Controls, in particular Event Information Dialog and Event Set-up Dialog.

Playback Editor

The HVE Playback Editor panel contains the following components:

- Add Report Window pushbutton (*Add New Object* button on the toolbar)
- Add Playback Window (added from Playback mode via the Playback Information dialog; the former *Options, Add Playback Window* menu path has been removed)
- Delete Window pushbutton (*Delete* button on the toolbar)

These components work for playback windows exactly like the other editors. The Playback Editor also has a Playback Controller that allows the user to execute, pause and review trajectory simulation windows.

Unlike the other editors, the Playback Editor may display multiple viewers, called Report Windows, each containing different output results for a particular event. These results may be:

- Numeric Reports
- Graphic Reports
- Trajectory Simulations
- Variable Output Tables

The Playback Editor is used to combine several individual trajectory simulations into a single Playback Window. The Playback Window may display multiple humans and vehicles from any number of events. This window also provides a professional video output capability.

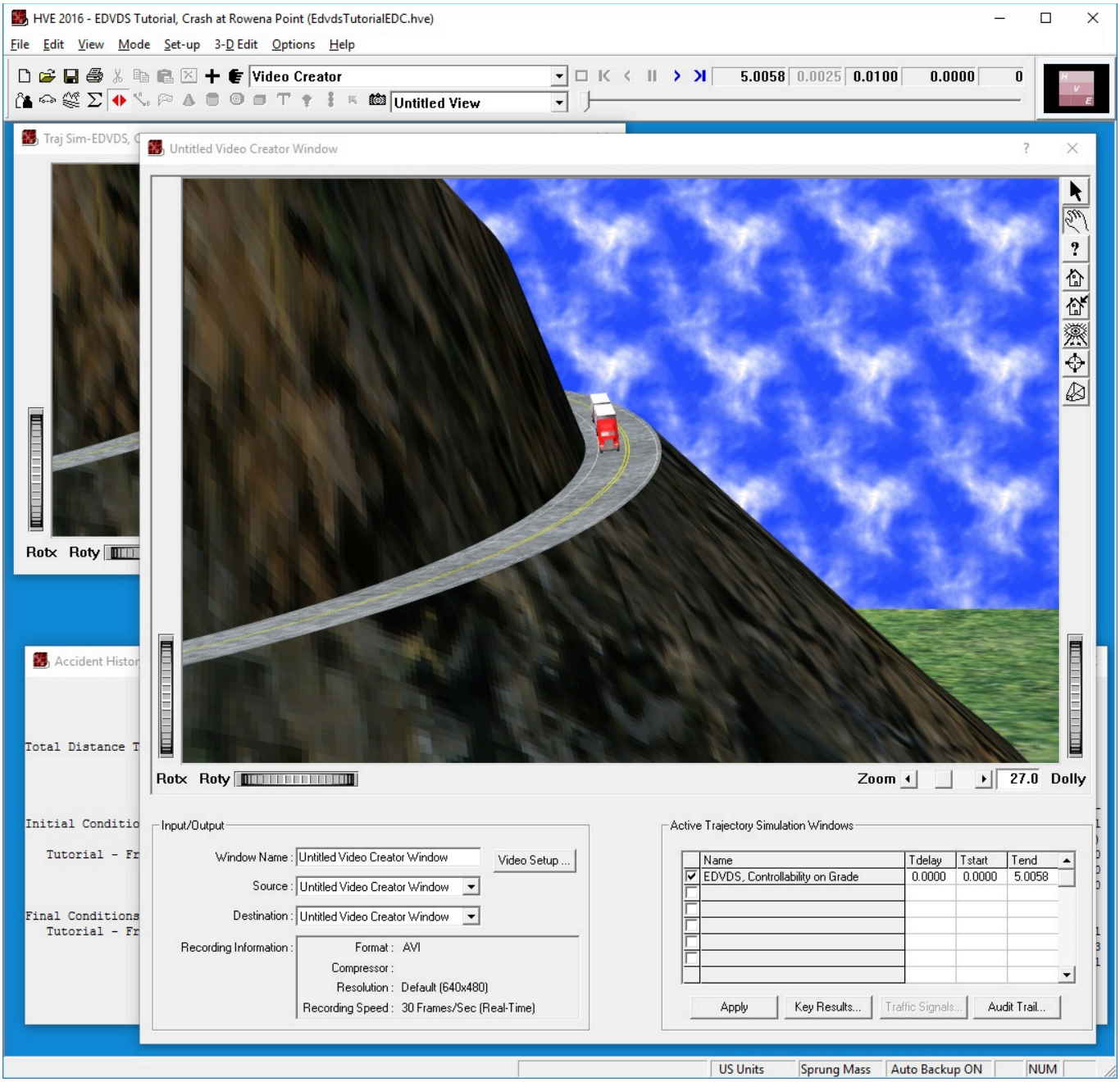


Figure 1-5: HVE Playback Editor, illustrating several types of windows and viewers.

For detailed information about each of these five HVE editors, refer to the individual editor's section in this manual.

See also: Reports & Output and Playback Controls.

Chapter 2 — How To Use HVE

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 2, pages 2-1 through 2-104. Revised for the current HVE release. Because of its length, Chapter 2 is split into three parts:

Part	Contents
Part A (this file)	Getting Started, HVE Menu Bar, HVE Modes, Starting a New Case, Creating Humans/Vehicles/Environments, 3-D Editor, Creating Events, Setting Up Events, Executing Events, Choosing Options
Part B — 02b	Creating Report and Playback Windows, Playback Controller, Audit Trail, Creating Case Files, Creating Databases, Printing Results, Setting the View (cameras, overlays)
Part C — 02c	Selecting User Options (Key Results, show toggles, Grid, Units, Render, Simulation Controls, Playback, Calculation Options, DyMESH, Get Surface Info, Preferences), Getting Help, Video Interface

We think you will find HVE easy to use. All the operations are simple and intuitive. You'll find yourself thinking more about the *objects* in your study (*humans*, *vehicles* and *environments*) and much less about *data* (weight, wheelbase and other details).

Using HVE involves performing the following steps:

- Use the HVE Human Editor to select and create all the humans involved in your study.
- Use the HVE Vehicle Editor to select and create all the vehicles involved in your study.
- Use the HVE Environment Editor to select and create the environment involved in your study.
- Use the HVE Event Editor to set up and execute reconstruction and simulation models which analyze the interactions between the selected humans, vehicles and environment.
- Use the HVE Playback Editor to combine multiple events into a single coherent sequence involving multiple humans and vehicles. You can also use the Playback Editor to create printed, plotted and video output showing and documenting the events.

Getting Started

To start HVE, use the Start, Programs, HVE cascade menu to select the HVE program icon, just as you would start any other program on your computer. After starting HVE, the current Editor will be displayed, along with the menu bar.

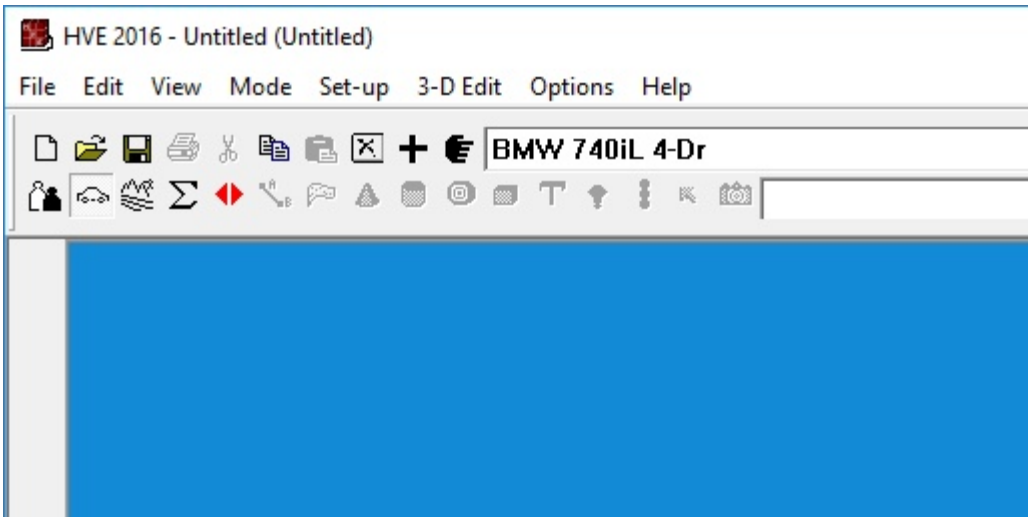


Figure 2-1: HVE, ready to begin. The HVE Menu Bar and current editor (in this case, the Vehicle Editor) are displayed. The Mode pushbuttons on the toolbar allow the user to select the current editor.

HVE Menu Bar

The HVE Menu Bar allows the user to select from the following pull-down menus:

- **File Menu** — Provides access to file handling, printing and video output information.
- **Edit Menu** — Provides access to editing functions used in various portions of HVE, such as the 3-D Editor.
- **View Menu** — Provides access to various viewing options.
- **Mode Menu** — Provides access to select the current editor, and to add new or previous objects to the editor.
- **Set-up Menu** — Provides access to event set-up options.
- **3-D Edit Menu** — Provides access to 3-D Editor functionality.
- **Options Menu** — Provides access to various user options and preferences.
- **Help Menu** — Provides access to HVE's extensive on-line help system, HVE version information and also EDC Technical Support information.

Each of these menus provides access to important user information and options which apply to all HVE operations. Refer to the Menu Reference section of this manual for detailed information about each option available in the HVE Menu Bar.

(updated: the current HVE menu bar is File, Edit, View, Mode, Set-up, 3-D Edit, Options, Help. The current File menu includes New, Open, Save, Save As, Export Preview Case, Print, Print All, Video Creator, Export, FBX Export, recent files and Exit. The Help menu now also includes a User Manuals submenu — HVE, EDCRASH, EDGEN, EDHIS, EDSMAC, EDSMAC4, EDVDS, EDSVS, EDVSM, EDVTS, SIMON, Damage Studio, GATB and ReadDataFile — plus Online Licensing options for registering a User ID code and refreshing licenses.)

HVE Modes

HVE's functional design is based on the following five modes:

- **Human Mode** — Provides access to the Human Editor
- **Vehicle Mode** — Provides access to the Vehicle Editor
- **Environment Mode** — Provides access to the Environment Editor
- **Event Mode** — Provides access to the Event Editor

- **Playback Mode** — Provides access to the Playback Editor

Each of these modes has an editor used for creating and studying the objects (humans, vehicles and environment) of interest. The desired mode is selected by pressing the desired mode button at the top of the current editor (see Figure 2-1).

(updated: modes may also be selected from the Mode menu, which assigns the keyboard shortcuts Ctrl+1 (Human), Ctrl+2 (Vehicle), Ctrl+3 (Environment), Ctrl+4 (Event) and Ctrl+5 (Playback), and provides Add... New/Previous and Information... (Ctrl+I) options.)

Starting A New Case

A *case* is a file containing the results of an HVE session. If you didn't open a case when you started HVE, you can do so using the *Open Case* option found in the File menu. The Open Case option displays the File Selection dialog which displays all your previously run case filenames. Simply double-click on the desired filename and the case file will be loaded.

If you choose not to open a case file, you will begin a new case.

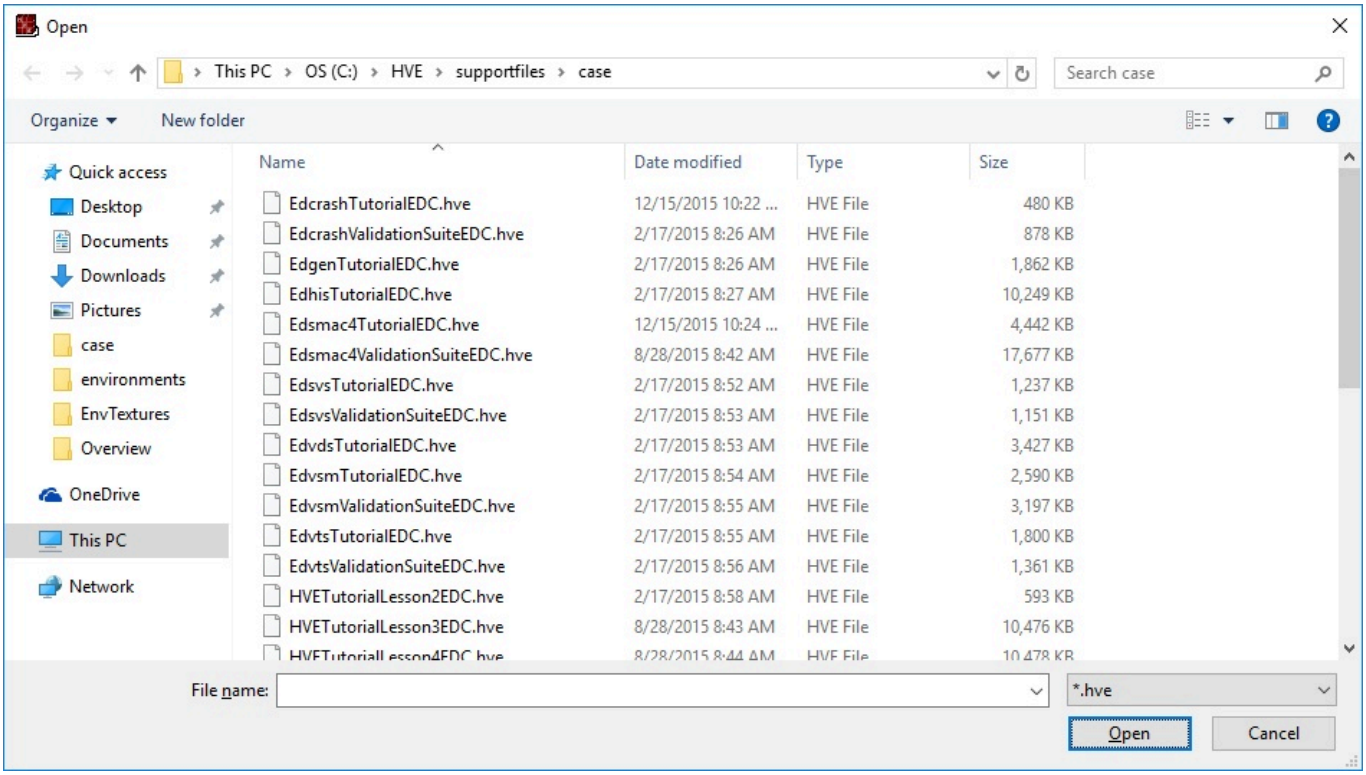


Figure 2-2: The Open Case File Selection dialog allows the user to open previously saved cases (`*.hve` files).

Creating Humans, Vehicles and Environments

The first step is to create the humans, vehicles and environment to be studied.

Creating Humans

Humans are created and edited using the HVE Human Editor. To create the required humans:

1. Click on the Human Mode button on the toolbar. This puts HVE in Human Mode.

2. Click on *Add New Object* to display the Human Information dialog (see Figure 2-3).
3. Choose the human location (any of nine occupant positions or pedestrian).
4. Choose the human according to *Sex*, *Age*, *Weight* percentile and *Height* percentile.
5. Choose *OK*.

The selected human is now displayed in the Human Viewer. At this point you may choose to edit the current human's properties (refer to the Human Editor section of this manual for details; see also the Human Information dialog reference). The above steps are repeated for each human to be included in the study.

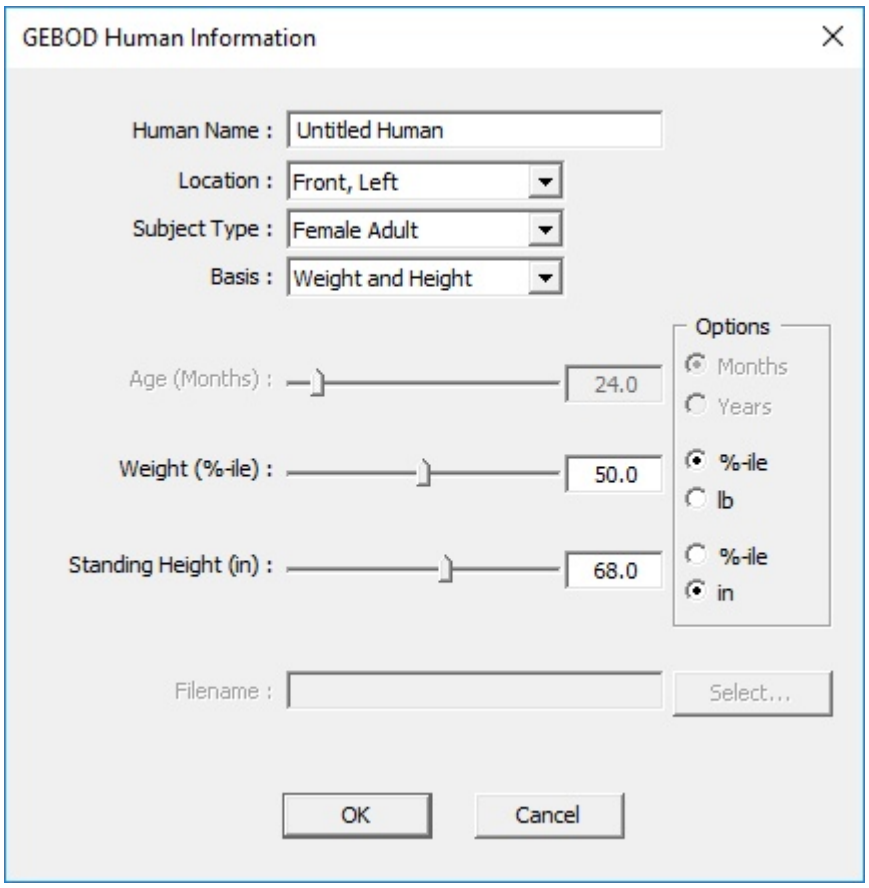


Figure 2-3: The HVE Human Editor and Human Information dialog (Human Name, Location, Sex, Age, Weight Percentile, Height Percentile).

Creating Vehicles

Vehicles are created and edited using the HVE Vehicle Editor. To create the required vehicles:

1. Click on the Vehicle Mode button on the toolbar.
2. Click on *Add New Object* to display the Vehicle Information dialog (see Figure 2-4).
3. Choose the vehicle *Type*, *Make*, *Model*, *Year* and *Body Style*.
4. If desired, edit the *Number of Axles*, *Driver Location*, *Engine Location* and *Drive Axle*.
5. Choose *OK*.

The selected vehicle is now displayed in the Vehicle Viewer. At this point you may choose to edit the current vehicle's properties (refer to the Vehicle Editor section of this manual for details; see also the Vehicle Information dialog

reference).

Repeat the above steps for each vehicle to be included in the study.

> **NOTE:** A multi-vehicle train (e.g., a tractor-trailer) is created by creating its individual vehicle units, one at a time, using the Vehicle Editor.

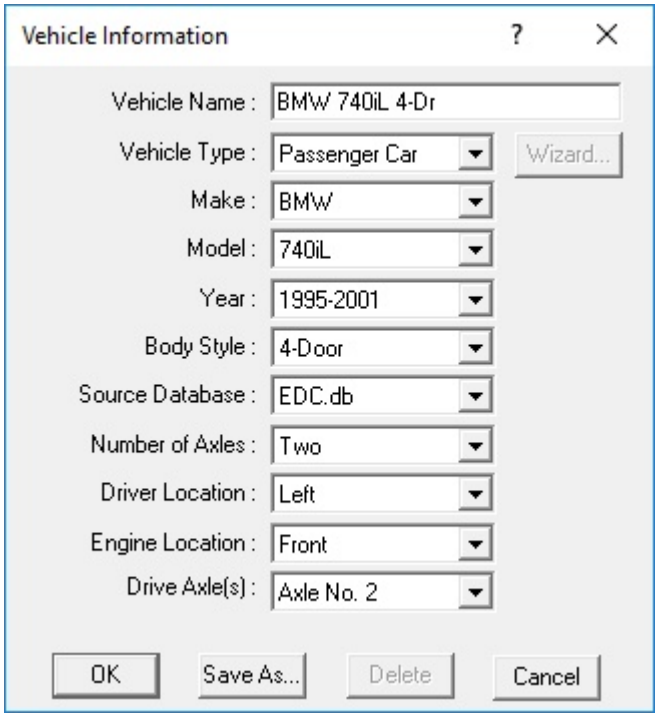


Figure 2-4: The HVE Vehicle Editor and Vehicle Information dialog (Vehicle Name, Vehicle Type, Make, Model, Year, Body Style, Source Database, Number of Axles, Driver Location, Engine Location, Drive Axle(s)).

Creating Environments

To create the environment:

1. Click on the Environment Mode button on the toolbar.
2. Click on *Add New Object* to display the Environment Information dialog.
3. Select the *City, State* and *Country* using the Location Database.
4. Enter the *Time* and *Date* of the Event.
5. Enter the *Angle of the X axis* (relative to North) of the earth-fixed coordinate system used in your study.
6. Enter the *Wind Speed* and *Direction*.
7. Enter the *Barometric Pressure* and *Ambient Temperature*.
8. Enter the *Gravitational Constant*.

> **NOTE:** The latitude, longitude and hours from GMT, along with the time and date of the accident and angle of the X axis, are used to position the sun in your environment.

1. Choose *Sky Attributes* to display the Sky Attributes dialog. Enter or modify the atmospheric visibility conditions (*Sky Color, Fog Type, Maximum Visibility* and *Fog Color*).

2. Choose *Open* to load an existing environment 3-D geometry file and/or a scanned photograph of the environment.

> **NOTE:** The File Selection dialog includes an option list allowing the user to select the file format (h3d, dxf, rgb, etc.) and the desired input file. Where necessary, HVE converts the selected file into the HVE file format.
>
> **NOTE:** The Options pushbutton allows the user to scale and rotate the environment model as required for HVE's SAE coordinate system.

1. Choose *OK*.

The environment is now displayed in the Environment Viewer.

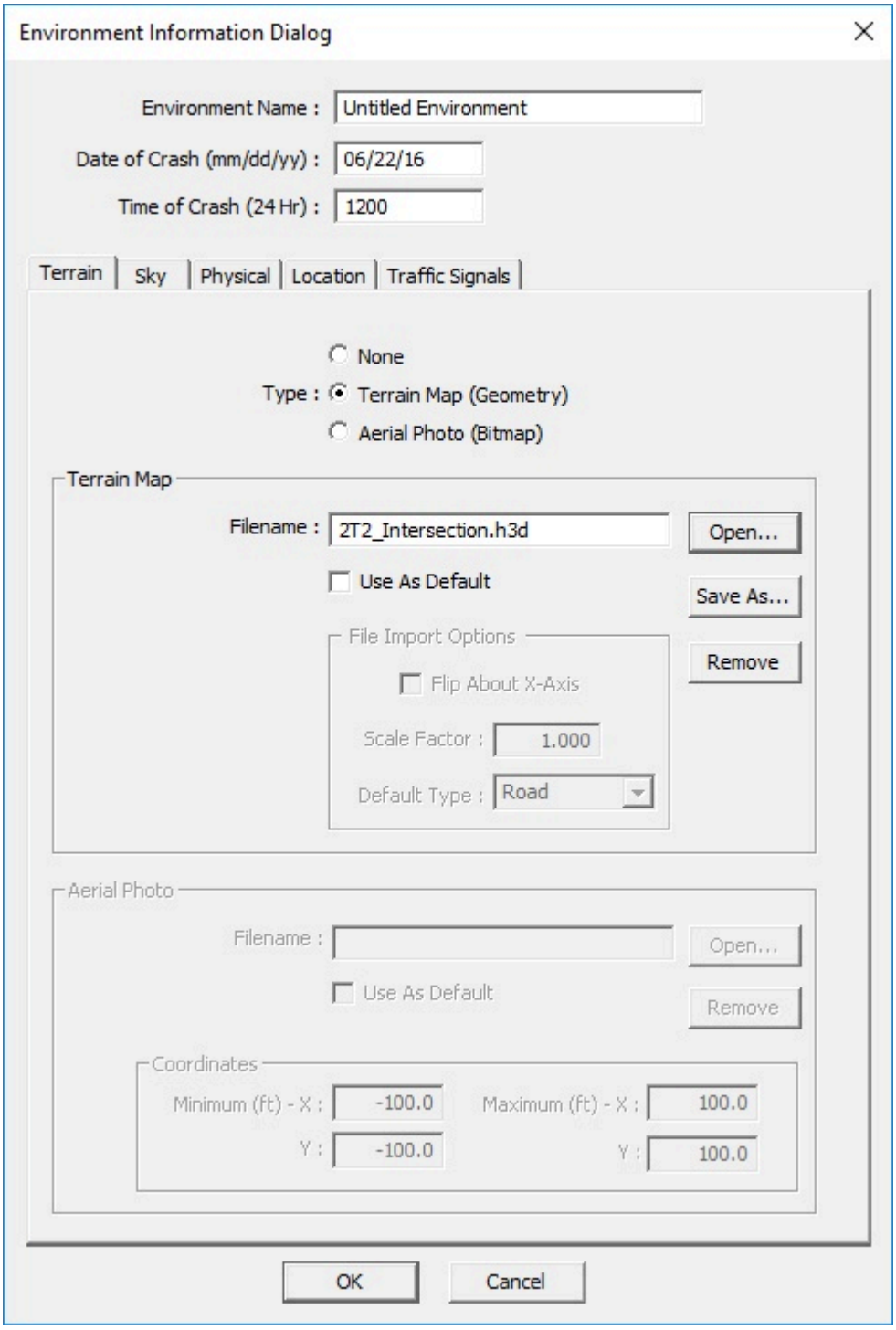


Figure 2-5: The Environment Information dialog (Current Location, Latitude, Longitude, GMT offset, Environment Name, Date and Time of accident, Angle of X Axis, Wind Speed and Direction, Barometric Pressure, Temperature, Gravity, Environment 3-D Geometry File, Environment Scanned Image File, Sky Attributes).

See also: Environment Information dialog reference and Sky Attributes dialog.

3-D Editor

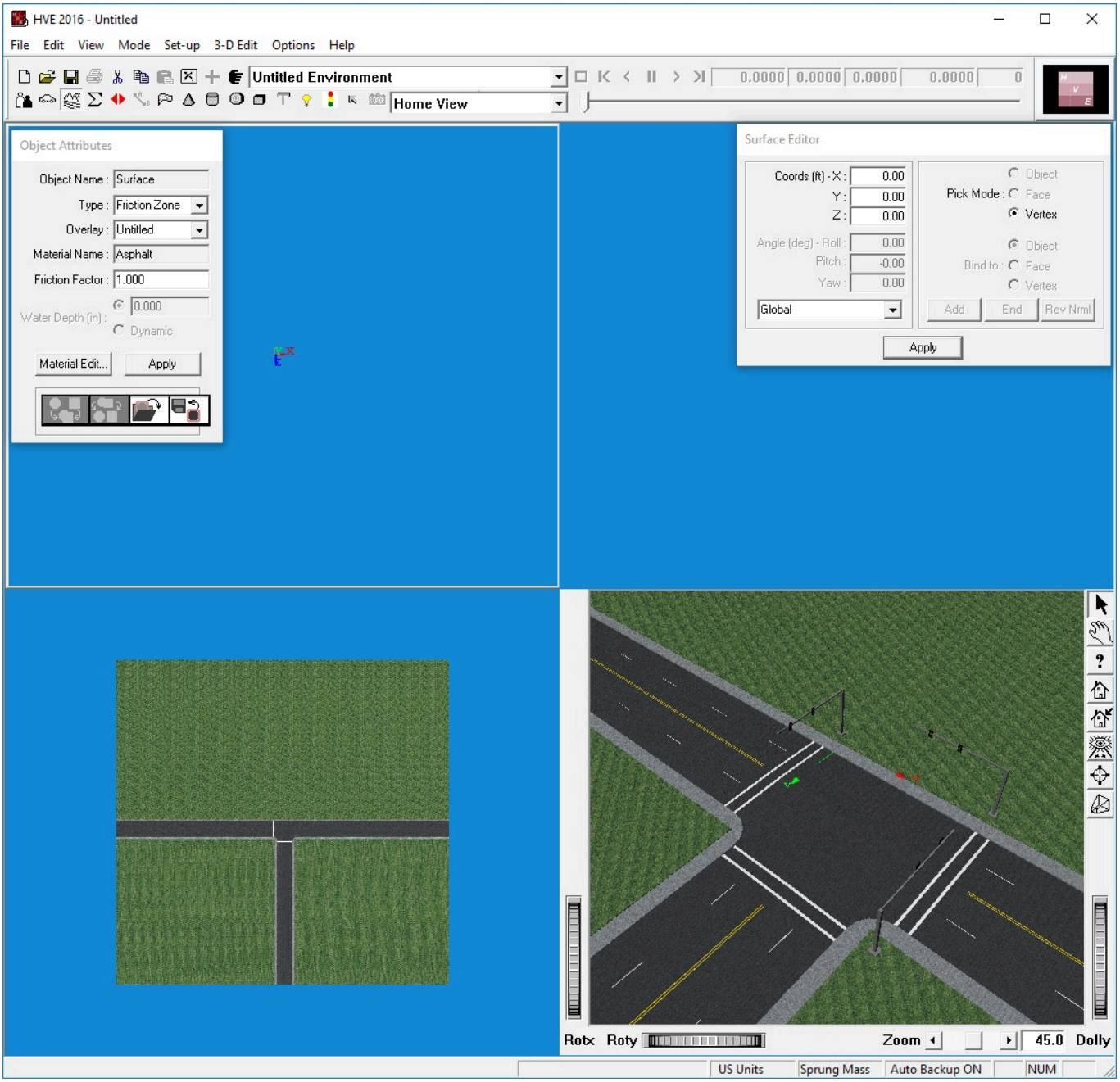


Figure 2-6: HVE 3-D Editor, used for creating and editing 3-D geometry data used to visualize the humans, vehicles and environment for the current case. The 3-D Editor has four viewers (Perspective plus X-Y, Y-Z and X-Z).

The 3-D Editor is a 3-D modeling tool which may be used to create and edit the 3-D geometry data used for visualizing environments.

The 3-D Editor is started in Environment Mode by selecting the 3-D Edit menu option, then selecting

Launch 3-D Editor

. The current object is loaded for editing. In this case, it has the current environment loaded and ready for editing its visual and surface friction properties.

For more information about the 3-D Editor, refer to the *3-D Editor* tab in the *Tools* section of this manual (see also the 3-D Editor reference).

(updated: the current 3-D Edit menu contains Launch 3-D Editor (Ctrl+L), Material Color..., Material Texture..., a Manipulators submenu (Direct, Jack, Trackball, Centerball, Handle Box, Transform Box, Tab Box), Object Attributes..., Viewer, and Close 3-D Editor.)

Creating Events

An HVE event is a single reconstruction or simulation analysis of one or more humans and vehicles in the current environment. Event Mode is where all the action (or

interaction

) is. The calculation model is actually executed in Event Mode.

To create an HVE event:

1. Click on the Event Mode button on the toolbar. The Event Editor is displayed.
2. Click on *Add New Object* to display the Event Information dialog (see Figure 2-7).
3. Select the human and vehicle objects and the calculation model (see below).
4. Choose *Calculation Options* to display and edit any program options.
5. Choose *OK*.

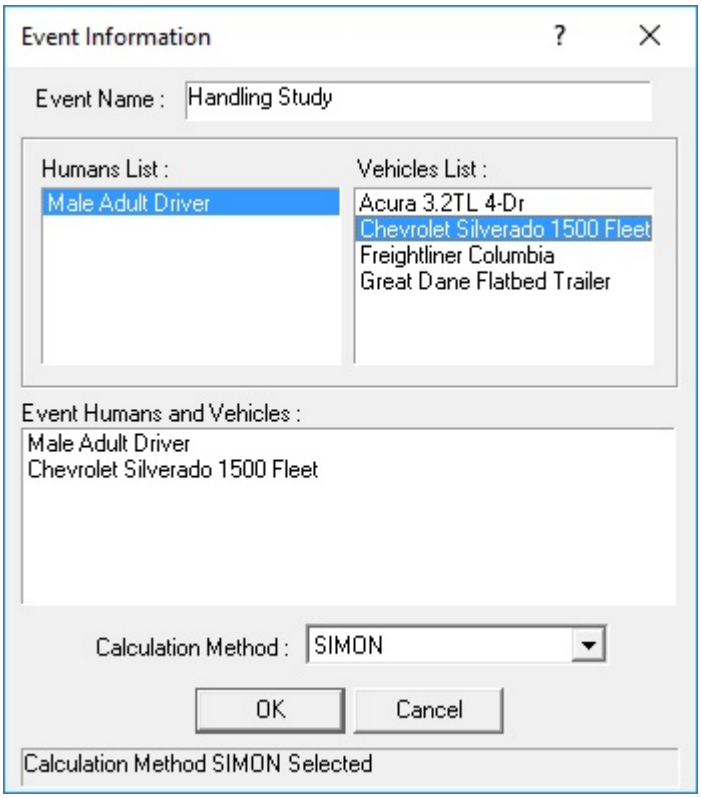


Figure 2-7: The Event Information dialog — Event Name, Humans List, Vehicles List, Event Humans and Vehicles, Calculation Method.

See also: Event Information and the per-model Calculation Options references.

Choosing A Calculation Model

The choice of calculation models is dependent upon the needs of the study. Normally, reconstruction models are executed first, because they provide part or all of the initial conditions required to execute simulations.

Reconstruction Models

According to SAE [6.4], a

reconstruction

is the determination of initial conditions (usually impact speed) based on post-accident evidence. To execute an HVE reconstruction program in the HVE Event Editor, the user normally supplies (for each vehicle) some or all of the following information during Event Mode:

- Pre-impact position
- Impact position
- Path positions between impact and rest
- Rest position
- Vehicle damage

> **NOTE:** The actual requirements are dependent on the particular reconstruction method.

The reconstruction calculations usually provide some or all of the following results:

- Initial Velocity
- Velocity at Begin Perception
- Velocity at Begin Reaction
- Velocity at Begin Braking
- Velocity at Impact
- Velocity at Separation
- Final Velocity
- Delta-V during Impact
- Energy absorbed during Impact
- PDOF during Impact

Simulation Models

According to SAE [6.4], a

simulation

is the prediction of human or vehicle paths and damage based on initial conditions (position and velocity). To execute an HVE simulation program in the HVE Event Editor, you will usually supply (for each human and/or vehicle) some or all of the following information during Event Mode:

- Initial position and velocity
- Driver Controls (for vehicles)
- Collision Pulse (for vehicles)
- Mesh (for vehicles)

- Payload (for vehicles)
- Wheels (for vehicles)
- Accelerometers (for vehicles)
- Restraints (for occupant simulation)
- Contacts (for occupant and pedestrian simulation)

> **NOTE:** The actual requirements are dependent on the particular simulation method.

The simulation calculations usually provide some or all of the following results (reported as a function of time):

- Sprung Mass Kinematics (position, velocity and acceleration)
- Sprung Mass Kinetics (forces)
- Accelerometers (velocity and acceleration)
- Damage (vehicle exterior crush geometry and force)
- Tire Conditions (contact patch location, forces, slip angles, skidding)
- Wheel Conditions (position and orientation, forces and moments at wheels and suspensions)
- Connections (articulation angles, forces and moments)
- Driver Controls (throttle, braking, steering and gear selection, HVE Driver Path Follower results)
- Contact Surface Data (force, deflection)
- Belt Data (stretch and tension)
- Airbag Data (pressure, radius, force)

User-written Reconstruction and Simulation Models

HVE has an open architecture which allows users to write their own reconstruction and simulation models. Users are urged to consider this feature, especially when existing models don't quite consider all the issues. Refer to Appendix VII (HVE Developer's Toolkit Overview) and references 1.2 and 1.3 for specific instructions on how to produce your own reconstruction and simulation models.

Setting Up Events

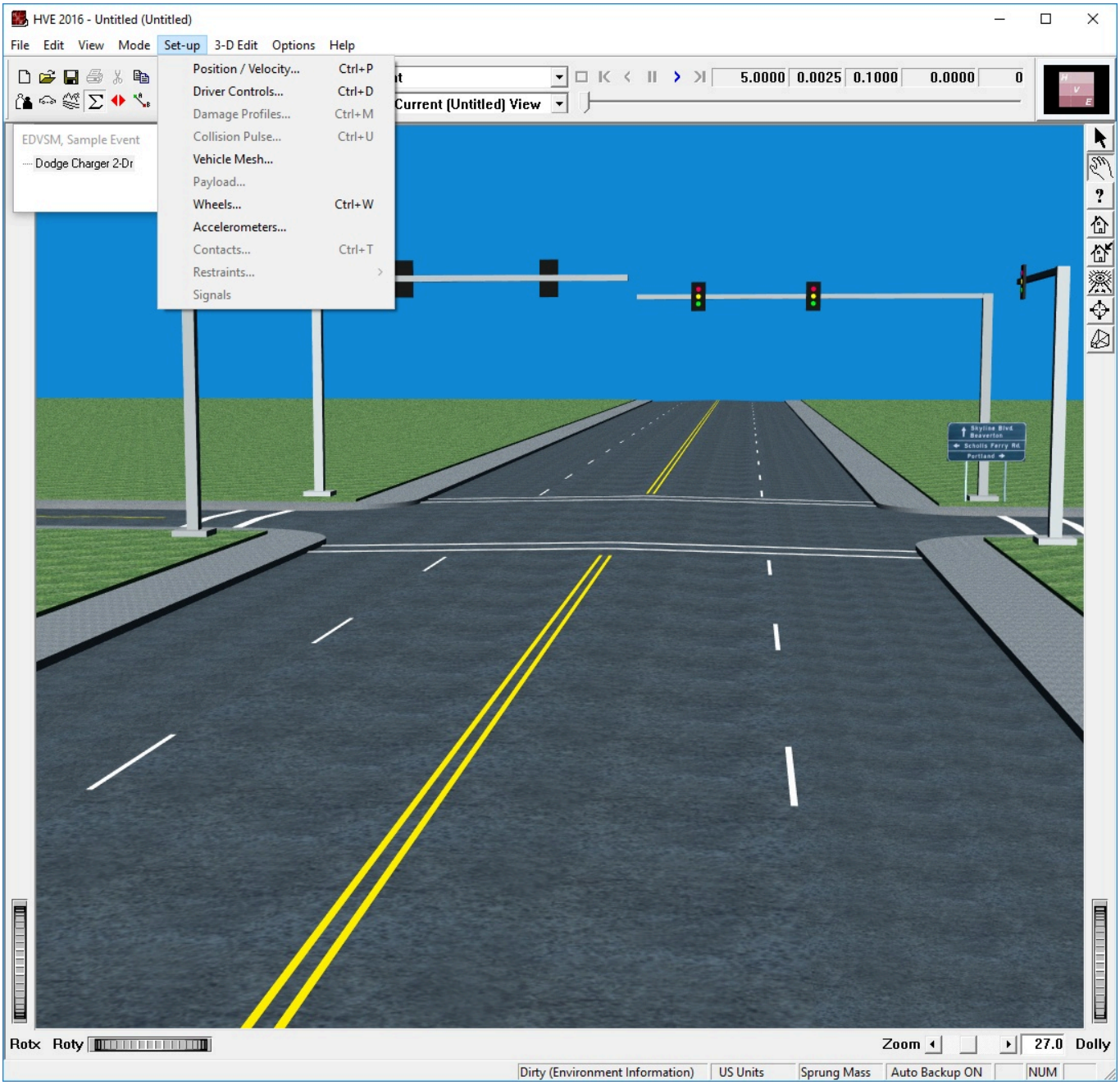


Figure 2-8: The Set-up Menu allows the user to choose from the available Event Set-up options.

Events are

set up

after the objects (humans and vehicles) and calculation method are selected. Event set-up involves the following steps:

1. Assign the required positions and velocities for each object
2. Assign the desired driver controls (throttle, braking, steering and gear selection tables for simulations; wheel data for reconstructions)
3. Assign any damage profile data
4. Assign any payload data

5. Assign any collision pulse
6. Assign any mesh data (tessellation, inter-vehicle friction)
7. Assign any wheel data (tire blow-out, wheel displacement, brake adjustment)
8. Assign any accelerometer positions
9. Assign restraints (belts and airbag usage information) for occupant simulations
10. Assign contacts (allowable interactions between human ellipsoids and vehicle contact surfaces) for occupant and pedestrian simulation

Event setup options are available using the Set-up Menu (see Figure 2-8).

(updated: the current Set-up menu contains Position/Velocity... (Ctrl+P), Driver Controls... (Ctrl+D), Damage Profiles... (Ctrl+M), Collision Pulse... (Ctrl+U), Vehicle Mesh..., Payload..., Wheels... (Ctrl+W), Accelerometers..., Contacts... (Ctrl+T), a Restraints... submenu with Airbags... (Ctrl+G) and Belts... (Ctrl+B), plus Signals and Notes items that postdate the 2006 manual.)

Assigning Positions and Velocities

Every simulation or reconstruction model requires positioning of humans and/or vehicles within the environment. Simulations and some reconstruction models also require that velocities be entered. Positioning is always the first step during event set-up.

HVE allows the user to position humans and vehicles within the environment using direct manipulation of the object, or by entering position coordinates and orientations in the Position/Velocity dialog (see Figure 2-9).

Using direct manipulation, first select the object within the Event Viewer, then

drag and drop

it at the desired location. Roll, pitch and yaw manipulators allow rotation of the human or vehicle about its coordinate axis system to provide the proper orientation (see Figures 2-9 and 2-10 for vehicles and humans, respectively).

If the actual coordinates and orientation of the selected human or vehicle are known, it is easiest to simply enter them directly in the Position/Velocity dialog.

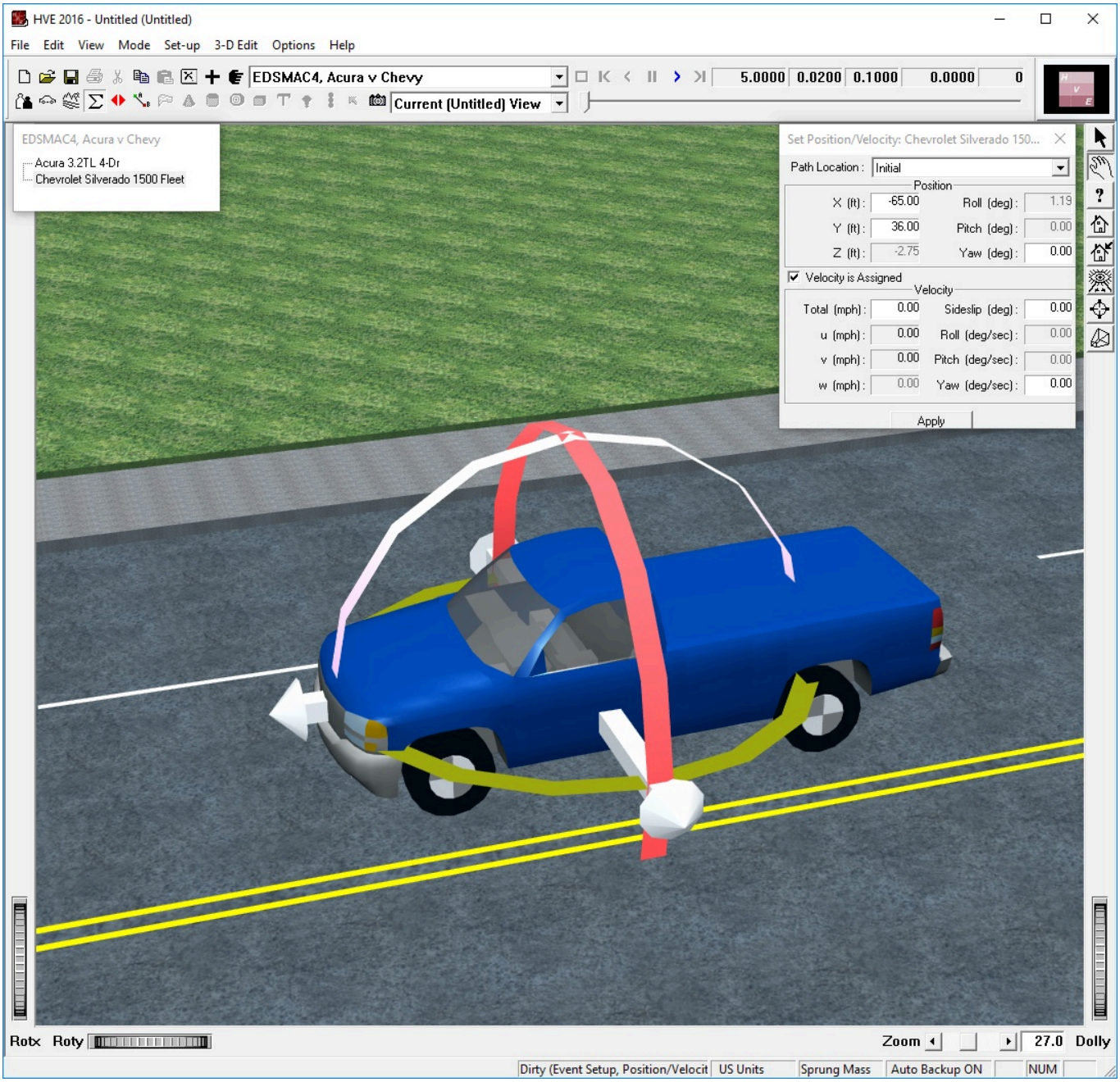


Figure 2-9: Vehicles are positioned using the translation and rotation manipulators, or by entering data directly into the Position/Velocity dialog.

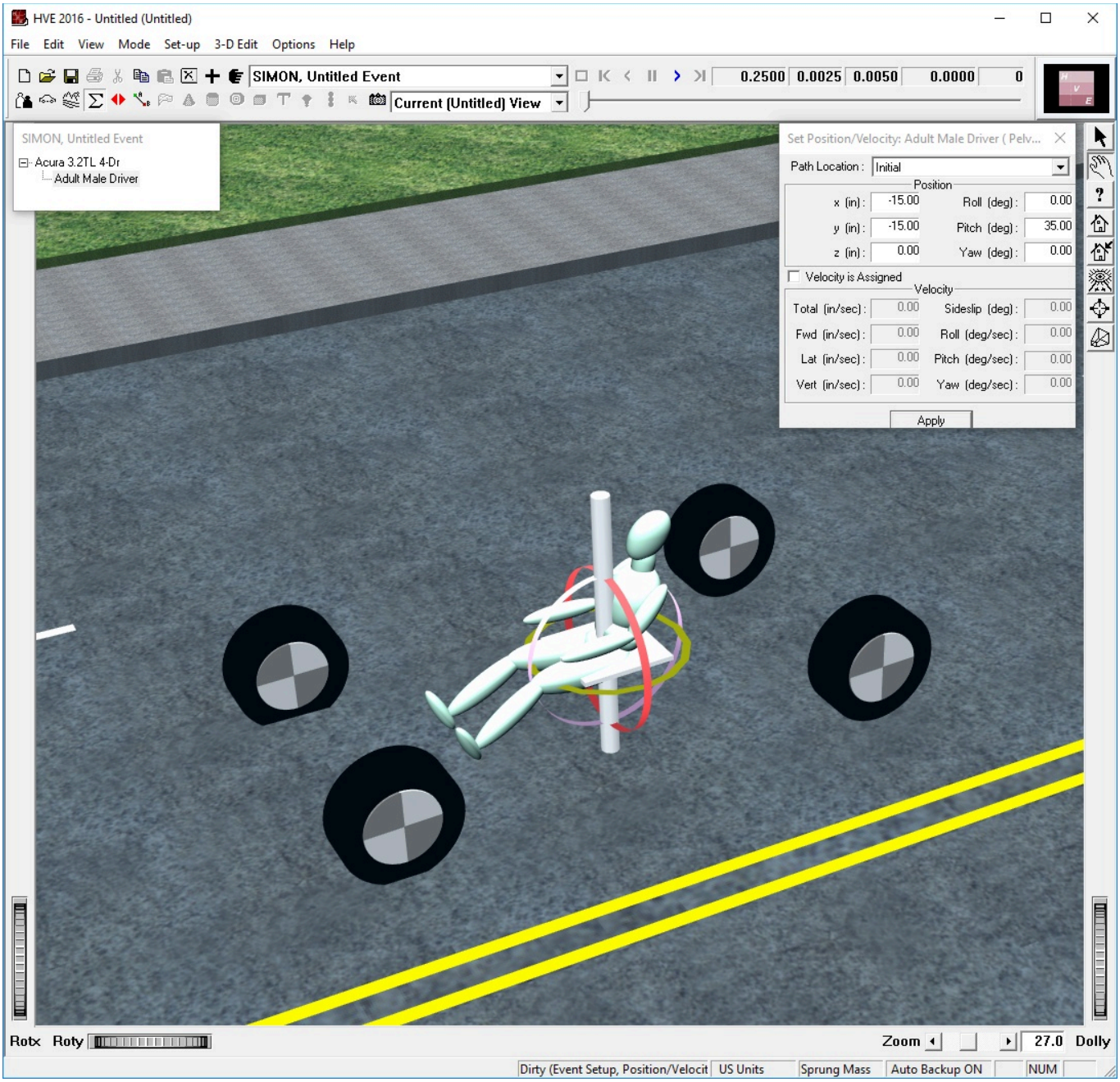


Figure 2-10: Humans are positioned, like vehicles, using the translation and rotation manipulators, or by entering data directly into the Position/Velocity dialog.

> **NOTE:** For occupant simulations, the human is positioned relative to the vehicle. Therefore, the vehicle should be positioned first.

HVE allows the user to enter positions and velocities for the following path locations:

- Initial Position
- Begin Perception
- Begin Braking
- Impact

- Separation
- Point On Curve
- End Of Rotation
- Final/Rest

The positions and velocities provided by the user are dependent upon the requirements of the selected model. Simulations *require* only initial positions and velocities. Reconstructions normally *require* impact and rest positions.

See also: Position/Velocity dialog reference.

Target Positions

The user may enter human and vehicle positions even though they are not actually used by the simulation or reconstruction. These are called

target positions

, and appear as translucent objects. Target positions provide visual feedback to the user regarding the desired path of the human or vehicle. If a simulated object passes through the target, the user can tell very quickly it is on the correct path. By observing that the simulated object misses the target, the user can make the necessary adjustments and rerun the simulation.

Target positions are also used by the HVE Path Follower. In this case, the user-entered target positions are used to define a path. Using the HVE Path Follower, simulation models determine the driver steering, braking and throttle inputs required to follow the path. The HVE Path Follower is further described in Chapters 4 and 16 of the legacy manual.

Assigning Driver Controls

The HVE vehicle model is controlled by driver inputs. The available driver controls are:

- Throttle Table
- Brake Table
- Steering Table
- Gear Selection Table
- HVE Path Follower
- Wheel Data

Steering, Throttle, Brake, Path Follower, and Gear Selection driver controls are available for simulation models; only Wheel Data controls are available for reconstruction models.

To supply a vehicle's driver controls, you must first select a vehicle, then choose *Driver Controls* from the Set-up menu *(updated: the 2006 manual says "from the Edit menu" here, but driver controls are on the Set-up menu — Set-up, Driver Controls..., Ctrl+D)*. There are two ways to select a vehicle:

- Choosing from the list of vehicles in the Event Editor
- Clicking on the vehicle in the Event Viewer (this also displays its current Position/Velocity dialog)

> **NOTE:** Because trailers do not appear in the list of vehicles in the Event Editor, to apply Driver Controls to a trailer, you must click on the trailer.

After the vehicle is selected, use the Driver Controls option to choose the desired method. The various methods are described below.

(See also the Driver Controls reference.)

Throttle Table — The Throttle Table allows you to provide tractive effort at each drive wheel on the selected vehicle. Data are entered using a table. Three options are available:

- *Percent Wide-Open-Throttle* vs time
- *Percent Available Friction* vs time
- *Tractive Effort* vs time

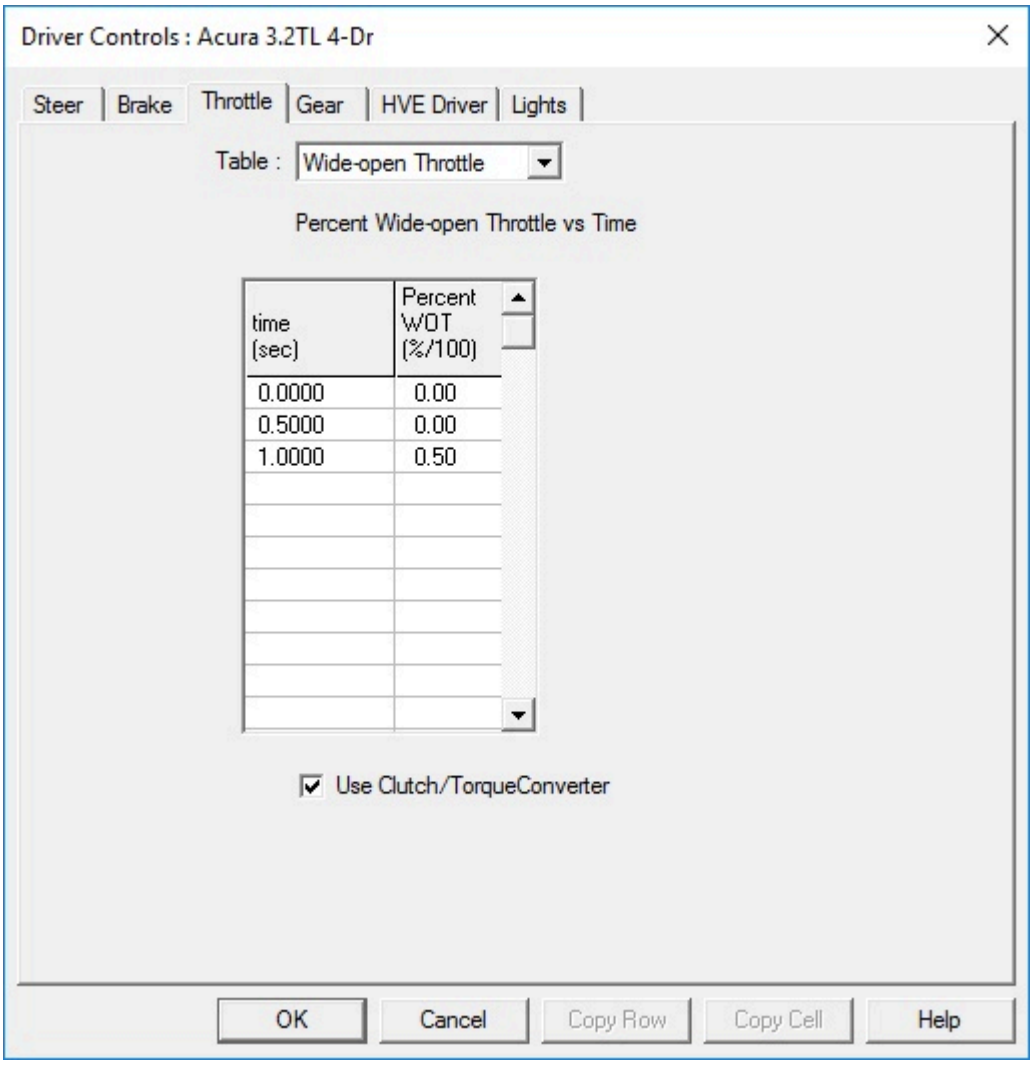


Figure 2-11: The Throttle Table allows the user to enter a table of tractive effort vs time. The Percent Wide-Open-Throttle option is shown.

The availability of these options is dependent upon the selected calculation model. For example, if the simulation is not capable of using the HVE Vehicle's engine model, the

Percent of Wide-open Throttle

option is not available.

Brake Table — The Brake Table allows you to provide longitudinal braking force at each wheel of the selected vehicle. Data are entered using a table. Three options are available:

- *Pedal Force* vs time
- *Percent Available Friction* vs time
- *Braking Force* vs time

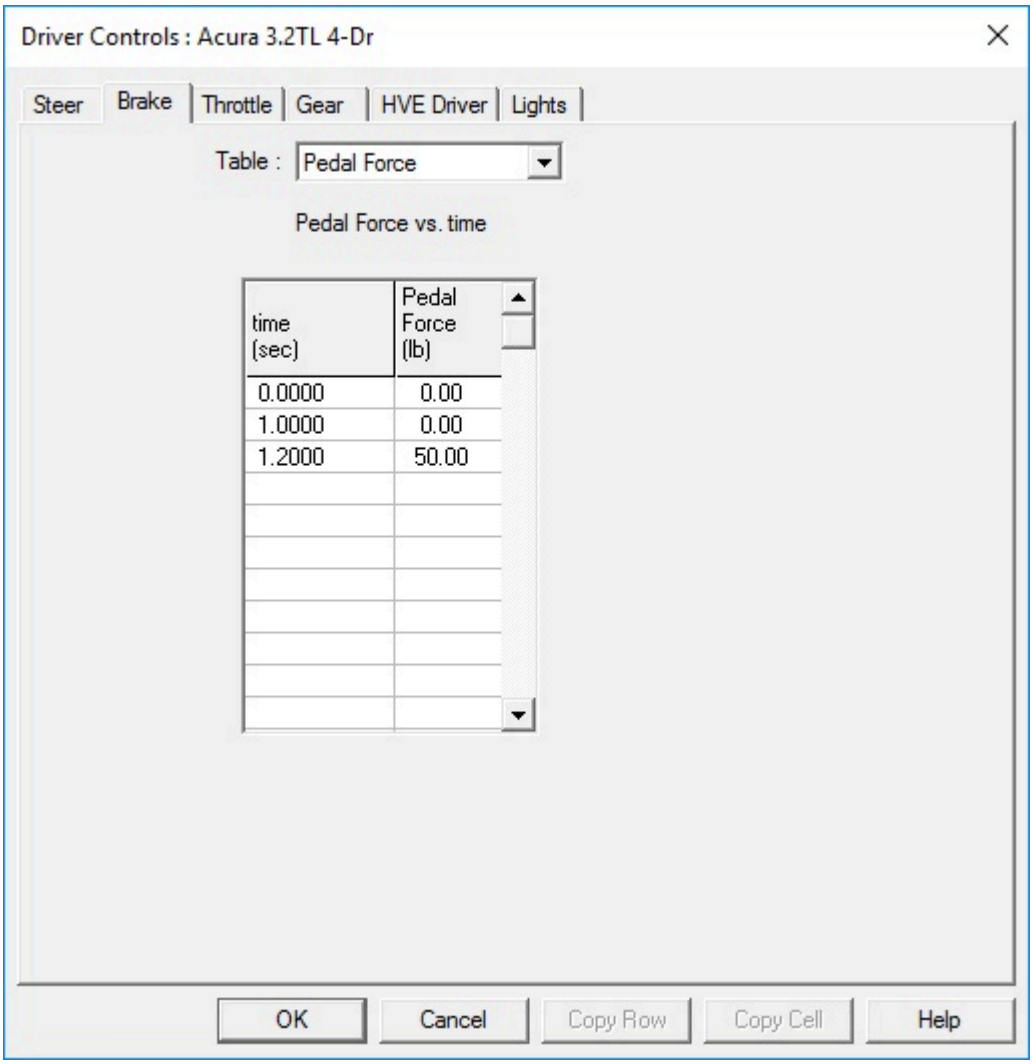


Figure 2-12: The Brake Table allows the user to enter a table of braking force vs time. The Pedal Force option is shown.

Again, the availability of these options is dependent upon the selected calculation model. For example, if the simulation is not capable of using the HVE Vehicle's brake system model, the

Pedal Force

option is not available.

Steer Table — The Steer Table allows you to provide steer angles at each steerable axle for the selected vehicle. Data are entered using a table. Two options are available:

- *Steering Wheel Angle* vs time
- *Tire Angle* vs time

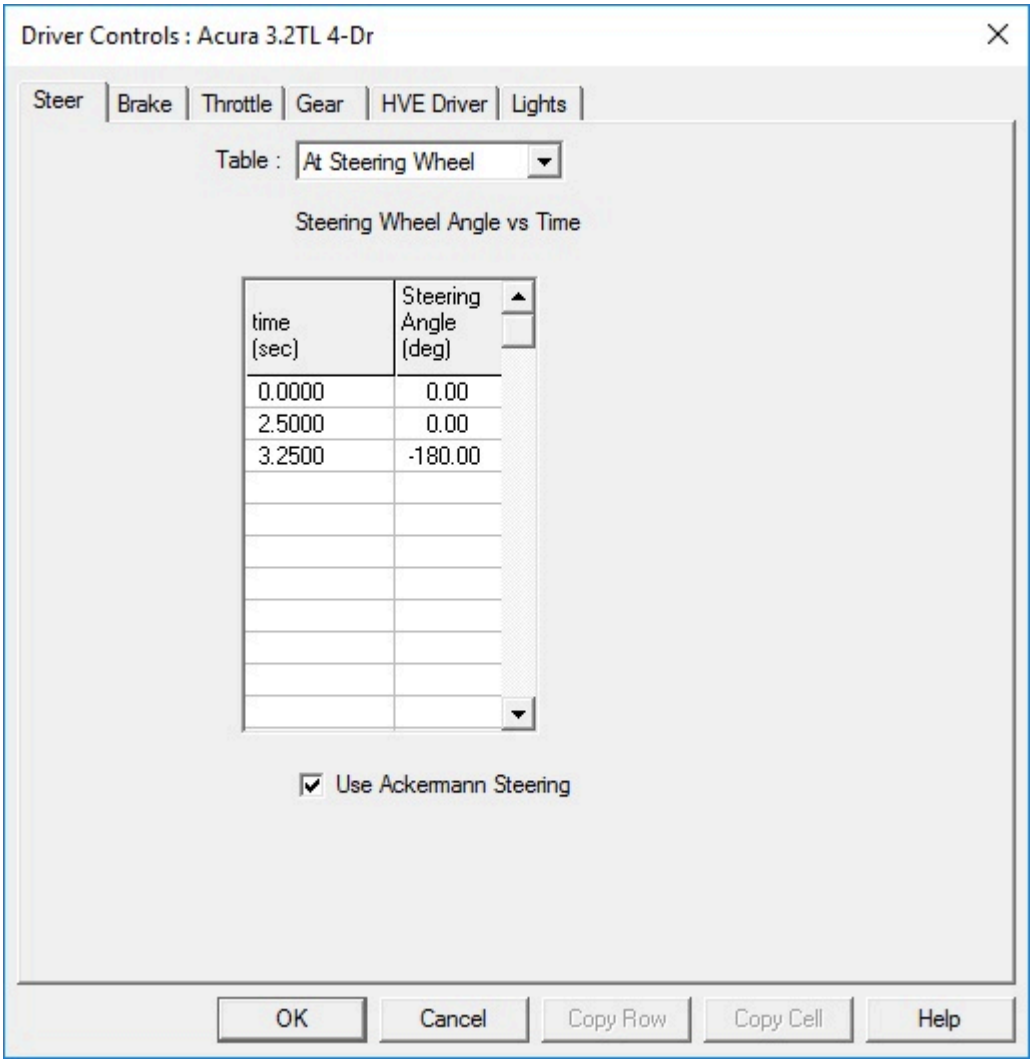


Figure 2-13: The Steer Table allows the user to enter a table of steer angles vs time for each steerable axle. The At Steering Wheel option is shown.

Gear Table — The Gear Table allows the user to provide the current gear selection for both the transmission and differential (differential gear selection only applies if the selected vehicle's differential has more than one gear ratio). Data are entered using a special table, as shown in Figure 2-14.

The Gear Table option is used for backing a vehicle, as well as changing its forward gear selection. The Gear Table option is available only if the current simulation model can use the gear information (some simulation models do not allow tractive effort in reverse). Thus, use of the Gear Table also requires that the simulation uses the HVE Engine Model.

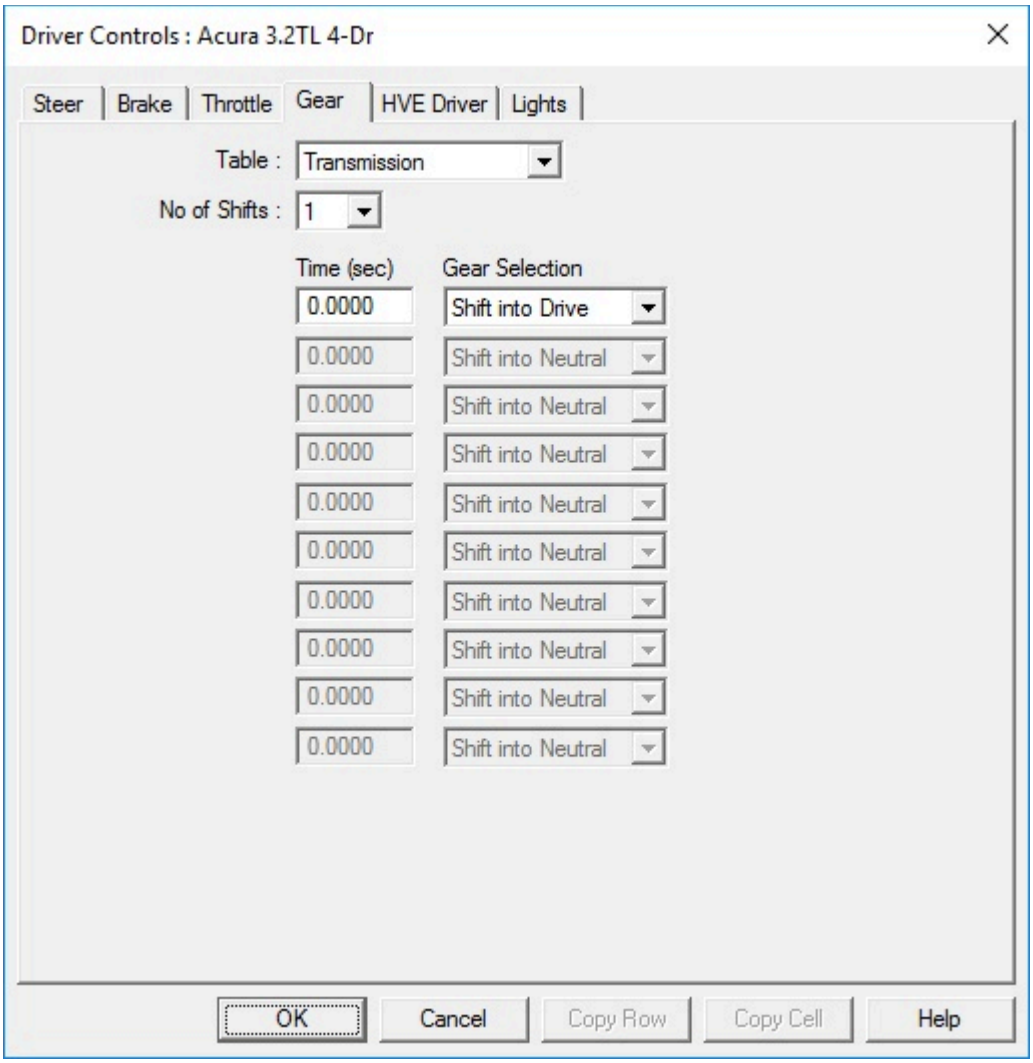


Figure 2-14: The Gear Table allows the user to enter a table of gear selections vs time. This table is only available for simulations which include an engine model, and requires that the current Throttle Table uses the Percent Wide-open Throttle method.

HVE Path Follower — The HVE Path Follower dialog allows the user to define a 3-D vehicle path using target positions. Using this path, the simulation model determines the steering, throttle and braking driver inputs required to cause the vehicle to follow the path.

The HVE Path Follower dialog includes options for path following by *Required Steering* or *Required Torque*. If target velocities are entered, the *Speed Follower* option is enabled. Finally, a *Driver Filter* option is available that allows simulation of the effects of driver physical characteristics on the ability to follow a path.

The HVE Path Follower dialog is shown in Figure 2-15. A detailed description of the HVE Path Follower is provided in Chapter 4 (Event Set-up) and Chapter 16 (HVE Event Model) of the legacy manual.

> **NOTE:** The Torque Method and Speed Follower options were not enabled as of the 2006 manual.

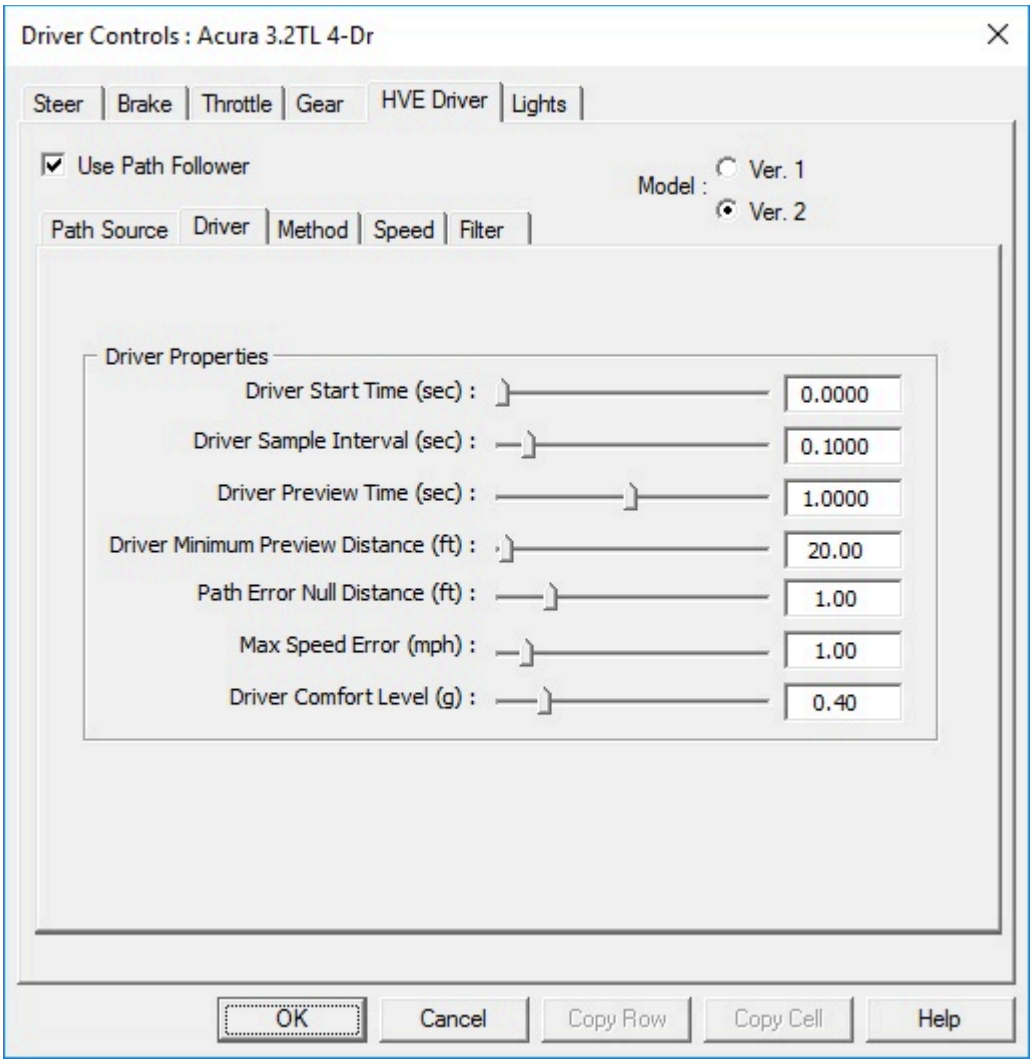


Figure 2-15: The HVE Driver tab of the Driver Controls dialog — Driver page (Driver Start Time, Driver Sample Interval, Driver Preview Time, Path Error Null Distance, Driver Comfort Level).

Wheel Data — The Wheel Data dialog is used for reconstruction models. It differs from the other driver control tables in that its values are fixed, rather than time-dependent, because reconstructions do not calculate in the time domain.

The Wheel Data dialog allows the user to enter the wheel lock-up (sometimes called *percent of available friction*) and steer angle for each wheel on the selected vehicle. You can also enter the total lock-up for pre-impact braking, and click on a check box to select a different calculation method for handling rotating and spinning vehicles between impact and rest (*Rot/Lat Skidding*). A typical Wheel Data dialog is shown in Figure 2-16.

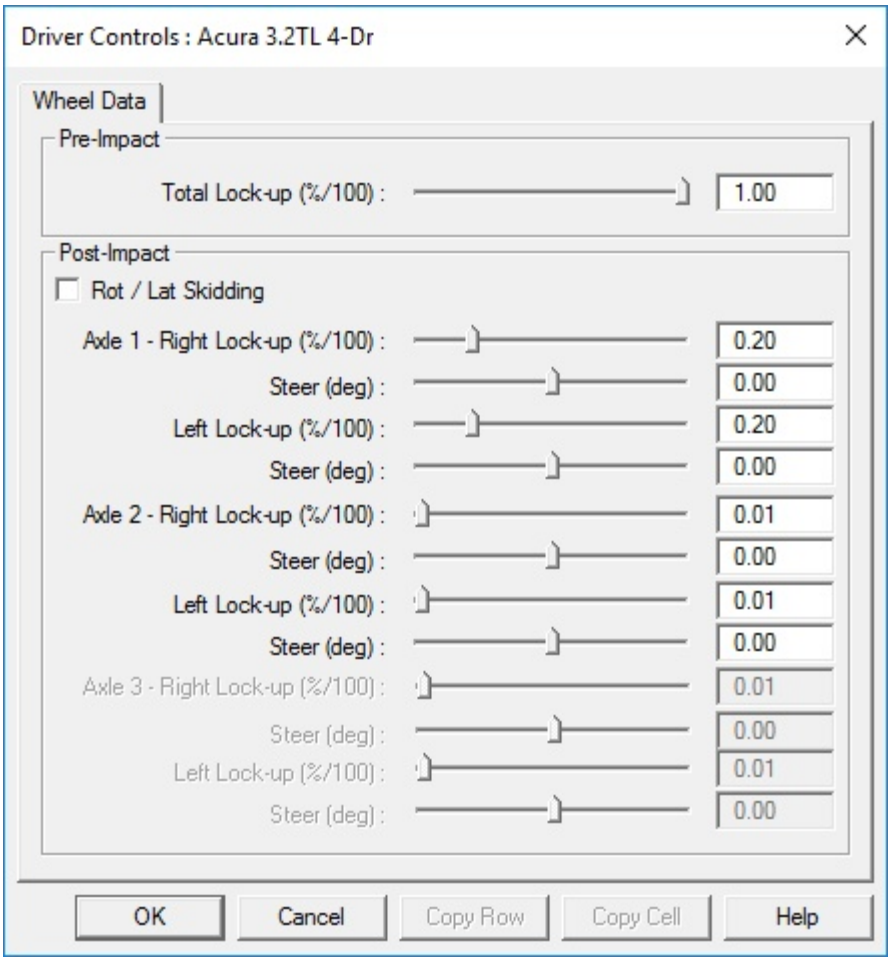


Figure 2-16: The Wheel Data dialog is used by reconstruction models that use constant wheel forces and steer angles.

Assigning a Vehicle Damage Profile

You can assign a damage profile to the selected vehicle. To supply a damage profile, you must first select a vehicle, then choose

Damage Profiles

from the Set-up menu. (This option will be unavailable if the selected reconstruction or simulation model does not allow user-entered damage profiles.) A typical Damage Profiles dialog is shown in Figure 2-17.

The damage profile defines the location and character of vehicle damage by allowing the user to specify:

- the width of the damaged surface
- the depth of crush at up to 10 equally-spaced intervals along the damage width
- the location of the center of damage relative to the vehicle's CG
- the principal direction of force (PDOF)

This information is initially provided in the form of a Collision Deformation Classification, or CDC (see SAE J224 [6.3]) which HVE uses to assign the default damage profile. The user can then over-ride the default values by replacing the assigned data (see Figure 2-17).

HVE uses the damage profile to calculate and display the Equivalent Energy Speed, or EES [2.25, 6.4]. The default value is set equal to the Total Delta-V (see below). The user may over-ride the default EES by clicking on the

EES

radio button and entering the desired value.

Damage Results

The Damage Data dialog uses the damage profile and stiffness coefficients to calculate the following results:

- **Delta-V** — Change in velocity during the collision. Total, forward and lateral components are displayed.
- **Damage Energy** — Unconserved energy during the collision.
- **Peak Force** — The peak force during the collision.

> **NOTE:** The Damage Profiles dialog calculates the delta-V for a vehicle striking a barrier, **not** another vehicle! To determine the delta-V during impact with another vehicle requires information about the other vehicle. Use EDCRASH or a similar program for such an analysis.
>
> **NOTE:** Actually, EDCRASH performs the above analysis on both vehicles, then apportions the delta-V according to the total damage energy and respective vehicle masses.

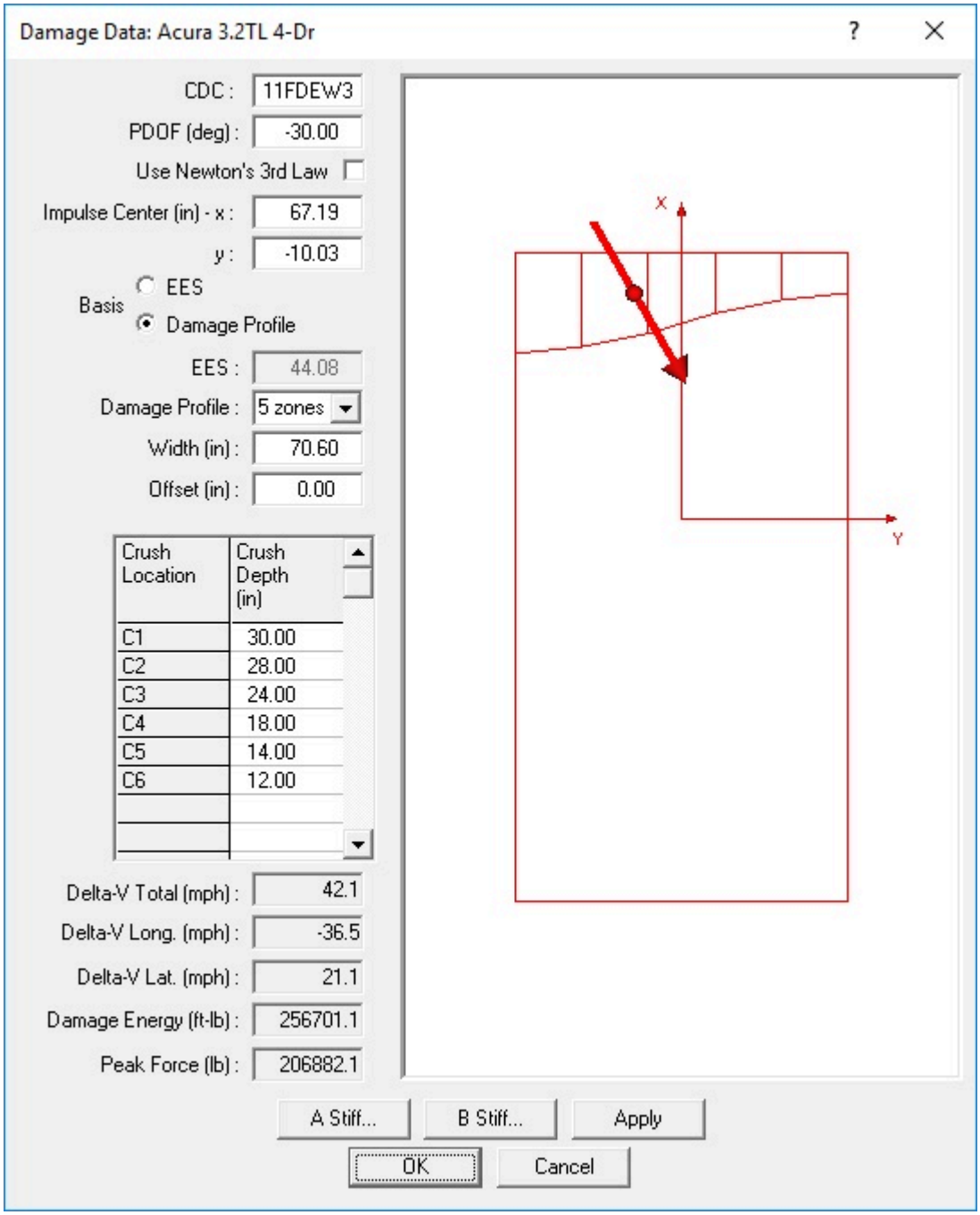


Figure 2-17: The Damage Data dialog — CDC, PDOF, Use Newton's 3rd Law, Impulse Center, EES/Damage Profile basis, Damage Profile zones, Width, Offset, crush locations and depths, Delta-V Total/Long./Lat., Damage Energy, Peak Force, A Stiff... and B Stiff... buttons.

See also: Damage Data reference.

Stiffness Coefficients

The selected vehicle's stiffness coefficients are normally used for damage-based calculation methods. Choosing *A Stiffness* or *B Stiffness* displays a dialog showing the current stiffness values for each crush zone. These coefficients are initially assigned by the Vehicle Editor when the vehicle is created. These original coefficients may be replaced by user-entered values for each crush zone (see Figure 2-18).

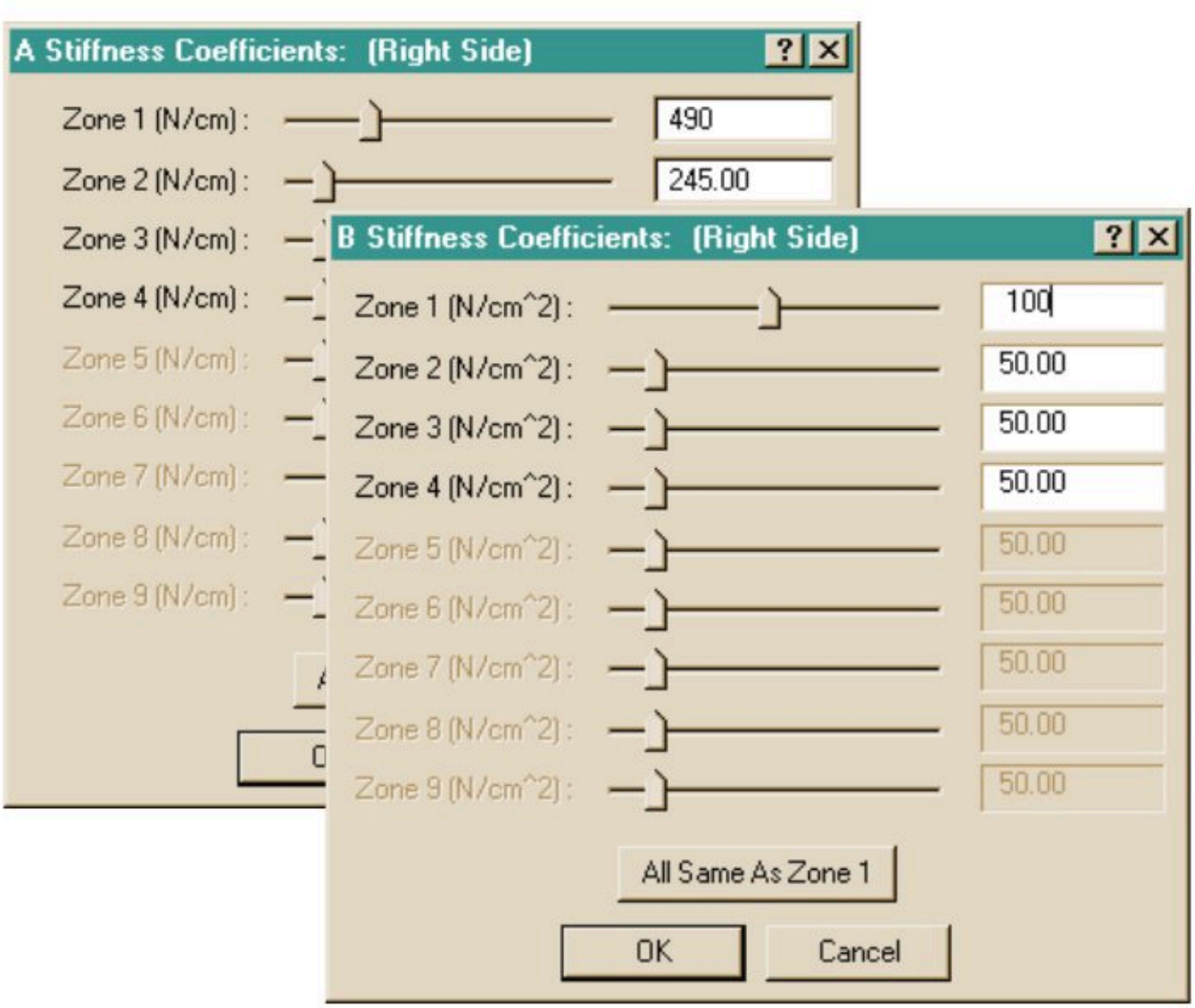


Figure 2-18: A and B Stiffness Coefficients dialogs allow the user to enter stiffness coefficients for the selected vehicle, per crush zone, with an "All Same As Zone 1" shortcut.

See also: A Stiffness and B Stiffness dialog references.

Assigning a Collision Pulse

A collision pulse is the motion vs time history during impact that defines the vehicle's motion during impact. If the current simulation model is an occupant simulation, the user must supply a collision pulse to the selected vehicle. Also, some vehicle simulations can include a collision pulse.

To supply a collision pulse, first select a vehicle, then choose

Collision Pulse

from the Set-up menu. (This option will be unavailable if the selected reconstruction or simulation model does not use a collision pulse.)

The collision pulse is provided in the form of a table of linear and angular motion inputs (e.g., accelerations) at specified times (see Figure 2-19). Because the collision pulse normally lasts 100 to 200 milliseconds, the timestep is usually very

small, on the order of 0.001 to 0.01 seconds.

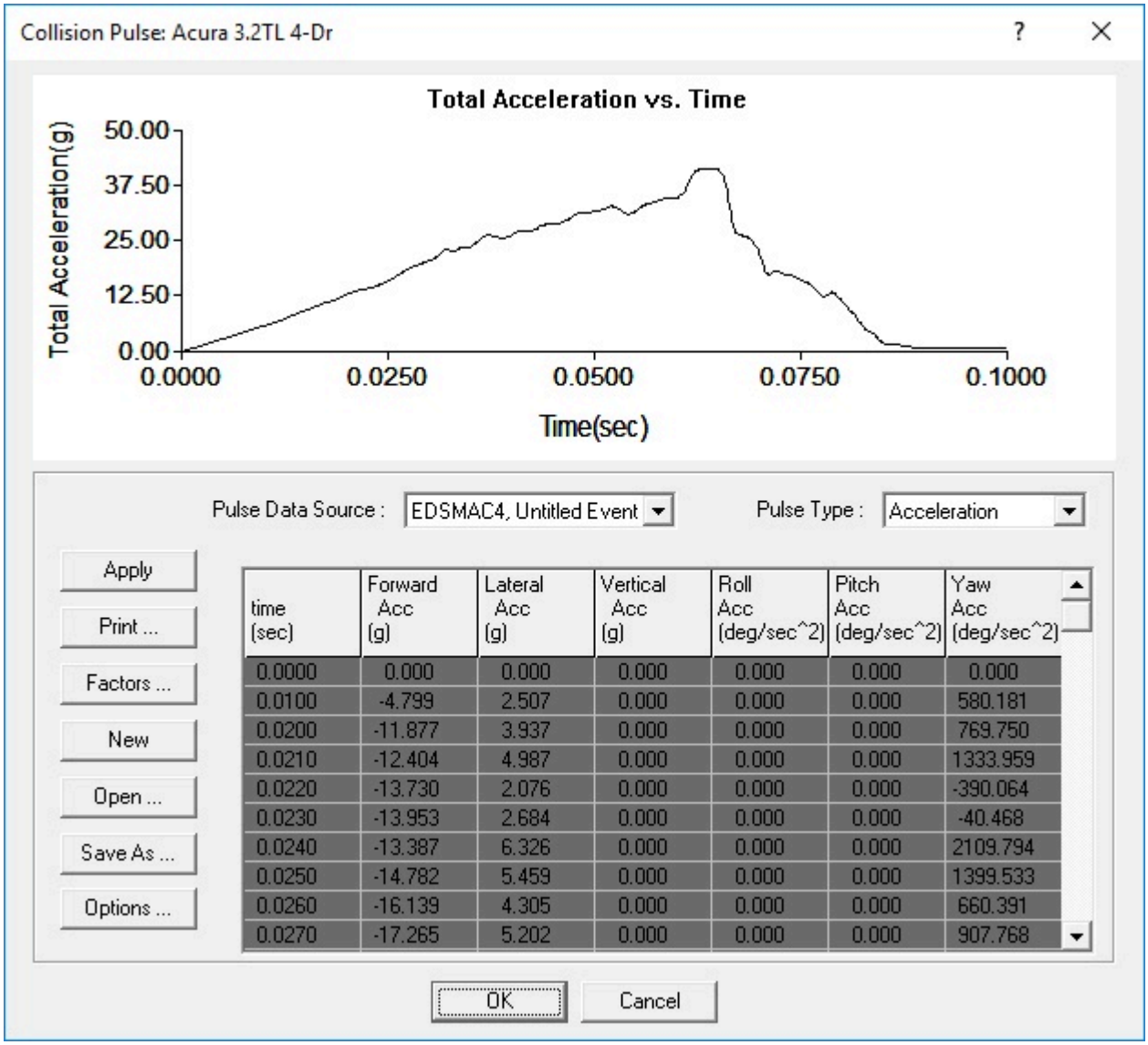


Figure 2-19: The Collision Pulse dialog — Total Acceleration vs. Time graph, Pulse Data Source and Pulse Type option lists, table of Forward/Lateral/Vertical/Roll/Pitch/Yaw accelerations vs time, with Apply, Print, Factors, New, Open, Save As and Options buttons.

Pulse Data Source

The collision pulse data may be assigned in a user-entered table or the data may be transferred automatically from a previously run event (e.g., EDSMAC4). The user determines the source of the collision pulse data using the Pulse Data Source option list.

User-entered Collision Pulse — A user-entered collision pulse is defined by direct table entry of time and associated accelerations. Collision pulses are actually stored in a file; the default collision pulse filename is 'Untitled'. The current table may be saved and opened for use in another event or case. Save and Open buttons in the Collision Pulse dialog are used for this purpose.

Event-defined Collision Pulse — HVE keeps track of all previously executed events that produced a collision pulse. If the selected vehicle was used in one or more of those events, those event names will be accessible by clicking the *Pulse*

Data Source option list. By selecting one of these events, the collision pulse table is automatically loaded using the selected event's output tracks. The table uses the output timestep from the source event.

> **NOTE:** HVE will display a warning message if the timestep assigned by the event is too large.

If the source event (e.g., EDSMAC4) is changed and rerun, the data it supplies to the collision pulse table will be different. Therefore, if the user reruns the occupant simulation the results will be different because they reflect the changes to the source event.

Pulse Type

Four types of collision pulses are allowed:

- Position vs Time
- Velocity vs Time
- Acceleration vs Time
- Force/Moment vs Time

The current calculation method determines which, if any, methods are selectable.

Collision Pulse Factors

The individual linear and angular components of the collision pulse may be increased or decreased using the Pulse Factors dialog (see Figure 2-20). The Collision Pulse Factors provide a convenient way to assess changes in the vehicle and/or occupant response due to an increase or decrease in collision force components.

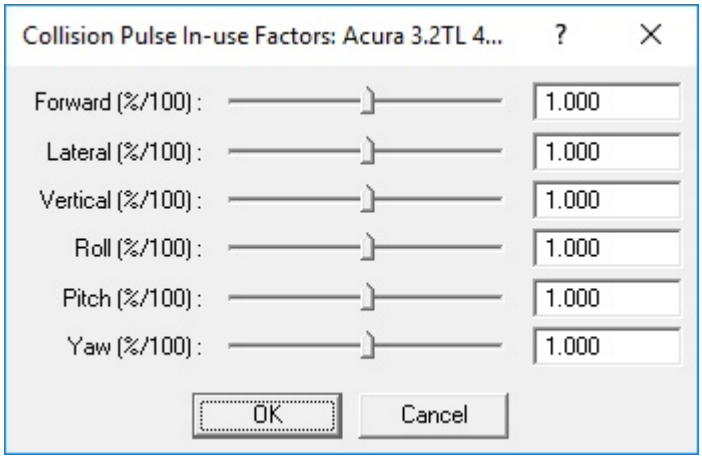


Figure 2-20: The Collision Pulse In-use Factors dialog — Forward, Lateral, Vertical, Roll, Pitch and Yaw amplification factors.

Collision Pulse Options

The Collision Pulse Options dialog allows the user to focus on a particular portion of the collision pulse. This feature is useful because a rollover simulation may require the entire pulse, while a collision simulation only requires that portion during which the acceleration is greater than some threshold acceleration (normally 1 g). The pulse is edited by supplying a threshold acceleration and a starting time that specifies that portion of interest for the entire pulse.

If the current pulse type is

Force/Moment

, the user can specify the vehicle-fixed coordinates for the point of application of the force on the vehicle.

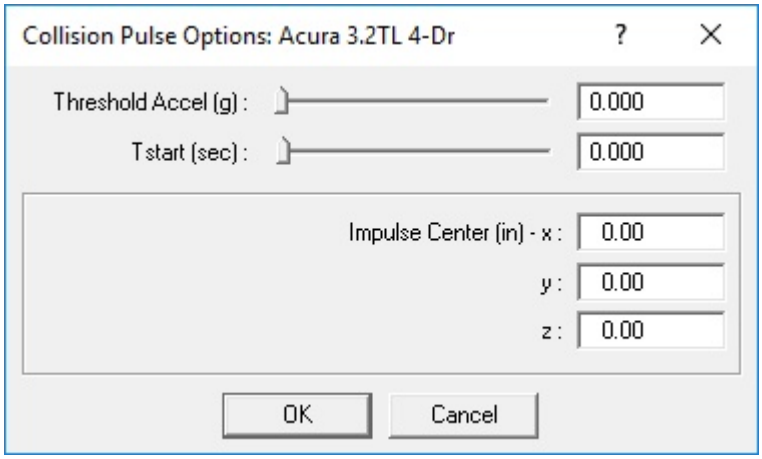


Figure 2-21: The Collision Pulse Options dialog — Threshold Accel, Tstart, Impulse Center coordinates.

See also: Collision Pulse, Pulse Factors and Pulse Options references.

Vehicle Mesh

The Mesh dialog is used for changing the attributes of the mesh (3-D geometry) for the selected vehicle. Vertices within the Weld Distance share the same vertex coordinates. Clicking on *Tessellate* and then entering a *Maximum Side Length* ensures that no side of any triangle is longer than the entered value. Identifying the *watertightness* provides quality information about the mesh. These options are useful for *DyMESH* and other simulation models employing a deformable mesh.

The Mesh dialog is also used for assigning *Inter-vehicle Friction* and *Restitution* (this option takes on various forms, depending on the selected physics model).

To assign Mesh data, first select a vehicle, then choose

Vehicle Mesh

from the Set-up menu (this option is unavailable if the current reconstruction or simulation model does not use the Mesh Options dialog). The Mesh dialog will be displayed as shown in Figure 2-22.

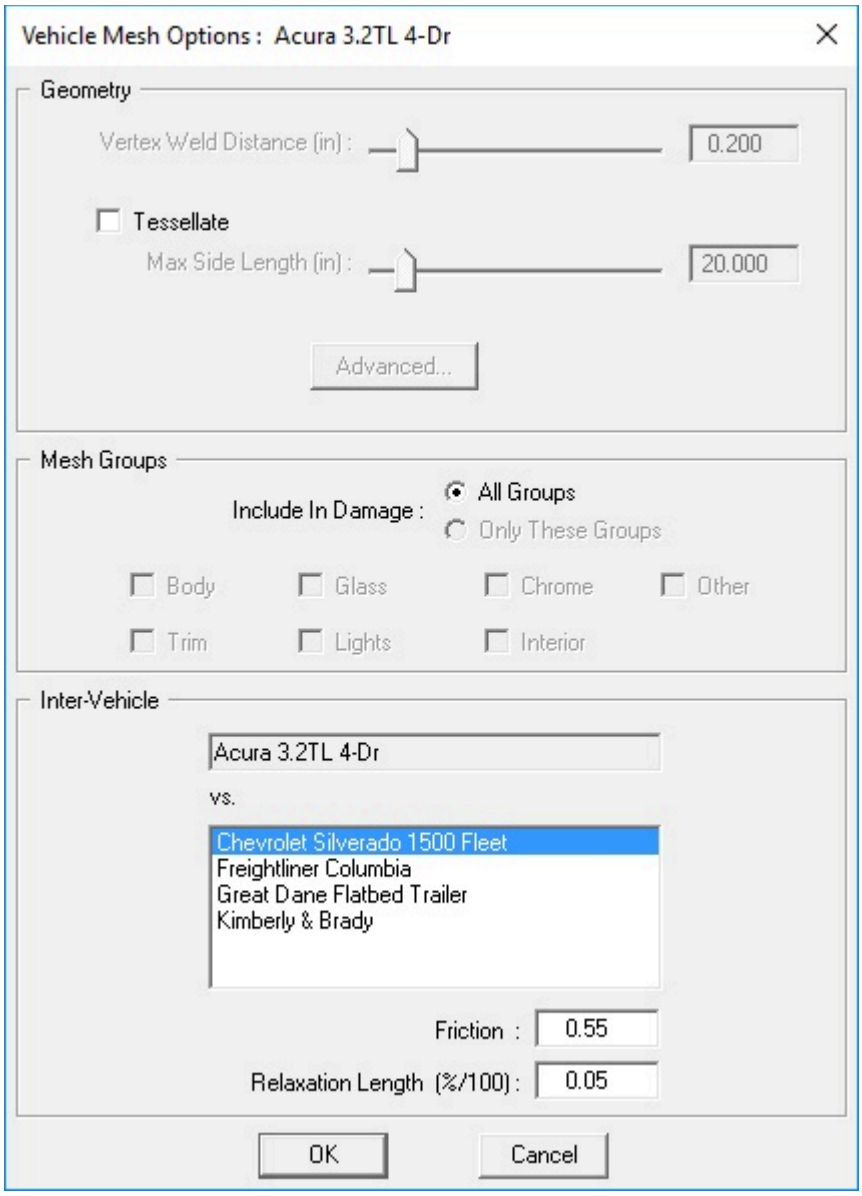


Figure 2-22: The Vehicle Mesh Options dialog — Vertex Weld Distance, Tessellate/Max Side Length, Is Mesh Water Tight? (Yes/No/Unknown), Mesh Groups included in damage (Body, Glass, Chrome, Trim, Lights, Interior, Other), and Inter-Vehicle friction/relaxation length vs each other object.

Assigning Payloads

You can assign a payload to the selected vehicle. To supply a payload, you must first select a vehicle, then choose *Payload* from the Set-up menu. See Figure 2-23. (This option will be unavailable if the selected reconstruction or simulation model does not allow vehicle payloads.)

The addition of a payload causes HVE to update the earth-fixed position of the vehicle's center of gravity. In addition, the physics model may also automatically alter the inertial properties and weight distribution of the selected vehicle.

> **NOTE:** This behavior is physics model-dependent.

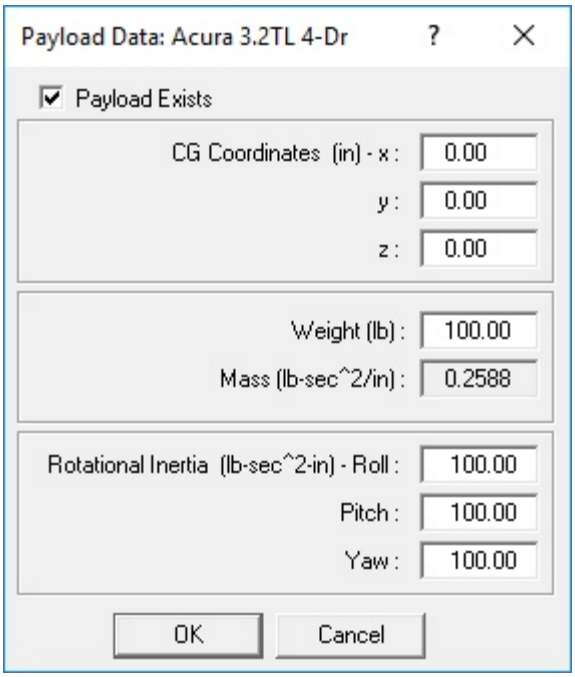


Figure 2-23: The Payload Data dialog — Payload Exists, CG Coordinates (x,y,z), Weight, Mass, Rotational Inertia (Roll, Pitch, Yaw).

See also: Payload dialog reference.

Wheel Data Options (Set-up, Wheels)

The Wheels dialog (Set-up menu, *Wheels...*) contains four tabbed pages: Blow-out, Damage, Brake and Tire-Terrain. *(See also the Wheels dialog reference.)*

HVE Tire Blow-out Model

The selected vehicle may incur a tire blow-out during an event. The

HVE Tire Blow-out Model™

allows the user to study the transient effects of a loss of inflation pressure on vehicle handling and behavior during an event.

The Tire Blow-out dialog (see Figure 2-24) allows the user to select one or more tire locations and specify the time and duration of the blow-out(s), as well as the various factors used for the tire blow-out simulation. Some simulations can determine when the blow-out is to occur; this feature is activated by clicking the *AutoStart* check box. This option replaces the user-entered *Start Time*.

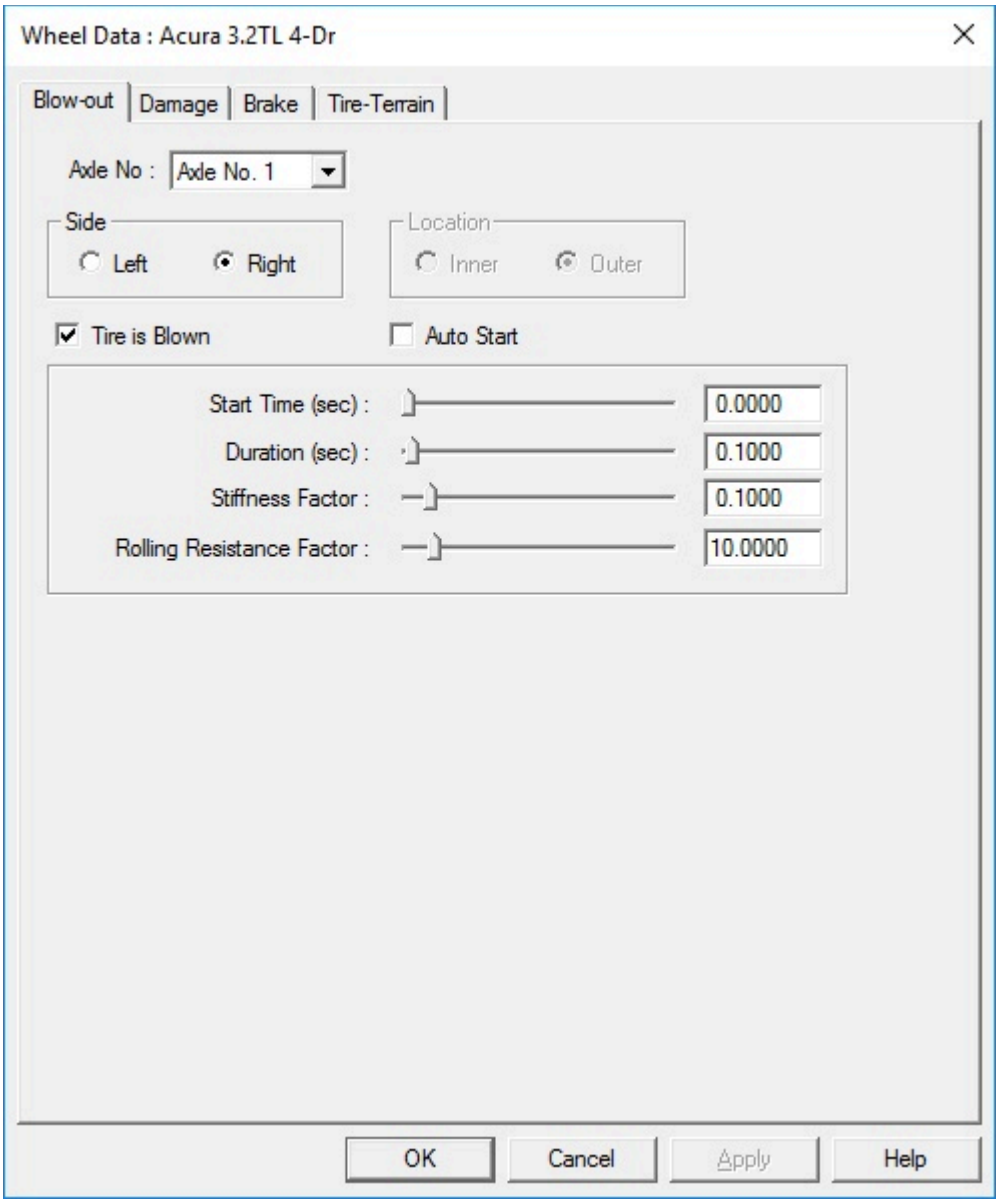


Figure 2-24: The Tire Blow-out dialog — Axle No, Side, Location (Inner/Outer), Tire is Blown, Auto Start, Start Time, Duration, Stiffness Factor, Rolling Resistance Factor.

Wheel Damage Option

The selected vehicle's wheel may be damaged during an event. A damaged wheel affects vehicle handling in several ways: it alters the location of the wheel (thus altering the tire moments and forces acting on the vehicle); it alters the camber and thus may result in different lateral force at a given wheel location; and the wheel may be partially or totally locked up due to damage.

The Wheel Damage dialog (see Figure 2-25) allows the user to select one or more wheel locations and specify the time and duration of the wheel damage(s), the wheel displacement (Δx, Δy, Δz), the change in camber for each wheel and the percentage of wheel lock-up for each wheel. Some simulations can determine when the damage is to occur; this feature is activated by clicking the *AutoStart* check box. This option replaces the user-entered *Start Time*.

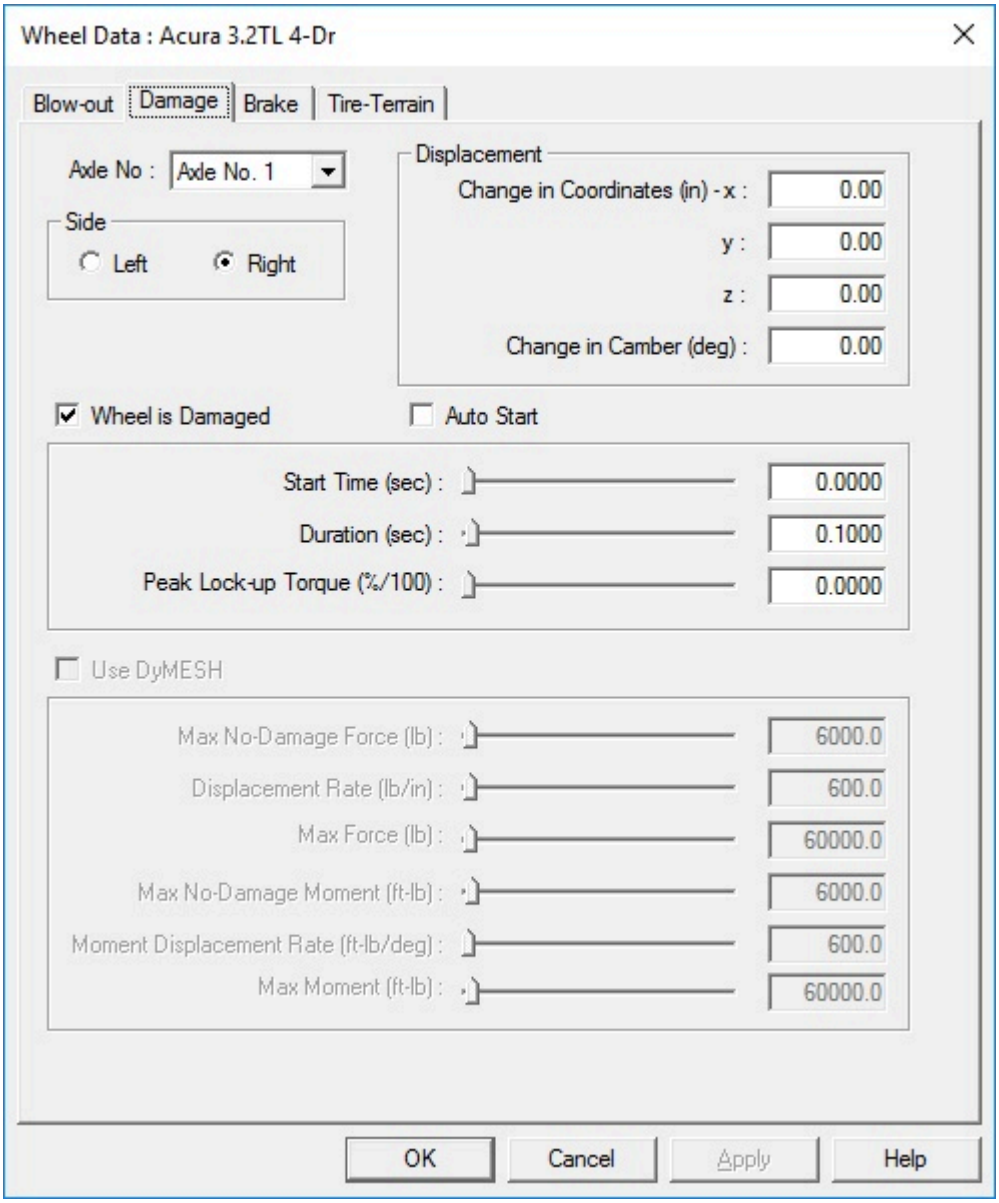


Figure 2-25: The Wheel Damage dialog — Axle No, Side, Displacement (change in coordinates x, y, z), Change in Camber, Wheel is Damaged, Auto Start, Start Time, Duration, Peak Lock-up Torque, Max No-Damage Force, Deformation Rate, Max Force.

Wheel Brake Option

The selected vehicle's wheel may have an improperly adjusted brake, or the brake may be completely failed due to a mechanical malfunction, such as leakage. These factors influence the brake torque at the selected wheel, and thus, the total forces and moments acting on the sprung mass.

The Wheel Brake dialog (see Figure 2-26) allows the user to select one or more wheel locations and specify the initial lining and drum/rotor temperatures, the initial stroke, and the time and duration of the brake failure. Some simulations can automatically assign the time at which the brake failure begins. This option is activated by clicking the *AutoStart* check box. This option replaces the user-entered *Start Time*.

The Wheel Brake dialog is particularly useful for studying the degradation in braking on long, down-hill grades.

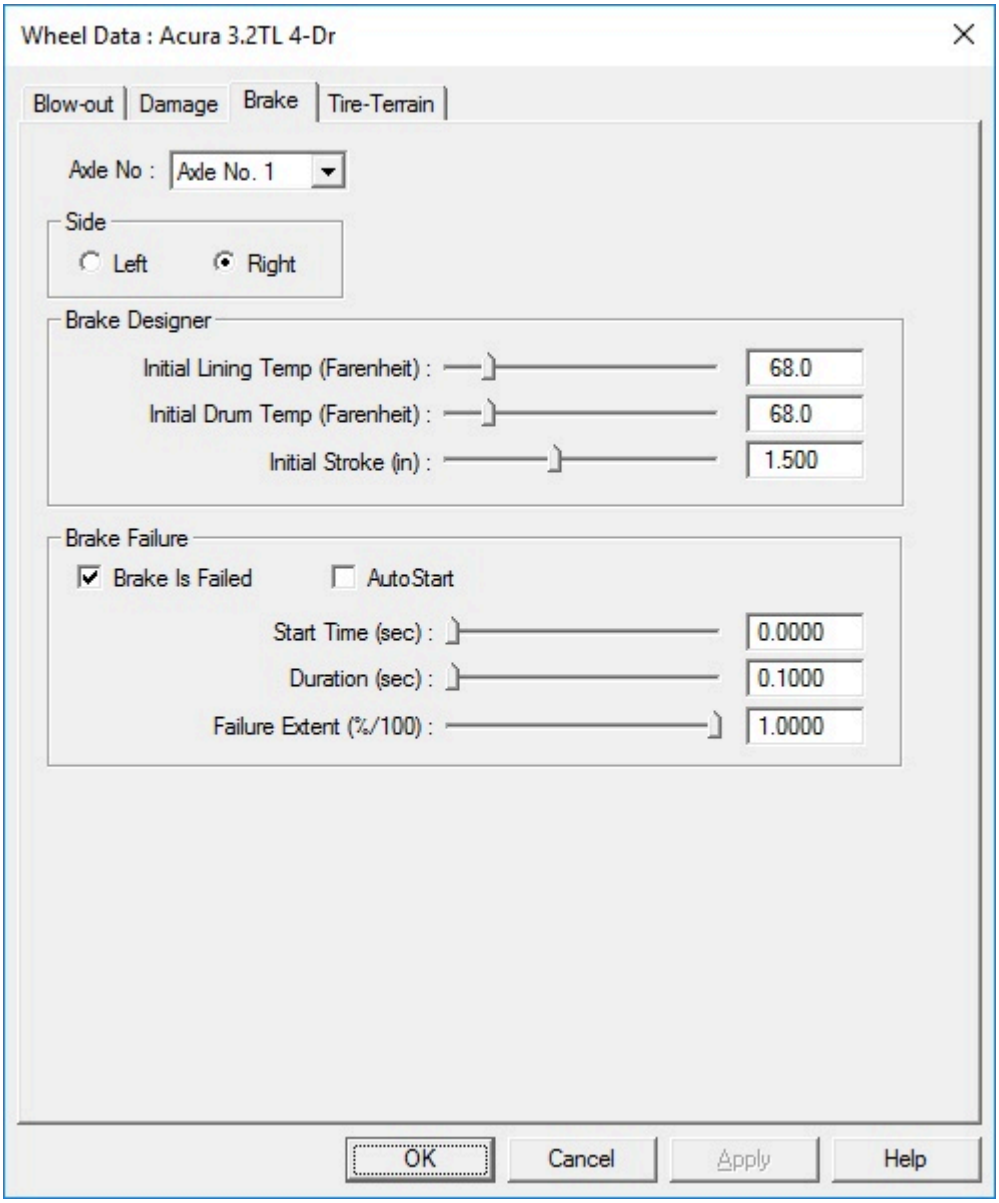


Figure 2-26: The Wheel Brakes dialog — Axle No, Side, Brake Designer (Initial Lining Temp, Initial Drum Temp, Initial Stroke), Brake Failure (Brake Is Failed, AutoStart, Start Time, Duration, Failure Extent).

Wheel Tire-Terrain Option

The selected vehicle may travel over irregular terrain (e.g., a pot hole or curb) or soft soil during an event. The Tire-Terrain dialog allows the user to select from the following tire-terrain models:

- Point Contact Model
- Radial Spring Model
- Sidewall Impact Model
- Soft Soil Model

The Wheel Tire-Terrain dialog (see Figure 2-27) allows the user to select one or more wheel locations and then select the appropriate model for the terrain conditions. The dialog also provides access to the parameters used by each of the

models. Note that different wheels may use different models. For example, the right front tire might use the Sidewall Impact model, while the left front tire uses the Radial Spring model.

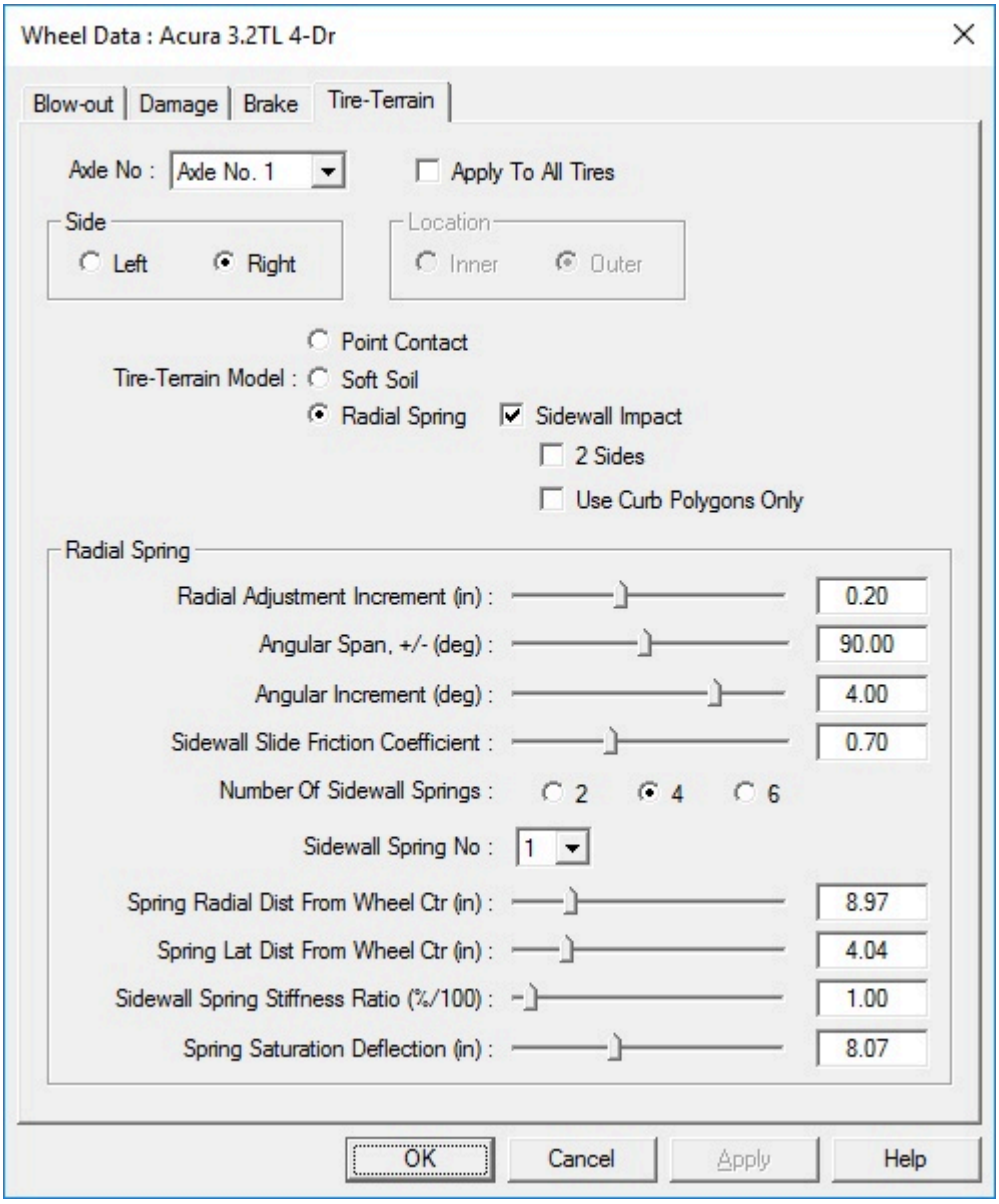


Figure 2-27: The Wheel Tire-Terrain dialog — Axle No, Side, Location, Tire-Terrain Model (Point Contact / Soft Soil / Radial Spring, with Sidewall Impact check box), Radial Spring parameters (Radial Adjustment Increment, Angular Span, Angular Increment, Sidewall Slide Friction Coefficient, Number of Sidewall Springs, per-spring distances and stiffness ratios).

Accelerometers

The Accelerometers dialog (see Figure 2-28) allows the user to select up to five vehicle-fixed locations for the placement of accelerometers. These accelerometers allow the user to study the acceleration and velocity change at the specified location. Such studies are useful for determining the collision severity experienced by occupants seated in various positions within the vehicle. Accelerometers are also useful for determining the threshold for airbag deployment for specific airbag sensor locations.

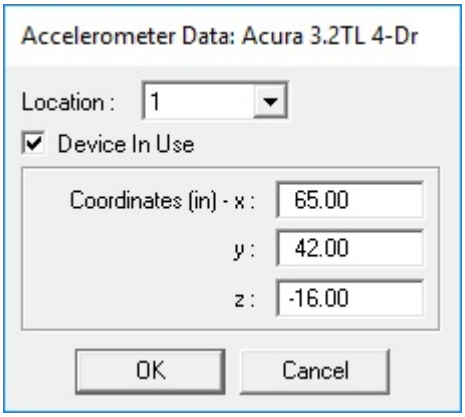


Figure 2-28: The Accelerometer Data dialog — Location (1-5), Device In Use, Coordinates (x, y, z).

See also: Accelerometers dialog reference.

Assigning Restraint System Usage

The selected vehicle may have belt and/or airbag restraint systems. However, as we all know, just because the vehicle has restraint systems doesn't necessarily mean they are being used!

If the current simulation model is an occupant simulation, you can supply in-use parameters for belt and airbag restraint systems for the selected vehicle. To supply restraint system in-use parameters, you must first select a vehicle, then choose *Restraints* from the Set-up menu *(updated: the 2006 manual says "from the Edit menu"; the current Set-up menu provides a Restraints... submenu with separate Airbags... and Belts... items)*. (This option will be unavailable if the selected simulation model does not model restraint systems.)

Using the Restraints dialog, you can select from the available restraints and supply appropriate parameters (see Figure 2-29). An individual system (lap belt, shoulder belt or airbag) is available only if the selected vehicle has that system installed at the occupant's seat position.

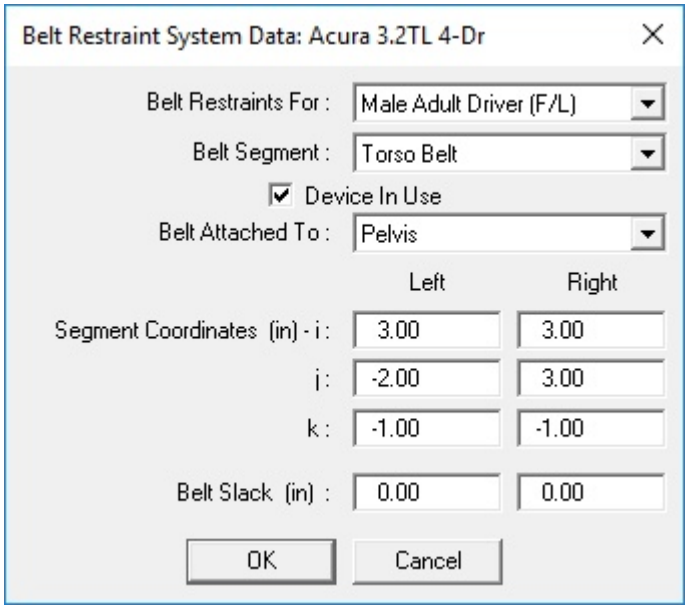


Figure 2-29: The Belt Restraint System Data dialog — Belt Restraints For (occupant), Belt Segment, Device In Use, Belt Attached To, left/right segment coordinates (i, j, k), Belt Slack.

See also: Belt Restraint Systems and Airbag Systems references.

Assigning Contacts

The human model is composed of 15 segments; each segment has up to three contact ellipsoids. The vehicle model has up to 25 contact surface planes. Occupant and pedestrian simulations work by calculating the force between each of these ellipsoids and surface planes at each timestep. It follows that up to 1,125 (15x3x25) sets of force calculations must be performed. Because these force calculations are rather lengthy, execution may be extremely slow.

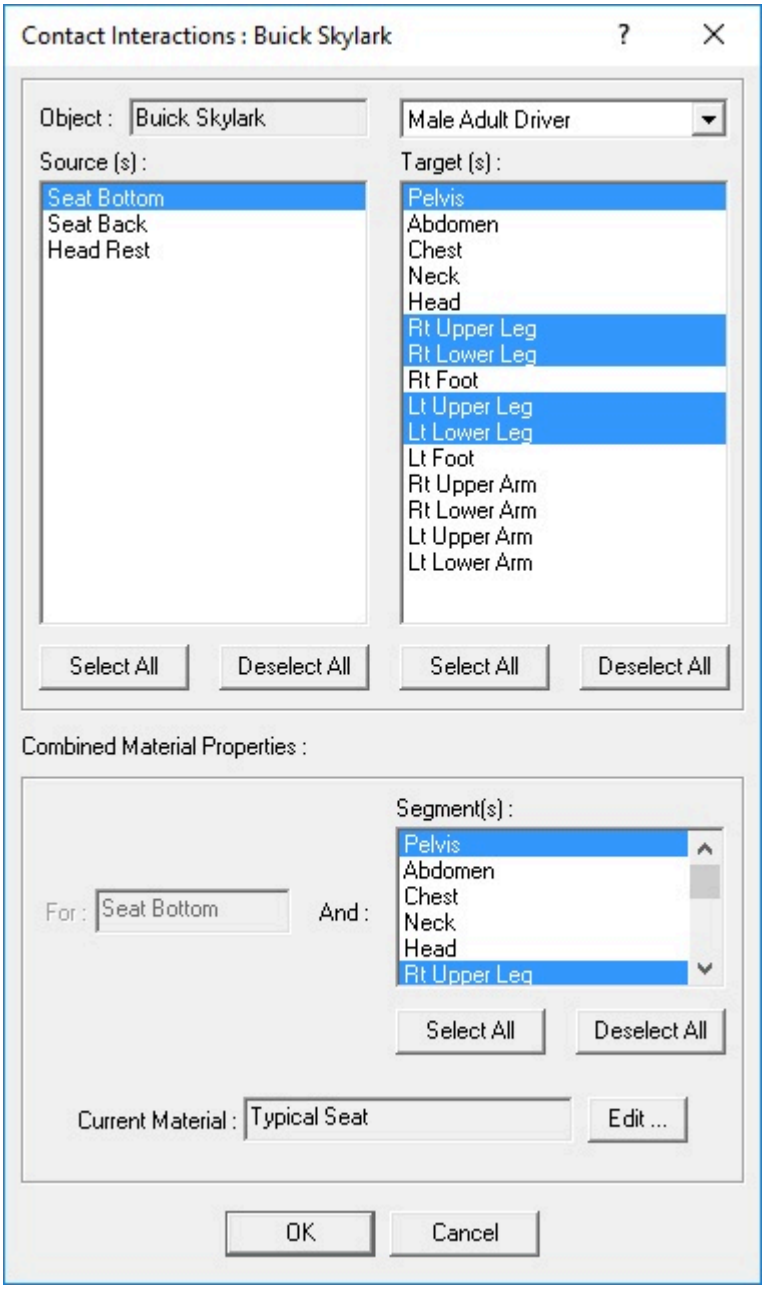
If it is clear that certain ellipsoid-surface pairs will never come into contact (e.g., interior surface planes during a pedestrian simulation), it makes sense to tell the simulation model to ignore the contact force calculations for these ellipsoid-surface pairs. The Contacts dialog is used for this purpose. This option is only available for human simulation models.

To select and deselect ellipsoid-surface pairs, first select a vehicle, then choose

Contacts

from the Set-up menu.

The Contacts dialog displays two list boxes: the one on the left contains all the defined human ellipsoids; the one on the right contains all the defined vehicle contact surfaces. To allow or inhibit interaction between ellipsoid-surface pairs, select an ellipsoid then choose the contacts you wish to allow by either selecting or deselecting from the list of contacts (see Figure 2-30). If the current event is an occupant simulation, all the vehicle interior surfaces are initially selected; if the current event is a pedestrian simulation, all the exterior surfaces are selected.


Contact Interactions : Buick Skylark
Object : Buick Skylark
Male Adult Driver
Source (s) :
Seat Bottom
Seat Back
Head Rest
Target (s) :
Pelvis
Abdomen
Chest
Neck
Head
Rt Upper Leg
Rt Lower Leg
Rt Foot
Lt Upper Leg
Lt Lower Leg
Lt Foot
Rt Upper Arm
Rt Lower Arm
Lt Upper Arm
Lt Lower Arm
Select All
Deselect All
Select All
Deselect All
Combined Material Properties :
For : Seat Bottom
And :
Segment(s) :
Pelvis
Abdomen
Chest
Neck
Head
Rt Upper Leg
Select All
Deselect All
Current Material : Typical Seat
Edit ...
OK
Cancel

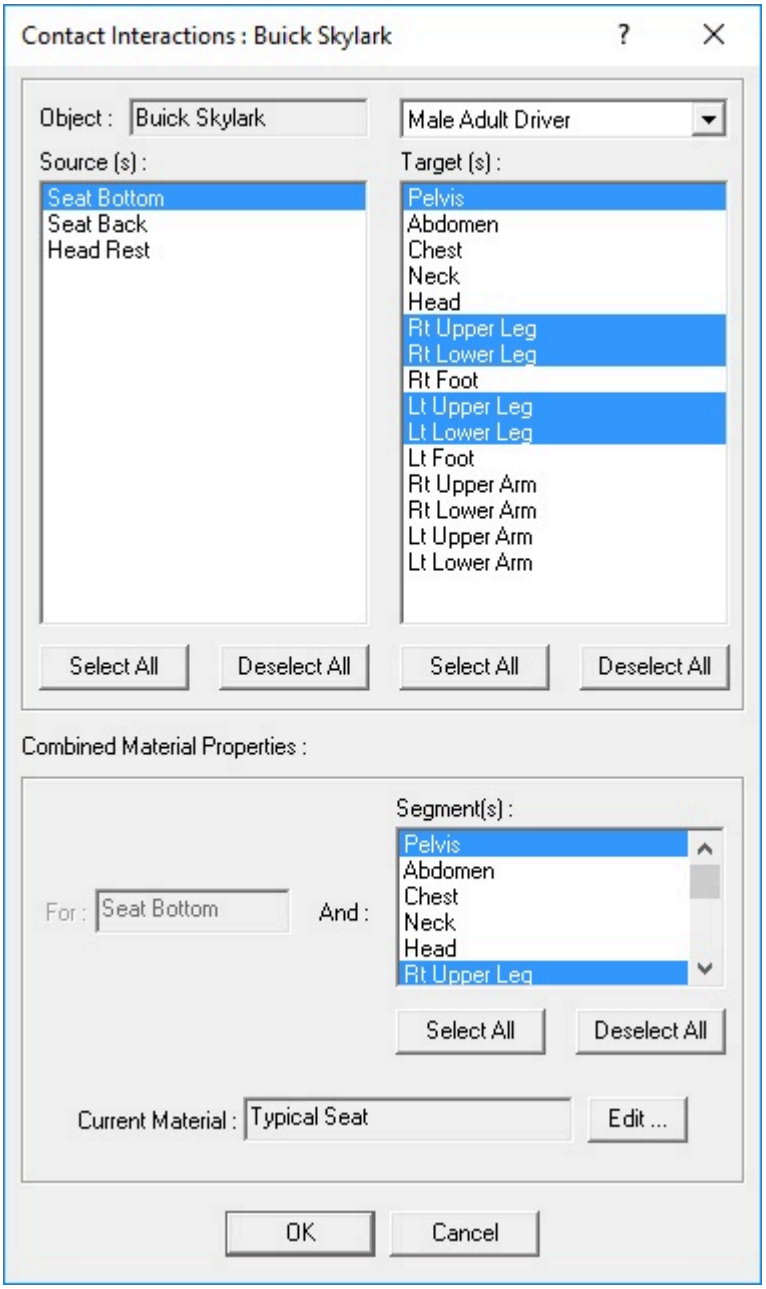


Figure 2-30: The Contact Interactions dialog — Object, Source(s), Target(s), Combined Material Properties, Current Material and Edit... button.

See also: Contacts dialog reference.

Executing Events

After an event has been set up, it is ready for execution. For simulation models, this means starting the simulation and watching the Event Viewer to study the interactions between the human(s), vehicle(s) and environment. The goal is to match the simulated trajectories and damage profiles with the measured trajectories and damage. For reconstruction models, this means using the measured path positions and damage profiles to estimate the initial velocities and velocity changes. In either case, event execution is controlled using the Event Controller.

Event Controller

The Event Controller is similar to a typical VCR controller in both form and function (see Figure 2-31).

The Event Controller has the following buttons:

- **Reset** — Reinitialize the calculation model for recomputation
- **Rewind** — Return to the start of the simulation
- **Reverse** — Run the simulation backwards
- **Pause** — Pause the simulation
- **Play** — Run the simulation forwards or execute a reconstruction model
- **Advance To End** — Advance to the end of the simulation

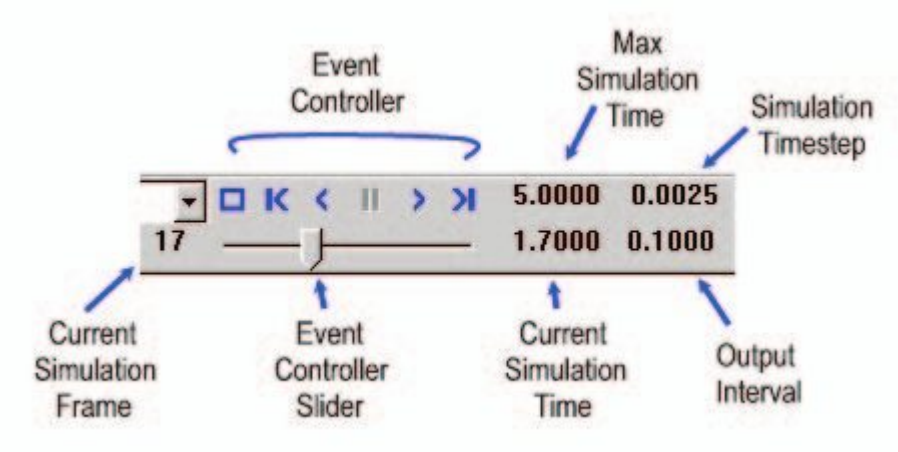


Figure 2-31: The Event Controller — Reset/Rewind/Reverse/Pause/Play/Advance buttons, Max Simulation Time, Simulation Timestep, Current Simulation Frame, Event Controller Slider, Current Simulation Time, Output Interval.

The Event Controller also displays the current simulation time and includes a calculation speed control which allows you to slow down execution, if desired, to observe details during calculations.

Use the Event Controller to execute the current event as follows:

1. Press *Play* to start the reconstruction or simulation calculations.
2. Press *Pause* to pause the calculations.
3. Make any changes to the event set-up options (e.g., *Driver Controls*).
4. Press *Reset* to re-initialize the calculation model.
5. Press *Play* to re-execute the reconstruction or simulation model with the updated driver controls.

> **NOTE:** If Reset is not pressed, any changes to the event set-up options will be ignored.

Choosing Options

HVE provides a number of useful options. These options are available from the Options menu. The available options are:

- **Key Results** — A window displaying the current calculation results for each human and vehicle
- **Coordinate Axes** — Display coordinate axis systems on each human, vehicle and environment

- **Contacts** — Display the human ellipsoids and vehicle surfaces used in occupant/pedestrian force calculations
- **Belt Anchors** — Display an icon showing the location of restraint system seat belt anchors installed in each vehicle
- **Velocity Vectors** — Display the current direction of each object's velocity vector
- **Skidmarks** — Display skidmarks resulting from each skidding tire
- **Targets** — Display user-entered target positions as translucent objects

These options are implemented as toggles: you can show and hide these features. These options are described later in this section, as well as in the Menu Reference section of the manual.

(updated: the current Options menu contains a larger set of show toggles — Show Key Results, Show Axes, Show Contacts, Show Belt Anchors, Show Velocity Vectors, Show Skidmarks, Show Tracks, Show CGs, Show CG Paths, Show Accelerometers, Show Accelerometer Paths, Show Position Sequences, Show Targets, Show Connections — plus Create DB, Distance Tool..., Show Path..., and Autoposition. The settings items are Grid..., Units..., Shadows..., Render... (Ctrl+R), Playback..., Simulation Controls... (Ctrl+Y), Calculation Options... (Ctrl+J), DyMESH..., Get Surface Info..., and Preferences... (Ctrl+F). See the Options Menu reference.)

Chapter 2 continues in Part B: Creating Report and Playback Windows.

Chapter 2: How To Use HVE — Part B

Updated edition, revised for the current HVE release. Original: HVE User's Manual, Seventh Edition (Jan 2006), printed pages 2-40 through 2-69.

Part A: Basic concepts through Event Mode • Part C: Selecting User Options, Getting Help, Video

Contents (Part B)

- Creating Report and Playback Windows
- Creating Case Files
- Creating Databases
- Printing Results
- Setting the View

Creating Report and Playback Windows

Select Playback Mode after creating one or more events. Playback Mode is used to prepare output reports, view trajectory simulations and combine multiple simulation events into a single, coherent sequence. Playback Mode is also used to route these results to your printer or video recording device.

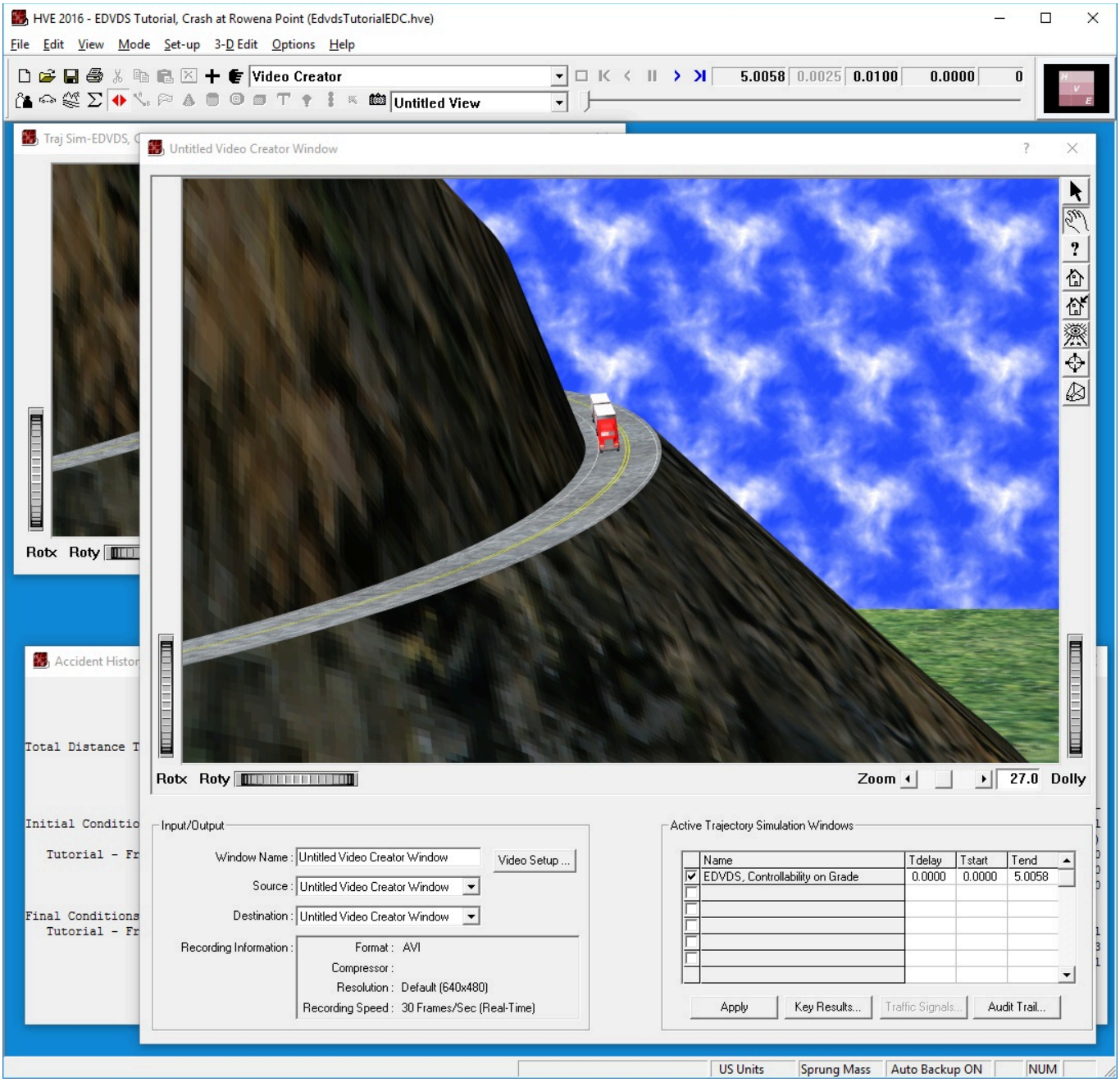


Figure 2-32: The HVE Playback Editor combining the results of multiple simulations into a single coherent sequence.

In Playback Mode, the user creates two types of windows:

- **Report Windows**
- **Playback Windows**

Report and Playback Windows are described in the following sections.

Creating Report Windows

Each Report Window contains the results for a single event. To create a Report Window, perform the following steps:

1. Select Playback Mode.

2. Click on *Add New Object*. The Report Window dialog will be displayed, containing a list of all the executed events (see Figure 2-33).

> **NOTE:** The Event Status tells you if the selected event has been executed. If the event has not been executed, there will not be reports available.

1. Choose an event from the *Active Events* list.
2. If desired, edit the event name.
3. Select the desired output report(s).
4. Press *OK*.

(updated: the current Report Window dialog contains the Active Events list, a read-only Status field, an editable Name field, and a multiple-selection Report Type list with Select All and Clear All buttons — so several report windows for one event can be created in a single step.)

The selected output window will be displayed. Output types may be broadly categorized into the following types:

- Numeric Outputs
- Graphic Outputs
- Trajectory Simulations
- Variable Output Table

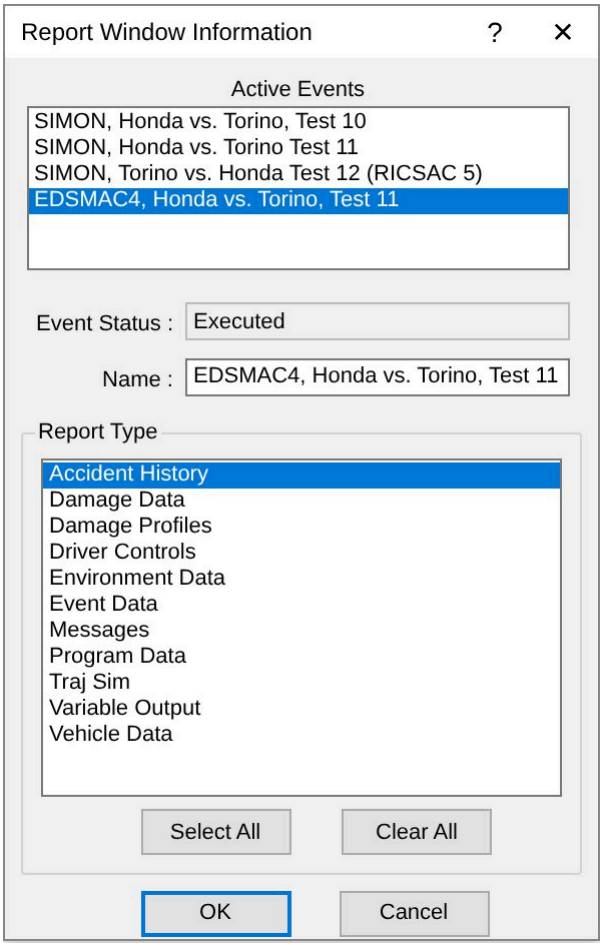


Figure 2-33: The Report Window Information dialog.

Numeric Outputs

Numeric outputs are displayed in Preview Windows (see the Common Reports window reference). The types of numeric outputs are described below.

> **NOTE:** The specific data available in any numeric output window is dependent upon the calculation model.

Accident History — The Accident History output report is a table of positions and velocities for each user-entered path position (see the Accident History reference).

Damage Data — The Damage Data output report normally contains user-entered damage profile data for reconstruction models and calculated damage profile information for simulation models (see the Damage Data reference).

Driver Data — The Driver Data output report normally contains user-entered driver control tables (steering, braking, throttle and gear selection vs time) for simulation models.

Environment Data — The Environment Data output report normally contains the environment parameters used by the reconstruction or simulation model.

Event Data — The Event Data output report normally contains the event set-up data (positions, velocities and event options) used by the reconstruction or simulation model. *(updated: the 2006 manual repeated the Damage Data description here by mistake; the Event Data report actually contains event set-up data.)*

Human Data — The Human Data output report contains the human anthropometric information supplied by the HVE Human Model and actually used by the reconstruction or simulation model.

> **NOTE:** Although the HVE Human Model contains several hundred parameters, the reconstruction or simulation model may use only a few.

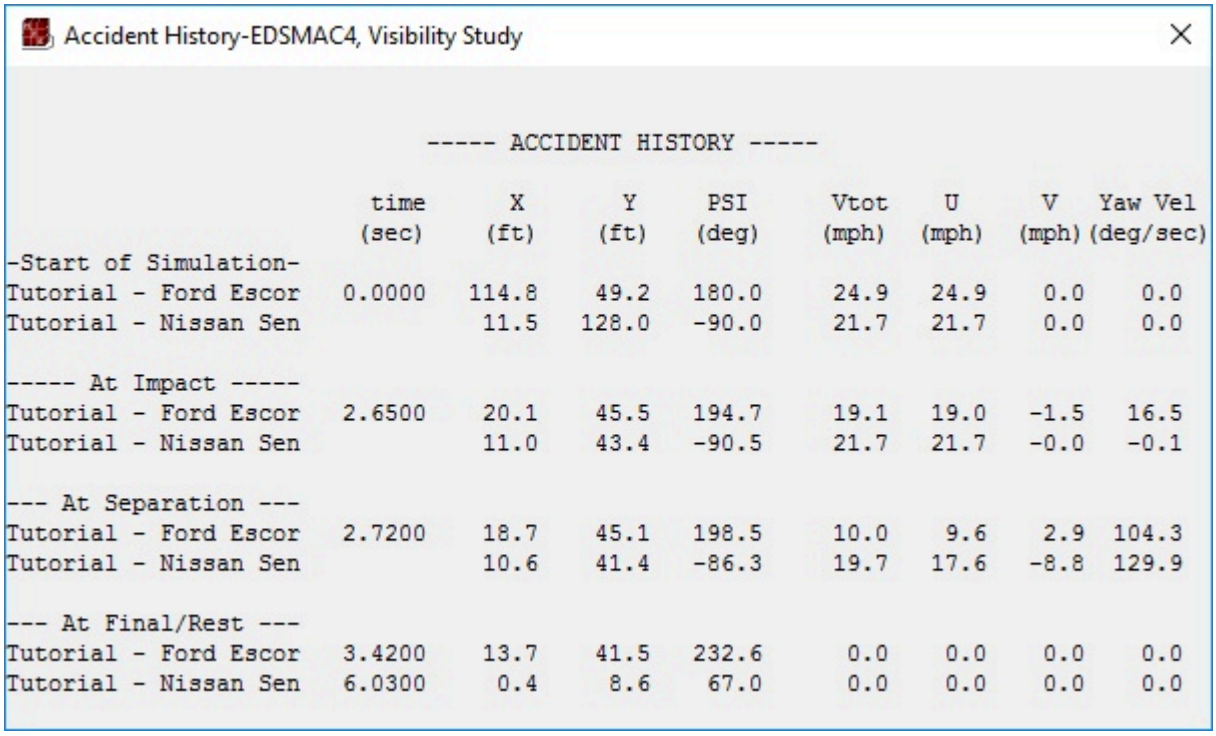
Injury Data — The Injury Data output report contains injury results predicted by an occupant or pedestrian simulator.

Messages — The Messages output report contains information about the run. This information does not normally contain calculated results; rather it contains warnings and diagnostics which describe the run.

Program Data — The Program Data output report normally contains general program input data for documentation purposes. For example, simulation models normally report the simulation control data in this report.

Vehicle Data — The Vehicle Data output report contains the vehicle parameters actually used by the reconstruction or simulation model.

> **NOTE:** Although the HVE Vehicle Model contains several hundred parameters, the reconstruction or simulation model may not use them all.



Accident History-EDSMAC4, Visibility Study

----- ACCIDENT HISTORY -----

	time (sec)	X (ft)	Y (ft)	PSI (deg)	Vtot (mph)	U (mph)	V (mph)	Yaw Vel (deg/sec)
-Start of Simulation-								
Tutorial - Ford Escor	0.0000	114.8	49.2	180.0	24.9	24.9	0.0	0.0
Tutorial - Nissan Sen		11.5	128.0	-90.0	21.7	21.7	0.0	0.0
----- At Impact -----								
Tutorial - Ford Escor	2.6500	20.1	45.5	194.7	19.1	19.0	-1.5	16.5
Tutorial - Nissan Sen		11.0	43.4	-90.5	21.7	21.7	-0.0	-0.1
--- At Separation ---								
Tutorial - Ford Escor	2.7200	18.7	45.1	198.5	10.0	9.6	2.9	104.3
Tutorial - Nissan Sen		10.6	41.4	-86.3	19.7	17.6	-8.8	129.9
--- At Final/Rest ---								
Tutorial - Ford Escor	3.4200	13.7	41.5	232.6	0.0	0.0	0.0	0.0
Tutorial - Nissan Sen	6.0300	0.4	8.6	67.0	0.0	0.0	0.0	0.0

Figure 2-34: A typical Accident History numeric output window.

Graphic Outputs

Graphic output reports are displayed in Report Windows. The types of graphic outputs are described below.

Site Drawing — The Site Drawing output window is a static view of the accident site showing the vehicles at the user-entered positions for each vehicle (e.g., Initial, Impact, Final/Rest). The vehicle speeds at each position are also displayed.

> **NOTE:** The Site Drawing may show the results for one or two vehicles, depending on the number of vehicles in the reconstruction model.
>
> **NOTE:** The Site Drawing is normally used for reconstruction models.

Damage Profiles — The Damage Profiles output window is a view of the vehicle damage on each vehicle. In addition to the profile itself, the Damage Profile output window may also display the PDOF and other damage-based information.

> **NOTE:** The Damage Profiles window may be either a 2-dimensional or 3-dimensional view, depending on the calculation model.

Momentum Diagram — The Momentum Diagram is a static display which shows a momentum (vector) diagram describing the collision between two vehicles. The Momentum Diagram may be based on vehicle damage data or accident site data.

> **NOTE:** In any case, the Momentum Diagram window requires that sufficient data be entered to establish impact and separation positions and velocities.

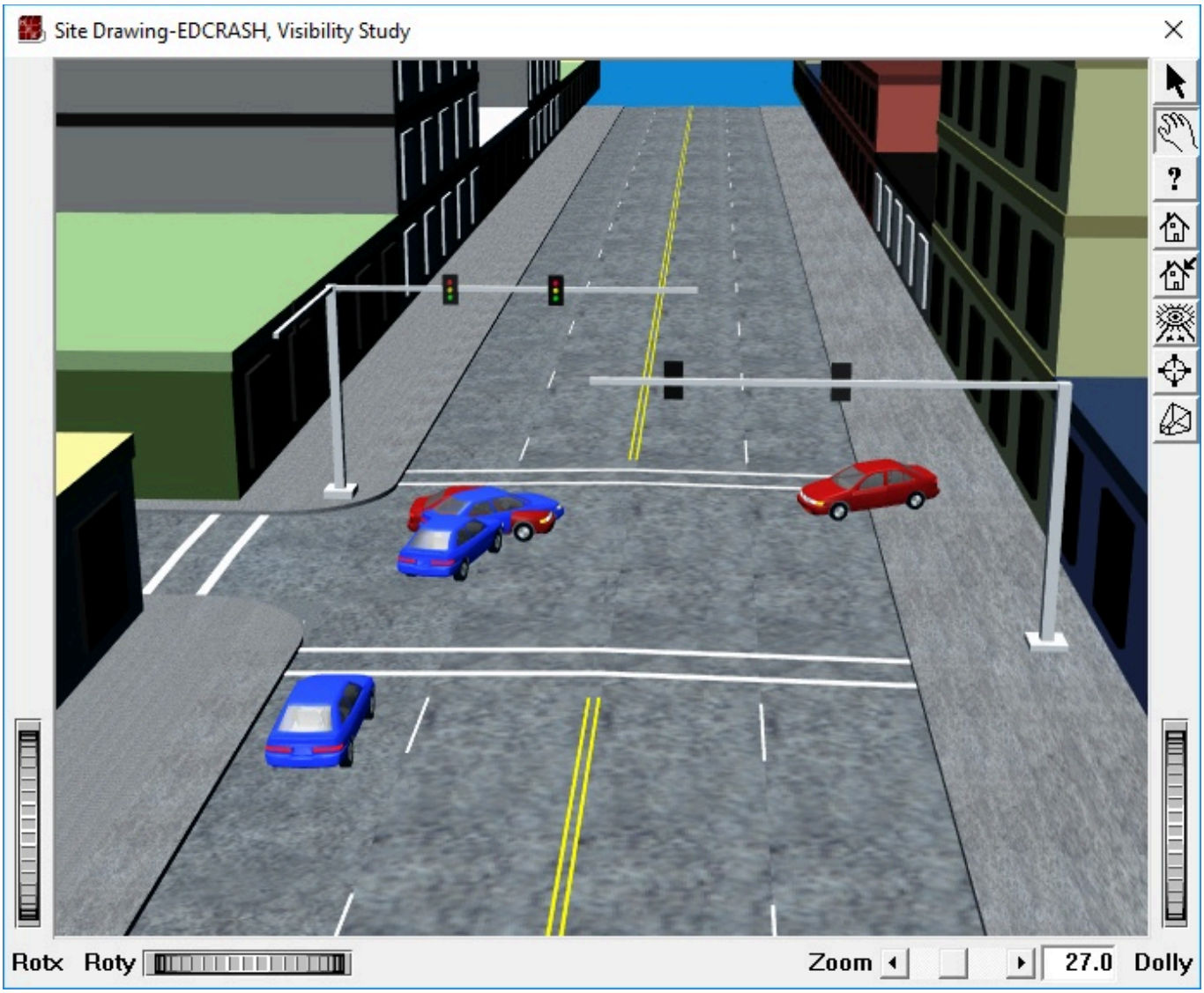
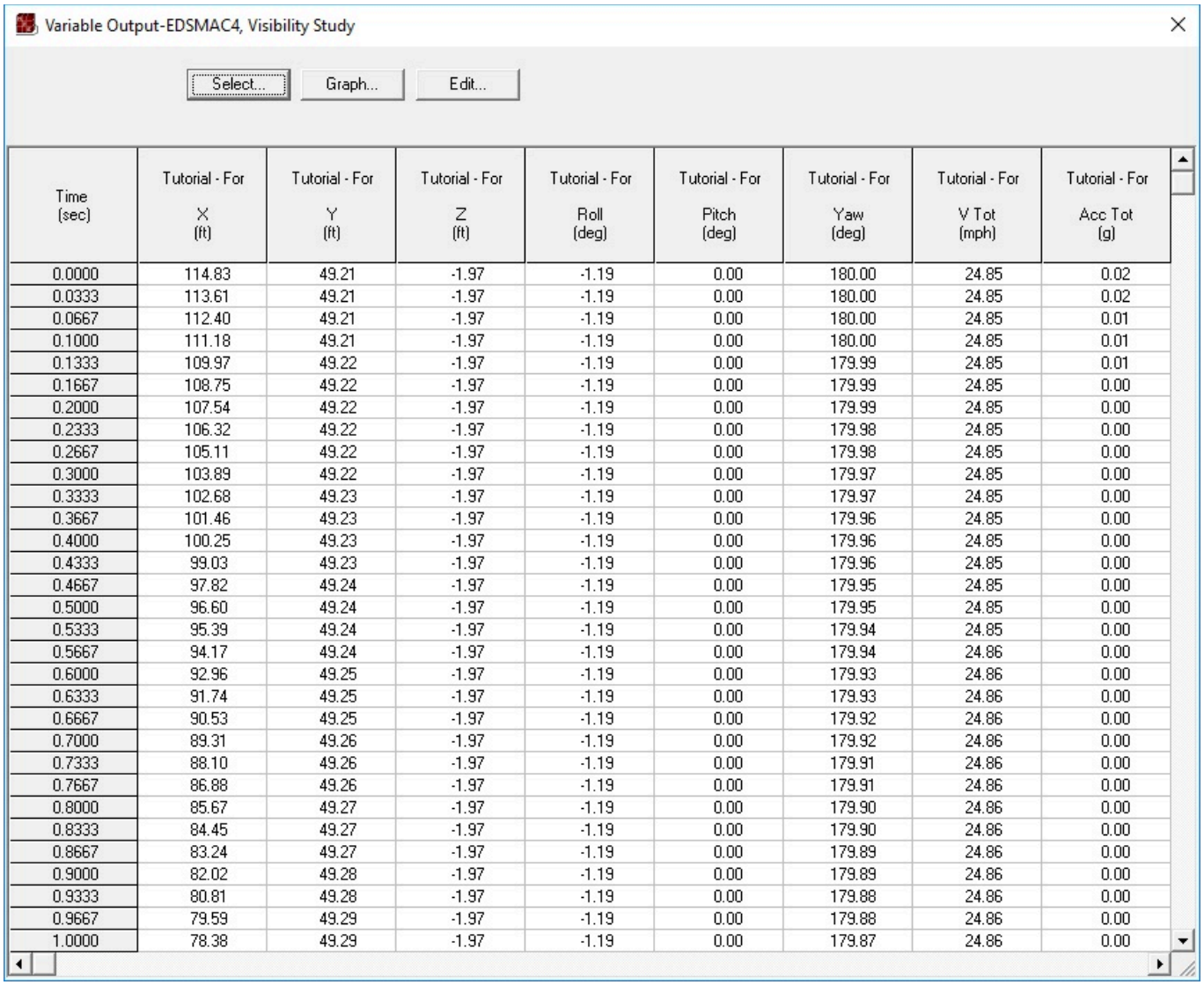


Figure 2-35: A Site Drawing graphic output from EDCRASH.

Variable Output Table

The Variable Output Table is a special output table of simulation results for an individual event, displayed as a function of time. The Variable Output Table can show a tremendous amount of data and is scrollable both horizontally and vertically (see the Variable Output reference).



Time (sec)	Tutorial - For X (ft)	Tutorial - For Y (ft)	Tutorial - For Z (ft)	Tutorial - For Roll (deg)	Tutorial - For Pitch (deg)	Tutorial - For Yaw (deg)	Tutorial - For V Tot (mph)	Tutorial - For Acc Tot (g)
0.0000	114.83	49.21	-1.97	-1.19	0.00	180.00	24.85	0.02
0.0333	113.61	49.21	-1.97	-1.19	0.00	180.00	24.85	0.02
0.0667	112.40	49.21	-1.97	-1.19	0.00	180.00	24.85	0.01
0.1000	111.18	49.21	-1.97	-1.19	0.00	180.00	24.85	0.01
0.1333	109.97	49.22	-1.97	-1.19	0.00	179.99	24.85	0.01
0.1667	108.75	49.22	-1.97	-1.19	0.00	179.99	24.85	0.00
0.2000	107.54	49.22	-1.97	-1.19	0.00	179.99	24.85	0.00
0.2333	106.32	49.22	-1.97	-1.19	0.00	179.98	24.85	0.00
0.2667	105.11	49.22	-1.97	-1.19	0.00	179.98	24.85	0.00
0.3000	103.89	49.22	-1.97	-1.19	0.00	179.97	24.85	0.00
0.3333	102.68	49.23	-1.97	-1.19	0.00	179.97	24.85	0.00
0.3667	101.46	49.23	-1.97	-1.19	0.00	179.96	24.85	0.00
0.4000	100.25	49.23	-1.97	-1.19	0.00	179.96	24.85	0.00
0.4333	99.03	49.23	-1.97	-1.19	0.00	179.96	24.85	0.00
0.4667	97.82	49.24	-1.97	-1.19	0.00	179.95	24.85	0.00
0.5000	96.60	49.24	-1.97	-1.19	0.00	179.95	24.85	0.00
0.5333	95.39	49.24	-1.97	-1.19	0.00	179.94	24.85	0.00
0.5667	94.17	49.24	-1.97	-1.19	0.00	179.94	24.86	0.00
0.6000	92.96	49.25	-1.97	-1.19	0.00	179.93	24.86	0.00
0.6333	91.74	49.25	-1.97	-1.19	0.00	179.93	24.86	0.00
0.6667	90.53	49.25	-1.97	-1.19	0.00	179.92	24.86	0.00
0.7000	89.31	49.26	-1.97	-1.19	0.00	179.92	24.86	0.00
0.7333	88.10	49.26	-1.97	-1.19	0.00	179.91	24.86	0.00
0.7667	86.88	49.26	-1.97	-1.19	0.00	179.91	24.86	0.00
0.8000	85.67	49.27	-1.97	-1.19	0.00	179.90	24.86	0.00
0.8333	84.45	49.27	-1.97	-1.19	0.00	179.90	24.86	0.00
0.8667	83.24	49.27	-1.97	-1.19	0.00	179.89	24.86	0.00
0.9000	82.02	49.28	-1.97	-1.19	0.00	179.89	24.86	0.00
0.9333	80.81	49.28	-1.97	-1.19	0.00	179.88	24.86	0.00
0.9667	79.59	49.29	-1.97	-1.19	0.00	179.88	24.86	0.00
1.0000	78.38	49.29	-1.97	-1.19	0.00	179.87	24.86	0.00

Figure 2-36: The Variable Output Table.

The Variable Output Table includes the following features:

Variable Selection — Choosing *Select* displays the Variable Selection dialog, allowing the user to choose variables from the available output groups. The selected variables are displayed in the Variable Output Table as a function of time (see the Variable Selection dialog reference).

> **NOTE:** The selectability of variables is determined by the simulation model.

Variable Editing — Each HVE simulation event produces results calculated by the simulation model. Certain non-calculated results may also be assigned by the simulation model; these non-calculated results are editable by the user. For example, a 2-D simulator, which calculates X, Y and Heading, may allow the user to edit the roll, pitch and Z elevation values to produce the appearance of a 3-D simulation. Pressing *Edit* displays the user-editable table of these non-calculated variables (see the Variable Edit dialog reference).

> **NOTE:** The results made available for editing by HVE are determined by the simulation model.

Graphing — The *Graph* button allows the user to produce a graph of the first six selected variables in the Variable Output Table (see the Variable Graphing reference).

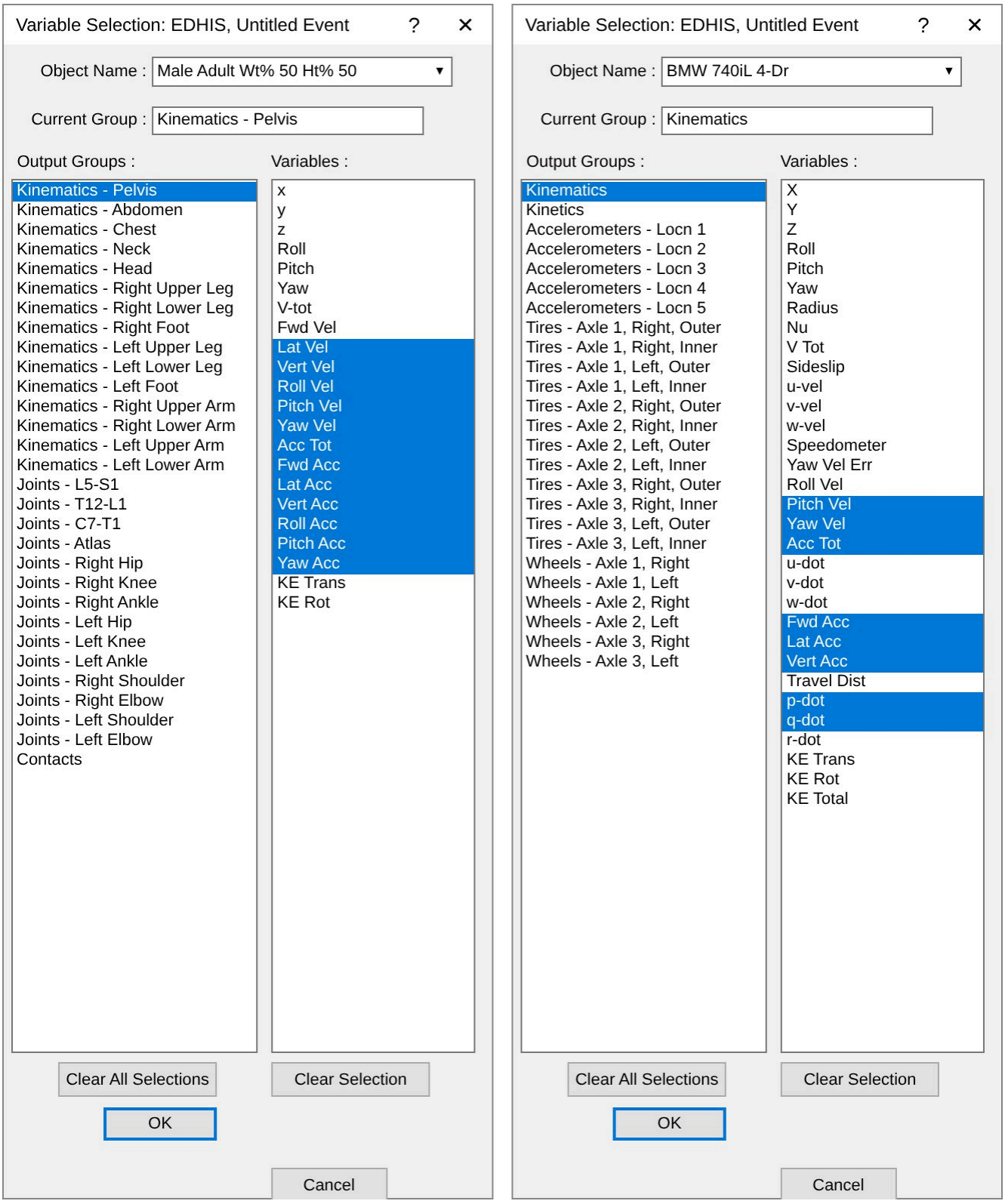
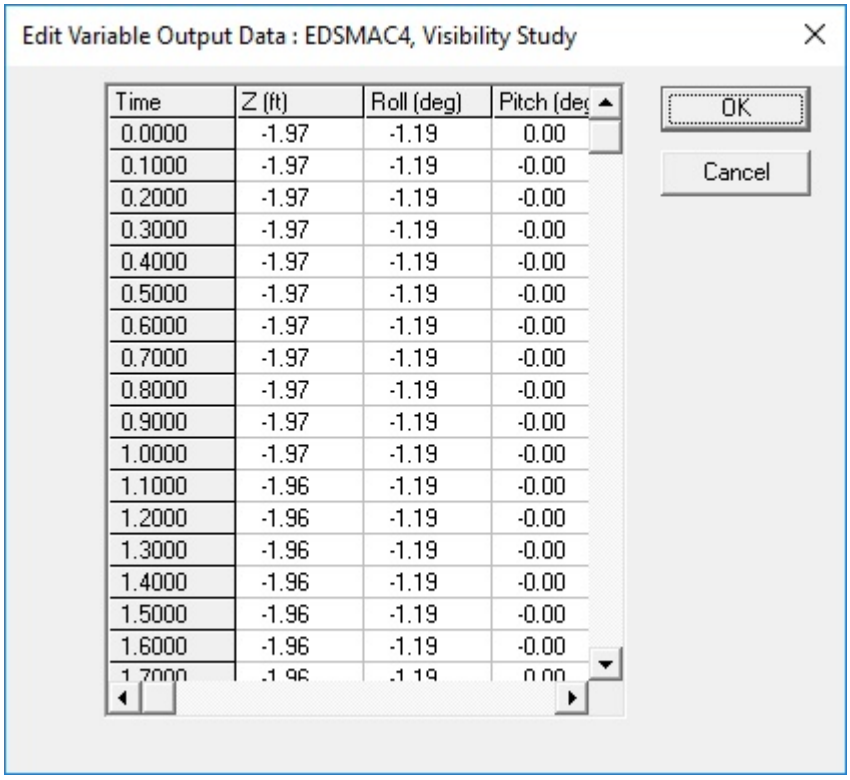


Figure 2-37: The Variable Selection dialogs for humans and vehicles.



Time	Z (ft)	Roll (deg)	Pitch (deg
0.0000	-1.97	-1.19	0.00
0.1000	-1.97	-1.19	-0.00
0.2000	-1.97	-1.19	-0.00
0.3000	-1.97	-1.19	-0.00
0.4000	-1.97	-1.19	-0.00
0.5000	-1.97	-1.19	-0.00
0.6000	-1.97	-1.19	-0.00
0.7000	-1.97	-1.19	-0.00
0.8000	-1.97	-1.19	-0.00
0.9000	-1.97	-1.19	-0.00
1.0000	-1.97	-1.19	-0.00
1.1000	-1.96	-1.19	-0.00
1.2000	-1.96	-1.19	-0.00
1.3000	-1.96	-1.19	-0.00
1.4000	-1.96	-1.19	-0.00
1.5000	-1.96	-1.19	-0.00
1.6000	-1.96	-1.19	-0.00
1.7000	-1.96	-1.19	0.00

Figure 2-38: The Variable Edit dialog.

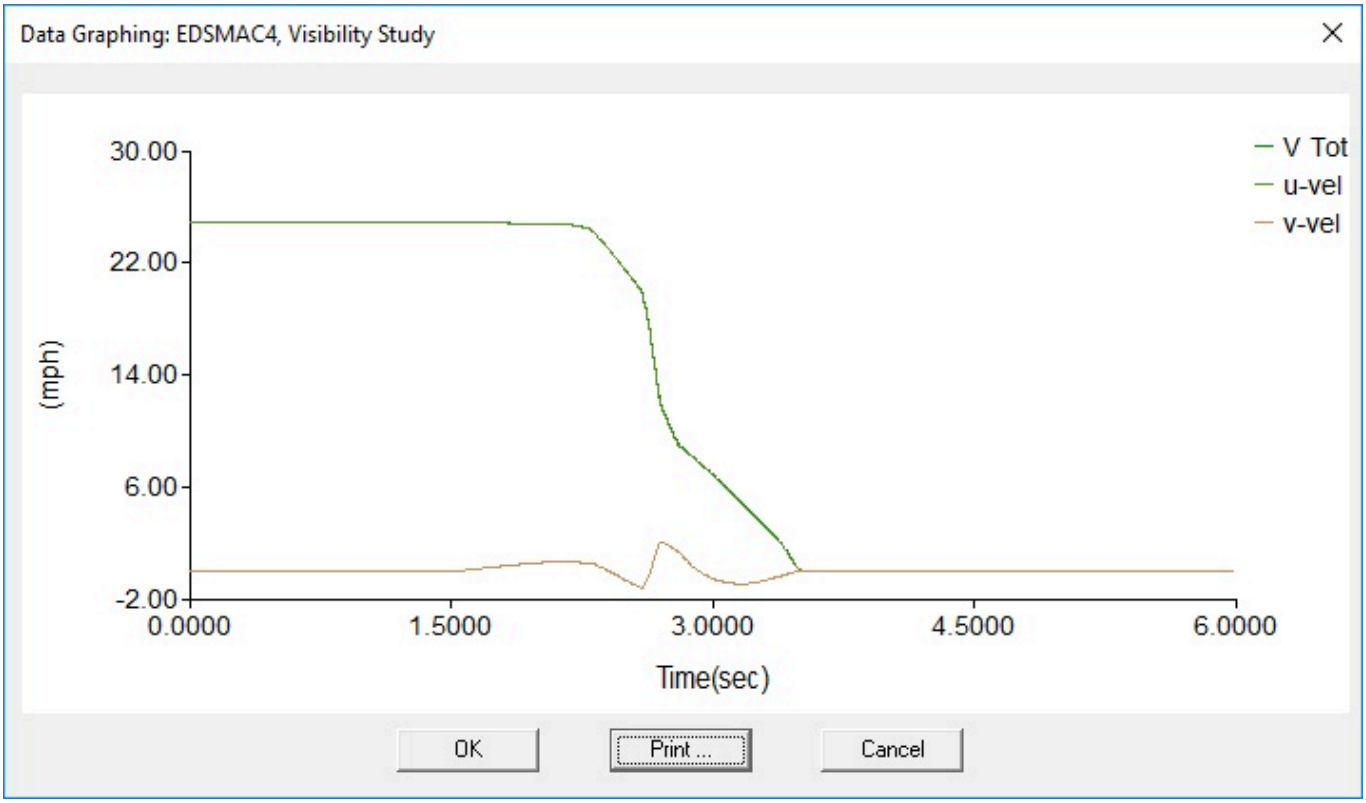


Figure 2-39: The Variable Graphing window.

Trajectory Simulation

The Trajectory Simulation output window is a dynamic visualization of the human and vehicle motion, the same as during Event mode, beginning at the user-entered initial position and continuing until the end of the simulation. Like other output reports, Trajectory Simulations are also displayed in Report Windows (see the Trajectory Report reference).



Figure 2-40: The Trajectory Simulation window.

The motion in the Trajectory Simulation window is controlled using the Playback Controller, described below.

Creating a Playback Window

The Playback Window allows the user to combine several events (previously set up in Report Windows) to be displayed in a single window. The Playback Window can be used to produce sequences showing multi-vehicle accidents. The Playback Window is also used to produce video output.

To create a Playback Window, perform the following steps:

1. Create one or more Trajectory Simulation windows (see previous section).
2. Add a Playback Window. *(updated: the 2006 manual said to click "Add Playback Window" on the Options menu; in the current version the Playback Window is created from the Playback Editor the same way as other playback*

objects, and its dialog is titled "Playback Information".) The Playback Window dialog will be displayed, containing a list of all the Report Windows containing Trajectory Simulations (see Figure 2-41).

3. Choose one or more Report Windows.
4. Enter a name for the Playback Window.

> **NOTE:** Only one Playback Window can be created. *(updated: in the current version the Playback Information window is resizable rather than fixed-size.)*

The Playback Window initially displays the humans and vehicles for each event at their initial positions. However, because not all action begins at the same time, some editing is usually required. For example, if an impact occurs 3.0 seconds into a sequence, the start of the occupant simulation must be delayed by 3.0 seconds. The editing procedure is described briefly in the following section.

The Playback Window also provides the Video Source and Destination selection, allowing the user to view simulation results, create a compressed movie and also route a simulation movie to a video file:

- **Video Source** — Option list which allows the user to choose the source of the information displayed in the Playback Window.
- **Video Destination** — Option list which allows the user to route the Playback Window to a user-selectable destination.

(updated: the current Playback Information window also contains a Video Setup... button, a read-only Recording Information panel showing the current Format (e.g., AVI), Compressor, Recording Size (e.g., HDTV 1080p 1920x1080) and Recording Speed, plus Key Results..., Traffic Signals..., Reorder Events... and Audit Trail... buttons.)

> **NOTE:** For more information about Video Source and Destination, see Video Interface, later in this chapter, and Section Nine, Video Output.

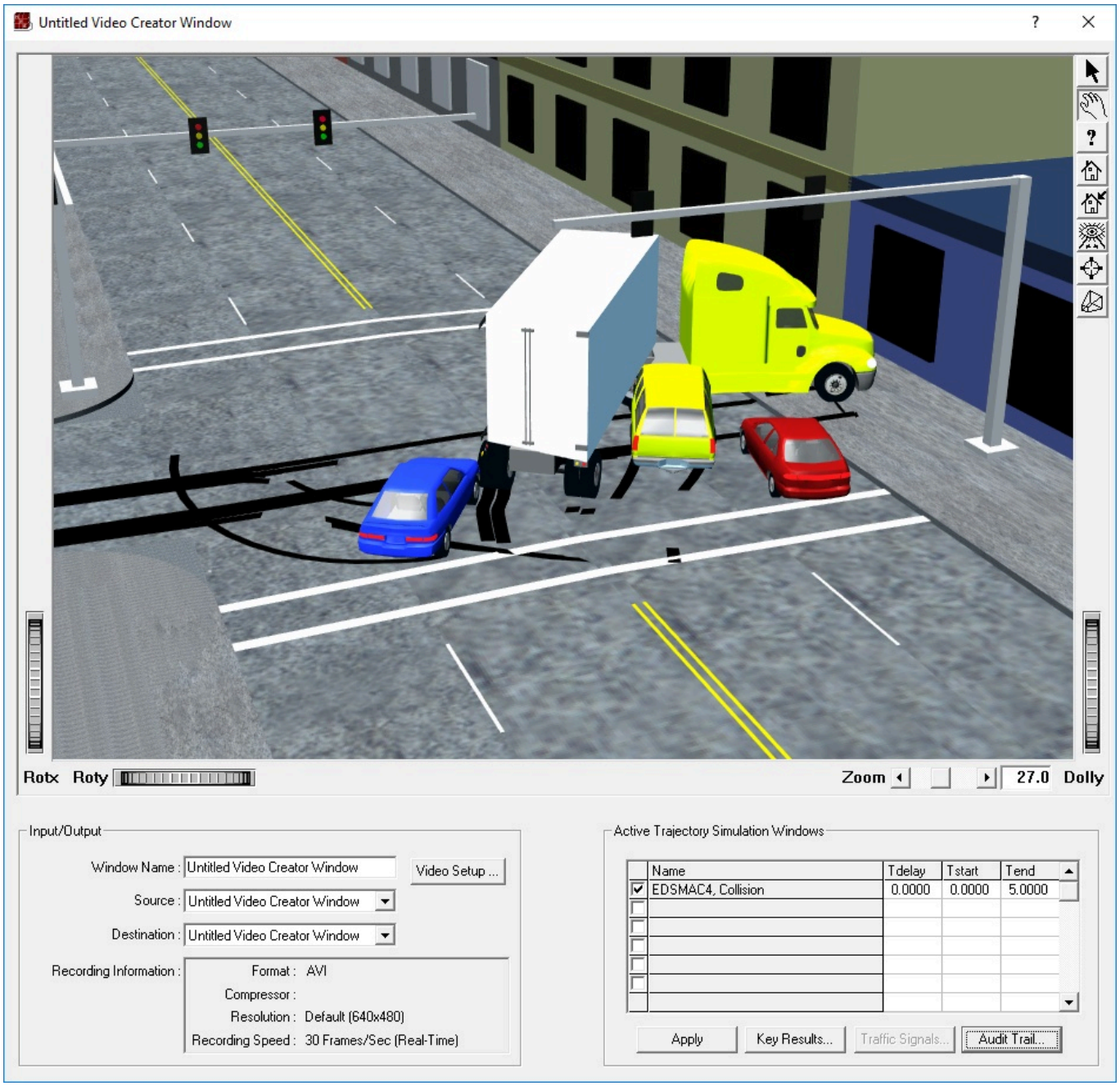


Figure 2-41: The Playback Window dialog.

Playback Controller

The Playback Controller (see Figure 2-42) is used for controlling the motion in Trajectory Simulations and the Playback Window. Like the Event Controller described in Part A, the Playback Controller is much like a VCR in both form and function. It has the same buttons:

- **Stop** — Stop the simulations displayed in Trajectory Simulation and/or Playback Windows
- **Rewind to Start** — Return to the start of the simulation
- **Reverse** — Run the simulation backwards
- **Pause** — Pause the simulation(s)

- **Play** — Run the simulation forward
- **Advance To End** — Advance to the end of the simulation

In addition to the above trajectory simulation control features, the Playback Controller also includes the following components:

- **Time Display** — Displays the current simulation time
- **Frame Control Slider** — Allows the user to visually choose a frame within the sequence. To move to the first frame, move the slider all the way to the left; to choose a frame near the middle of the sequence, move the slider to the middle, and so forth. If the user knows the exact frame number, it may be entered directly.

See the Playback Controls reference for the current control layout.



Figure 2-42: The Playback Controller.

Combining and Editing Multiple Events

The timing of multiple trajectory simulation events is affected by the following editing operations, performed using the *Active Trajectory Simulations* list in the Playback Window (see Figure 2-41):

- **Editing Tdelay** — Set Tdelay to delay the onset of an event by the user-entered time value
- **Editing Tstart and Tend** — Set Tstart and Tend to display a portion of the entire event, defined from Tstart to Tend

> **NOTE:** Tstart and Tend allow you to select a specific portion of the event. For example, an occupant simulation may contain 50 milliseconds of irrelevant motion while the human settles into the seat and reaches equilibrium. By selecting Tstart as 50 milliseconds after the output begins, you can remove this undesired motion.

To edit multiple events into the desired time sequence, perform the following steps:

1. Open the Playback Window dialog if you have not already done so. It displays a list of active Trajectory Simulation windows for your events.
2. Select an event.
3. Enter the delay time, start time and end time for the event.
4. Repeat the above steps for any number of additional events.

The motion for each event is displayed separately in its Trajectory Simulation Report Window. Use the Report Window to confirm the event(s) are synchronized correctly. The Playback Window shows all the events correctly synchronized in a single window.

(updated: the current Playback Information dialog also provides a Reorder Events... button for changing the order in which events are applied.)

Audit Trail

Each Playback Window may contain the results from several events. The merging of these events requires a considerable amount of logic. For example, the pre-impact phase of an accident sequence may use one simulation model, the impact phase may use a second, and one of the vehicles might use a third model for the post-impact phase.

The HVE Playback Window has an internal table to determine which objects are driven by which events during which time intervals. This table is called the Audit Trail; it may be displayed and printed. The Audit Trail is a scrollable text box containing the following information about each event in the Playback Window:

- Report Window Name
- Event Name
- Event Objects used in the Playback Window

> **NOTE:** Even though an event may include two objects, the motion of one of its objects may be controlled by a different event. For example, after impact, a vehicle might roll over. You can use a different (3-D) model to control this portion of the object's motion in the Playback Window.

- The starting and ending times during which the object's motion is controlled by that event

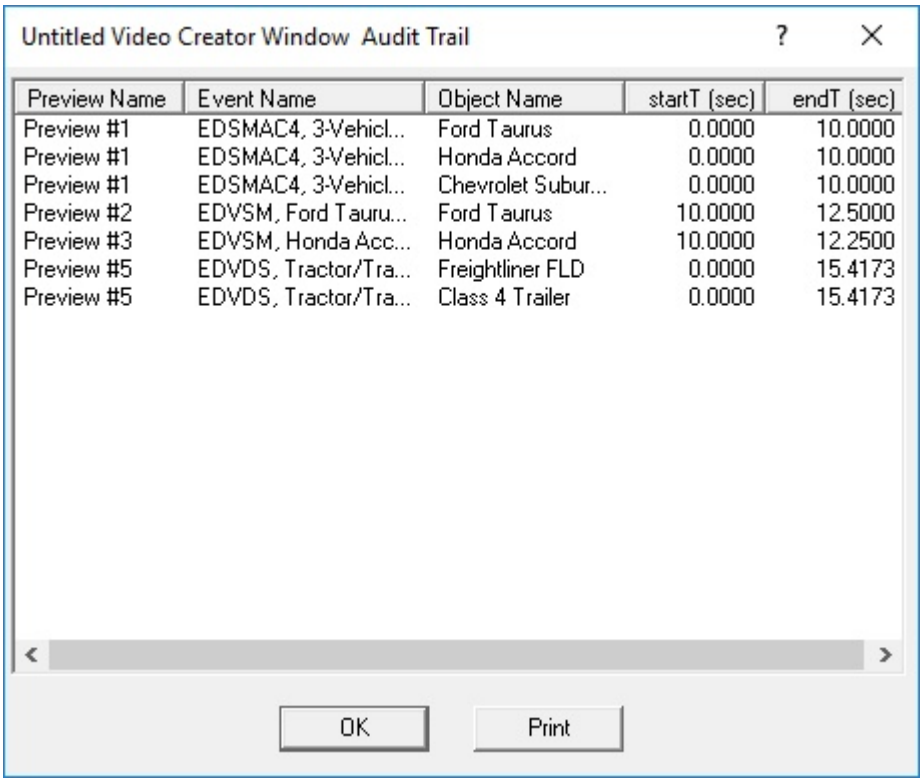


Figure 2-43: The Audit Trail.

The Audit Trail is a convenient way to document the simulation models which control the motion of each object in the Playback Window. To display the Audit Trail for a Playback Window, perform the following steps:

1. Choose *Audit Trail*. The Audit Trail will be displayed, as shown in Figure 2-43.
2. After viewing, and possibly printing, the Audit Trail, press *OK*.

Creating Case Files

HVE creates a file, called the Case file, containing all the active humans and vehicles and the environment. All the events and playback windows are also saved in the case file. At any time, it is possible that many events involving several humans and vehicles have been executed. Therefore, cases can become very large; a 25 MB case file is not uncommon.

Case files are stored in the `\case` subdirectory of HVE. The directory must be defined by HVE because these case files are also used to create a virtual database of humans, vehicles and environments used in previous cases; HVE needs to know where they are when you first start HVE. See Appendix VIII, Databases, for more information about the Case Database.

Opening Cases

To open an existing case, choose

Open

from the File menu (Ctrl+O), as shown in Figure 2-44. If another case is open, HVE will ask the user to save the current case before opening a different one.

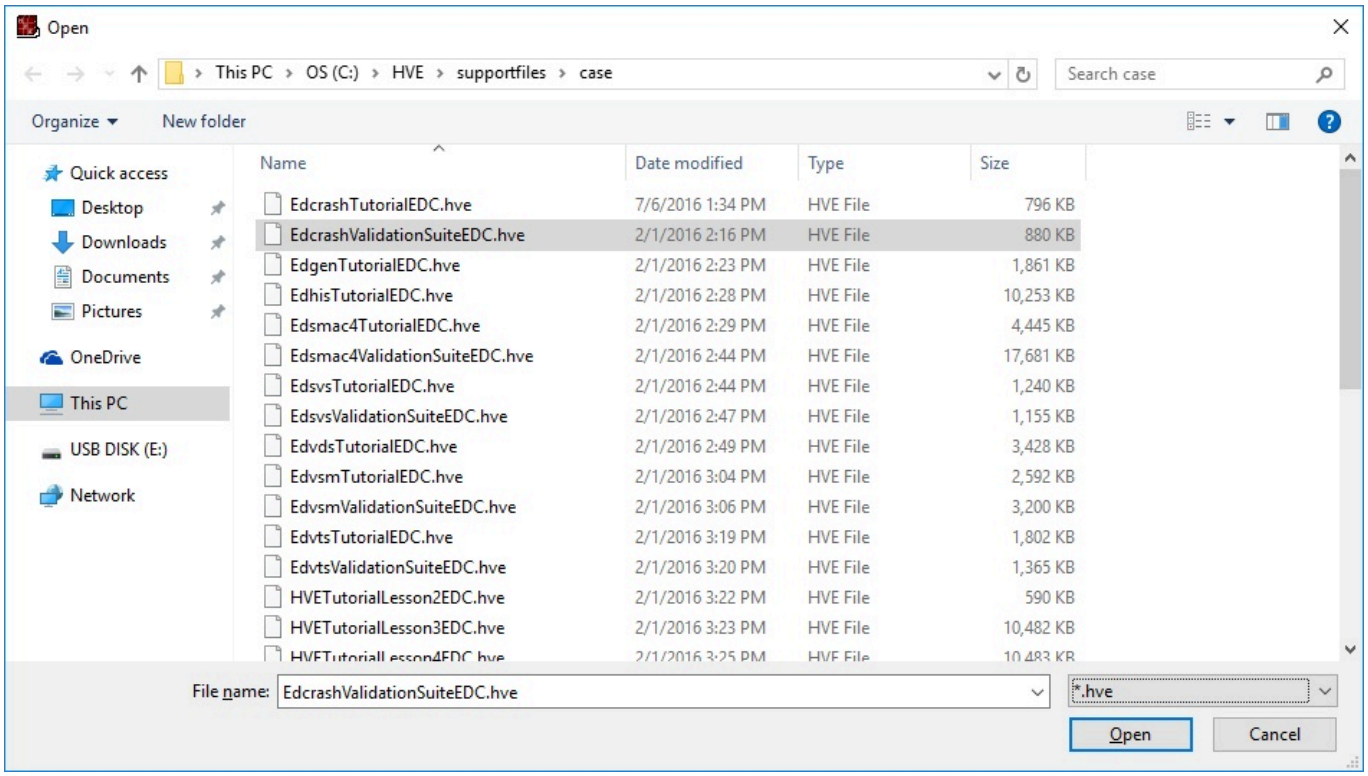


Figure 2-44: The Open Case File Selection dialog.

Saving Cases

To save the current case, choose

Save

from the File menu (Ctrl+S). HVE will save the case information using the current Case filename.

Save As

If the case hasn't been saved before, HVE will display the Save As file dialog, which requests the user to supply a filename for the case file. The user may also add a case title to the case using the Save As dialog, shown in Figure 2-45. Every case should have a case title, because the title appears in the HVE main menu bar, as well as in the heading on printed output reports.

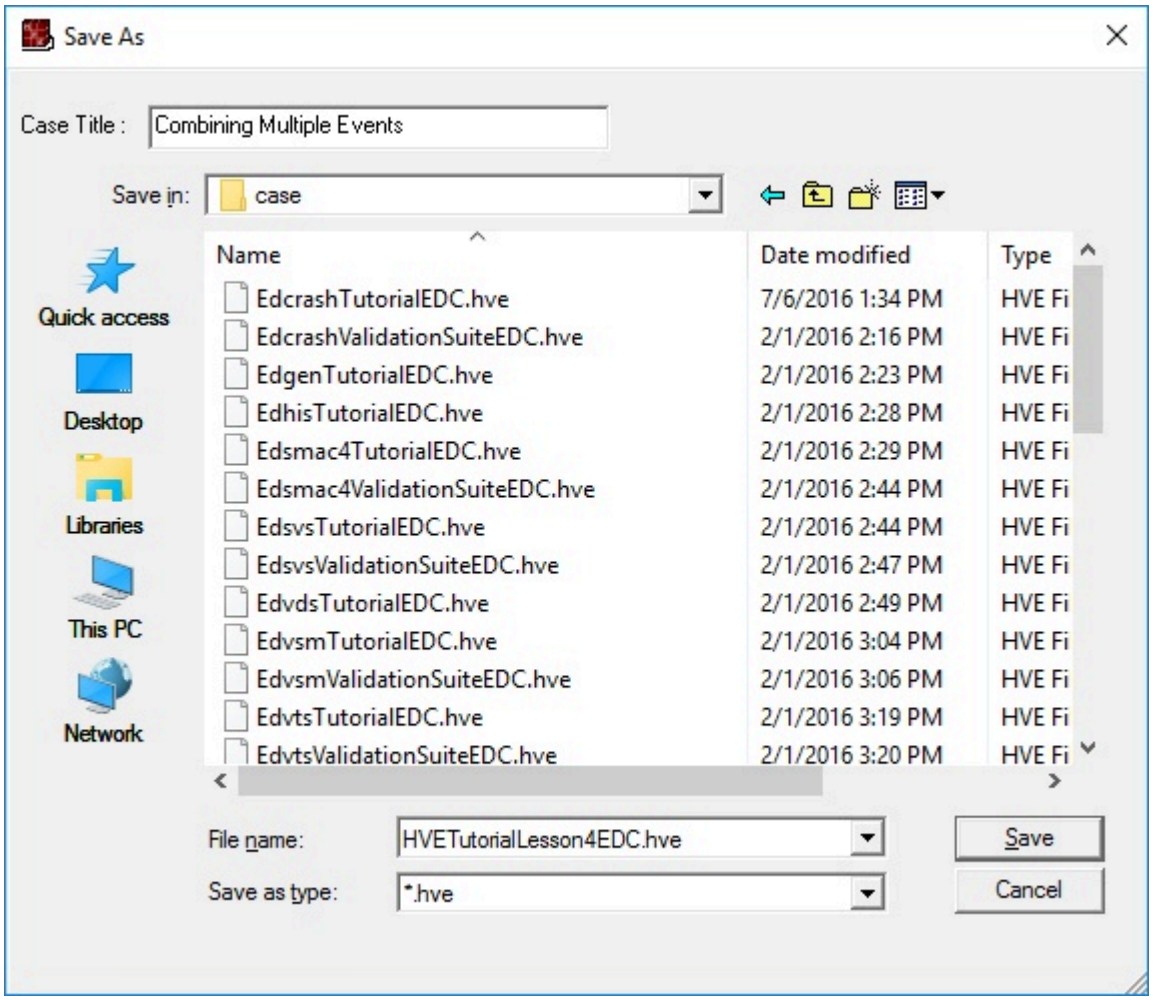


Figure 2-45: The Save Case File Selection dialog.

Starting New Cases

To start a new case, choose

New

from the File menu (Ctrl+N). If another case file is open, HVE will ask to save the current case before opening a new one. Choosing New completely removes all the case information associated with the current case.

Configuration File

HVE maintains a file, called the configuration file, that keeps track of all the user's current settings and preferences not directly tied to any case. Examples of user preferences include the current status of user options, such as Show Key Results, Units and Autoposition.

Creating Databases

One of HVE's most powerful features is its user-extendible databases for humans, vehicles and tires. By using these databases, the user can truly realize HVE's greatest strength: its object-oriented design. Users will find themselves thinking less about data and more about the interaction of objects (humans, vehicles and environments).

Human Database

The Human database allows the user to select pre-defined humans, according to the following keys:

- Sex
- Age
- Weight Percentile
- Height Percentile

The Human Database is available in the Human Editor's Human Information dialog (see Figure 2-46). By using these keys, HVE creates a human model based on anthropometric studies (the GEBOD databank). This human may then be edited by the user.

(updated: the current dialog is titled "GEBOD Human Information"; see the Human Information reference.)

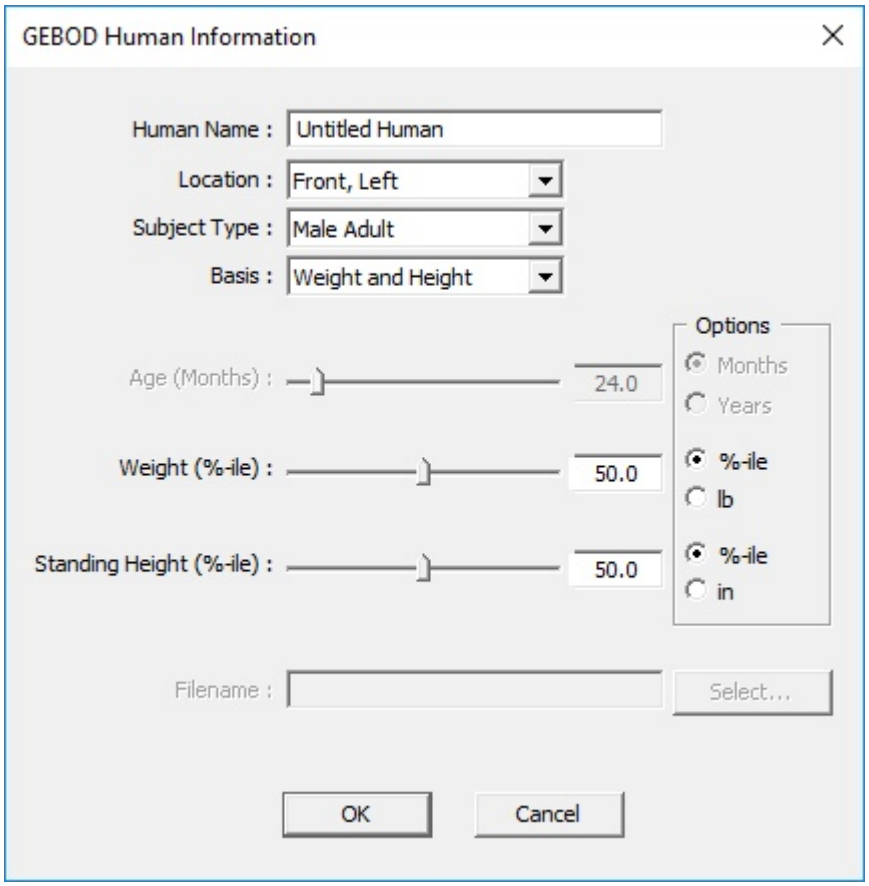


Figure 2-46: The Human Editor's Human Information dialog.

Vehicle Database

The Vehicle Database allows the user to select pre-defined vehicles, according to the following keys:

- Type
- Make
- Model
- Year
- Body Style

The Make, Model, Year and Body Style keys are user-definable (see User Databases, below).

The selectable vehicle types are:

- Passenger Car
- Pickup
- Van
- Sport-Utility
- Truck
- Trailer
- Dolly
- Fixed Barrier
- Movable Barrier

(updated: the 2006 manual listed "SAE Movable Barrier" and "SAE Fixed Barrier"; the current labels are simply "Movable Barrier" and "Fixed Barrier", and an additional "Other" type is available.)

The Vehicle Database is available in the Vehicle Editor's Vehicle Information dialog (see Figure 2-47 and the Vehicle Information reference). After choosing the desired vehicle, HVE loads it into the Vehicle Editor for possible editing and/or use in the current case.
(updated: the current version also provides a Vehicle Information search dialog with a Vehicle Search text field for finding vehicles by name.)

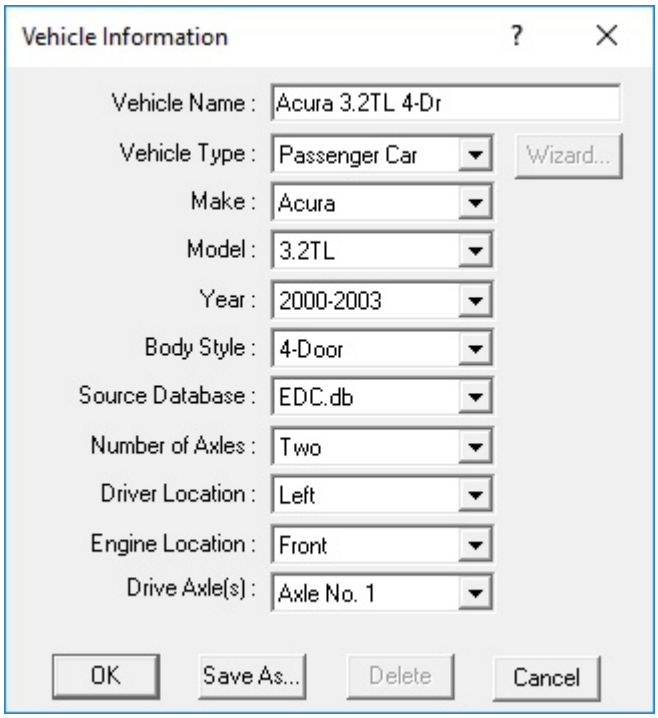


Figure 2-47: The Vehicle Editor's Vehicle Information dialog.

Tire Database

The Tire Database allows the user to select pre-defined tires, according to the following keys:

- Type
- Manufacturer
- Model
- Size

The Manufacturer, Model and Size keys are user-definable (see User Databases, below). The selectable tire types are:

- Passenger Car
- Light Truck
- Heavy Truck
- Mobile Home

(updated: the 2006 manual labeled the third type "Heavy (on-highway) Truck"; the current label is "Heavy Truck".)

The Tire Database is available in the Vehicle Editor's Tire Information dialog (see Figure 2-48 and the Tires and Wheels reference). After choosing the desired tire, HVE assigns it to the current vehicle at the selected wheel location (e.g., Axle 1, Right Side).

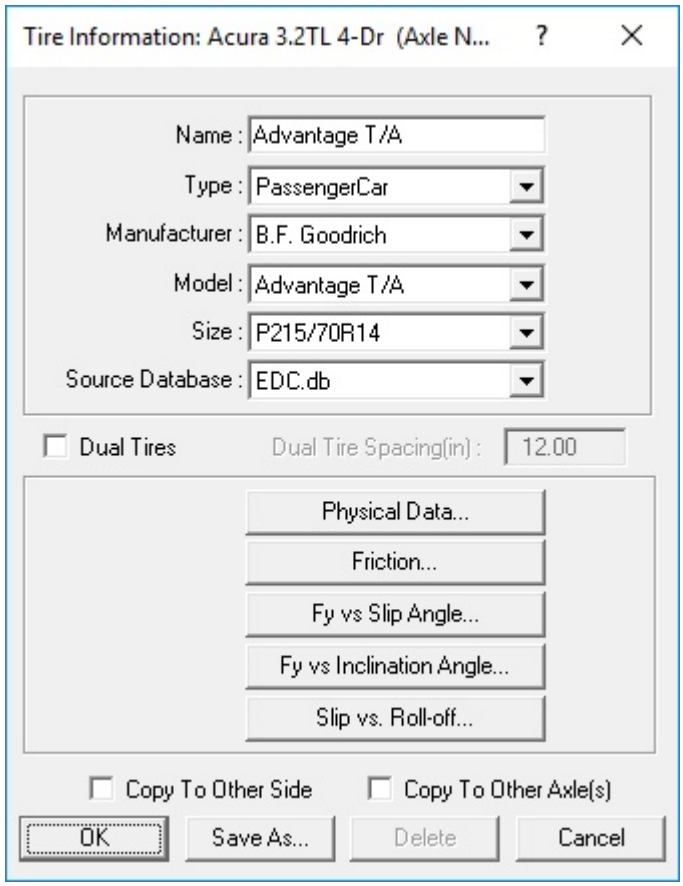


Figure 2-48: The Vehicle Editor's Tire Information dialog.

Generic Databases

HVE includes databases for humans, vehicles and tires. These databases include generic data, that is, data which do not represent any specific human, vehicle or tire, but are representative of typical humans, vehicles and tires for a given size range. The generic databases were produced by the statistical analysis of a large number of humans, vehicles and tires (see references 3.10, 4.9, 4.10, 4.11, 4.26), and contain reasonable estimates for all data necessary to execute any HVE-compatible reconstruction or simulation model. Humans, vehicles and tires created using HVE's generic database may be modified and saved in custom databases, as described below.

Custom Databases

Vehicles and tires may be purchased in custom databases. A custom database differs from a generic database in that it contains actual vehicles (e.g., 1996 Ford Taurus 2-Dr Coupe) and tires (e.g., Goodyear Arriva, P195/75R14). Custom databases are available from EDC, as well as from independent suppliers. Custom databases often include 3-D geometry files for the vehicle body for use in DyMESH simulations, resulting in a life-like appearance. Individual vehicles are also available from EDC.

User Databases

The user may create and save vehicles and tires in his or her own database. These databases are called User Databases. Objects from both generic and custom databases may be saved.

> **NOTE:** The supplier of custom databases may include a license key which prevents unauthorized duplication.

To save the current vehicle in the User Database, perform the following steps:

1. Double-click on the current vehicle in the Active Vehicles list. The Vehicle Information dialog will be displayed, showing the database information for the current vehicle.
2. Choose *Save As*. The Save As New Vehicle dialog will appear, displaying the current Vehicle Type, Make, Model, Year and Body Style information. The vehicle's data filename, version number, license key and image filename are also displayed (see Figure 2-49).
3. Update these fields by entering the desired data.
4. Press *OK* to save the new vehicle in the User Database.

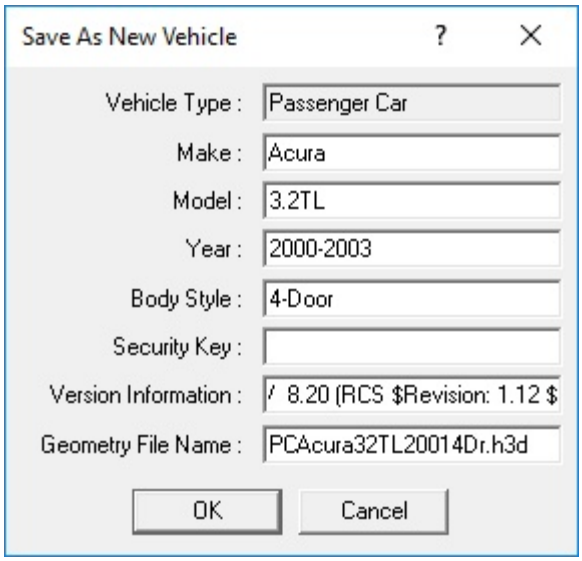


Figure 2-49: The Vehicle Save-As dialog.

> **NOTE:** It is possible to have the same vehicle in two different databases (i.e., EDC and User); however, it is not possible to have the same vehicle twice in the same database. HVE's Database Manager will ask if you want to replace the existing vehicle or enter a different make, model, year or body style.

To save a tire on the current vehicle in the User Database, perform the following steps:

1. Click on the desired tire in the Vehicle Viewer. A pop-up menu will be displayed.
2. Choose *Tire* from the pop-up menu. The Tire Information dialog for the selected tire will be displayed.
3. Choose *Save As*. The Save As New Tire dialog will appear, displaying the current Tire Type, Manufacturer, Model and Size information. The tire's data filename, version number and license key are also displayed.
4. Update these fields by entering the desired data.
5. Choose *OK* to save the new tire in the User Database.

> **NOTE:** It is possible to have the same tire in two different databases (i.e., EDC and User); however, it is not possible to have the same tire twice in the same database. HVE's Database Manager will ask if you want to replace the existing tire or enter a different manufacturer, model, or size.

A number of issues arise when extending and maintaining databases. Users are encouraged to plan ahead and decide how to approach these issues. For more information about HVE's databases, see Appendix VIII, Databases.

Case Database

Every case includes a number of humans and vehicles. Every time the user saves a case, these humans and vehicles are saved in the case file. These humans and vehicles are made available for use in future cases. When starting HVE, all cases in the `\case` subdirectory are opened and the humans, vehicles, environments and tires are extracted to form what is called the Case Database.

To select humans from previous cases, perform the following steps:

1. Select Human mode, then choose *Mode* > *Add...* from the main menu. HVE will display a cascade menu containing the options *New...* and *Previous....*
2. Choose *Previous....* HVE displays a dialog listing every case in the `\case` subdirectory and the names of each human included in that case (see Figure 2-50).

> **NOTE:** If the selected case does not include the desired human, select a different case.

1. Select the desired human from the list of human names.
2. Press *Open* to add the selected human to the current case.

The selected human will be added to the current case. Use the same procedure to add vehicles from previous cases.

The purpose of the Case Database is not to extend your standard database of humans and vehicles — you should use Save As and create your own User Database for that purpose. Rather, the purpose of the Case Database is to allow you access to unusual vehicles created for, and used in, previous cases. For example, you may remember creating a pickup with a camper. Rather than recreating that vehicle, you can select it directly from the previous case in which it was created and used.

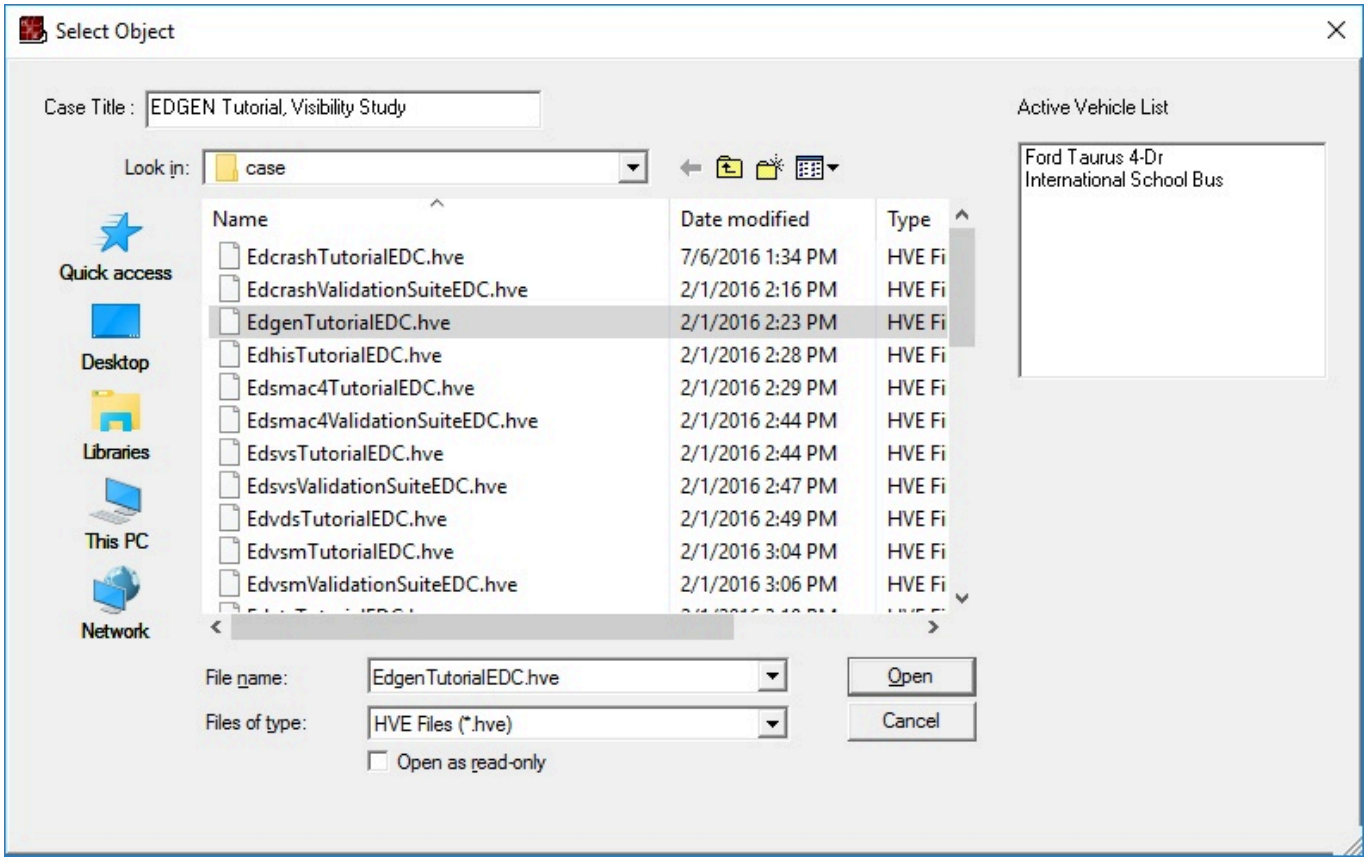


Figure 2-50: The Case Database selection dialog.

Printing Results

HVE allows the user to print several types of output reports, graphs and 3-D views of the accident sequence. The method for printing depends on the object being printed.

Printing Output Windows

In Playback Mode, all output windows may be printed by choosing

Print

from the File menu. The printable output reports are:

- All Numeric Output Reports (e.g., Accident History, Vehicle Data)
- All Graphic Output Reports (e.g., Damage Profiles)
- Variable Output Table
- Trajectory Simulations (including the Playback Window)

To print an output window, perform the following steps:

1. Choose Playback Mode.

> **NOTE:** The output windows are displayed in Playback Mode.

1. Choose the desired output report.
2. Choose *Print...* from the File menu (Ctrl+P). The Print dialog is displayed, providing several print options (see Figure 2-51).
3. Select from the available print options.
4. Press *OK*. The selected output window is printed on the system printer.

(updated: the current File menu also provides a "Print All..." item for printing all open report windows at once.)

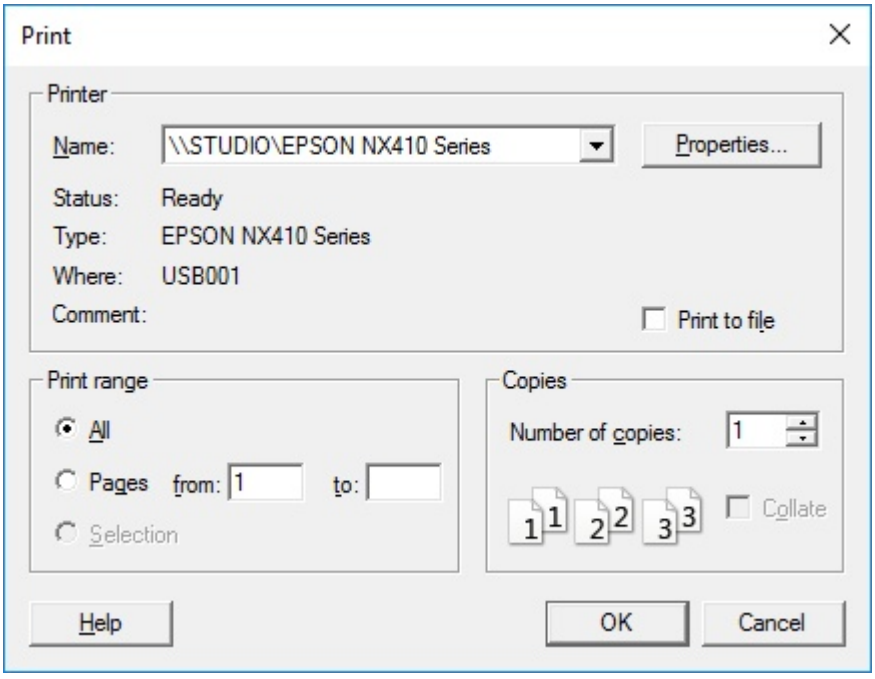


Figure 2-51: The Print dialog.

Printing From Modal Dialogs

If a graph or report is included as part of a modal dialog (i.e., a dialog that requires the user to press OK before continuing), the dialog will contain a

Print

button.

> **NOTE:** The Print button is required because you cannot select anything (including the Print option in the File menu) until you press OK on the current dialog.

The following modal dialogs have a Print button:

- Several graphs in the Vehicle Editor (e.g., Tire Lateral Force vs Slip Angle)
- The Audit Trail for the current Playback Window
- The Graphing dialog for Variable Output

All printed reports and graphs are printed on the current system printer.

> **NOTE:** The current system printer is installed using the Windows Control Panel.

Printing To a File

The HVE print dialog allows the user to print to file, as well as to the system printer. Printing to file has several important uses:

- Saving the desired information for later printing
- Exporting selected results to a file to be used by another program

To print the current results to file, perform the following steps:

1. Using the Print dialog, click *Print To File*.
2. Press *OK*. The Print to File dialog is displayed (see Figure 2-52).
3. Change the current printer name to *Generic / Text Only*.

> **NOTE:** If a Generic / Text Only printer is not included in the list of printers, you will need to install one using the standard Windows printer installation procedure.

1. Enter the desired filename, using `.txt` as the extension.
2. Press *OK*. The selected report is saved using the specified filename and format.

(updated: the current File menu also provides "Export..." and "FBX Export..." items for exporting data and 3-D geometry directly, which is often more convenient than printing to a file.)

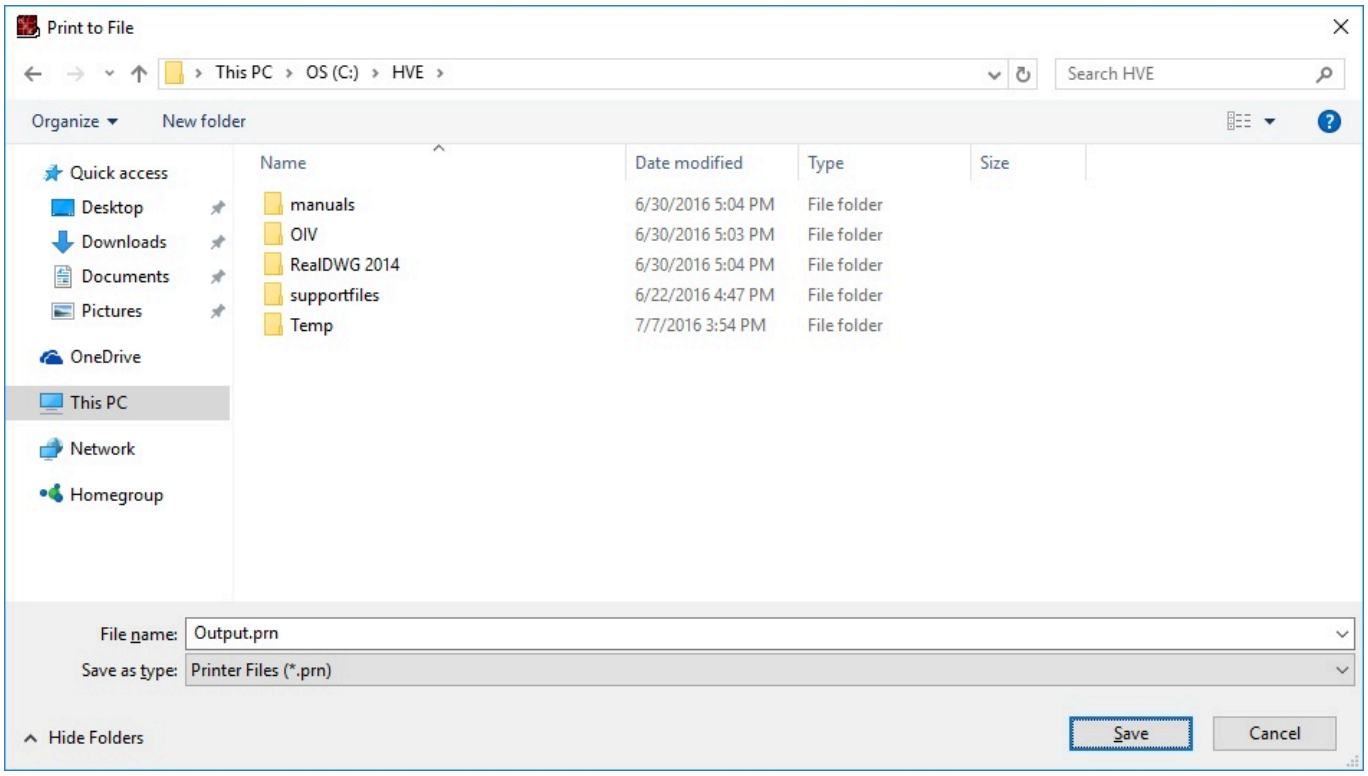


Figure 2-52: The Print To File selection dialog.

Setting the View

The view determines what is visible in any viewer. HVE uses a camera as its viewing paradigm; that is, you choose where you are looking from (the Camera Position), where you are looking at (the Picture Center), and the focal length of the lens. You can also set near and far clipping planes, which determine whether close-up and distant objects are visible.

There are two different ways to set the view:

- Use the thumb wheels on the viewer window borders to set the view interactively, as shown in Figure 2-53.
- Select *Set Camera...* from the View menu. The Set Camera dialog allows you to directly enter the viewing information described above, as shown in Figure 2-54 (see the Set Camera dialog reference).

Using the thumb wheels has the advantage of allowing the user to quickly and interactively select a view. Using the Set Camera dialog has the advantage of allowing the user to specify the exact camera location and view. The user can also attach the camera to one object and view another object as it moves by. The latter option is useful for visibility studies where the exact viewing position is an issue.

Note that, because HVE's view is 3-dimensional, the term "scale" has no relevance. In a 3-D world, the concept of scale is replaced by the position and focal length of the camera lens used in the view.

Object-based Cameras

By default, the camera is attached to the environment. However, the Set Camera dialog (see Figure 2-54) allows the user to attach the camera to any object (human, vehicle or environment) and attach the view to the same or any other object. This is useful for situations where visibility of drivers or witnesses is an issue.

To set the view using the viewer's thumb wheels, perform the following steps:

1. Using the viewer's pick/manipulate icons, choose Pick mode. The cursor will turn into a pointing arrow.
2. Click on the *Rot X* thumb wheel and drag up or down to rotate the entire scene about the viewer's horizontal axis.
3. Click on the *Rot Y* thumb wheel and drag up or down to rotate the entire scene about the viewer's vertical axis.

> **NOTE:** The Rot X and Rot Y thumb wheels rotate the scene about the viewer's axes, not the earth's axes!

1. Click the *Dolly* thumb wheel and drag up to move the camera closer to the scene; drag down to move the camera farther from the scene.
2. Using the viewer's pick/manipulate icons, choose Manipulate mode. The cursor will turn into a hand.
3. Click in the viewer using the left mouse button and drag the mouse. The scene will rotate about its center.
4. Click in the viewer using the middle mouse button and drag the mouse. The scene will translate in the direction of the mouse movement.

> **NOTE:** Setting the view using Manipulate mode takes a little practice. Once you get the hang of it, however, this technique works extremely well!

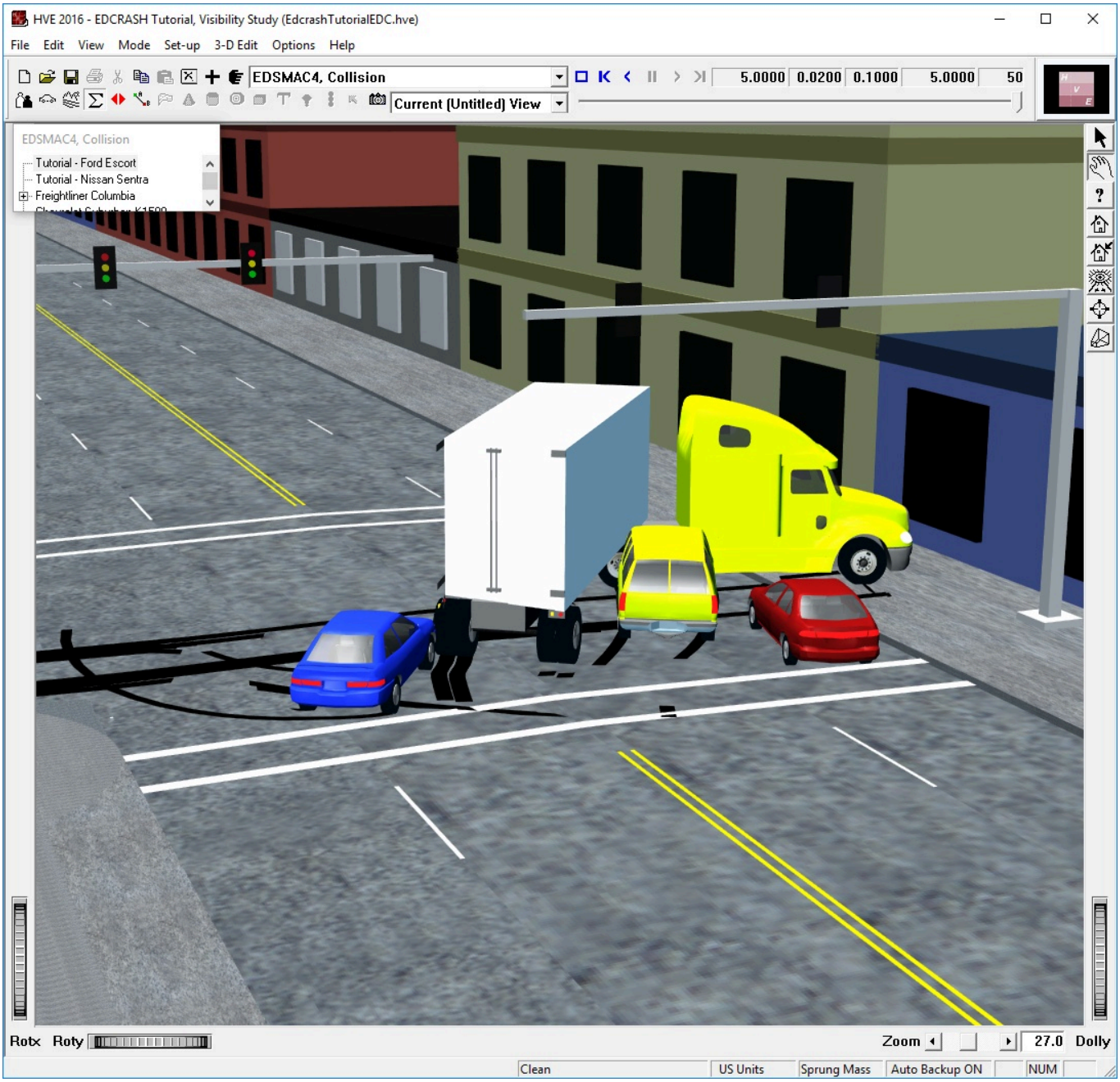


Figure 2-53: HVE viewer thumb wheels.

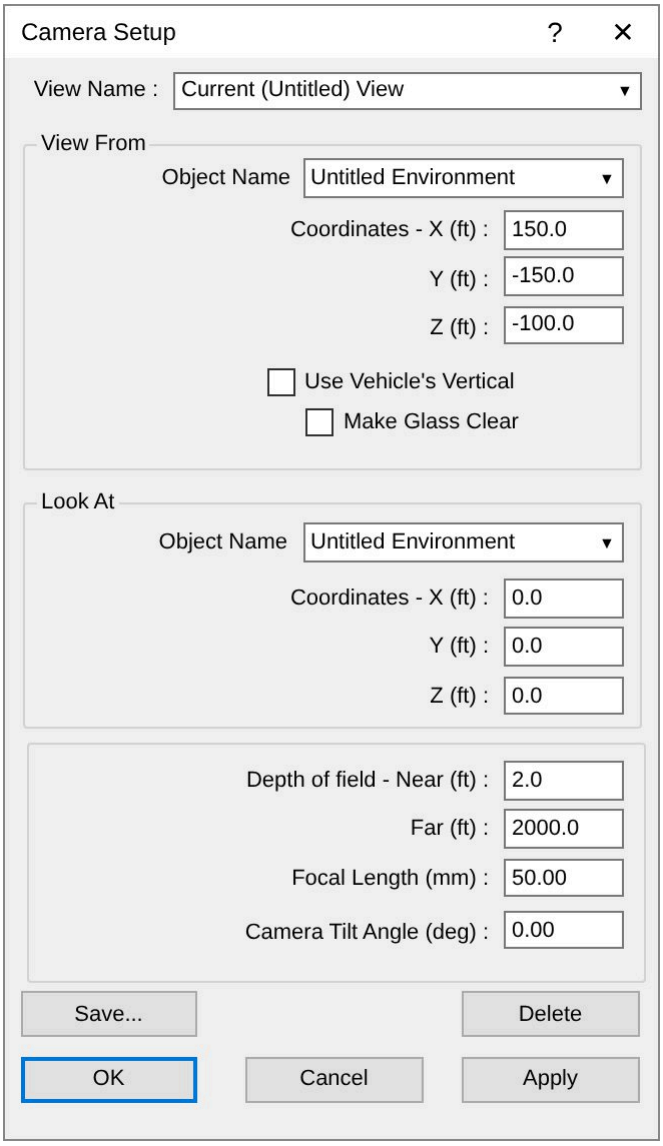


Figure 2-54: The Set Camera dialog.

Overlays

An Overlay is a collection of environment objects which can be made visible or invisible as a group. These objects are usually related in some way. For example, by placing all accident-related artifacts (debris, gouges, skidmarks) on an overlay, the user may turn this overlay off to show the scene's appearance before the accident, and turn it on to show how the scene appeared after the accident. The Overlays dialog, shown in Figure 2-55, is used for displaying and removing selected overlays from the scene (see the Overlays dialog reference).

Overlays are created when the environment is created or edited using the 3-D Editor. Refer to the 3-D Editor section for details on creating and naming overlays.

> **NOTE:** If you use another editor to create your environment, you can still use HVE's 3-D Editor to create overlays.

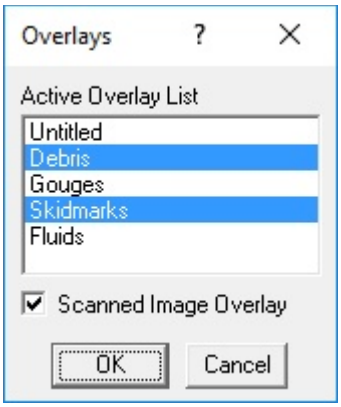


Figure 2-55: The Overlays dialog.

Scanned Image Overlay

A scanned image overlay is a special overlay created from a scanned photograph of the accident scene. The scanned image may be included as part of the environment background using the Environment Information dialog (see the Environment Editor reference).

A scanned background image adds photo-realism to the scene (see Figure 2-56). However, the use of a scanned image requires that the earth-fixed coordinates are known for both the location where the photograph was taken, as well as for the center of the photograph. In addition, the focal length of the camera lens must be known. Finally, to use a scanned image, the view must be fixed; that is, the camera cannot be attached to a moving human or vehicle.

Two excellent uses for scanned background images are:

- To show that a 3-D model of a scene matches the actual scene. This may be accomplished by first showing the scanned image, then showing the 3-D model from the same camera position.
- For a distant sky or background. If a 3-D model of the environment has a relatively flat horizon that fades into the distance, using a photograph of a cloudy sky (which HVE places beneath the 3-D model) creates a realistic environment.

> **NOTE:** The sky remains fixed, even if the camera is attached to a moving human or vehicle. This may reduce the realism.

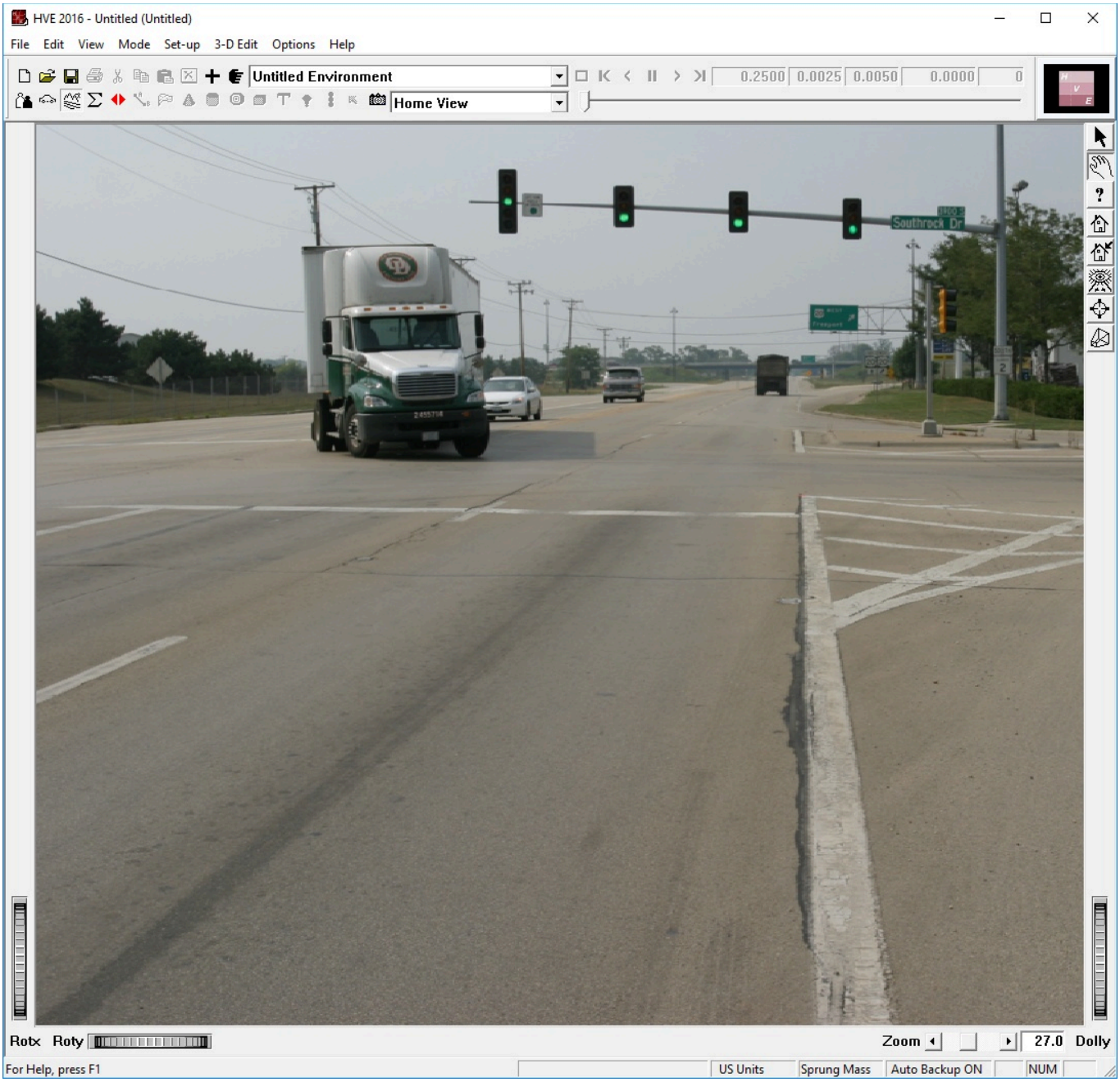


Figure 2-56: A scanned background image.

The individual overlays and scanned background image are displayed or hidden using the Overlays dialog, available in the View menu (see Figure 2-55). The Overlays dialog includes a multiple-selection list box containing the names of every overlay created using the 3-D Editor (the default name is "Untitled"). By default, all overlays are displayed (highlighted). The Overlays dialog also includes a Scanned Image check box to turn the scanned image on and off. To choose the desired overlays and scanned image for display, perform the following steps:

1. Choose *Overlays...* from the View menu. The Overlays dialog will be displayed; all the currently displayed overlays will be highlighted, and the Scanned Image check box will show the current status (on or off) of the scanned image.
2. Click on one or more highlighted overlay names to hide them.
3. Click on one or more unhighlighted overlay names to show them.

4. Click on the Scanned Image check box to show or remove it.
5. Press *OK*.

The selected overlays and scanned background image will be displayed.

Chapter 2 continues in Part C: Selecting User Options, Getting Help, and the Video Interface.

Chapter 2: How To Use HVE — Part C

Updated edition, revised for the current HVE release. Original: HVE User's Manual, Seventh Edition (Jan 2006), printed pages 2-70 through 2-104.

Part A: Basic concepts through Event Mode • Part B: Report/Playback Windows, Cases, Databases, Printing, View

Contents (Part C)

- Selecting User Options
- Getting Help
- Video Interface

Selecting User Options

You can customize HVE through the selection of several user options. Each of these options is selected using the Options menu. The current status of these options is saved when you exit HVE, so your user environment will be the same the next time you start HVE. These options are described below.

See the Options Menu reference for the complete list of current Options menu items.

Key Results Windows

Key Results windows display important calculation results for each human and vehicle in the event. Key Results windows are available for both reconstruction and simulation models (see Figures 2-57 and 2-58). The results for reconstruction models are static, while the results for simulations are time-dependent and are updated every output timestep (see Simulation Controls, later in this section).

To display Key Results windows, check *Show Key Results* on the Options menu. To remove Key Results windows, uncheck *Show Key Results*.

> **NOTE:** The Key Results option is a toggle.

See the Key Results window reference for details.

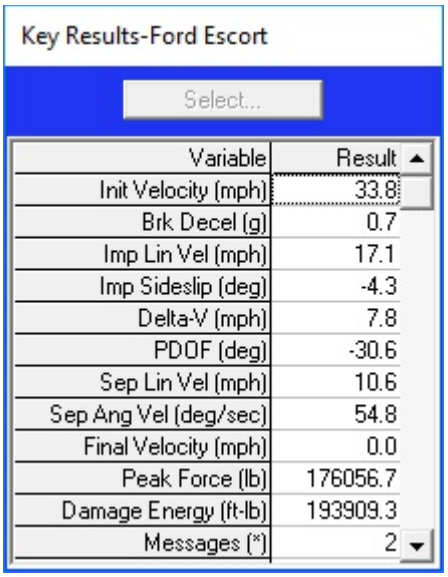


Figure 2-57: Key Results window for reconstructions.

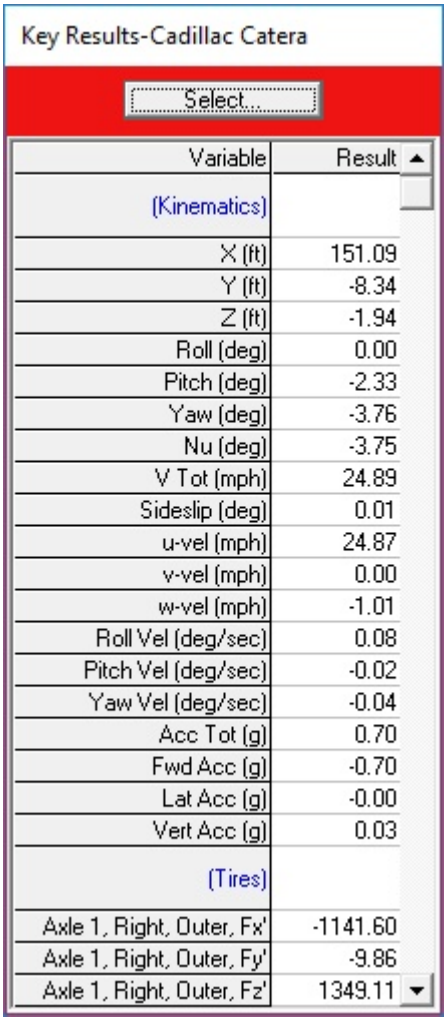


Figure 2-58: Key Results window for simulations.

Selecting Key Results

The results for simulation models are user-selectable, so the user can decide what data are important to display for the current event.

> **NOTE:** Although the typical simulation model produces an enormous amount of output data, the user is normally interested in the results for only a few variables. These are called "Key Results".

To select the desired variables during Event mode, perform the following steps:

1. If not already displayed, check *Show Key Results* on the Options menu.
2. Locate the Key Results window for the desired human or vehicle (each object has its own Key Results window) and click on *Select Variables*. The Variable Selection dialog will be displayed (see Figure 2-59) and the currently selected variables will be highlighted.
3. Choose the desired variable group. For humans, the available groups are:
4. Kinematics
5. Joints
6. Contacts
7. Belts
8. Airbags

For vehicles, the variable groups are:

- Kinematics
- Kinetics
- Accelerometers
- Damage
- Tires
- Wheels
- Inter-vehicle Connections
- Driver
- Contacts
- Belts
- Airbags

- After selecting the variable group, choose any required subgroups (e.g., Axle Number, Side, Inner/Outer for tire variables; Segment Name, Ellipsoid Name, Contact Name for contact variables). Once the required subgroups are selected, a multiple-selection list box will be displayed, showing all the variables available for selection in the chosen group.
- Choose the desired variables from the multiple-selection list box. Selected variables are highlighted. Click on any highlighted variables to deselect them.

> **NOTE:** The list box includes the name of every HVE simulation variable. If you choose a variable the current simulation model does not calculate, the Variable Selection list box will automatically deselect it.

1. Repeat the above steps for each variable to be displayed in the Key Results window.
2. Press *OK*.

The selected variables will be added to the existing list of variables in the Key Results window.

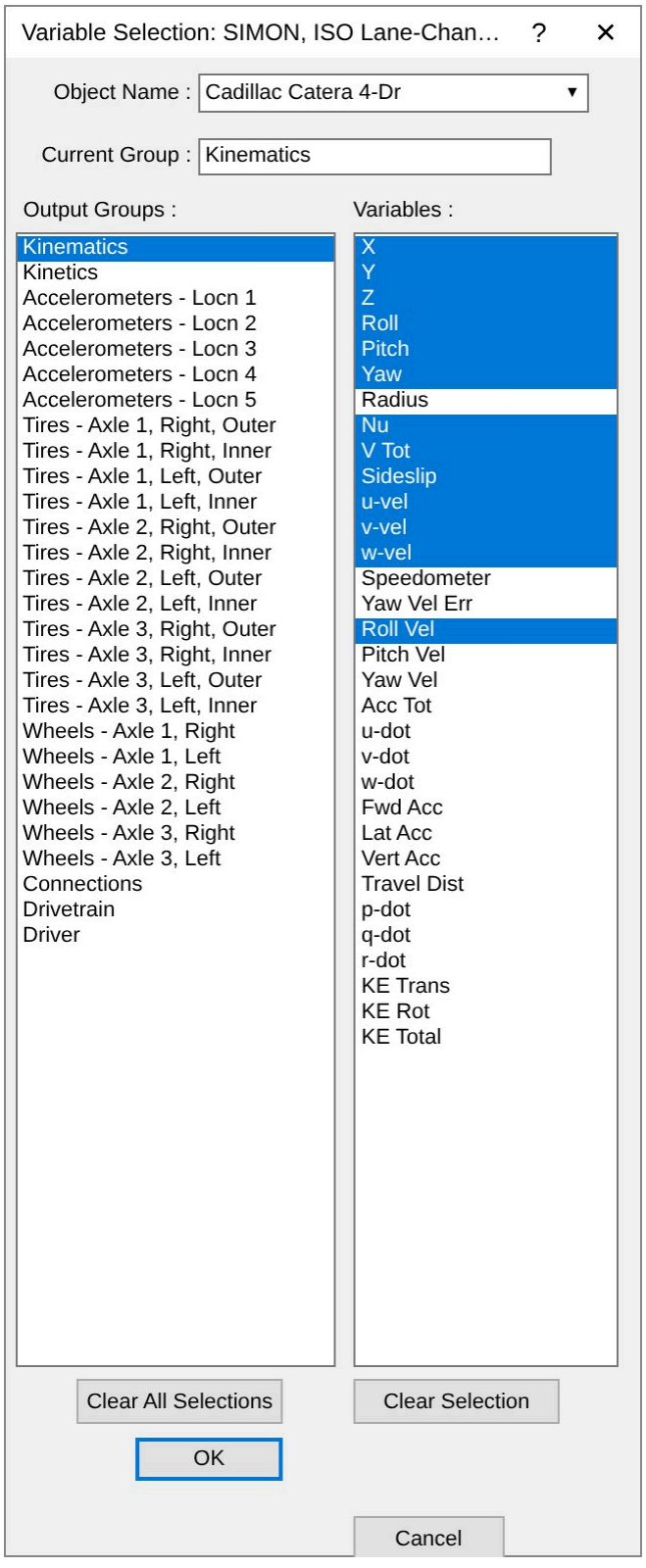


Figure 2-59: The Variable Selection dialog.

Coordinate Axes

The Coordinate Axes option allows the user to display the X,Y,Z coordinate system for each human or vehicle, as well as the environment (see Figure 2-60). The coordinate axes are shown in all modes, and serve as a useful reminder when

dimensions are being assigned and positions are being entered.

To display the coordinate axis system for each object, check *Show Axes* on the Options menu. To remove the axes, uncheck *Show Axes*. The user may choose to turn off the coordinate axes once in Playback mode in order to reduce the level of detail.

> **NOTE:** The Axes option is a toggle.

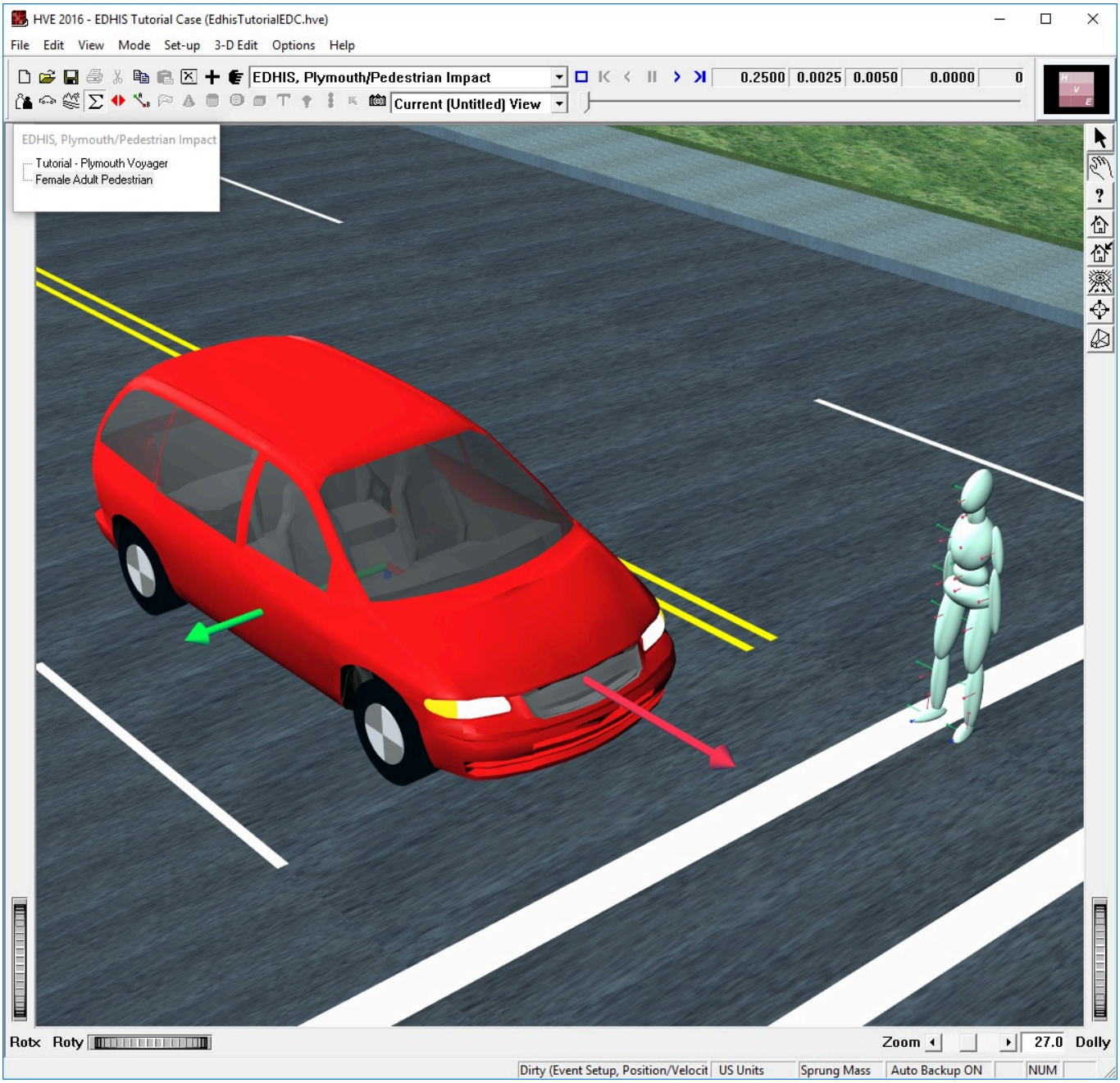


Figure 2-60: The Show Axes option.

Contact Surfaces

Contact surfaces are the physical surfaces of humans and vehicles which interact to produce forces during impact. For humans, the contact surfaces are ellipsoids attached to each human segment. These ellipsoids may be assigned or edited using the HVE Human Editor. For vehicles, these contact surfaces are flat planes attached to the vehicle's interior (dashboard, seat, floorboard and other interior surfaces) or exterior (bumper, grill, hood or other exterior surface). The viewer in Figure 2-61 includes a human and a vehicle with the Show Contacts option active.

The user may choose to make the contact surfaces visible while creating human ellipsoids during Human mode, and while creating contact surface planes during Vehicle mode, to ensure the surfaces are correctly positioned. The user may also choose to make the contact surfaces visible while in Event mode while setting up and executing events. This helps to ensure the correct ellipsoid vs contact surface interactions are being simulated.

While in Playback mode, the user may choose to turn off the contacts to reduce the level of detail and to make the simulation appear more realistic.

To display the human ellipsoids and vehicle contact surfaces for each object, check *Show Contacts* on the Options menu. To remove the contacts from the display, uncheck *Show Contacts*.

> **NOTE:** The Contacts option is a toggle.

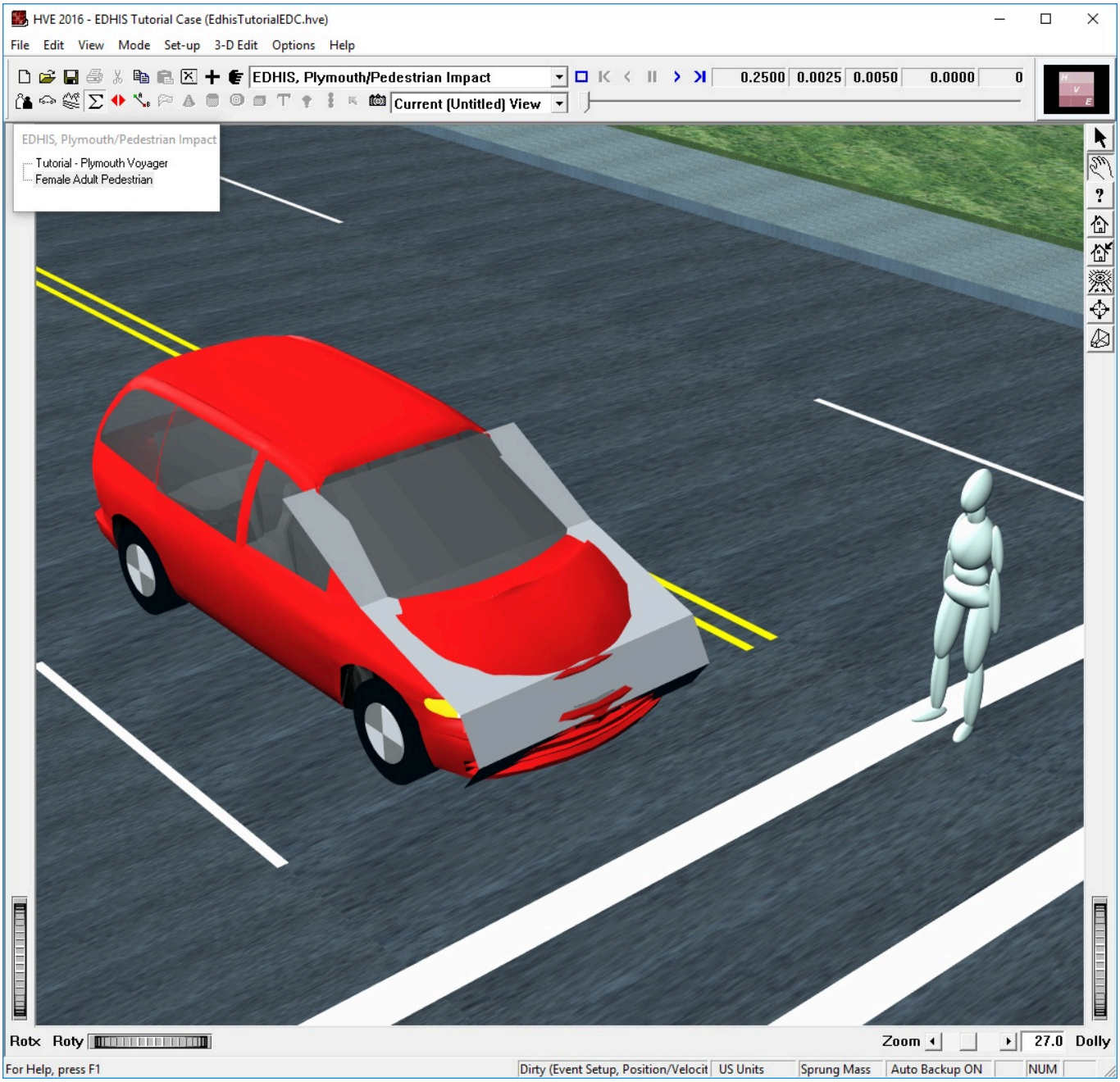


Figure 2-61: The Show Contacts option.

Velocity Vectors

Velocity vectors are represented by arrows drawn from each human segment CG and each vehicle CG which represent the magnitude and direction of the object's motion. These vectors are useful during Event mode while studying the motion during a simulated sequence. Figure 2-62 shows a skidding vehicle with the Velocity Vectors option active. The vectors may be turned off to reduce the level of detail in the scene.

> **NOTE:** The length of the vector is scaled in proportion to speed; the longer the vector, the higher the speed.

To display the velocity vectors for each human and vehicle, check *Show Velocity Vectors* on the Options menu. To remove the vectors, uncheck *Show Velocity Vectors*.

> **NOTE:** The Velocity Vectors option is a toggle.

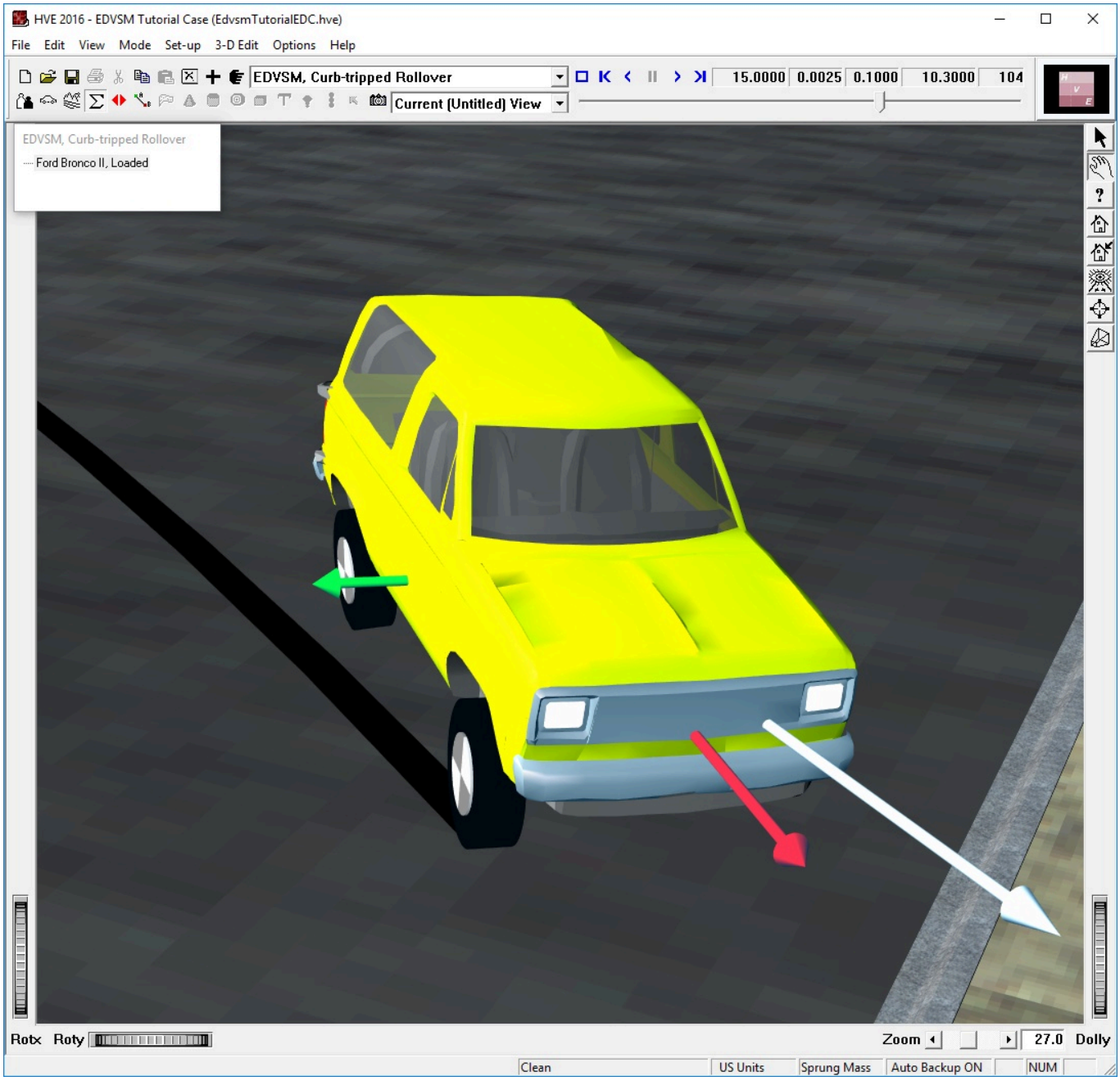


Figure 2-62: The Show Velocity Vectors option.

Skidmarks

Simulated skidmarks and scuffmarks are created by vehicle simulators. These marks on the environment surface are created by vehicle simulations at each output interval according to the value of Tire Skid (see Variable Selection dialog, Tires output group). Skidmarks are drawn for each tire according to the vertical tire load (increased tire load results in a darker mark) and longitudinal and lateral tire slip (increased slip results in a darker mark). The width of the mark is determined by the width of the tire.

The researcher may compare these simulated marks with any actual marks created during an accident sequence to help verify the accuracy of the simulation. A scene with simulated skidmarks is shown in Figure 2-63.

To display the simulated skidmarks for each vehicle's tires, check *Show Skidmarks* on the Options menu. To remove the simulated skidmarks from the display, uncheck *Show Skidmarks*.

> **NOTE:** The Skidmarks option is a toggle.
>
> **NOTE:** Some simulation models do not support a variation in the darkness of the skidmarks; they simply display solid black lines.

(updated: the current Options menu also provides a related "Show Tracks" toggle that displays the path of each tire, and "Show CG Paths" / "Show Accelerometer Paths" toggles for the paths of vehicle CGs and accelerometers.)

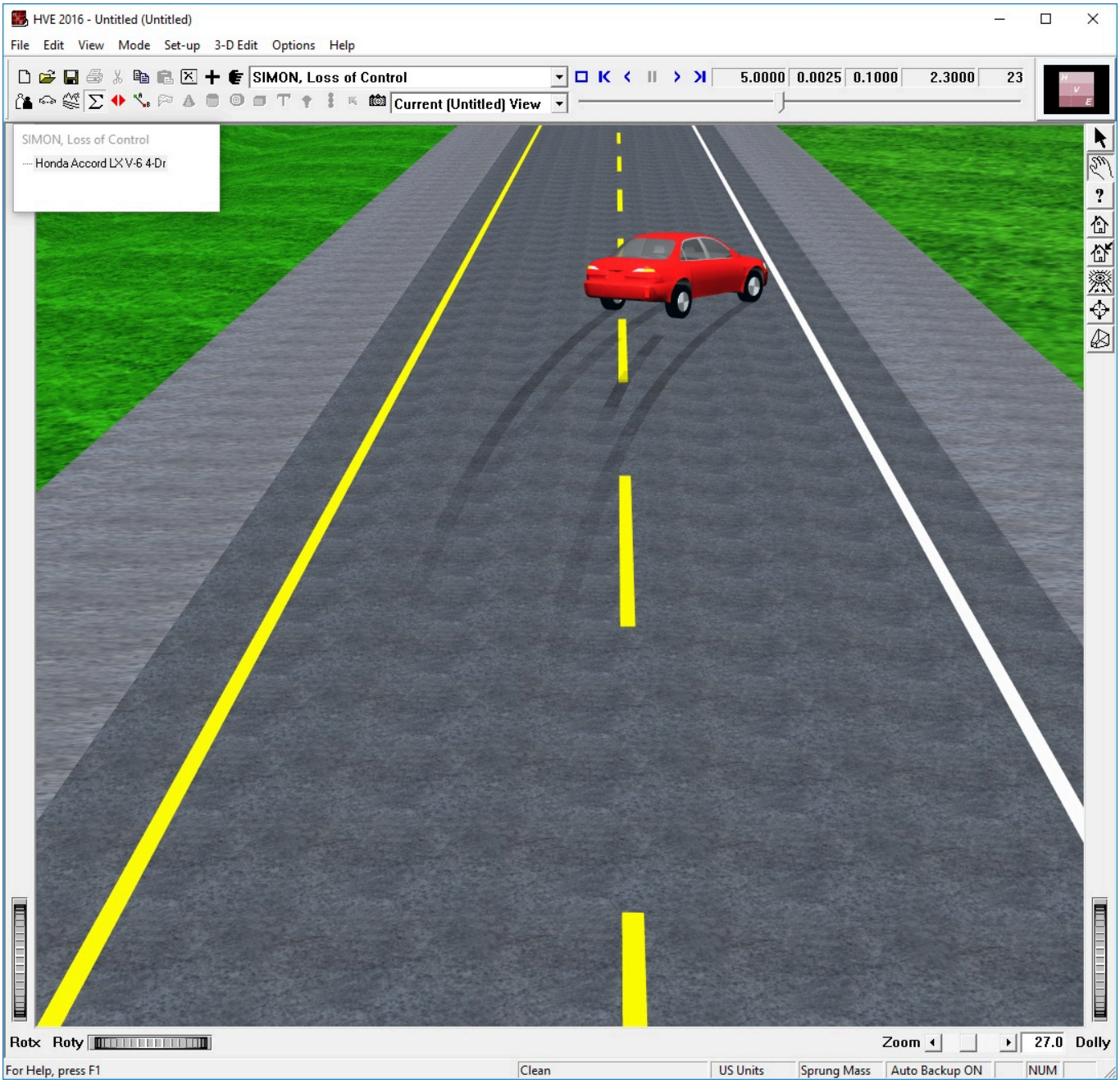


Figure 2-63: The Show Skidmarks option.

Target Positions

During Event mode, the user positions each human and vehicle in the environment. Up to eight positions may be supplied:

- Initial
- Begin Perception
- Begin Braking
- Impact
- Separation
- Point-on-curve
- End-of-rotation
- Final/Rest

Even though any or all of these positions may be supplied, they may not actually be used by the reconstruction or simulation model. For example, simulations only use the Initial position. All other positions are irrelevant to the simulation.

Then why supply them? Because, as the name suggests, targets provide feedback to the user regarding how well the simulated path matches the actual path. This is quite useful because, in general, the better the match between simulated and actual paths, the greater the level of confidence the researcher has about his/her conclusions.

Another important reason for entering target positions is the HVE Path Follower. In this case, the targets are used to define a path. Using this path, the simulation model determines the steering, throttle and braking driver inputs required to cause the vehicle to follow the path.

Target positions are displayed as translucent humans or vehicles, thus distinguishing them from the positions used by the simulation model. Target positions are most useful during Event mode, when the analysis is being conducted. A scene which includes target vehicles is shown in Figure 2-64.

To display the target positions for each human and vehicle, check *Show Targets* on the Options menu. To remove the targets from the display, uncheck *Show Targets*.

> **NOTE:** The Targets option is a toggle.

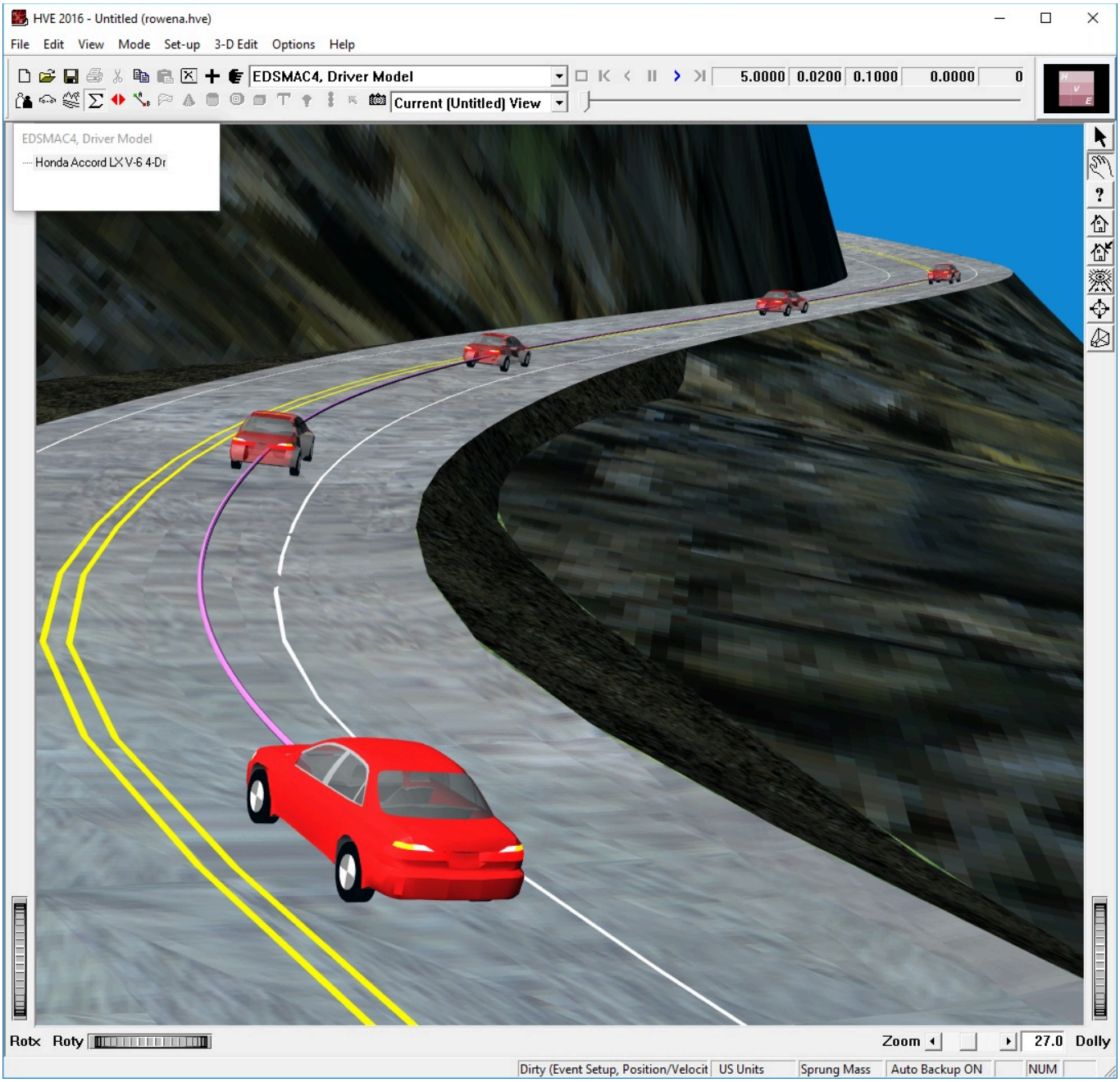


Figure 2-64: The Show Targets option.

Grid

The Grid option allows the user to display a set of evenly spaced lines (i.e., a grid) in the environment. It is used to provide spatial context while positioning objects during Event mode and in the 3-D Editor. The Grid dialog (see Figure 2-65 and the Grid dialog reference) allows the user to turn the grid on and off, as well as to define the grid spacing.

To display the grid and set the grid spacing, perform the following steps:

1. Choose *Grid...* from the Options menu. The Grid dialog will be displayed, showing the current grid status (on or off) and grid spacing.
2. If the Turn Grid On check box is not checked, click on it to turn on the grid.

3. Enter the desired grid spacing.
4. Choose *OK* to remove the Grid dialog.

> **NOTE:** The grid is displayed in the 3-D Editor viewers only.

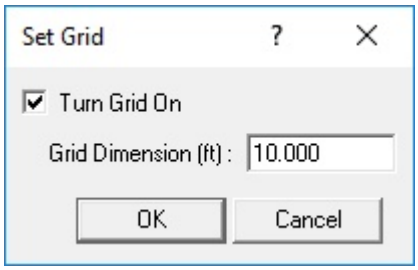


Figure 2-65: The Grid dialog.

Selecting Units

The Units dialog (see Figure 2-66 and the Units dialog reference) allows the user to specify different units (inches or millimeters, pounds force or Newtons, ...) for each type of object.

HVE incorporates a simple, yet extremely robust, units implementation. In general, it works like this:

Every object (human, vehicle, environment) and some component objects (tires, brakes, engine) has its own unit name for force, length and time (and all derivatives thereof). These are called Program Units. A list of these unit names may be found in Appendix IV.

For every program unit, the user may define the units of his/her own liking. These are called User Units.

For every unit name, HVE has a conversion factor between its program units and user units, as well as a human-readable label.

For example, a vehicle's forward velocity component has program units of in/sec. Although HVE includes several sets of user units, two possible options are miles per hour (having the user unit name "mph" and factor 17.6000, which converts from mph to in/sec), and meters per second (having the user unit name "m/s" and factor 0.0254, which converts from m/s to in/sec).

(updated: the m/s conversion factor to in/sec is 39.37; 0.0254 is the in-to-m factor as printed in the original manual.)

HVE includes two files, `units.us` and `units.si`, which contain two complete sets of user units. The Units dialog simply switches between these two files.

> **NOTE:** One file, `units.us`, contains typical user units for the US (Imperial) system of units; the other file, `units.si`, contains typical user units for the SI (Système International, or "metric") system of units.

To change the current system of units, perform the following steps:

1. Choose *Units...* from the Options menu. The Units dialog will be displayed, showing the current units (US or metric).
2. Click on the radio button to choose the other system of units.
3. Press *OK* to remove the Units dialog.

The selected units will be displayed.

> **NOTE:** The units currently displayed in any window while the units are changed will not be updated until the window is removed and redisplayed.

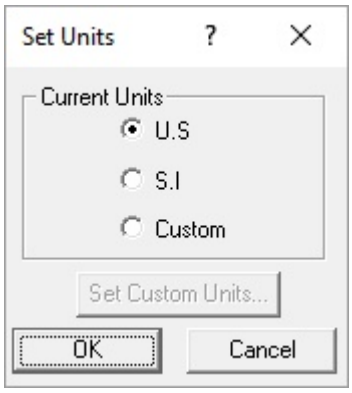


Figure 2-66: The Units dialog.

Render Options

The Render Options dialog (see Figure 2-67 and the Render dialog reference) allows the user to determine how objects are displayed and to choose the level of rendering quality. These options are important because high-quality rendering requires more time than low-quality rendering. There are times in which high-quality rendering is less important than speed (for example, during Event mode, when the goal is to conduct a thorough analysis), and there are other times when high-quality rendering is more important than speed (for example, during the final rendering of a Playback sequence to video). The various Render options are briefly described below.

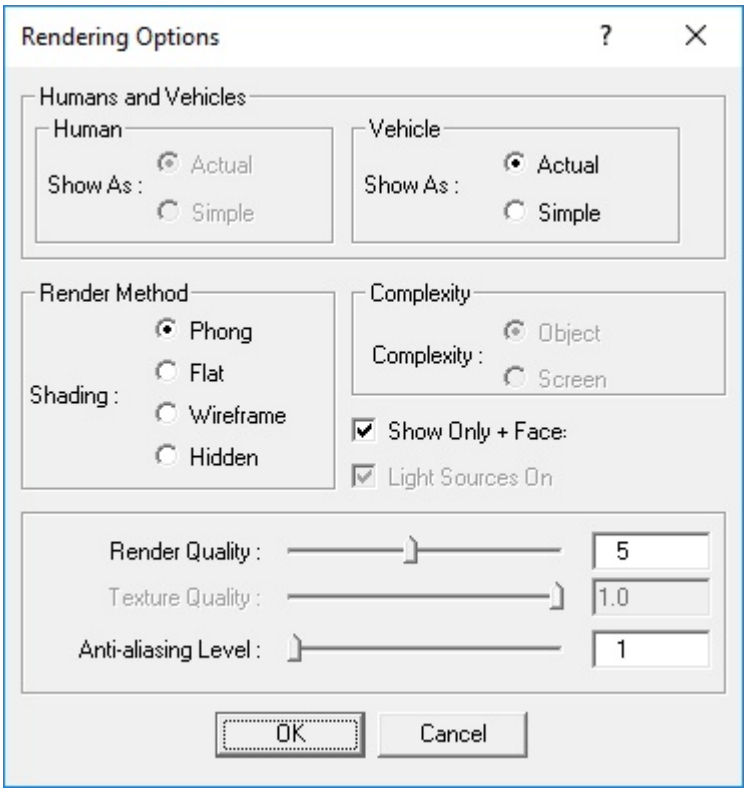


Figure 2-67: The Rendering Options dialog.

Show As

The Show As radio button has two choices: Actual and Simple. Choosing *Actual* causes HVE to display humans and vehicles using a complex 3-D model, if one is supplied. Choosing *Simple* causes humans and vehicles to be displayed as simple shapes: humans are shown using 15 basic body ellipsoids, and vehicles are shown using a simplified face set. *(updated: the current Render dialog provides separate Show As choices for humans and for vehicles. The Human Show As choices (Actual/Simple) are currently disabled — grayed out — in the dialog.)*

> **NOTE:** Because of the time required to render an actual human or vehicle containing several thousand polygons, you should choose the Simple option for all but your final runs. This option will greatly improve your productivity.

Shading

The Shading radio button has four choices: Phong, Flat, Wireframe and Hidden Line. Choosing *Phong* causes HVE to perform its highest level of lighting and shading calculations, and results in the most realistic appearance. Because it provides the necessary visual feedback crucial to the use of HVE, Phong is the preferred choice for most work. Choosing *Flat* causes HVE to perform minimal shading (no lighting) calculations, and results in a less realistic appearance than Phong shading. It is only slightly faster than Phong. Sometimes, Flat shading aids during Event mode or when using the 3-D Editor, when technical accuracy is important and the blending associated with Phong shading causes difficulty. The *Wireframe* and *Hidden Line* options are most useful as construction tools while using the 3-D Editor. In these modes, no lighting or shading of any surfaces occurs; in fact, surfaces are not drawn. Only lines representing the edges of each surface are drawn. Wireframe is considerably faster than Hidden Line because it requires no depth cueing.

> **NOTE:** Depth cueing is the process whereby closer objects obstruct the view of objects behind them.

Complexity

The Complexity radio button has two choices: Object and Screen. Choosing *Object* causes HVE to render every polygon of an object, regardless of how small or distant. Choosing *Screen* causes HVE to render fewer polygons for distant objects. *(updated: the Complexity choices (Object/Screen) are currently disabled — grayed out — in the dialog.)*

> **NOTE:** Logic dictates there is no reason to render 10,000 polygons if they occupy only 5 or 10 pixels!

Show Only + Faces

All polygons have a front side and a back side, as determined by the right-hand rule. The positive (+) side is determined by a counter-clockwise order of the vertices comprising the polygon. Choosing this option is a way to determine if one or more polygons has the vertices in the wrong order.

> **NOTE:** This is important to DyMESH. A force must be applied to the + side of a surface to cause damage.

Render Quality

The Render Quality slider has a range from 1 to 10. The default value is 5. Choosing a value less than 5 results in reduced tessellation of geometric shapes, such as spheres and cylinders. As a result, these objects are rendered faster, but begin to appear rather blocky. Choosing a value greater than 5 results in smoother objects, but they take longer to render.

A value of 5 results in a reasonable level of tessellation for most purposes. A value of 10 may be used for final production to video.

Anti-aliasing Level

The Anti-aliasing Level slider has a range from 1 to 10. The entered value determines how many times the scene is rendered. The default value is 1. Increasing the Anti-aliasing Level greatly improves the image quality, but at a rather significant increase in rendering time.

> **NOTE:** Because of the time required to re-render the scene several times, you should set the value to 1 until your final production to video.

To change the current rendering options, perform the following steps:

1. Choose *Render...* from the Options menu (Ctrl+R). The Rendering Options dialog will be displayed, showing the current status of each option.
2. Click on the radio buttons or check boxes to change any of the current options.
3. Set the desired Render Quality and Anti-aliasing Level using the sliders.
4. Press *OK* to update the selected options.

The viewer will be re-rendered using the new options.

(updated: the current Options menu also includes a "Shadows..." item for configuring rendered shadows; see the Options Menu reference.)

Simulation Controls

The Simulation Controls dialog (see Figure 2-68 and the Simulation Controls dialog reference) allows the user to set parameters affecting the execution of simulations. These options may be divided into two basic categories: Integration Timesteps and Termination Conditions.

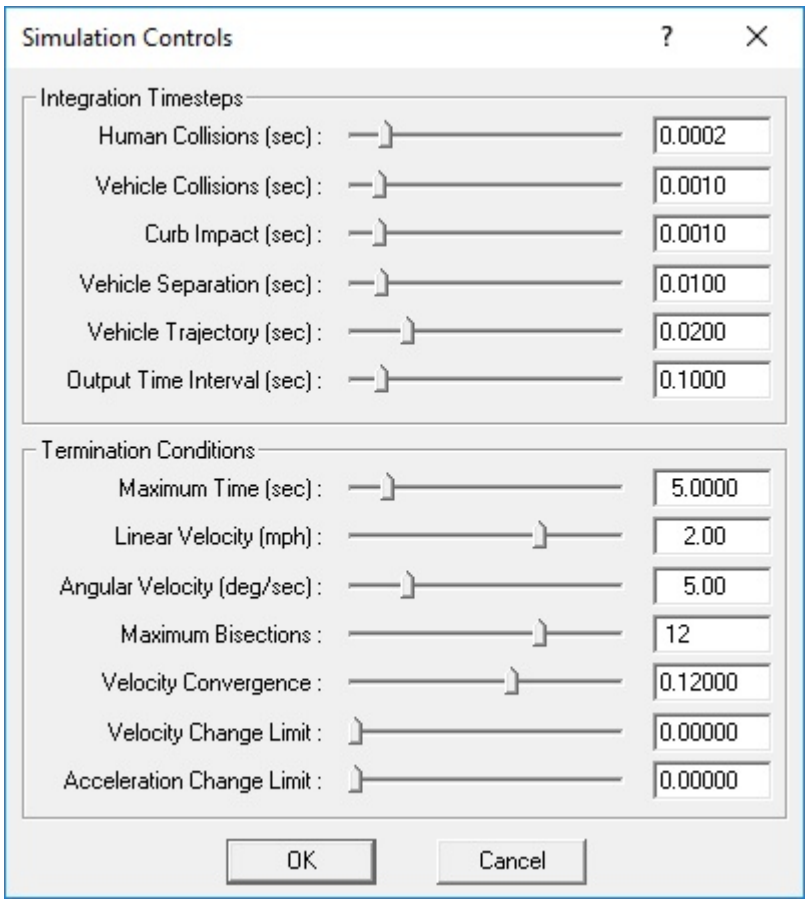


Figure 2-68: The Simulation Controls dialog.

Integration Timesteps

The Integration Timestep variables control the basic integration timesteps for the simulation model. HVE provides the following timestep controls:

- **Human Collisions** — This timestep may be used for occupant and pedestrian simulations. Because of the dynamic nature of the interaction between human ellipsoids and vehicle or environment contacts, this value should be relatively small. The default value is 0.0005 seconds.
- **Vehicle Collisions** — This timestep may be used during the collision phase of a collision between two vehicles. Because of the dynamic nature of motion during a collision, this value should be relatively small. The default value is 0.001 seconds.
- **Curb Impacts** — This timestep may be used during a simulation when the tire model detects a tire is mounting a curb. Because of the dynamic nature of the interaction between a tire and curb during a curb impact, this value should be relatively small. The default value is 0.001 seconds.
- **Vehicle Separation** — This timestep may be used for the separation phase of a collision simulation. Because of the dynamic nature of the motion just following impact, and because a secondary impact might occur, this value should be relatively small. The default value is 0.010 seconds.
- **Vehicle Trajectory** — This timestep may be used during the normal simulation of a vehicle on regular terrain. The default value is 0.050 seconds for 2-D simulations and 0.0025 seconds for 3-D simulations.
- **Output Interval** — This is the time increment at which the simulation sends the current simulation results to HVE for display and storage. The default value is 0.100 seconds for vehicle simulations and 0.010 seconds for human

simulations.

> **NOTE:** The Output Interval should be reduced if the output of the current event is to be used for the collision pulse of a human occupant simulation. 0.0050 seconds is a reasonable value. See Collision Pulse for further information.

These timesteps are sufficient for all known simulation models. Some simulators only need one or two of the above parameters; some require many more. The user should refer to the documentation for the individual simulation program for specific requirements.

Termination Conditions

Termination Conditions determine what causes a simulation to stop running. Any of several conditions may arise which dictate the end of a simulation. HVE provides the following Termination Condition controls:

- **Maximum Simulation Time** — The maximum simulation time determines how long the simulation is allowed to run before normal termination.
- **Termination Linear Velocity** — The velocity below which the simulated vehicle(s) are assumed to have stopped moving; a velocity of zero is assumed and the simulation terminates.
- **Termination Angular Velocity** — The angular velocity below which the simulated vehicle(s) are assumed to have stopped rotating; a velocity of zero is assumed and the simulation terminates.
- **Maximum Bisections** — For predictor-corrector (PC) numerical integration methods, the Maximum Bisections is the number of times the nominal timestep is allowed to be halved before the simulation terminates (normally an indication of abnormally high forces and resulting accelerations).
- **Velocity Convergence** — For predictor-corrector (PC) numerical integration methods, the Velocity Convergence parameter is the criterion used to determine if the current integration results for position and velocity are acceptably close to the predicted values. If the difference is greater than the velocity convergence parameter, the timestep is halved and the force (and resulting acceleration) calculations are repeated.
- **Velocity Change Limit** — For numerical integration methods which restrict the maximum value of the change in velocity during each timestep, the Velocity Change Limit sets that value. If exceeded, the timestep is halved and the force (and resulting acceleration) calculations are repeated.
- **Acceleration Change Limit** — For numerical integration methods which restrict the maximum value of the change in acceleration during a single timestep, the Acceleration Change Limit sets that value. If exceeded, the timestep is halved and the force (and resulting acceleration) calculations are repeated.

These Termination Conditions are sufficient for all known simulation models. Some simulators only need one or two of the above parameters; some require many more. The user should refer to the documentation for the individual simulation program for specific requirements.

> **NOTE:** Any integration weighting factors must be supplied by the simulator.

To change the current simulation controls, perform the following steps:

1. Choose *Simulation Controls...* from the Options menu (Ctrl+Y). The Simulation Controls dialog will be displayed, showing the current status of each simulation parameter.
2. Update the desired parameters using the sliders or by entering the value directly.

> **NOTE:** It's usually easier to enter the desired value directly in the field.

1. Press *OK* to update the selected Simulation Control parameters.

> **NOTE:** Changing the values during the middle of a simulation has no effect. You must restart the simulation for the new values to be used.

Playback Options

Playback options specify the timestep and display type during Playback mode. The timestep is simply the time increment between each frame of output. It is used for Variable Output, Variable Graphing, Trajectory Simulations, Playback Windows and Video Output.

> **NOTE:** The default playback output interval is set according to the current video setting: for NTSC video, the output interval is 0.0333 seconds (30 frames/sec); for PAL video, the output interval is 0.0400 seconds (25 frames/sec).

To change the current Playback Control options, perform the following steps:

1. Choose *Playback...* from the Options menu. The Playback Options dialog will be displayed, showing the current value of each option (see Figure 2-69).
2. Set the desired Playback Output Interval using the slider or by entering the desired value directly into the field.
3. Choose *OK* to update the current Playback Controls.

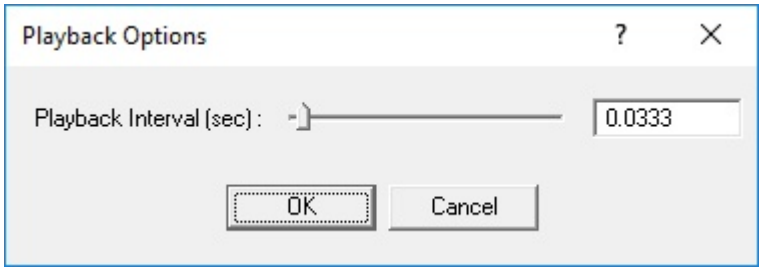


Figure 2-69: The Playback Options dialog.

Calculation Options

Each calculation method (e.g., SIMON, EDVSM or EDVDS) may have one or more calculation-specific options not included elsewhere in the HVE simulation environment. An example is EDVDS: it has three tire modeling options. To accommodate these options, HVE provides a Calculation Options dialog written specifically for each reconstruction or simulation model. This dialog is accessed from the Options menu.

See the Calculation Options reference for descriptions of each model's Calculation Options dialog (EDCRASH, EDGEN, EDHIS, EDSMAC, EDSMAC4, EDVDS, EDVSM and SIMON; EDSVS and EDVTS do not have a Calculation Options dialog).

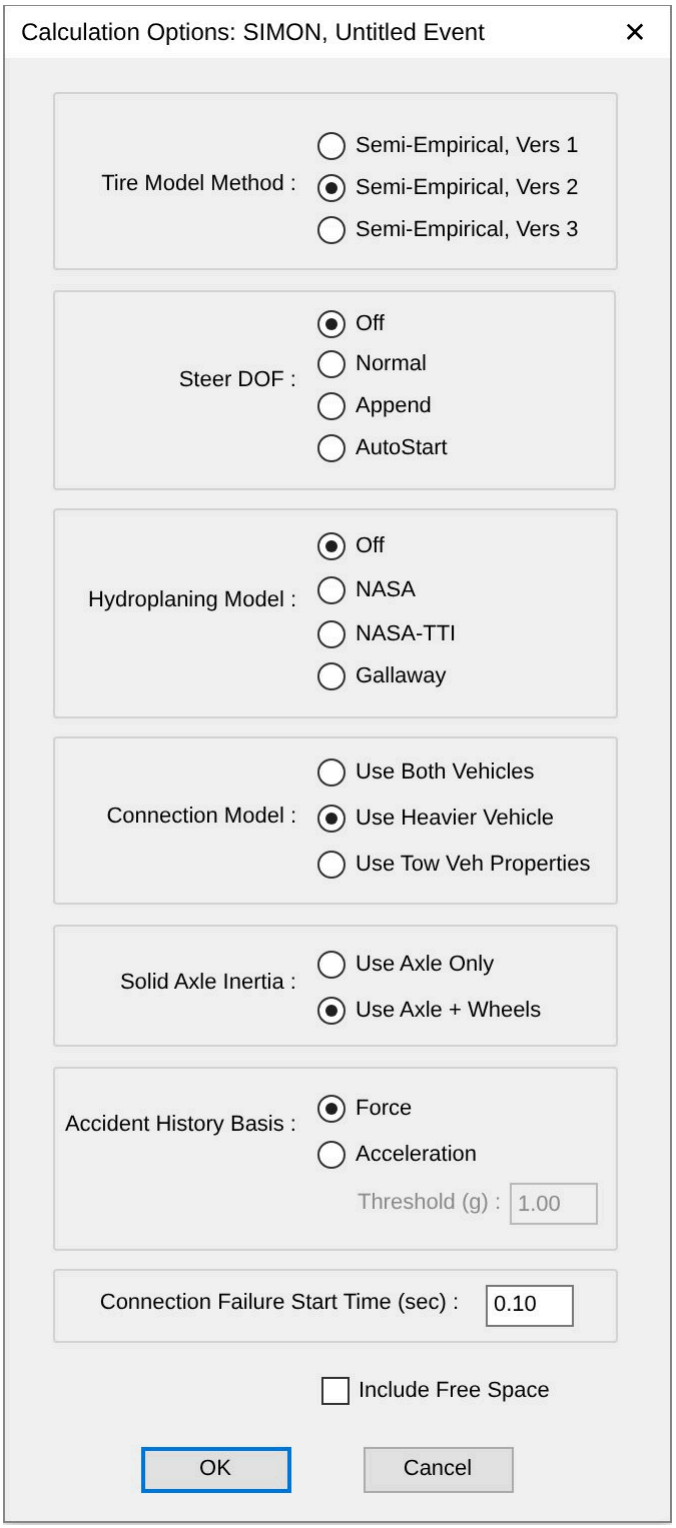


Figure 2-70: A Calculation Options dialog.

To view or edit the current Calculation Options, perform the following steps:

1. Create an event in the Event Editor.

> **NOTE:** You must be in Event mode to select the Calculation Options dialog. In addition, the Calculation Options dialog is not selectable unless an event has been created.

1. Choose *Calculation Options...* from the Options menu (Ctrl+J). The Calculation Options dialog for the current event will be displayed, showing the current value of each option (an example is shown in Figure 2-70).
2. Set the desired options using the sliders, radio buttons or data fields provided by the dialog.
3. Choose *OK* to update the current Calculation Options.

DyMESH Options

The DyMESH Options dialog (see Figure 2-71) turns the DyMESH collision model on for the event and sets how it is applied: whether contact with the environment and between connected vehicles is included, which DyMESH version and force-smoothing model are used, when contact detection begins, and how the contact search is sized.

> **NOTE:** See the Menu Reference, Options Menu, DyMESH Options for a detailed description of the DyMESH user options.

DyMESH simulations typically require a smaller integration timestep and output interval; set these in the Simulation Controls dialog.

After creating an event, the DyMESH Options dialog is selected by choosing

DyMESH...

from the Options menu.

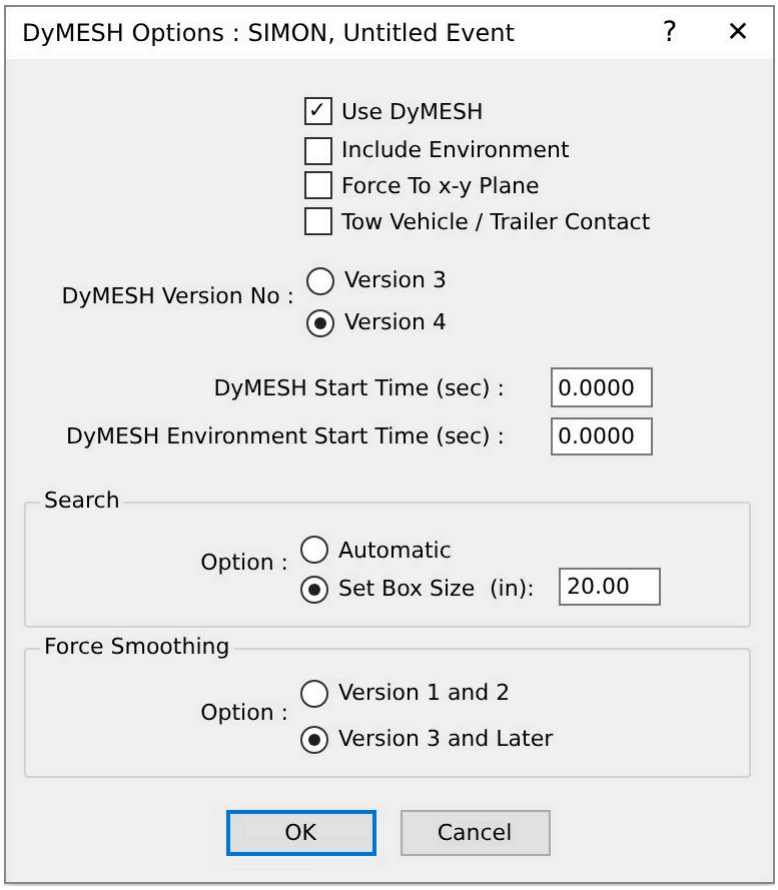


Figure 2-71: The DyMESH Options dialog.

Get Surface Information

All vehicle simulation models (e.g., SIMON, EDVSM or EDVDS) must calculate interaction forces between the tires and the terrain. This is accomplished by the simulation's tire model. Various options are available for calculating this interaction. The Get Surface Information dialog displays these options (see Figure 2-72), selectable from the Options menu.

While a simulation is executing, during each integration timestep the tire model searches through the database of terrain polygons to determine the characteristics of the polygon beneath each tire. The Get Surface Information dialog includes two options that affect this search:

- **Search Method** — There are three options:
- *From First Polygon*: The search starts at the top of the polygon database for every timestep.
- *From Previous Polygon*: The search starts by looking at the polygon found during the previous timestep. If it is not the correct polygon, the search spreads in both directions until the correct polygon is found.
- *By Elevation*: The entire polygon database is searched. The polygon closest to the underside of the undeflected tire is selected.

> **NOTE:** By Elevation is not implemented.

- **Search Direction** — Each polygon in the terrain has a surface normal whose direction is determined according to the right-hand rule. In general, it makes sense to be driving only on surfaces that have their normals facing upwards.
- *All Directions*
: Choose this option if you wish to include all terrain polygons, regardless of the direction of their surface normals. This option may be useful if the terrain includes ill-behaved surface normals (i.e., some normals pointing up, others pointing down).

  > **NOTE:** Choose this option if you are simulating a stunt driver performing a loop-the-loop.

- *Upward Facing Only*
: Choose this option if you wish only to include terrain polygons with upward-facing normals. This is the most common situation.

- *Z Component Greater Than*
: This option allows the user to assign the range for polygon normals. It is useful when curbs with near-vertical faces are included in the polygon database.

  > **NOTE:** This option may be useful for the Radial Spring and Sidewall Impact tire models.
  >
  > **NOTE:** In a vertical face, the Z component should be exactly zero. Because of rounding error when the surfaces were created, it is possible that a vertical face may have a small Z component.

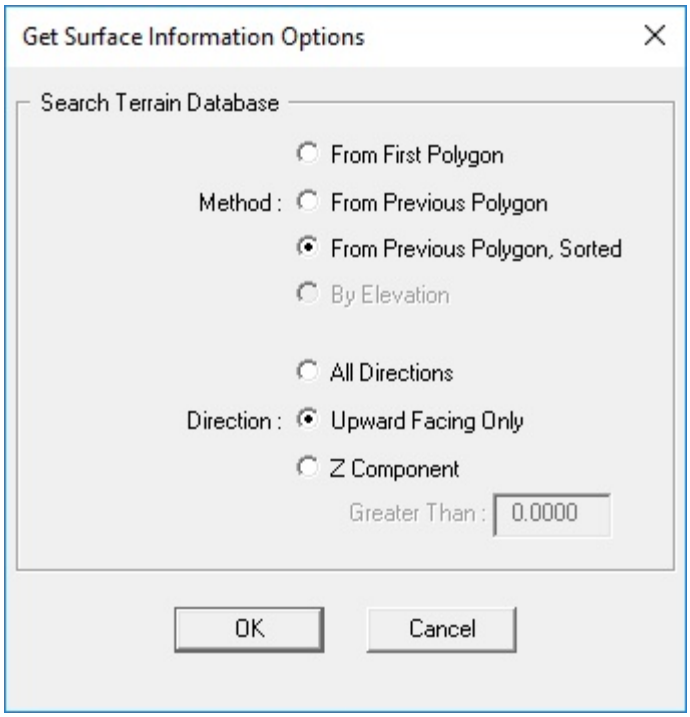


Figure 2-72: The Get Surface Information Options dialog.

To assign these options, perform the following steps:

1. Choose *Get Surface Info...* from the Options menu. The Get Surface Information dialog will be displayed, showing the current state of each option.
2. Click on the radio button to select the desired option.
3. Enter the range for surface normal Z Component, if applicable.
4. Choose *OK* to update the current Get Surface Information options.

User Preferences

HVE allows the user to set certain preferences to customize his/her HVE working environment. These preferences are saved when exiting HVE, so the next time HVE is used, the environment will be the same. The Preferences dialog is shown in Figure 2-73 (see the User Preferences dialog reference for the complete option list).

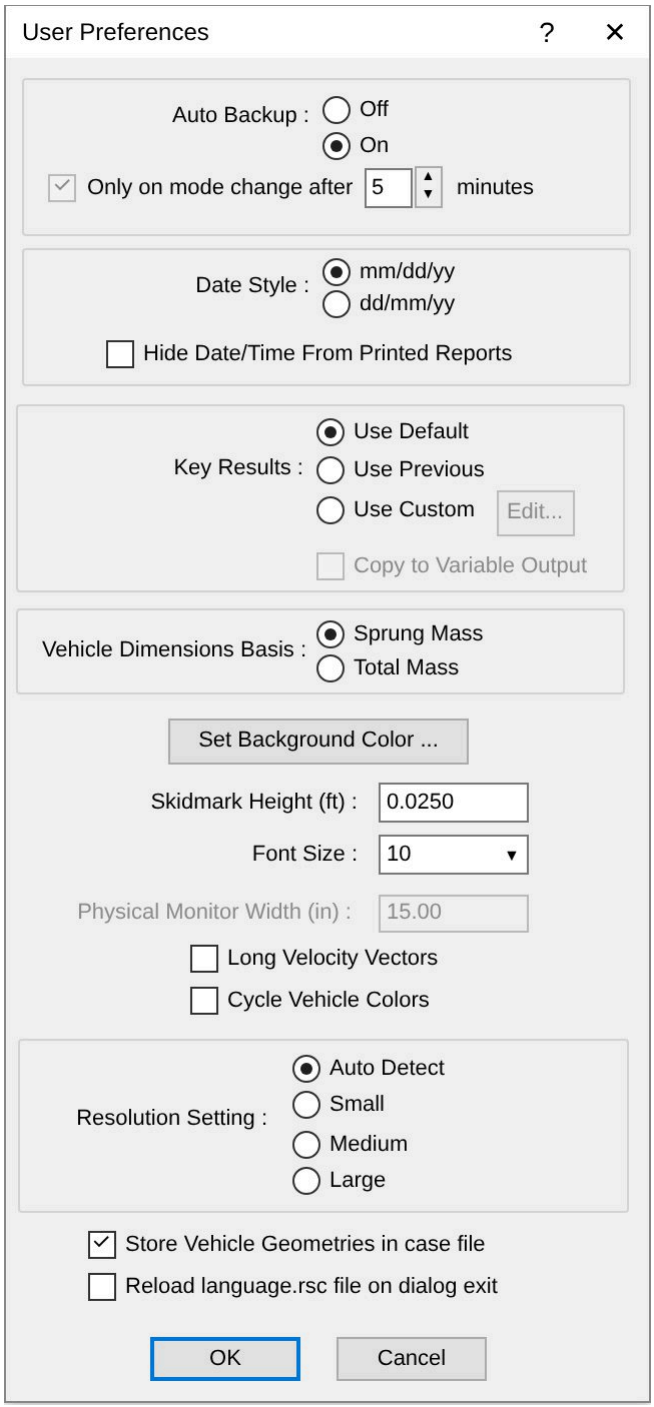


Figure 2-73: The User Preferences dialog.

The User Preferences dialog allows the user to set the following preferences:

- **Auto Backup** — The Auto Backup option allows the user to save his/her work at specified time intervals.

> **NOTE:** The backup file is created only when the user changes modes.

- **Date Style** — The Date Style radio button allows the user to switch between the common style (month/day/year) and military or European style (day/month/year). This field affects the Environment Editor's Date of Accident field.

- **Background Color** — Displays a color wheel allowing the user to set the background color in the Human and Vehicle Editor viewers.
- **Skidmark Height** — Skidmarks must be placed above the road surface in order to be visualized. If placed too close to the surface, rendering errors will occur when the surface is viewed from a distance. If placed too high, the skidmarks will be visually incorrect. It is a good idea to place the skidmarks as low as possible to the terrain, while also ensuring they are properly rendered.
- **Font** — Allows the user to change the size of the font displayed in Numeric Report windows and also in printed output reports.
- **Other options** — The dialog also includes Hide Date/Time From Printed Reports, Key Results, Vehicle Dimensions Basis, Physical Monitor Width, Long Velocity Vectors, Cycle Vehicle Colors, Resolution Setting, Store Vehicle Geometries in case file and Reload language.rsc file on dialog exit. Each is described in the User Preferences dialog reference.

To change the current User Preferences, perform the following steps:

1. Choose *Preferences...* from the Options menu (Ctrl+F). The User Preferences dialog will be displayed, showing the current status of each option.
2. Click on the radio buttons to change the Auto Backup status and Date Style.
3. If desired, edit the current skidmark height.
4. Choose *Background Color* to display the HVE color wheel. Click on the color wheel at the desired location, and use the intensity slider to set the lightness or darkness of the color.
5. If desired, change the current Font Size for output reports.
6. Choose *OK* to update the current User Preferences.

Getting Help

HVE has an on-line help system which provides user assistance as well as general information about human and vehicle dynamics and accident reconstruction. HVE's Help System is described below.



Figure 2-74: The Help Contents/Index.

Help Topics

The HVE Help Topics provides an index for important terms related to the use of HVE. In addition, the HVE Help Index includes portions of SAE J670e, Vehicle Dynamics Terminology [6.2]; SAE J885, Human Tolerance To Impact As Relates To Motor Vehicle Design [6.1]; SAE J224c, Collision Deformation Classification [6.3]; and SAE J1675, Accident Reconstruction Terminology [6.4]

(updated: J1675 has since been published)

.

To access the HVE Help Topics, perform the following steps:

1. Choose *Help Topics* from the Help menu. The Help Index dialog will be displayed (see Figure 2-74).
2. Choose an item from the list box. Follow the instructions in order to print out the help topic or to choose other topics.
3. Close the Help System when you are finished.

(updated: the current Help menu also contains a "User Manuals" submenu with direct access to the HVE, EDCRASH, EDGEN, EDHIS, EDSMAC, EDSMAC4, EDVDS, EDSVS, EDVSM, EDVTS, SIMON, Damage Studio, GATB and ReadDataFile user manuals, and an "Online Licensing" submenu with "Register User ID Code" and "Refresh Licenses" items.)

Technical Support

The

Tech Support

option on the Help menu provides the user with important information they will need to have available when contacting EDC for technical support. This includes:

- User Name
- Company Name
- User ID Number

The Technical Support dialog also displays your system software and licensing information. This includes:

- HVE Version and Release (service pack)
- System codes (Machine Id and User ID Code)
- License file information (Options and Versions)

A sample Technical Support dialog is shown in Figure 2-75.

> **NOTE:** This information is required for assistance when contacting EDC Technical Support regarding licensing issues. The report may be copied to the system clipboard and pasted into an email to EDC Technical Support staff.

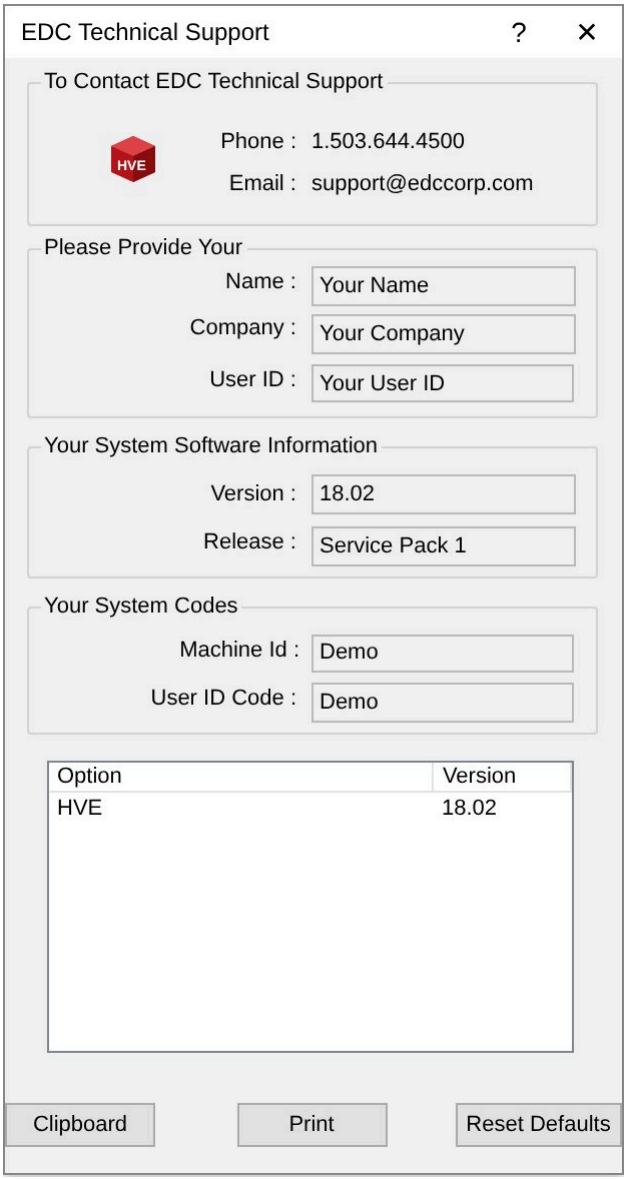


Figure 2-75: The Technical Support Help dialog.

About

About...

displays a dialog containing the current release information. This information includes the following:

- Program name, Service Pack and Version Number
- Build Date and Time

The About HVE dialog is shown in Figure 2-76.

For more information about getting Help, refer to the Help Menu in the Menu Reference section of this manual.

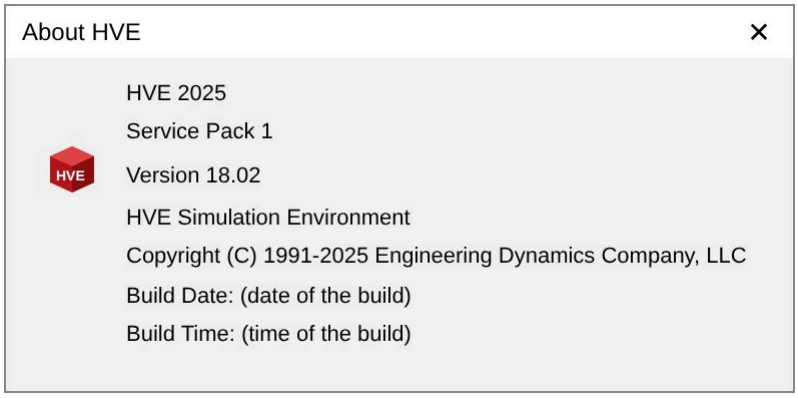


Figure 2-76: The About HVE dialog.

Video Interface

One of HVE's most powerful features is its integrated video interface.

> **UPDATED:** This section of the 2006 manual described output to video tape recorders (VTR) and S-Video/Composite Video devices with NTSC/PAL-based recording. The current version of HVE instead creates digital video files and image frame sequences directly. The Video dialog is opened with *Video Creator...* on the File menu, or with the *Video Setup...* button in the Playback Information window. The current Video dialog provides:
>
> - **Movie** output — an AVI movie file, with a user-selected name and output directory, an *Overwrite Existing Movie* option, a *Compressor* selection (e.g., Cinepak Codec, Full Frames/uncompressed, or MPEG compression with a selectable compression ratio), a *Resolution* selection (e.g., HDTV 1080p 1920x1080) and a *Recording Speed* slider with a frames-per-second field (e.g., 30 frames/sec normal, or slow motion rates).
> - **Single Frames** output — a sequence of individual image frames, with an *Overwrite Existing Frames* option.
>
> The Playback Information window shows the current Format, Compressor, Recording Size and Recording Speed in its Recording Information panel.
>
> The original procedures are preserved below for users of legacy HVE versions; the *Source*/*Destination* concept in the Playback Controller still applies.

The interface includes two important tools:

- Video Device Output
- Video Compression

These tools are made available in the Video Set-up dialog (see Figure 2-77). The following sections describe how to set up and use these tools.

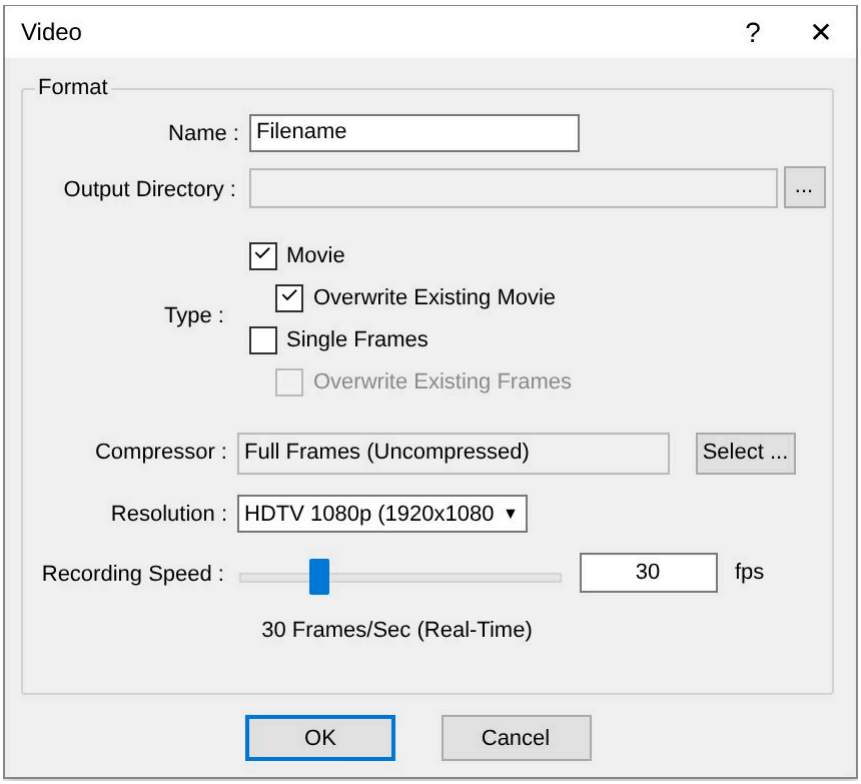


Figure 2-77: The Video dialog.

Video Device Set-up

The HVE Video Interface provides for output to a video device, either a video tape recorder (VTR) or S-Video/Composite Video. The video output device allows the user to create real-time or slow-motion sequences and route them to a video tape.

Before a video output device can be used, it must be installed. To install the video device, perform the following steps:

1. Choose *Video Set-up* from the File menu *(updated: now "Video Creator..." on the File menu, or the "Video Setup..." button in the Playback Information window)*. The Video Set-up dialog will be displayed (see Figure 2-77).
2. Click on the Device Name option list and select a device from the list.

> **NOTE:** Choose S-Video/Composite Video for real-time Video for Windows (AVI) recording or for output to video tape.

1. Press *OK* to install the device.

The device is now ready for use. Use the HVE Playback Controller (part of the Playback Editor) to record a simulation sequence to the video device. This process is described next.

Video Compression Set-up

If your computer system has hardware or software compression installed, the compressed video output can be copied to and read from disk. Video compression has two benefits:

- Entire sequences may be replayed from disk in real time. It is not necessary to record the sequence and play it back on a VCR before visualizing the sequence in real time.
- The sequence can be routed to the video device in real time.

Before a video compressor can be used, it must be installed. To install the video compressor, perform the following steps:

1. Choose *Video Set-up* from the File menu. The Video Set-up dialog will be displayed (see Figure 2-77).
2. Click on the Compressor Type option list and select a compressor from the list.

> **NOTE:** The user typically selects Full Frames (uncompressed) or Cinepak Codec as the compressor.

1. Press *OK* to install the device.

The compressor is now ready for use.



Figure 2-78: The Playback Controller with video source and destination.

Recording A Video

After setting up the required video options, as described in the previous section, the video devices are ready for use. To create real-time video output to tape or disk, perform the following steps:

1. Create a simulation using the HVE Event Editor.
2. Combine and edit the event sequence(s) using the HVE Playback Editor.

From here, several options are available, including:

- Recording to disk using video compression
- Playing back compressed video from disk
- Real-time recording from disk to the video device
- Saving and replaying compressed video files

All these options involve using the Playback Controller (see Figure 2-78) to select a video source and a video destination. These options are described below.

Recording To Disk Using Video Compression

Using video compression, the simulation in the Playback Window may be routed directly to the computer's hard disk, one frame at a time. To record the current sequence in the Playback Window, perform the following steps:

1. Click on the *Source* option list (in the Playback Controller), and choose the current Playback Window as the source.

2. Click on the *Destination* option list and choose the current video compressor type (e.g., AVI Compressor) as the destination.
3. Press *Play*. The simulation in the Playback Window will be routed to the computer's hard disk, one frame at a time.

> **NOTE:** Recording to disk takes only slightly longer than normally displaying the sequence in the Playback Window.

Playing Back Compressed Video From Disk

After recording the sequence to disk, it may be replayed in real time in the Playback Window. To play back the sequence in real time in the Playback Window, perform the following steps:

1. Click on the *Source* option list (in the Playback Controller), and select AVI as the source.
2. Click on the *Destination* option list and choose the Playback Window as the destination.
3. Press *Play*. The simulation will be routed directly from disk to the Playback Window. The sequence will be displayed in real time.

Real-time Recording From Disk To The Video Device

(legacy procedure — applies to video tape output in older HVE versions)

After recording the sequence to disk, it may also be routed directly to a special window designed to integrate with the computer's video subsystem for routing to a video recording device.

To record the sequence in real time, perform the following steps:

1. Click on the *Source* option list (in the Playback Window), and choose AVI as the source.
2. Click on the *Destination* option list and choose the video recording device (i.e., S-Video/Composite Video) as the destination.
3. Activate the computer's video subsystem.
4. Press the *Play* button on the Playback Controller to activate the special playback window.
5. Press the VCR's record button to begin recording.
6. Press *Play*. The simulation will be routed directly from disk to both the video recording device and the Playback Window. The sequence will be recorded and displayed in real time.

Saving and Replaying Compressed Video Files

HVE routes compressed images to a default file, named `default.<compressor type>` (e.g., `default.avi`). This file may be saved and played back at a later date. *(updated: in the current Video dialog the user chooses the movie filename and output directory directly.)* To save the current file, perform the following steps:

1. Choose *Video Set-up* from the File menu. The Video Set-up dialog will be displayed.
2. Choose *Save As*. A file selection dialog will be displayed, showing a list of previously saved files.
3. Enter a new filename or choose an existing one and press *OK*.

> **NOTE:** If an existing file is selected, HVE will ask if you wish to overwrite the existing file.

To replay a previously saved compressed file, perform the following steps:

1. Choose *Video Set-up* from the File menu. The Video Set-up dialog will be displayed.
2. Choose *Open*. A file selection dialog will be displayed, showing a list of previously saved files.
3. Choose a filename and press *OK*. The selected filename will be displayed as the compressed source file.
4. In the Playback Window, click on the *Source* option button and choose the compressed filename as the source.
5. Press *Play* to display the previously saved file.

For more information about using video, refer to Section Nine, Video Output.

End of Chapter 2. Return to the section table of contents.

HVE User's Manual — Section Two: Menu Reference

Updated Markdown edition of Section Two of the legacy *HVE User's Manual* (HVE Version 5, Seventh Edition, January 2006). Content has been revised for the current HVE release; differences from the printed edition are marked with italic *(updated: ...)* notes.

Menu items covered here that have detailed dialog pages elsewhere in this manual set are cross-linked rather than duplicated (see in particular 01-user-interface and 09-events-driver-controls).

Contents

Chapter	Menu	Manual pages
3	File Menu — New, Open, Save, Save As, Export Preview Case, Print, Print All, Video Creator, Export, FBX Export, Recent Files, Exit	3-1 … 3-14
4 (Part A)	Set-up Menu — Position/Velocity, Driver Controls (Throttle, Brake, Steering, Gear, Path Follower, Wheel Data)	4-1 … 4-31
4 (Part B)	Set-up Menu — Damage Profiles, Collision Pulse, Vehicle Mesh, Payload, Wheels, Accelerometers, Contacts, Restraints, Signals, Notes	4-32 … 4-79
5	View Menu — Toolbar, Status Bar, Set Camera, Overlays	5-1 … 5-10
6	Options Menu — display toggles, Grid, Units, Shadows, Render, Playback, Simulation Controls, Calculation Options, DyMESH, Get Surface Info, Preferences	6-1 … 6-38
7	Help Menu — Help Topics, User Manuals, Tech Support, About, Online Licensing	7-1 … 7-6

Notable updates relative to the 2006 printed edition

- **File menu:** "Video Set-up" has been replaced by **Video Creator**; **Export Preview Case**, **Export** (Export Variables dialog) and **FBX Export** were added.
- **Set-up menu:** "Vehicle Damage" is now titled **Damage Profiles**, "Mesh" is now **Vehicle Mesh**; **Signals** (traffic signal set-up) and **Notes** (event notes) were added; the Driver Controls dialog gained a **Lights** page.
- **Options menu:** added **Create DB**, **Show Tracks**, **Show CGs**, **Show CG Paths**, **Show Accelerometers**, **Show Accelerometer Paths**, **Show Position Sequences**, **Show Connections**, **Distance Tool**, **Show Path** and **Shadows**; **Add Playback Window** is no longer in the Options menu (playback windows are added from Playback mode).
- **Help menu:** added the **User Manuals** cascade (PDF manuals for HVE and all calculation models) and **Online Licensing** (Register User ID Code, Refresh Licenses).

Chapter 3: File Menu

HVE User's Manual — Section Two: Menu Reference. Updated edition, revised for the current HVE release.

The File Menu includes the following options:

- **New** — Begin a new case
- **Open** — Open a previously saved case file for review and editing
- **Save** — Save the current case file
- **Save As** — Save the current case file using a different filename
- **Export Preview Case** — Export the current case for use in HVE Preview Mode *(updated: added since the 2006 manual)*
- **Print** — Route the currently selected report window to a printer or plotter
- **Print All** — Route all of the currently selected report windows to a printer or plotter
- **Video Creator** — Create video output from a Playback window *(updated: replaces the former "Video Set-up" option)*
- **Export** — Export variable output results to a file *(updated: added since the 2006 manual)*
- **FBX Export** — Export the simulation as an FBX 3-D animation file *(updated: added since the 2006 manual)*
- **Recent File List** — Displays the last few cases worked on by the user
- **Exit** — End the current HVE session

These options allow the user to perform basic case file operations, as well as interfacing with peripheral devices. Each of these options is explained in this chapter.

NEW

Menu Option: NEW (Ctrl+N)

Purpose: Start a new case

Description: If the user is currently working on a case, choosing New from the HVE File Menu allows the user to close the current case (with or without saving it) and begin a new case. Choosing New also causes HVE to remove all current humans, vehicles and the environment, and start fresh, as if you had just started a new HVE session.

See Also: Open, Save, Save As

OPEN

Menu Option: OPEN (Ctrl+O)

Purpose: Open an existing case

Description: Choosing Open allows the user to load a previously created case into HVE. The user may simply review prior results, create new outputs or add new events with new humans and vehicles. The user may also change the

environment and re-execute each event. The main purpose of opening a previous case is to extend the user's previous work.

To Open an Existing Case:

1. Choose File from the Menu Bar. The File menu is displayed.
2. Choose Open. The Case File Selection dialog will be displayed.
3. Select the filename of the previously created case.
4. Press Open.

The selected case will be loaded and ready for further work.

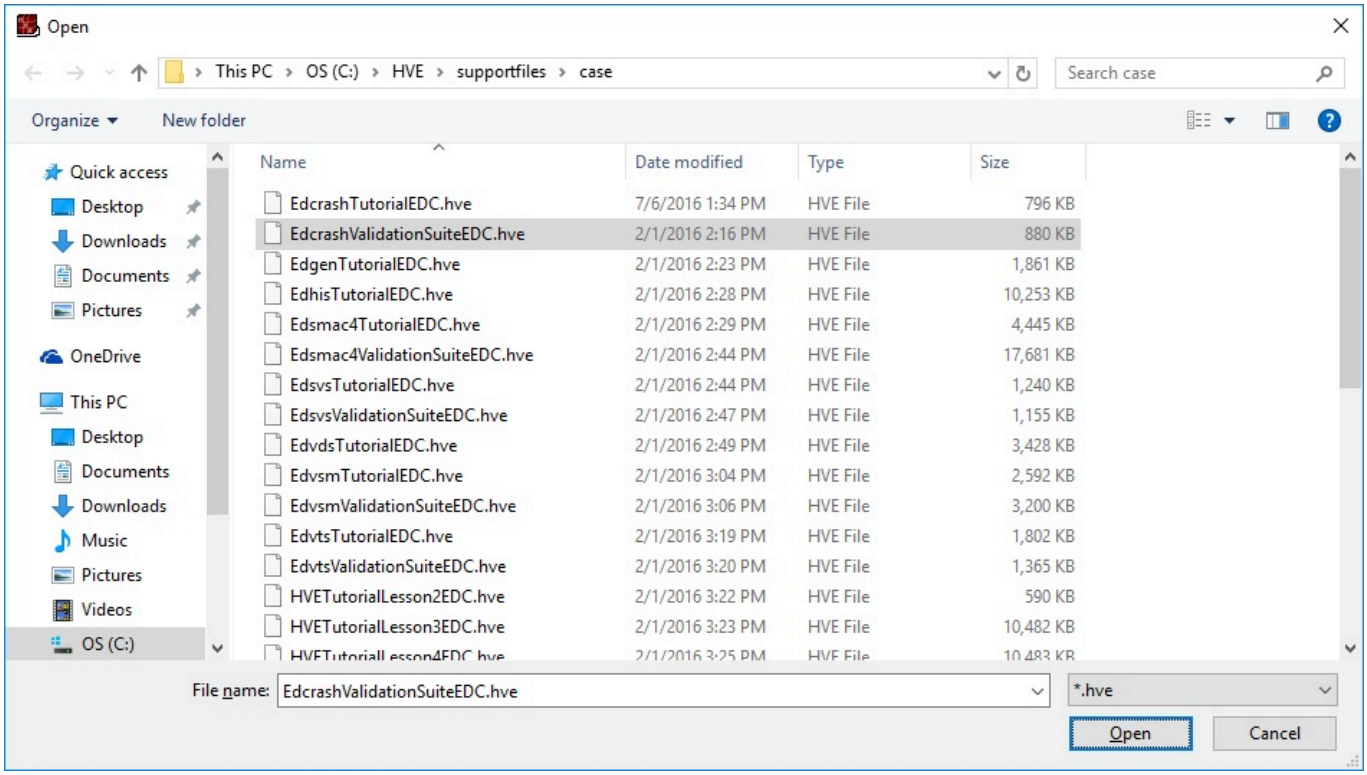


Figure 3-1: Use the Case File Selection dialog to open existing cases.

See Also: Case Files, Save, Save As

SAVE

Menu Option: SAVE (Ctrl+S)

Purpose: Save the current case

Description: Choosing Save from the HVE File Menu allows the user to save the current status of the case on the computer's disk storage system. This case file may then be loaded using Open. The practice of frequently saving the case file is also encouraged in order to protect against loss of information in the event of a computer hardware or software malfunction.

> **NOTE:** If the current case has not been saved previously, HVE will display the Save As dialog.

See Also: Save As, Open

SAVE AS

Menu Option: SAVE AS

Purpose: Save the current case using a user-specified filename

Description: Choosing Save As from the HVE File Menu allows the user to save the current status of the case on the computer's disk system according to a user-specified filename. This case file may then be opened at a later date for review and/or editing of the case file contents.

Choosing Save As also allows the user to enter or edit the Title of the current case file.

> **NOTE:** The Title is displayed as a heading on all HVE output reports.

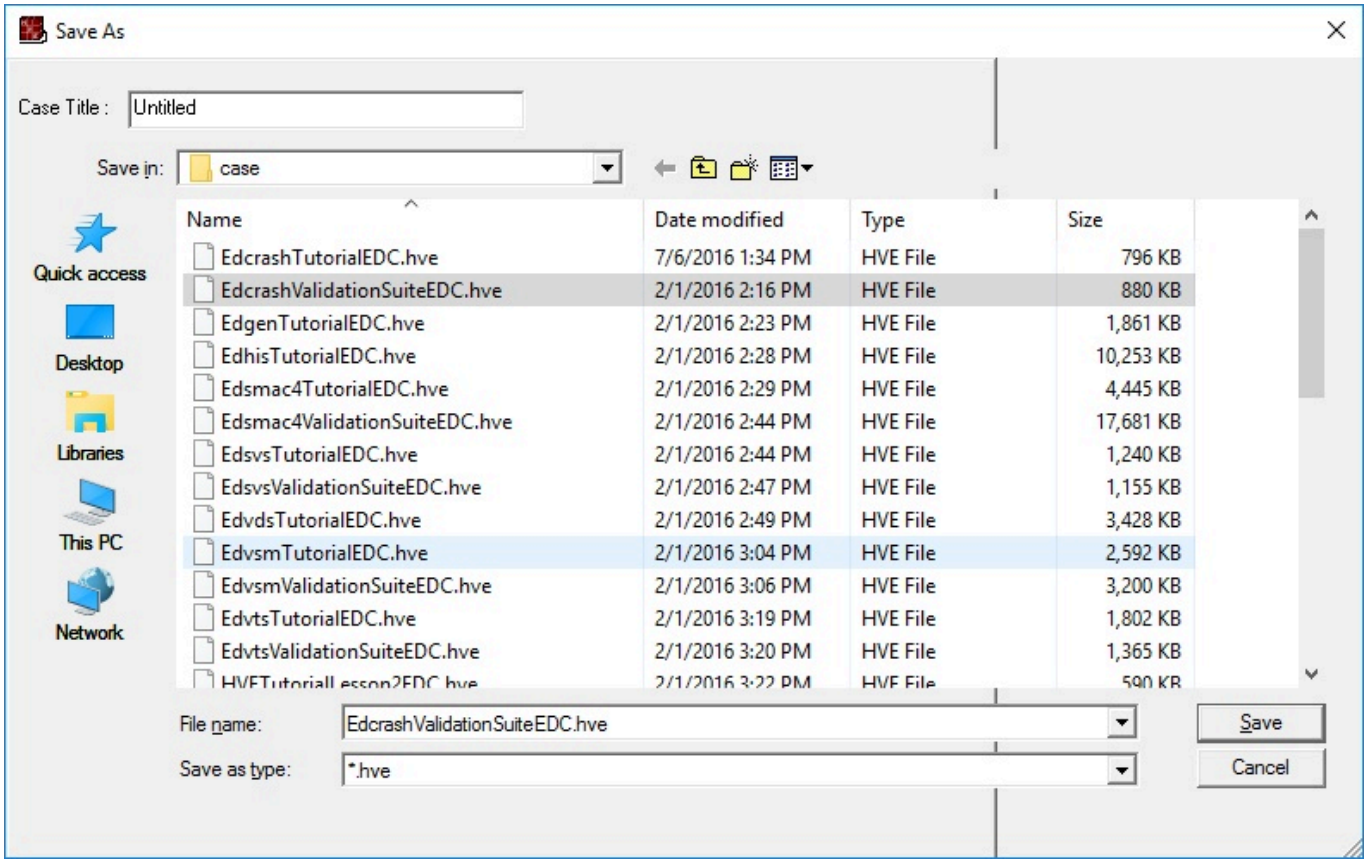


Figure 3-2: Save As Case Files dialog.

To Save an Existing Case by a User-specified Filename:

1. Choose File from the Menu Bar. The File menu is displayed.
2. Choose Save As. The Case File Selection dialog will be displayed.

3. Enter the filename.
4. Optionally, enter a Case Title.
5. Press Save.

The case will be saved. In addition, the Case Title and filename will be displayed in the HVE Main Menu Bar.

See Also: Save, Open

EXPORT PREVIEW CASE

(updated: this option was added after the 2006 manual)

Menu Option: EXPORT PREVIEW CASE

Purpose: Export the current case in a form that can be opened by users running HVE in Preview Mode

Description: Choosing Export Preview Case saves a copy of the current case for review by other users using the free HVE Preview Mode (playback-only) configuration. This allows colleagues or clients who do not have a full HVE license to view the case and its results.

PRINT

Menu Option: PRINT (Ctrl+P)

Purpose: Print the current object

Description: Choosing Print allows the user to print the contents of the selected Playback window. The Print Dialog is displayed, allowing the user to choose from the selected printer's possible print options by pressing the Properties button.

Selecting the check box for Print to file causes a File Selection Dialog to be displayed (see Figure 3-4), prompting the user for a filename. The File Selection dialog also includes a Save as type option list. Choose the desired format, enter a filename and press Save. The selected results are routed to a file according to the selected format and filename.

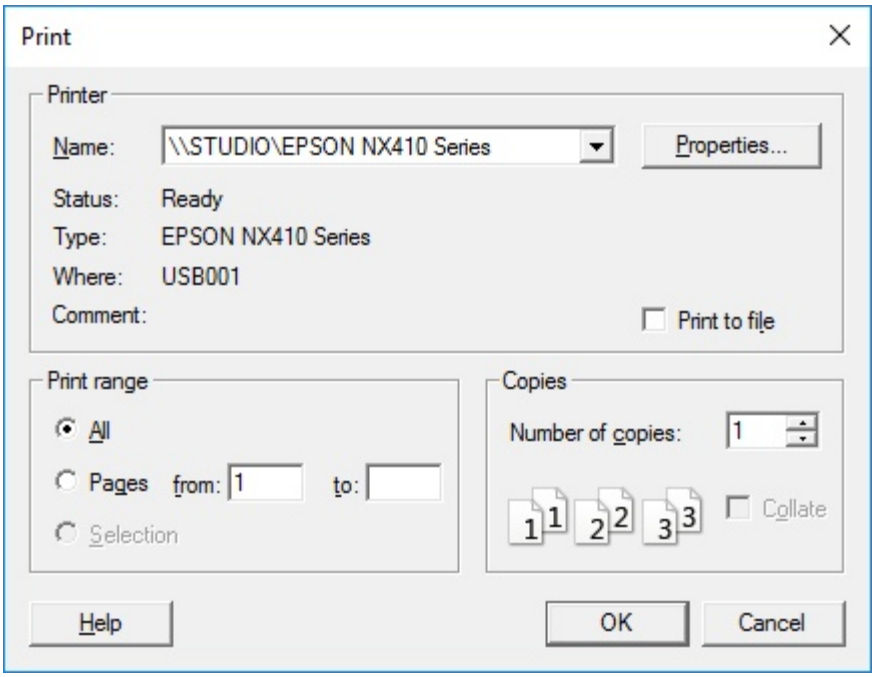


Figure 3-3: Use the Print Dialog to select the printer you wish to use. The Print Dialog also allows you to route output to a file. Routing selected Variable Output results to a file allows you to export the simulation results to other programs.

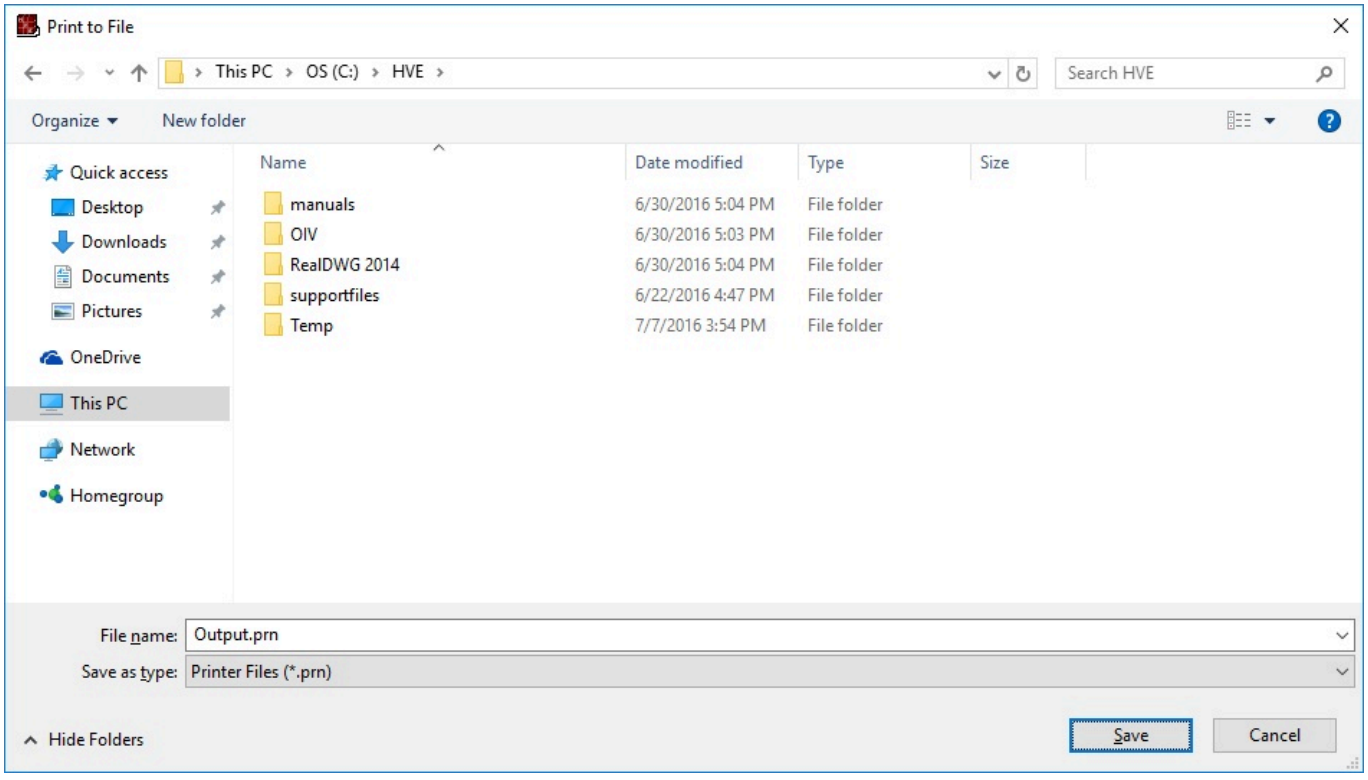


Figure 3-4: Print to File Selection dialog.

The following Playback windows may be printed:

- Accident History
- Audit Trail
- Damage Data

- Damage Profiles
- Driver Data
- Environment Data
- Event Data
- Human Data
- Injury Data
- Messages
- Momentum Diagram
- Program Data
- Site Drawing
- Variable Output Table
- Vehicle Data

> **NOTE:** Because the width of some reports (especially Variable Output) can vary, you should experiment to determine which page layout works best for a particular report.

The availability and contents of each report is calculation model-dependent. To print any of these windows, perform the following steps:

1. Using the HVE Event Editor, create the desired event(s).
2. Using the HVE Playback Editor, add a Report Window containing the desired report.
3. If there is more than one report displayed in the Playback Editor, choose the desired report by clicking on its dialog header, making it the current window.

> **NOTE:** The current window pops to the top of the desktop. 4. Choose Print. The Print dialog (see Figure 3-3) is displayed. 5. Select the available print options as described above. 6. Press OK to print the selected output report.
>
> **NOTE:** The selected report is printed on the system printer. Use the Windows Control Panel for information about selecting and configuring your system printer.

In addition to the above output windows available in Playback Mode, the Vehicle Editor contains many printable graphs, including:

- Anti-pitch vs Wheel Jounce/Rebound
- Camber vs Wheel Jounce/Rebound
- Half-track Change vs Wheel Jounce/Rebound
- Engine Power vs Speed
- Tire Fx vs Longitudinal Slip
- Tire Fy vs Slip Angle
- Tire Fy vs Camber Angle
- Tire Rolloff vs Slip

Exporting Results

The Print option provides a powerful, general purpose export utility for HVE simulation results. Exporting of any simulation results is easily accomplished using the Print option.

To export the desired results, perform the following steps:

1. Create a Variable Output report containing the desired results.
2. Select the File menu and choose Print. The Print dialog is displayed.
3. Choose Print To File.
4. Change the current printer name to Generic / Text Only.

> **NOTE:** If a Generic / Text Only printer is not included in the list of printers, you will need to install a Generic / Text Only printer using your Windows installation media. 5. Press OK. The Print To File dialog is displayed (see Figure 3-4). 6. Enter the desired filename.
>
> **NOTE:** The .txt extension is typically used. 7. Press OK. The selected report is saved in the /hve directory according to the requested filename.

The selected results are printed to the selected filename.

> **NOTE:** Refer to the documentation for the target program (e.g., Excel) for information about importing results into that program.

*(updated: the current File menu also provides a dedicated **Export** option that displays the Export Variables dialog, writing selected variable output directly to a file without going through the printer subsystem — see the Export section below.)*

See Also: Playback Editor, Vehicle Editor, Variable Output, Variable Selection

---

PRINT ALL

Menu Option: PRINT ALL

Purpose: Print all current objects

Description: Choosing Print All allows the user to print all open Report Windows in Playback Mode using the same functionality described for the Print menu option. The Print Dialog is displayed, allowing the user to choose from the selected printer's possible print options by pressing the Properties button.

> **NOTE:** Print All only prints text reports and viewer windows that the user has open in Playback Mode.

See Also: Print

VIDEO CREATOR

*(updated: the 2006 manual documented a "Video Set-up" option based on the legacy Video for Windows recorder/compressor interface. In the current version this has been replaced by the **Video Creator** option, which opens a dedicated Video Creator window used to produce video files (e.g., AVI) directly from a Playback window. The original Video Set-up description is retained below for reference.)*

Menu Option: VIDEO CREATOR (formerly VIDEO SET-UP)

Purpose: Set up and select various video output options

Description (legacy Video Set-up): Selecting Video displays the Video Set-up dialog, allowing the user to choose various video output options. These options are used during Playback Mode when routing a Playback window to the video output device. The available devices were:

- Video Recorder (pre-set to S-Video/Composite Video)
- Video Compressor

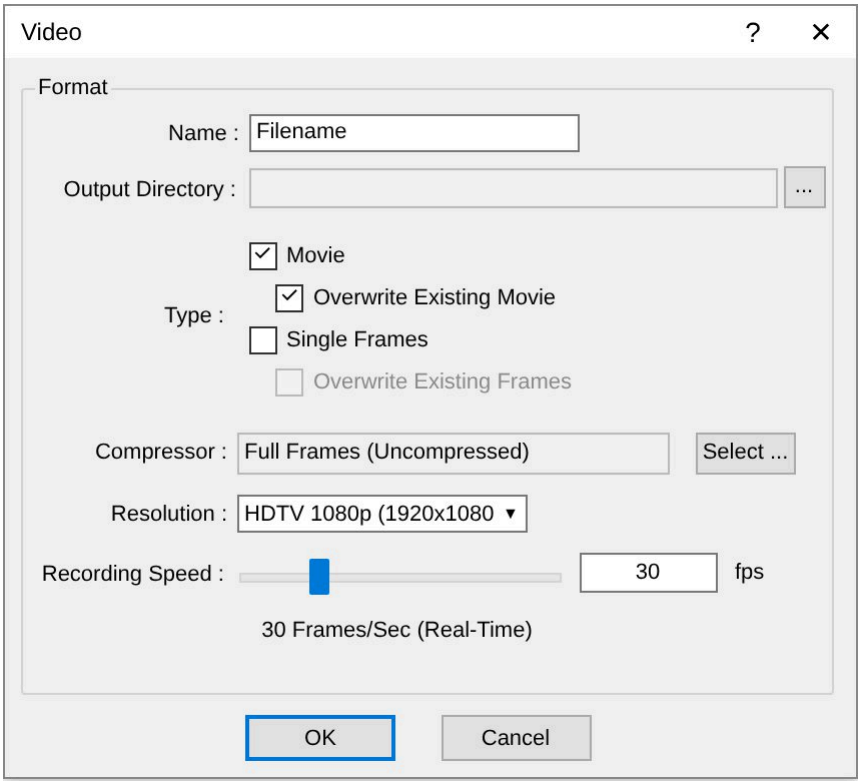


Figure 3-5: The Video dialog sets the movie and single-frame output options, including the compressor, resolution and recording speed.

Installing a Video Compressor

To install a video compressor, choose the appropriate compressor from the Compressor option list. This list will be different for every computer, depending on the codecs (COmpressor-DECompressor) provided with the Windows operating system and/or other video-related optional software programs.

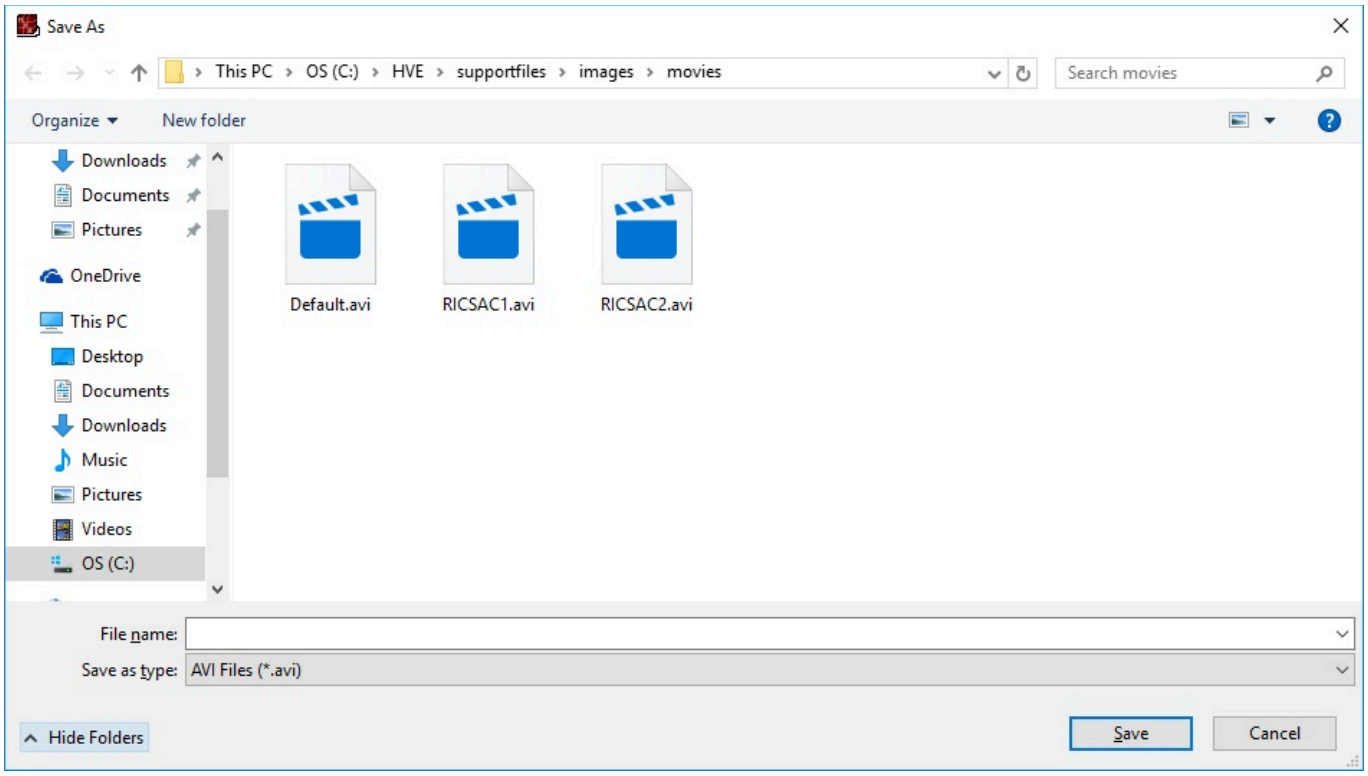


Figure 3-6: Compressed Video File Selection dialog allows the user to open and save video files using the video compression system.

The typical choice when creating a Video for Windows (AVI) movie is the Full Frame (Uncompressed) or the Cinepak codec.

> **NOTE:** If you have installed a custom video sub-system, you should use the compressor that is recommended for that sub-system.

Opening and Saving Video Files

HVE always routes the compressed video to a default file. To save this file, choose Save As using the Video Set-up dialog (see Figure 3-6). HVE will display a file selection dialog listing all the current compressed files. Choose an existing file or enter a new filename.

> **NOTE:** If you choose an existing file, HVE will ask you if you wish to replace it.

The Video Set-up dialog also allows the user to play previously saved video files. To play a previously saved file, choose Open; then choose the desired file for replay. The Video File Selection dialog, used for saving and opening video files, is shown in Figure 3-6.

See Also: Playback Editor, Playback Controller, Playback Source and Destination, Video Tools

---

EXPORT

(updated: this option was added after the 2006 manual)

Menu Option: EXPORT

Purpose: Export variable output results to a file

Description: Choosing Export displays the Export Variables dialog, which allows the user to select simulation output variables and write them directly to a data file (e.g., for import into a spreadsheet). When the export completes, the output pathname of the written file is displayed in the status bar.

FBX EXPORT

(updated: this option was added after the 2006 manual)

Menu Option: FBX EXPORT

Purpose: Export the simulation to an FBX file

Description: Choosing FBX Export displays the FBX Export dialog, which allows the user to export the current event's environment geometry and animated vehicle motion as an Autodesk FBX file for use in third-party 3-D animation and visualization packages.

RECENT FILE LIST

Menu Option: (most recently used case files)

Purpose: Quickly reopen a recent case

Description: The File menu displays the last few cases worked on by the user. Choosing a filename from this list opens that case, exactly as if it had been selected using the Open option.

EXIT

Menu Option: EXIT

Purpose: End the current HVE session

Description: Choosing Exit allows the user to end the current HVE session. HVE asks the user to save any changes made to the case since it was last saved. If the user chooses to save a case that has not been previously saved, HVE will display the Save As dialog.

See Also: New, Open, Save, Save As

Chapter 4: Set-up Menu (Part A — Position/Velocity, Driver Controls)

HVE User's Manual — Section Two: Menu Reference. Updated edition, revised for the current HVE release. This chapter is split into two files: Part A covers Position/Velocity and Driver Controls; Part B covers Damage Profiles through Contacts, plus the newer Signals and Notes options.

The Set-up menu is used by HVE's Event Editor. The Event Editor uses the Set-up menu for setting up events prior to execution.

The Set-up Menu includes the following options:

- **Position/Velocity** (Ctrl+P) — Allows the user to assign positions and velocities for each human and vehicle
- **Driver Controls** (Ctrl+D) — Allows the user to assign vehicle driver controls
- **Damage Profiles** (Ctrl+M) — Allows the user to assign vehicle damage profiles
- **Collision Pulse** (Ctrl+U) — Allows the user to assign vehicle collision pulses
- **Vehicle Mesh** — Allows the user to assign vehicle mesh options
- **Payload** — Allows the user to assign vehicle payloads
- **Wheels** (Ctrl+W) — Allows the user to assign vehicle tire blow-outs, wheel displacements, and brake adjustment and temperature
- **Accelerometers** — Allows the user to assign vehicle accelerometer locations
- **Contacts** (Ctrl+T) — Allows the user to select and remove interactions between selected human ellipsoids and vehicle contact surfaces
- **Restraints** (Airbags Ctrl+G, Belts Ctrl+B) — Allows the user to assign belt and airbag usage factors
- **Signals** — Allows the user to set up traffic signals for the current event *(updated: added since the 2006 manual)*
- **Notes** — Allows the user to attach text notes to the current event *(updated: added since the 2006 manual)*

This chapter explains how to use each of these options to supply various event-related inputs for humans and vehicles. Before supplying these inputs, a human or vehicle must first be selected. To select a human or vehicle, use one of the following methods:

- Select the desired human or vehicle from the Event Editor's Humans and Vehicles list, or
- Click on the desired human or vehicle.

> **NOTE:** Since trailers are not included in the Event Humans and Vehicles list, the latter method must be used for selecting trailers.

POSITION/VELOCITY

Menu Option: POSITION/VELOCITY (Ctrl+P)

Purpose: Assign position and velocity for the selected human or vehicle

Description: Selecting Position/Velocity from the Set-up Menu allows the user to assign positions and velocities for each human and vehicle in the current event. See also the dialog reference: Position/Velocity Dialog Box.

Positions and velocities may be entered for each of the following path locations:

- **Initial** — Position and velocity of the selected human or vehicle at the beginning of execution
- **Begin Perception** — Position and velocity of the selected human or vehicle at the beginning of perception (normally the initial part of the perception/reaction period for a driver about to begin an accident avoidance maneuver)
- **Begin Braking** — Position and velocity of the selected human or vehicle when the brakes are applied
- **Impact** — Position and velocity of the selected human or vehicle at the time of impact
- **Separation** — Position and velocity of the selected human or vehicle at the time of separation
- **Point-on-curve** — Position and velocity of the selected human or vehicle on a curved path following separation
- **End-of-rotation** — Position and velocity of the selected human or vehicle at the end of the rotating/lateral skidding phase
- **Final/rest** — Position and velocity of the selected human or vehicle at the end of the post-impact path

> **NOTE:** The human or vehicle need not have a zero velocity at this point. For example, the Final/rest position of a vehicle may be at the point of impact with a tree; the vehicle's velocity might be 20 mph at this point.

The position(s) and velocities required by the current reconstruction or simulation model vary from model to model. Refer to the User's Manual for the particular reconstruction or simulation program for a list of positions and velocities used by the model (see also Target Positions later in this section).

Coordinate System

Positions and velocities are assigned according to SAE Recommended Practice SAE J670e. In this coordinate system, the Z axis points down (in the direction of the gravity vector). See Appendix III and reference 6.2 for details.

Rotation Direction

According to the SAE coordinate system, rotation direction is defined using the right-hand rule. Thus, positive rotation is clockwise about a given axis. For example, looking down on the X-Y plane from above, a vehicle turning right will have a positive rotation direction.

It follows that the coordinate system convention has implications when entering a sequence of positions (e.g., Impact, Separation, Rest). When two successive positions are entered, the direction of rotation is defined by the difference in the heading angles. For example, if the heading angle at impact, Ψimp, is 35 degrees and the heading angle at rest, $\Psi rest$, is 80 degrees, then the difference from impact to rest is $\Psi rest - \Psi imp = 80 - 35 = +45$ degrees. The positive sign indicates the direction of rotation from impact to rest is positive (i.e., clockwise). In general, any rotation direction is defined as follows:

> $\Delta\Psi = \Psi n - \Psi n{-}1$

where:

- n = a position in a sequence of positions (e.g., rest)
- $n{-}1$ = the previous position in a sequence of positions (e.g., impact)

If $\Delta\Psi$ is positive, the direction is clockwise; if $\Delta\Psi$ is negative, the direction is counter-clockwise. The above applies to roll and pitch rotations as well as yaw (heading angle) rotations.

> **NOTE:** This definition is especially important to EDVAP/EDCRASH users. In that earlier program, the direction of rotation was entered directly by the user as 'Clockwise' or 'Counter-clockwise' or 'None'. In HVE, the direction of rotation is not entered directly; it is determined as shown above.

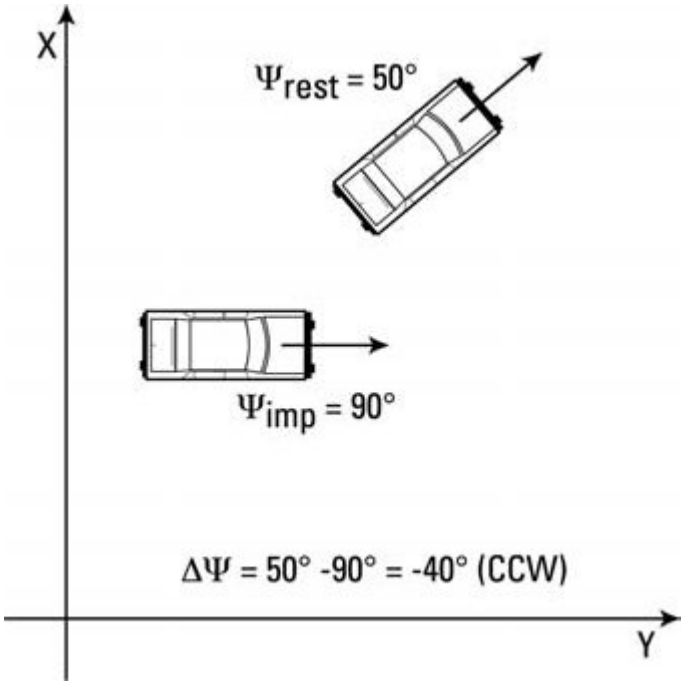


Figure 4-1: Examples of rotation direction for various position sequences.

To supply an object's position and velocity, perform the following steps:

1. Select the human or vehicle from the Event Editor dialog.
2. Choose Position/Velocity from the Set-up menu. The Position/Velocity dialog will be displayed (see Figure 4-2).
3. Click on the Path Location option list and select the desired location instance (e.g., Initial or Impact). The object will appear at its current location.

> **NOTE:** The default location is the origin. 4. Position the human or vehicle. Two different methods are available: - Enter the position and orientation data using the Position/Velocity dialog, or - Use the manipulators attached to the selected human or vehicle to drag the object to the desired location. 5. If desired, click on the Velocity is Assigned checkbox and enter the velocity using the Position/Velocity dialog.

Perform the above steps for the selected vehicle at each desired position instance. Each of these steps is described in detail in the next section.

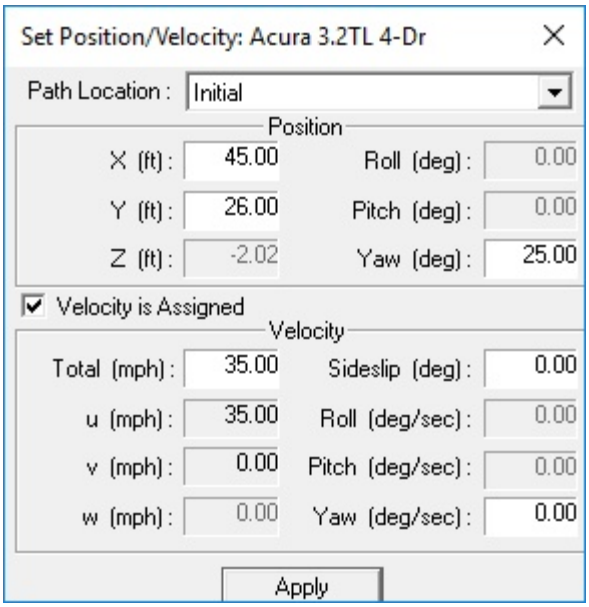


Figure 4-2: The Position/Velocity dialog is used for assigning positions and velocities for humans and vehicles.

Position/Velocity Dialog

When a human or vehicle is selected for positioning, its Position/Velocity dialog is displayed. The Position/Velocity dialog (see Figure 4-2) allows the user to enter the position and velocity according to the following protocol:

- **Single vehicles**, and the tow vehicle of vehicle-trailer combinations, are positioned relative to the earth-fixed coordinate system. X,Y,Z linear positions and roll, pitch, yaw orientations may be entered. Total velocity and sideslip angle are entered. Vertical velocity may be entered for 3-D models. For the user's reference, the forward and lateral velocity are calculated and displayed.
- **Trailers and dollies** are positioned relative to the vehicle pulling them; thus only relative roll, pitch and yaw positions and velocities are required.

> **NOTE:** Trailer and dolly names are not displayed in the selected objects list. To select a trailer or dolly, click on the desired object in the viewer. - **Human occupants** are positioned using the CG of the pelvis segment, and are positioned relative to the vehicle-fixed coordinate system. x,y,z linear positions and roll, pitch, yaw orientations of the pelvis may be entered. i,j,k linear velocities and roll, pitch, yaw angular velocities are entered relative to the pelvis segment's local coordinate system. - **Human pedestrians** are positioned using the CG of the pelvis segment, and are positioned relative to the earth-fixed coordinate system. X,Y,Z linear positions and roll, pitch, yaw orientations of the pelvis may be entered. i,j,k linear velocities and roll, pitch, yaw angular velocities are entered relative to the pelvis segment's local coordinate system. - For both human occupants and pedestrians, segments other than the pelvis are positioned using joint articulation angles relative to the upstream (i.e., closer to the pelvis) adjoining segment; thus only relative roll, pitch and yaw positions and velocities are required.
>
> **NOTE:** 2-D, yaw plane models do not use Z, roll or pitch information. HVE will automatically make these fields non-user-editable and calculate the values using AutoPosition.

Direct Manipulation

When a human or vehicle is selected for positioning, a set of manipulators is attached to it. These manipulators may be used to drag the human or vehicle to the desired location using the mouse.

> **NOTE:** This action is referred to as "drag-and-drop."

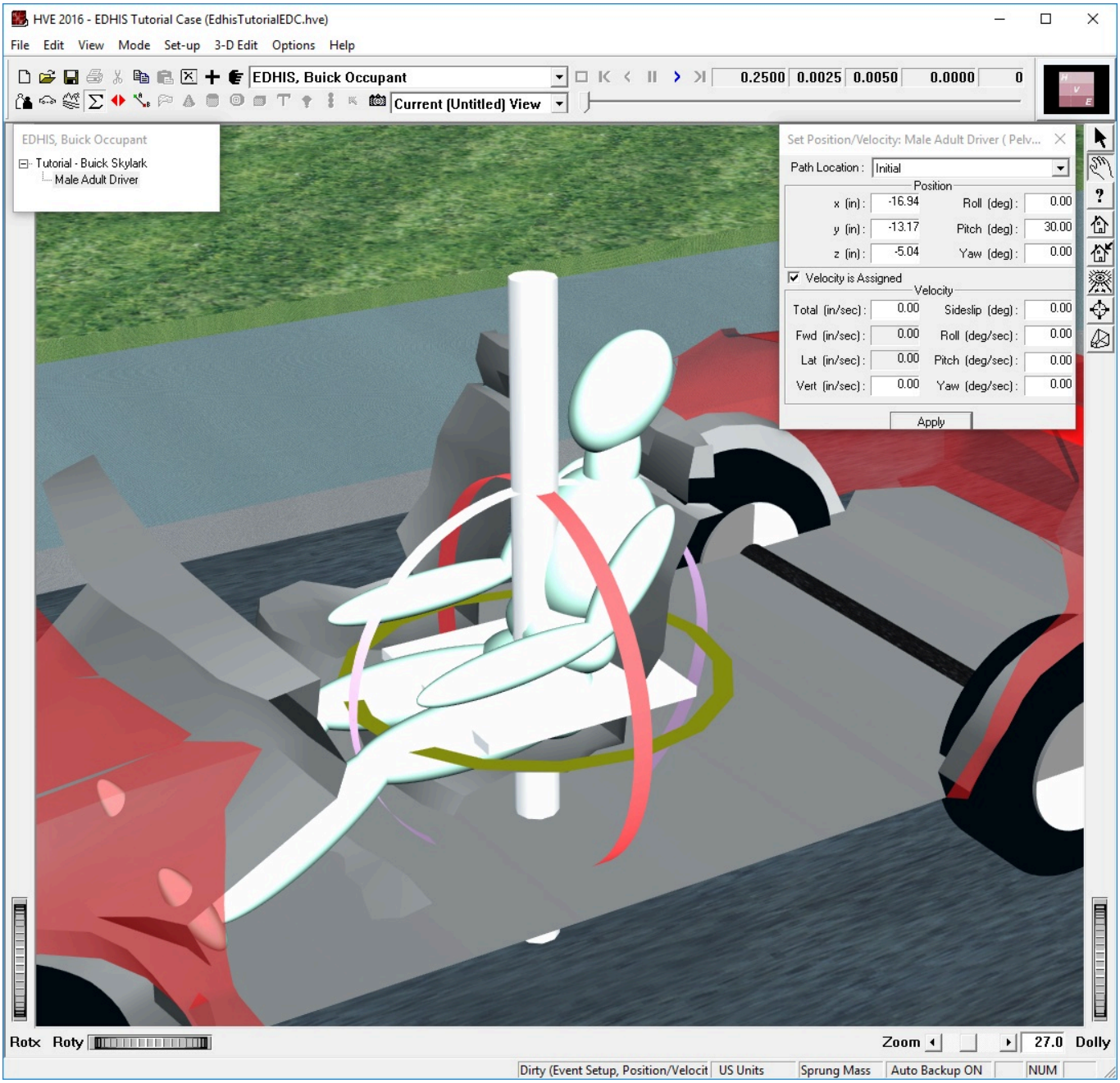


Figure 4-3: The Position Manipulator for humans has a set of controls (a) that move the human in the x-y plane, a vertical manipulator (b) for moving the human vertically, and a set of roll, pitch and yaw manipulators (c, d and e, respectively) for orienting the human.

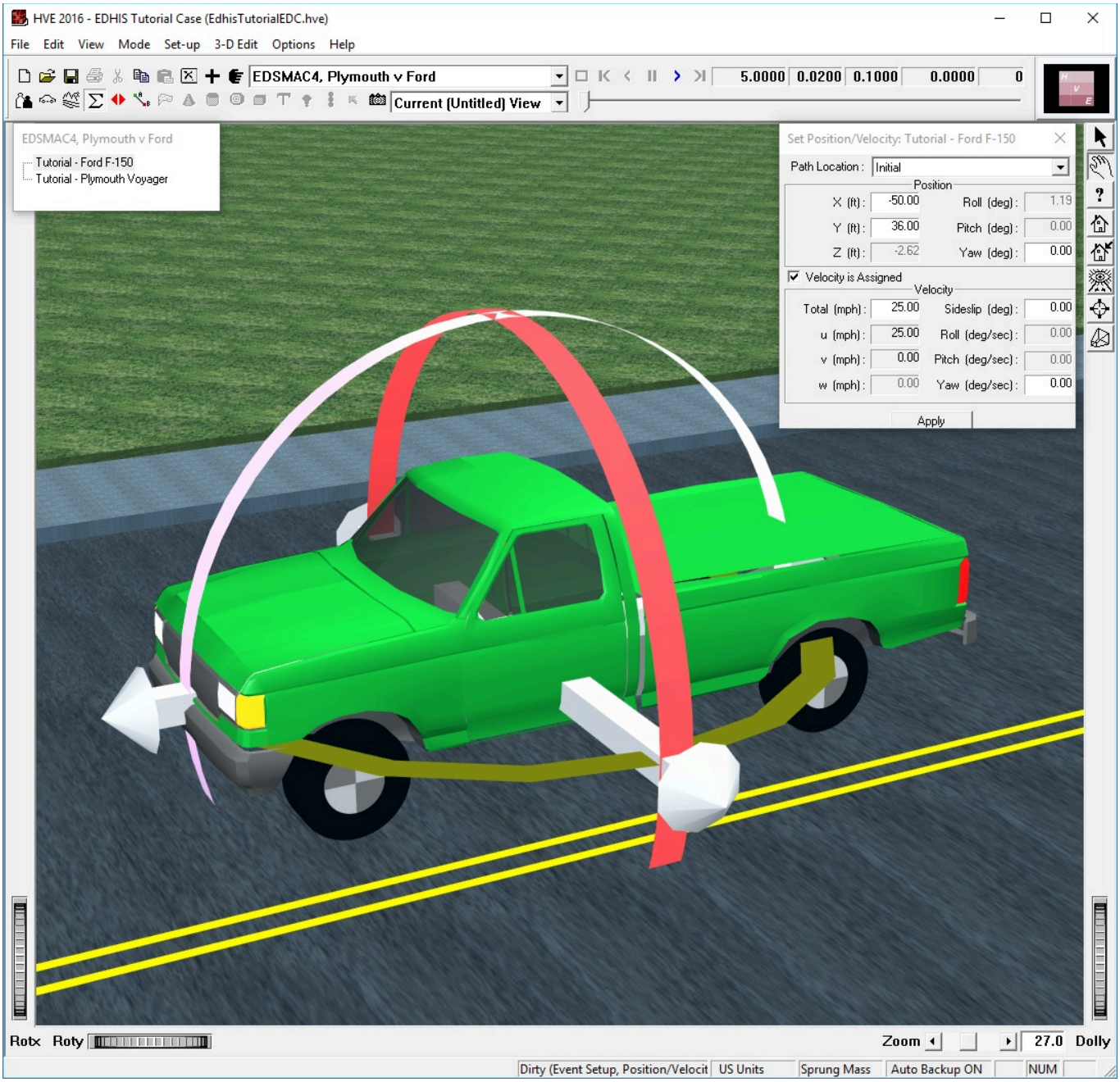


Figure 4-4: The Position Manipulator for vehicles has a set of controls (a) that move the vehicle in the X-Y plane and a set of roll, pitch and yaw manipulators (b, c and d, respectively) for orienting the vehicle. Vehicle elevation (Z) is assigned using the Position/Velocity dialog.

The object has separate manipulators for translation and rotation. The translation manipulator is a set of cross-bars which lie in the X-Y plane. This manipulator may be used to drag the human or vehicle to the desired X,Y coordinates. The Z coordinate is entered using the Position/Velocity dialog.

> **NOTE:** If AutoPosition is on, vehicle elevation will be determined automatically according to the elevation of the earth.

The rotation manipulator is a set of three bands which rotate the object about the x axis (roll), y axis (pitch) and z axis (yaw). To use the rotation manipulator, click on the desired band and drag it around its axis to cause the object to rotate.

Figures 4-3 and 4-4 show the human and vehicle manipulators, respectively.

Target Positions

Any of the eight positions (e.g., Initial, Impact, or Separation) may be assigned to the selected human or vehicle. It is even possible to assign a position that is not actually used by the current calculation model. For example, the user may enter a vehicle rest position for a simulation program. Positions which are entered but not used are called target positions because they provide visual feedback to the user regarding how close the simulated path matches the desired path.

> **NOTE:** Target humans and vehicles are translucent, to distinguish them from positions which are used by the model.

In general, simulation-type models require only initial positions and velocities. Reconstruction-type models normally require positions at impact and rest. The reconstruction or simulation model knows which positions are actually used by the model and reports this information back to HVE so targets are correctly displayed.

Target positions are also used to define the path for the HVE Path Follower (see Driver Controls, Path Follower, for more information).

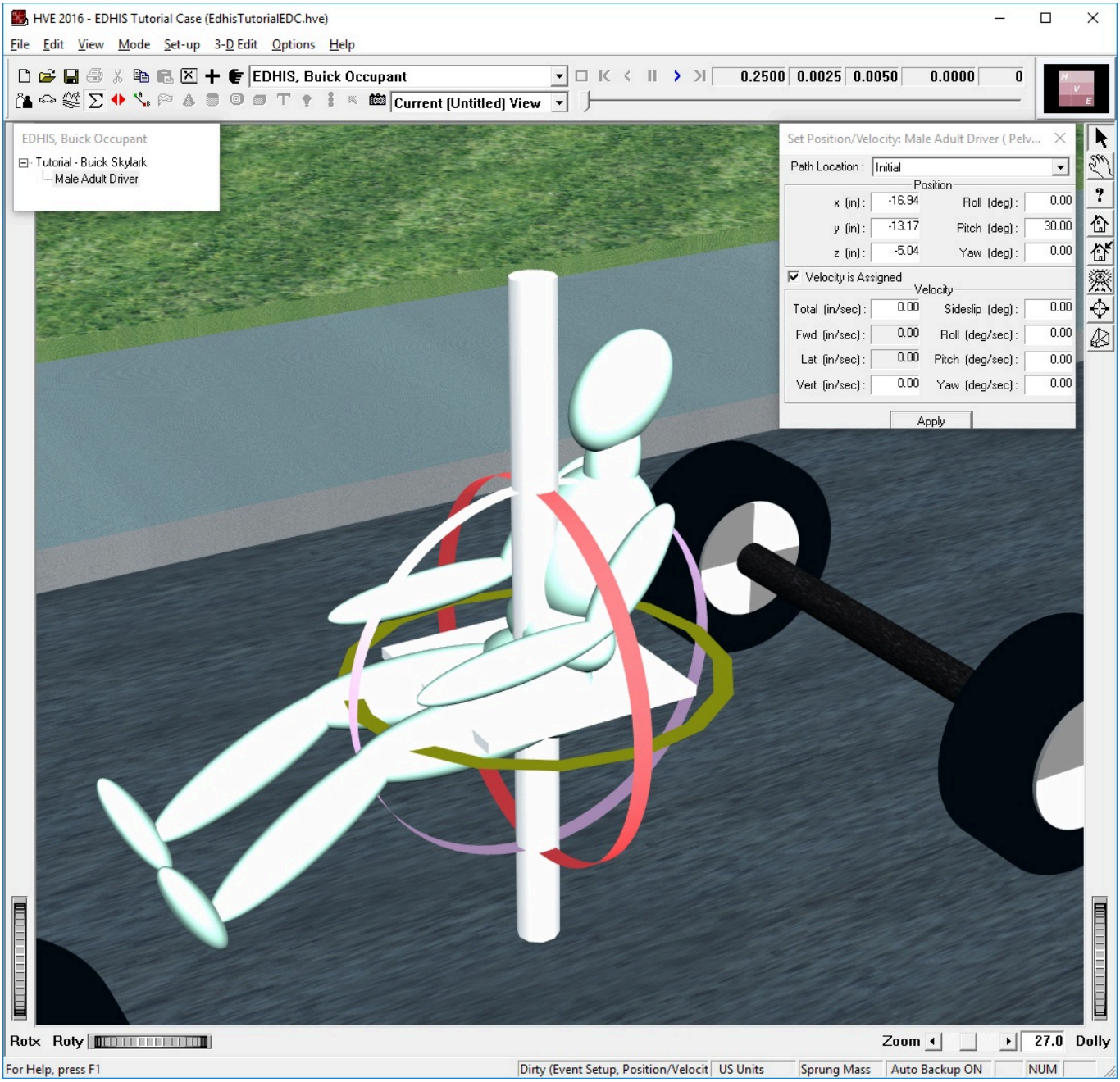


Figure 4-5: Positioning humans inside of vehicles.

Positioning Humans in Vehicles

While positioning a human occupant inside a vehicle, the vehicle becomes transparent to assist in positioning.

> **NOTE:** Although it is not required, it makes sense to position the vehicle before positioning its occupant; otherwise you'll have no visual reference point to use while positioning the human occupant.
>
> **NOTE:** You might wish to remove the vehicle's exterior body from the vehicle for positioning humans in occupant simulations. To do this, use the Vehicle 3-D Geometry File Selection dialog, and choose NoBody.h3d. This is illustrated in Figure 4-5. See Vehicle Editor for more information.

Deleting Humans and Vehicles

Any of the eight position instances (i.e., Initial, Begin Braking, Impact, etc.) may be deleted by clicking on the human or vehicle in the viewer and choosing Delete. Only the selected instance is deleted.

Parameters: The parameters assigned using this dialog are shown in Table 4-1.

Table 4-1: Parameters assigned using the Position/Velocity dialog

Parameter	Units (US / SI)	Description
X,Y,Z or x,y,z	in / cm or ft / m *	Linear distance from earth-fixed origin (vehicles and human pedestrians) or vehicle-fixed origin (human occupants)
Roll, Pitch, Yaw	deg	Angular displacement about the vehicle-fixed z, y and x axes (in that order) or human segment-fixed z, y and x axes (again, the order is important). For trailers and human segments other than the pelvis, these angles are articulation angles relative to the upstream vehicle or human segment.
Fwd, Lat, Vert velocity	in/sec, mph / km/h *	Linear velocity components in the vehicle-fixed or human segment-fixed forward, lateral and vertical directions
Roll, Pitch, Yaw velocity	deg/sec	Angular velocities about the vehicle-fixed z, y and x axes or human segment-fixed z, y and x axes. For trailers and human segments other than the pelvis, these are relative to the upstream segment.

* Vehicle units (in / cm; mph / km/h) for human occupants; environment units (ft / m; mph / km/h) for human pedestrians and vehicles.

See Also: Event Set-up, Targets, User's Manual for selected reconstruction or simulation model

---

DRIVER CONTROLS

Menu Option: DRIVER CONTROLS (Ctrl+D)

Purpose: Assign Throttle, Braking, Steering, Gear Selection and Wheel Data for the selected vehicle

Description: Selecting Driver Controls from the Set-up Menu allows the user to enter parameters which describe how the driver attempted to control the vehicle. See also the dialog references: Driver Controls and its sub-pages for Throttle, Brake, Steering, Gear, Path Follower (with Method, Speed Follower and Neuro-muscular Filter tabs) and Wheel Data.

> **NOTE:** The word "attempted" is important here. Although a driver may attempt a certain level of braking or steering, the vehicle may not necessarily respond to those inputs due to limitations in the available frictional force at the tire-road interface.

The available driver controls are:

- **Throttle Table** — Applies longitudinal tire forces that attempt to accelerate the vehicle.
- **Brake Table** — Applies longitudinal tire forces that attempt to slow the vehicle.
- **Steer Table** — Applies steer angles that attempt to steer the vehicle.
- **Gear Selection Table** — Applies gear selections that change the current transmission or differential ratio.

- **Path Follower** — Allows the vehicle simulation to determine the required steering, throttle and braking inputs required to follow a user-defined path.
- **Wheel Data** — Applies constant wheel forces and steer angles.
- **Lights** — Assigns vehicle exterior light states (e.g., headlights, brake lights, turn signals) during the event. *(updated: added since the 2006 manual — see Driver Controls)*

The Driver Controls options are described below.

Throttle Table

Throttle controls are available for simulations, and are used to accelerate the vehicle. Three methods of throttle control tables are available:

- Percent Wide-open Throttle (WOT)
- Tractive Effort
- Percent Available Friction

Percent Wide-Open Throttle (WOT)

This method allows the user to enter a table of throttle position versus time, as shown in Figure 4-6. The resulting table determines how much engine power is applied. An entry of 100 percent applies a drive torque associated with 100 percent of the current vehicle's engine power at the current engine speed.

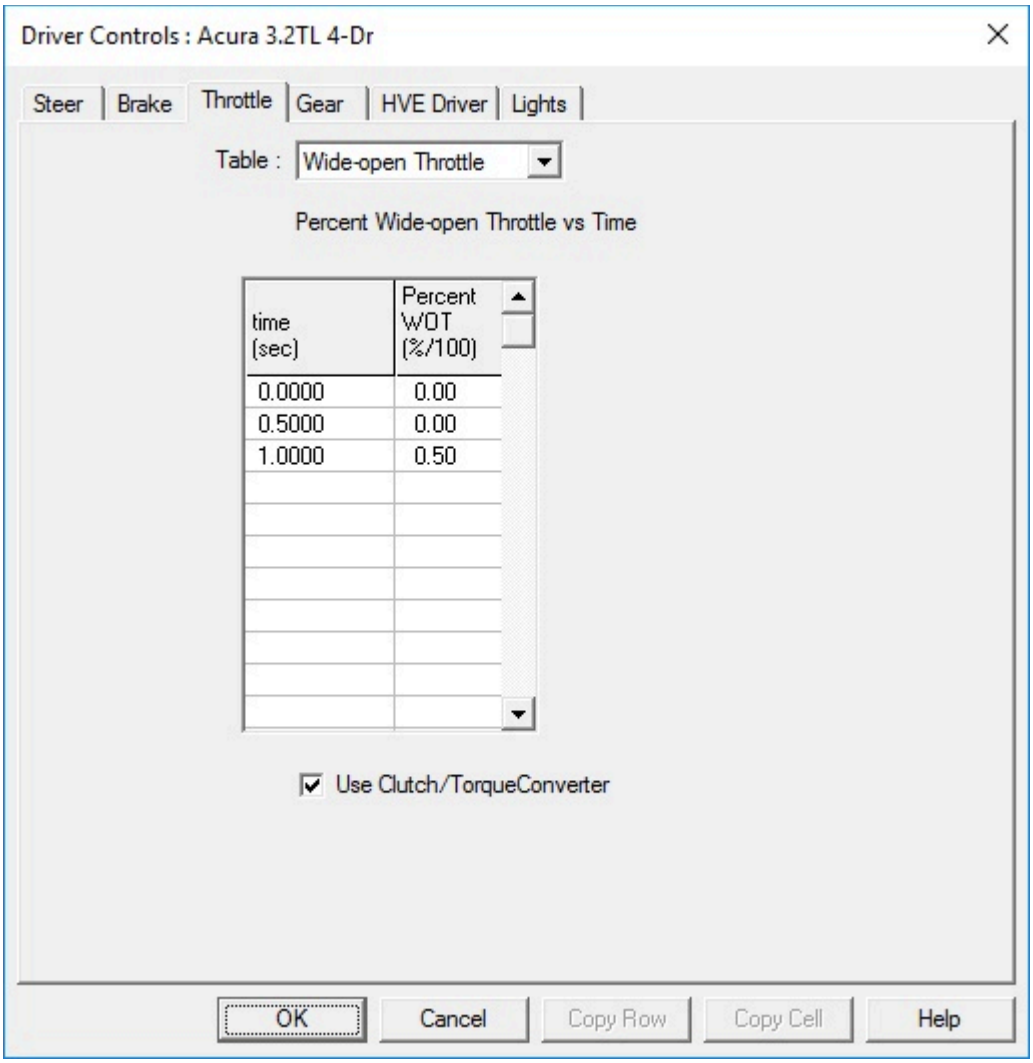


Figure 4-6: The Throttle Table, Percent Wide-open Throttle option, allows the user to enter a table of throttle position vs time for the selected vehicle.

> **NOTE:** The Percent Wide-open Throttle method is available only if the current simulation includes an engine model with the ability to calculate drive torque according to the current engine speed and throttle position.

To enter a Throttle table for the current vehicle using the Percent Wide-open Throttle method, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Throttle from the cascade menu. The Throttle Table will be displayed.
4. Choose Percent Wide-open Throttle from the Table Is option list. The table will be displayed.
5. Enter the table of times and throttle positions.
6. Press OK when the table is complete.

Tractive Effort

This method allows the user to enter a table of tractive effort (total force accelerating the vehicle) versus time, as shown in Figure 4-7. The resulting table determines how much accelerating force is applied at each drive wheel.

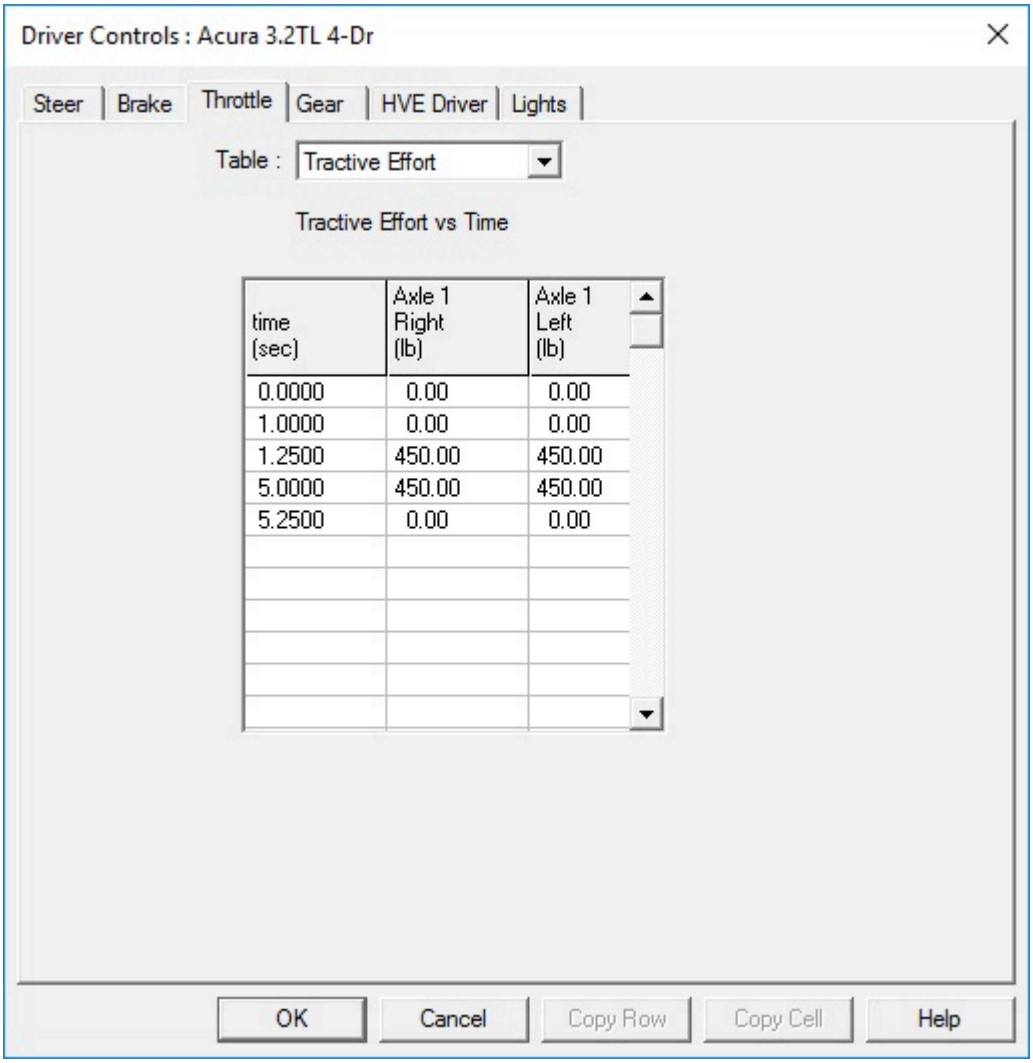


Figure 4-7: Vehicle Throttle Table dialog, Tractive Effort option.

> **NOTE:** The Tractive Effort Table allows entries only at the vehicle's drive wheels, as specified by the Vehicle Information dialog.

To enter a Throttle table for the current vehicle using the Tractive Effort method, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Select Throttle from the cascade menu. The Throttle Table will be displayed.
4. Select Tractive Effort from the Table Is option list. The table will be displayed.
5. Enter the table of times and tractive effort forces for each drive wheel.
6. Press OK when the table is complete.

Percent Available Friction

This method allows the user to enter a table of available friction force (percentage of the total available frictional force accelerating the vehicle) versus time, as shown in Figure 4-8. The resulting table determines how much accelerating

force is applied at each drive wheel. The calculations are performed by the simulation model, which multiplies the entered value by the currently available tire friction and vertical tire load.

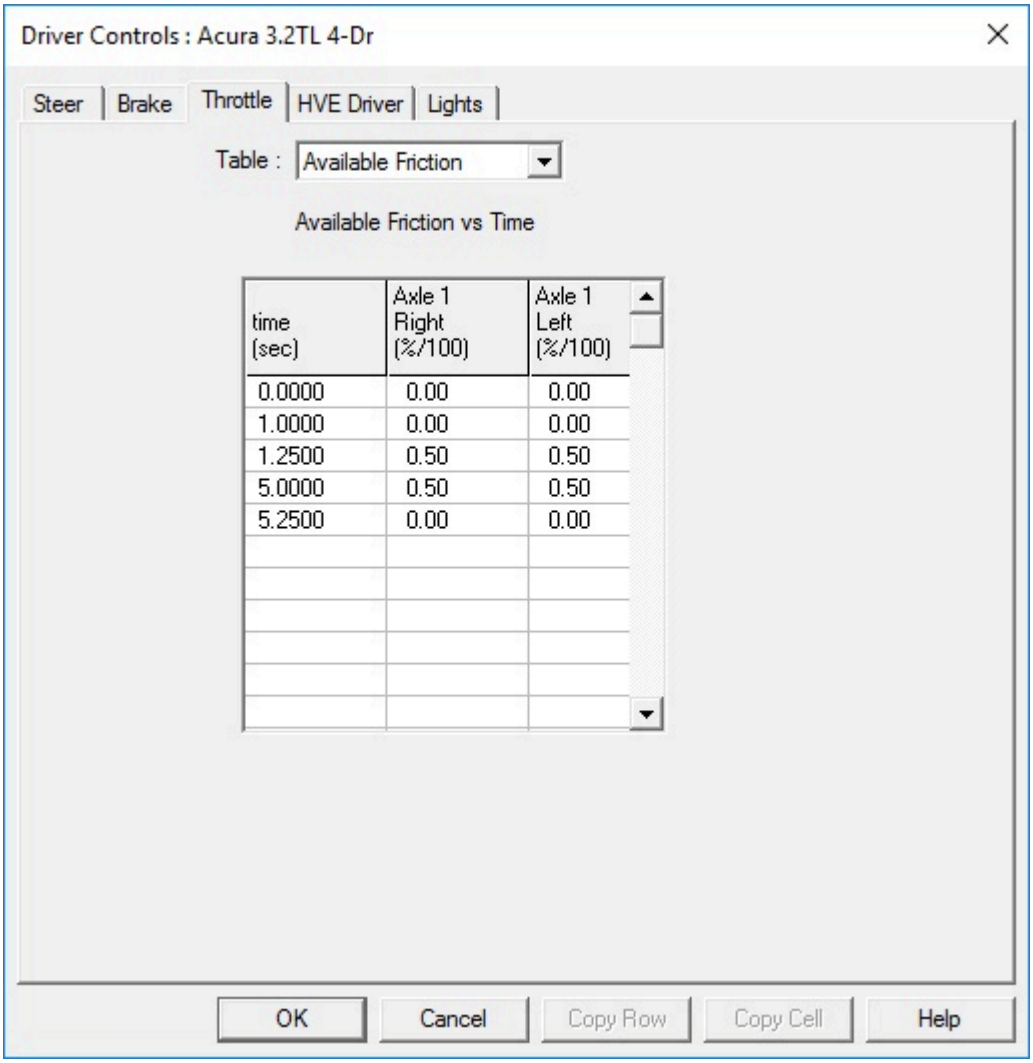


Figure 4-8: Vehicle Throttle Table dialog, Percent of Available Friction option.

> **NOTE:** The Percent Available Friction Table only allows entries at drive wheels, as specified by the Vehicle Information dialog.

To enter a Throttle table for the current vehicle using the Percent Available Friction method, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Throttle from the cascade menu. The Throttle Table will be displayed.
4. Choose Percent Available Friction from the Table Is option list. The table will be displayed. *(updated: the original manual mistakenly repeated "Tractive Effort" here)*
5. Enter the table of times and friction percentages for each drive wheel.
6. Press OK when the table is complete.

Parameters: The parameters assigned using the Throttle Table dialog are shown in Table 4-2.

Table 4-2: Throttle Table Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of throttle input
Throttle Input	%/100 or lb / N	Current value for throttle input

Brake Table

Brake controls are available for simulations. Three methods are available:

- Pedal Force
- Wheel Force
- Percent Available Friction

Pedal Force

This method allows the user to enter a table of brake pedal force versus time, as shown in Figure 4-9. The resulting table determines how much brake torque is applied at each wheel. The brake torque at each wheel is the product of the entered table value, master cylinder ratio, proportioning rate and wheel brake torque ratio.

> **NOTE:** This method is available only if the current simulation includes a brake model with the ability to calculate brake torque according to the brake system parameters and brake pedal force.

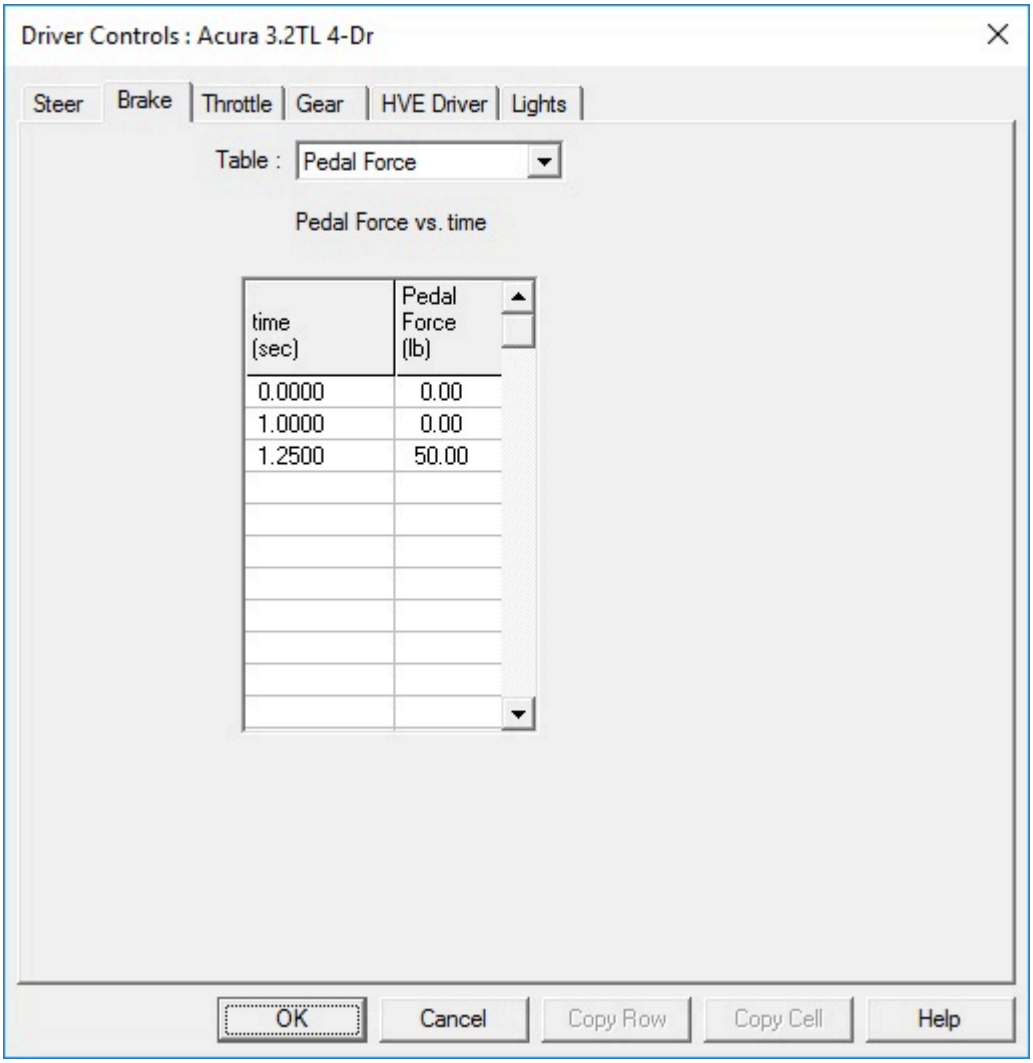


Figure 4-9: The Brake Table, Pedal Force option, allows the user to enter a table of brake pedal force vs time for the selected vehicle.

To enter a brake table for the current vehicle using the Pedal Force option, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Brakes from the cascade menu. The Brake Table will be displayed.
4. Choose Pedal Force from the Table Is option list. The selected table will be displayed.
5. Enter the table of times and brake pedal forces.
6. Press OK when the table is complete.

Wheel Force

This method allows the user to enter a table of brake force (attempted longitudinal force decelerating the vehicle) versus time, as shown in Figure 4-10. The resulting table determines how much decelerating force is applied at each wheel.

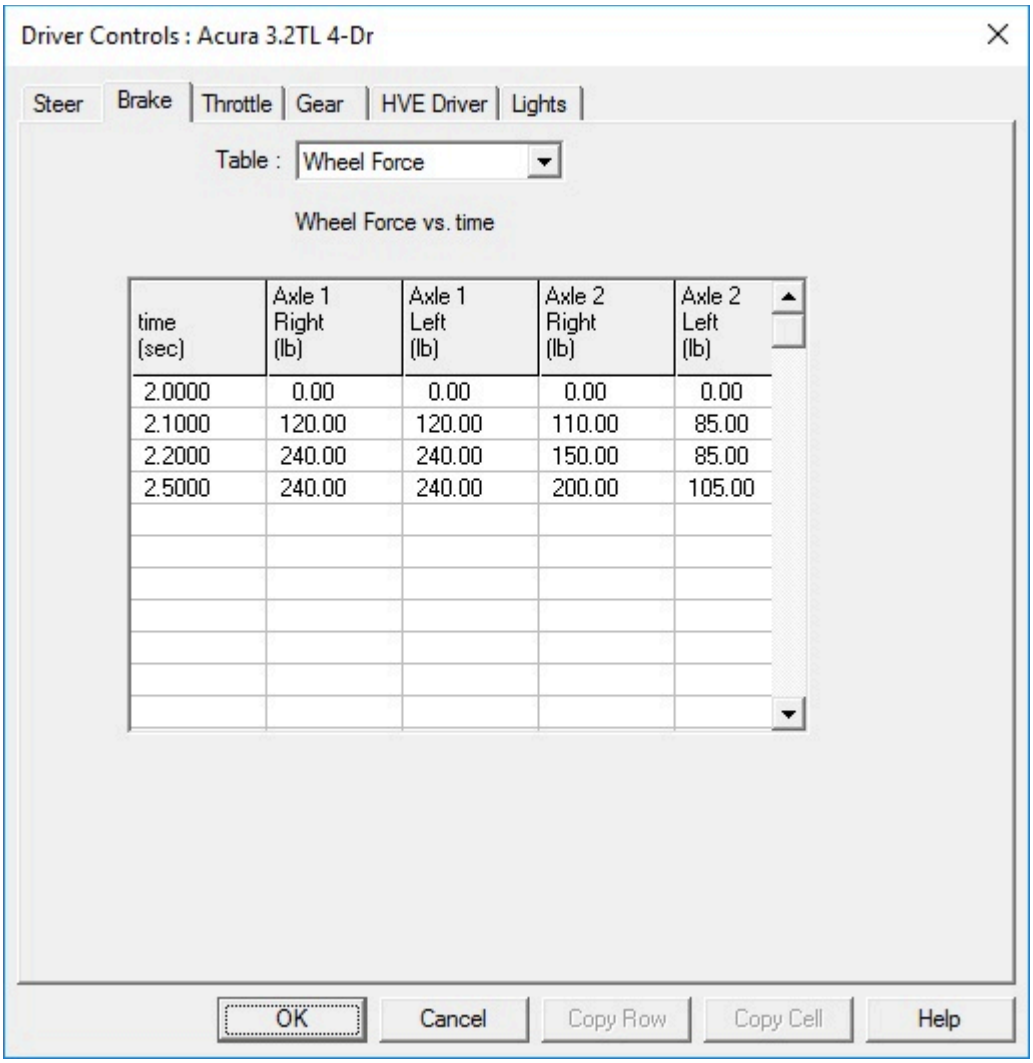


Figure 4-10: Vehicle Brake Table dialog, Wheel Force option.

To enter a Brake table for the current vehicle using the Wheel Force option, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Brakes from the cascade menu. The Brake Table will be displayed.
4. Choose Wheel Force from the Table Is option list. The selected table will be displayed.
5. Enter the table of times and brake forces at each wheel.
6. Press OK when the table is complete.

Percent Available Friction

This method allows the user to enter a table of available frictional braking force (percentage of total available frictional force braking the vehicle) versus time, as shown in Figure 4-11. The resulting table determines how much brake force is applied at each wheel. The calculations are performed by the simulation model, which multiplies the entered value by the currently available tire friction and vertical tire load.

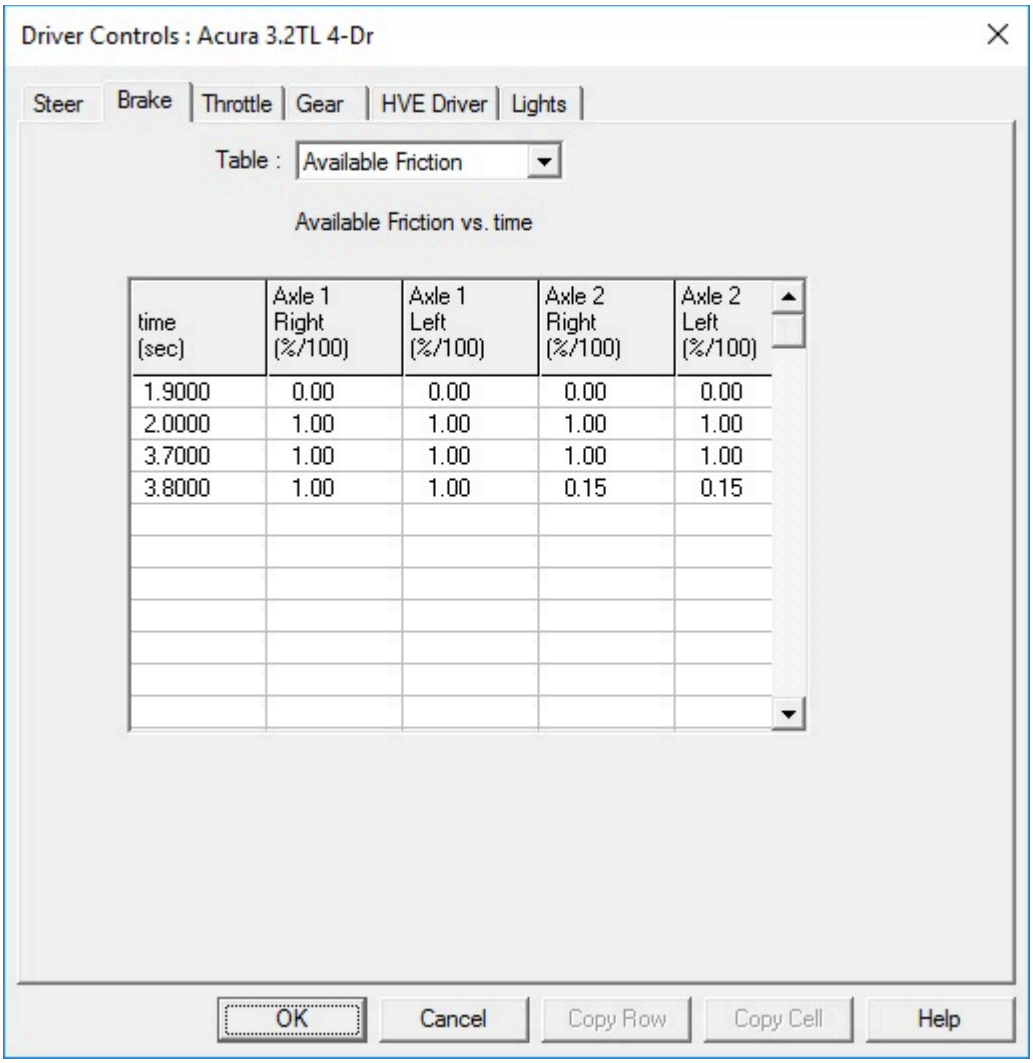


Figure 4-11: Vehicle Brake Table dialog, Percent of Available Friction option.

To enter a Brake table for the current vehicle using the Percent of Available Friction option, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Brakes from the cascade menu. The Brake Table will be displayed.
4. Choose Available Friction from the Table Is option list. The selected table will be displayed.
5. Enter the table of times and friction percentages at each wheel.
6. Press OK when the table is complete.

Parameters: The parameters assigned using the Brake Table dialog are shown in Table 4-3.

Table 4-3: Brake Table Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of brake input
Brake Input	lb / N or %/100	Current value for brake input

Steering Table

Steering controls are available for simulations. Two methods are available:

- At Steering Wheel
- At Axle

At Steering Wheel

This method allows the user to enter a table of steering wheel angle versus time, as shown in Figure 4-12. The entered value is divided by the axle's steering gear ratio to determine the nominal steer angle at the tire.

> **NOTE:** The actual steer angle for an individual tire may be affected by Ackerman angle, roll steer and/or toe-in. The inclusion of these parameters is simulation-dependent.

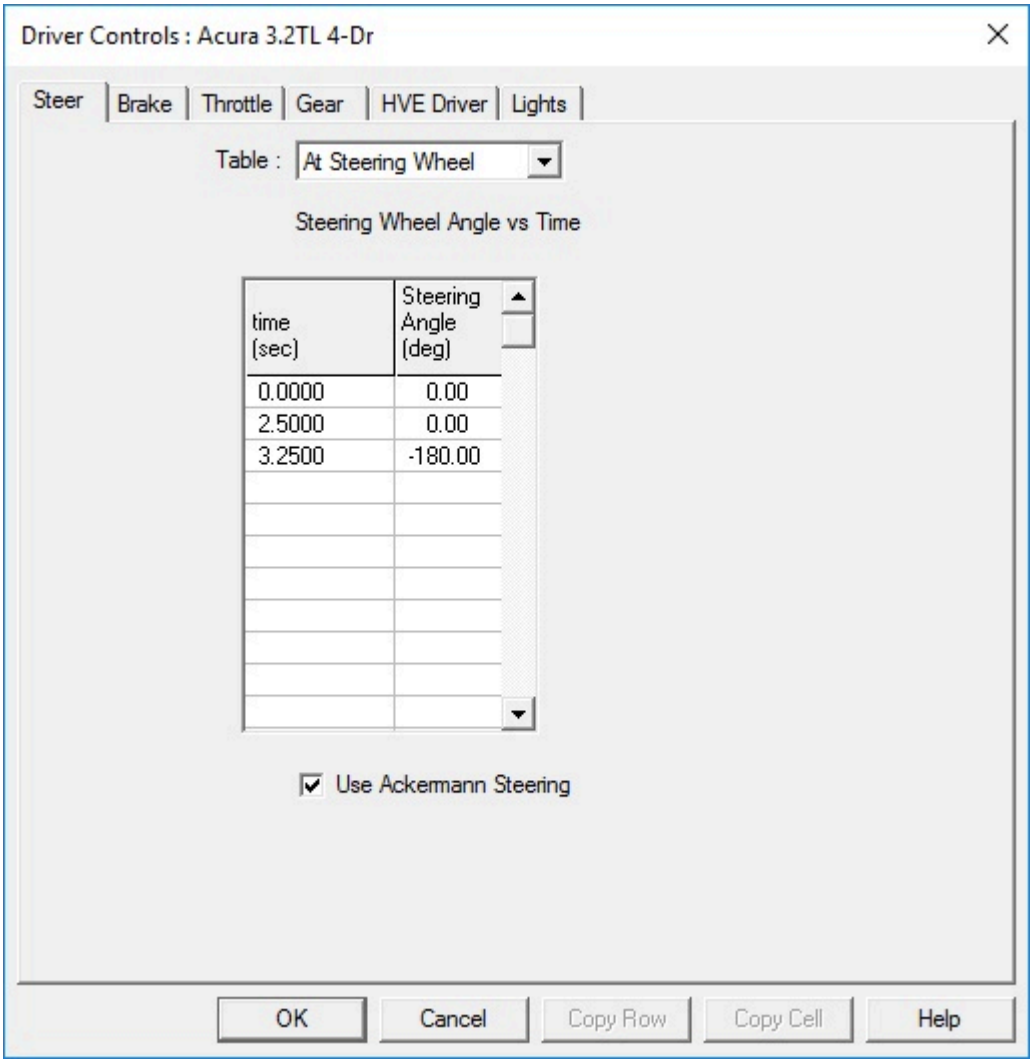


Figure 4-12: The Steer Table, At Steering Wheel option, allows the user to enter a table of steering wheel angle vs time for the selected vehicle.

To enter a Steer table for the current vehicle using the At Steering Wheel option, use the following steps:

1. Select the vehicle.

2. Choose Driver Controls from the Set-up menu.
3. Choose Steering from the cascade menu. The Steering Table will be displayed.
4. Choose At Steering Wheel from the Table Is option list. The selected table will be displayed.
5. Enter the table of times and steering wheel angles.
6. Press OK when the table is complete.

At Axle

This method allows the user to enter a table of steer angle at each tire of a steerable axle, as shown in Figure 4-13. The entered value is applied directly to the tire, and is unaffected by any roll steer or toe-in.

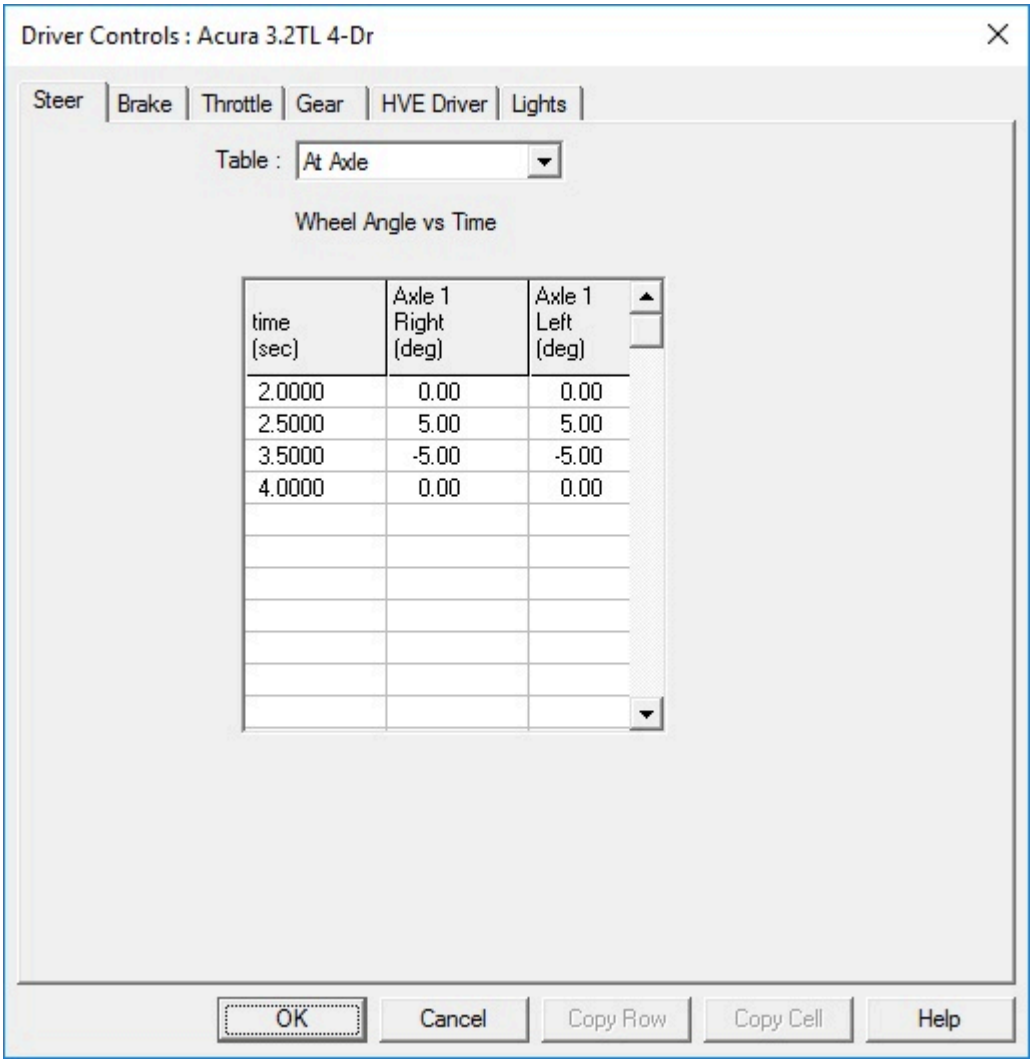


Figure 4-13: The Vehicle Steer Table dialog, At Axle option, allows the user to directly enter steer angles for the left and right wheels on each steerable axle.

To enter a Steer Table for the current vehicle using the At Axle option, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Steering from the cascade menu. The Steering Table will be displayed.
4. Choose At Axle from the Table Is option list. The selected table will be displayed.

5. Enter the table of times and steer angles for each wheel.
6. Press OK when the table is complete.

Parameters: The parameters assigned using the Steer Table dialog are shown in Table 4-4.

Table 4-4: Steer Table Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of steer input
Steer Input	deg	Current value for steer input

Gear Selection

The Gear Selection dialog is available for simulations. This dialog allows the user to enter a table of gear selection versus time, as shown in Figure 4-14. Separate tables may be entered for the transmission and differential.

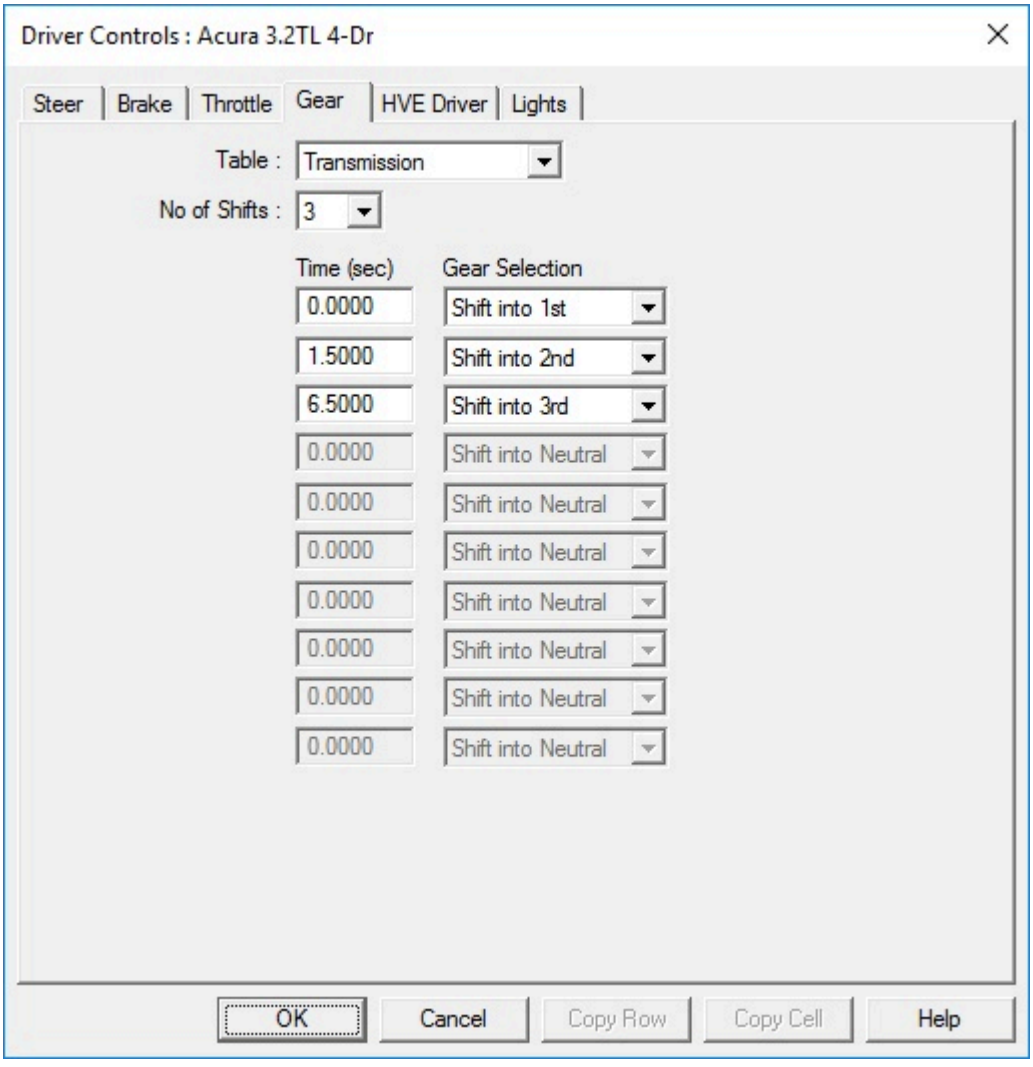


Figure 4-14: The Gear Selection Table allows the user to enter a table of gear selections vs time for the selected vehicle. Separate tables are available for the transmission and differential.

> **NOTE:** This option is available only if the current Throttle Method is Percent Wide-Open Throttle.

The dialog has a user-selectable option for transmission and differential.

> **NOTE:** The differential option is available only if the selected vehicle has more than one differential ratio.

To enter a Gear Table for the current vehicle, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Throttle from the cascade menu. The Throttle Table will be displayed. Confirm that the Percent Wide-open Throttle option is used. If that is not the current option, click on Table Is and choose Percent Wide-open Throttle, then enter a throttle table using the Percent Wide-open Throttle option, as described earlier.

> **NOTE:** If the Percent Wide-open Throttle option is not selectable, the current calculation method probably does not support the HVE Drivetrain model. In this case, the Gear Table option is not available. 4. Choose Driver Controls from the Set-up menu and select the Gear Selection option. The Gear Selection table will be displayed. 5. Choose the At Transmission option. 6. Enter the table of times and gear numbers for the transmission. 7. If the vehicle has a multi-speed differential, choose the At Differential option and enter a table of gear numbers vs time for the differential. 8. Press OK when the table is complete.

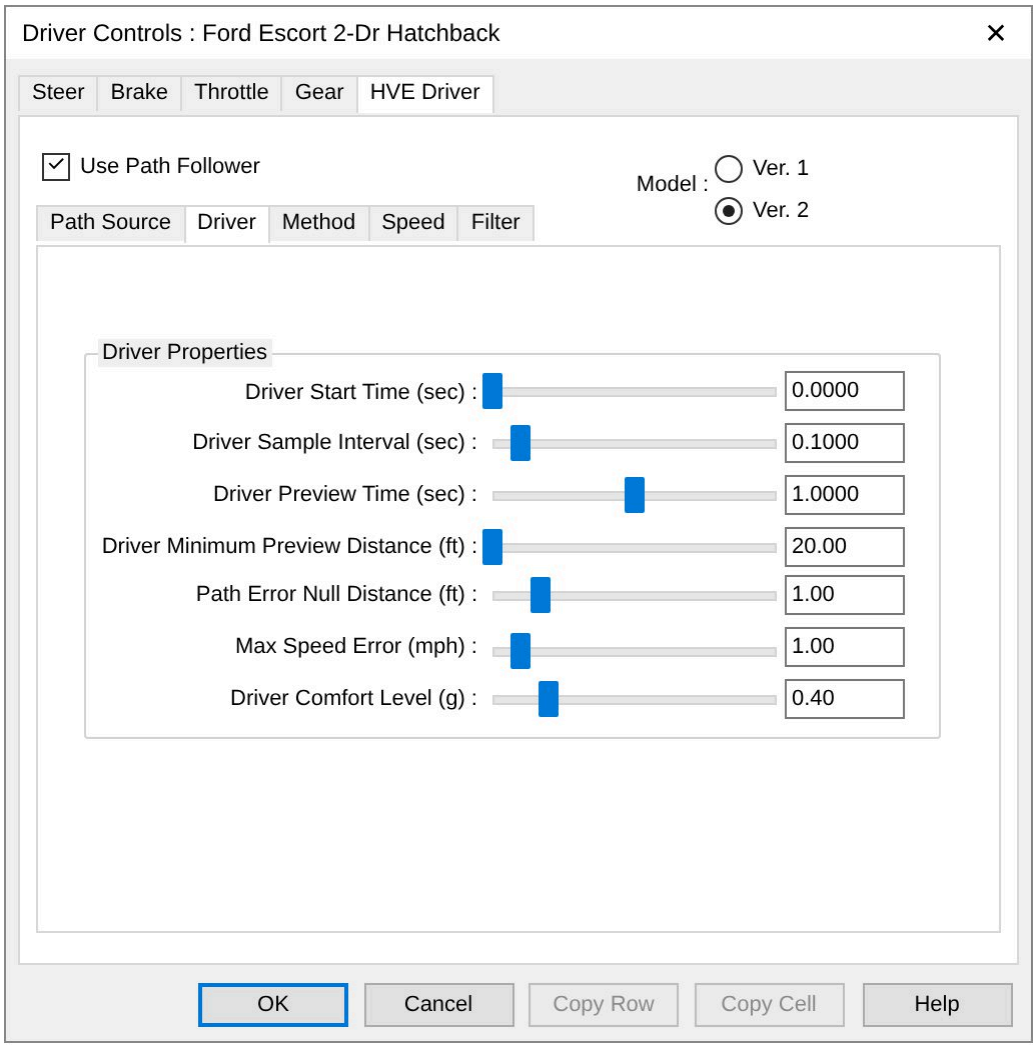
Parameters: The parameters assigned using the Gear Selection dialog are shown in Table 4-5.

Table 4-5: Gear Selection Table Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current gear shift
Gear Selection	—	Current gear number for transmission or differential

HVE Path Follower

The HVE Path Follower allows the user to define a 3-D path using target positions. Using this path, the simulation model determines the steering, throttle and braking inputs required to make the vehicle follow the path. The HVE Path Follower is set up on the **HVE Driver** tab of the Driver Controls dialog, shown in Figure 4-15 (see also Driver Controls Path Follower). Its settings are arranged on the Path Source, Driver, Method, Speed and Filter tabs.


Driver Controls : Ford Escort 2-Dr Hatchback
Steer
Brake
Throttle
Gear
HVE Driver
Use Path Follower
Model :
Ver. 1
Ver. 2
Path Source
Driver
Method
Speed
Filter
Driver Properties
Driver Start Time (sec) :
0.0000
Driver Sample Interval (sec) :
0.1000
Driver Preview Time (sec) :
1.0000
Driver Minimum Preview Distance (ft) :
20.00
Path Error Null Distance (ft) :
1.00
Max Speed Error (mph) :
1.00
Driver Comfort Level (g) :
0.40
OK
Cancel
Copy Row
Copy Cell
Help

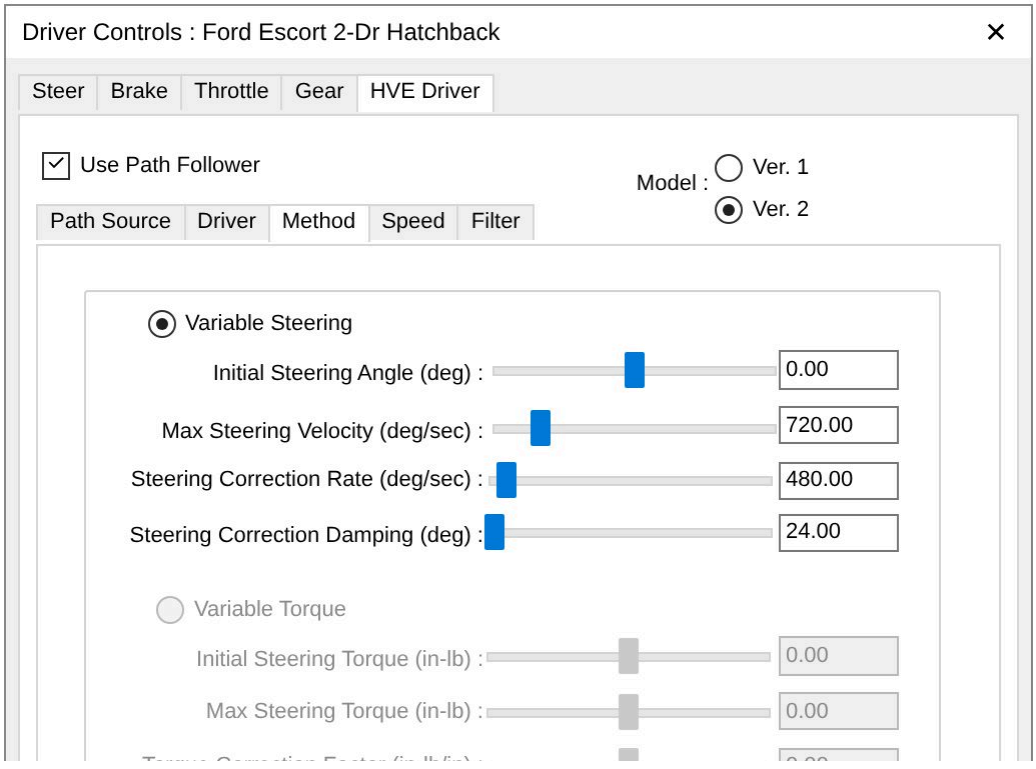

Driver Controls : Ford Escort 2-Dr Hatchback
Steer
Brake
Throttle
Gear
HVE Driver
Use Path Follower
Model :
Ver. 1
Ver. 2
Path Source
Driver
Method
Speed
Filter
Variable Steering
Initial Steering Angle (deg) :
0.00
Max Steering Velocity (deg/sec) :
720.00
Steering Correction Rate (deg/sec) :
480.00
Steering Correction Damping (deg) :
24.00
Variable Torque
Initial Steering Torque (in-lb) :
0.00
Max Steering Torque (in-lb) :
0.00



Figure 4-15: The HVE Driver tab of the Driver Controls dialog, showing the Driver (top) and Method (bottom) tabs. It is used to simulate the driver inputs required to follow a user-defined path.

The HVE Path Follower includes several features, some required and some optional. These features are:

- **Path Generator** (required) — user-defined path positions (e.g., Initial, Begin Braking, etc.) to define the attempted path
- **General Parameters** (required) — user-entered parameters required for HVE Path Follower operation
- **Method** (required) — user-entered parameters defining how the correction is accomplished
- **Speed Follower** (optional) — user-entered velocities at each path position are used to define an attempted speed at each point on the path
- **Neuro-muscular Filter** (optional) — user-entered parameters that define driver physiological capabilities [3.17]

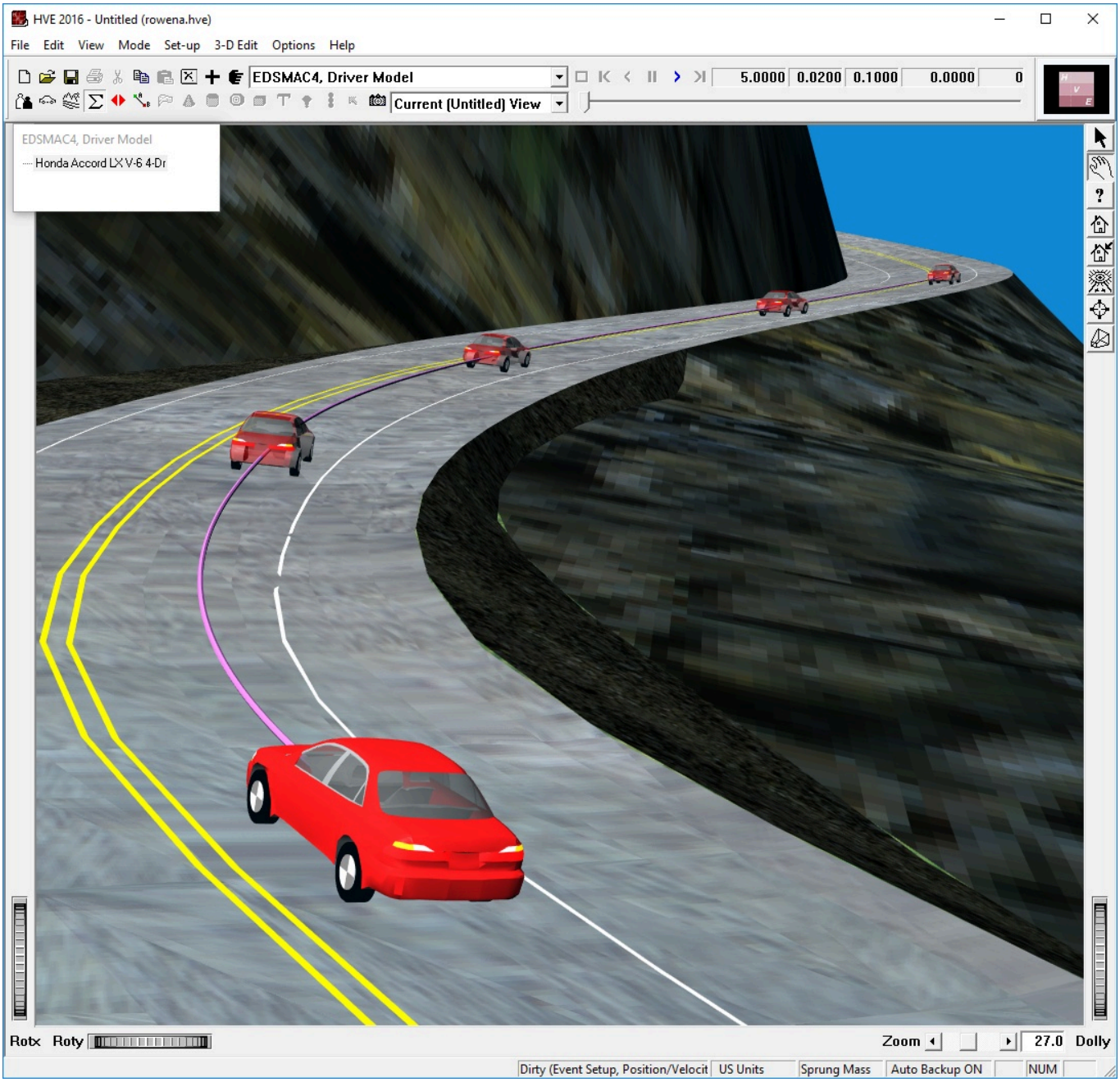


Figure 4-16: The HVE Path Follower uses up to eight user-assigned positions to define a 3-D path. Intermediate path positions are determined using a 3-D spline.

Path Generator

A primary input to a path follower is the procedure used to define the attempted path. The HVE Path Follower uses up to eight user-defined path positions (see Event Set-up, Assigning Positions). These positions and orientations are used as nodes in a 3-D spline path (see Figure 4-16). It is this spline path that the vehicle attempts to follow.

General Parameters

The basic parameters required for use of the HVE Path Follower are the path (described above), Start Time and Sample Interval, Preview Distance (the point ahead of the vehicle where the driver is actually looking and presumably wants to go), the Allowable Path Error and the level of Lateral Acceleration acceptable to the driver.

Method

Two methods are available in the HVE Path Follower. These are:

- **Variable Steering** — The path correction is accomplished by means of a steering correction. In this case, the user supplies an Initial Steer Angle, Maximum Steering Wheel Velocity, Steering Correction Factor and Steer Damping.
- **Variable Torque** — The path correction is accomplished by means of a torque application at the steerable wheels. In this case, the user supplies an Initial Steering Wheel Torque, Maximum Steering Wheel Torque, Torque Correction Factor and Torque Damping.

Speed Follower

The speed follower option attempts to maintain the required speed, as established by the user-entered velocities at each path position (if a velocity for any path position is not entered, an error message will be issued and the method will abort). In addition, the user supplies an Allowable Speed Error, Maximum Throttle Application and Maximum Brake Pedal Force.

Neuro-muscular Filter

The neuro-muscular filter from the original HVOSM VD-2 model has been included in the HVE Path Follower. The neuro-muscular filter represents a simplified model of the physiological operator which incorporates a Time Delay, Lead Time and Lag Time. These parameters correspond to the first-order effects of the neurological and muscular systems of a human driver, as described in reference 3.17.

To use the HVE Path Follower, first you must assign at least two path positions for the vehicle, then perform the following steps:

1. Choose Driver Controls from the Set-up menu.
2. Choose Path Follower from the cascade menu. The HVE Path Follower dialog will be displayed.
3. Assign the desired HVE Path Follower parameters.
4. Press OK when the desired parameters are assigned.

Parameters: The parameters assigned using the HVE Path Follower dialog are shown in Table 4-6.

Table 4-6: HVE Path Follower Parameters

Parameter	Units (US / SI)	Description
Start Time	sec	Simulation time for start of path follower
Sample Time	sec	Time increment for path sampling
Driver Preview Distance	ft / m	Distance ahead of vehicle where driver is looking
Max Path Error	ft / m	Allowable distance from desired path to point projected ahead of vehicle at the Driver Preview Distance
Max Lat Accel	g	Driver's limit for comfortable lateral acceleration
Path Follower Method	—	Method used to accomplish path following (Variable Steer or Torque)
Initial Steer Angle	deg	Steering wheel angle at Driver Start Time
Max Steer Velocity	deg/sec	Limit on steering wheel angular velocity
Steer Correction Factor	deg/ft / deg/m	Amount of steering correction per unit of path error
Steer Correction Damping	deg-sec/ft / deg-sec/m	Damping, used to limit steering activity
Initial Steering Torque	in-lb / N-m	Steering wheel torque at Driver Start Time
Max Steering Torque	in-lb / N-m	Limit on steering wheel torque
Torque Correction Factor	in-lb/ft / N-m/m	Amount of steering torque correction per unit of path error
Torque Damping	in-lb-sec/ft / N-m-sec/m	Damping, used to limit steering activity
Max Speed Error	mph / km/h	Allowable difference between current speed and desired speed
Max Throttle Pedal	%/100	Maximum allowable pedal position
Max Brake Pedal	lb / N	Maximum allowable pedal force
Driver Lag Time	sec	Neuro-muscular filter lag time
Driver Lead Time	sec	Neuro-muscular filter lead time
Driver Time Delay	sec	Neuro-muscular filter time delay

Wheel Data

The Wheel Data dialog is available for reconstruction models. This dialog, shown in Figure 4-17, allows the user to enter the following information:

- Pre-impact Total Wheel Lock-up
- Impact-To-Rest Motion Type (Rotating/Lateral Skidding or Constant Sideslip)
- Wheel Lock-up (Percent Available Friction used) at each wheel
- Steer Angle at each wheel

These parameters are described below.

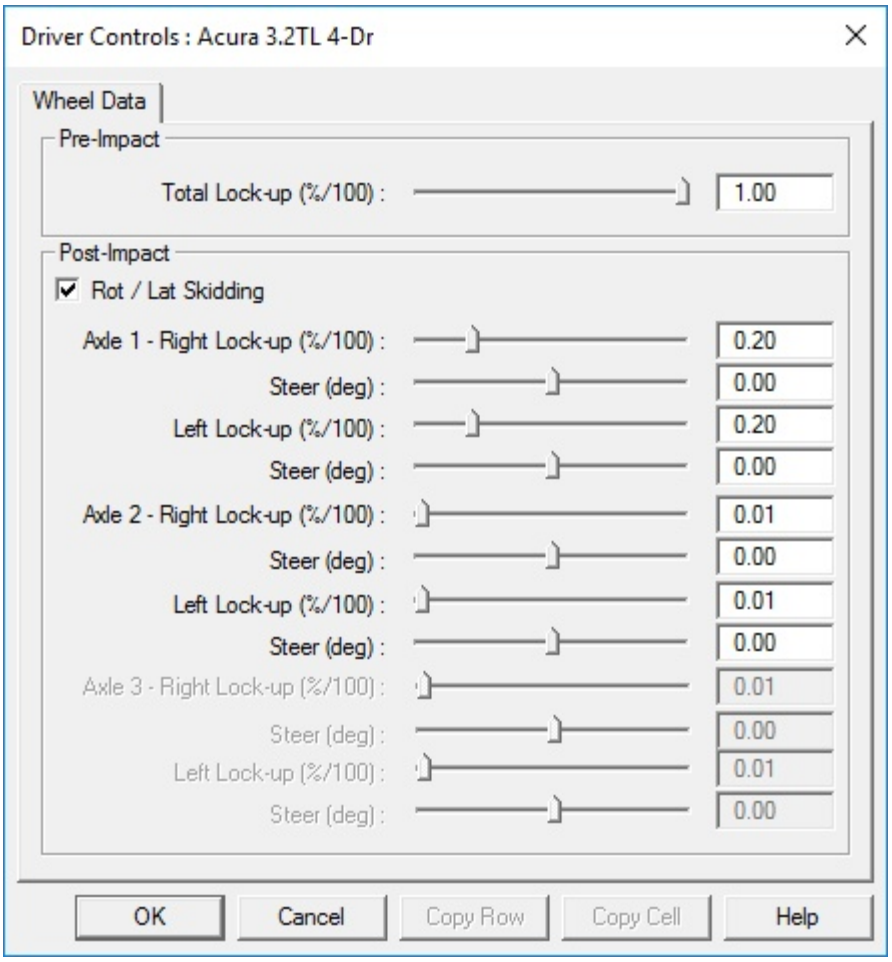


Figure 4-17: Vehicle Wheel Data dialog.

Pre-impact Total Wheel Lock-up

The entered value is multiplied by the average tire-ground friction coefficient to determine the total vehicle deceleration between the user-entered Begin Braking and Impact positions. This value is relevant only if the user has supplied a Begin Braking position for the current vehicle.

Rotating/Lateral Skidding

This check box may be used by reconstruction models to trigger special calculations for vehicles that spin between separation and rest.

> **NOTE:** Spinning vehicles have tire forces which vary, resulting in a variable deceleration rate between impact and rest!

Wheel Lock-up

The Wheel Lock-up (sometimes called Percent Available Friction) is multiplied by the available friction force to determine the average longitudinal force at each wheel. The entered value should not account for lateral skidding; the program's tire model will determine the tire slip angle and account for the force associated with lateral skidding.

> **NOTE:** Wheel lock-up for simulation models may be applied using the Wheel Damage dialog (see Wheels menu option later in this chapter).

Steer Angle

The entered value is used to determine the steer angle at each wheel. This angle is assumed to remain constant, and applies only during the post-impact phase.

To enter Wheel Data for the current vehicle, use the following steps:

1. Select the vehicle.
2. Choose Driver Controls from the Set-up menu.
3. Choose Wheel Data from the cascade menu. The Wheel Data dialog will be displayed.
4. Enter the desired pre-impact drag factor, post-impact rotation option and percent of available friction and steer angle for each wheel.
5. Press OK when the desired data are entered.

> **NOTE:** Wheel Lock-up and Steer Angle values assigned for each wheel are assumed to remain constant throughout the run.

Parameters: The parameters assigned using the Wheel Data dialog are shown in Table 4-7.

Table 4-7: Wheel Data Dialog Parameters

Parameter	Units (US / SI)	Description
Total Wheel Lock-up	%/100	Average wheel lock-up for all wheels, used to determine the pre-impact deceleration rate
Wheel Lock-up	%/100	Constant percent of available friction at each wheel used to decelerate vehicle between impact and rest
Steer Angle	deg	Constant steer angle at each wheel

See Also: Wheels Menu Option, Vehicle Model, Event Model

Continue with Part B: Damage Profiles, Collision Pulse, Vehicle Mesh, Payload, Wheels, Accelerometers, Restraints, Contacts, Signals, Notes.

Chapter 4: Set-up Menu (Part B — Damage Profiles through Notes)

HVE User's Manual — Section Two: Menu Reference. Continued from Part A (Position/Velocity, Driver Controls).

DAMAGE PROFILES (VEHICLE DAMAGE)

Menu Option: DAMAGE PROFILES (Ctrl+M) *(updated: the 2006 manual titled this section "Vehicle Damage"; the current menu item is "Damage Profiles")*

Purpose: Assign a damage profile to the selected vehicle

Description: The Vehicle Damage dialog (see Figure 4-18) allows the user to assign a damage profile to the selected vehicle. The damage profile is obtained by post-crash inspection of the vehicle, and provides a significant amount of useful information about the collision, including:

- Damage Energy
- Peak Force
- Linear and angular velocity change
- Principal Direction of Force (PDOF)

This information may be used to help estimate impact speed and collision severity. The major advantage is that scene data are not required.

CDC

HVE's Damage Profile dialog requests from the user a Collision Deformation Classification (CDC), a seven-character code describing the vehicle damage. The CDC is an SAE Recommended Practice, and is defined in SAE J-224C [6.3], portions of which are included in the HVE Help Index.

The entered CDC is used to define a default PDOF and damage profile, which includes damage width, depth and location. The default data provided by the CDC are used to fill in the fields in the Damage Profile dialog. The resulting delta-V, damage energy and peak force are calculated and displayed in the dialog.

HVE uses the damage profile to calculate and display the Equivalent Energy Speed, or EES [2.25, 6.4]. The default value is set equal to the total delta-V. The user may override the default EES by clicking on the EES radio button and entering the desired value.

> **NOTE:** The calculated results define what would happen if the current vehicle struck a rigid barrier, not another vehicle! To determine what would happen during a collision with another vehicle requires further analysis which includes information about the second vehicle.

To enter Vehicle Damage Data for the current vehicle, use the following steps:

1. Select the vehicle.
2. Choose Damage Profiles from the Set-up menu. The Vehicle Damage dialog will be displayed, as shown in Figure 4-18.

3. Enter a CDC and press Apply. The default damage profile (based on the CDC) and associated damage results will be displayed.
4. Where actual measurements are available, replace any of the default results assigned by the CDC with the available data, followed by Apply.
5. Press OK after the desired vehicle damage data are entered.

The specific damage profile parameters are defined in this section.

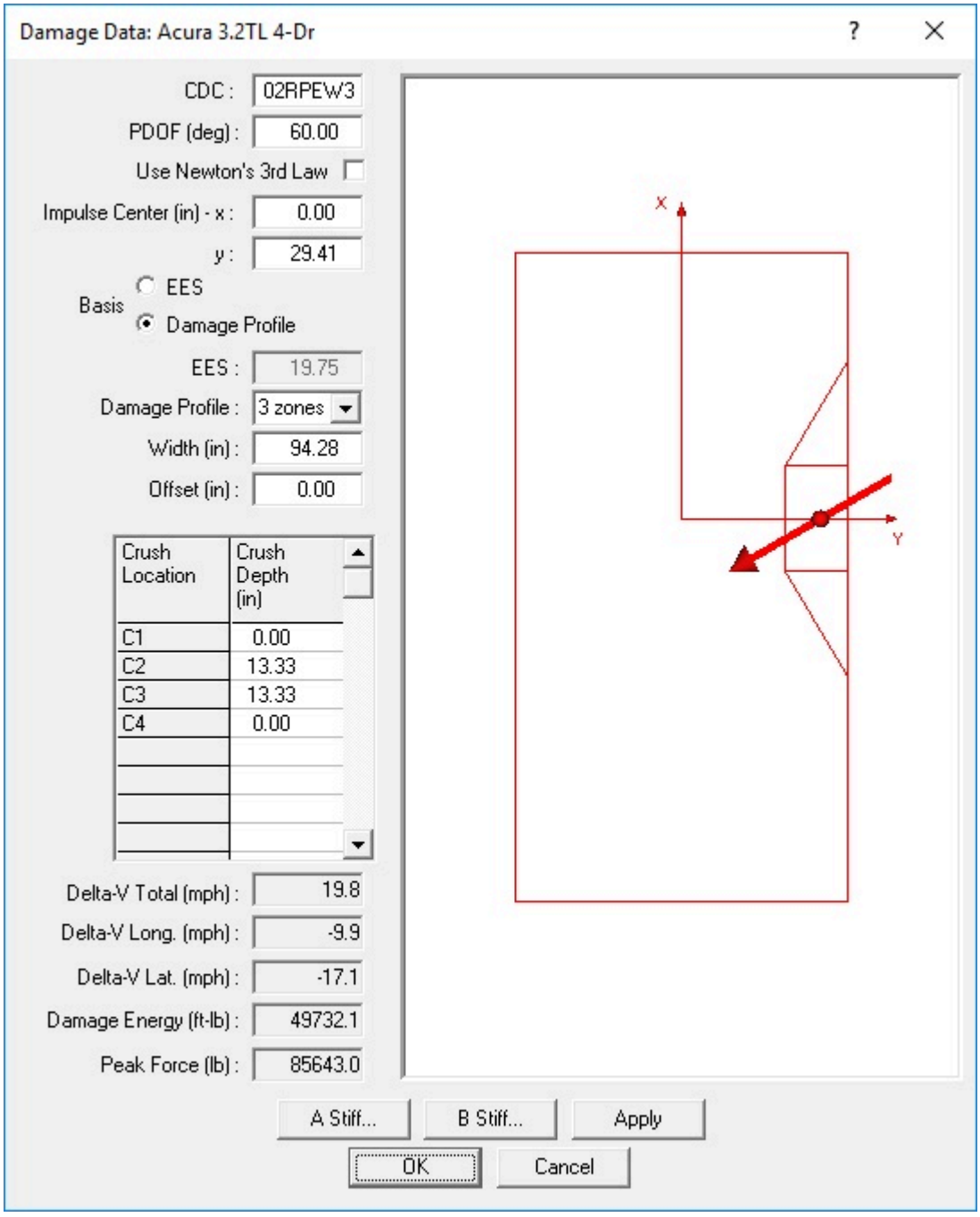


Figure 4-18: The Damage Data dialog (Vehicle Damage Profile) is used to assign damage data for the current vehicle.

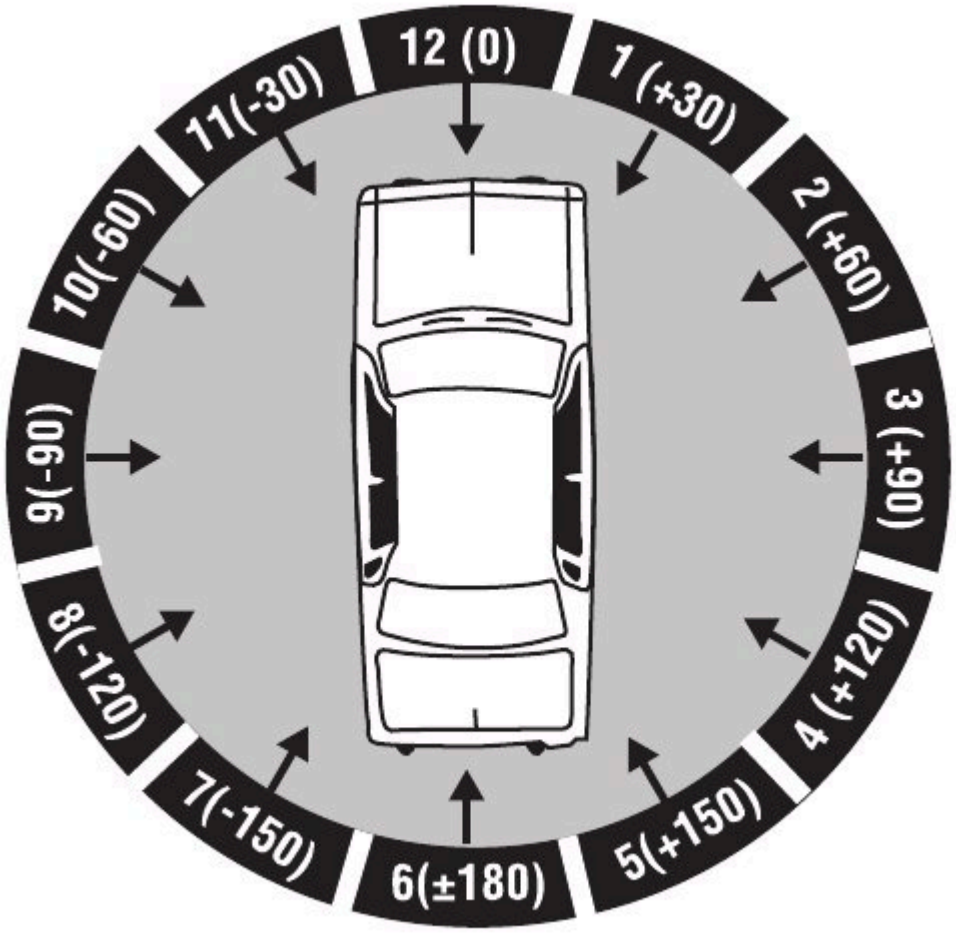


Figure 4-19: The default PDOF is assigned by its clock direction (12, 01, 02, etc.) in the first two characters in the CDC.

PDOF

Simply stated, the PDOF is the direction of the impulse. This, of course, has physical significance, because the direction of the impulse is the same as the direction of the delta-V. In fact, because the damage analysis has no way of computing the forward and lateral components of the delta-V, the user-entered PDOF is used for this purpose.

> **NOTE:** If scene data are entered, the PDOF can often be calculated. This serves as an excellent cross-check on the user-entered value.

The PDOF is assigned by the CDC as the closest hour angle (or clock direction), as shown in Figure 4-19. The damage profile dialog displays the resulting PDOF in degrees. This value may be edited by the user if a more precise value is known (note that the precision of the clock direction is limited by the number of degrees in one hour, or 30 degrees).

Use Newton's 3rd Law

Some reconstruction programs (e.g., EDCRASH) are able to calculate the PDOF of one of the vehicles in a two-vehicle collision based on the impact heading angles and the PDOF of the other vehicle. Clicking this check box executes that

option.

> **NOTE:** You can only choose this option for one of the vehicles; if you choose this option for both vehicles, the program will normally display an error message.

Damage Profile

The vehicle damage profile is defined by three parameters:

- Damage Width
- Crush Depths
- Damage Midpoint Offset

These three parameters are described below.

Damage Width

The Damage Width is the width of the damaged region. The default value assigned by the CDC does not include induced damage (non-contact damage adjacent to the area of actual contact with another vehicle). In most cases, the user should include induced damage if the goal is to estimate delta-V [2.26].

Crush Depths

A table of Crush Depths defines the shape of the damage profile. Up to 10 equally spaced crush depths may be provided along the total damage width, as shown in Figures 4-18 and 4-20.

Damage Midpoint Offset

The Damage Midpoint Offset is simply the longitudinal or lateral distance from the CG to the midpoint of the damage profile, as shown in Figures 4-18 and 4-20.

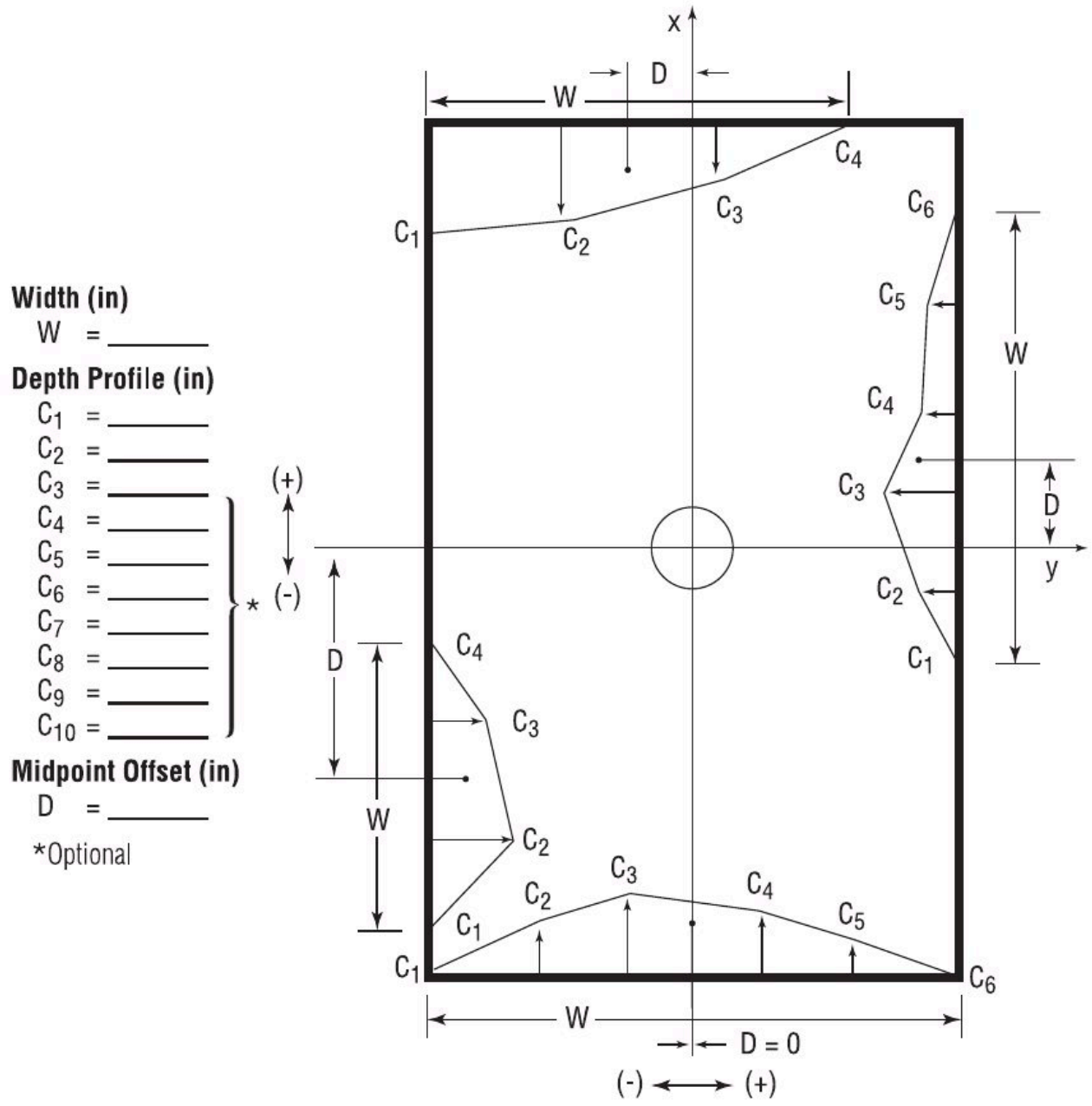


Figure 4-20: Examples of damage profiles for front, back and side impacts.

Stiffness Coefficients

The damage analysis requires empirical coefficients determined by crash tests. These coefficients are called stiffness coefficients, because they define the structural stiffness of the vehicle.

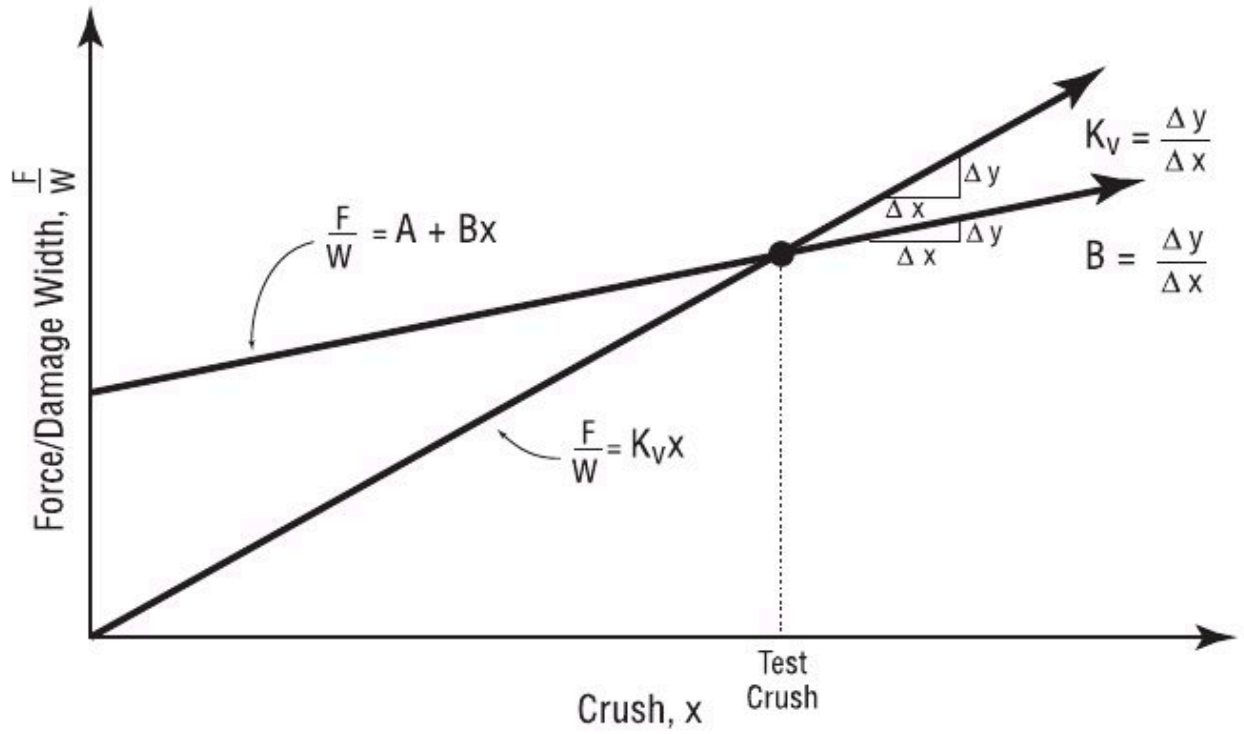


Figure 4-21: Force vs Crush characteristics.

The HVE Vehicle Damage dialog includes two user-assigned stiffness coefficients:

- **A** — The A Stiffness Coefficient has units of force per unit of damage width, and defines the force necessary to begin crushing the vehicle's exterior.
- **B** — The B Stiffness Coefficient has units of force per unit of crush depth per unit of damage width, and defines the linear spring rate of the vehicle's exterior.

Given values for A and B, the crush force per unit of damage width is assumed to have a linear relationship with crush depth. Figure 4-21 shows this relationship. The A and B coefficients are provided in the A Stiffness and B Stiffness dialogs, respectively (see Figure 4-22). The A Stiffness and B Stiffness dialogs allow the user to assign equal coefficients for every zone or vary the coefficients from zone to zone.

A and B stiffness coefficients may be calculated from barrier crash test data. Methods are outlined in references 2.13, 4.28 and 4.29.

> **NOTE:** Before using any of these methods, ensure the type of crash test is known! Crash tests involving a deformable barrier are not useful, because the fraction of energy absorbed by each vehicle is unknown. In addition, moving barrier crash tests require a more complete equation (for fixed barrier tests, several parameters drop out of the equation).

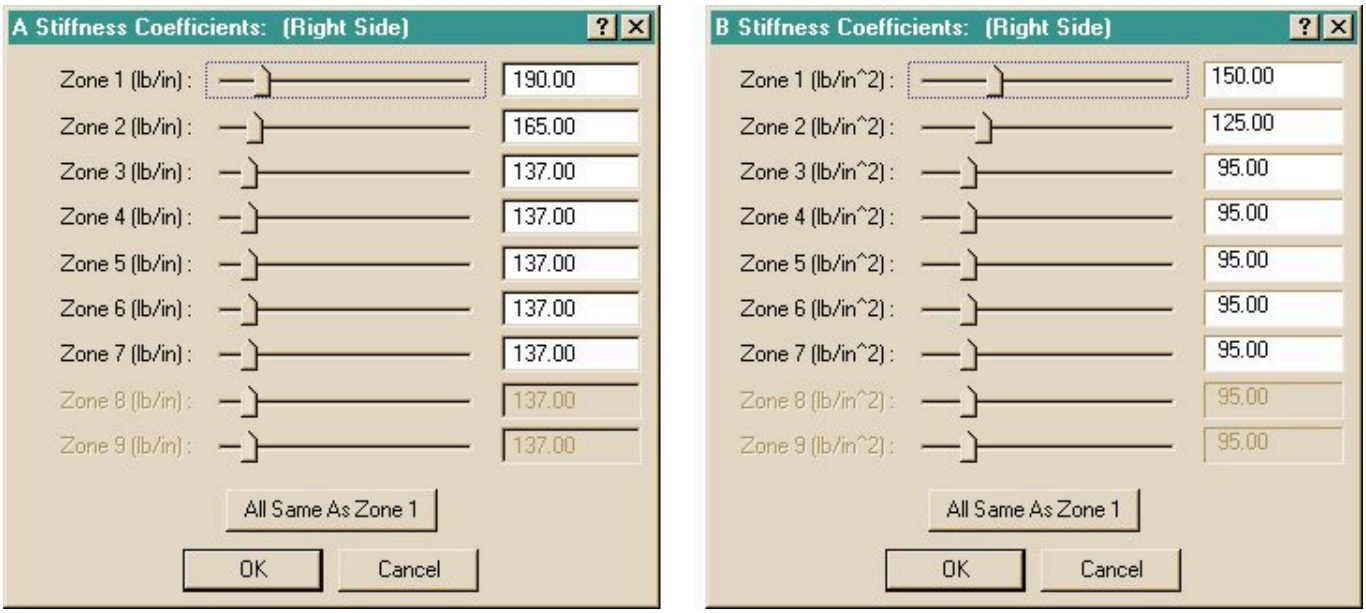


Figure 4-22: The A and B Stiffness dialogs allow the user to supply stiffness coefficients for each crush zone.

Crush Zones

The area between any two crush depth measurements is called a crush zone; thus the number of crush zones will always be one less than the number of entered crush depths (see Figures 4-18 and 4-20).

Obviously, the actual stiffness of a vehicle varies according to the structure absorbing the crush, especially on the side of a vehicle, where one area, such as the quarter panel, contains unsupported sheet metal, while another area, such as a wheel, might be quite rigid. A separate set of A and B coefficients may be assigned to each crush zone.

> **NOTE:** The A and B coefficients are not independent from each other; one should never be adjusted without adjusting the other. In addition, the percentage of adjustment should be the same for both the A and B stiffness coefficients.

Parameters: The parameters assigned using this dialog are shown in Table 4-8.

Table 4-8: Damage Profile Parameters

Parameter	Units (US / SI)	Description
CDC	(none)	Collision Deformation Classification, 7-character damage code describing location and character of damage
PDOF	deg	Principal Direction Of Force, the direction of the collision impulse (same as direction of delta-V)
Impulse Center	in / cm	Vehicle-fixed x,y coordinates of the impulse center
EES	mph / km/h	Energy Equivalent Speed, the speed associated with the energy required to cause the observed damage if the vehicle had struck a fixed rigid barrier
Width	in / cm	Total width of damage
Offset	in / cm	Longitudinal or lateral distance from the CG to the center of the damage profile
Crush Depth	in / cm	Measured depth of crush at up to 10 points along the damage profile
A Stiffness	lb/in / N/cm	Force per unit of damage width required to initiate measurable damage (for each crush zone)
B Stiffness	lb/in^2 / N/cm^2	Force per unit of crush depth per unit of damage width required to cause the measured damage (for each crush zone)

See Also: Event Editor, Vehicle-fixed Coordinate System, Reconstruction Models, References 2.23, 2.26, 4.28, 4.29

COLLISION PULSE

Menu Option: COLLISION PULSE (Ctrl+U)

Purpose: Assign a collision pulse to the selected vehicle

Description: The Collision Pulse dialog allows the user to assign a collision pulse to the selected vehicle. A collision pulse is a vehicle's position, velocity, acceleration or force/moment vs time history, and is used by occupant simulation models to define the vehicle's motion during impact. The Collision Pulse dialog is shown in Figure 4-23.

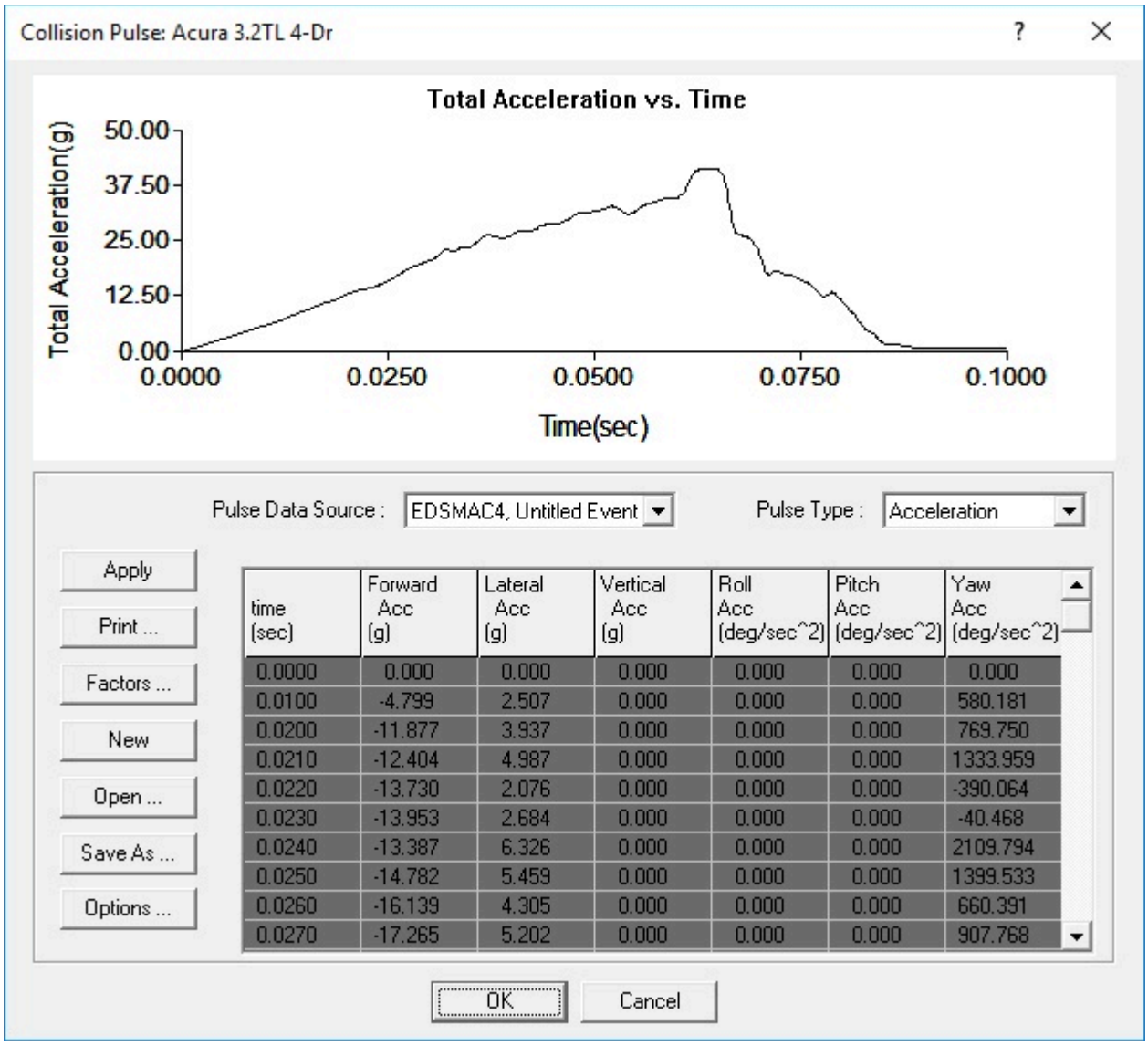


Figure 4-23: The Collision Pulse dialog allows the user to assign a collision pulse for the current vehicle. A collision pulse is used by occupant simulation models and by some vehicle dynamics models that include an impulse.

Collision Pulse Table

To enter Collision Pulse data for the current vehicle, use the following steps:

1. Select the vehicle.
2. Choose Collision Pulse from the Set-up menu. The Collision Pulse dialog will be displayed (see Figure 4-23).
3. Enter a table of time and dependent values (Position, Velocity, Acceleration (default) or Force/Moment data; see Pulse Type, below) for the duration of the collision.

> **NOTE:** The duration of the table is normally about 75 to 150 milliseconds. 4. Press OK when the desired data are entered.

Collision Pulse Source

The user may assign a collision pulse by entering a table of acceleration vs time, as described above. However, it is much easier (and faster) if the collision pulse is assigned by another program, such as SIMON/DyMESH or EDSMAC4. To assign the collision pulse using this method, perform the following steps:

1. Click on the Pulse Data Source option list. The option list contains all of the previously executed events which produced a collision pulse for the selected vehicle.
2. Choose the desired event. Its collision pulse will be loaded in the table and displayed in the graph.

> **NOTE:** If the Pulse Source is from an event, the table is non-editable. If you wish to edit the pulse data from an event, you can save it first, then open the table as a user-saved pulse (see Opening and Saving Collision Pulse Files later in this section).
>
> **NOTE:** If the selected event's output interval is too large, HVE will display a message suggesting you rerun the event with a smaller output interval.

Collision Pulse Type

The HVE Collision Pulse dialog allows the user to apply four types of pulses to a vehicle:

- Position vs time
- Velocity vs time
- Acceleration vs time
- Force and Moment vs time

To select a particular pulse type for the current event, perform the following steps:

1. Click on the Pulse Type option list. The available pulse types are displayed.
2. Select the desired pulse type from the list.

> **NOTE:** Not all simulation models support every pulse type; the current simulation model will determine which pulse types are selectable from the list.

The Collision Pulse Table for the selected pulse type is displayed.

Opening and Saving Collision Pulse Files

Collision pulses may be assigned from previous cases, as well as saved for use in future cases. For example, a measured acceleration pulse for a specific vehicle's 35 mph barrier crash test may be available from test data. This pulse may be used for several individual simulations wherein the restraint system pre-load is varied. To assign a collision pulse from a previous case, perform the following steps:

1. Choose Open. The Collision Pulse Selection dialog will be displayed.
2. Select the filename of the desired collision pulse.
3. Press OK to display the selected collision pulse.

To save the current collision pulse for use in a future case, perform the following steps:

1. Choose Save As. The Collision Pulse Selection dialog will be displayed.
2. Enter a filename for the current collision pulse.
3. Press OK to save the current collision pulse.

Collision Pulse Factors

It may be useful to study the sensitivity of the occupant motion to changes in the magnitude of the collision pulse. The Pulse Factors dialog (see Figure 4-24) allows the user to quickly assign multipliers to each degree of freedom (forward, lateral, vertical, roll, pitch and yaw).

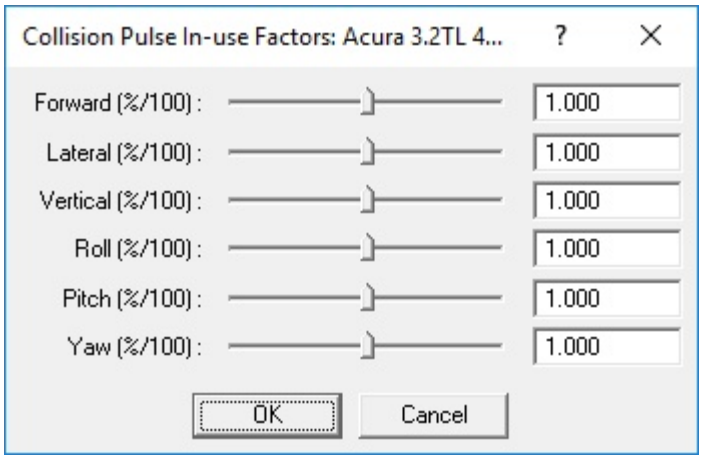


Figure 4-24: The Collision Pulse Factors dialog allows the user to modify the time-dependent pulse data by a user-specified amount for each degree of freedom.

To assign collision pulse factors to the selected vehicle, perform the following steps:

1. Choose Factors. The Collision Pulse Factors dialog will be displayed.
2. Use the individual sliders to increase or decrease the linear and angular values.

> **NOTE:** The default slider value is 1.0. 3. Press OK.

The selected multipliers are applied to each degree of freedom.

Collision Pulse Options

The user may need to select a portion of the total pulse for use in a simulation. For example, an entire left turn may last 3 seconds, while the resulting collision with an oncoming vehicle lasts perhaps only 125 milliseconds. Thus, the user would like to extract that portion of the total pulse during which the collision occurs. This task can be performed using the Pulse Options dialog (see Figure 4-25). The Pulse Options dialog is also used for assigning the impulse center for a 3-dimensional impulse when the Force Pulse Type is used.

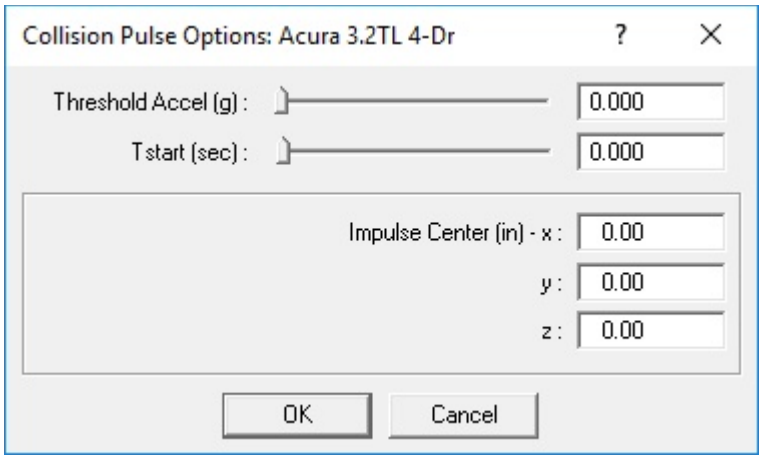


Figure 4-25: The Collision Pulse Options dialog allows the user to trim the unwanted portion of the collision pulse and assign an impulse center for Force/Moment pulse tables.

To use the Collision Pulse Options dialog for the selected vehicle, perform the following steps:

1. Choose Options. The Collision Pulse Options dialog will be displayed (see Figure 4-25).
2. Enter the desired starting time for the pulse.

> **NOTE:** Only the portion of the pulse beginning at the starting time is included in the pulse table. 3. Enter the desired acceleration threshold for the pulse.
>
> **NOTE:** Only the portion of the pulse equal to or greater than the entered value is included in the pulse table; the default value is 1.0 g. 4. If the pulse type is Force/Moment, enter the x,y,z coordinates for the impulse center.
>
> **NOTE:** This option is only available if the current pulse type is Force/Moment. 5. Press OK to assign the pulse options.

Parameters: The parameters assigned using this dialog are shown in Table 4-9.

Table 4-9: Collision Pulse Parameters

Parameter	Units (US / SI)	Description
Pulse Source	N/A	Available pulse sources
Pulse Type	N/A	Available pulse types (Position, Velocity, Acceleration or Impulse)
Fwd, Lat, Vert linear degrees of freedom for selected pulse type	length, velocity, acceleration or force units, depending on the variable	Forward, lateral and vertical components of the vehicle's linear motion
Roll, Pitch, Yaw angular degrees of freedom for selected pulse type	angle, angular velocity, angular acceleration or torque units, depending on the variable	Angular motion about vehicle's roll, pitch and yaw axes
Pulse Start	sec	Starting time for pulse
Pulse Threshold	g	Minimum total acceleration required to be considered part of the collision pulse (applies only to acceleration pulses)

* Unit Name is dependent on pulse type: Disp (Position), Vel (Velocity), Accel (Acceleration) or Force (Impulse).

See Also: Output Tracks, Event Model, User's Manual for selected occupant simulation model

---

VEHICLE MESH

Menu Option: VEHICLE MESH *(updated: the 2006 manual titled this option "Mesh"; the current menu item is "Vehicle Mesh")*

Purpose: Perform various mesh-related operations on the selected vehicle

Description: The Mesh dialog allows the user to do the following:

- Modify vehicle geometry file (mesh) properties
- Select and deselect mesh groups for inclusion in damage
- Select a pair of vehicles (or a vehicle and environment) and edit their inter-vehicle friction and restitution properties

These options (see Figure 4-26) are described in the following sections.

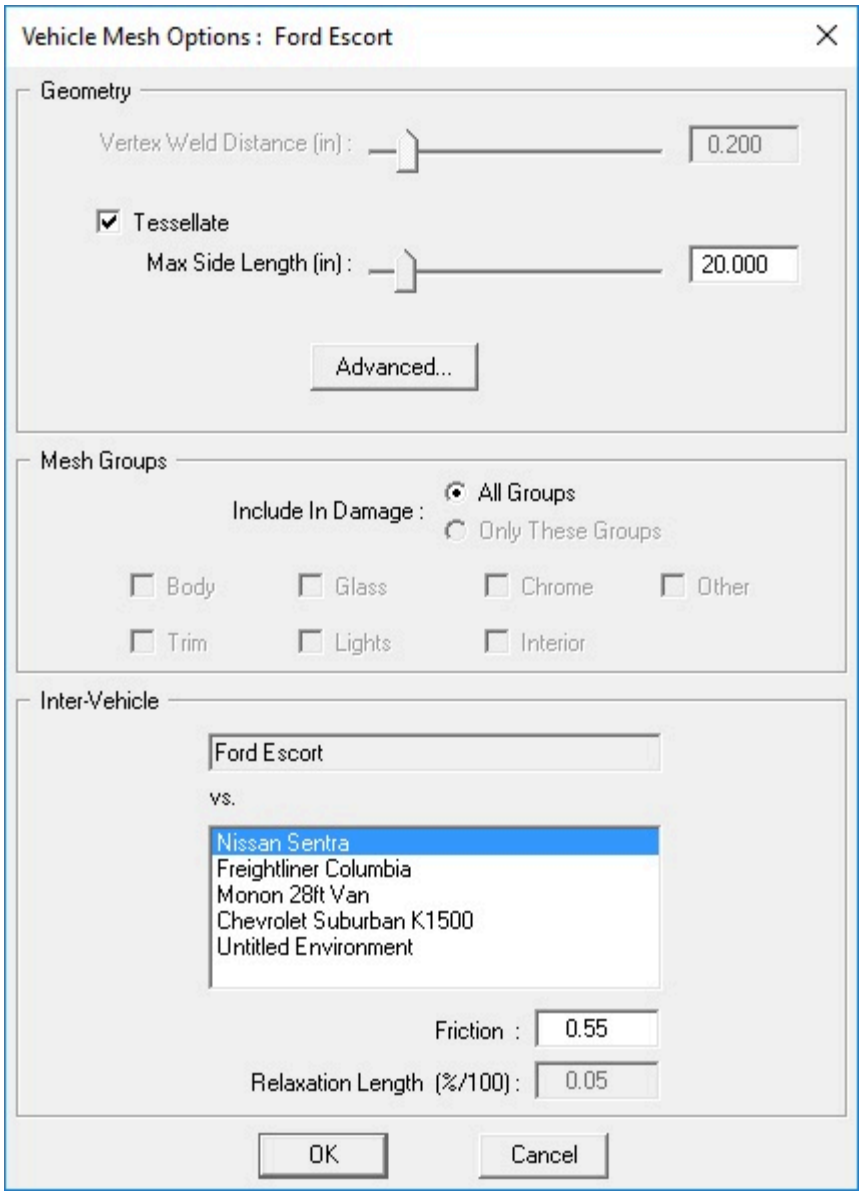


Figure 4-26: The Mesh dialog allows the user to tessellate the geometry file and edit inter-vehicle friction and restitution.

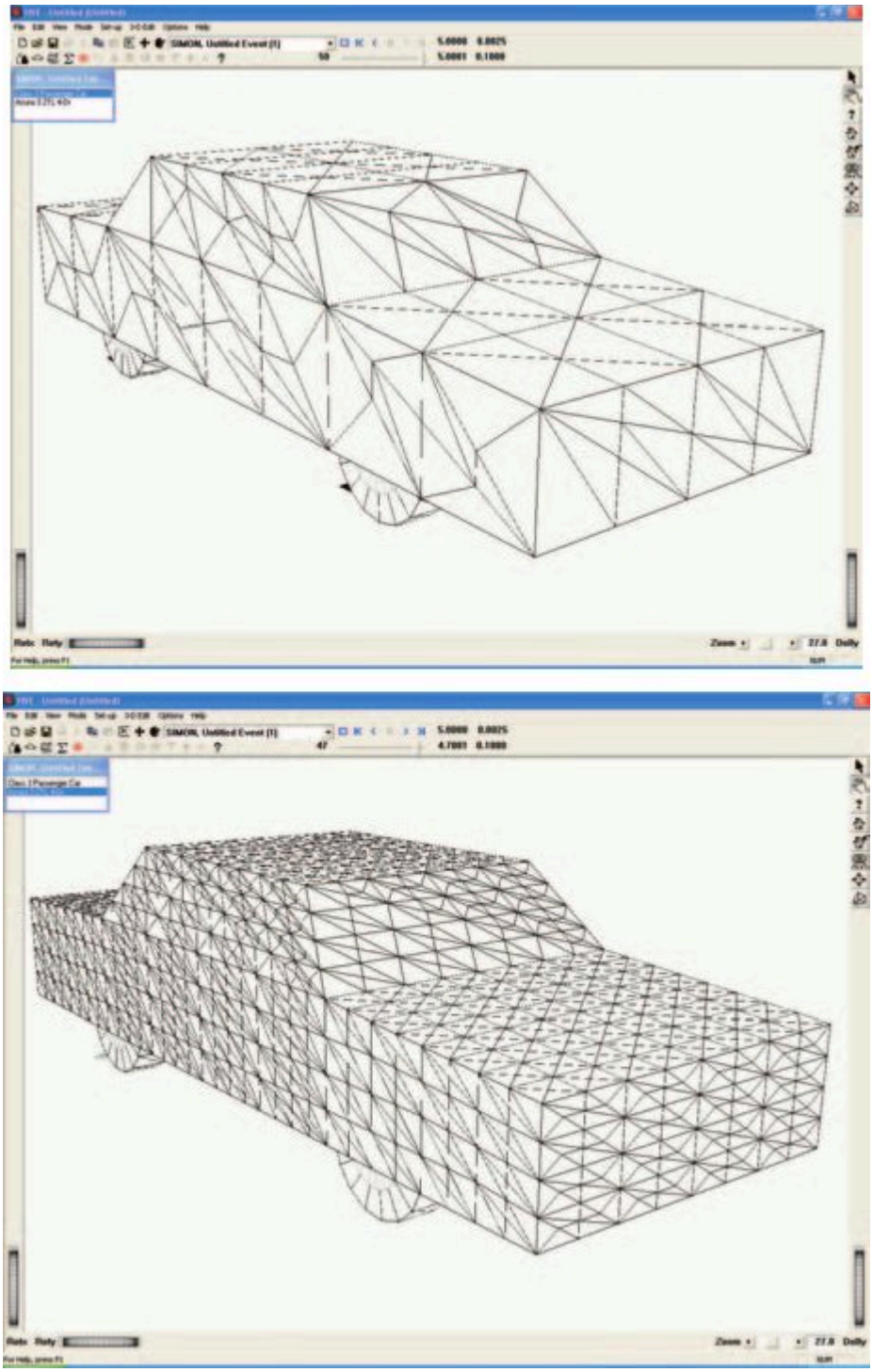


Figure 4-27: Generic Vehicle with 30 inch tessellation (above) and 10 inch tessellation (below).

Geometry

The Geometry group parameters provide various options that affect how the vehicle mesh behaves when it is deformed. The available options are:

- Weld Distance
- Tessellate

- Water-tightness

Weld Distance — If two vertices on the mesh lie within this distance from each other, they are assumed to be the same point. This helps to prevent two adjacent polygons from moving independently when the mesh deforms.

> **NOTE:** Two vertices a small distance from each other are usually the result of an error during the measuring or building of the mesh.

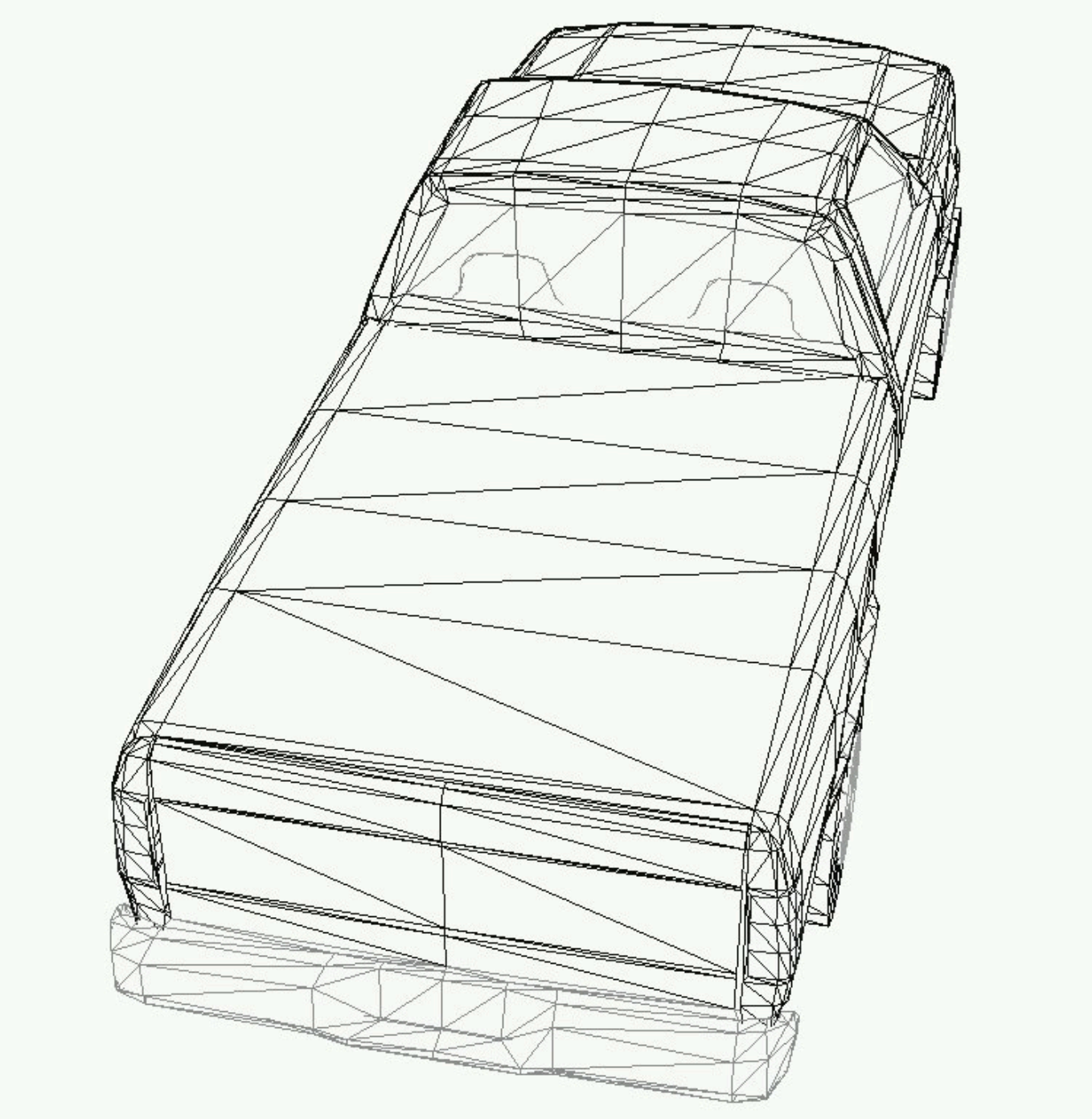


Figure 4-28: Region of vehicle with triangles having a large aspect ratio.

Tessellation — The Tessellation parameter establishes a maximum length for the side of any polygon in the mesh. Setting this value has two important effects: First, it establishes the maximum size of triangles in the mesh. Obviously a mesh with 10 inch polygons will provide more detail than a mesh with 30 inch polygons (see Figure 4-27). Second, it prevents a mesh from having triangles with large aspect ratios (i.e., triangles that are long and skinny; these triangles can be problematic for DyMESH); see Figure 4-28.

Although increased tessellation (i.e., smaller triangles) increases resolution, there is a cost. Increased tessellation also requires more computer memory, disk space and processing time.

> **NOTE:** Using a 10 inch Maximum Side Length for a 45-ft semi-trailer will bring your computer to its knees.

Water-tightness — A mesh is water-tight if each neighboring polygon shares its vertices. For example, the mesh at the left in Figure 4-29 is water-tight, while the mesh on the right is not. Note what happens when the vertex in the center of the mesh on the right is displaced: it opens up, resulting in a hole, as shown in Figure 4-30. This hole represents a region through which a secondary vertex on the other vehicle may pass. If that occurs, the secondary vertex will not find contact with any surface on the primary vehicle mesh, so the vertex will remain undeformed, resulting in what is called a rogue vertex (see Figure 4-31). The Water-tight test has three options:

- Yes (the mesh is water-tight)
- No (the mesh is known not to be water-tight)
- Unknown (it may or may not be water-tight)

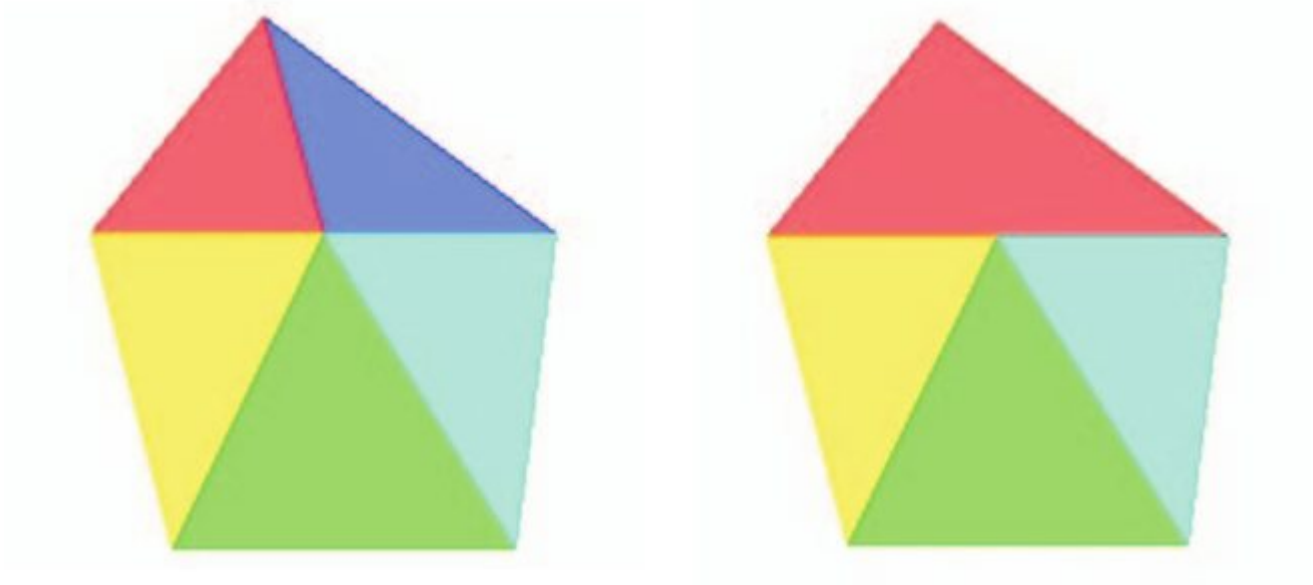


Figure 4-29: Water-tight mesh (left) and non-water-tight mesh (right).



Figure 4-30: Displacing the vertex in a non-water-tight mesh opens a hole in the mesh.

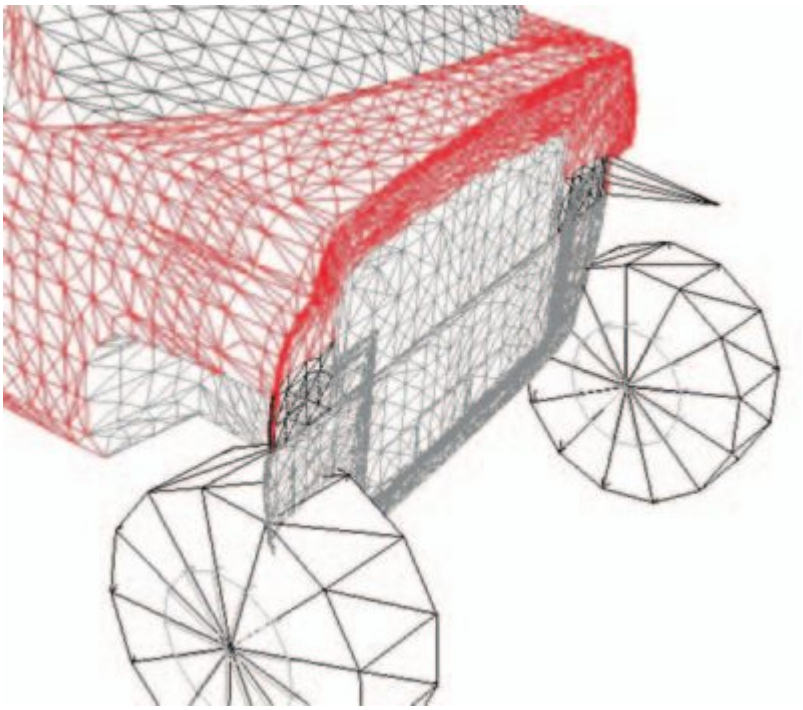


Figure 4-31: Rogue vertex resulting from collision with a barrier having a non-water-tight mesh.

If the mesh is water-tight, DyMESH can use an optimized procedure, thus speeding up processing. If the mesh is not water-tight, DyMESH can use a different procedure that prevents rogue vertices, but requires longer processing times. If the state of the mesh is unknown, DyMESH can determine its status at the start of the run, again taking longer to execute.

Inter-vehicle Options

The Inter-vehicle Options section of the dialog includes a Target Vehicle list box, and Inter-vehicle Friction and Restitution (relaxation length or relaxation energy) data fields.

> **NOTE:** These options operate on the selected pair of vehicles.

To edit the Vehicle Mesh properties (weld distance, tessellate, water-tight status, inter-vehicle friction and restitution), perform the following steps:

1. Select a vehicle from the Active Vehicles list.
2. Choose Set-up, Vehicle Mesh. The Mesh dialog is displayed.
3. Edit the weld distance as desired.
4. Click on the Tessellate checkbox and edit the current Maximum Side Length as desired.
5. Select the desired Water-tightness option (Yes, No, Unknown).
6. Select a vehicle or the environment from the list and edit the Friction and Relaxation Length as desired.
7. Press OK when the desired parameters are set or entered.

Parameters: The parameters assigned using this dialog are shown in Table 4-10.

Table 4-10: Mesh Parameters

Parameter	Units (US / SI)	Description
Maximum Side Length	in / cm	Determines the longest side of a polygon
Inter-vehicle Friction	—	Inter-vehicle friction coefficient for selected pair of vehicles
Relaxation Length	%/100	Determines the amount of reduction in displacement vector

See Also: DyMESH Options, User's Manual for selected simulation model

---

PAYLOAD

Menu Option: PAYLOAD

Purpose: Assign a payload to the selected vehicle

Description: The Payload dialog allows the user to add a payload to the selected vehicle. The payload may be any object adding inertia to a vehicle that is not accounted for in the vehicle's inertial properties. Examples include cargo and occupants. The Payload dialog is shown in Figure 4-32.

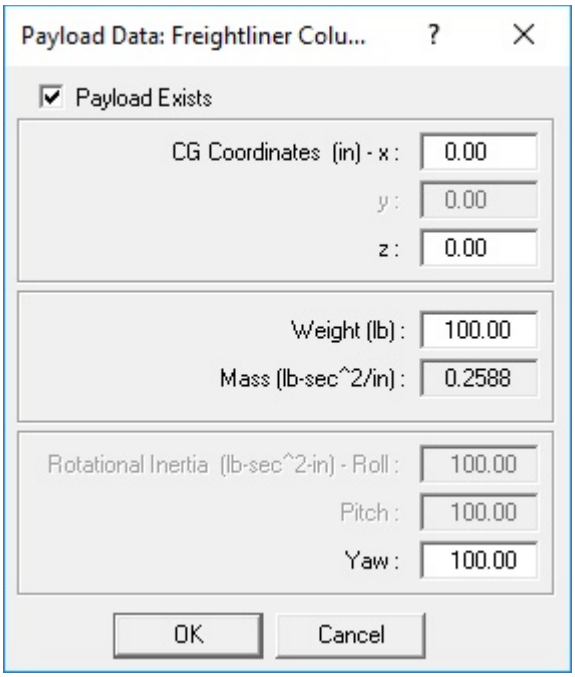


Figure 4-32: The Payload dialog allows the user to assign a payload to the current vehicle.

The payload is defined by the following information:

- **Location** — The x,y,z distances from the vehicle's center of gravity to the payload's center of gravity.
- **Weight** — The weight of the payload.
- **Rotational Inertia** — The roll, pitch and yaw moments of inertia of the payload.

> **NOTE:** The entered weight value is divided by the current gravitational constant and stored as mass. Therefore, if you take the current vehicle to the moon, the payload's inertial properties will still be correct!
>
> **NOTE:** The vehicle's CG moves when you add a payload. HVE automatically updates the earth-fixed coordinates of the vehicle's CG to account for this fact. You will see the updated CG coordinates in the Position/Velocity dialog.

To enter Payload Data for the current vehicle, use the following steps:

1. Select the vehicle.
2. Choose Payload from the Set-up menu. The Payload dialog will be displayed.
3. Enter the desired payload parameters.

> **NOTE:** You can often use a simple mechanics formula to estimate rotational inertia, e.g., Izz = (m/12)(a^2 + b^2). 4. Press OK when the desired data are entered.

Parameters: The parameters assigned using this dialog are shown in Table 4-11.

Table 4-11: Payload Parameters

Parameter	Units (US / SI)	Description
x,y,z coordinates	in / cm	x, y and z distances from the vehicle CG to the Payload CG
Weight	lb / N	Payload weight
Rotational Inertia	lb-sec²-in / kg-m²	Payload rotational inertia about its x, y and z axes

See Also: User's Manual for individual reconstruction or simulation model

---

WHEELS

Menu Option: WHEELS (Ctrl+W)

Purpose: Assign event-related parameters for each wheel

Description: The Wheels option allows the user to select from a cascade menu the following options:

- **Blow-out** — Simulate the transient effects of an air loss at one or more tires
- **Damage** — Simulate the transient effects of wheel damage (normally the result of a collision) at one or more wheels
- **Brakes** — Simulate the effects of slack adjuster adjustment and initial temperature at one or more wheels on braking effectiveness

> **NOTE:** The default adjustment is assigned using the Vehicle Editor. This dialog allows further adjustment at individual wheels.
>
> **NOTE:** Initial temperature is the environment ambient temperature. - **Tire-Terrain** — Assign the desired Tire-Terrain model for each tire

Each of the Wheels options has an appropriate dialog; the use of these dialogs is described below.

Tire Blow-out Option

The HVE Tire Blow-out Model is set up using the Tire Blow-out dialog, shown in Figure 4-33. The tire blow-out model allows the user to vary tire performance parameters (these parameters are cornering, camber and radial stiffness and rolling resistance) while the simulation is running.

To assign tire blow-out parameters for one or more tire locations, perform the following steps:

1. Choose Wheels from the HVE Set-up menu. The Wheel Data dialog is displayed; the options are Blow-out, Damage, Brakes and Tire-Terrain.
2. Choose Blow-out. The HVE Tire Blow-out dialog is displayed, as shown in Figure 4-33.
3. Click on the Axle option list and select the desired axle location for the tire blow-out.
4. Click on the Side option list and select the Right or Left side.
5. Click on the Location radio button and select the Inner or Outer tire location.

The current blow-out options for the selected tire are displayed. By default, the tire is in its normally inflated state. To simulate an air loss, perform the following steps:

1. Click the Is Blown check box. The HVE Tire Blow-out Model parameters become editable.
2. Click AutoStart if you want the simulation model to determine when the air loss begins. Normally, the air loss begins when the simulation detects a collision.
3. Assign a Start Time for the air loss.

> **NOTE:** Start Time is disabled if AutoStart is selected. 4. Assign a Time Duration for the air loss.
>
> **NOTE:** The default duration is 0.1 seconds. High speed film of tire blow-outs shows this is a reasonable value for a 1 inch (2.5 cm) hole in the sidewall. You should adjust this value if the size of the hole is larger or smaller.
>
> **NOTE:** You can simulate a slow air loss by using a large time duration. 5. Assign Stiffness and Rolling Resistance Factors for the air loss.
>
> **NOTE:** Review the documentation for the particular simulation model you are using to determine whether all the parameters are used. 6. If desired, use the Axle, Side and Location option lists to select additional tires for assigning blow-out parameters.
>
> **NOTE:** Blow-out parameters may be assigned to any number of tires. 7. Press OK to accept the tire blow-out parameters.

When the simulation is executed, the selected blow-out parameters will be incorporated into the simulation.

> **NOTE:** The HVE Tire Blow-out Model is further described in Chapter 16, Event Model.

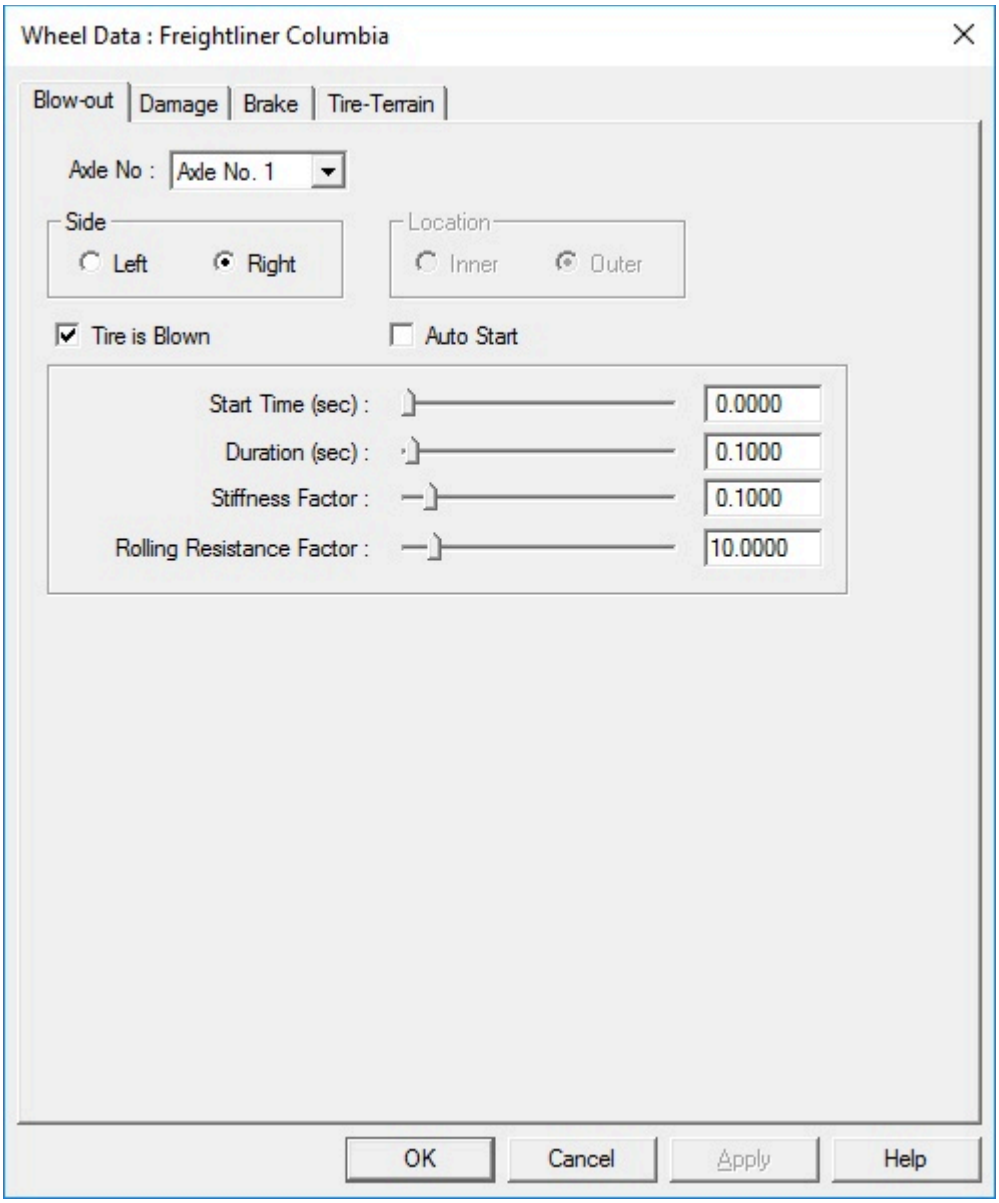


Figure 4-33: The Tire Blow-out dialog allows the user to assign tire blow-out parameters to one or more tires.

Wheel Damage Option

The Damage option is set up using the Wheel Damage dialog, shown in Figure 4-34. By specifying wheel displacement(s), the user can dynamically vary wheel position and orientation during the simulation. Another application for the Wheel Damage dialog is assigning wheel lock-ups, normally caused by damage during a collision.

To assign wheel damage parameters for one or more wheel locations, perform the following steps:

1. Choose Wheels from the HVE Set-up menu. The Wheel Options cascade menu is displayed; the options are Blow-out, Damage, Brakes and Tire-Terrain.
2. Choose Damage. The Wheel Damage dialog is displayed, as shown in Figure 4-34.
3. Click on the Axle option list and select the desired axle location for the damaged wheel.
4. Click on the Side option list and select the Right or Left side.

The current damage options for the selected wheel position are displayed. By default, the wheel is at its normal position and orientation. To simulate a wheel displacement, perform the following steps:

1. Click the Is Displaced check box. The Wheel Displacement parameters become editable.
2. Click AutoStart if you want the simulation model to determine when the displacement begins. Normally, the displacement begins when the simulation detects a collision.
3. Assign a Start Time for the displacement.

> **NOTE:** Start Time is disabled if AutoStart is selected. 4. Assign a Time Duration for the displacement.
>
> **NOTE:** The default duration is 0.1 seconds, the length of a typical collision. High speed film of crash tests reveals this is a reasonable value. You should adjust this value if the displacement occurs over a shorter or longer interval. 5. Assign Δx, Δy, Δz wheel displacements.
>
> **NOTE:** The displacements are vehicle-fixed; a negative value moves the wheel in the negative x, y or z vehicle-fixed directions. 6. Assign the Camber Change.
>
> **NOTE:** The Camber Change is a vehicle-fixed angle. A positive value moves the top of the wheel outward; a negative value moves the top of the wheel inward.

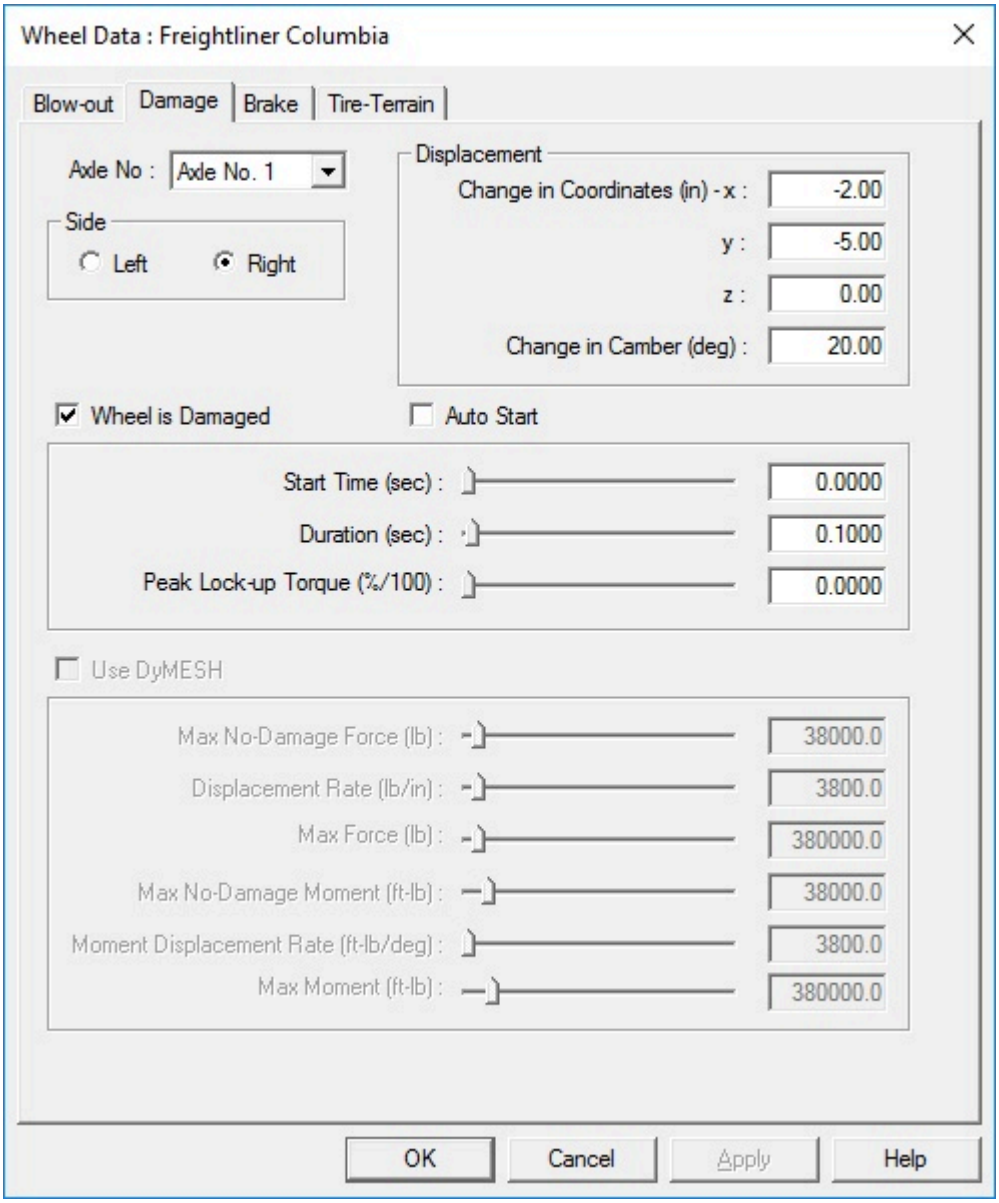


Figure 4-34: The Wheel Damage dialog allows the user to assign a dynamic change in wheel position and orientation during the simulation.

To simulate wheel lock-up, perform the following steps:

1. Assign the desired percentage of the nominal friction required to fully lock the wheel under static loading conditions.

> **NOTE:** Entering a value of 2.00 will ensure that the wheel locks under any reasonable condition. 2. If desired, use the Axle and Side option lists to select additional wheel positions for assigning displacement parameters.
>
> **NOTE:** Damage parameters may be assigned to any number of wheel positions. 3. Press OK to accept the wheel damage parameters.

When the simulation is executed, the selected wheel displacement parameters will be incorporated into the simulation.

Wheel Brakes Option

The Wheel Brakes option allows the user to simulate the effects of brake adjustment on air brake systems by specifying initial stroke and lining and drum temperature at each wheel.

To assign slack adjuster and temperature conditions at each wheel location, perform the following steps:

1. Choose Wheels from the HVE Set-up menu. The Wheel Options cascade menu is displayed; the options are Blow-out, Damage, Brakes and Tire-Terrain.
2. Choose Brakes. The Wheel Brakes dialog is displayed, as shown in Figure 4-35.
3. Click on the Axle option list and select the desired axle location for the brake data.
4. Click on the Side option list and select the Right or Left side.

The default conditions for the selected wheel position are displayed. By default, the brake adjustment is that which was assigned using the Vehicle Editor. The temperature is the environment ambient temperature. To assign new conditions, perform the following steps:

1. Assign an Initial Lining Temperature for the brake.
2. Assign an Initial Drum Temperature for the brake.
3. Assign Adjuster Slack for the brake.

Brake failure may also be simulated. To simulate a brake failure at the selected wheel location, perform the following steps:

1. Click the Brake Is Failed checkbox. The Brake Failure parameters are enabled.
2. If desired, click the AutoStart checkbox to have the simulation automatically set the time for the start of the brake failure.
3. Enter the Start Time and Duration (the time interval during which the brake effectiveness goes from normal to failed) for the brake failure.
4. Enter the Failure Extent as a percentage of total brake failure for the selected brake.
5. If desired, use the Axle and Side option lists to select additional wheel positions for assigning brake temperature, adjustment and failure parameters.

> **NOTE:** Brake parameters may be assigned to any number of wheel positions. 6. Press OK to accept the wheel brake parameters.

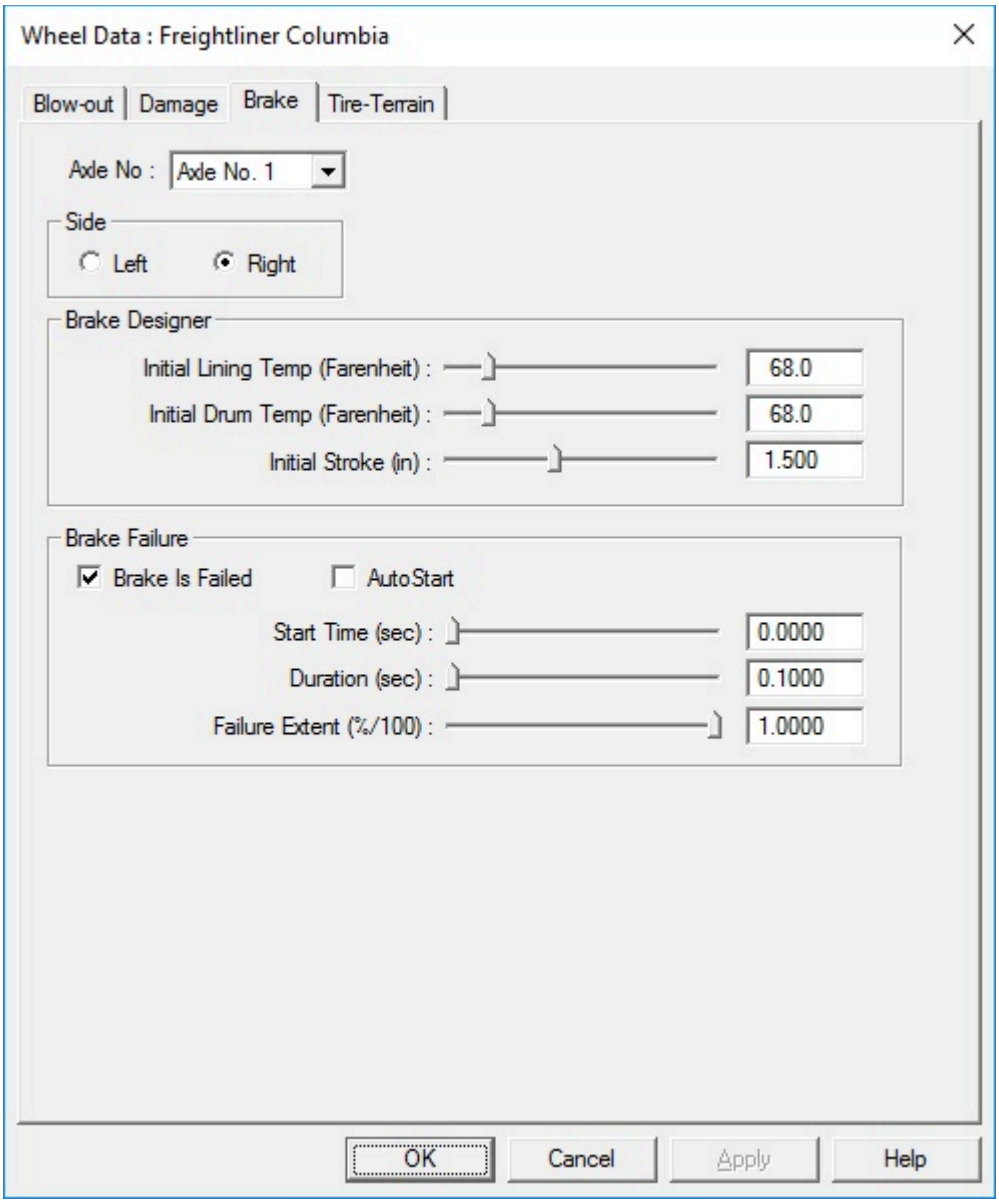


Figure 4-35: The Wheel Brakes dialog allows the user to assign initial brake slack adjuster adjustment and lining and drum temperatures for air brake systems. The Wheel Brakes dialog is also used for simulating brake failure at one or more wheel locations.

When the simulation is executed, the selected wheel brake parameters will be incorporated into the simulation.

Tire-Terrain Option

The Tire-Terrain option provides the user with several methods for simulating tire interaction with various types of terrains, such as flat surfaces, potholes, curbs and soft soil. This is done by selecting an appropriate tire-terrain model. Three tire models are available:

- Point Contact Model
- Radial Spring Model (with Sidewall Impact Option)
- Soft Soil Model

Each of these tire-terrain models is described below.

Point Contact Tire-Terrain Model

The Point Contact Tire-Terrain model assumes that the tire force, Fx', Fy', is a shear force acting at the ground contact plane at a single point directly beneath the wheel center (see Figure 4-36). This is a good assumption for a majority of conditions. In particular, the Point Contact model assumes a flat (not necessarily horizontal), rigid terrain. The Point Contact model is the default Tire-Terrain option.

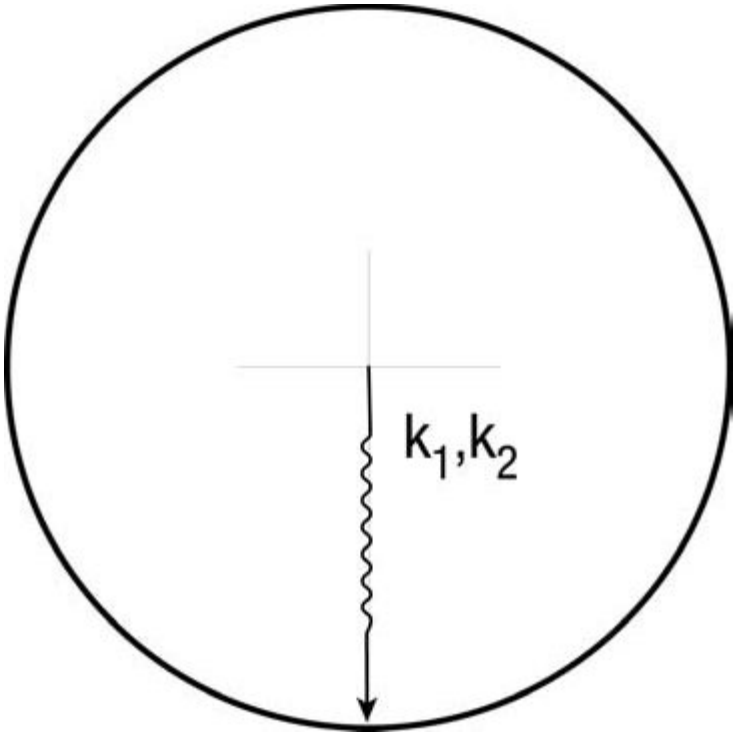


Figure 4-36: The Point Contact Tire-Terrain Model (default) assumes the tire force acts at a single point directly below the wheel center.

Radial Spring Tire-Terrain Model

The Radial Spring Model models the tire as a series of radial springs lying in the tire plane projecting radially outward from the wheel center (see Figure 4-37). Whereas the Point Contact Model may be thought of as a single spring projecting downward and interacting with the terrain at a single point, the Radial Spring Model provides numerous springs that may interact with multiple surfaces. This model is ideal for simulating tires mounting curbs and rolling over potholes.

The radial springs are defined by the following parameters:

Radial Spring Rate, K1, K2 — This 2-stage, bi-linear spring rate is derived from the tire's physical properties (initial and secondary stiffness). These rates are distributed between individual springs when the model is initialized. This initialization process results in an equivalent tire deflection for the Radial Spring Model when compared to the Point Contact model.

Angular Span, Ω — It would be computationally unnecessary (and wasteful) to calculate radial spring forces about the entire perimeter of the tire, since only a small portion at the bottom of the tire is normally in contact with the ground. Starting at the ground, the Angular Span is an angle that defines how much of the tire's perimeter is modeled by radial springs (see Figure 4-37). The default value is +/-90 degrees, meaning that the entire lower half of the tire is modeled. For normal contact with relatively flat terrain, this value may be reduced to +/-30 degrees. A logical test produces a message if the Angular Span needs to be increased.

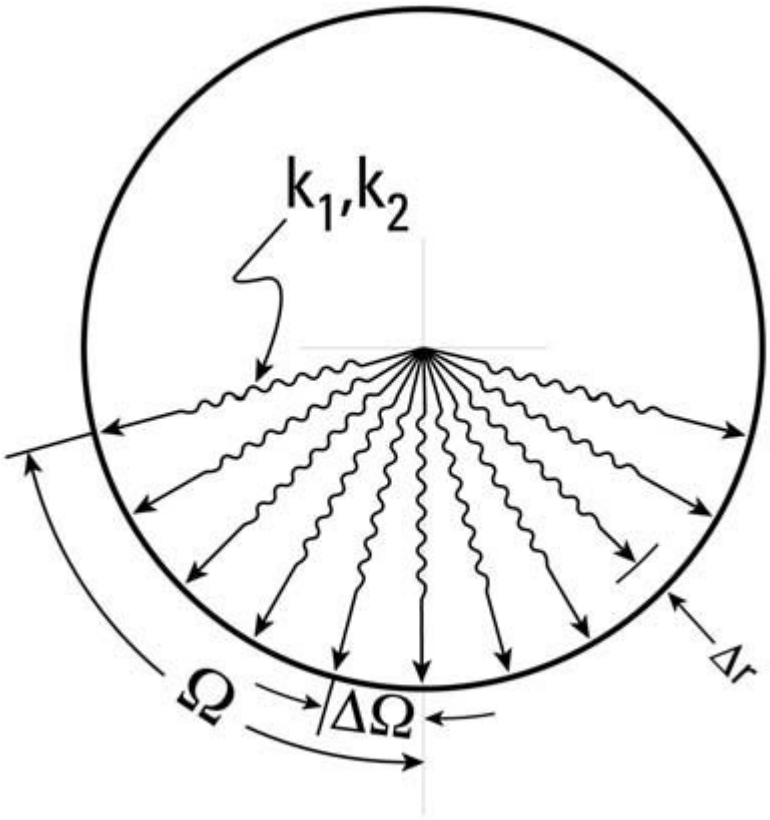


Figure 4-37: The Radial Spring Tire-Terrain Model uses evenly spaced springs, rather than a single spring, to model the radial tire force.

Angular Increment, ΔΩ — The radial springs are placed at discrete angular intervals about the tire. This angle defines the increment between radial springs (see Figure 4-37).

Radial Adjustment Increment, Δr — The current force on each radial spring is determined by its displacement (see Figure 4-37) according to the terrain elevation beneath the spring (actually calculated as the intersection of the radial spring vector with the terrain). The Radial Adjustment Increment is used, along with quadratic interpolation of the radial spring look-up table stored for each radial spring, to determine the radial spring force.

Sidewall Impact Tire Model

The Sidewall Impact Tire Model is an extension of the Radial Spring Tire Model. In the Sidewall Impact Tire Model, each radial spring (see above) has springs projecting laterally from the wheel plane to the tire sidewall plane (see Figure 4-38).

The lateral springs are defined by:

Sidewall Slide Friction Coefficient, μs — This parameter defines the coulomb friction between the tire sidewall and the terrain. This value has a null band that requires relative velocity between the tire and terrain in order to produce frictional force (in the direction of the tire sidewall tangential velocity relative to the terrain).

Number of Sidewall Springs, n — This parameter defines the number of lateral springs for each radial spring. The user may specify two, four or six lateral springs.

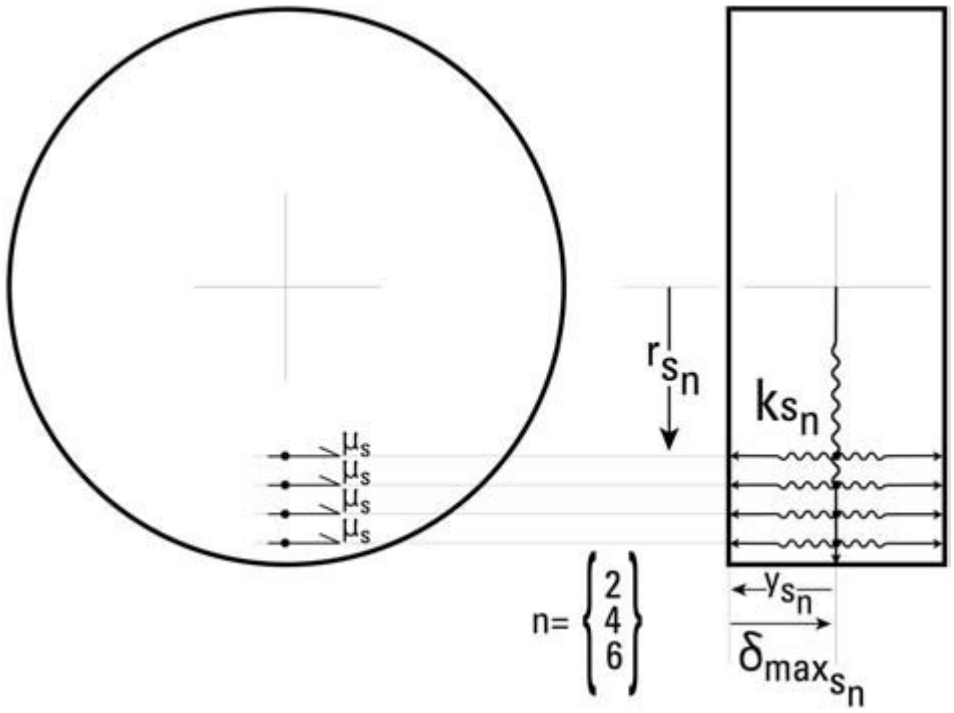


Figure 4-38: The Sidewall Impact Tire-Terrain Model is an extension of the Radial Spring Model, and simulates forces applied to the tire sidewall.

Radial Distance from Wheel Center, rsn — The springs are evenly spaced at this radial distance from the wheel center. By default, the springs are located along the section height of the tire (the spacing differs according to the tire section height and the number of sidewall springs, above).

Lateral Distance from Wheel Center, ysn — This distance specifies the resting length of the springs. By default, this length is one-half of the tire width.

Sidewall Spring Stiffness Ratio, ksn — This value provides the linear stiffness of each sidewall spring. The stiffness is assigned as a ratio of the tire radial spring stiffness. By default, the sidewall stiffness is equal to the tire radial spring stiffness divided by the number of sidewall springs.

Saturation Deflection, dmax,sn — This value represents the maximum sidewall spring deflection. The simulation terminates if the lateral spring deflection exceeds this value.

The forces acting at the tire sidewall produce additional forces and moments. Because these forces are not friction-limited, they can be much greater than forces generated at the tire-road interface. Sidewall forces also produce moments about the steering axis, affecting vehicle behavior when the Steer DOF Model option is used.

Soft Soil Tire Model

The Soft Soil Tire Model is based on research performed by M.G. Bekker [5.3] at the University of Michigan. The purpose of part of Bekker's research was to predict the performance and tractive effort requirements of the lunar rover (since no one had previously driven on the surface of the moon!). In the model (see Figure 4-39), tire sinkage into the soil is calculated using an empirical model developed by Bekker. The empirical model uses three soil descriptors:

Bekker Soil Deformation Exponent, N — An empirically derived value describing soil deformation characteristics.

Soil Frictional Modulus, Kphi — An empirically derived value describing soil frictional characteristics.

Soil Cohesive Modulus, Kc — An empirically derived value describing soil cohesive characteristics.

The reference includes a table of these soil descriptors for approximately 40 different soils of varying clay and moisture content. Many of these soils are included in the HVE Terrain Soils Database. Each terrain polygon includes a material attribute having these descriptors. The Soft Soil Tire Model uses these descriptors and current vertical tire load and size (unloaded radius and current rolling radius, and section width) to calculate the current soil sinkage beneath the tire. Sinkage also affects the earth-fixed elevation of the tire contact patch. Based on the calculated sinkage, the projected frontal and sidewall areas of the tire sinkage are calculated. These areas are then used in conjunction with the three soil descriptors (above) to calculate the Fx' and Fy' tire forces.

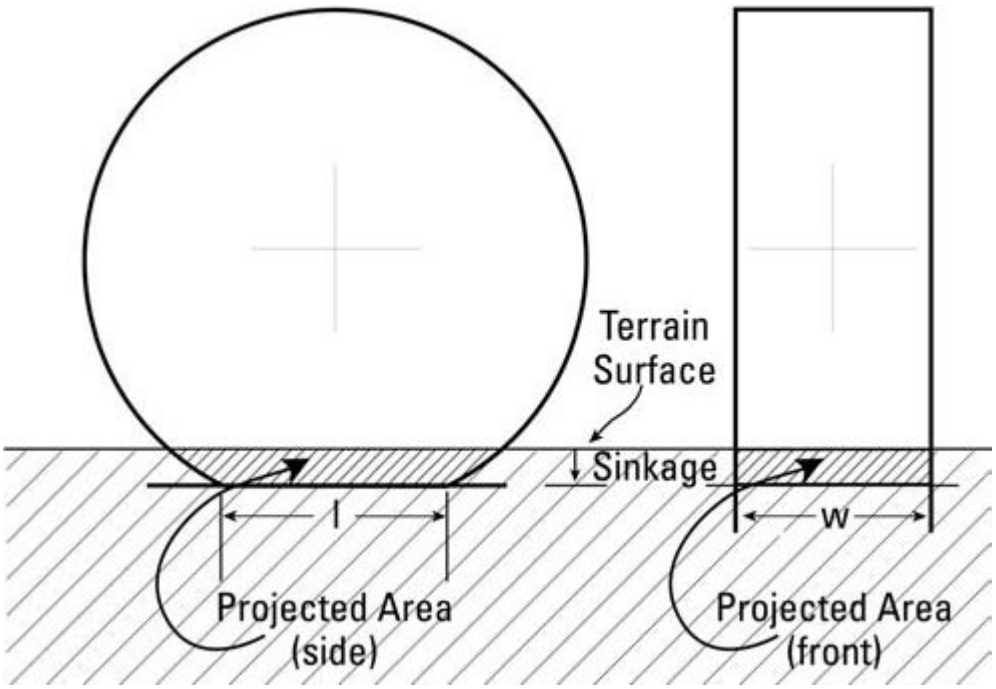


Figure 4-39: The Soft Soil Tire-Terrain Model calculates the forces on a tire that is sinking into the terrain.

Assigning Tire-Terrain Model Attributes

To assign the desired tire-terrain model and associated parameters, perform the following steps:

1. Choose Wheels from the HVE Set-up menu. The Wheel dialog is displayed, showing the option tabs: Blow-out, Damage, Brakes and Tire-Terrain.
2. Choose Tire-Terrain. The Tire-Terrain dialog is displayed, as shown in Figure 4-40.
3. Click on the Axle option list and select the desired axle location for the tire-terrain data.
4. Click on the Side option list and select the Right or Left side.
5. Click on the Location radio button and select the Inner or Outer tire location.

The Point Contact Tire-Terrain Model is the default selection. To choose a different model:

1. Click the Soft Soil radio button to select that option. When the tire travels over a soft soil (as determined using the Point Contact model approach), HVE's surface search will apply the Bekker soil parameters for the terrain beneath the tire.
2. Click the Radial Spring radio button to select that option. The additional attributes required by the Radial Spring Tire-Terrain Model will be enabled. Edit the default parameters, if desired.

3. Click the Sidewall Impact check box to select that option. The additional attributes required by the Sidewall Impact option will be enabled. Edit the default parameters, if desired.
4. If desired, use the Axle, Side and Location option lists to select additional tire positions and select the appropriate tire-terrain model.

> **NOTE:** Different Tire-Terrain models may be assigned to each tire location. 5. Press OK to accept the tire-terrain parameters.

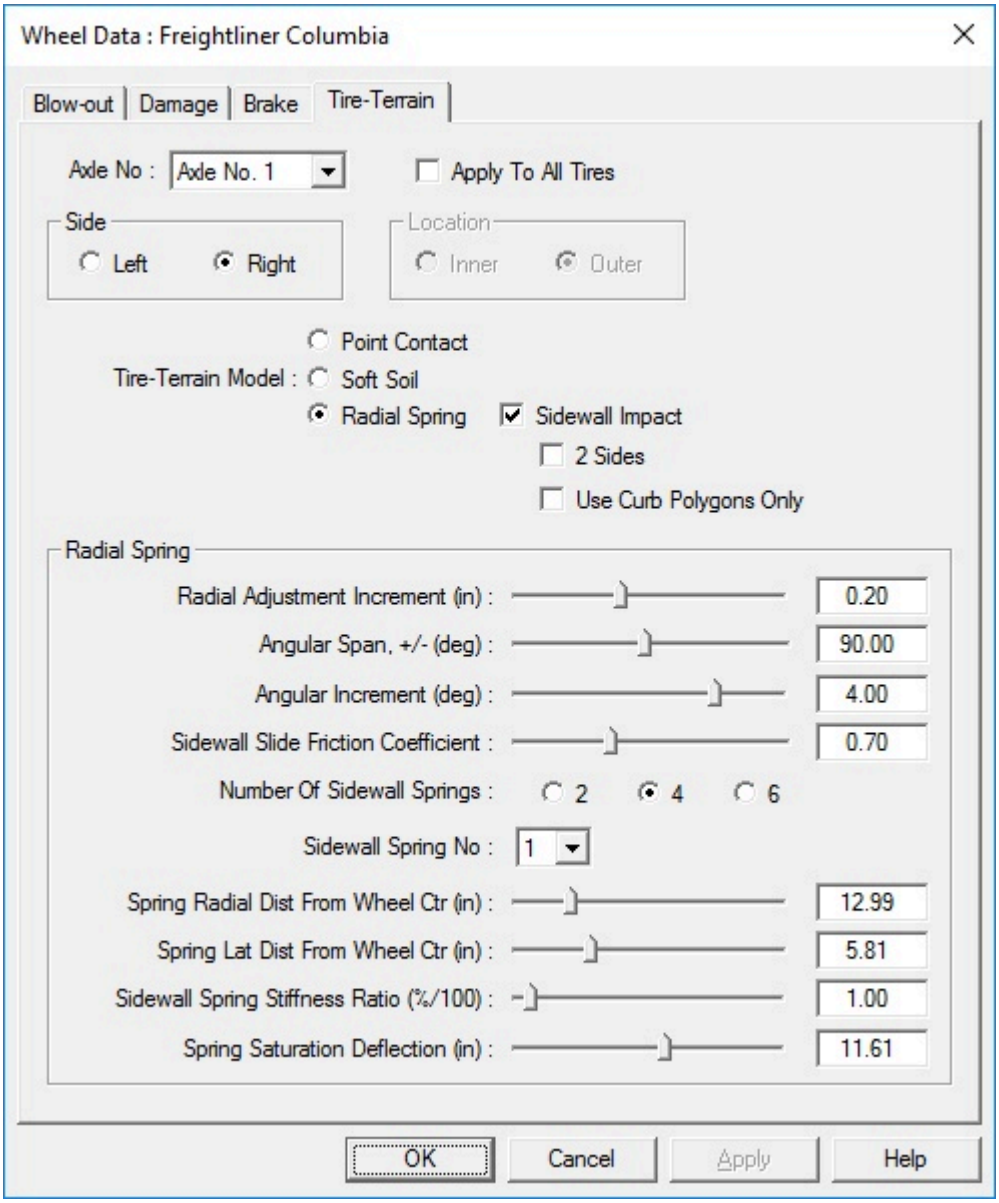


Figure 4-40: The Tire-Terrain dialog allows the user to select from several available tire-terrain models useful for simulating tire interaction with potholes, curbs and soft soils.

See Also: Vehicle Editor, Brake Information dialog, Vehicle Output Tracks

Parameters: The parameters assigned using the Wheels dialogs are shown in Table 4-12.

Table 4-12: Wheel Editing Parameters

Parameter	Units (US / SI)	Description
Axle Number	—	Axle index (1, 2 or 3)
Side	—	Side index (Right or Left)
Location	—	Tire location index (Inner or Outer)
Tire Is Blown	—	Flag indicating tire is blown
AutoStart	—	Flag indicating start time is set by simulation
Start Time	sec	Simulation time at which blow-out or wheel displacement starts
Duration	sec	Duration over which air loss or wheel displacement occurs
Stiffness Factor	—	Multiplier for tire stiffness parameters
Rolling Resistance Factor	—	Multiplier for tire rolling resistance
Change in Coordinates	in / cm	Vehicle-fixed wheel displacement distance
Change in Camber	deg	Vehicle-fixed wheel camber change angle
Percent Peak Lock-up Torque	%/100	Percentage of nominal torque required to lock wheel
Initial Lining Temp	deg F / deg C	Temperature of brake lining at the start of the simulation
Initial Drum Temp	deg F / deg C	Temperature of brake drum at the start of the simulation
Adjuster Slack	in / cm	Slack adjuster stroke before brake torque begins
Percent Brake Failure	%/100	Fraction of total brake failure at the selected wheel location (0 = no failure; 1 = total failure)
Tire-Terrain Model	—	Tire-Terrain Model option
Sidewall Impact	—	Flag indicating the Sidewall Impact option is to be used
Radial Adjustment Increment	in / cm	Incremental change in radial spring length used in spring force calculation
Angular Span	deg	Sweep angle for radial springs
Angular Increment	deg	Angular increment between radial springs
Sidewall Slide Friction Coefficient	—	Tire sidewall slide friction coefficient
Number of Sidewall Springs	—	Number of sidewall springs attached to each radial spring
Spring Number	—	Spring index
Spring Radial Dist from Wheel Center	in / cm	Radial distance from wheel center to current lateral spring
Spring Lateral Dist from Wheel Center	in / cm	Distance from tire centerline plane to tire sidewall for current spring
Sidewall Spring Stiffness Ratio	—	Ratio of sidewall spring stiffness to radial spring stiffness
Spring Saturation Deflection	in / cm	Maximum allowable deflection for current sidewall spring

ACCELEROMETERS

Menu Option: ACCELEROMETERS

Purpose: Assign the locations for up to five vehicle-fixed accelerometers

Description: The Accelerometers dialog allows the user to assign the vehicle-fixed x,y,z coordinates for up to five accelerometers. The simulation model can then return the velocity and acceleration at each selected location. Accelerometers are assigned using the Accelerometers dialog, as shown in Figure 4-41.

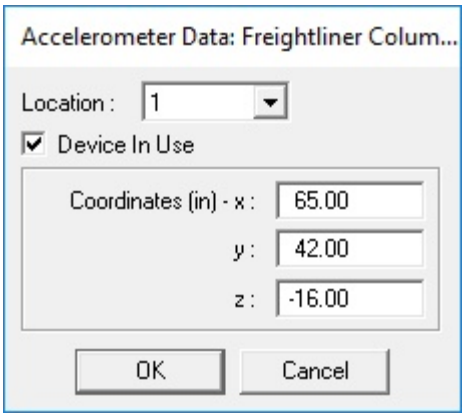


Figure 4-41: Accelerometer Data dialog (Vehicle Accelerometers).

To assign one or more accelerometers to the current vehicle, perform the following steps:

1. Click on the Location Number option list and choose a device location.
2. Click the Device In Use check box. The Device Coordinates become editable.
3. Enter the vehicle-fixed coordinates of the accelerometer.
4. Repeat the above three steps for each additional location.
5. Press OK to assign the accelerometer location data.

The simulation reports the velocity and acceleration for the current timestep in the output tracks, Accelerometers group.

Parameters: The parameters assigned using this dialog are shown in Table 4-13.

Table 4-13: Accelerometer Parameters

Parameter	Units (US / SI)	Description
Device Number	—	Device selector number
DeviceIsUsed	—	Flag indicating the selected device is in use
x-coord, y-coord, z-coord	in / cm	Vehicle-fixed coordinates of accelerometer

See Also: Event Model Definition, Output Tracks. *(updated: see also the Options menu items Show Accelerometers and Show Accelerometer Paths, which visualize these locations — Options Menu)*

CONTACTS

(updated: in the current menu, Contacts appears before Restraints; the manual's original order is retained here to match the printed edition's cross-references. See also the dialog reference: Contacts Dialog Box.)

Menu Option: CONTACTS (Ctrl+T)

Purpose: Choose allowable human ellipsoid vs vehicle contact surface interactions during calculations and assign combined material attributes for the selected pair of contacts

Description: The Contacts dialog allows the user to select the allowable interactions between human ellipsoids and vehicle contact surfaces. For example, there is little chance of a human pedestrian's pelvis interacting with the vehicle's interior contact surfaces (e.g., the seat) during the simulation. Therefore, it makes little sense to perform force calculations for this ellipsoid/contact surface pair. By selecting only those contact surfaces having a chance of interaction, far fewer calculations need to be performed. As a result, the calculation time can be reduced significantly. The Contacts dialog is shown in Figure 4-44.

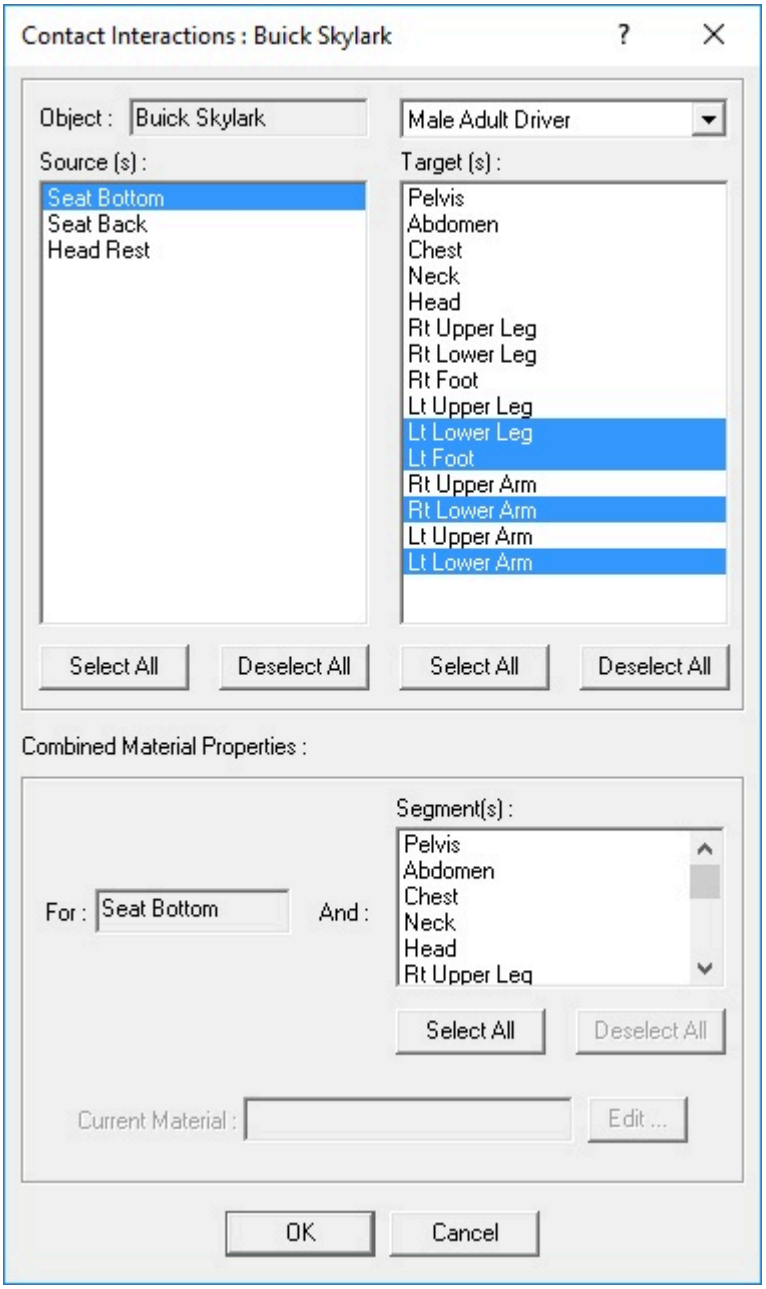


Figure 4-44: The Contacts dialog allows the user to select the allowable interactions between human ellipsoids and vehicle contact surfaces during an occupant or pedestrian simulation.

The Contacts dialog has three list boxes:

- **Source Segment** — A multiple selection list box containing all the contacts (human ellipsoids or vehicle contact surfaces) defined for the source object
- **Target Segment** — A multiple selection list box containing all the contacts (human ellipsoids or vehicle contact surfaces) defined for the target object
- **Combined Materials Target Segment** — A multiple selection list box containing all the contacts (human ellipsoids or vehicle contact surfaces) defined for the target object; only those segments selected in the multiple selection Target Segment list box (described above) are selectable in this list box

The source object is the object selected in the Event Editor dialog's Event Humans and Vehicles list box. This object is automatically selected before the Contacts dialog is displayed. The target object is selected from the Contacts dialog's Target Object option list. Once selected, the available contacts are displayed in the Source Segment and Target Segment list boxes. For humans, the list box will display all the human ellipsoids. For vehicles, the list box will display all the vehicle contact surfaces.

> **NOTE:** The Source Object and Target Object may be the same object. For example, the same human object may be selected from both option lists. This allows the user to define the interaction between the selected human's upper arm and torso, for example.

By default, all the vehicle interior contact surfaces are pre-selected for occupant simulations; all vehicle exterior contact surfaces are pre-selected for pedestrian simulations.

Combined Material Properties

Some simulation models define the human ellipsoid and vehicle contact surface material properties according to the sole properties of the ellipsoid or contact surface; other simulations define the material properties as a combined property shared by the two interacting objects. In the latter case, a combined material property must be assigned for each interacting ellipsoid-contact surface pair.

> **NOTE:** In the former case, the material property is assigned directly in the Human and Vehicle Editors.

The combined material attributes for the selected Source and Target Segments are assigned using Combined Material Properties. By selecting a source and target segment, the Combined Materials Edit pushbutton is enabled. Pressing this button causes the Combined Materials dialog to be displayed (see Figure 4-45).

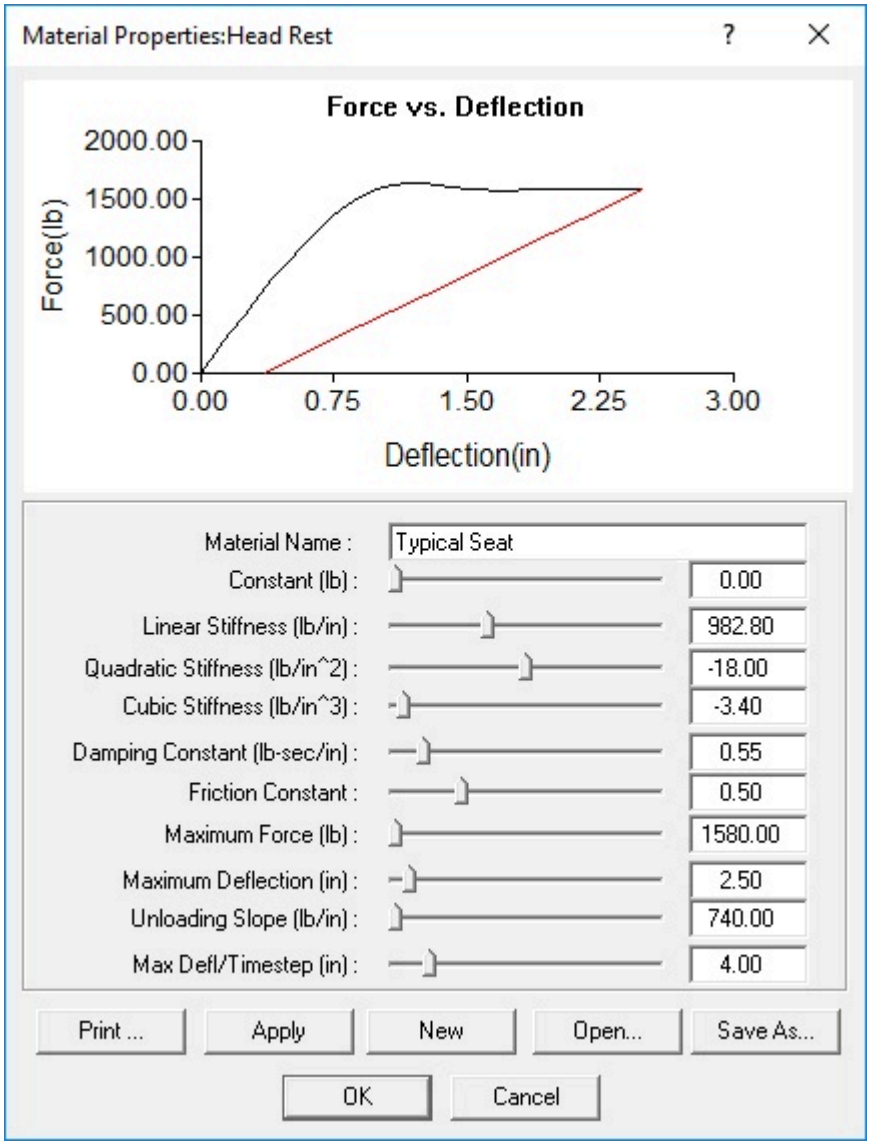


Figure 4-45: The Combined Materials dialog allows the user to review and edit the material physical parameters associated with interaction between the selected source and target segments. The dialog also includes a graph of the force-deflection relationship for the selected segments.

To select the allowable interactions for the current event, use the following steps:

1. Select either the human or the vehicle from the Event Humans and Vehicles list box. This object will be the source object.

> **NOTE:** For vehicle occupant simulations, it is convenient to select the human. 2. Choose Contacts from the Set-up menu. The Contact Interactions dialog is displayed. The selected human or vehicle (see above step) becomes the Source Object. 3. Select one or more segments from the Source Object list box. If the selected object is a human, it may be convenient to click Select All. This will assign contacts for all human segments in a single step. 4. Click on the Target Object option list and choose a target object. The Target Segments list box displays all the available target segments; the default selections are highlighted. 5. Select (highlight) or deselect (unhighlight) the desired segments in the Target Segments list box so only the desired interactions are selected.

If the simulation model requires combined material properties, the Combined Material Properties' Target Segment list box will be enabled. In this case continue with the following optional steps:

1. Choose a target segment from the Target Segment list box.

> **NOTE:** Only those segments selected earlier (see above) are selectable in this list box.
>
> **NOTE:** If multiple segments are selected in the Source Segment list box, combined material attributes are assigned to all source segments. 2. Press Edit. The Combined Materials dialog is displayed for the selected source and target segments (see Figure 4-45).
>
> **NOTE:** The default attributes are calculated using an arithmetic average of the individual properties for the selected segments. 3. Edit the desired material attributes. 4. Press Apply. The force-deflection curve for the entered material properties is displayed.
>
> **NOTE:** The user also enters a Maximum Force. An error message is displayed if the maximum force cannot be achieved using the entered material constants.

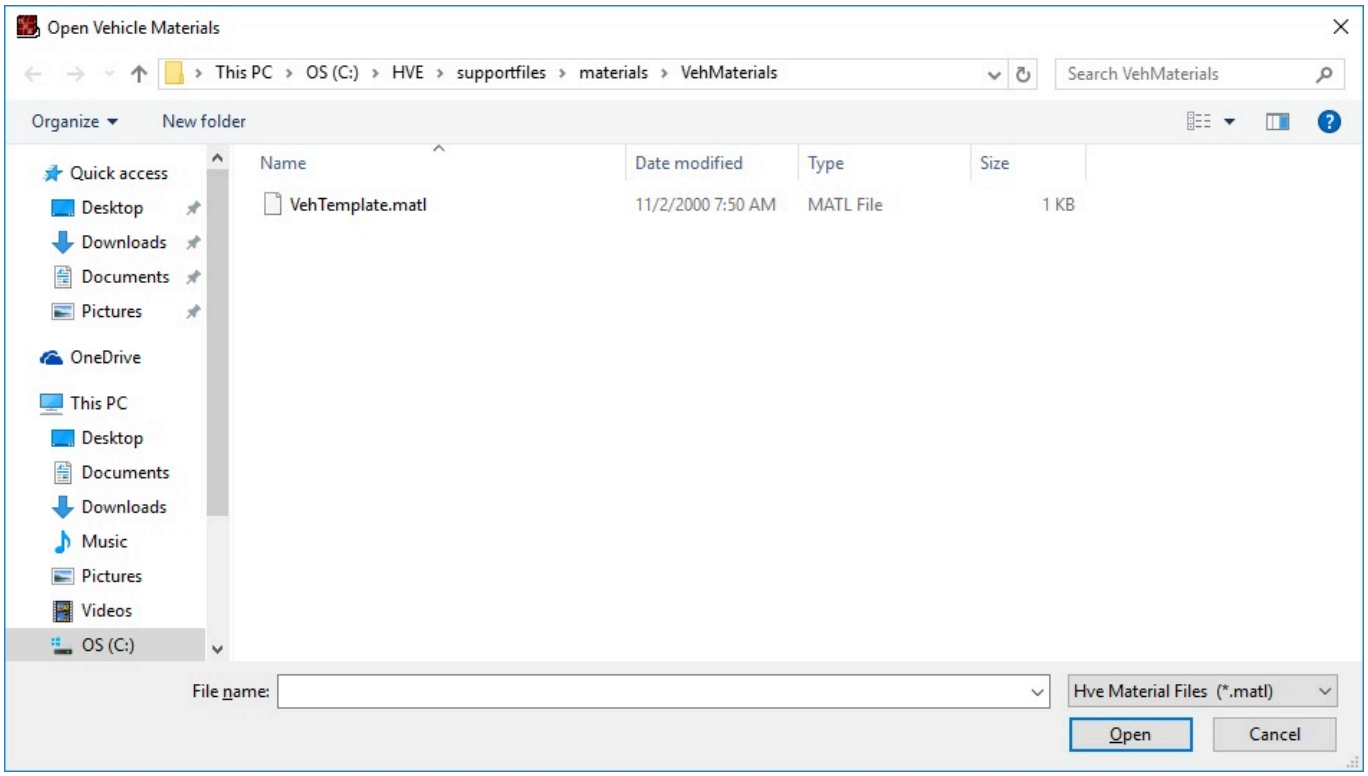


Figure 4-46: The Combined Materials File Selection dialog allows the user to open and save combined material files.

Opening and Saving Material Files

Combined material properties may also be assigned from materials files. To assign combined material properties from a file, perform the following steps:

1. Press Open. The Combined Material File Selection dialog is displayed (see Figure 4-46).
2. Select the desired filename and choose Open. The selected material is displayed in the Combined Material dialog. The force-deflection characteristics are also graphed.
3. If desired, edit the materials properties as described above.

4. If desired, choose Save As. The Combined Material File Selection dialog is again displayed. Enter a filename, followed by Save, to save the new material for use on other events or cases.
5. Press OK to assign the combined material attributes to the selected Source/Target segment pair.
6. Repeat the above steps for each Source/Target segment pair.
7. Press OK to assign the contact interactions for the current event.

Parameters: The parameters assigned using the Contacts dialog are shown in Table 4-15.

Table 4-15: Contact Interactions Dialog Parameters

Parameter	Units (US / SI)	Description
Source Segment	n/a	Name of the source segment (may be either a human ellipsoid or a vehicle contact surface)
Target Segment	n/a	Name of the target segment (may be either a human ellipsoid or a vehicle contact surface)
Material Name	—	User-editable material name
Constant	lb / N	Force required to initiate deflection
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient
Quadratic Stiffness	lb/in^2 / N/cm^2	Quadratic material deformation coefficient
Cubic Stiffness	lb/in^3 / N/cm^3	Cubic material deformation coefficient
Damping Constant	lb-sec/in / N-sec/m	Material velocity-dependent deformation constant
Friction Constant	—	Inter-segment friction coefficient
Maximum Force	lb / N	Force at which 3rd-order force-deflection relationship is abandoned
Maximum Deflection	in / cm	Deflection at which 3rd-order force-deflection relationship is abandoned
Unloading Slope	lb/in / N/cm	Linear unloading slope beginning at max deflection
Max Defl/Timestep	in / cm	Logical distance defining the amount of deformation allowed during one timestep (used to define front/back)
Edge Constant	—	Force reduction coefficient for edges

See Also: Human Information dialog, Human Editor, Human Ellipsoids dialog, Vehicle Editor, Vehicle Contact Surfaces dialog, User's Manual for simulation model

---

RESTRAINTS

Menu Option: RESTRAINTS (cascade: Airbags Ctrl+G, Belts Ctrl+B)

Purpose: Assign restraint system usage parameters to the selected human

Description: The Restraints dialog allows the user to assign a restraint system to the selected human/vehicle pair. Two options are available:

- **Airbag Restraint** — Simulate an airbag restraint system

- **Belt Restraint** — Simulate torso belt and lap belt restraint systems

> **NOTE:** The belt restraint system is defined as part of the vehicle using the Vehicle Editor. However, as we all know, just because a vehicle has a restraint system does not necessarily mean it is being used!
>
> **NOTE:** If a particular restraint system is not installed in the vehicle at the selected human's seat location, the option will not be selectable.
>
> **NOTE:** The selected human's seat position (e.g., R/F, L/F, etc.) is defined in its Human Information dialog (see Human Editor).

The Restraint set-up options are described in the following sections.

Airbag Restraints Option

The Airbag set-up option allows the user to assign an airbag restraint system and its in-use parameters for the selected occupant. To assign the airbag in-use parameters, perform the following steps:

1. Select the desired human from the Active Humans and Vehicles list.
2. Click on the Set-up menu and choose Restraints, then choose Airbags from the cascade menu. The Airbag Restraint System dialog for the selected vehicle is displayed (see Figure 4-42).
3. Click on the Airbag Restraints For option list and choose the desired human occupant.
4. Click on the Device In Use check box. The airbag restraints parameters become enabled for the selected occupant.
5. Using the Allowed Contacts list box, select the contact ellipsoids with which airbag contact is allowed.

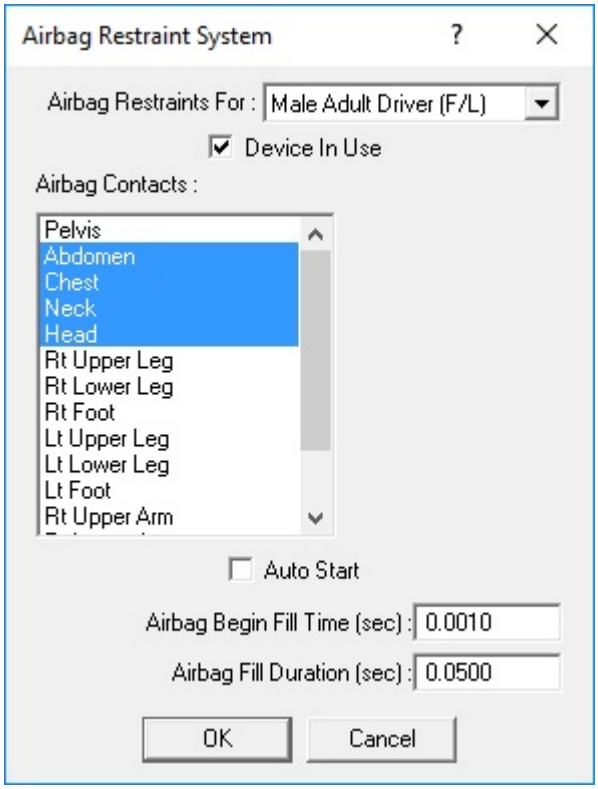


Figure 4-42: Vehicle Airbag Restraint Systems dialog.

1. Click on the AutoStart checkbox if you wish to have the airbag automatically deployed. Deployment is normally triggered when the vehicle's acceleration exceeds a threshold.

> **NOTE:** Support of the AutoStart feature is calculation model-dependent. Refer to the User's Manual for your simulation program to determine if AutoStart is supported. 7. Assign the Airbag Begin Fill Time for the selected occupant.
>
> **NOTE:** Airbag Begin Fill Time is disabled if AutoStart is selected. 8. Assign the Fill Duration for the selected occupant. 9. If desired, choose additional human occupants from the Airbag Restraints For option list and repeat the above steps to assign the airbag parameters for each occupant. 10. Press OK to accept the airbag restraint system options for the selected vehicle and its occupant(s).

When the simulation is executed, the airbag parameters will be incorporated into the simulation.

Belt Restraints Option

The Belt Restraints set-up option allows the user to assign torso and lap belt restraints and in-use parameters for the selected occupant. To assign the belt restraint parameters, perform the following steps:

1. Select the desired vehicle from the Event Humans and Vehicles list.
2. Choose Restraints from the Set-up menu, and choose Belts from the Restraints cascade menu. The Belt Restraints System dialog for the selected vehicle will be displayed (see Figure 4-43).
3. Click on the Belt Restraints For option list and choose the desired human occupant.
4. Click on the Belt Segment option list and choose the desired belt restraint system (Torso Belt or Lap Belt).
5. Click the Device In Use check box. The belt restraints parameters become enabled for the selected occupant.
6. Click on the Attached To option list and select a segment to attach the end of the belt section (the other end is attached to the anchor point on the vehicle; see Vehicle Editor).

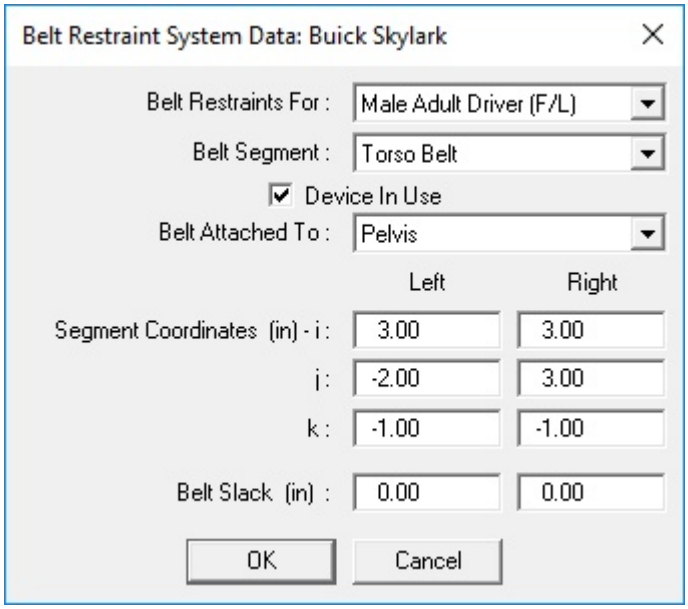


Figure 4-43: Vehicle Belt Restraint System dialog.

1. Enter the segment attachment coordinates for the left and right belt endpoints according to the human segment's coordinate system.

> **NOTE:** These are segment-fixed coordinates, and normally lie on or near the surface of a contact ellipsoid.
>
> **NOTE:** The left-side and right-side attachment points should be approximately symmetrical about the segment i-k plane; otherwise, belt tension may cause the human to rotate in an unexpected manner. 8. Enter the desired belt slack for the right and left sections of the selected belt. 9. Repeat the above steps for the other belt system, if desired. 10. Press OK to accept the belt restraint system parameters for the selected vehicle and its occupants.

When the simulation is executed, the belt restraint system parameters will be incorporated into the simulation.

Parameters: The parameters assigned using this dialog are shown in Table 4-14.

Table 4-14: Airbag and Belt Restraint System Parameters

Parameter	Units (US / SI)	Description
x, y and z Attachment Coordinates	in / cm	Segment-based x,y,z coordinates for the attachment point of the ends of the right and left belt segments
Belt Slack	in / cm	Belt slack. The amount of displacement between the ends of the belt allowed before creating belt tension (a negative value implies pre-load)
Airbag Begin Fill Time	sec	Simulation time at which the airbag is deployed and begins to fill
Airbag Fill Duration	sec	Length of time during which airbag filling occurs

See Also: Human Editor, Human Information dialog, Vehicle Editor, Vehicle Model (Restraint Systems), Occupant Simulation, Human Output Tracks, Vehicle Output Tracks, Vehicle Data Report, User's Manual for selected simulation model

---

SIGNALS

(updated: this option was added after the 2006 manual)

Menu Option: SIGNALS

Purpose: Set up traffic signals for the current event

Description: Choosing Signals from the Set-up menu displays the Event Signals Set-up dialog. Traffic signal lights defined in the environment can be assigned timing and state sequences (e.g., red/amber/green phases) for the current event, so that signal displays are synchronized with the simulation timeline and appear correctly in Event and Playback modes.

NOTES

(updated: this option was added after the 2006 manual)

Menu Option: NOTES

Purpose: Attach text notes to the current event

Description: Choosing Notes from the Set-up menu displays the Event Notes dialog, which allows the user to enter one or more free-form text notes for the current event. The notes are stored with the event in the case file, providing a convenient way to document assumptions, data sources and set-up decisions.

Chapter 5: View Menu

HVE User's Manual — Section Two: Menu Reference. Updated edition, revised for the current HVE release.

The View Menu includes the following options:

- **Toolbar** — Displays the toolbar containing shortcuts for performing common tasks
- **Status Bar** — Displays the status bar of current user actions and selections at the bottom of the HVE window
- **Set Camera** — Define the viewing region and the camera reference point
- **Overlays** — Display or remove from the view user-named sets of objects

These options allow you to set up the HVE user interface and to determine how the scene is viewed. Each of the View Menu options is explained in this chapter.

TOOLBAR

Menu Option: TOOLBAR

Purpose: Displays the toolbar of shortcuts for performing common tasks

Description: Choosing Toolbar from the View Menu displays the HVE Toolbar. The Toolbar (see Figure 5-1) provides a set of button choices for performing common tasks, such as changing modes, adding objects, opening and saving case files and executing events. All of these options are also available using HVE's pull-down menus. The Toolbar is simply quicker.

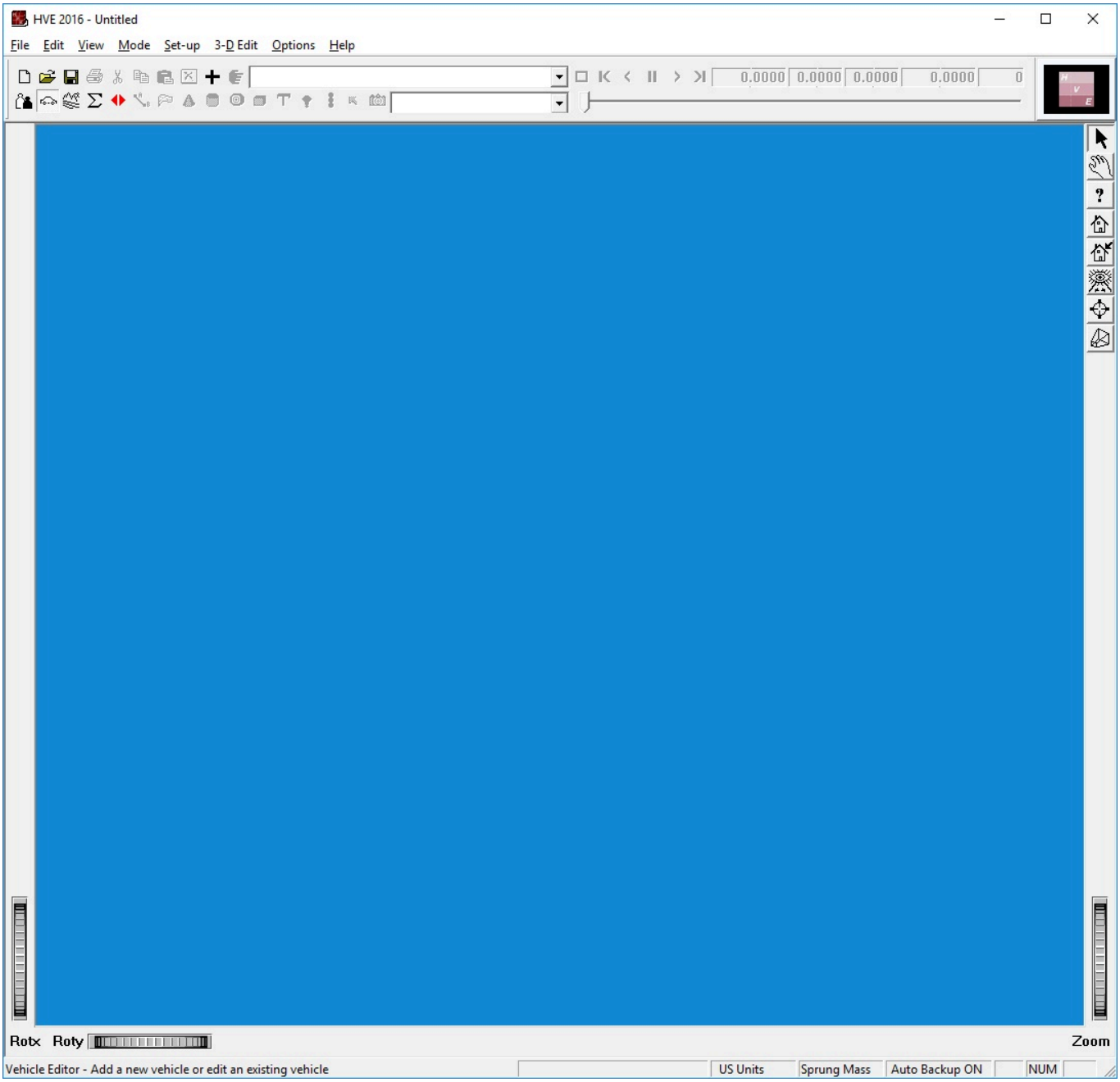


Figure 5-1: The HVE Toolbar (near the top of the HVE window) provides a set of button choices for performing common tasks. The HVE Status Bar (near the bottom of the HVE window) displays a brief description of user actions and selections.

STATUS BAR

Menu Option: STATUS BAR

Purpose: Displays the HVE Status Bar at the bottom of the HVE window

Description: Choosing Status Bar from the View Menu displays the HVE Status Bar. The Status Bar (see Figure 5-1) displays a brief description of the user's actions or selections. The Status Bar may be thought of as a first-level help system.

See Also: Help

SET CAMERA

Menu Option: SET CAMERA

Purpose: Determine how the scene is viewed

Description: Choosing Set Camera from the View Menu displays the Set Camera dialog. The Set Camera dialog allows the user to define what is visible in the viewers. It is called Set Camera because a camera metaphor is used to perform this task: determining how a scene is viewed is much like looking through a camera. See also the dialog reference: Camera Set-up Dialog.

The Set Camera dialog is shown in Figure 5-2.

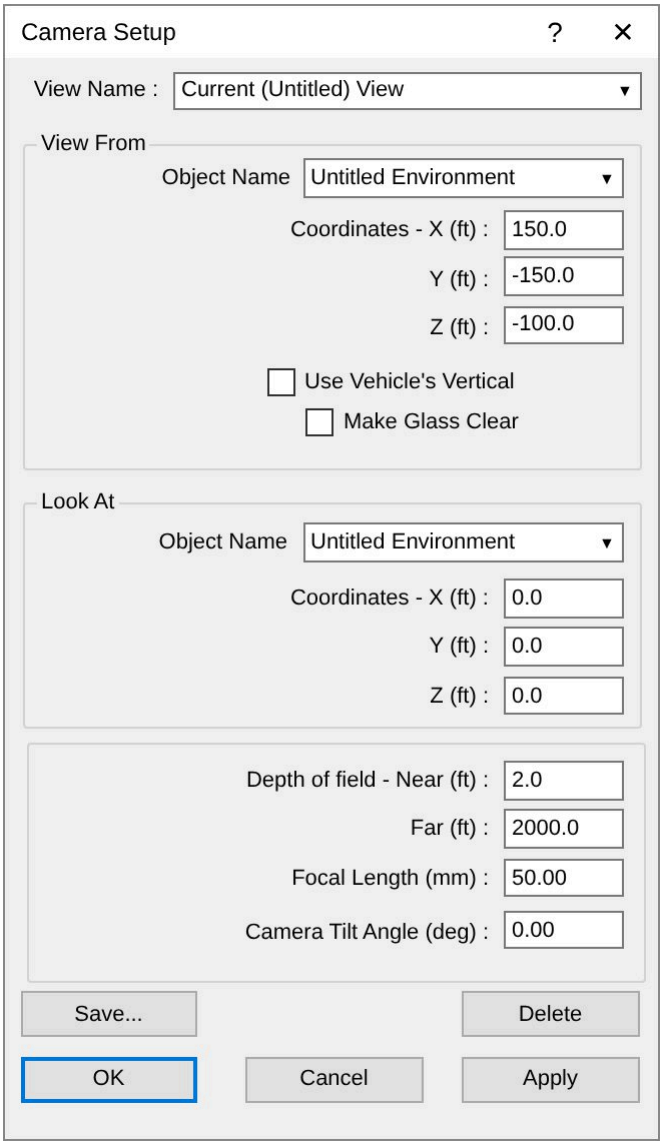


Figure 5-2: The Set Camera dialog is used to determine how the scene is viewed.

Setting the camera allows the user to perform the following functions:

- Select, create and store user-named views
- Attach the camera to a fixed or moving object
- Determine what the camera is looking at
- Set the depth of field
- Set the focal length of the lens
- Set the camera tilt (bank) angle
- Use the View From vehicle's vertical for the camera's up direction
- Make the View From vehicle's glass clear

> **NOTE:** Many of these functions may be performed interactively without using the Set Camera dialog. Instead, use the Viewer's pan, dolly, zoom and rotate thumb wheel controls.

To set up the camera, perform the following steps:

1. Choose Set Camera from the View menu. The Set Camera dialog will be displayed.
2. Click on the View From option list to select an object to attach the camera to.

> **NOTE:** The option list contains the names of the environment and each vehicle in the current window (Event, Preview or Playback). 3. Enter the desired X,Y,Z (or x,y,z) coordinates for the View From object. 4. Click on the Look At option list to select an object to attach the camera to.
>
> **NOTE:** The option list also contains the names of the environment and each vehicle in the current window (Event, Preview or Playback). 5. Enter the desired X,Y,Z (or x,y,z) coordinates for the Look At object.
>
> **NOTE:** The View From and Look At coordinates are relative to the object the camera is attached to. For example, if View From and Look At are both Mason Convertible (a vehicle), the View From and Look At coordinates will both be relative to the Mason Convertible's CG, and the camera will move with the vehicle! 6. Enter the Near and Far Depth of Field (Clipping Planes). 7. Enter the Focal Length of the lens. 8. Press OK when the desired camera set-up data are entered.

Each of these steps is described more fully in the following paragraphs.

View Name

The View Name option list displays and allows the user to select from a list of previously created views. The default View Name is Default Camera. Selecting a different view causes the Set Camera parameters (described below) to be updated to those assigned for the selected view. See also Save/Delete View below.

View From: Object Name

The Object Name option list in the View From group allows the user to select the object from which the scene is viewed. This option list contains all the objects (humans, vehicles and environment) in the event (see Figure 5-1). Attaching the camera to a human or vehicle causes the camera to move with the human or vehicle.

View From Coordinates

Next, the camera Coordinates are entered. These coordinates are relative to the selected View From object. For example, if the View From object is a vehicle, setting the View From coordinates at 0,0,0 means the scene will be viewed from the vehicle's CG, regardless of the vehicle's location in the environment.

Use Vehicle's Vertical and Make Glass Clear

When the camera is attached to a vehicle, *Use Vehicle's Vertical* makes the camera's "up" direction follow the vehicle, so the view rolls and pitches with it; when unchecked, the camera stays level with the earth. *Make Glass Clear* renders the vehicle's glass transparent, which is useful when viewing from inside the vehicle (for example, a driver's-eye view).

Look At: Object Name

The Object Name option list in the Look At group allows the user to select the object being viewed. This option list contains all the objects (humans, vehicles and environment) in the event (see Figure 5-1). Attaching the camera to a human or vehicle causes the view to remain fixed on that human or vehicle.

> **NOTE:** This is a very useful feature! For example, if the View From object is Vehicle A and the Look At is Vehicle B, the camera will automatically pan as required to keep viewing Vehicle B from Vehicle A.
>
> **NOTE:** If the View From and the Look At objects are the same object, the view will not pan. This is essentially the same behavior as existed in earlier versions of HVE.

Look At Coordinates

The Look At Coordinates define the coordinates on the Look At object to which the view is attached. The coordinates are also relative to the View From object. For example, if the View From object is a vehicle, setting the Coordinates to 100,0,0 causes the view to look at a point 100 feet ahead of the vehicle, regardless of the vehicle's position or orientation.

> **NOTE:** A line drawn from the View From X,Y,Z coordinates to the Look At X,Y,Z coordinates is called the View Axis.

Depth Of Field

The depth of field is defined by near and far clipping planes. Any object closer to the camera than the near clipping plane, or farther from the camera than the far clipping plane, will not be rendered in the scene.

Focal Length

The Focal Length is the traditional focal length of a virtual 35 mm camera used to visualize the scene. A Focal Length of 50 mm is the default, and represents a normal view. Focal lengths less than 50 mm are wide-angle lenses; greater than 50 mm are telephoto lenses.

> **NOTE:** You can also set the focal length using the Zoom slider in the viewer. The number reported by the Zoom slider is the included angle of the viewer.

Camera Tilt Angle

The Camera Tilt Angle rotates the camera about its viewing direction (the View Axis). A value of 0 keeps the camera level.

Save...

(updated: this button is labeled Save... in the current build.)

Pressing Save... saves the current camera settings as a named view. The named view is added to the View Name list so it can be recalled later. This lets the user quickly change between previously created views. Up to ten views may be stored.



Figure 5-3: The New Camera View dialog is used to assign and store up to ten views for later retrieval.

> **NOTE:** A separate Rename View control existed in earlier versions but is hidden and non-functional in the current build.

Delete

Pressing Delete removes the current view from the list of views. The previous view in the View Name list box becomes the current view.

Parameters: The parameters assigned using this dialog are shown in Table 5-1.

Table 5-1: Parameters assigned using the Set Camera dialog

Parameter	Units (US / SI)	Description
View From Object Coordinates	in / cm or ft / m *	Components of the X,Y,Z (or x,y,z) distances from the selected object origin to the camera
Look At Object Coordinates	in / cm or ft / m *	Components of the X,Y,Z (or x,y,z) distances from the selected object origin to the center of the viewer
Near, Far Depth of Field (Clipping Planes)	in / cm or ft / m *	Distance from the camera to the closest (Near) and farthest (Far) objects which can be viewed
Focal Length	mm	Focal length of the virtual camera lens

* in / cm when the camera is attached to a human or vehicle; ft / m when attached to the environment.

See Also: Viewers, Using HVE

OVERLAYS

Menu Option: OVERLAYS

Purpose: Select or remove overlays from the selected viewer

Description: Choosing Overlays from the View Menu displays the Overlays dialog. The Overlays dialog (see Figure 5-4) allows the user to display or remove objects from the scene. The Overlays dialog displays a multiple-selection list box displaying all the overlay names of the objects in the viewer. By default, all objects are selected, and therefore, are displayed in the viewer. Click on an overlay name to remove all objects using that name. See also the dialog reference: Overlays Dialog.

> **NOTE:** A good example of the use of overlays is accident site debris. By using HVE's 3-D Editor to name all objects 'Debris' when they are created, the debris may be displayed or removed from the scene to show how an accident site appeared before and after the accident.

The Scanned Image check box is used to display or remove a scanned image from the environment. A scanned image may be used for the entire environment, or just the sky.

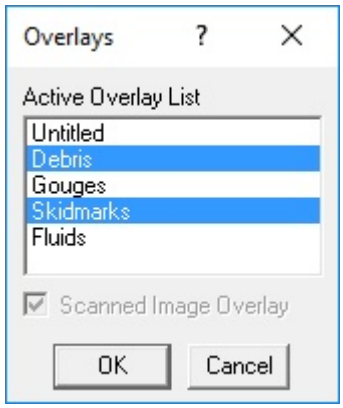


Figure 5-4: Set Overlays Dialog, used for turning on and off various layers of geometry.

> **NOTE:** If a scanned image is used for the entire environment, the user must enter the X,Y,Z coordinates from which the picture was taken, the X,Y,Z coordinates for the center of the picture, and the focal length of the camera lens used to take the picture. This information is required for registration between the scanned image and the moving humans and vehicles.
>
> **NOTE:** You should confirm the scale of the bitmap is correct by noting the time required for a vehicle to travel a known distance between two objects in the photograph.

To select or deselect overlays, perform the following steps:

1. Choose Overlays from the View menu. The Overlays dialog will be displayed, showing a list box with all the overlay names in the current scene. The currently selected overlays will be highlighted.
2. Select additional overlays you wish to be displayed, and deselect those overlays you wish to remove.
3. Click on the Scanned Image Overlay check box to display or remove any scanned environment image.
4. Press OK when the desired overlays and scanned image have been selected.

See Also: 3-D Editor, Current Object Attributes, Scanned Images (Bitmaps), Environment Information dialog, Set Camera dialog

Chapter 6: Options Menu

HVE User's Manual — Section Two: Menu Reference. Updated edition, revised for the current HVE release. For a compact summary of every current Options menu item, see Options Menu.

The Options Menu includes the following options

(updated: list and order verified against the current menu; items marked "new" were added after the 2006 manual)

:

- **Create DB** *(new)* — Rebuilds the HVE object database from the database source files.
- **Show Key Results** — Displays a window during Event and Playback mode containing user-selectable results.
- **Show Axes** — Displays a coordinate axis system for each human and vehicle, as well as the environment.
- **Show Contacts** — Displays the physical contact surfaces on each human and vehicle.
- **Show Belt Anchors** — Displays an icon at the vehicle-fixed anchor point for lap and shoulder belt restraints on each vehicle.
- **Show Velocity Vectors** — Displays a velocity vector on each human and vehicle according to its current velocity and direction.
- **Show Skidmarks** — Displays simulated skidmarks for each vehicle.
- **Show Tracks** *(new)* — Displays the path traced by each tire.
- **Show CGs** *(new)* — Displays a symbol at each vehicle's center of gravity.
- **Show CG Paths** *(new)* — Displays the path traced by each vehicle's CG.
- **Show Accelerometers** *(new)* — Displays a symbol at each vehicle's accelerometer locations.
- **Show Accelerometer Paths** *(new)* — Displays the path traced by each vehicle's accelerometers.
- **Show Position Sequences** *(new)* — Displays position sequence overlays.
- **Show Targets** — Displays positions not used by the current calculation model. These positions are displayed as translucent to distinguish them from positions used by the current calculation model.
- **Show Connections** *(new)* — Displays connections between connected vehicles (e.g., tractor-trailer).
- **Distance Tool** *(new)* — Displays a modeless dialog for measuring distances in the viewer.
- **Show Path** *(new)* — Displays a smooth curved path based on the event's path positions.
- **Autoposition (On/Off)** — During event positioning by the user, causes the selected vehicle to be positioned on top of the environment's physical surface.
- **Grid** — Displays a set of construction lines at user-specified intervals.
- **Units** — Allows the user to select the current system of units.
- **Shadows** *(new)* — Displays the Shadow Options dialog controlling rendered shadows.
- **Render** (Ctrl+R) — Allows the user to choose how objects are drawn on the screen.
- **Playback** — Allows the user to choose the output time interval used for trajectory simulations and reports in Playback Mode.
- **Simulation Controls** (Ctrl+Y) — Allows the user to set various simulation control parameters for the current event.
- **Calculation Options** (Ctrl+J) — Allows the user to assign optional parameters required by the current calculation model.
- **DyMESH** — Allows the user to choose various modeling options available to the DyMESH collision model.

- **Get Surface Info** — Sets options that determine which terrain surface polygons vehicle tires may drive on.
- **Preferences** (Ctrl+F) — Allows the user to choose various user-selectable preferences.

> *(updated: the "Add Playback Window" option described at the end of the 2006 chapter is no longer located in the Options menu. Playback windows are now added from Playback mode; the original description is retained at the end of this chapter for reference.)*

The checked/unchecked options are toggles allowing the user to choose which information HVE displays. Other selections in the Options Menu allow users to customize their HVE user environment. Each of these options is explained on the following pages.

CREATE DB

(updated: this option was added after the 2006 manual)

Menu Option: CREATE DB

Purpose: Rebuild the HVE object database

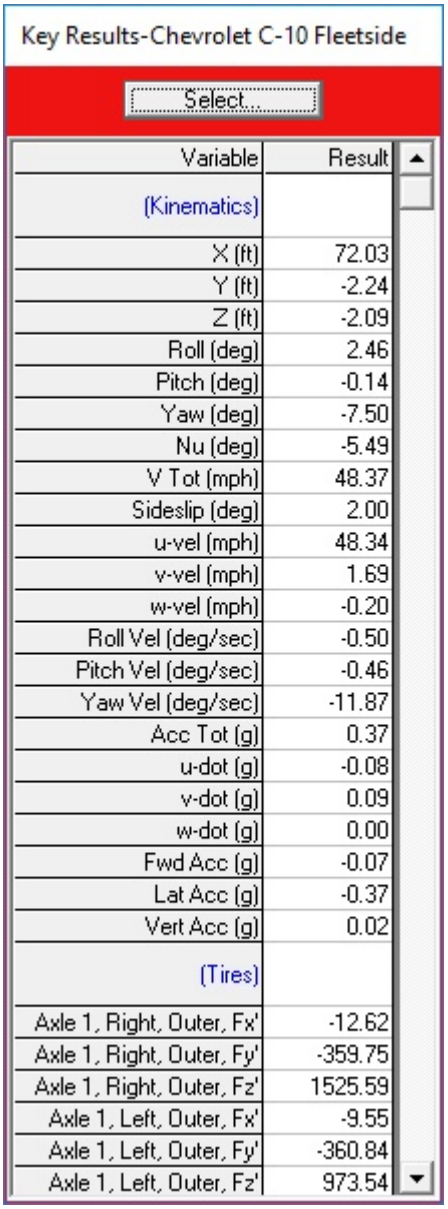
Description: Choosing Create DB rebuilds the HVE object database (ESNDB.db) from the database source files. All vehicle and human binary files are regenerated and the rebuilt database is reloaded. This may take a few moments; the cursor changes to an hourglass while the database is being created.

SHOW KEY RESULTS

Menu Option: SHOW KEY RESULTS

Purpose: Display or remove Key Results windows for all displayed events

Description: Choosing Show Key Results from the Options Menu displays a Key Results window for each object. Key Results windows display user-specified results during Event and Playback modes. Key Results are especially useful during Event mode to monitor the values of one or more variables at each timestep. Use the Variable Selection dialog to select which variables are displayed in the Key Results window. See also the dialog reference: Key Results Dialog Box.



Variable	Result
(Kinematics)	
X (ft)	72.03
Y (ft)	-2.24
Z (ft)	-2.09
Roll (deg)	2.46
Pitch (deg)	-0.14
Yaw (deg)	-7.50
Nu (deg)	-5.49
V Tot (mph)	48.37
Sideslip (deg)	2.00
u-vel (mph)	48.34
v-vel (mph)	1.69
w-vel (mph)	-0.20
Roll Vel (deg/sec)	-0.50
Pitch Vel (deg/sec)	-0.46
Yaw Vel (deg/sec)	-11.87
Acc Tot (g)	0.37
u-dot (g)	-0.08
v-dot (g)	0.09
w-dot (g)	0.00
Fwd Acc (g)	-0.07
Lat Acc (g)	-0.37
Vert Acc (g)	0.02
(Tires)	
Axle 1, Right, Outer, Fx'	-12.62
Axle 1, Right, Outer, Fy'	-359.75
Axle 1, Right, Outer, Fz'	1525.59
Axle 1, Left, Outer, Fx'	-9.55
Axle 1, Left, Outer, Fy'	-360.84
Axle 1, Left, Outer, Fz'	973.54

Figure 6-1: The Key Results window displays the current simulation results for the selected human or vehicle.

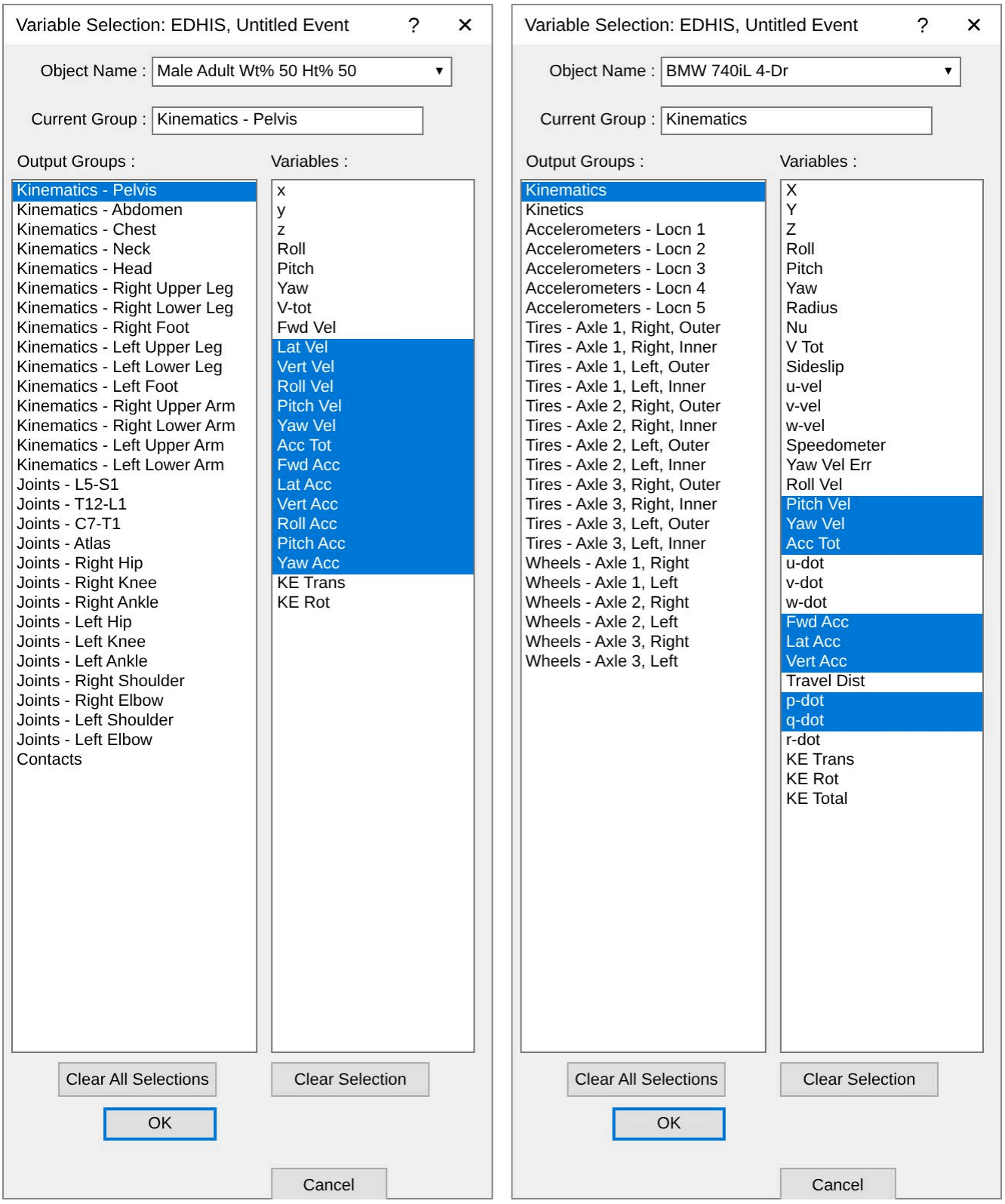


Figure 6-2: The Variable Selection dialogs for humans (left) and vehicles (right) allow the user to select the variables displayed in the Key Results dialog as well as in the Variable Output window (see Playback Mode).

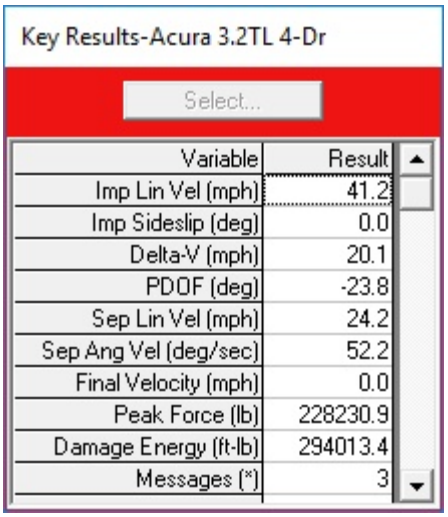


Figure 6-3: Reconstruction programs also have Key Results windows.

See Also: Variable Selection dialog, Event Execution, Variable Output

---

SHOW AXES

Menu Option: SHOW AXES

Purpose: Display or remove coordinate axis systems from all humans, vehicles and environments

Description: Choosing Show Axes from the Options Menu displays a set of vectors in the X,Y,Z directions for each human and vehicle, as well as the environment. The origin of the axis system is at the origin of the object. The axes are displayed in all modes.

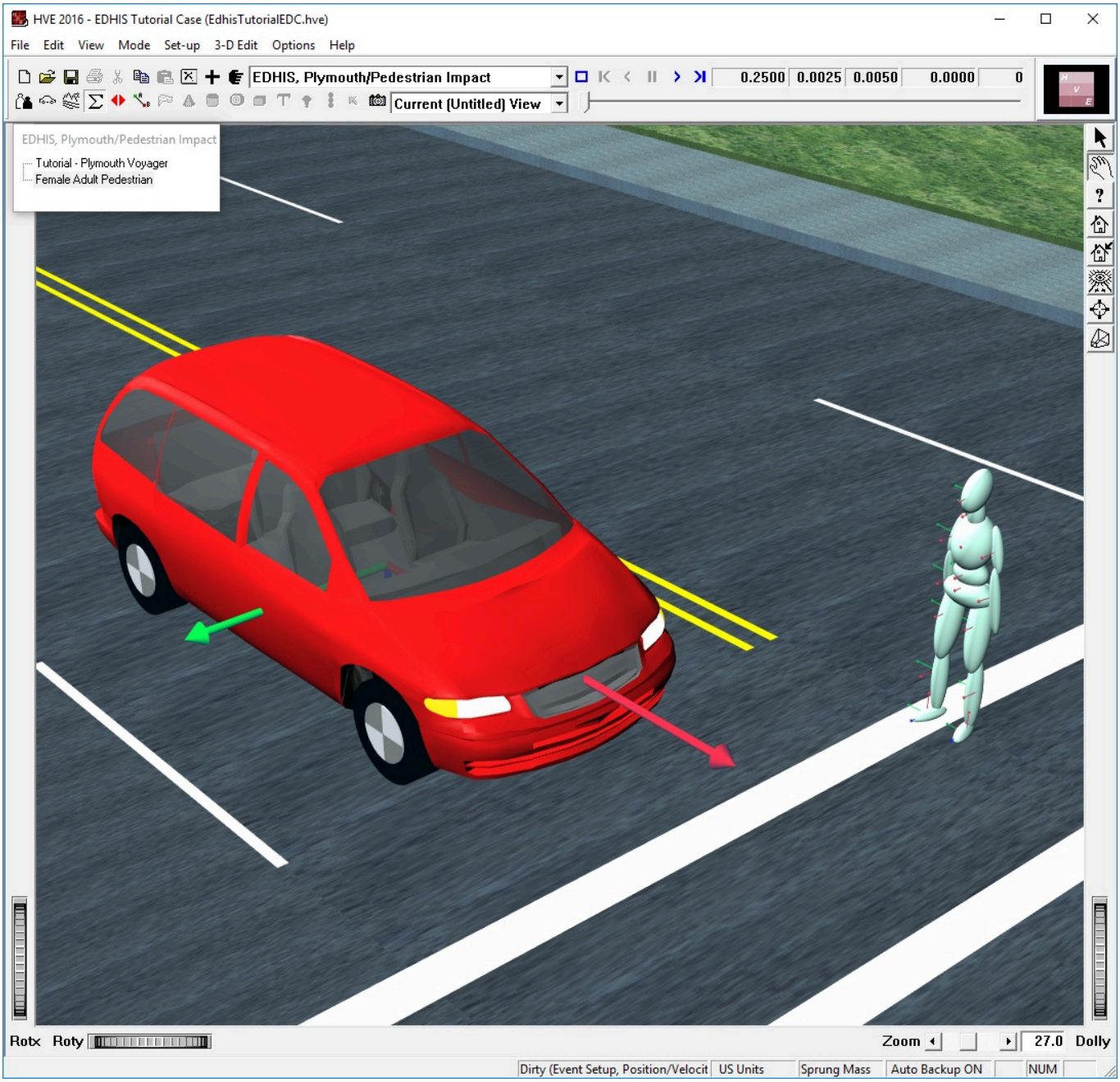


Figure 6-4: Show Axes Option.

See Also: Coordinate Systems

SHOW CONTACTS

Menu Option: SHOW CONTACTS

Purpose: Display or remove physical contact surfaces from all humans, vehicles and environments

Description: Choosing Show Contacts from the Options Menu displays the physical contact surfaces used for generating forces between human ellipsoids and vehicle contact surface planes. Visualizing these planes can be very useful while

creating the ellipsoids and surface planes, and also during Event mode when the interactions are being simulated. However, you may choose to hide contacts during Playback mode if you find them visually distracting.

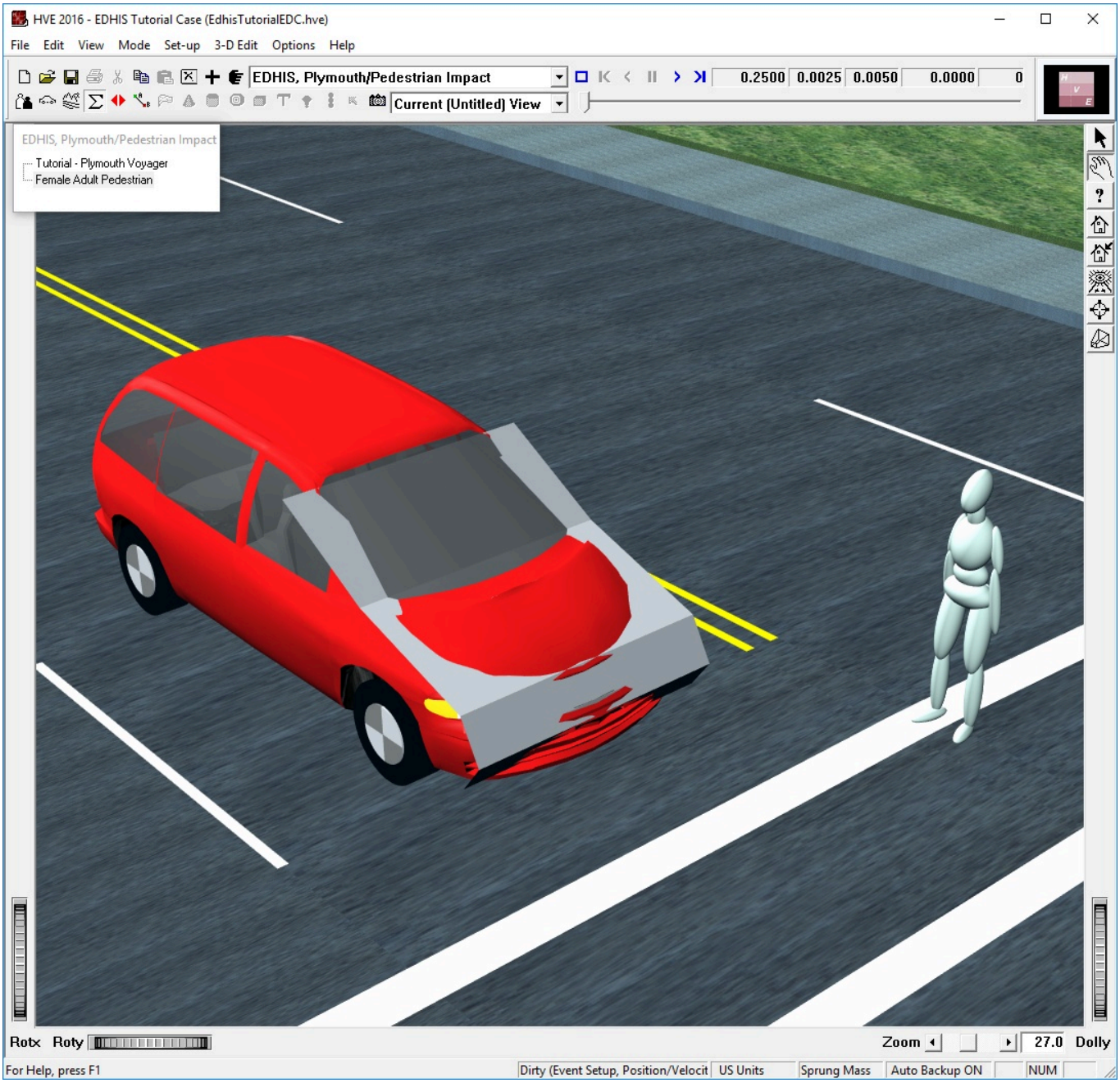


Figure 6-5: Show Contacts Option for Vehicles.

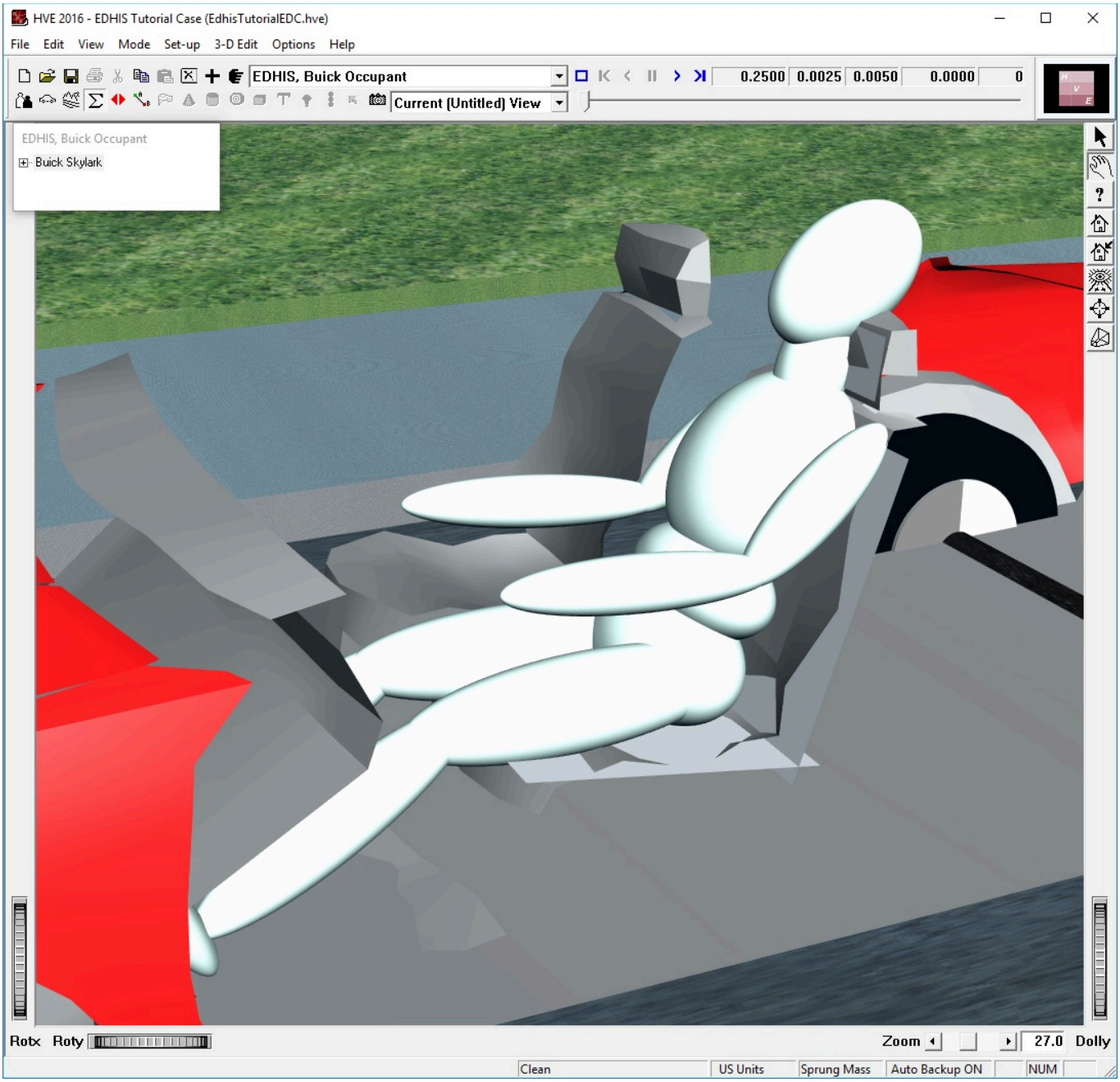


Figure 6-6: Show Contacts Option for Humans.

See Also: Human Editor, Human Contact Ellipsoids, Vehicle Editor, Vehicle Contact Surfaces, Event Set-up (Inhibiting Contacts)

SHOW BELT ANCHORS

Menu Option: SHOW BELT ANCHORS

Purpose: Display or remove the icons defining the vehicle-fixed x,y,z coordinates for the lap and torso belt restraints for each vehicle

Description: Choosing Show Belt Anchors from the Options Menu displays the vehicle-fixed belt anchor locations for each belt restraint system installed in the vehicle. The location of the icon can provide important visual cues during event set-up as well as during execution.

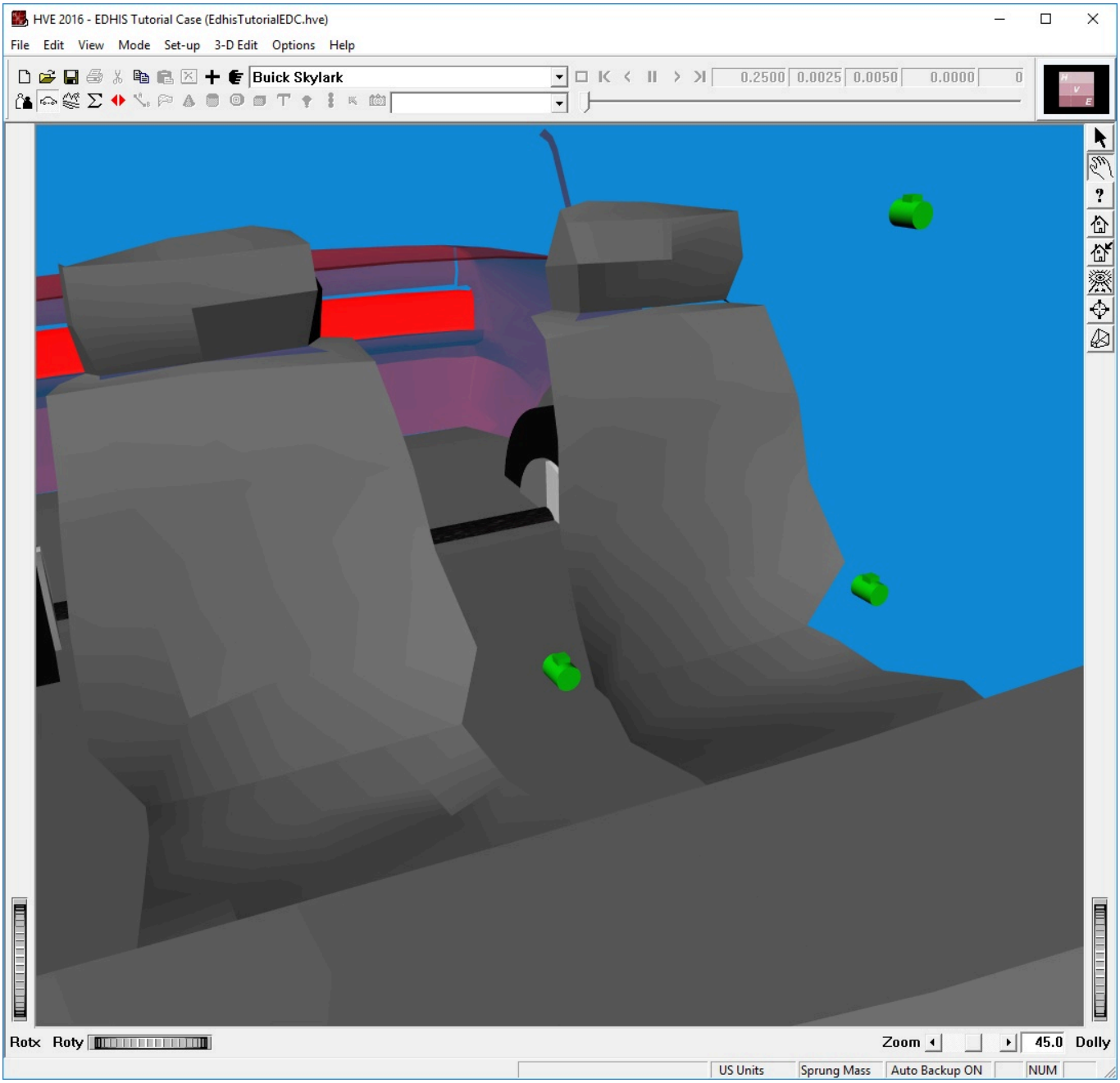


Figure 6-7: Show Belt Anchors Option for Vehicles.

See Also: Event Set-up, Variable Output

SHOW VELOCITY VECTORS

Menu Option: SHOW VELOCITY VECTORS

Purpose: Display or remove a vector (arrow) representing the magnitude and direction of the total velocity for all humans and vehicles

Description: Choosing Show Velocity Vectors from the Options Menu displays the current velocity vector during Event and Playback modes. Both relative magnitude and direction are shown. (The vector's length shortens as the vehicle slows.) This vector can provide important visual cues during event set-up as well as during execution. *(updated: see also the Long Velocity Vectors option in the User Preferences dialog, which lengthens the displayed vectors.)*

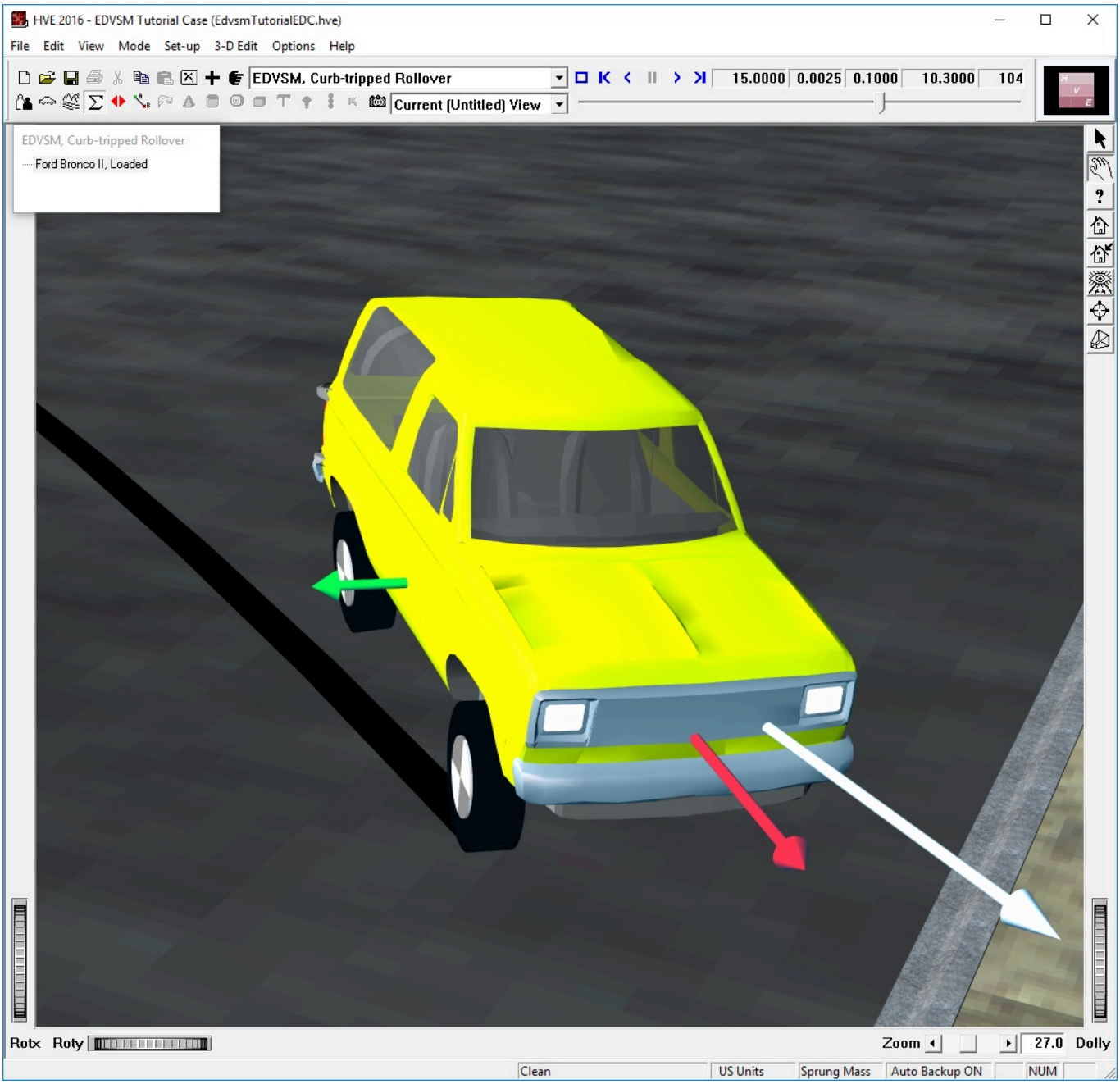


Figure 6-8: Show Velocity Vectors Option for Vehicles.

See Also: Event Set-up, Variable Output

SHOW SKIDMARKS

Menu Option: SHOW SKIDMARKS

Purpose: Display or remove simulated vehicle skid and scuff marks in the environment

Description: Choosing Show Skidmarks from the Options Menu displays tire skidmarks and scuffmarks calculated by the simulation model. Simulated tire marks are displayed for each tire, including duals, and are displayed according to the current value of Tire Skid in Variable Output (see Variable Selection, Tire Group).

Tire marks are drawn according to the current vertical tire load (higher loads result in darker marks), and longitudinal and lateral tire slip (higher slips result in darker marks). The width of a tire mark is determined by the tire width (according to its tire size designation).

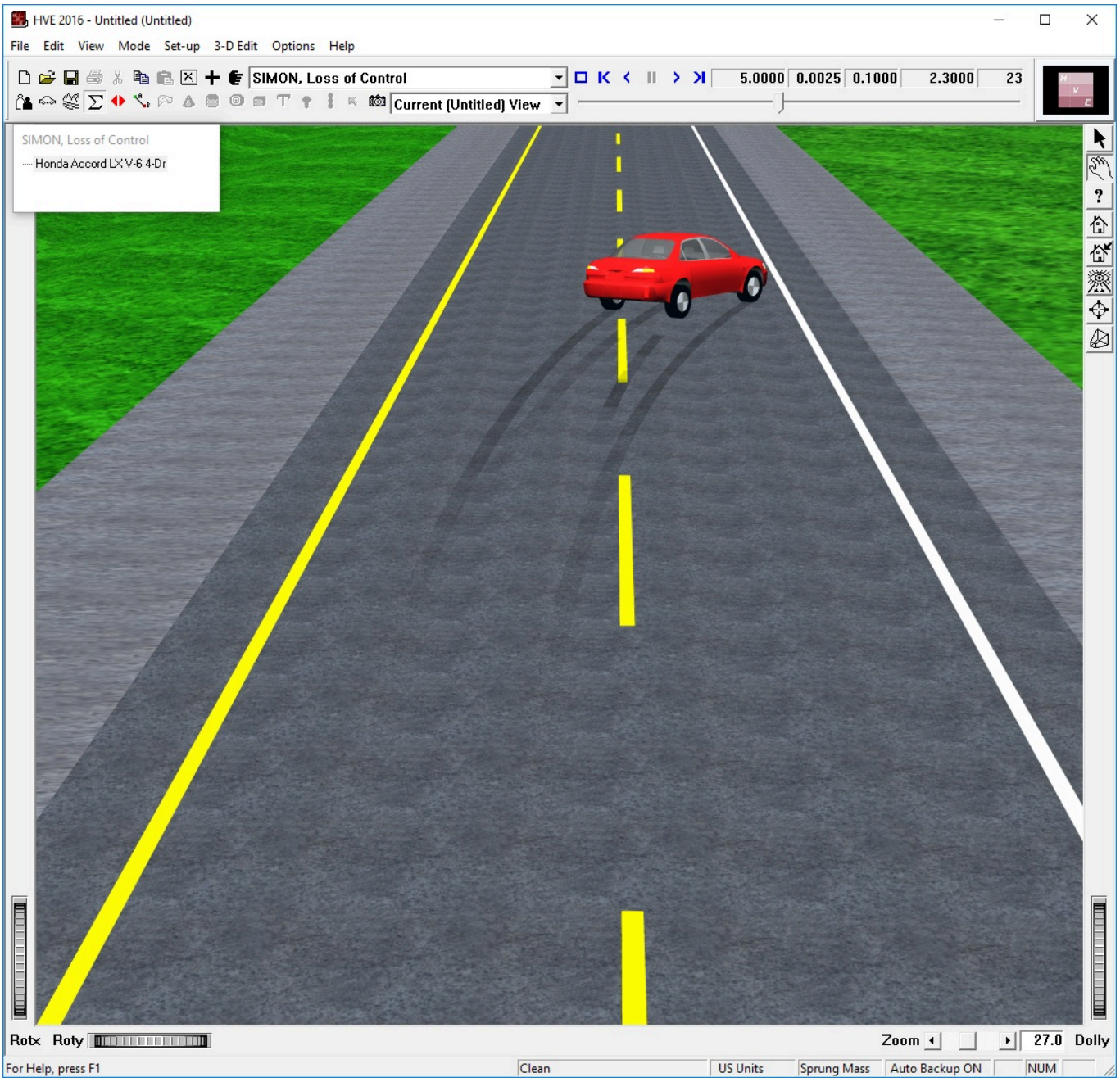


Figure 6-9: The Show Skidmarks Option displays simulated skidmarks from each tire generated by the simulation.

See Also: Simulation Models, Vehicle Editor (Tire Information), Skidmark Elevation (User Preference)

SHOW TRACKS

(updated: this option was added after the 2006 manual)

Menu Option: SHOW TRACKS

Purpose: Display or remove the wheel track overlay

Description: Choosing Show Tracks toggles the display of the path traced by each tire, whether or not the tire is skidding. Wheel tracks are useful for comparing the simulated wheel paths against evidence at the scene.

SHOW CGs / SHOW CG PATHS

(updated: these options were added after the 2006 manual)

Menu Options: SHOW CGs, SHOW CG PATHS

Purpose: Display or remove CG symbols and CG path traces

Description: Show CGs toggles the display of a symbol at each vehicle's center of gravity. Show CG Paths toggles the display of the path traced by each vehicle's center of gravity during the event.

SHOW ACCELEROMETERS / SHOW ACCELEROMETER PATHS

(updated: these options were added after the 2006 manual)

Menu Options: SHOW ACCELEROMETERS, SHOW ACCELEROMETER PATHS

Purpose: Display or remove accelerometer symbols and path traces

Description: Show Accelerometers toggles the display of a symbol at each vehicle's accelerometer locations (assigned using the Set-up menu Accelerometers option; see Chapter 4, Part B). Show Accelerometer Paths toggles the display of the path traced by each accelerometer during the event.

SHOW POSITION SEQUENCES

(updated: this option was added after the 2006 manual)

Menu Option: SHOW POSITION SEQUENCES

Purpose: Display or remove position sequence overlays

Description: Choosing Show Position Sequences toggles the display of position sequence overlays — a series of vehicle positions displayed at selected times during the event.

SHOW TARGETS

Menu Option: SHOW TARGETS

Purpose: Display or remove target humans and vehicles from the environment

Description: Choose Show Targets from the Options Menu to display target humans and vehicles. Targets are positions you can enter during event set-up that do not enter into simulation calculations (except for the HVE Path Follower), but allow you to visually compare how closely the simulated path matches the actual path represented by the targets. Simulation models may also perform error calculations based on the target positions and velocities to quantify the match between simulated and actual paths.

Target humans and vehicles are translucent to distinguish them from positions used by the calculation model.

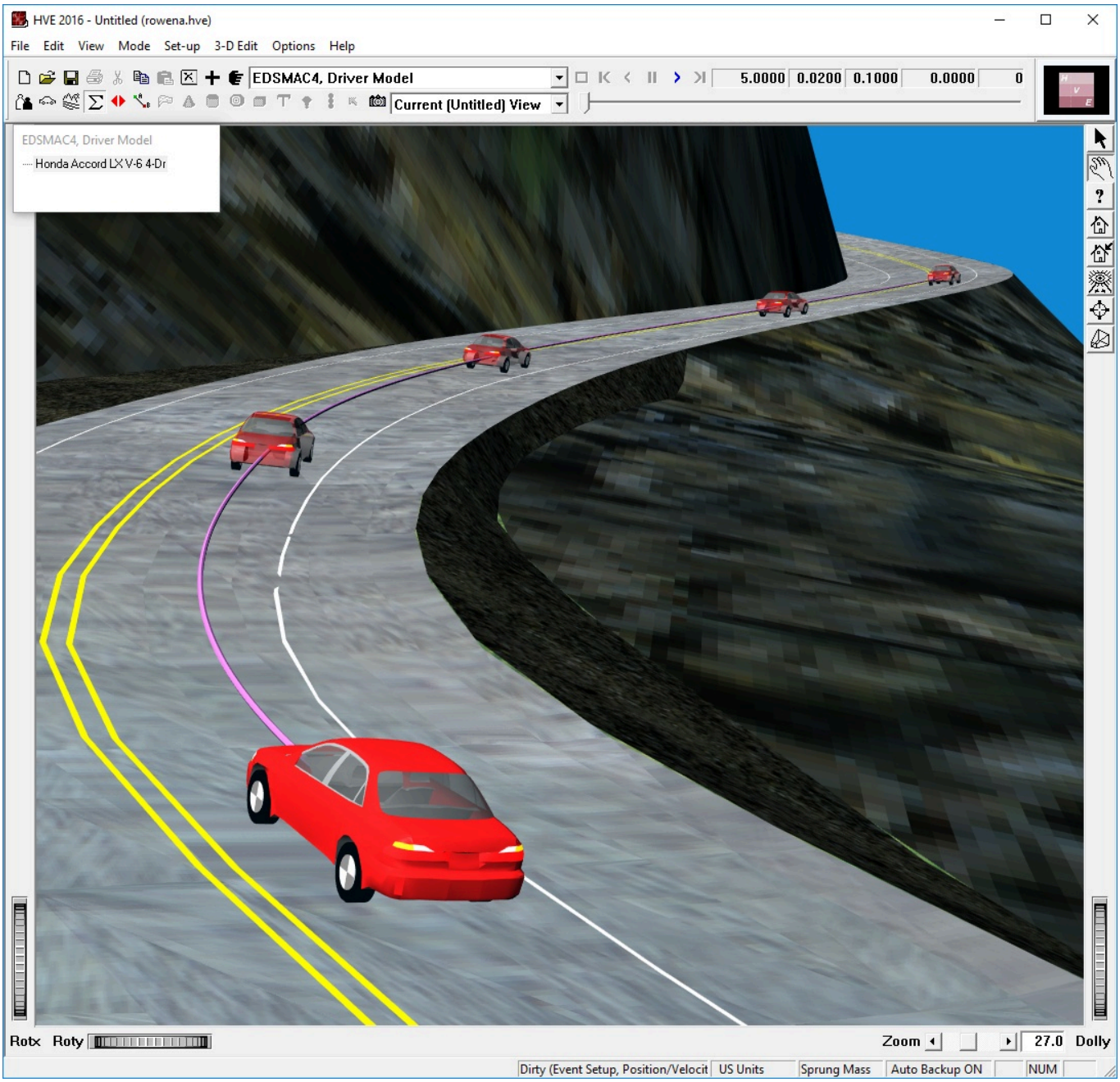


Figure 6-10: The Show Targets Option displays user-entered vehicle target positions (i.e., positions not required by the reconstruction or simulation model). Target positions are translucent, to distinguish them from other positions. Targets are a useful means of assessing how well the simulated path matches the actual path.

See Also: Event Editor, Event Set-up

SHOW CONNECTIONS

(updated: this option was added after the 2006 manual)

Menu Option: SHOW CONNECTIONS

Purpose: Display or remove vehicle connection markers

Description: Choosing Show Connections toggles the display of the connections (e.g., hitches and articulation points) between connected vehicles, such as a tractor and its trailer(s). See also: Inter-Vehicle Connections Dialog.

DISTANCE TOOL

(updated: this option was added after the 2006 manual)

Menu Option: DISTANCE TOOL

Purpose: Measure distances in the viewer

Description: Choosing Distance Tool displays the modeless Distance Tool dialog, used for measuring distances in the viewer during Event mode. This option is available only when the case contains at least one event.

SHOW PATH

(updated: this option was added after the 2006 manual)

Menu Option: SHOW PATH

Purpose: Display a smooth curved path based on the event's path positions

Description: Choosing Show Path displays the Show Curved Path dialog for the current event. Use it to display a smooth (spline) curved path in the viewer based on the event's path positions; the path display is updated when the dialog is closed with OK.

AUTOPOSITION (On/Off)

Menu Option: AUTOPOSITION (On/Off)

Purpose: Allows automatic 3-D positioning of vehicles

Description: Choosing AutoPosition from the Options Menu causes vehicles to be positioned on top of the environment surface geometry. AutoPosition uses the current surface elevation beneath each wheel and the vehicle's tire radii and ride height to determine the earth-fixed Z coordinate of the vehicle CG which will place the vehicle in equilibrium at the start

of a simulation. AutoPosition greatly simplifies initial positioning for 3-D simulations, which otherwise tend to take a while for the simulation to reach equilibrium.

AutoPosition does not include suspension and tire deflections. Therefore, a small disturbance may still be experienced, especially on 3-D terrain which does not form a plane beneath the tires.

AutoPosition is always used for 2-D reconstructions and simulations. Because these models do not include suspension effects, on severely non-planar surfaces tires will appear to come off the ground. This visual effect does not affect calculations.

See Also: Event Set-up (Position/Velocity)

GRID

Menu Option: GRID

Purpose: Set a user-definable distance interval for display and data entry in the environment

Description: Choosing Grid from the Options Menu displays markers at user-defined intervals. These markers are used for spatial cues to help determine distances while creating and editing 3-D models of humans, vehicles and environments. See also the dialog reference: Set Grid Dialog Box.

Parameters: The parameters assigned using this dialog are shown in Table 6-1.

Table 6-1: Grid Parameters

Parameter	Units (US / SI)	Description
Grid Space	in / cm or ft / m *	Distance between grid lines or markers in the 3-D Editor viewers

* in / cm for humans and vehicles; ft / m for environments, depending on the object loaded in the 3-D Editor.

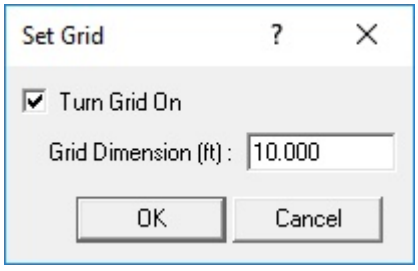


Figure 6-11: Grid Dialog used for the 3-D Editor.

See Also: 3-D Editor

UNITS

Menu Option: UNITS

Purpose: Allow the user to switch between US, SI and user-defined systems of units

Description: Choosing Units from the Options Menu displays the Units dialog, allowing the user to specify the current system of units. HVE maintains two basic sets of units: program units and user units. Program units are always pounds, inches and seconds (and derivatives of these basic quantities). These units are used for all the internal workings, and the user never sees them. Instead, the user sees the user units, which may be defined according to the user's needs through the use of the proper conversion factors. This conversion factor converts all the user's entries into program units so HVE can operate on them, and also converts all the program's results into user units. See also the dialog reference: Units Dialog Box.

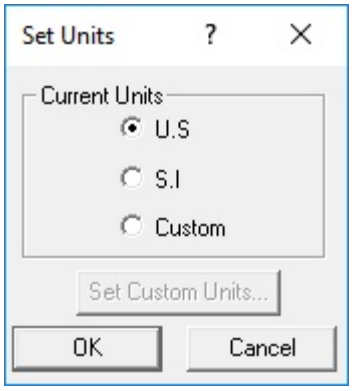


Figure 6-12: The Set Units dialog allows the user to change the current units.

See Also: Appendix IV

SHADOWS

(updated: this option was added after the 2006 manual)

Menu Option: SHADOWS

Purpose: Control rendered shadows

Description: Choosing Shadows displays the Shadow Options dialog. Options include Show Shadows (turns shadow rendering on/off) and user-editable Radius, Quality, Precision and Intensity values that control the size, smoothness and darkness of the rendered shadows.

RENDER

Menu Option: RENDER (Ctrl+R)

Purpose: Set various options that affect the visual appearance of all 3-D objects

Description: Choosing Render from the Options Menu displays the Render dialog. The Render dialog includes several user-definable options which affect the visual quality of the objects displayed by HVE. See also the dialog reference: Render Options Dialog.

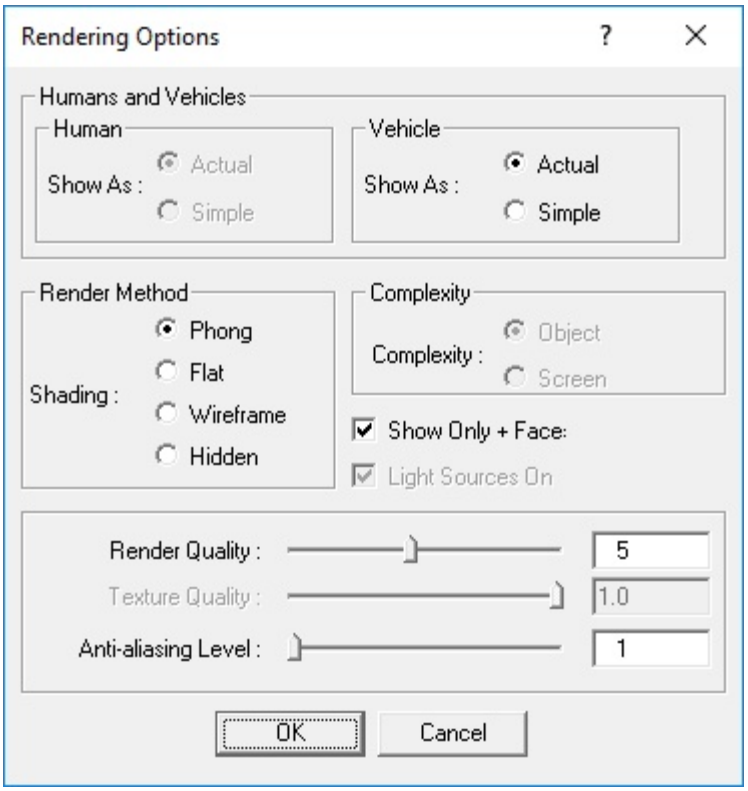


Figure 6-13: The Render Options dialog allows the user to set various rendering options which determine the image quality and rendering speed.

> **NOTE:** Increasing render quality makes objects look better, but always increases the time required to display the objects.

The available render options are:

- Show As
- Render Method
- Complexity
- Render Quality
- Anti-aliasing Level

Show As

Show As is an option for both humans and vehicles. When Show As Actual is selected, HVE displays humans or vehicles using the object's 3-D geometry file. If Show As Simple is selected, humans are displayed as ellipsoids and vehicles are displayed as simplified boxy-shaped objects.

> **NOTE:** The Human Show As choices (Actual/Simple) are currently disabled (grayed out) in the dialog.
>
> **NOTE:** If a human or vehicle does not have a 3-D geometry file, it will be displayed as a Simple object, regardless of which option is selected.

Render Method

HVE provides four Render Methods:

- Phong Shading
- Flat Shading
- Wireframe
- Hidden Line

Phong Shading is required for realistic images. Only if Phong shading is selected are special lighting and shading calculations performed. Because of these calculations, Phong shading also has the longest rendering time.

Flat Shading displays each object according to its user-defined color(s). However, no lighting or shading calculations are performed, so objects appear less realistic when compared to Phong shading; however, objects are rendered more quickly.

Wireframe mode shows only the edges of each object; no surfaces are displayed. The resulting display is faster, but realism is lost. Wireframe mode is often useful as a construction technique during model building in the 3-D Editor.

Hidden Line is like Wireframe, except it includes depth calculations: objects hidden behind other objects are not displayed. Like Wireframe, Hidden Line mode is often useful as a construction technique during model building in the 3-D Editor.

Complexity

The Complexity option allows the user to specify whether objects are drawn relative to object space or screen space. If Object is selected, all the object's polygons are rendered, regardless of the distance from the camera. If Screen is selected, HVE uses the object size and distance from the camera to decide whether to render all the object's polygons, or just render its pixels at some average color.

For example, consider an object which is a great distance from the camera. By selecting Object mode, rendering calculations are performed for every polygon, even though the result may simply be only 25 or 30 pixels in size and not even detectable to most viewers. By selecting Screen mode, these objects are displayed much more quickly, although close inspection may reveal a lesser quality image.

> **NOTE:** The Complexity choices (Object/Screen) are currently disabled (grayed out) in the dialog.

Show Only + Faces

All polygons have a front side and a back side, as determined by the right-hand rule. The positive (+) side is determined by a counter-clockwise order of vertices comprising the polygon. Choosing this option is a way to determine if one or more polygons has the vertices in the wrong order.

> **NOTE:** This is important to DyMESH.

Render Quality

Render Quality is a user-definable range (1 to 10) which defines how finely objects are subdivided (tessellated). The default value is 5. If a value less than 5 is selected, objects like cones and spheres appear more blocky; surfaces may

have unrendered areas. Selecting a value greater than 5 results in smoother cones and spheres; the appearance of surfaces may or may not improve (depending on how they were created). Again, there is a trade-off between speed and image quality.

Anti-aliasing Level

Anti-aliasing Level is a user-definable range (1 to 10) which defines how smoothly objects are rendered. The default value is 1. If a value greater than 1 is selected, objects are re-rendered up to 10 times (depending on the selected value), resulting in a significant improvement in image quality. However, rendering time is increased significantly.

> **NOTE:** In general, it is best to use the lowest possible rendering quality while editing objects and events. Boost the rendering quality before rendering the final result.

See Also: 3-D Editor

PLAYBACK

Menu Option: PLAYBACK

Purpose: Set the time step for the current playback window

Description: Choose Playback from the Options Menu to display the Playback Controls dialog. The Playback Controls dialog allows the user to select the output interval during Playback Mode. This value is used for all outputs, including the Variable Output table and Trajectory Simulations. The value is also used for the Playback Window, which may be routed to video.

> **NOTE:** Options → Playback opens this output-interval dialog. It is distinct from the toolbar transport controls (Reset/Rewind/Play) documented in the reference Playback Controls.
>
> **NOTE:** For real-time recording to video, this value should be set to 0.0333 sec (1/30 second) for NTSC video or 0.0400 sec (1/25 second) for PAL video.

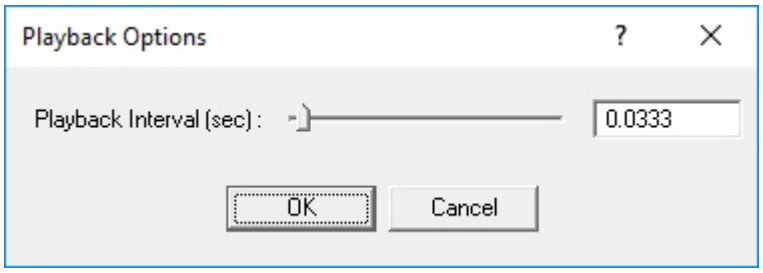


Figure 6-14: The Playback Options dialog allows the user to edit the time interval for results displayed in Playback Mode.

Parameters: The parameters assigned using this dialog are shown in Table 6-2.

Table 6-2: Playback Parameters

Parameter	Units (US / SI)	Description
Playback Interval	sec	Time interval used during Playback Mode. Affects Variable Output and Trajectory Simulation windows.

> **NOTE:** For slow-motion recording to video, the Playback Interval may be set to a value less than 1/30th second. For example, setting the value to 0.0033 will result in slow-motion video output at 1/10th real time.

See Also: Variable Output, Playback Mode, Trajectory Simulation, Playback Window

---

SIMULATION CONTROLS

Menu Option: SIMULATION CONTROLS (Ctrl+Y)

Purpose: Display and allow editing of all simulation control data for the current event

Description: Choosing Simulation Controls from the Options Menu displays the Simulation Controls dialog. See also the dialog reference: Simulation Controls Dialog.

> **NOTE:** The Simulation Controls option is only available in Event Mode, and requires that a current event be selected.

Simulation controls options are used by all simulation programs. All simulations require two basic types of simulation controls:

- Integration Timesteps
- Termination Conditions

Integration Timesteps

Integration Timesteps define the discrete timestep used by the simulation, and are dependent on the purpose and type of the simulation. The individual Integration Timestep Controls provided by HVE are shown in Table 6-3.

Termination Conditions

Termination Conditions are used to define when the simulation terminates. Termination Linear and Angular Velocities are natural termination conditions which define when a simulated object has stopped moving.

> **NOTE:** Owing to the nature of discrete timestep integration, a simulated object will never stop moving unless some non-zero termination velocity is supplied.

The Simulation Controls dialog is shown in Figure 6-15.

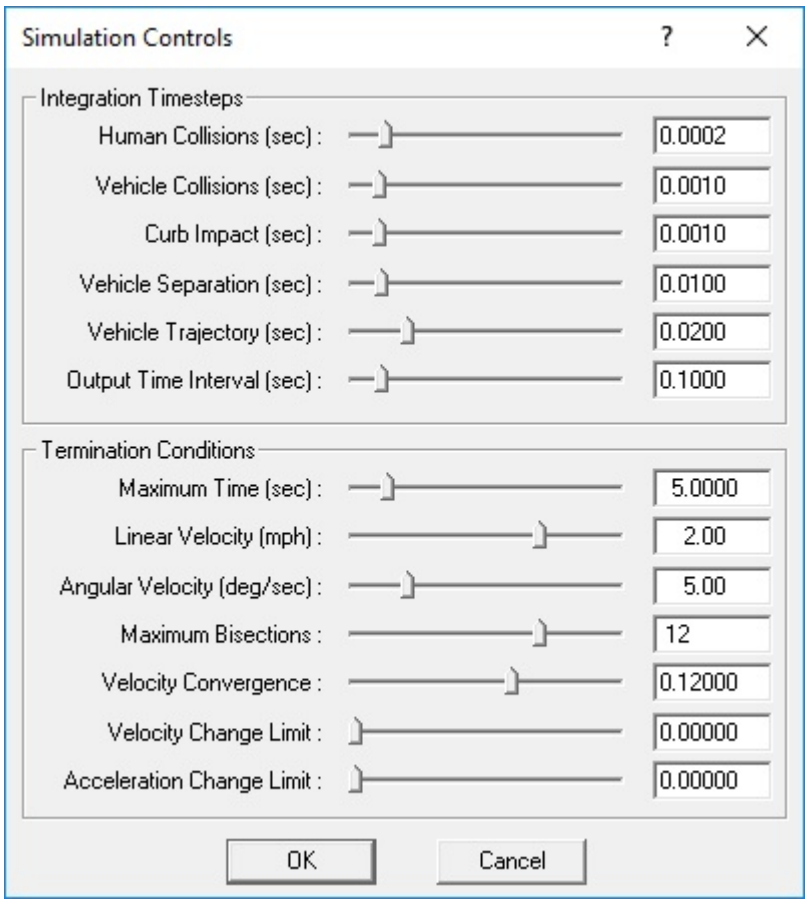


Figure 6-15: The Simulation Controls dialog allows the user to edit the simulation control parameters for the current event.

Parameters: The parameters assigned using this dialog are shown in Table 6-3.

Table 6-3: Simulation Parameters

Parameter	Units (US / SI)	Description
Dt, Human Collision	sec	Integration timestep used during the collision phase for occupant and pedestrian simulations
Dt, Vehicle Collision	sec	Integration timestep used during the collision phase for vehicle simulations
Dt, Curb Impact	sec	Integration timestep used during curb impact for vehicle simulations
Dt, Vehicle Separation	sec	Integration timestep used during the initial portion of the post-impact phase for vehicle simulations
Dt, Vehicle Trajectory	sec	Integration timestep used for vehicle trajectory simulations
Dt, Output	sec	Event output time interval
Maximum Time	sec	Maximum simulation time
Linear Termination Velocity	in/sec, mph / km/h *	Minimum linear velocity. Event termination may result if current linear velocity is less than this value.
Angular Termination Velocity	deg/sec	Minimum angular velocity. Event termination may result if current angular velocity is less than this value.
Max. Bisections	—	Maximum number of times the integration timestep may be halved
Velocity Convergence	—	Allowable error between predicted and actual integration results
Velocity Change Limit	—	Maximum allowable change in velocity during one timestep
Acceleration Change Limit	—	Maximum allowable change in acceleration during one timestep

* in/sec (US) for humans or mph (US) for vehicles; km/h in SI units, depending on the current objects.

See Also: Event Model, Simulation Model, User's Manual for calculation method

---

CALCULATION OPTIONS

Menu Option: CALCULATION OPTIONS (Ctrl+J)

Purpose: Display and allow editing of any calculation-specific parameters

Description: Choosing Calculation Options from the Options Menu displays the Calculation Options dialog.

> **NOTE:** The Calculation Options option is available in Event Mode only, and requires that the current calculation model use a Calculation Options dialog.
>
> **NOTE:** The Calculation Options dialog is different for each calculation model (e.g., EDCRASH, EDGEN, EDHIS, EDSMAC, EDSMAC4, EDVDS, EDVSM, SIMON). EDSVS and EDVTS do not have a Calculation Options dialog.

A sample Calculation Options dialog is shown in Figure 6-16.

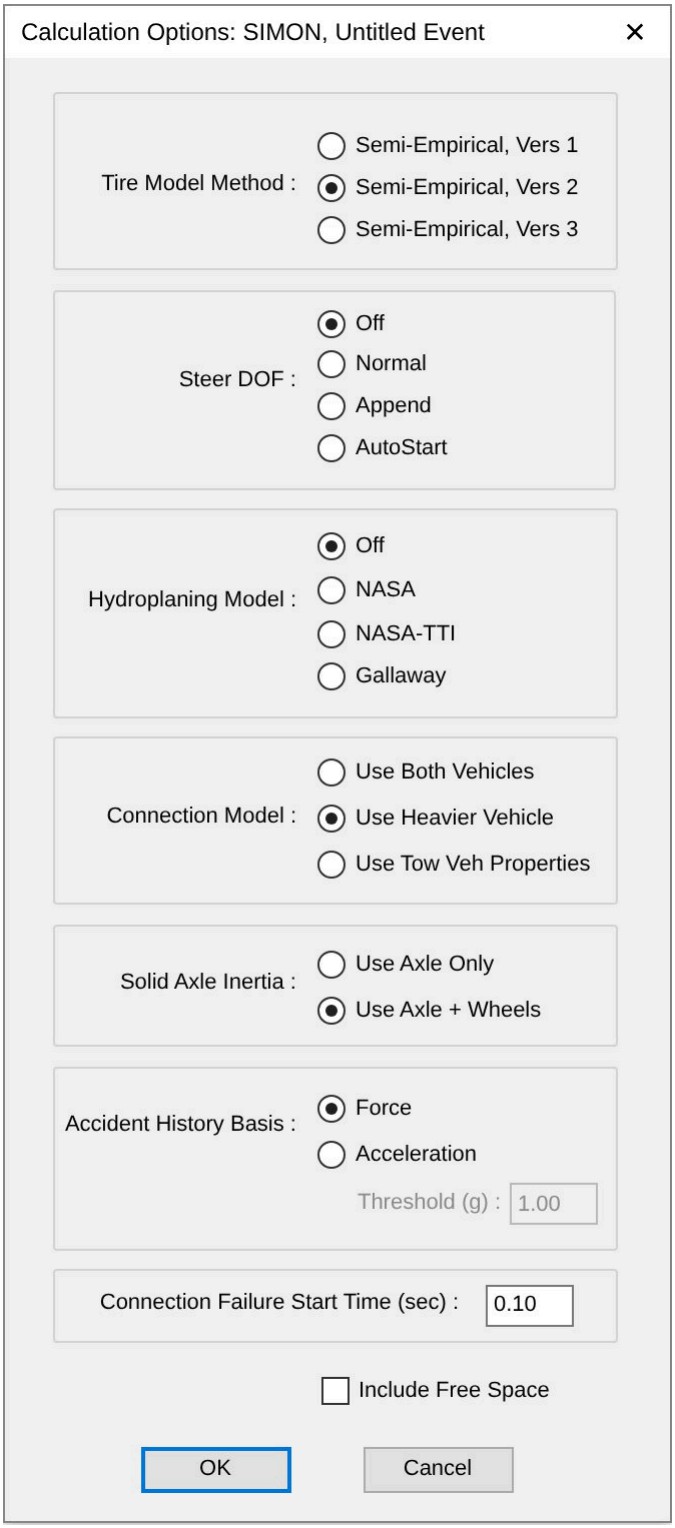


Figure 6-16: The Calculation Options dialog allows the user to supply calculation-specific parameters.

See Also: Event Model, User's Manual for the specific calculation model

DYMESH

Menu Option: DYMESH

Purpose: Allows the user to choose various modeling options available to the DyMESH collision model

Description: DyMESH is a 3-dimensional collision model used for simulating the collision phase of a collision between two or more vehicles (or a vehicle and the environment). Choosing DyMESH from the Options Menu displays the DyMESH Options dialog.

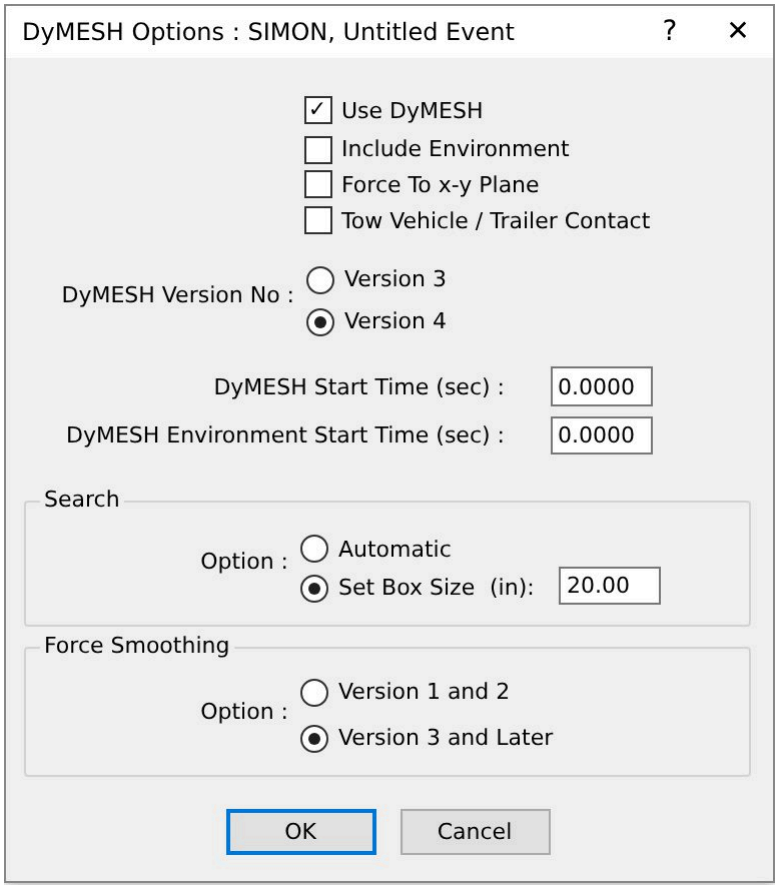


Figure 6-17: The DyMESH Options dialog.

DyMESH is enabled by clicking on the Use DyMESH check box; all other options are available only when it is checked. Using DyMESH requires a DyMESH license.

(updated: earlier releases presented this as a tabbed dialog with General and Advanced pages, including search, pushback, restitution, timestep and debug options. The current dialog is a single page containing only the options described below. The settings that were removed are fixed in the current release — see Fixed settings below.)

Options

- **Use DyMESH** — Turns the DyMESH 3-D collision model on for the event.
- **Include Environment** — Extends DyMESH contact to the environment (for example a barrier or terrain feature), in addition to vehicle-to-vehicle contact.
- **Force To x-y Plane** — Keeps the DyMESH deformation (pushback) direction horizontal. In collisions with an extremely high closing velocity (say 80 mph and above), DyMESH sometimes predicts a severe over-ride ("ski

jump"); the effect is worse with a large size mismatch, and mesh characteristics also play a role. Choosing this option improves the resulting simulation. It is not applied to contact with the environment.

- **Tow Vehicle / Trailer Contact** — Enables DyMESH contact between vehicles joined by an inter-vehicle connection, such as a tractor and trailer in a jackknife.
- **DyMESH Version No** (Version 3 / Version 4) — Selects the DyMESH algorithm. Version 4 (the default for new events) adds a refinement to the restoration step that reduces pushback; Version 3 reproduces results from cases created with earlier releases.
- **DyMESH Start Time** (sec) — Simulation time at which DyMESH vehicle-to-vehicle contact detection begins.
- **DyMESH Environment Start Time** (sec) — Simulation time at which DyMESH vehicle-to-environment contact detection begins.

Search

For each vertex on the secondary vehicle, DyMESH must find the candidate polygons on the primary vehicle that it may have penetrated.

Option — Two methods are available:

- **Automatic** — Locates the polygons directly attached to the polygon through which the vertex has penetrated, by searching that polygon's neighbors at each timestep. Potentially the fastest method.
- **Set Box Size** (default) — Finds all polygons on the primary vehicle within the entered distance (in) of the secondary vertex.

Force Smoothing

Option — Selects how the collision forces are averaged from one timestep to the next as DyMESH alternates the primary and secondary roles (see the SIMON manual, Chapter 4, Primary and secondary vehicles):

- **Version 1 and 2** — The smoothing used by earlier releases, kept so that older cases reproduce their original results.
- **Version 3 and Later** (default) — The current smoothing model.

Fixed settings

The following settings were adjustable in earlier releases but are fixed in the current release. Every new event uses the values shown.

Setting	Value used
Pushback Method	By Primary — surface normals of nearby polygons on the primary vehicle, weighted by nearness
Don't Restore Toward Other CG / Don't Restore Away From This CG	Off
Restitution Model	Relaxation Length — the percentage of vector displacement recovered during rebound, set for each pair of colliding vehicles in the Vehicle Mesh dialog (Set-up menu)
Stop After	All Vertices — every candidate polygon is examined and the one needing the smallest pushback is used
Search Method	Use Polygon Vertices — a polygon is a candidate if its vertices are found in the search
Inside Vehicle Test	Primary normal — a vertex lying on the outward side of the primary polygon's normal is outside the vehicle and is ignored
Inside Polygon Test	Cross product
Don't Deform A Single Vertex / Don't Restore A Large Distance	Off

> **NOTE:** A case created in an earlier release keeps whatever values were chosen for these settings at the time, and the simulation still uses them, although the current dialog does not display them.

The DyMESH timesteps are set in the Simulation Controls dialog (see Figure 6-15). Collision simulation using DyMESH typically requires smaller integration timesteps, and because the collision phase is short it is often helpful to use a smaller output time interval as well.

See Also: Event Model, User's Manual for the specific calculation model

---

GET SURFACE INFORMATION

Menu Option: GET SURFACE INFO

Purpose: Sets various options that determine which terrain surface polygons vehicle tires may drive on

Description: Choosing Get Surface Info from the Options Menu displays the Get Surface Information Options dialog. All vehicle simulation models (e.g., SIMON, EDVSM or EDVDS) must calculate interaction forces between the tires and the terrain. This is accomplished by the simulation's tire model. Various options are available for calculating this interaction. The Get Surface Information dialog displays these options (see Figure 6-19), selectable from the Options menu.

While a simulation is executing, during each integration timestep the tire model searches through the database of terrain polygons to determine the characteristics of the polygon beneath each tire. The Get Surface Information dialog includes two options that affect this search:

- **Search Method** — There are three options:
- **From First Polygon:** The search starts at the top of the polygon database for every timestep.

- **From Previous Polygon:** The search starts by looking at the polygon found during the previous timestep. If not the correct polygon, the search spreads in both directions until the correct polygon is found.
- **By Elevation:** The entire polygon database is searched. The polygon closest to the underside of the undeflected tire is selected.

> **NOTE:** By Elevation is not implemented. - **Search Direction** — Each polygon in the terrain has a surface normal whose direction is determined according to the right-hand rule. In general, it makes sense to be driving only on surfaces that have their normals facing upwards. - **All Directions:** Choose this option if you wish to include all terrain polygons, regardless of the direction of their surface normals. This option may be useful if the terrain includes ill-behaved surface normals (i.e., some normals pointing up, others pointing down).
>
> **NOTE:** Choose this option if you are simulating a stunt driver performing a loop-the-loop. - **Upward Facing Only:** Choose this option if you wish only to include terrain polygons with upward-facing normals. This is the most common situation. - **Z Component Greater Than:** This option allows the user to assign the range for polygon normals. It is useful when curbs with near-vertical faces are included in the polygon database.
>
> **NOTE:** This option may be useful for the Radial Spring and Sidewall Impact Tire Models.
>
> **NOTE:** In a vertical face, the Z component should be exactly zero. Because of rounding error when the surfaces were created, it is possible that a vertical face may have a small Z component.

To assign these options, perform the following steps:

1. Choose Get Surface Info from the Options Menu. The Get Surface Information dialog will be displayed, showing the current state of each option.
2. Click on the radio button to select the desired option.
3. Enter the range for surface normal Z Component, if applicable.
4. Choose OK to update the current Get Surface Information options.

(updated: closing the dialog with OK re-tessellates the environment surfaces used by physics.)

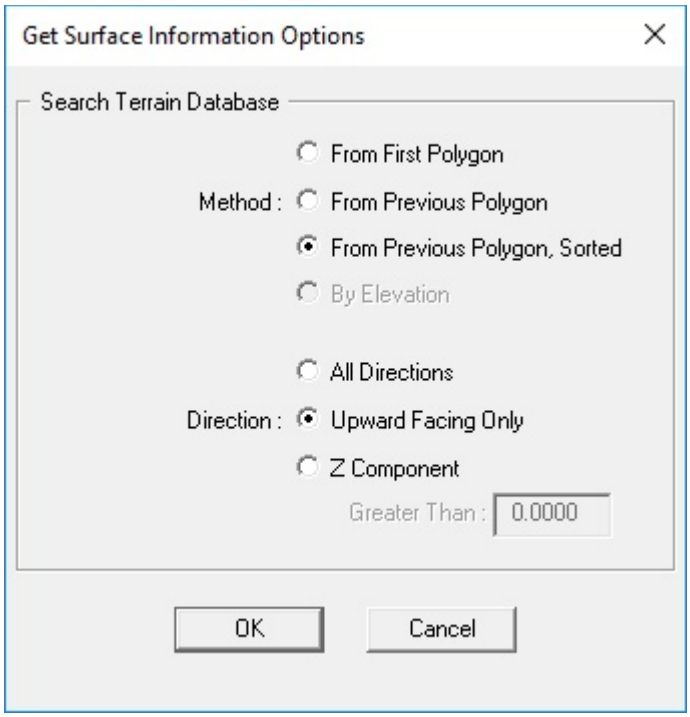


Figure 6-19: The Get Surface Information Options dialog allows the user to set various options related to tire-terrain interaction.

PREFERENCES

Menu Option: PREFERENCES (Ctrl+F)

Purpose: Customize HVE according to various user-selectable preferences

Description: Choosing Preferences from the Options Menu displays the User Preferences dialog. See also the dialog reference: User Preferences Dialog Box. The Preferences dialog allows the user to select the following preferences:

- Auto Backup
- Date Style
- Hide Date/Time From Printed Reports
- Key Results
- Vehicle Dimensions Basis
- Background Color
- Skidmark Height
- Font Size (for report windows)
- Physical Monitor Width
- Long Velocity Vectors
- Cycle Vehicle Colors
- Resolution Setting
- Store Vehicle Geometries in case file
- Reload language.rsc file on dialog exit

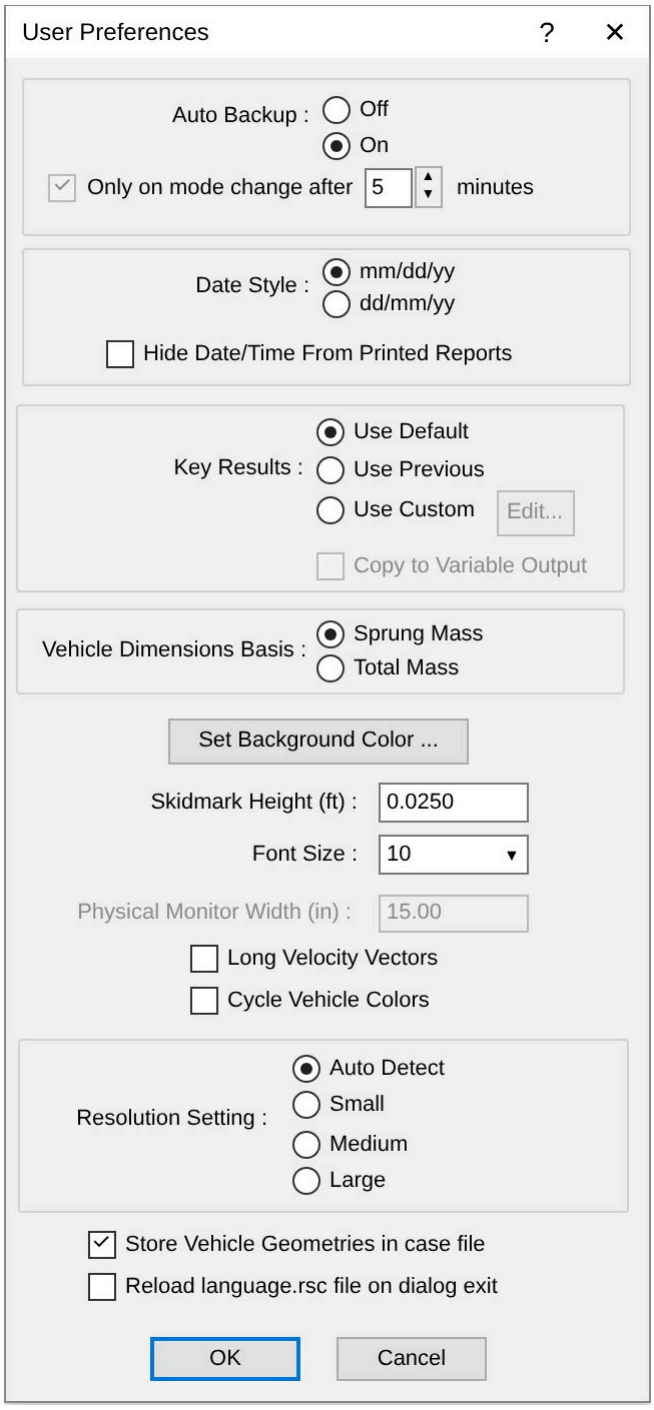


Figure 6-20: The Preferences dialog allows the user to customize HVE according to various user-selectable preferences.

Auto Backup

The Auto Backup option allows the user to save his/her work at specified time intervals.

> **NOTE:** To prevent interruption while working, the backup process occurs only when changing modes (e.g., switching between the Event Editor and the Playback Editor).

Date Style

HVE allows the user to select from two date styles:

- US Standard (month/day/year)
- Military (day/month/year)

Military style is also used in Europe and many other countries.

Background Color

Use the Background Color option to set the background color of the viewer in the Human and Vehicle Editors.

> **NOTE:** Changing the background color helps while using the Human or Vehicle Editor if the human or vehicle is nearly the same color as the background.

Skidmark Height

If a skidmark and the road surface are both drawn at the same elevation (earth-fixed Z coordinate), the two objects will visually fight each other to determine which one is visible. Placing the skidmarks slightly above the road surface ensures that the skidmark is properly visualized. The Skidmark Height is the distance above the road surface at which skidmarks are positioned.

By keeping the Skidmark Height small (the default value is 0.025 ft, or 0.3 in), it is not possible to see that the skidmark does not lie directly on the surface.

> **NOTE:** If a skidmark occurs over a non-uniform surface (e.g., a curb) it may be necessary to increase the Skidmark Height.
>
> **NOTE:** If the scene is viewed from a great distance, it may be necessary to increase the Skidmark Height.
>
> **NOTE:** Pavement striping must also be placed above the road surface (for exactly the same reason). Therefore, it is possible that skidmarks may be displayed beneath road stripes. This is another situation in which it may be necessary to increase the Skidmark Height.

Font Size

Adjusting the size of the font changes the font used in both displayed and printed report windows in the Playback Editor.

Other Preferences

Hide Date/Time From Printed Reports, Key Results, Vehicle Dimensions Basis, Physical Monitor Width, Long Velocity Vectors, Cycle Vehicle Colors, Resolution Setting, Store Vehicle Geometries in case file and Reload language.rsc file on dialog exit are described in the User Preferences Dialog Box reference.

See Also: Playback Mode, Messages, Human Editor, Vehicle Editor, Environment Options

---

ADD PLAYBACK WINDOW (legacy)

(updated: this command is no longer located in the Options menu. Playback windows are now added from Playback mode — see the Playback Information dialog. The original 2006 description is retained below for reference.)

Menu Option: ADD PLAYBACK WINDOW

Purpose: Create a Playback Window used for combining multiple events into a complete sequence, creating a compressed simulation movie file and also routing a compressed movie file out to videotape

Description: Choosing Add Playback Window displayed the Playback Window dialog. Previously created trajectory simulations were listed in the Active Events table portion of the Playback Window.



Figure 6-21: The Playback Window allows the user to combine multiple events into a coherent sequence.

The Playback Window includes the following components:

- **Playback Window Viewer** — a 3-D window used for viewing trajectory simulations.
- **Active Trajectory Simulation Windows List** — A table used for selecting and editing simulation sequences that are to be included in the 3-D Playback Window viewer.
- **Video Source List** — An option list used for selecting the current video source.
- **Video Destination List** — An option list used for selecting the current video destination.

These components are described below.

Playback Window Viewer

The Playback Window Viewer is a 3-D viewer that displays trajectory simulations for multiple events in a single window. This viewer is like all other 3-D viewers in HVE. In addition, it has the capability of outputting its contents to a compressed movie file, as well as replaying a previously created movie file in real time.

Active Trajectory Simulation Windows List

The Active Trajectory Simulation Windows List is a table that is used for selecting trajectory simulations to be included in the Playback Window Viewer, and for editing the sequence of the individual trajectory simulation events. The table includes the following components:

- **Checkbox** — Used to select or deselect trajectory simulations that are to be included in the Playback Window Viewer
- **Name** — The name of the trajectory simulation, assigned by the user when the trajectory simulation report was created
- **Tdelay** — A time offset, used to delay the beginning of the trajectory simulation
- **Tstart** — A time offset, used to remove from the Playback Window Viewer a portion at the start of the trajectory simulation
- **Tend** — A time offset, used to remove from the Playback Window Viewer a portion at the end of the trajectory simulation

Video Source List

The Video Source List is an option list that displays the current video source, that is, the source of the information displayed in the Playback Window Viewer. There are two options:

- **Playback Window Name** — This is the user-assigned name given to the sequence of Trajectory Simulations defined by the Active Trajectory Simulation Windows Table (described earlier). This source is rendered directly from the simulation output tracks, and thus is typically slower than real time.
- **Movie File Name** — This is the user-assigned name given to a previously saved movie file. Alternatively, it may simply be the name of the last movie file to be created. By default, its name is AVI. This movie file can normally be displayed in real time (depending on the user's hardware).

Video Destination List

The Video Destination List is an option list that displays the current video destination, that is, the destination of the information displayed in the Playback Window Viewer. There are three options:

- **Playback Window Name** — This is the user-assigned name of the Playback Window Viewer. In this case, the viewer is displaying the sequence of the trajectory simulations defined in the Active Trajectory Simulation Windows Table (described earlier). Choosing Movie File Name or AVI as the source and Playback Window Name as the destination results in the real-time playback of a compressed movie file.
- **AVI** — This is the default destination for the video compressor, selected using the Video Set-up dialog (see Chapter 3, File Menu, Video). Choosing Playback Window Name as the source and AVI as the destination results in the creation of a compressed movie file.
- **Composite Video/S-Video** — Choosing this destination causes the current video source to be routed to the computer's video output port.

> **NOTE:** This option requires the installation of a video card capable of outputting an NTSC or PAL video signal. See Section Nine, Video Output, for more information.

See Also: Playback Editor, Combining Multiple Events, Creating a Compressed Movie File, Creating a Video, Video Set-up, Playback Controller

Chapter 7: Help Menu

HVE User's Manual — Section Two: Menu Reference. Updated edition, revised for the current HVE release.

The Help Menu contains the following options:

- **Help Topics** — Provides an on-line reference system for terms related to human and vehicle dynamics and general motor vehicle safety. Terms related to accident reconstruction are available as well.
- **User Manuals** — Opens the PDF user manual for HVE or any installed calculation model *(updated: added since the 2006 manual)*
- **Tech Support** — Provides information about the user's name, company name and user ID number. Also provides contact details for EDC Technical Support.
- **About** — Provides information about the user's version of HVE, including the version number and release date.
- **Online Licensing** — Provides commands for registering a User ID Code and refreshing licenses over the Internet *(updated: added since the 2006 manual)*

This chapter describes these help systems.

HELP TOPICS

Menu Option: HELP TOPICS

Purpose: Provide an on-line help reference, organized by category (Contents) and by term (Index)

Description: Choosing Help Topics from the Help Menu displays the Help Topics dialog. This is a single menu item; the Contents and Index views are tabs within the dialog.

The **Contents** tab provides four basic categories of program information.

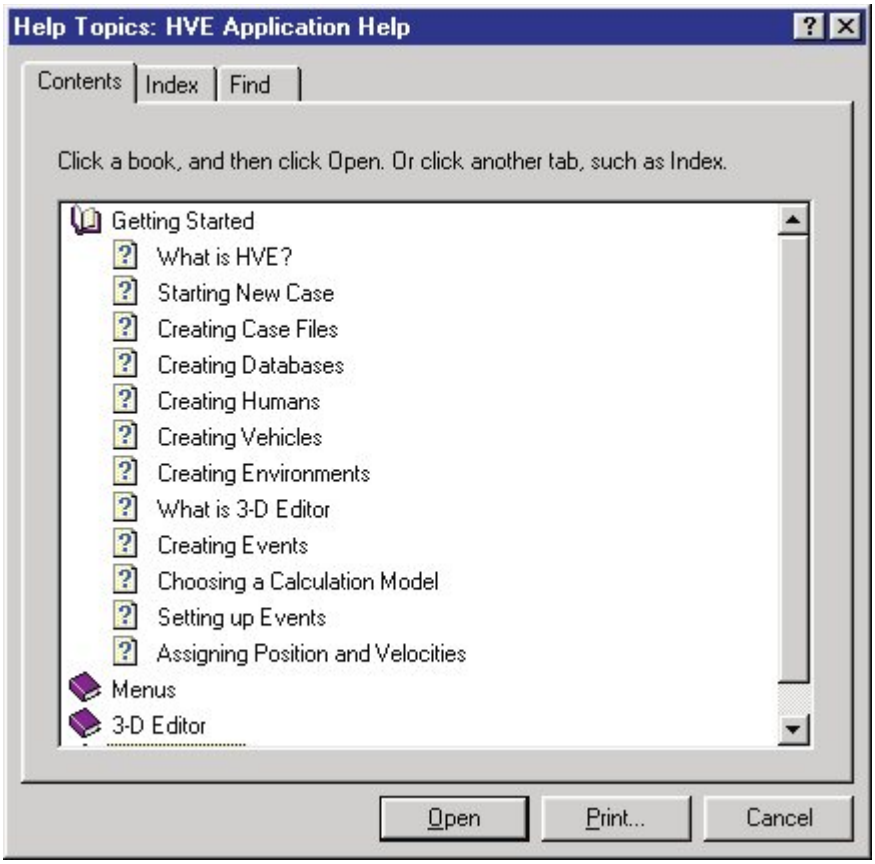


Figure 7-1: Typical Help Topics — Contents tab.

The **Index** tab provides a detailed glossary of terms, including SAE J670e (Vehicle Dynamics Terminology; reference 6.2), SAE J885 (Human Tolerance to Impact Conditions As Relates To Motor Vehicle Design; reference 6.1), and SAE J1675 (Accident Reconstruction Terminology, ref. 6.4).

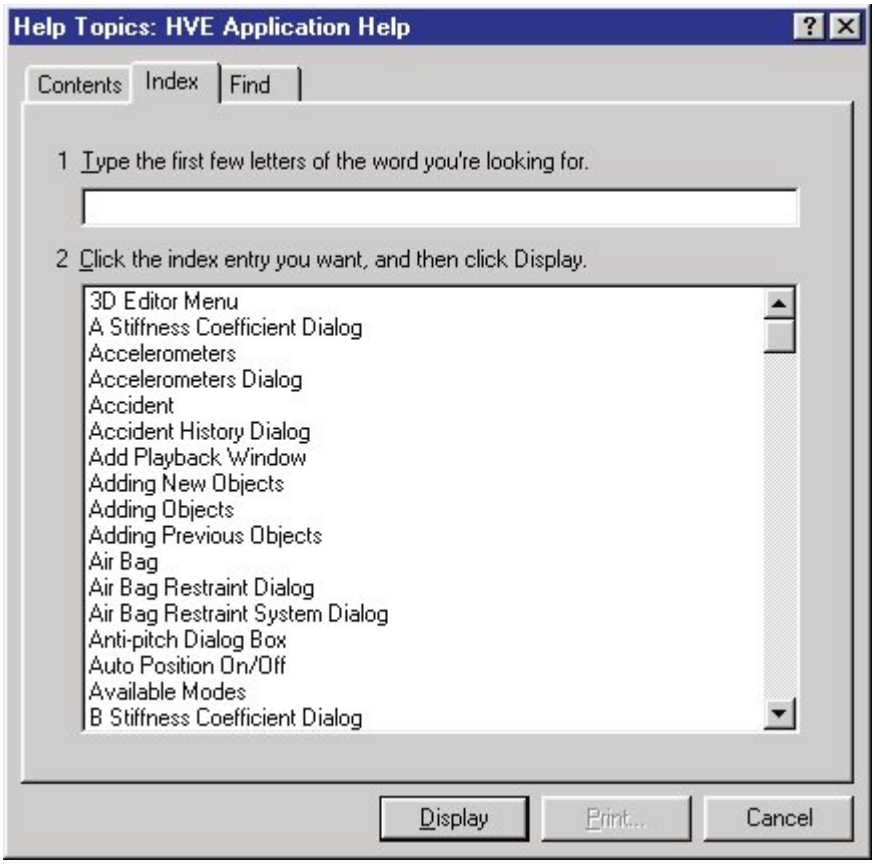


Figure 7-2: The Help Topics — Index tab provides definitions of many human and vehicle dynamics terms and terms related to accident reconstruction and motor vehicle safety.

USER MANUALS

(updated: this cascade menu was added after the 2006 manual)

Menu Option: USER MANUALS

Purpose: Open the PDF user manual for HVE or an installed calculation model

Description: Choosing User Manuals displays a cascade menu of the manuals installed with HVE. Selecting an entry opens the corresponding PDF manual. The available manuals include:

- HVE User Manual
- EDCRASH User Manual
- EDGEN User Manual
- EDHIS User Manual
- EDSMAC User Manual
- EDSMAC4 User Manual
- EDVDS User Manual
- EDSVS User Manual
- EDVSM User Manual

- EDVTS User Manual
- SIMON User Manual
- Damage Studio User Manual
- GATB User Manual
- ReadDataFile User Manual

TECH SUPPORT

Menu Option: TECH SUPPORT

Purpose: Display EDC Technical Support contact details and user information

Description: Choosing Tech Support displays the EDC Technical Support dialog. The EDC Technical Support dialog includes the following information about the installed version of HVE:

- EDC Telephone Number
- EDC Email Address
- Licensed User Name
- Company Name
- User ID Number

The Technical Support dialog also displays your system software and licensing information. This includes:

- Your HVE Version and Release (service pack)
- Your System Codes (Machine Id and User ID Code)
- Your License File Information (Options and Versions)

A sample Technical Support dialog is shown in Figure 7-3.

> **NOTE:** This information is required for assistance when contacting EDC Technical Support regarding licensing issues. The report may be copied to the system clipboard and pasted into an email, or printed.

To access the HVE Technical Support Help Information, perform the following steps:

1. Choose Tech Support from the Help Menu. The Help Technical Support dialog will be displayed (see Figure 7-3).
2. Press Clipboard to copy the License information to your system clipboard (you may then paste this in an email message to the EDC Technical Support staff).
3. Press Print to print a hard copy of the License information.
4. Close the dialog when you are finished.

The

Reset Defaults

button returns the Key Results windows to their default screen positions.

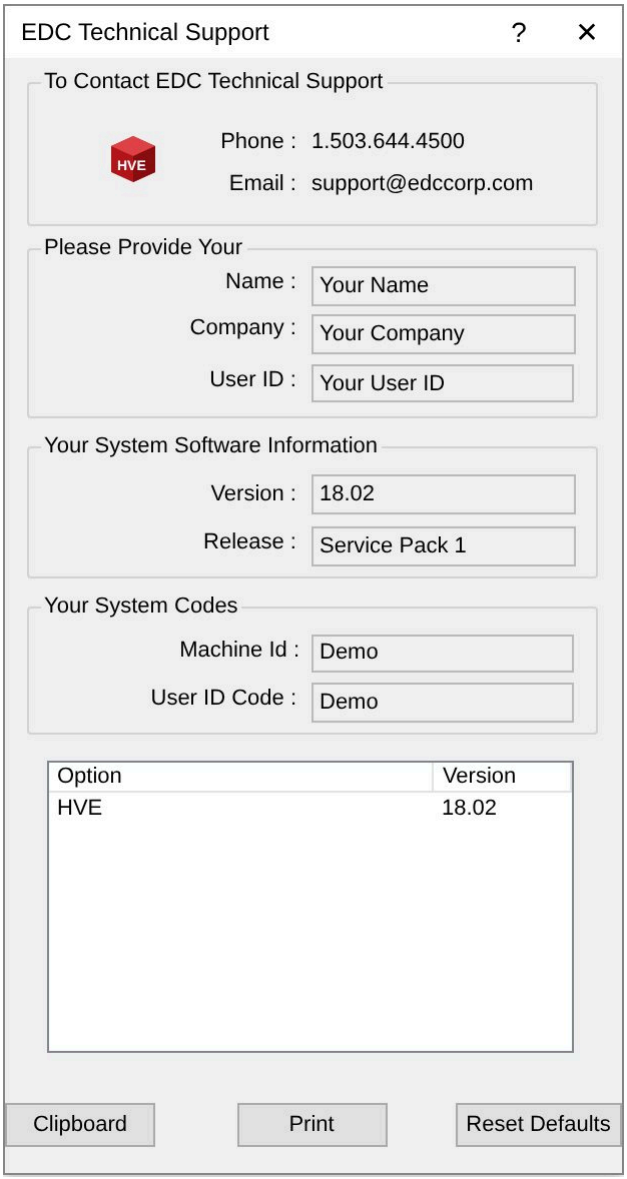


Figure 7-3: The Help Technical Support dialog explains how to contact EDC for Technical Support.

See Also: (none)

ABOUT

Menu Option: ABOUT

Purpose: Display release information about the user's version of HVE

Description: Choosing About displays the About HVE dialog. The About HVE dialog includes the following information about the installed version of HVE:

- Program name (e.g., HVE 2025)
- Service Pack
- Version Number

- Build Date and Build Time

> **NOTE:** You should have this information available when you call for Technical Support.

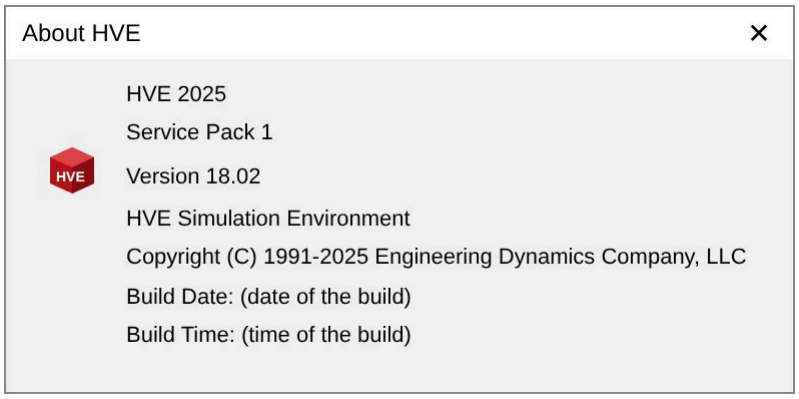


Figure 7-4: The About HVE Dialog contains useful release information.

See Also: (none)

ONLINE LICENSING

(updated: this cascade menu was added after the 2006 manual)

Menu Option: ONLINE LICENSING

Purpose: Register and refresh HVE licenses over the Internet

Description: The Online Licensing cascade menu contains two commands:

- **Register User ID Code** — Displays a dialog for entering the User ID Code supplied by EDC, registering this installation of HVE for online licensing.
- **Refresh Licenses** — Contacts the EDC licensing service and refreshes the local license information for HVE and the installed calculation models.

Section Three: Human Editor

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Section Three "Human Editor". Revised for the current HVE release.

The purpose of the HVE Human Editor is to provide anthropomorphically correct 3-D human models for use in dynamic reconstructions and simulations involving motor vehicle occupants and pedestrians. This section of the HVE User's Manual describes how to create and edit these human models. It also provides a detailed description of the human model parameters.

The basic physical/mathematical model of the HVE human is a 15-segment, 14-joint model. Each of these segments has dimensional and inertial properties that may be edited by the user. In addition, each segment may have attached to it one or more contact ellipsoids (up to three per segment), which are physical surfaces used by simulations to detect interaction with other objects and to calculate the resulting forces. The details of the HVE Human Model are provided in Chapter 9.

General Procedure

To use the Human Editor, perform the following steps:

1. Select Human mode.
2. Select one or more humans, either from the Human Database (via the GEBOD body generator in current versions — see Chapter 8) or from a previous HVE case.
3. If desired, modify the properties of the selected human(s) to suit the needs of the current case. Properties which may be modified are:
4. Inertial Properties
5. Segment Color
6. Contact Ellipsoid Properties
7. Injury Tolerances
8. Joint Properties

Overview of Section Three

This section of the HVE User's Manual provides a detailed explanation of the process of creating and editing humans, and also presents a detailed overview of the human model's physical properties:

- Chapter 8 — Creating and Editing Humans describes the process of creating and editing humans.
- Chapter 9 — Human Model Definition provides a detailed overview of the HVE Human Model.

Related Dialog Reference Pages

The individual Human Editor dialogs are documented in detail in the Human Dialog Reference:

- Human Information Dialog (classic and GEBOD)
- Human CG (segment popup) Dialog
- Inertial Data Dialog

- Contact Ellipsoids Dialog
- Human Material Properties Dialog
- Injury Tolerance Data Dialog
- Joint Data Dialog
- Joint Properties Dialog
- Selected Humans & Vehicles Dialog

Chapter 8 — Creating and Editing Humans

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 8, manual pages 8-1 through 8-8. Revised for the current HVE release.

The Human Editor is used for creating and editing humans for the current case. These humans may also be used in other cases.

This chapter describes how to use the Human Editor to create and edit humans, beginning with a description of its components.

> **NOTE:** Refer to the next chapter, Human Model Definition, for a detailed description of each human model parameter.

Human Editor Components

To use the Human Editor, choose Human Mode using the mode selector. This puts HVE in Human Mode and allows access to the Human Editor's components:

- **Human Editor Dialog** — The Human Editor dialog is used for adding humans to the current case and to select the current human.
- **Human Viewer** — The Human Viewer is used for visualizing the current human and editing his or her parts.

These components are shown in Figure 8-1 and are described later in this chapter.

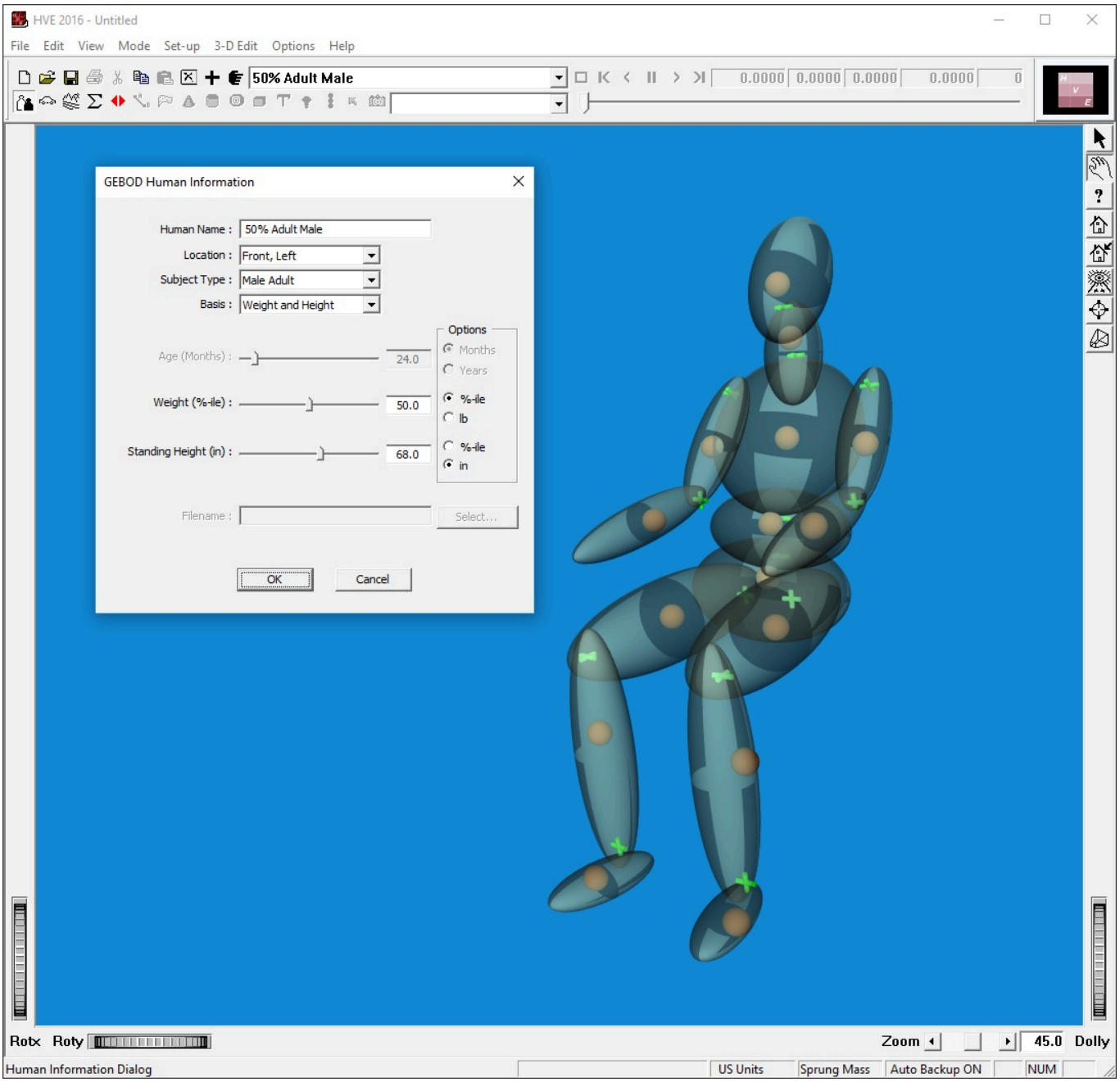


Figure 8-1: Human Editor, Human Information Dialog and Viewer.

Human Editor Dialog

The Human Editor dialog is the heart of the HVE Human Editor. It is used to manage (create and edit) all the humans in the current case. The Human Editor dialog includes the following functionality:

- **Add Human** — Allows the user to add humans to the current case and provides two options: New and Previous. Choose New to add a human from the Human Database; choose Previous to add a human from a previous HVE case. These options are further described below.
- **Active Humans List** — Displays a list of the names for all the humans defined in the current case. The name of the current human is displayed in the minimized list. Click on the combo box to view the full list and select a different

name to change the current human in the viewer. Click on the Object Info button on the toolbar to display its Human Information dialog.

- **Delete Human** — Removes the current human from the case. Select Edit, Delete on the main menu, or click on the Delete button on the toolbar to delete the current human.

> **NOTE:** Pressing Delete deletes the human only from the current case, NOT the database!
>
> **NOTE:** The Delete option is not selectable if the current human is used in an event.

Human Viewer

The Human Viewer displays the current human. It contains the following components:

- **Current Human** — Allows the user to visualize the current human. Also displays the pickable spheres on each of the 15 human segments used for editing the current human's parameters (see Selecting and Editing Humans, later in this chapter; also see Chapter 9 — Human Model Definition).
- **Viewer Controls** — Allow the user to Rotate, Pan, Zoom and set the current Pick Mode (see Chapter 2 for details on using the viewer controls).

> **NOTE:** Although every human's properties are defined using the Human Editor, humans are not positioned using the Human Editor. The position of a human in any event is an event-related issue, and is performed using the Event Editor. Think of it this way: At this point, we don't care what the human is doing. All we care about is that the human is a male, weighs 180 pounds and has other properties of our choosing.

Adding New Humans

New humans are added to the current case using the Human Information dialog.

*(updated: As of HVE Version 10, new humans are created using the **GEBOD Human Information dialog**, which generates the human data set directly from the GEBOD body generator instead of selecting from a fixed database of body types. The classic Human Information dialog described in the original manual is still displayed when editing a human that was created from the older, pre-Version 10 human database. Both dialogs are documented in detail in the Human Information Dialog reference page.)*

Classic Human Information Dialog (pre-Version 10 humans)

To add a new human using the classic dialog, perform the following steps:

1. Click on Add New Object. The Human Information dialog (see Figure 8-2) will be displayed, containing the following options:
2. **Name** — A user-editable field allowing the user to assign a name to the current human.
3. **Location** — Displays a list allowing the user to choose a seat position for the current human occupant (Right Front, Left Front, etc.); refer to the Vehicle Editor for a description of the available seating positions within a vehicle. The user may also choose Pedestrian.

> **NOTE:** You will notice that human occupants assume a seated position by default, while human pedestrians assume a standing position. This is a reminder, since the simulation program often treats occupants and pedestrians differently. Refer to your simulation program manual for more information.

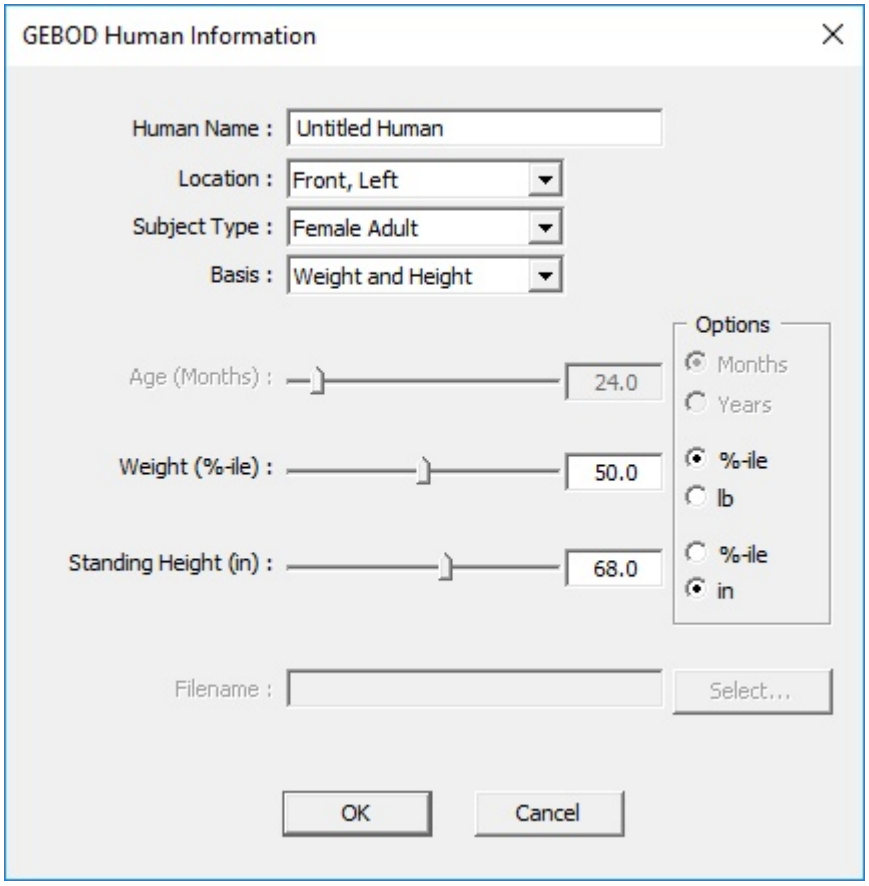


Figure 8-2: Human Information dialog, used for adding a new human from the Human Database.

- **Sex** — An option list. The user may select either Male or Female.
- **Age** — An option list allowing the user to select Adult or 3, 6, 9 or 12 year-old children's ages.
- **Weight Percentile** — An option list containing the available weight percentiles.
- **Height Percentile** — An option list containing the available height percentiles.

> **NOTE:** Regarding Weight and Height Percentiles: 50 percentile is average; 2.5 percentile means that only 2.5 percent of the population is smaller; 97.5 percentile means that only 2.5 percent of the population is larger.

1. Enter a name for the current human. The name will be used to identify the human in each event as well as in all output reports.

> **NOTE:** The default human name is supplied according to the selected human attributes (Sex, Age, Weight and Height Percentile).

1. Choose a position for the current human.

> **NOTE:** If a seat position (e.g., Left Front) is selected, the human is assumed to be a vehicle occupant and its motion will be defined relative to the vehicle. If Pedestrian is selected, the human's motion will be defined relative to the earth.

1. Choose the Sex, Age, Weight Percentile and Height Percentile for the current human.

2. Choose OK to create the selected human and add it to the case.

The selected human will be added to the Active Humans List and become the current human.

Table 8-1: Human Dummy Data, Child

Age (years)	Height (in)	Weight (lb)
3	38.94	23.08
6	45.26	46.93
9	51.58	70.78
12	57.91	94.63

Table 8-2: Human Dummy Data, Adult

Size (%-ile)	Female Height (in)	Female Weight (lb)	Male Height (in)	Male Weight (lb)
2.5	51.19†	94.75	65.04	131.50
5.0	59.94	—	65.81	—
50.0	63.82	127.24	69.82	173.50
95.0	67.70	—	73.83	—
97.5	68.45	159.70	74.60	215.50

† Value as printed in the original manual; it is inconsistent with the 5.0 percentile value (59.94 in) and appears to be a misprint in the original edition.

GEBOD Human Information Dialog *(updated: Version 10 and later)*

In current versions of HVE, clicking Add New Object opens the GEBOD Human Information dialog. Instead of picking a body type from fixed database keys, the dialog runs the GEBOD (GEnerator of BOdy Data) regression program to compute the complete anthropomorphic data set for the requested subject. The dialog contains:

- **Name** and **Location** — Same purpose as in the classic dialog.
- **Subject Type** — Child, Female, Male or User-defined.
- **Basis** — Selects which body measurement(s) the GEBOD data set is based on: Age, Weight, Height or All.
- **Age** — Slider and edit field, entered in Months or Years (Options radio buttons).
- **Weight** — Slider and edit field, entered as a percentile or directly in weight units.
- **Standing Height** — Slider and edit field, entered as a percentile or directly in length units.
- **Filename / Select** — For a user-defined subject, selects the GEBOD data file used to create the human.

> **NOTE:** The *User-defined* subject type is not yet supported; selecting it reverts to the previous type, and its *Filename / Select* controls are inactive.

See the Human Information Dialog reference page for the complete field-by-field description of both dialogs.

Human Database

The Sex, Age, Weight Percentile and Height Percentile are actually keys used to select the current human from the Human Database. These keys provide inputs to the GEBOD program [3.10] that are used to calculate anthropomorphic data for the 15-segment, 14-joint human model described in the next chapter.

(updated: with the GEBOD Human Information dialog, GEBOD is run directly from the entered subject type, basis and body measurements, so the available body descriptions are no longer limited to the discrete database keys listed in Tables 8-1 and 8-2.)

The Human Editor creates a copy of the selected human for use in the current case. The properties of this human may be edited. Changes made to the current human do not affect the Human Database.

Adding Humans From Previous Cases

The user may also include humans from other cases in the current case. This option is useful when another case includes a uniquely modified human which is substantially similar to a human required for the current case.

To choose a human from a previous case for use in the current case, perform the following steps:

1. On the Main Menu, select Mode, Add and choose Previous. The Previous Human Selection dialog (see Figure 8-3) will be displayed. This dialog contains two list boxes: Active Cases and Active Humans.
2. Click on a case name in the Active Cases list box. The Active Humans List for that case will be displayed in the Active Humans list box.
3. Choose a name from the Active Humans List.
4. Press Open to add the selected human to the current case.

The selected human will be added to the Active Humans List and become the current human.

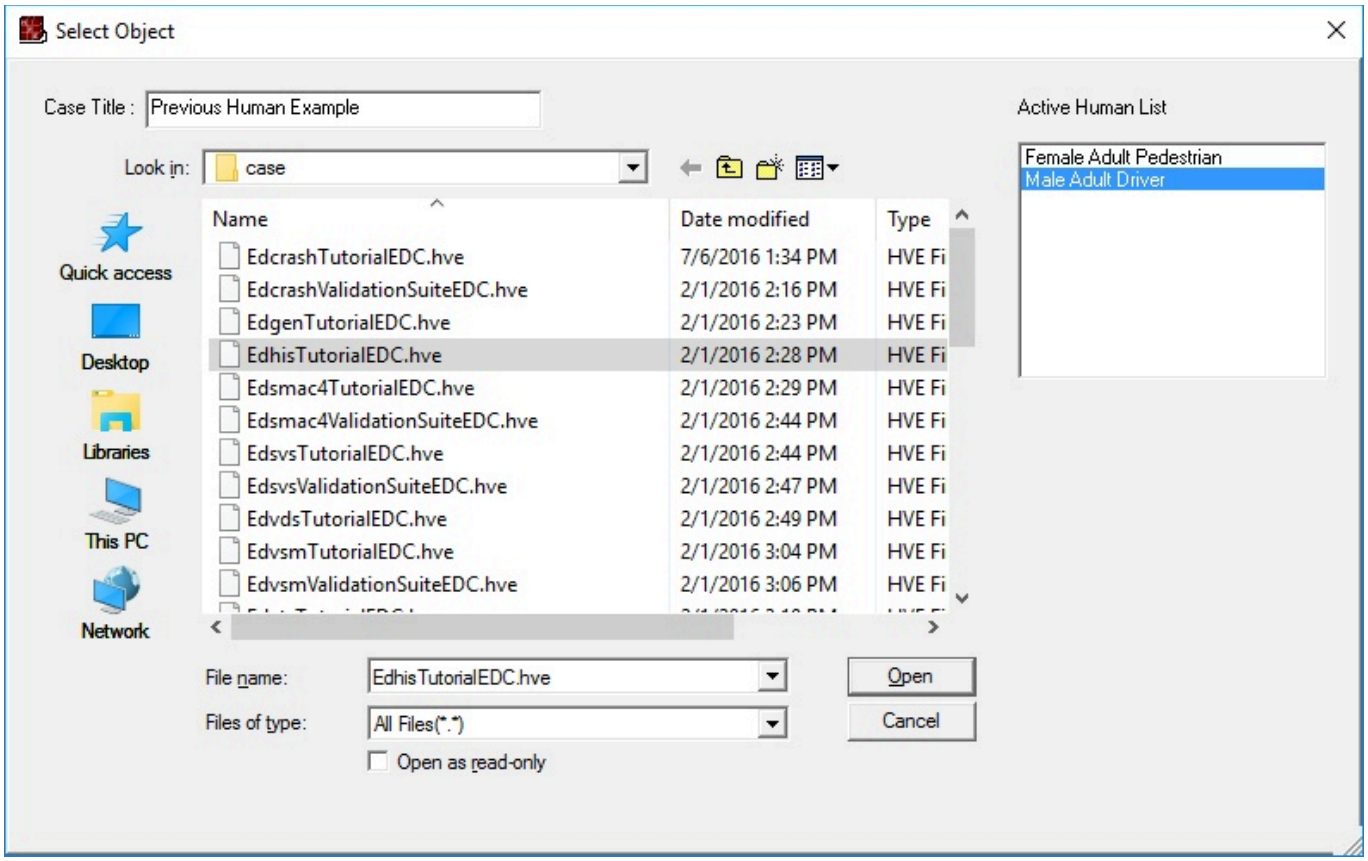


Figure 8-3: Add Humans, Previous dialog.

Selecting and Editing Humans

After a New or Previous human is created, its name is added to the Active Humans List. The last human added to the case becomes the current human. The current human is displayed in the Human Viewer and may be edited.

To select and edit any human in the Active Humans List, perform the following steps:

1. Click on the desired human name in the Active Humans List. The selected human becomes the current human and will be displayed in the Human Viewer.
2. If desired, click on the Object Info button on the toolbar to display its Human Information dialog (described earlier). This allows the user to change the basic attributes of the current human (Name, Position and body description). Press OK to update the current human.

> **NOTE:** If the selected human is from a previous case, the database keys are not editable. *(updated: the body-description controls are also disabled if the human already has event output tracks.)*

1. If desired, edit the current human's properties. Humans are edited on a per-segment basis. Click on any of the human's segment CG spheres to display a menu containing the human data categories (see the Human CG Dialog reference page):
2. **Inertias** — Displays and allows editing of the inertial properties for the selected segment.
3. **Contact Ellipsoids** — Displays and allows editing of the contact ellipsoids for the selected segment.
4. **Joints** — Displays and allows editing of the joint properties for the selected segment.

5. **Injury Tolerance** — Displays and allows editing of the injury tolerance properties for the current human.

Each of these data categories is described in detail in the following chapter, Human Model Definition.

Chapter 9 — Human Model Definition

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 9, manual pages 9-1 through 9-16. Revised for the current HVE release.

Overview

The HVE Human Editor displays the current human in the Human Viewer (see Figure 9-1). The HVE Human model is a physical/mathematical model based on GEBOD [3.9, 3.10], a program developed at Wright-Patterson Air Force Base under Federal sponsorship. The model has 15 segments and 14 joints. A schematic diagram of the HVE Human Model is shown in Figure 9-2. The original purpose of GEBOD was to produce anthropomorphically correct models of human pilots for purposes of design of the cockpits of fighter aircraft, although GEBOD is now used by several occupant dynamics simulators used in the study of motor vehicle safety [3.15, 3.16].

The Human Model parameters for the HVE Human Model are selected by clicking on the individual segment CG spheres in the Human Viewer. Clicking on a segment displays an option list of parameter groups for that segment (see the Human CG Dialog reference page). The Human Model parameters are divided into the following groups:

- Inertial Properties for each segment
- Color for each segment
- Contact Ellipsoid Properties for each segment
- Injury Tolerances for the selected human
- Joint Properties for each segment

These categories, and their associated parameters, are described in the following sections.

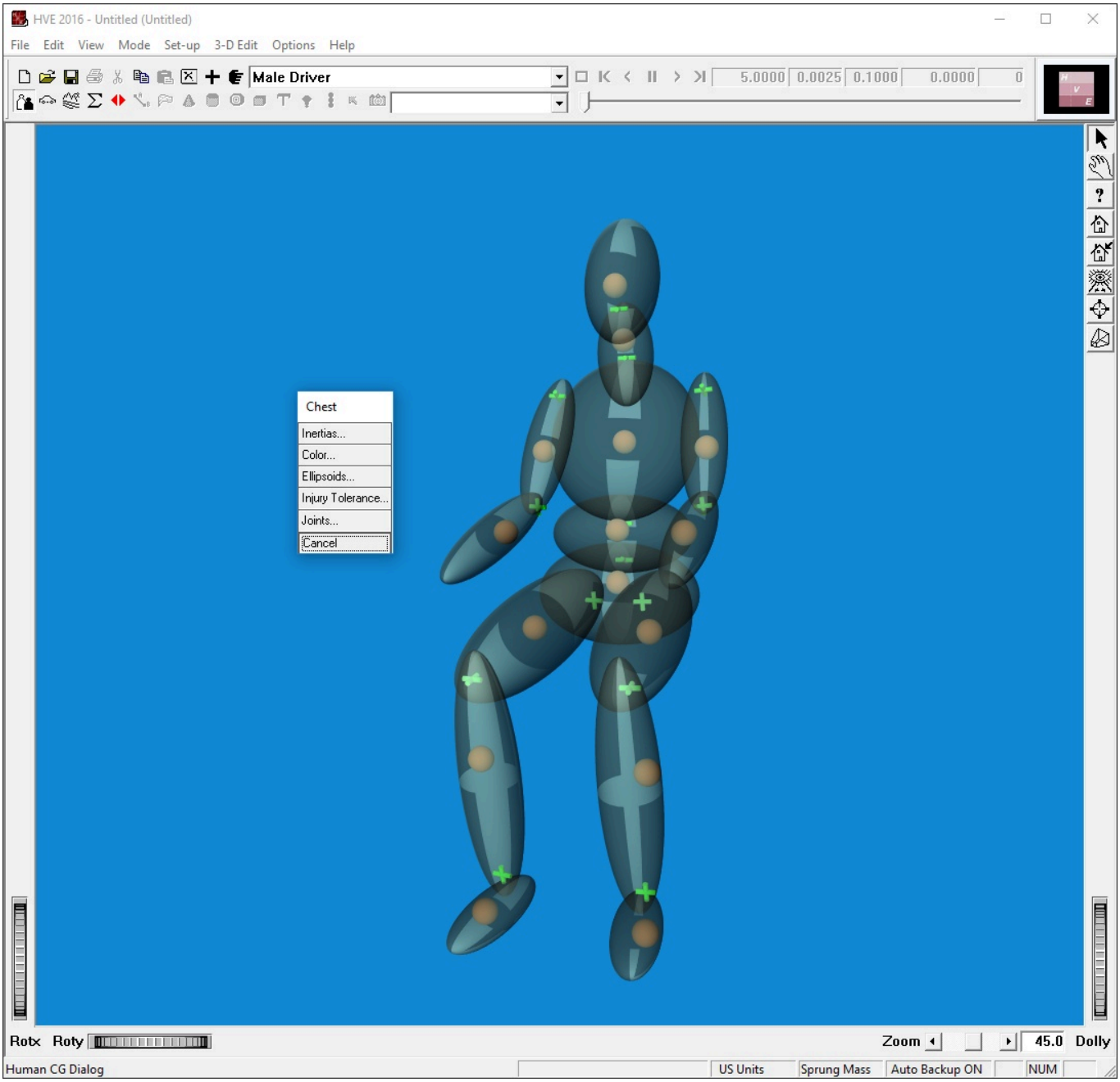


Figure 9-1: Human Editor, Viewer and Segment Data pop-up menu.

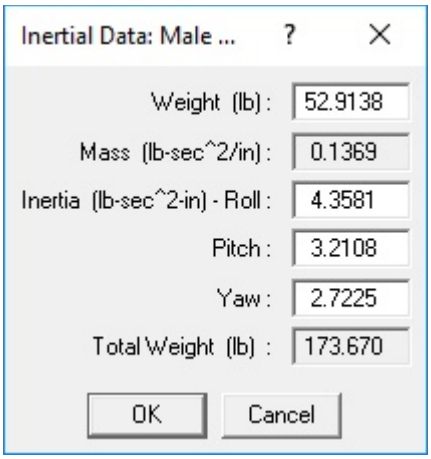


Figure 9-2: HVE Human Model with 15 segments and 14 joints. The HVE Human Model [3.9] is based on GEBOD [3.10].

The 15 segments are: Pelvis, Abdomen, Chest, Neck, Head, right and left Upper Legs, right and left Lower Legs, right and left Feet, right and left Upper Arms, and right and left Lower Arms

(segment list per the current segment popup dialog; see the Human CG Dialog reference page)

.

Inertial Properties

Inertial parameters for the HVE Human Model are defined for each of the 15 segments. To view and edit the current human's inertial parameters, click on the desired segment and choose Inertias from the cascade menu. The individual parameters are displayed in the Inertias dialog, as shown in Figure 9-3 and Table 9-1. See also the Inertial Data Dialog reference page.

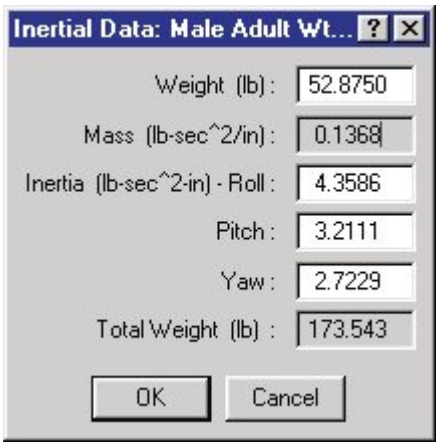


Figure 9-3: Human Segment Inertias dialog.

Table 9-1: Inertial Parameters for each human segment

Parameter	Units (US / SI)	Description
Weight	lb / N	Weight of selected segment (HVE calculates and stores the segment mass by dividing the entered weight by the current acceleration of gravity)
Rotational Inertias	lb-sec²-in / kg-m²	Rotational inertias about the segment i, j and k (roll, pitch and yaw) axes

(updated: the current dialog also displays the read-only computed segment Mass and the read-only Total Weight of the entire human — the sum of all 15 segment masses multiplied by the current acceleration of gravity.)

Segment Color

The color of each of the 15 segments is user-definable. To view and edit the current segment color, click on the desired segment and choose Color from the cascade menu. The Human Color dialog (see Figure 9-4 and Table 9-2) is displayed. The current color is displayed in the color patches on the left side of the dialog (the left patch is the current color, the right patch is the previous color). To edit the color, click on the color wheel at the desired color location. To edit the intensity (brightness), drag the intensity slider to the left (darken) or right (lighten). Click on the Copy To All Segments checkbox to apply the selected color to all segments.

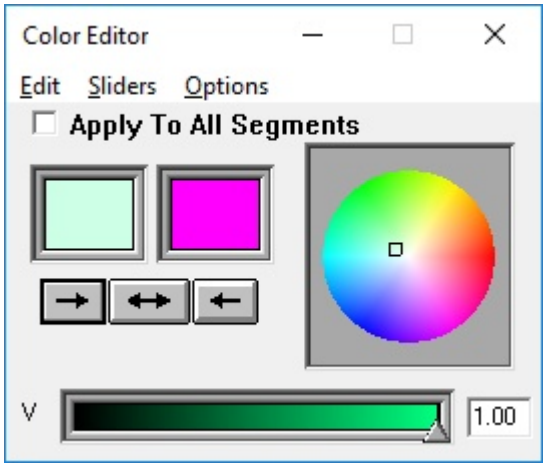


Figure 9-4: Human Color dialog.

Table 9-2: Color Parameters for each human segment

Parameter	Units (US / SI)	Description
RGB Color Values	—	Red, green and blue color values for the selected segment
Color Intensity	—	Brightness (0 = dark, 1 = bright)

Contact Ellipsoid Properties

Ellipsoid properties for the HVE Human Model are defined for each of the 15 segments. To view and edit the current human's ellipsoid parameters, click on the desired segment and choose Ellipsoids from the cascade menu. The individual parameters are displayed in the Contact Ellipsoids dialog, as shown in Figure 9-5 and Table 9-3. See also the Contact Ellipsoids Dialog reference page.

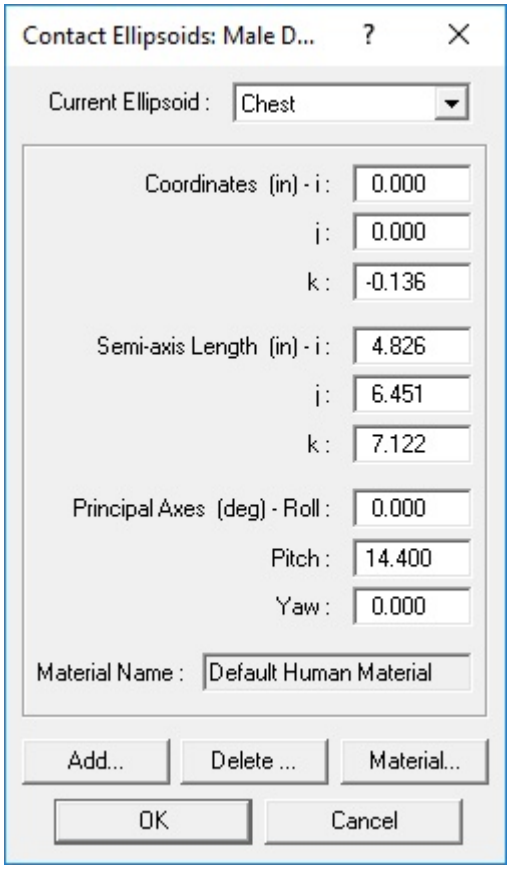


Figure 9-5: Human Contact Ellipsoids dialog.

Up to three individual ellipsoids may be supplied for each segment. To create a new ellipsoid, click on Add, and enter a new name. To view and edit the properties of an existing ellipsoid, click on the combo box arrow to display the list of available ellipsoids, then select the desired ellipsoid.

Table 9-3: Contact Ellipsoid Parameters for each human segment

Parameter	Units (US / SI)	Description
Ellipsoid Name	30-character text string	User-supplied names for each contact ellipsoid
Center Coordinates	in / cm	The coordinates of the center of the selected ellipsoid, defined relative to the segment principal axis system
Semi-axis Length	in / cm	The semi-axis i, j and k lengths of the selected ellipsoid
Principal Axes	deg	The angles of the i, j and k ellipsoid axis system relative to the segment axis system (see Figure 9-6)
Ellipsoid Material Name	—	Name for material attached to current ellipsoid

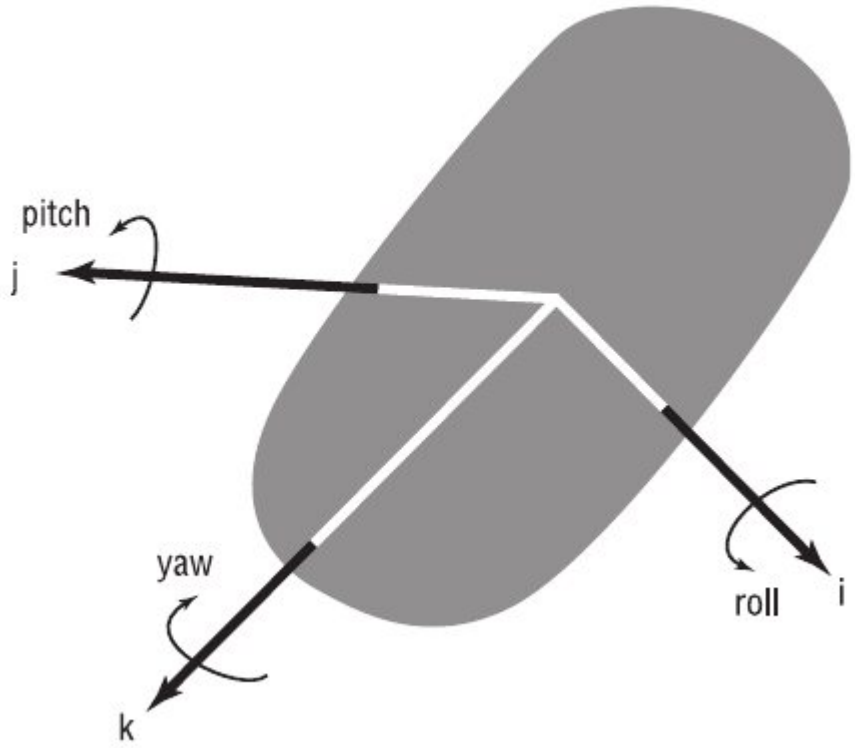


Figure 9-6: Human Segment Coordinate System.

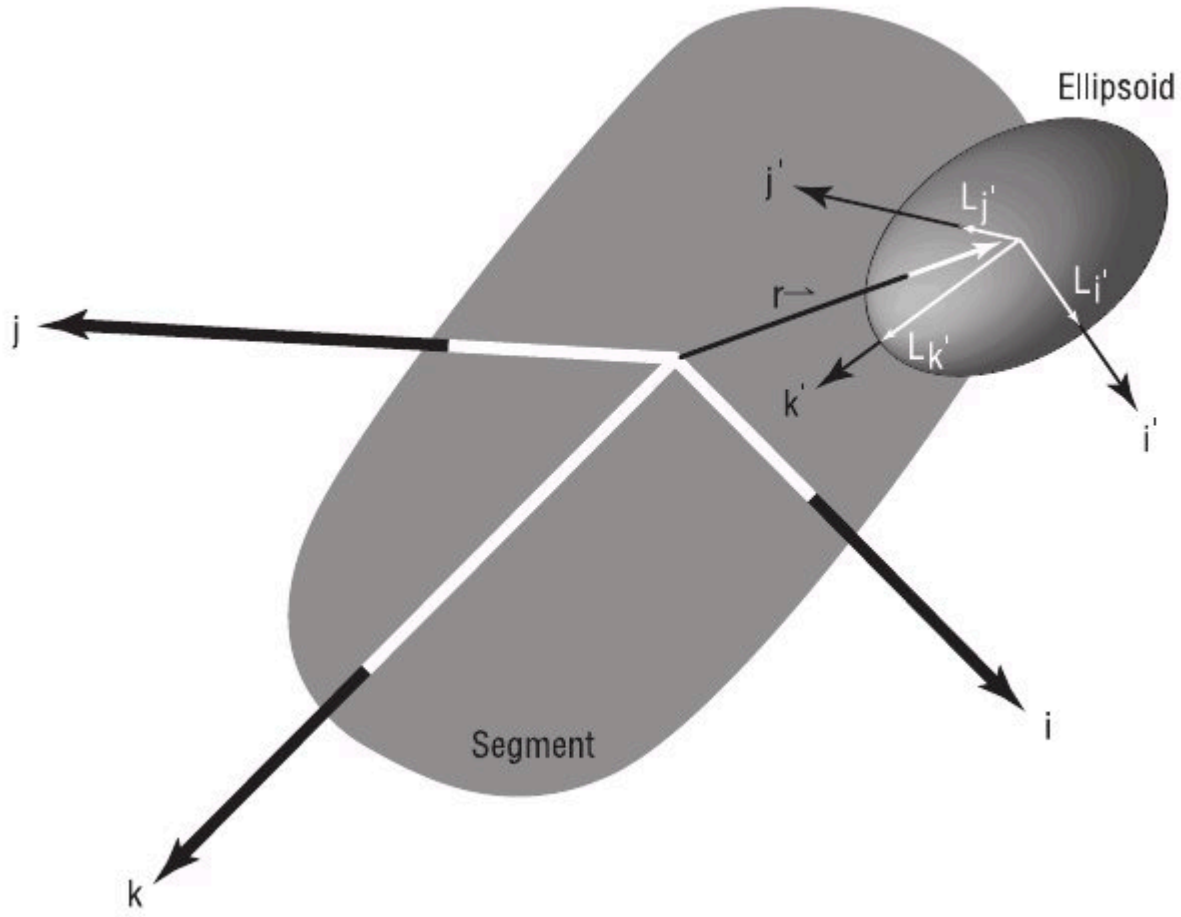


Figure 9-7: Human Contact Ellipsoids Coordinate System.

Contact Ellipsoid Material Properties

Each contact ellipsoid has material attributes. The material attributes are assigned by selecting a material file. These material files are user-editable. In addition, material files may be opened and saved for later application.

To view and edit the human ellipsoid material properties, select an ellipsoid using the Contact Ellipsoids dialog, then click on Material. The Human Materials dialog is displayed, as shown in Figure 9-8. The material attributes are defined in Table 9-4. See also the Human Material Properties Dialog reference page, which describes the dialog's force-deflection graph display.

Table 9-4: Human Contact Ellipsoid Material Parameters

Parameter	Units (US / SI)	Description
Material Name	—	User-editable material name
Constant	lb / N	Force required to initiate deflection
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient
Quadratic Stiffness	lb/in² / N/cm²	Quadratic material deformation coefficient
Cubic Stiffness	lb/in³ / N/cm³	Cubic material deformation coefficient
Damping Constant	lb-sec/in / N-sec/cm	Material velocity-dependent deformation constant
Friction Constant	—	Inter-segment friction coefficient
Maximum Force	lb / N	Force at which 3rd-order force-deflection relationship is abandoned
Maximum Deflection	in / cm	Deflection at which 3rd-order force-deflection relationship is abandoned
Unloading Slope	lb/in / N/cm	Linear unloading slope beginning at maximum deflection

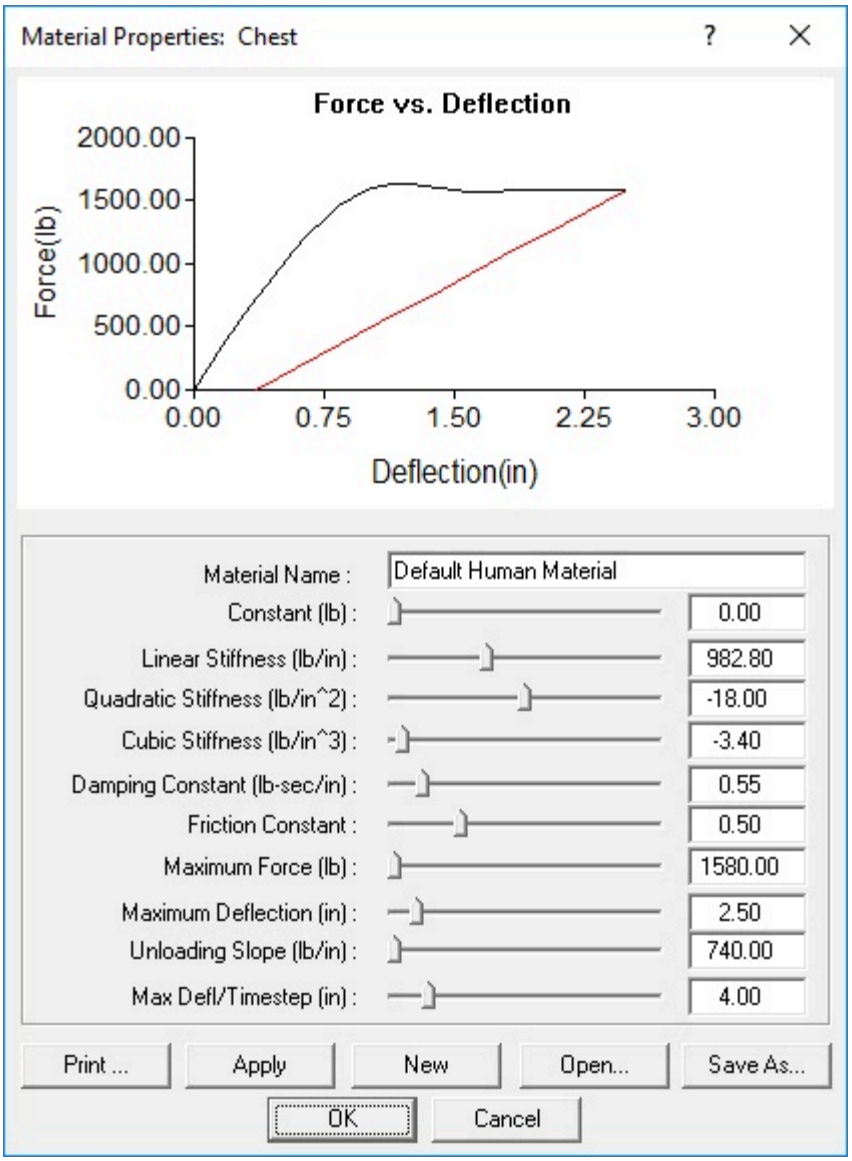


Figure 9-8: Human Materials Properties dialog displays the current material attributes for the selected ellipsoid. The dialog also allows the user to open and save human material files.

Injury Tolerances

Injury Tolerances may be defined for the current human. These injury tolerances are used during the current event as guidelines to estimate when specific injuries might occur.

> **NOTE:** Injury Predictions are an available Output Report.

To view and edit the current human's injury tolerances, click on any segment and choose Injury Tolerances from the cascade menu. The individual parameters are displayed in the Injury Tolerances dialog, as shown in Figure 9-9 and Table 9-5. See also the Injury Tolerance Data Dialog reference page.

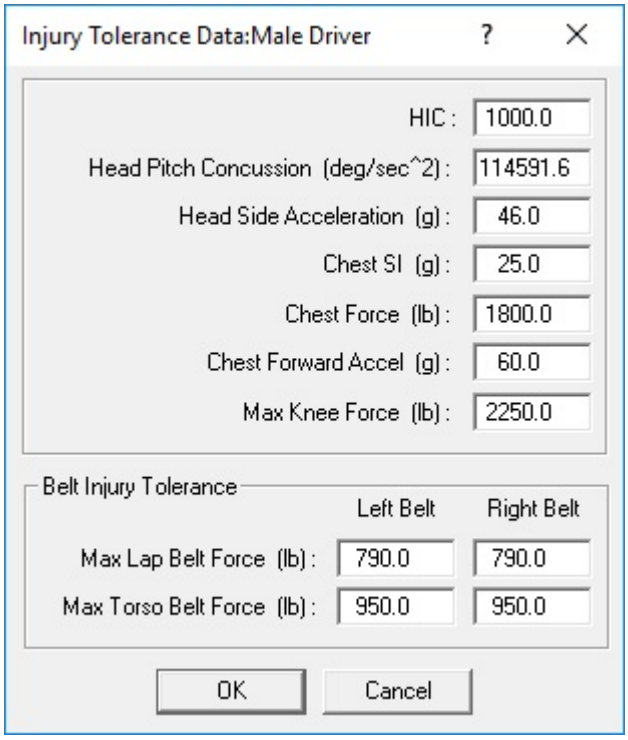


Figure 9-9: Human Injury Tolerance dialog.

Table 9-5: Injury Tolerance Parameters for the current human

Parameter	Units (US / SI)	Description
HIC	n/a	Head Injury Criterion, an empirically derived index [6.1] used to estimate the probability of a closed head injury
Head Pitch Concussion	deg/sec²	Angular acceleration of the head about the pitch axis of the neck
Head Side Acceleration	g	Linear acceleration of the head in the direction of the head-fixed j-axis
Chest SI	g	Chest Severity Index, an injury index indicating the maximum chest acceleration
Chest Force	lb / N	The tolerance for peak force against the chest in the direction of the chest-fixed i-axis
Chest Forward Acceleration	g	The tolerance for peak chest acceleration in the direction of the chest-fixed i-axis
Maximum Axial Femur Load (Max Knee Force)	lb / N	The tolerance for peak axial loading of the femur *(updated: labeled "Max Knee Force" in the current dialog)*
Maximum Lap Belt Force — Left Belt / Right Belt	lb / N	The tolerance for lap belt tension, above which injury is expected to occur to the abdomen
Maximum Torso Belt Force — Left Belt / Right Belt	lb / N	The tolerance for torso belt tension, above which injury is expected to occur to the torso

(updated: the original manual listed a single Maximum Lap Belt Force and a single Maximum Torso Belt Force. The current human model uses four separate belt injury tolerances — left lap, right lap, left torso and right torso — and the current dialog provides separate left-belt and right-belt entries for both the lap and torso webbing.)

Joint Parameters

Joint parameters for the HVE Human Model are defined for each of the 14 joints. To view and edit the current human's joint parameters, click on the desired segment and choose Joints from the cascade menu. The Joints dialog is displayed, as shown in Figure 9-10 and Table 9-6. The dialog contains an option list containing the names of each joint attached to the selected segment (a segment may have up to 4 attached joints), and the joint i, j, k coordinates relative to the selected segment coordinate system (see Figures 9-2 and 9-11). Click on the option list to display and select a different joint. See also the Joint Data Dialog reference page.

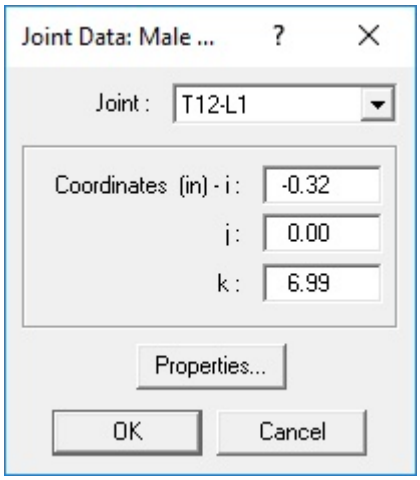


Figure 9-10: Human Joint Coordinates dialog.

Table 9-6: Joint Parameters for each human segment

Parameter	Units (US / SI)	Description
Joint Coordinates	in / cm	i, j and k coordinates of selected joint, defined relative to the segment principal axis system

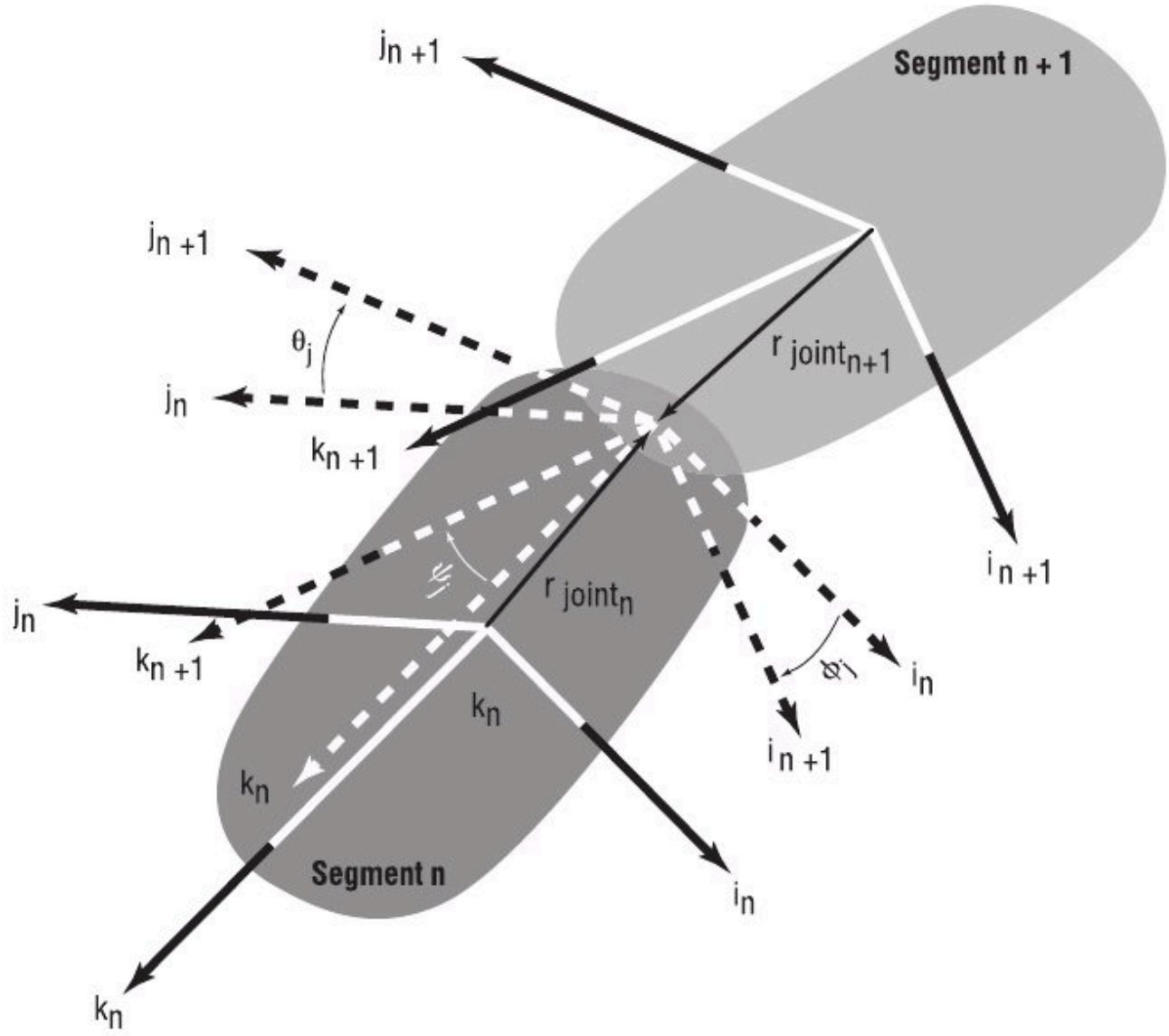


Figure 9-11: Human Joint Coordinate System.

Joint Properties

Each joint is defined by a number of physical properties. To view and edit these properties, first choose Joints to display the Joint Data dialog (see Figure 9-10), then click on Properties. The Joint Properties dialog is displayed, as shown in Figure 9-12. The individual properties are defined in Table 9-7. Except for the joint type, each property is entered separately for rotation about the joint's i, j and k axes. See also the Joint Properties Dialog reference page.

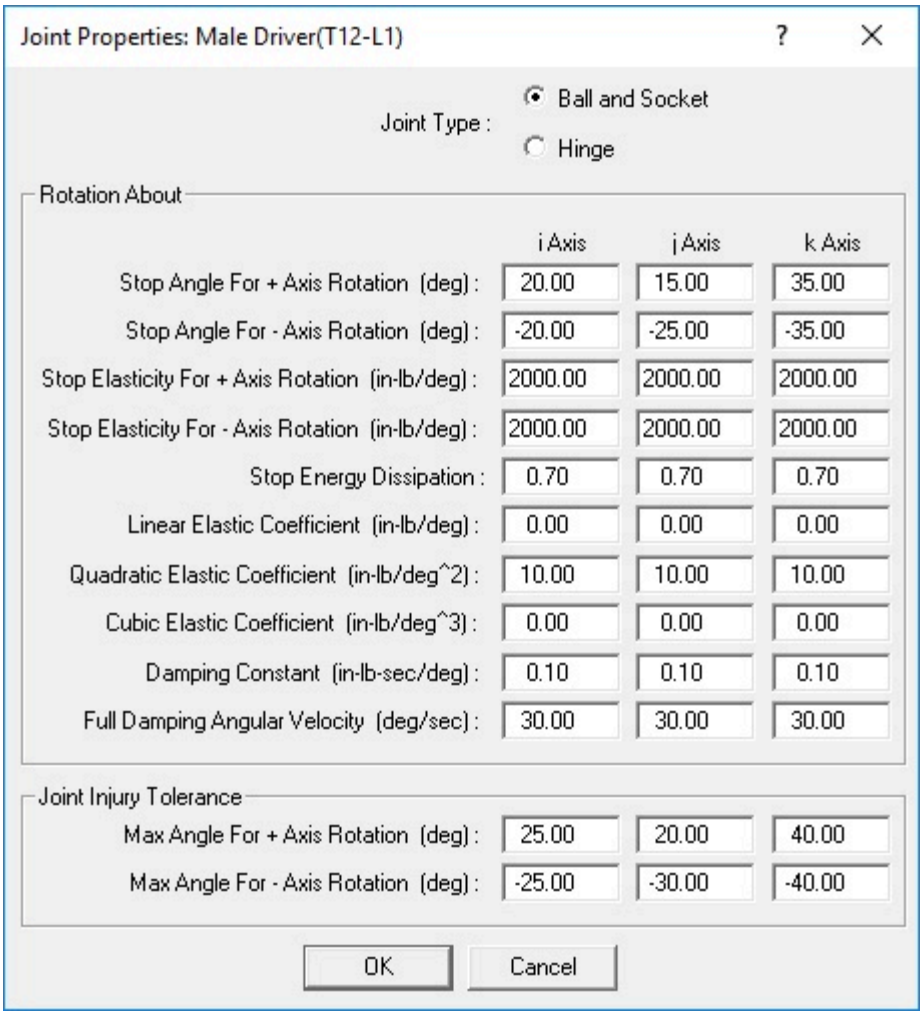


Figure 9-12: Human Joint Properties dialog.

Table 9-7: Joint Properties for each joint

Parameter	Units (US / SI)	Description
Joint Type	n/a	Either Ball-and-Socket or Hinge
Stop Angle, + Axis Rotation	deg	The angle at which the joint stop is applied for positive i, j and k segment rotations
Stop Angle, − Axis Rotation	deg	The angle at which the joint stop is applied for negative i, j and k segment rotations
Stop Elasticity, + Axis Rotation	in-lb/deg / N-m/deg	Joint stop linear elastic property for positive rotations about the i, j and k axes
Stop Elasticity, − Axis Rotation	in-lb/deg / N-m/deg	Joint stop linear elastic property for negative rotations about the i, j and k axes
Stop Energy Dissipation	—	Ratio of dissipated to total energy at the joint stop
Linear Elastic Constant	in-lb/deg / N-m/deg	Joint linear elastic property during normal range of motion for rotations about the i, j and k axes
Quadratic Elastic Constant	in-lb/deg² / N-m/deg²	Joint quadratic elastic property during normal range of motion for rotations about the i, j and k axes
Cubic Elastic Constant	in-lb/deg³ / N-m/deg³	Joint cubic elastic property during normal range of motion for rotations about the i, j and k axes
Damping Constant	in-lb-sec/deg / N-m-sec/deg	Joint damping property during normal range of motion for rotations about the i, j and k axes
Full Damping Angular Velocity	deg/sec	Angular velocity required to achieve full joint damping
Joint Injury Tolerance, Max + Rotation Angle	deg	The positive joint angle which, if exceeded, is predicted to cause injury to the joint
Joint Injury Tolerance, Max − Rotation Angle	deg	The negative joint angle which, if exceeded, is predicted to cause injury to the joint

Section Four: Vehicle Editor

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Section Four "Vehicle Editor". Revised for the current HVE release.

The purpose of the HVE Vehicle Editor is to provide physically correct 3-D vehicle models for use in dynamic reconstructions and simulations. The HVE Vehicle Model is a general purpose physical/mathematical model of a vehicle developed by EDC. Its generality of scope allows the same model to be used by any 2-D or 3-D reconstruction or simulation calculation method, and provides the capacity for human occupant and pedestrian simulation as well. The model applies to any type of vehicle having left-side and right-side wheels, including passenger cars, pickups, vans, sport-utility vehicles, trucks, trailers, dollies and barriers.

General Procedure

To use the Vehicle Editor, perform the following steps:

1. Select Vehicle Mode.
2. Select one or more vehicles from the Vehicle Database or from a previous HVE case.
3. If desired, modify the properties of the selected vehicle(s) to suit the needs of the current case (sprung mass, exterior, suspensions, brakes, tires, wheels, drivetrain, steering system and brake system).

Overview of Section Four

- **Chapter 10 — Creating & Editing Vehicles** describes the process of creating and editing vehicles: the Vehicle Editor dialog and Vehicle Viewer, adding new vehicles from the Vehicle Database, adding vehicles from previous cases, and selecting and editing vehicles.
- **Chapter 11 — Vehicle Model Definition** provides a detailed overview of the HVE Vehicle Model parameters. It is split here by major heading:
- **Part A — Sprung Mass**: Inertial Properties, Move CG, Color, Geometry File, Contact Surfaces, Belt Restraints, Airbag Restraints, Inter-vehicle Connections, Aerodynamic Drag and Body Torsion.
- **Part B — Exterior**: Vehicle Exterior Dimensions and Exterior Stiffness Coefficients (A, B, Kv).
- **Part C — Suspension**: Suspension Type, Roll Couple Distribution, Inter-tandem Load Transfer, Springs and Shocks, Axle Inertia, Jounce/Rebound, Spindle Axis (Alignment and Steering Stops), Camber & Half-track Tables, Anti-Pitch and Roll Steer.
- **Part D — Brakes, Tires and Wheels**: Brake Assembly Parameters (and Brake Designer access), Tire Properties (Physical Data, Friction Data, Fy vs Slip Angle, Fy vs Inclination Angle, Slip vs Rolloff Tables), Wheel Location and Wheel Image.
- **Part E — Drivetrain, Steering and Brake System**: Drivetrain (Engine, Transmission, Differential), Steering System Parameters, Brake System Parameters and the HVE Brake Designer overview.

Related Dialog References

Field-by-field dialog documentation is maintained separately and cross-linked from the chapters above:

- Vehicles (sprung mass, dimensions, stiffness)

- Suspension and Steering
- Brakes and Powertrain (incl. Brake Designer dialogs)
- Tires and Wheels

Chapter 10 — Creating & Editing Vehicles

The HVE Vehicle Editor is used for creating and editing vehicles for the current case. These vehicles may also be used in other cases.

This chapter describes how to use the Vehicle Editor to create and edit vehicles, beginning with a description of the Vehicle Editor's components.

> **NOTE:** Refer to the next chapter, Vehicle Model Definition, for a detailed description of each Vehicle Model parameter.

Vehicle Editor Components

To use the Vehicle Editor, choose Vehicle Mode using the mode selector. This puts HVE in Vehicle Mode and displays the Vehicle Editor's components:

- **Vehicle Editor Dialog** — The Vehicle Editor dialog is used for adding vehicles to the current case and to select the current vehicle.
- **Vehicle Viewer** — The Vehicle Viewer is used for visualizing the current vehicle and editing its parameters.

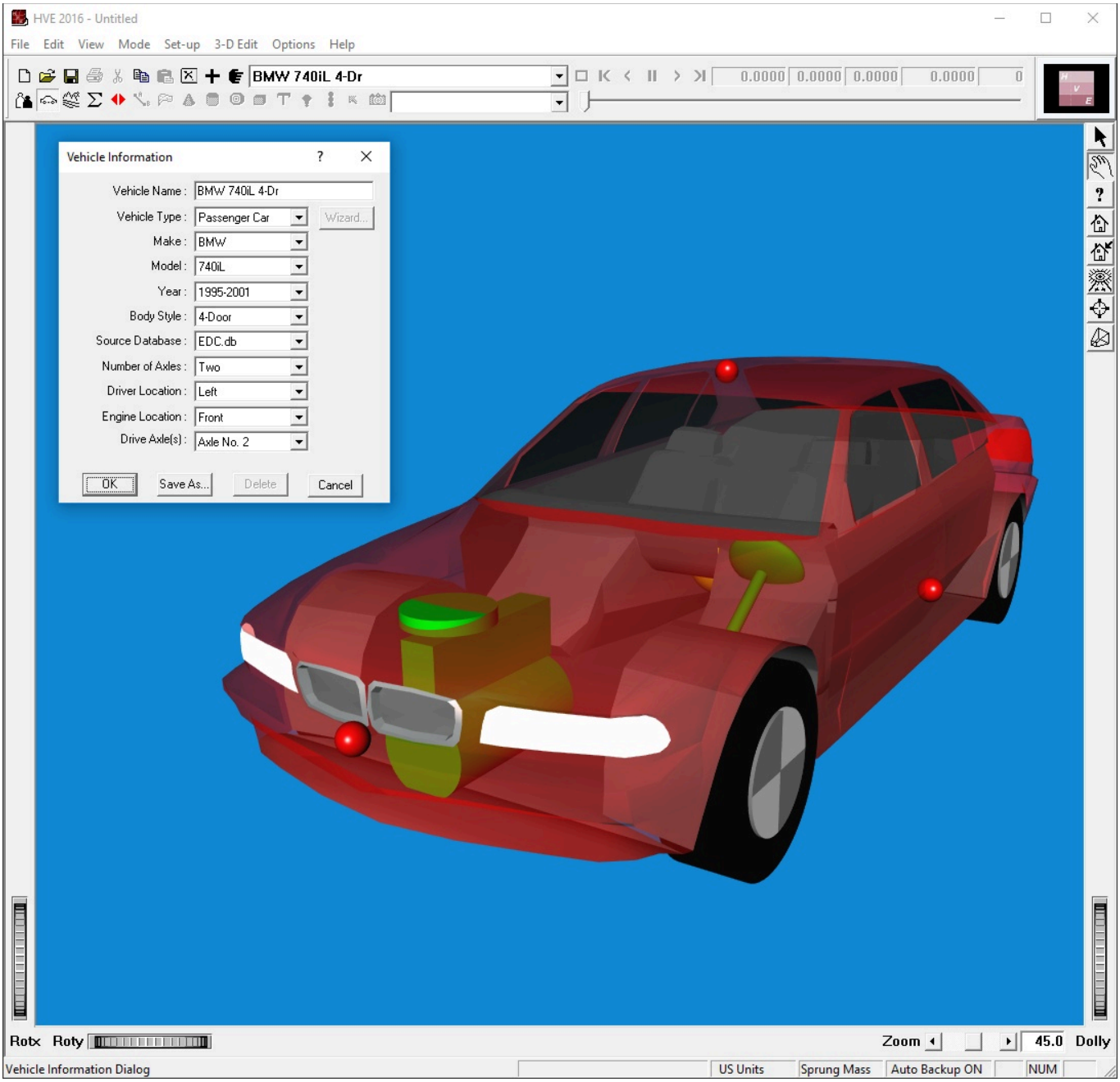


Figure 10-1: Vehicle Editor, Vehicle Information dialog and Viewer.

Vehicle Editor Dialog

The Vehicle Editor dialog is the heart of the HVE Vehicle Editor. It is used to manage (create and edit) all the vehicles in the current case. The Vehicle Editor dialog includes the following functions:

- **Add Vehicle** — Allows the user to add vehicles to the current case and provides two options: New and Previous. Choose *New* to add a vehicle from the Vehicle Database; choose *Previous* to add a vehicle from a previous HVE case. These options are further described below.
- **Active Vehicles List** — Displays a list of the names for all the vehicles in the current case. The name of the current vehicle is displayed in the minimized list. Click on the combo box to view the full list and select a different name to

change the current vehicle in the viewer. Click on the *Object Info* button on the toolbar to display its Vehicle Information dialog.

- **Delete Vehicle** — Removes the current vehicle from the case. Select *Edit, Delete* on the main menu, or click on the *Delete* button on the toolbar to delete the current vehicle.

> **NOTE:** Pressing Delete deletes the vehicle only from the current case, NOT the database!
>
> **NOTE:** The Delete button is not selectable if the current vehicle is used in an event.

Vehicle Viewer

The Vehicle Viewer displays the current vehicle. It contains the following components:

- **Current Vehicle** — Allows the user to visualize the current vehicle. Also displays the pickable vehicle components:
- **Center of Gravity** — Clicking on the CG allows the user to edit the sprung mass parameters: *Inertias, Move CG, Color, 3-D Geometry File, Contact Surfaces, Inter-vehicle Connections, Belt Restraints, Airbag Restraints, Aerodynamic Drag, Body Torsion*.
- **Wheels** — Clicking on a wheel allows the user to edit the suspension, tire and brake at the selected location. The x,y,z wheel location (relative to the CG) may also be edited.
- **Exterior** — Clicking on the exterior spheres allows the user to edit the exterior dimensions and structural stiffness.
- **Brake Master Cylinder** — Clicking on the brake master cylinder allows the user to edit the master cylinder or compressor properties.
- **Steering Wheel** — Clicking on the steering wheel allows the user to edit the steering system properties.
- **Engine** — Clicking on the engine allows the user to edit the drivetrain properties.
- **Viewer Controls** — Allow the user to *Rotate, Pan, Zoom* and set the current *Pick Mode* (see Chapter 2 for details on using the viewer controls).

> **NOTE:** Although every vehicle's properties are defined using the Vehicle Editor, vehicles are not positioned using the Vehicle Editor. The position of a vehicle in an event is an event-related issue, and is performed using the Event Editor. Think of it this way: At this point, we don't care if the Buick crashes, rolls over or simply slows to a stop. All we care about is that the Buick weighs 2970 pounds and has other properties of our choosing.

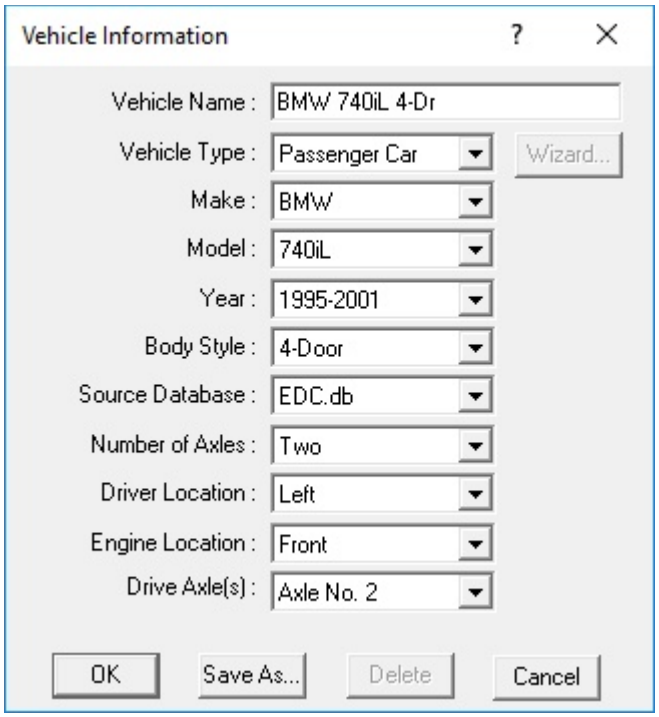


Figure 10-2: Vehicle Information dialog.

Adding New Vehicles

New vehicles are added to the current case using the Vehicle Editor dialog. To add a new vehicle, perform the following steps:

1. Click on *Add New Object*. The Vehicle Information dialog (see the reference page, Vehicle Information dialog) will be displayed, containing the following options:
2. **Name** — A user-editable field allowing the user to assign a name to the current vehicle. *(updated: if no name is entered, HVE generates a default name from the vehicle's year, make and model.)*
3. **Type** — Defines the basic vehicle type. The allowable selections are *Passenger Car, Pickup, Sport-Utility, Van, Truck, Trailer, Dolly, Fixed Barrier* and *Movable Barrier*. *(updated: only types actually present in the installed vehicle databases are listed.)*
4. **Make** — Vehicle manufacturer.
5. **Model** — Vehicle model.
6. **Year** — Model year.
7. **Body Style** — Vehicle body style (e.g., *2-Dr Hardtop, 4-Dr Sedan*).
8. Choose the *Type, Make, Model, Year* and *Body Style* for the current vehicle. Each selection filters the choices available in the lists below it.

The next option list displays the name of the database containing the vehicle. Three options available from EDC are *Generic.db, EDC.db* and *Tutorial.db*. When the user extends his/her database, the added vehicles are contained in the user's own database, called *User.db* (see Save-As below).

> **NOTE:** EDC.db is licensed. Unlicensed users will not be able to add vehicles from EDC.db.
>
> **NOTE:** The same vehicle Make, Model, Year and Body Style may be contained in more than one database.

The above parameters determine the basic vehicle. However, additional attributes are required to completely define its attributes. These additional attributes are:

- **Number of Axles** — Allowable selections are *None, One, Two* or *Three*. *(updated: for a Fixed Barrier, the only selection is None.)*
- **Driver Location** — The allowable selections are *None, Right, Center* or *Left*.
- **Engine Location** — Allowable selections are *None, Front, Mid* or *Rear*.
- **Drive Axle(s)** — Allowable selections depend on the number of axles: *None; Axle 1; Axle 2; Axle 3; Axles 1 & 2; Axles 2 & 3;* or *All Axles*.
- If desired, click on the *Number of Axles, Driver Location, Engine Location* and/or *Drive Axle(s)* to edit these attributes.
- Choose *OK* to create the selected vehicle and add it to the current case.

The selected vehicle will be added to the Active Vehicles List and become the current vehicle.

*(updated: the current Vehicle Information dialog also includes a **Wizard...** button that starts the Vehicle Wizard for building a custom vehicle. It is enabled for Passenger Car, Pickup, Sport-Utility, Van, Truck, Trailer and Dolly types having 1 to 3 axles; it is disabled for Fixed and Movable Barriers.)*

Vehicle Database

The vehicle *Type, Make, Model, Year* and *Body Style* are actually keys used to select the current vehicle from the Vehicle Database. These keys provide the basic vehicle definition, while the *Number of Axles, Driver Location, Engine Location* and *Drive Axle(s)* provide the exact configuration.

> **NOTE:** You can change the vehicle configuration using these keys!

The Vehicle Editor creates a copy of the selected vehicle for use in the current case. The properties of this vehicle may be edited. Changes made to the current vehicle do not affect the Vehicle Database.

Save-As New Vehicle

New vehicles may be added to the Vehicle Database, thus extending the vehicle library. To add a new vehicle to the database, perform the following steps:

1. First, make all required modifications of the current vehicle before saving it as a new vehicle.
2. Click the *Object Info* button on the toolbar for the current vehicle in the Active Vehicles List. The Vehicle Information dialog will be displayed, showing its current database keys.
3. Choose *Save-As*. The Vehicle Save-As dialog will be displayed, allowing the user to enter a new *Make, Model, Year* and *Body Style*. A *Version* and *3-D Geometry Filename* may also be entered (see the reference page, Save As New Vehicle dialog).

> **NOTE:** The 3-D geometry file must exist in the `HVE\supportFiles\images\vehicles` subdirectory. If no filename is supplied, or if the file is not found, the vehicle will be visualized as a simple vehicle body (defined according to Type).

1. If desired, enter a *Version Number*. This field may be used for licensing purposes and/or documentation.
2. Choose *OK* to add the current vehicle to the user's vehicle database.

Adding Vehicles from Previous Cases

The user may also include in the current case vehicles from other cases. This option is useful when another case includes a uniquely modified vehicle which is substantially similar to a vehicle required in the current case.

To choose a vehicle from a previous case for use in the current case, perform the following steps:

1. Select *Mode, Vehicle, Add* and choose *Previous*. The Previous Vehicle Selection dialog will be displayed (*Figure 10-3*). This dialog contains two list boxes: Active Cases and Active Vehicles.
2. Click on a case name in the Active Cases list box. The Active Vehicles List for that case will be displayed in the Active Vehicles list box.
3. Choose a name from the Active Vehicles List.
4. Press *OK* to add the selected vehicle to the current case.

> **NOTE:** You can click on the Object Info button for the vehicle's name in the Active Vehicles List to display its Vehicle Information dialog. However, if the selected vehicle is from a previous case, the database keys are not editable.

Selecting and Editing Vehicles

After a New or Previous vehicle is created, its name is added to the Active Vehicles List. The last vehicle added to the case becomes the current vehicle. The current vehicle is displayed in the Vehicle Viewer and may be edited.

To select and edit any vehicle in the Active Vehicles List, perform the following steps:

1. Click on the desired vehicle name in the Active Vehicles List. The selected vehicle becomes the current vehicle and will be displayed in the Vehicle Viewer.
2. If desired, click the *Object Info* button on the toolbar for the desired vehicle in the Active Vehicles List to display its Vehicle Information dialog (described earlier). This allows the user to change the basic attributes of the current vehicle (*Name, Type, Make, Model, Year* and *Body Style*). *Number of Axles, Driver Location, Engine Location* and *Drive Axle(s)* may also be edited. Press *OK* to update the current vehicle.

> **NOTE:** If the selected vehicle is from a previous case, the database keys are not editable.

If desired, edit the current vehicle's properties. A vehicle is edited by clicking on its components to bring up data dialogs. To edit the individual vehicle parameters:

- Click on the CG to edit the sprung mass parameters: *Inertias, Move CG, Color, 3-D Geometry File, Contact Surfaces, Inter-vehicle Connections, Belt Restraints, Airbag Restraints, Aerodynamic Drag, Body Torsion*.

- Click on a wheel to edit the *Suspension, Tire, Brake, Wheel x,y,z coordinates* (relative to the CG) and *Wheel Image* for the selected wheel location.
- Click on the front, right, back, left, top and bottom side exterior spheres to edit the exterior dimensions and structural stiffness.
- Click on the brake master cylinder to edit the master cylinder or compressor properties.
- Click on the steering wheel to edit the steering system properties.
- Click on the engine to edit the drivetrain properties.

Each of these data categories is described in detail in the following chapter, Vehicle Model Definition.

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 10, pages 10-1..10-8 — revised for the current HVE release, 2026-07-05.

Chapter 11 — Vehicle Model Definition (Part A: Sprung Mass)

The HVE Vehicle Editor displays the current vehicle in the Vehicle Viewer. The HVE Vehicle Model is a general purpose physical/mathematical model of a vehicle developed by EDC. The HVE Vehicle Model applies to any type of vehicle having left-side and right-side wheels, including passenger cars, pickups, vans, sport-utility vehicles, trucks, trailers, dollies and barriers.

The Vehicle Editor allows editing of the parameters which define the HVE Vehicle Model. The general categories for these parameters are:

- **Sprung Mass** — Clicking on the CG allows the user to edit the sprung mass parameter groups: *Inertias, Move CG, Color, Geometry File, Contact Surfaces, Inter-vehicle Connections, Belt Restraints, Airbag Restraints, Aerodynamic Drag, Body Torsion* (this part).
- **Unsprung Masses** — Clicking on a wheel allows the user to edit the unsprung mass parameter groups at the selected wheel location: *Suspension, Tire, Brake, Wheel x,y,z Coordinates* (see Part C — Suspension and Part D — Brakes, Tires and Wheels).
- **Exterior** — Clicking on the exterior spheres allows the user to edit the exterior dimensions and structural stiffness (see Part B — Exterior).
- **Brake Master Cylinder** — Clicking on the brake master cylinder allows the user to edit the master cylinder or brake system compressor properties; also used for activating the ABS system and editing the ABS system properties (see Part E).
- **Steering Wheel** — Clicking on the steering wheel allows the user to edit the steering system properties (see Part E).
- **Drivetrain** — Clicking on the engine allows the user to edit the drivetrain properties: *Engine, Transmission, Differential* (see Part E).

These categories and their associated parameters are described in this chapter.



Figure 11-1: Vehicle Editor, Sprung Mass Data pop-up menu.

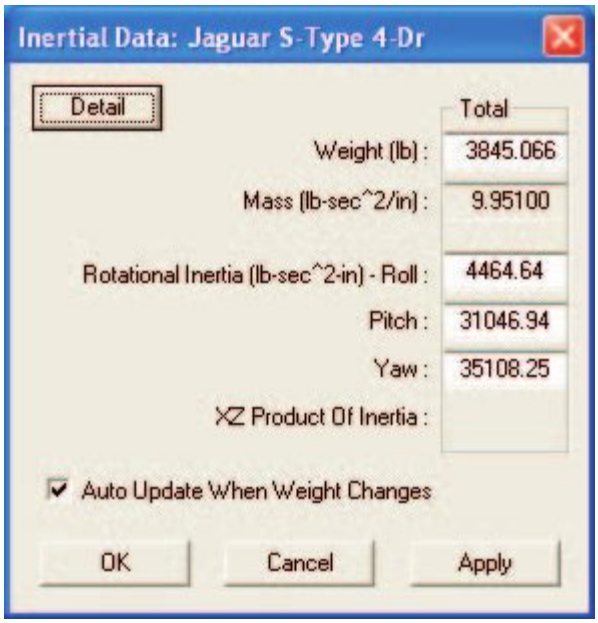
Inertial Properties

The inertial properties of the vehicle are displayed and edited using the Inertial Properties dialog. Two versions of the Inertias dialog are available: **Basic** and **Complex** *(updated: the legacy manual called the expanded version "Detailed"; the current dialog's toggle buttons are labeled Basic... and Complex...)*. The Basic version of the Inertias dialog displays only the sprung-mass properties in a narrowed dialog; the Complex version displays paired **Total** (i.e., sprung + unsprung) and **Sprung** columns side by side. *(updated: editing a value in one column automatically recalculates the corresponding value in the other column using the vehicle's unsprung masses and their locations. The last-used Basic/Complex state is remembered between sessions.)*

> **NOTE:** Typically, 2-D models use total vehicle inertial properties; 3-D models use the sprung mass properties. Why? The unsprung masses (i.e., wheels and axles) have their own independent equations of motion.

The Inertial Properties dialog also includes a feature that automatically updates the rotational inertias when the weight is changed.

> **NOTE:** The updated rotational inertias assume the mass distribution is unchanged by the weight change. The effect of this assumption is that the updated rotational inertia is directly proportional to the weight change.



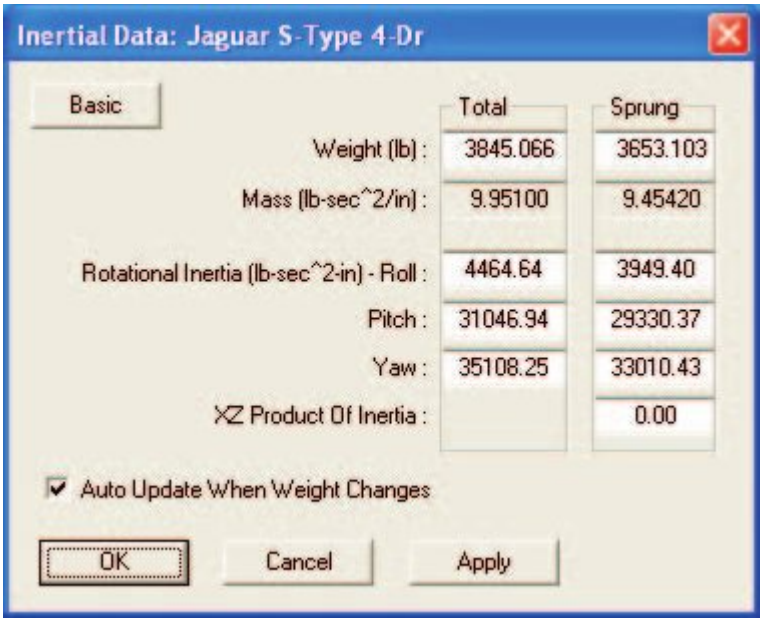


Figure 11-2: Vehicle Inertias dialog, Basic and Complex versions.

See also the reference page, Vehicle Inertia dialog.

The Inertial Properties dialog performs a number of calculations, depending on which values the user edits.

If the sprung weight, W_S, is edited, the total weight, W_T, is updated:

$$W_T = W_S + \sum_{n=1}^{N_{Axles}} W_U$$

where N_{Axles} = number of axles and W_U = unsprung weight of each wheel (plus axle for solid axle suspensions).

If the total weight, W_T, is edited, the sprung weight, W_S, is updated:

$$W_S = W_T - \sum_{n=1}^{N_{Axles}} W_U$$

If the sprung rotational inertia, I_{Sx}, I_{Sy} or I_{Sz} (roll, pitch or yaw, respectively), is edited, the total rotational inertia, I_{Tx}, I_{Ty} or I_{Tz}, is updated using the parallel axis theorem:

$$I_T = I_S + \sum_{n=1}^{N_{Axles}} (I_U + m_U r_U^2)$$

where

- I_U = rotational inertia of each wheel (and axle for solid axle suspensions) about its own x, y or z axis
- m_U = mass of each wheel (plus axle for solid axle suspensions)
- r_U = orthogonal distance from vehicle sprung mass CG to unsprung mass CG.

Similarly, if the total rotational inertia, I_{Tx}, I_{Ty} or I_{Tz} (roll, pitch or yaw, respectively), is edited, the sprung rotational inertia, I_{Sx}, I_{Sy} or I_{Sz}, is updated using the parallel axis theorem:

$$I_S = I_T - \sum_{n=1}^{N_{Axles}} (I_U + m_U r_U^2)$$

If the

Auto Update When Weight Changes

checkbox is checked, the rotational inertias are updated as follows:

a) If the sprung weight is edited, the updated sprung rotational inertias are:

```
I_S = I_S0 * (W_S / W_S0)
```

where

- I_{S0} = sprung rotational inertia before editing sprung weight
- W_{S0} = sprung weight before editing

The total rotational inertias, I_T, are then calculated using the parallel axis theorem, as shown above.

b) If the total weight is edited, the updated total rotational inertias are:

```
I_T = I_T0 * (W_T / W_T0)
```

The sprung rotational inertias, I_S, are then calculated using the parallel axis theorem, as shown above.

To display or edit the current vehicle's inertial properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Inertias* from the Sprung Mass option list. The Inertial Properties dialog will be displayed.

> **NOTE:** Either the Basic or the Complex version of the dialog will be displayed, depending on its state during your last visit to the Inertial Properties dialog.

1. View and/or edit the desired properties. The other properties will be updated and displayed as described above.
2. Press *OK* to accept the changes.

The Inertial Properties are described below.

- **Weight** — Vehicle total and sprung weight.

> **NOTE:** The Vehicle Editor actually divides your entered value by the current gravitational constant (see Environment Editor) and stores the resultant mass. That way, if you take the vehicle to the moon, it will behave correctly!

- **Mass (entered as weight)** — Vehicle total and sprung mass (weight/g), actually stored by the Vehicle Editor (see above note). This value is not user editable.
- **Roll, Pitch and Yaw Rotational Inertias** — Vehicle total and sprung mass rotational inertias.
- **x-z Product of Inertia** — Product of inertia about the y axis. This value represents the fact that the x-z plane is not a plane of symmetry for the vehicle sprung mass. *(updated: entered in the Sprung column only.)*

> **NOTE:** For all practical purposes, a vehicle is symmetrical in the x-y plane (i.e., about the z axis) and in the y-z plane (i.e., about the x axis). Therefore, these inertial products are zero.

(updated: in HVE-2D, the Roll, Pitch and XZ Product fields are read-only, the Total column and the Basic/Complex button are hidden, and only the Yaw inertia may be edited.)

Table 11-1: Inertial Properties

Parameter	Units (US / SI)	Description
Weight	lb / N	Vehicle total and sprung weight (HVE calculates and stores the mass by dividing the entered weight by the current acceleration of gravity)
Rotational Inertias	lb-sec²-in / kg-m²	Vehicle total and sprung rotational inertias about the vehicle x, y and z (roll, pitch and yaw) axes
x-z Product of Inertia	lb-sec²-in / kg-m²	Inertial product about the y axis

Move CG

The default CG (center of gravity) location of the vehicle sprung mass is determined from raw data (e.g., measured or table values). The user may choose to edit the CG location using the Move CG dialog. See also the reference page, Move CG dialog.

> **NOTE:** The default location of the CG is determined by the wheel locations (which are defined relative to the CG). Refer to the Wheel Location parameters for more information.
>
> **NOTE:** Changing wheel z coordinates or tire radii for any wheel location will change the calculated CG height.

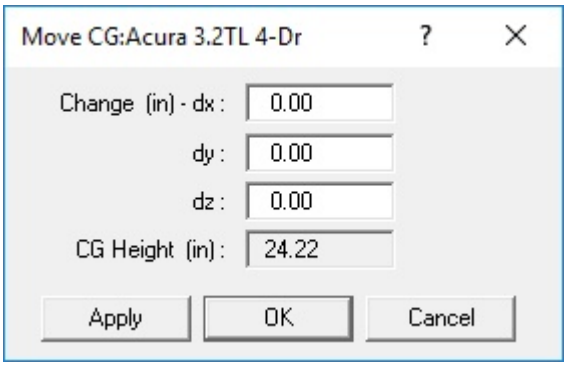


Figure 11-3: Move CG dialog.

To display or edit the current vehicle's CG location (relative to the default location), perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Move CG* from the Sprung Mass option list. The Move CG dialog will be displayed.
3. View and/or edit the desired properties.
4. Press *OK* to accept the changes.

The Move CG parameters are described below.

- **dx** — Moves the CG in the +x or −x direction, relative to the vehicle-fixed coordinate system (+x is *forward* in all coordinate systems).

- **dy** — Moves the CG in the +y or −y direction, relative to the vehicle-fixed coordinate system (+y is *to the right* in the SAE coordinate system).
- **dz** — Moves the CG in the +z or −z direction, relative to the vehicle-fixed coordinate system (+z is *down* in the SAE coordinate system).

Table 11-2: Move CG Parameters

Parameter	Units (US / SI)	Description
dx, dy, dz	in / cm	Adjustment of the default CG (center of gravity) location relative to the vehicle-fixed coordinate system
CG Height	in / cm	Approximate CG height, calculated from vehicle wheel positions and tire radius. Tire deflection is ignored.

Color

The color of the vehicle is obvious from looking at it. The color may be edited using the Color dialog.

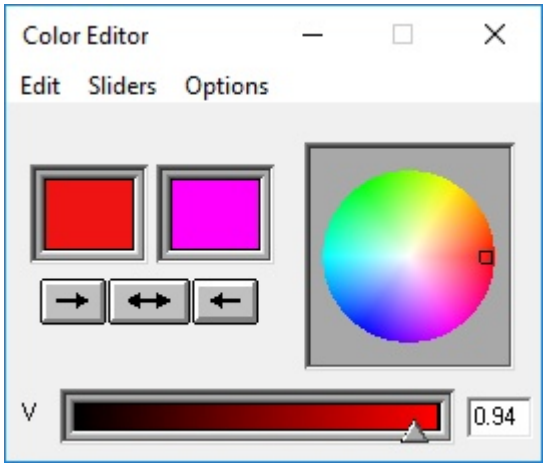


Figure 11-4: Color Editor dialog.

To display or edit the current vehicle's color properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Color* from the Sprung Mass option list. The Vehicle Color dialog will be displayed.
3. View and/or edit the color RGB value and intensity.
4. When finished, click on the Close button on the upper right corner of the dialog.

The Color Properties are described below.

- **RGB Color Wheel** — A small circle (*hot spot*) in the color wheel that determines the current color values. Move the hot spot to the desired color; towards the center of the color wheel creates neutral (gray) colors.
- **Color Intensity** — A slider ranging from 0 to 1 that determines the darkness or lightness of the color. Move the slider towards 0 to darken the vehicle.

> **NOTE:** Do not use a Color Intensity of 1.0 if you plan to produce a video! Saturated colors (especially red) look bad on video tape. See the Video Output section.

Geometry File

The current vehicle may be visualized using a simplified vehicle shape or an actual 3-D geometry file created by digitizing the vehicle. If a 3-D geometry file is available for the current vehicle, it may be selected using the Geometry File dialog.

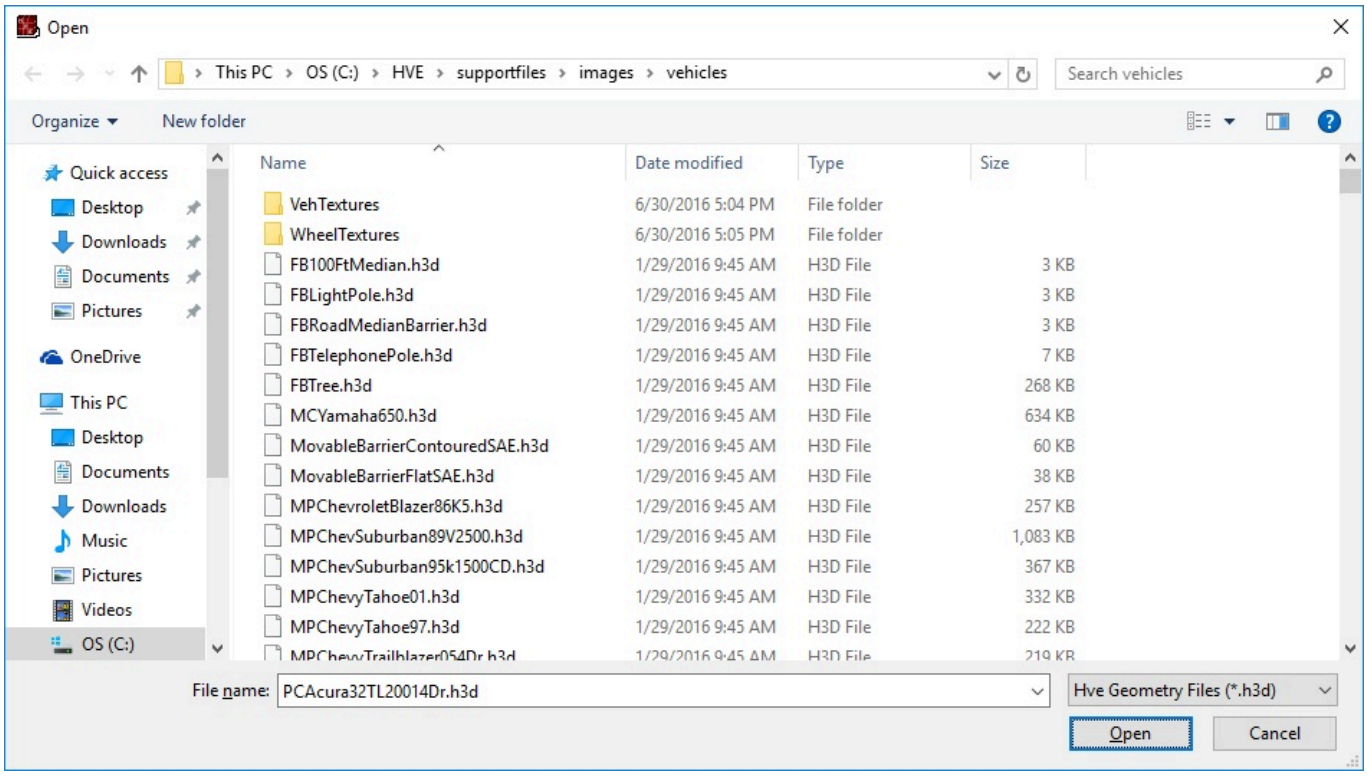


Figure 11-5: Geometry File dialog.

Mechanical Properties

The 3-D geometry file is not only used to visualize the vehicle, it is also used to define the mechanical/structural properties of the vehicle exterior. Each vertex includes a complete material specification. These material properties are:

- **x,y,z coordinates** — Vehicle-fixed coordinates for each vertex (a typical mesh includes approximately 2500 to 5000 vertices, not including the wheels).
- **Force Constant** — K_0, the force required to initiate vertex deformation.
- **Linear Stiffness** — F_1, the linear force-deflection constant for the vertex.
- **Quadratic Stiffness** — F_2, the 2nd-order force-deflection constant for the vertex.
- **Cubic Stiffness** — F_3, the 3rd-order force-deflection constant for the vertex.
- **Damping Constant** — The velocity-dependent force constant for the vertex.
- **Friction Constant** — The friction multiplier for the vertex.
- **Maximum Force** — Yield force for the vertex.

- **Maximum Deflection** — Yield at failure for the vertex.
- **Unloading Slope** — Linear force-deflection relationship during unloading.

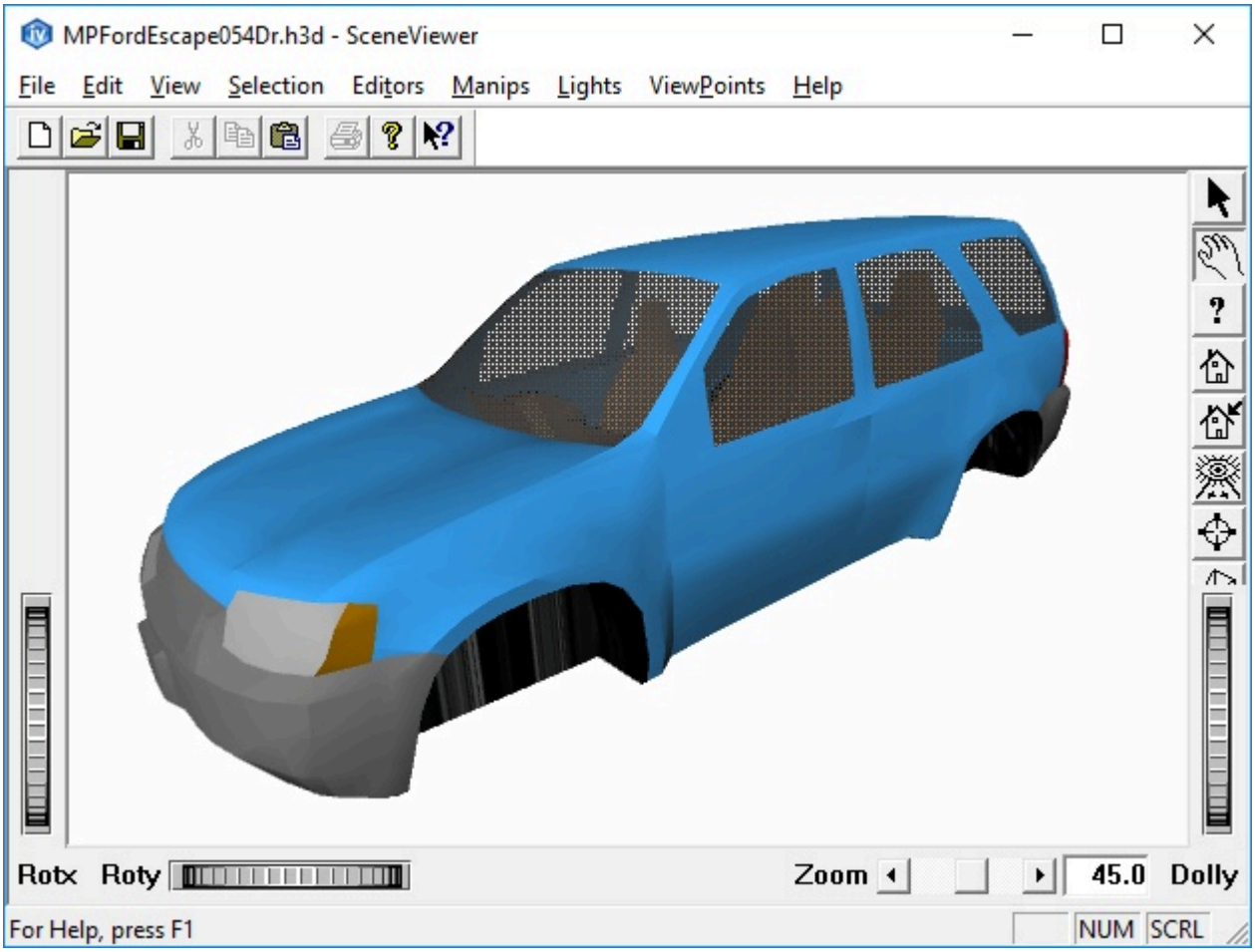


Figure 11-6: View of 3-D geometry file (mesh). The mesh provides visual properties, as well as mechanical properties that may be used by collision algorithms.

These material properties are available for collision algorithms to simulate collision forces and moments, as well as to visualize the resulting deformation to the vehicle exterior.

> **NOTE:** For more information regarding these parameters and how they are used, refer to the User's Manual for the particular simulation model being used.

Table 11-3: Vehicle 3-D Geometry Mechanical Parameters

Parameter	Units (US / SI)	Description
x,y,z coords	in / cm	Vehicle-fixed vertex coordinates
Material Name	—	Material name
Constant	lb / N	Force required to initiate deflection
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient
Quadratic Stiffness	lb/in² / N/cm²	Quadratic material deformation coefficient
Cubic Stiffness	lb/in³ / N/cm³	Cubic material deformation coefficient
Damping Constant	lb-sec/in / N-sec/m	Material velocity-dependent deformation constant
Friction Constant	—	Friction coefficient
Maximum Force	lb / N	Force at which 3rd-order force-deflection relationship is abandoned
Maximum Deflection	in / cm	Deflection at which 3rd-order force-deflection relationship is abandoned
Unloading Slope	lb/in / N/cm	Linear unloading slope beginning at max deflection

Geometry File Options

When Geometry File is selected from the Sprung Mass option list, HVE displays a cascade menu containing more options: *New, Open* and *Save-As*.

> **NOTE:** HVE requires a unique file format for geometry.
>
> **NOTE:** If you wish to use your own vehicle geometry file, you must place it in the `...images/vehicles` subdirectory for vehicle geometry files. That's the place HVE expects to find all vehicle 3-D geometry files.

New

The

New

option removes the existing 3-D geometry file (if any) and replaces it with the simple vehicle shape.

> **NOTE:** Choose this option if you accidentally select the wrong 3-D geometry file and you need to remove it.

To remove the existing 3-D geometry file, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Geometry File* from the Sprung Mass option list. The 3-D Geometry File Selection dialog will be displayed.
3. Choose *New*.

The 3-D geometry is removed from the current vehicle.

Open

The

Open

option displays the Vehicle Geometry File Selection dialog and allows the user to assign a digitized 3-D geometry file for the vehicle. To open a new 3-D geometry file and assign it to the current vehicle, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Geometry File* from the Sprung Mass option list. The 3-D Geometry File Selection dialog will be displayed.
3. Click on the *Format* option list to select the file format of the desired 3-D geometry file.
4. Select a 3-D geometry file from the list.
5. Press *OK*.

The selected 3-D geometry file is displayed on the current vehicle.

Save-As

The

Save-As

option displays the Vehicle Geometry File Selection dialog and allows the user to save the current 3-D geometry file using a new file format and file name. To save the current 3-D geometry file as a new file, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Geometry File* from the Sprung Mass option list. The 3-D Geometry File Selection dialog will be displayed.
3. Click on the *Format* option list to select the file format you wish to save the current 3-D geometry file as.
4. Enter a filename or select a filename from the list.
5. Press *OK*.

The selected 3-D geometry file is saved in the `images/vehicles` subdirectory according to the selected file format and filename.

Contact Surfaces

The current vehicle may include contact surfaces used for simulating interaction forces between the vehicle and its occupants or pedestrians.

Contact Surfaces are planes defined by 3 corners (the fourth corner is automatically assigned by assuming a parallelogram). Contact surfaces are created using the Contact Surfaces dialog.

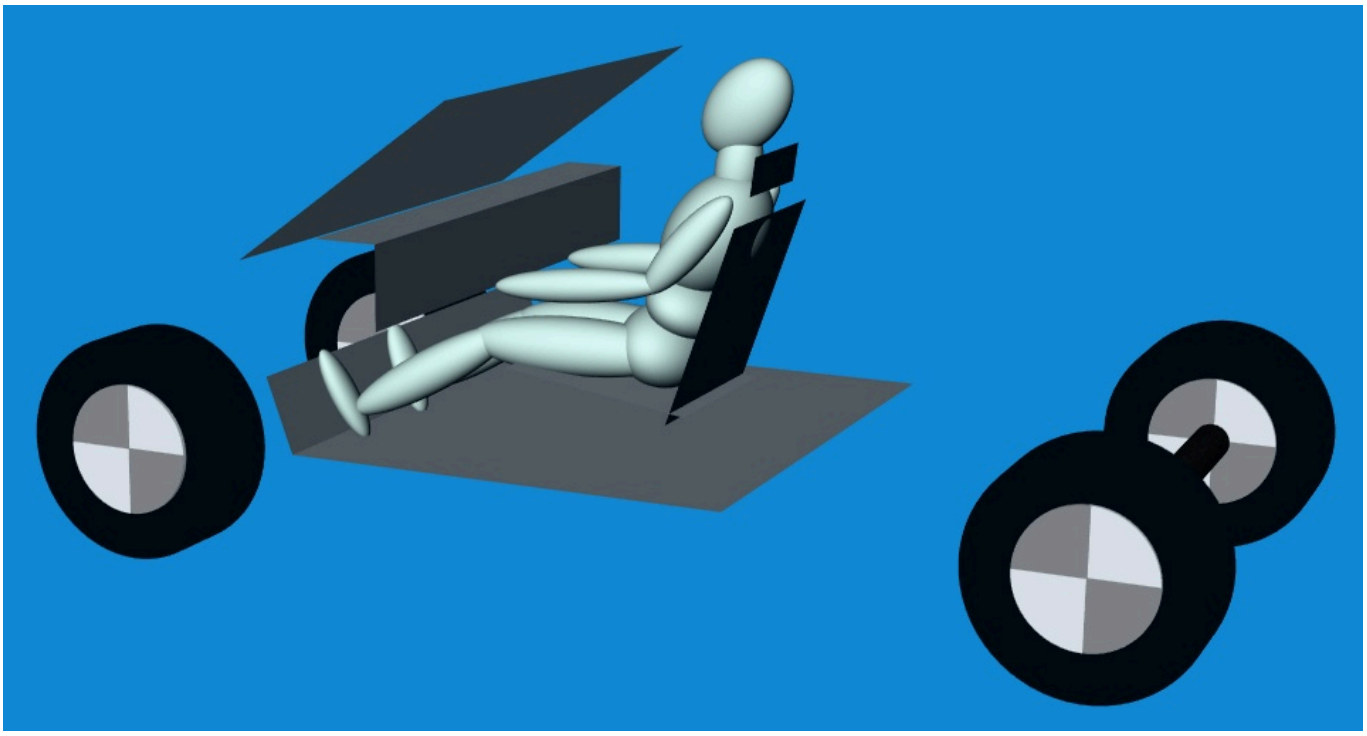


Figure 11-7: Vehicle contact surfaces — both interior and exterior contact surfaces are permissible.

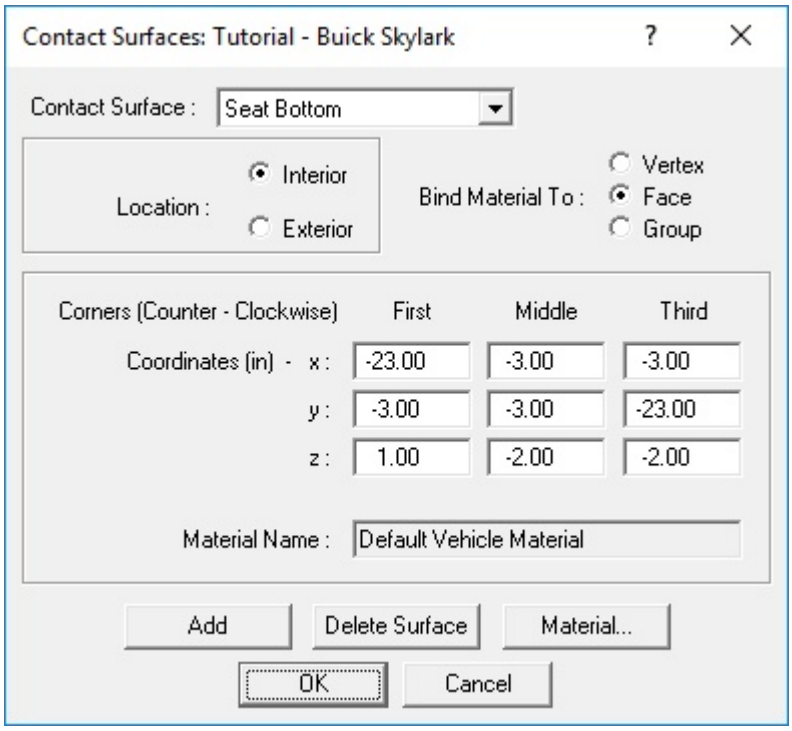


Figure 11-8: Contact Surfaces dialog.

To display or edit the current vehicle's contact surfaces properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Contacts* from the Sprung Mass option list. The Contact Surfaces dialog will be displayed.
3. To add a new contact surface, click on *Add* and enter a name for the contact surface, followed by *OK*.

> **NOTE:** If a name already exists in the text field, editing the existing name creates a new surface having the new name. The original name and surface still exist as well.

1. Click the appropriate *Bind Material To* option (*Vertex* or *Face*) to define whether the contact material attribute is attached to the vertex or face.
2. Click on the *Interior* or *Exterior* radio button to identify the contact as an interior or exterior surface.

> **NOTE:** The simulation may use this information to reduce the number of contacts for which it must calculate forces, based on whether the human is an occupant or pedestrian. For example, it is unnecessary to calculate the force between a pedestrian and seat cushion, since none will occur. (See Event Editor, Contacts Dialog, for more information.)

1. Enter the desired x,y,z (vehicle-fixed) coordinates for three corners of the contact surface. Be sure to choose three consecutive corners.

> **NOTE:** The "right hand rule" is used to determine the positive side of the contact surface; the positive side of the surface is light-colored, while the opposite side is black. Force on the contact surface is only calculated if it is struck on the positive side!
>
> **NOTE:** The fourth corner is created automatically assuming a parallelogram.

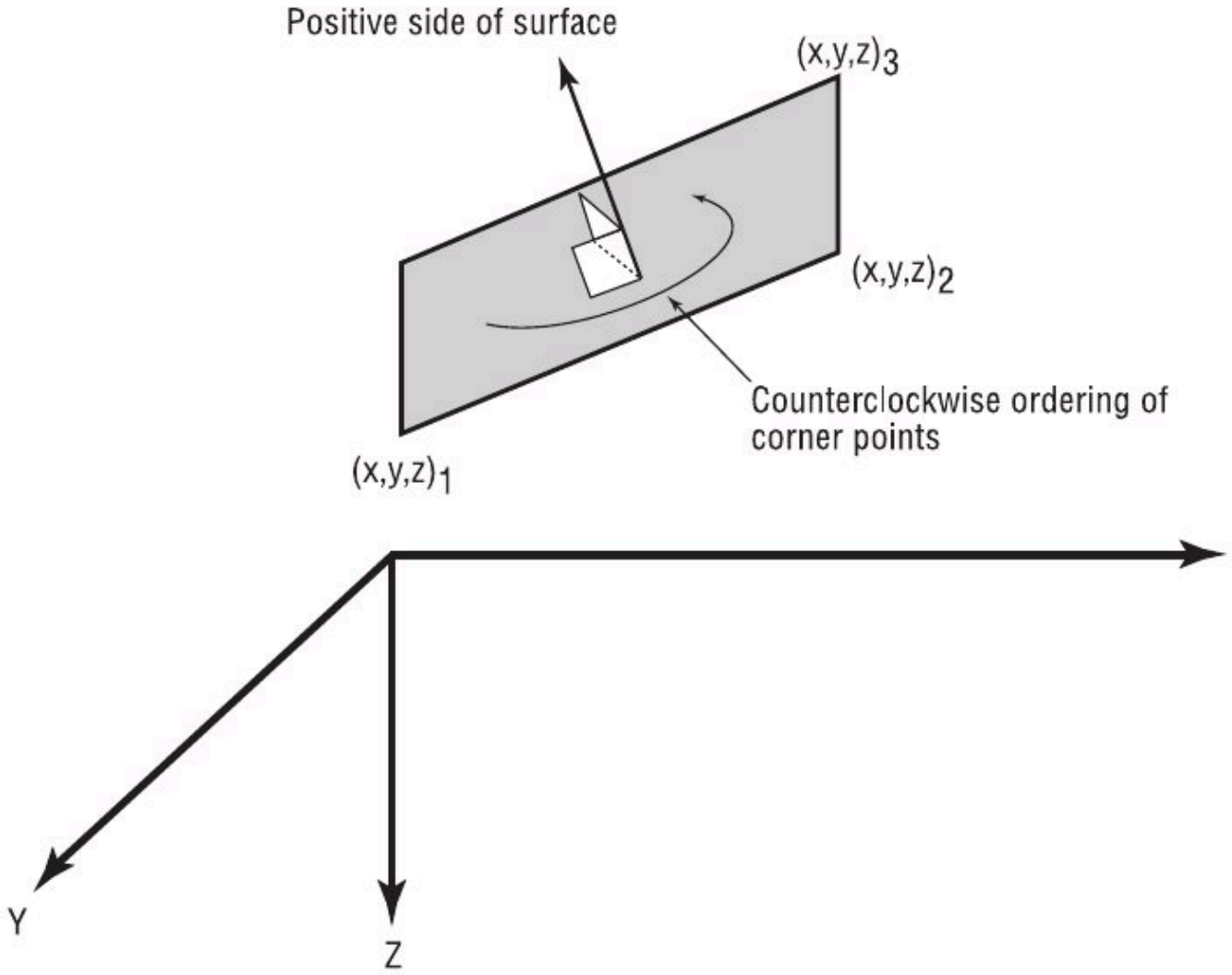


Figure 11-9: Convention for contact surfaces — three consecutive corners are selected; the right hand rule determines the positive side.

1. Choose *Material* to display the Contact Surface Properties dialog.
2. Click on the *Current Material* option button and choose the desired material from the materials option list. The properties of the selected material will be displayed.
3. If desired, edit the default material properties for the current surface.

> **NOTE:** Any editing applies only to the current contact surface and does not affect the Materials Database.

1. Press *OK* to accept the current material properties.
2. Press *OK* again to update the vehicle's contact surfaces.

It is important to confirm the contact surfaces are correctly positioned and facing the right direction within the vehicle. Carefully view each surface to ensure its correct position and orientation.

> **NOTE:** If the contact surfaces are not displayed, click on Options (main menu bar) and choose Show Contacts.

To edit an existing contact surface, perform the following steps:

1. If necessary, click on the vehicle CG and choose *Contacts* to display the Contact Surface dialog.
2. Click on the *Contact Surface Name* combo box button to display the list of contact surfaces.
3. Click on the desired name to make it the current surface. Its coordinates will be displayed.
4. If desired, edit the coordinates.
5. If desired, click on *Material* to display and edit the material properties or select a different material.
6. Click *OK* to update the list of contact surfaces for the current vehicle.

> **NOTE:** No changes take effect until you press OK.

The Contact Surface parameters are described below.

- **Bind Material To** — Assigns an attribute, either *Vertex* or *Face* (polygon); the simulation uses this attribute to determine whether the material is associated with the vertex or polygonal area.
- **Location** — Assigns an attribute, either *Interior* or *Exterior*, available for use by the simulation model to distinguish between surfaces on the vehicle's interior and exterior.
- **Material Name** — Assigns a user-editable name to the material; not used in calculations.
- **x,y,z** — Assigns the vehicle-fixed x,y,z coordinates for the three corners of the contact surface (the fourth is assigned by assuming a parallelogram).
- **Linear Stiffness** — The first-order force vs deflection constant for the current contact surface.
- **Quadratic Stiffness** — The second-order force vs deflection constant for the current contact surface.
- **Cubic Stiffness** — The third-order force vs deflection constant for the current contact surface.
- **Damping Constant** — The velocity-dependent force contribution for the current contact surface.
- **Friction Constant** — Assigns friction characteristics to the surface; normally the friction between two surfaces is the arithmetic average of their individual friction coefficients.
- **Maximum Force** — The maximum force resisted by the contact surface; if exceeded, force remains constant until the Maximum Deformation is reached.
- **Maximum Deflection** — The maximum allowable material deformation.
- **Unloading Slope** — A value used to assign hysteresis. The unloading slope defines the force vs deflection relationship during unloading.

> **NOTE:** The unloading slope should be steeper than the loading slope!

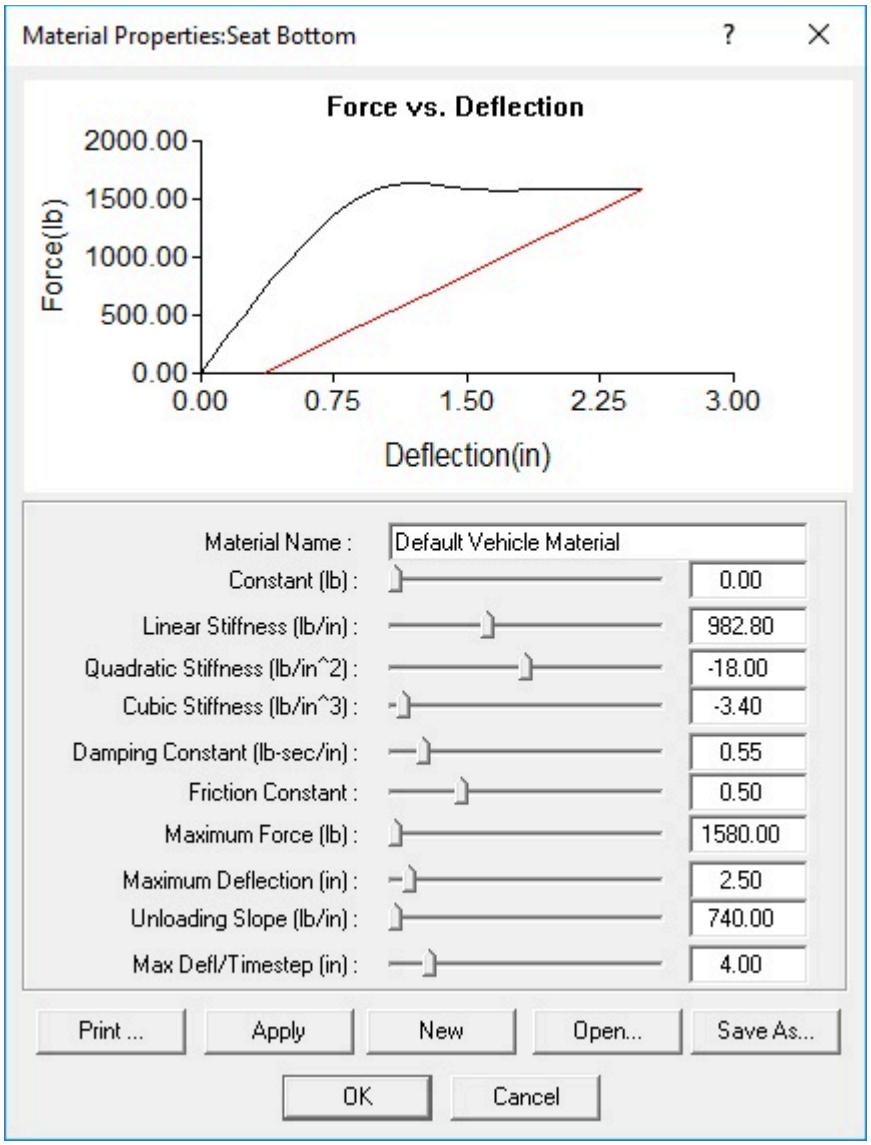


Figure 11-10: Contact Surfaces Materials dialog.

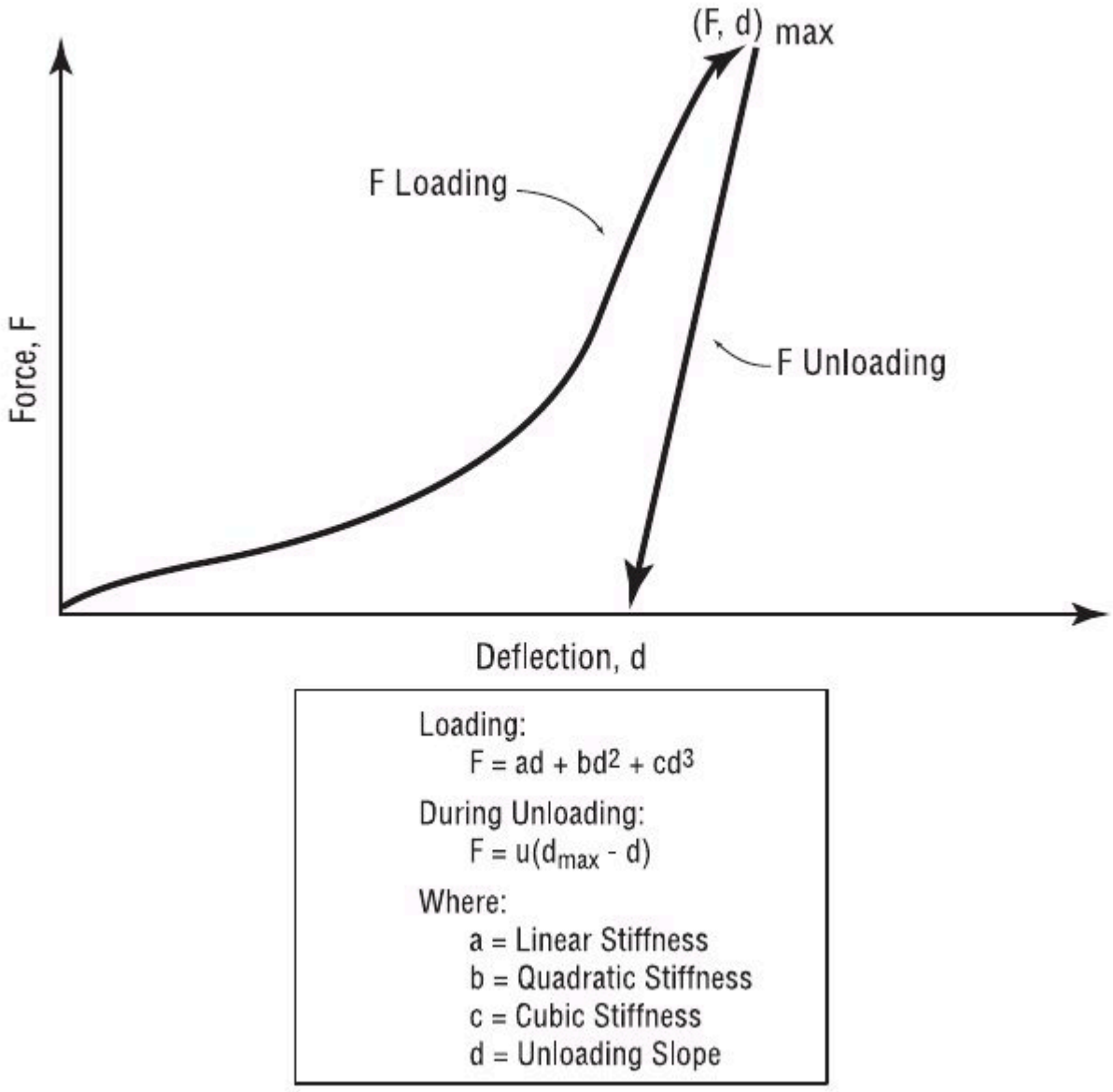


Figure 11-11: Typical force vs deflection curve for a contact surface. The loading slope (which occurs during penetration) may be defined by a 3rd-order polynomial: F = ad + bd² + cd³, where a = Linear Stiffness, b = Quadratic Stiffness, c = Cubic Stiffness. During unloading the slope is assumed linear: F = u(d_max − d), where u = Unloading Slope.

Table 11-4: Vehicle Contact Surface Parameters

Parameter	Units (US / SI)	Description
Bind To	—	Flag indicating whether material attributes are attached to each vertex or each face
Location	—	Flag indicating whether the surface is an interior or exterior surface
x,y,z coords	in / cm	Vehicle-fixed vertex coordinates for each of 3 corners
Material Name	—	Material name
Constant	lb / N	Force required to initiate deflection
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient
Quadratic Stiffness	lb/in^2 / N/cm^2	Quadratic material deformation coefficient
Cubic Stiffness	lb/in^3 / N/cm^3	Cubic material deformation coefficient
Damping Constant	lb-sec/in / N-sec/m	Material velocity-dependent deformation constant
Friction Constant	—	Friction coefficient
Maximum Force	lb / N	Force at which 3rd-order force-deflection relationship is abandoned
Maximum Deflection	in / cm	Deflection at which 3rd-order force-deflection relationship is abandoned
Unloading Slope	lb/in / N/cm	Linear unloading slope beginning at max deflection

Belt Restraints

The belt restraint systems of the vehicle are displayed and edited using the Belt Restraints dialog.

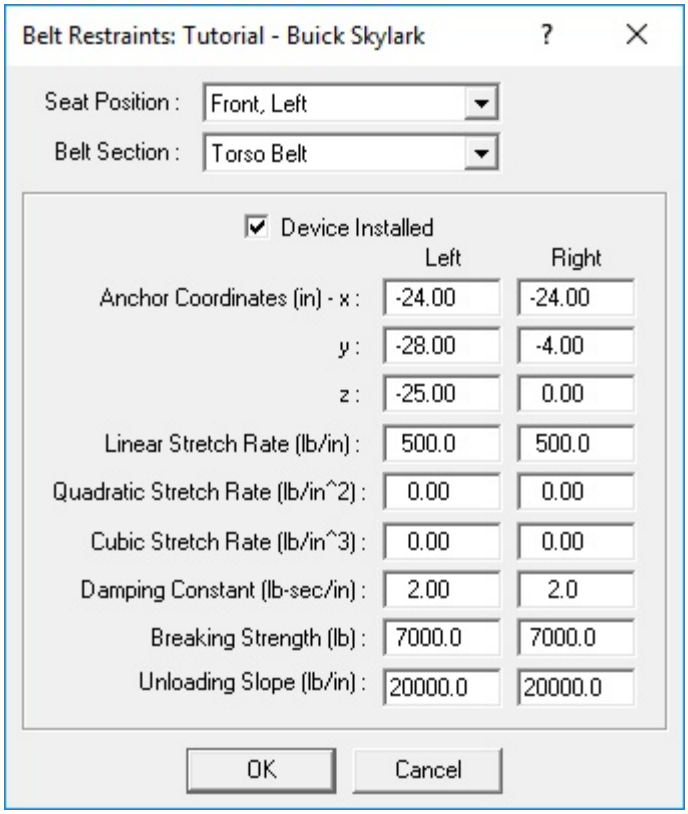


Figure 11-12: Belt Restraints dialog.

To display or edit the current vehicle's belt properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Belts* from the Sprung Mass option list. The Belt Restraints properties dialog will be displayed.
3. Click on the *Seat Position* option list and choose the desired seat position within the vehicle.
4. Click on the *Belt Section* option list and choose the desired belt section (*Torso* or *Lap*).
5. Click on the *Device Installed* checkbox to indicate a selected restraint system is installed at the selected seat location. The default belt properties for the right and left belt sections will become available for editing.
6. View and/or edit the desired properties for the current belt location and section.
7. Repeat the above steps for each seat location and belt section.
8. Press *OK* to accept the changes.

The Belt Restraint Properties are described below.

- **Seat Location** — A position index used to determine the location of the current belt restraint system.

> **NOTE:** When the human is created using the Human Editor, it is assigned a position (Right Front, Left Front, ..., Pedestrian). If the vehicle does not have a belt restraint system at a location that matches the occupant position, you will not be able to set up a restraint system during event mode!

- **Belt Section** — A position index used to determine the section (torso or lap) of the current belt restraint system.
- **Device Installed** — A check box to indicate the vehicle has a belt system at the current location and section.

The following belt restraint material properties may be assigned for the left and right belt sections:

- **x,y,z Anchor Points** — Assigns the vehicle-fixed x,y,z coordinates where the belt is attached to the vehicle.
- **Linear Stiffness** — The first-order force vs deflection constant for the belt section.
- **Quadratic Stiffness** — The second-order force vs deflection constant for the belt section.
- **Cubic Stiffness** — The third-order force vs deflection constant for the belt section.
- **Damping Constant** — The velocity-dependent contribution to stiffness for the current belt section.
- **Maximum Force** — The breaking strength of the belt. If exceeded, belt force becomes zero.
- **Unloading Slope** — A value used to assign hysteresis. The unloading slope defines the force vs deflection relationship during unloading.

> **NOTE:** The unloading slope should be steeper than the loading slope!

Table 11-5: Belt Restraint System Parameters

Parameter	Units (US / SI)	Description
Location	n/a	Location index (Right Front, Left Front, ..., Left Back) used to assign the location of the current belt section
Section	n/a	Torso or Lap belt section
Device Installed	BOOLEAN (True or False)	Switch that determines that a restraint is installed at the selected location and section
x,y,z	in / cm	Vehicle-fixed x,y,z coordinates for the anchor point of the belt
Linear Stiffness	lb/in / N/cm	First-order force vs deflection constant
Quadratic Stiffness	lb/in^2 / N/cm^2	Second-order force vs deflection constant
Cubic Stiffness	lb/in^3 / N/cm^3	Third-order force vs deflection constant
Damping Constant	lb-sec/in / N-sec/m	Velocity-dependent force contribution
Max Force	lb / N	Maximum allowable force
Unloading Slope	lb/in / N/cm	Force vs deflection constant during unloading

Airbag Restraints

The airbag restraint systems of the vehicle are displayed and edited using the Airbag Restraints dialog.

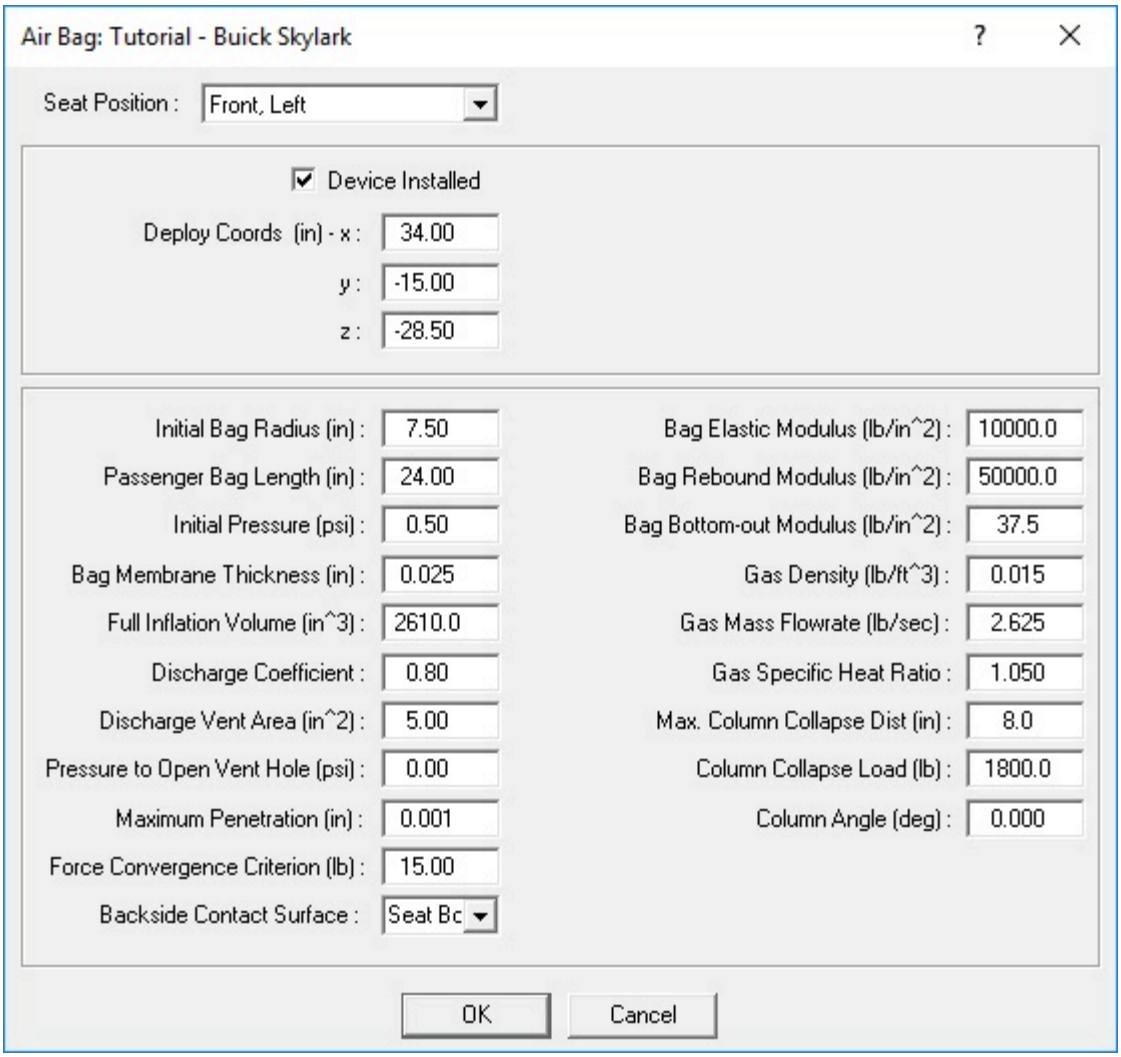


Figure 11-13: Air Bag dialog (Airbag Restraints).

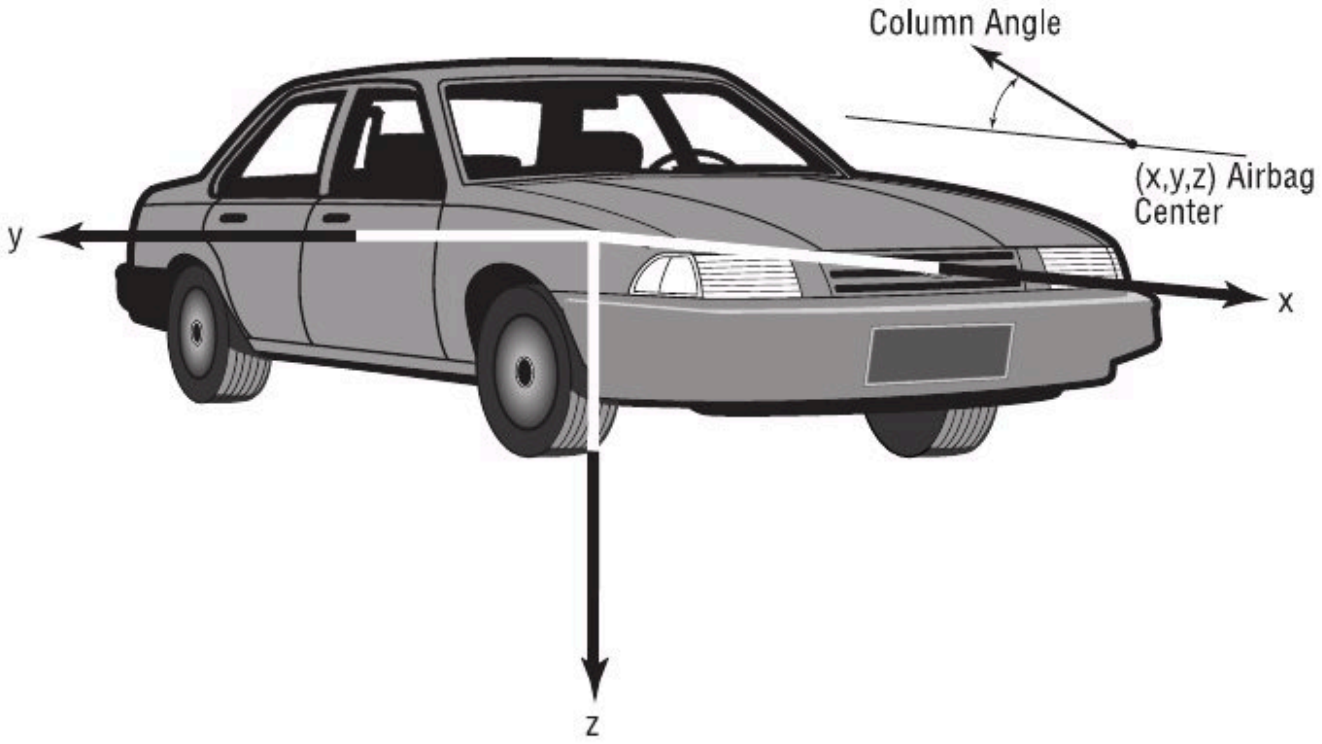


Figure 11-14: Airbag x,y,z deployment coordinates and deployment (column) angle.

To display or edit the current vehicle's airbag properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Airbags* from the Sprung Mass option list. The Airbag Restraints properties dialog will be displayed.
3. Click on the *Seat Position* option list and choose the desired seat position within the vehicle.
4. Click on the *Device Installed* checkbox to indicate the selected restraint system is installed at the selected seat location. The default airbag properties for the selected airbag location will become available for editing.
5. View and/or edit the desired properties for the selected airbag.
6. Repeat the above steps for each seat location.
7. Press *OK* to accept the changes.

The Airbag Restraint Properties are described below.

- **Seat Location** — A position index used to determine the location of the current airbag restraint system.

> **NOTE:** When the human is created using the Human Editor, he/she is assigned a position (Front Right, Front Center, Front Left, ..., Pedestrian). If this vehicle does not have an airbag restraint system at a location that matches the occupant position, you will not be able to set up a restraint system during event mode!

- **Device Installed** — A checkbox to indicate the vehicle has an airbag system at the current location.
- **x,y,z Deployment Coordinates** — Assigns the vehicle-fixed x,y,z coordinates where the airbag is installed and deployed.
- **Initial Bag Radius** — Assigns the initial radius for the airbag.

- **Passenger Airbag Length** — Assigns the length for non-driver-side airbags.

> **NOTE:** The driver's position is defined in the Vehicle Information dialog.

- **Initial Pressure** — Assigns the initial pressure for the airbag.
- **Bag Membrane Thickness** — Assigns the material thickness for the airbag.
- **Bag Full Inflation Volume** — Assigns the volume inside the airbag when fully inflated.
- **Vent Discharge Coefficient** — Assigns the thermodynamic discharge coefficient for the airbag vent hole.
- **Vent Discharge Area Coefficient** — Assigns the area of the airbag vent hole.
- **Pressure To Open Vent Hole** — Assigns the airbag pressure required to cause the vent hole to open.
- **Force Convergence** — Assigns the permissible force imbalance between the airbag and contact surfaces.
- **Maximum Penetration** — Assigns the penetration required before any force is computed.
- **Bag Elastic Modulus** — Assigns the elastic modulus for the airbag material during airbag filling.
- **Bag Rebound Modulus** — Assigns the elastic modulus for the airbag material during rebound.
- **Bag Bottom-out Modulus** — Assigns the elastic modulus for the airbag material after it has bottomed out.
- **Gas Density** — Assigns the gas density for the airbag filling agent.
- **Gas Mass Flow Rate** — Assigns the mass flow rate for the airbag filling rate.
- **Gas Specific Heat** — Assigns the specific heat (ratio) for the airbag filling agent.
- **Maximum Column Collapse Distance** — Assigns the maximum distance the steering column is allowed to collapse (applies to driver-side airbags only).
- **Column Collapse Load** — Assigns the axial load on the steering column required to begin collapsing the column (applies to driver-side airbags only).
- **Column Angle** — Assigns the steering column angle, about the vehicle's pitch axis (normally a negative angle).
- **Backside Contact Surface** — An option list containing all the vehicle interior contact surfaces. The selected surface is used to assign the material properties for the contact surface supporting the airbag.

> **NOTE:** An airbag must be supported on the backside. Otherwise, the bag would simply bounce away from the occupant.
>
> **NOTE:** This list of backside contact surfaces comes from the vehicle's interior contact surfaces.

Table 11-6: Airbag Restraint System Parameters

Parameter	Units (US / SI)	Description
Location	n/a	Location index (Right Front, Left Front, ..., Left Back) used to assign the location of the airbag
Device Installed	BOOLEAN (True or False)	Switch that determines that an airbag restraint is installed at the selected location
Deploy x,y,z	in / cm	Vehicle-fixed x,y,z coordinates for the airbag deployment location
Initial Bag Radius	in / cm	Radius of the (spherical) airbag at the start of filling
Passenger Bag Length	in / cm	Length of the (cylindrical) airbag used for passenger-side airbag systems
Initial Bag Pressure	psi / kPa	Airbag pressure at the start of filling
Membrane Thickness	in / cm	Thickness of the airbag material
Full Inflation Vol	in^3 / cm^3	Volume of the airbag at full inflation
Discharge Coef	—	Thermodynamic discharge coefficient of the airbag vent
Discharge Area	in^2 / cm^2	Area of the airbag vent
Vent Opening Pressure	psi / kPa	Pressure required to open the vent
Bag Penetration for Force	in / cm	Minimum deflection of an ellipsoid into the airbag required to produce a force
Force Convergence Criterion	lb / N	Allowable difference between force on the airbag and force on the human
Bag Elastic Modulus	lb/in^2 / N/cm^2	Elastic modulus of the airbag membrane material during force application
Bag Elastic Modulus on Rebound	lb/in^2 / N/cm^2	Elastic modulus of the airbag membrane material during unloading
Bag Elastic Modulus When Bottomed Out	lb/in^2 / N/cm^2	Elastic modulus of the airbag membrane material at maximum penetration
Gas Density	lb/ft^3 / kg/m^3	Density of the airbag filling agent
Max Column Collapse	in / cm	Maximum column stroke before it bottoms out
Column Collapse Load	lb / N	Force required to initiate column collapse
Column Angle	deg	Deployment (pitch) angle
Backside Contact	n/a	The name of the contact surface supporting the airbag (no force can exist between the human and airbag unless the airbag is supported on a contact surface that provides a reaction force)

Inter-vehicle Connections

The connections between tow vehicles and trailers are displayed and edited using the Inter-vehicle Connections dialog.

The available inter-vehicle connections are:

- Ball/Hitch
- Fifth Wheel/King Pin

- Pintle Hook/Eye (drawbars may be *Fixed* or *Hinged*)

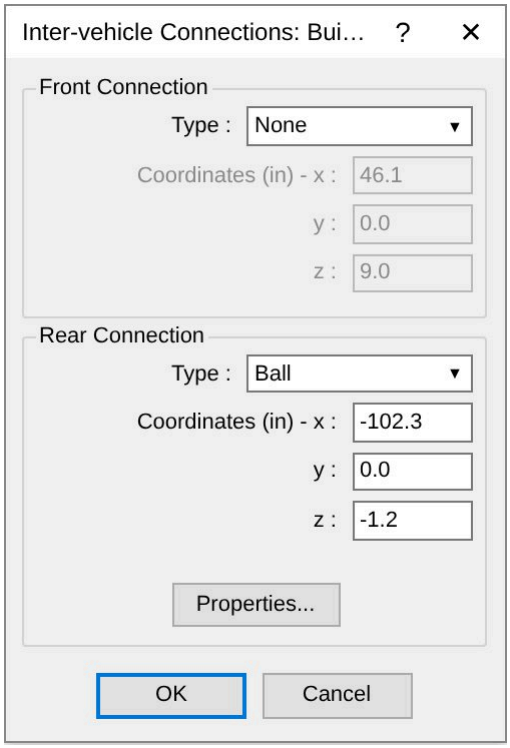


Figure 11-15: Inter-vehicle Connections dialog.

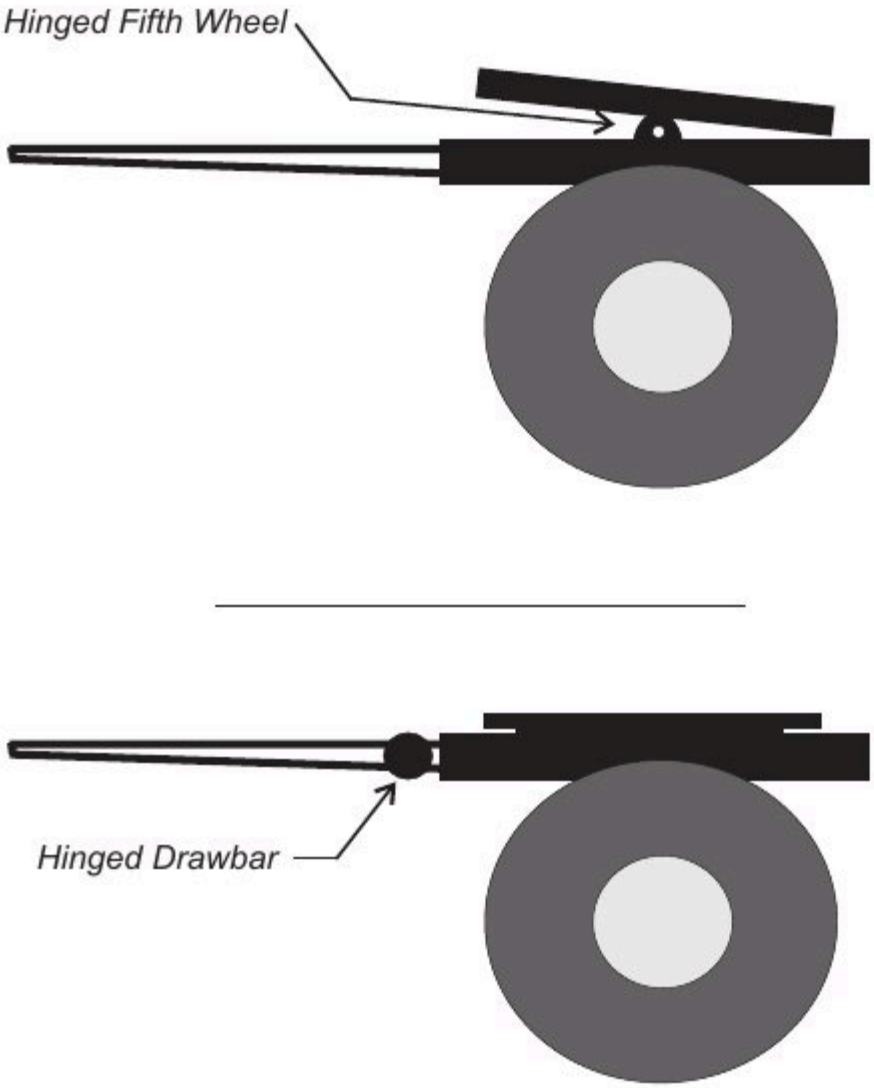


Figure 11-16: Converter (hinged fifth wheel) and Fixed (hinged drawbar) dollies.

To display or edit the current vehicle's connection properties, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Connections* from the Sprung Mass option list. The Inter-vehicle Connections Properties dialog will be displayed.
3. To put a connection on the front of a towed vehicle (e.g., a trailer), click on the *Front Connections* option button and select a connection type (*None, Hitch, King Pin, Fixed Drawbar* or *Hinged Drawbar*). Then enter the vehicle-fixed x,y,z coordinates for the connection.
4. To put a connection on the rear of a tow vehicle (e.g., a passenger car or truck tractor), click on the *Rear Connections* option button and select a connection type (*None, Ball, Fifth Wheel* or *Pintle Hook*). Then enter the vehicle-fixed x,y,z coordinates for the connection.
5. Press *Properties...* to enter the connection's radius, friction and other properties (see below). In HVE-2D there is no Properties... button; the radius, friction and maximum articulation angle are entered directly on the Inter-vehicle Connections dialog.

6. Press *OK* to accept the changes.

The Inter-vehicle Connection Properties are described below.

- **Front Connection Type** — The type of connection on the front of the current vehicle. The allowable types are *None, Hitch, King Pin, Fixed Drawbar* and *Hinged Drawbar* (the two drawbar types are pintle eyes).
- **Front Connection Coords** — The vehicle-fixed x,y,z coordinates of the front connection.
- **Rear Connection Type** — The type of connection on the rear of the current vehicle. The allowable types are *None, Ball, Fifth Wheel* and *Pintle Hook*.

> **NOTE:** During Event Mode, the simulation model normally checks to see the user has selected vehicles with compatible connections: a Hitch on the trailer must be connected to a Ball on the tow vehicle; a King Pin on the trailer must be connected to a Fifth Wheel on the tow vehicle; a Pintle Eye on the trailer must be connected to a Pintle Hook on the tow vehicle. Otherwise, the simulation model should issue a message: Incompatible Connections.

- **Rear Connection Coords** — The vehicle-fixed x,y,z coordinates of the rear connection.

> **NOTE:** The earth-fixed elevation of the connection above ground should be approximately equal on the rear connection of the tow vehicle and the front connection of the trailer. Otherwise, the trailer will be pitched up or down as necessary to make the connection. This can cause some physics models to compute an acceleration. For this reason, the physics model may terminate with an error message.

The following properties are entered in the Connection Properties dialog, displayed by pressing

Properties...

:

- **Friction** — The friction coefficient at the connection between the tow vehicle and trailer.
- **Friction Radius** — The moment arm at which the frictional torque acts at the connection between the tow vehicle and trailer.
- **Maximum Articulation Angle** — The maximum articulation angles (roll, pitch and yaw) allowable between the tow vehicle and trailer.
- **Linear and angular stiffness, damping, free play and breakaway force and torque** — Properties of the connection itself, also entered in the Connection Properties dialog.

Table 11-7: Inter-vehicle Connection Parameters

Parameter	Units (US / SI)	Description
Front Connection Type	n/a	Type of front connection (None, Hitch, King Pin or Pintle Eye)
Front Connection Coordinates	in / cm	Vehicle-fixed coordinates of the front connection
Rear Connection Type	n/a	Type of rear connection (None, Ball, Fifth Wheel or Pintle Hook)
Rear Connection Coordinates	in / cm	Vehicle-fixed coordinates of the rear connection
Connection Friction	n/a	Friction coefficient at the connection
Connection Friction Radius	in / cm	Frictional moment arm
Maximum Articulation Angle	deg	Maximum articulation angle between the tow vehicle and trailer

Aerodynamic Drag

The vehicle drag associated with aerodynamic pressure. The Aerodynamic Drag dialog allows the user to view or edit these properties. See also the reference page, Aerodynamic Drag dialog.

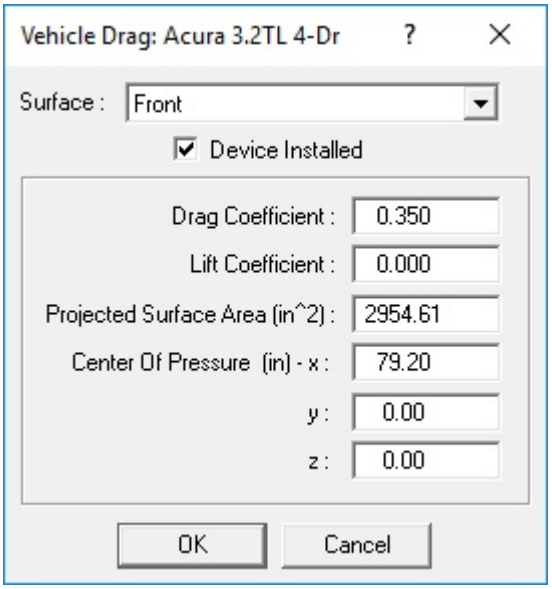


Figure 11-17: Vehicle Aerodynamic Drag dialog.

To display or edit the current vehicle's drag factors, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Aerodynamic Drag* from the Sprung Mass option list. The Aerodynamic Drag dialog will be displayed.
3. Select a surface (by default, the front surface has already been assigned those properties associated with velocity-dependent air drag; however, data for seven additional surfaces, including wings, air dams and spoilers, may also be assigned).
4. View and/or edit the desired properties.
5. Press *OK* to accept the changes.

The Aerodynamic Drag parameters are described below.

- **Surface Option List** — Assigns the selected surface. Available surface options are: *Front, Right, Back, Left, Top, Bottom, Front Device, Back Device*.
- **Device Installed** — Switch that activates the selected surface.
- **Drag Coefficient** — Assigns the properties resisting motion due to dynamic air pressure acting on the selected surface in the direction of the vehicle's total velocity vector. The force associated with this pressure is proportional to the velocity squared.
- **Lift Coefficient** — Assigns the properties resisting motion due to dynamic air pressure acting on the selected surface in the direction normal to the vehicle's total velocity vector.
- **Projected Surface Area** — Assigns the projected area parallel to the selected surface.
- **Center of Pressure** — Vehicle-fixed coordinates where the effective force from aerodynamic pressure is applied.

Table 11-8: Vehicle Aerodynamic Drag Parameters

Parameter	Units (US / SI)	Description
Surface Name	n/a	Name of selected aerodynamic surface
Device Installed	n/a	Flag indicating device is installed and is capable of producing aerodynamic force on the vehicle
Drag Coefficient	in² / m²	Coefficient of aerodynamic drag acting on surface
Lift Coefficient	in² / m²	Coefficient of aerodynamic lift acting on surface
Projected Surface Area	in² / cm²	Projected surface area parallel to selected surface
Center of Pressure	in / cm	Vehicle-fixed coordinates where the aerodynamic force is applied

Body Torsion

The HVE Vehicle Model includes the capability to model torsional distortion of the body. By default, the body is assumed to be rigid. However, the user may wish to edit these default parameters, if the actual torsional stiffness of the body is known, to study the effect of torsional distortion on vehicle handling. The Body Torsion dialog is used to edit these properties. See also the reference page, Body Torsion dialog.

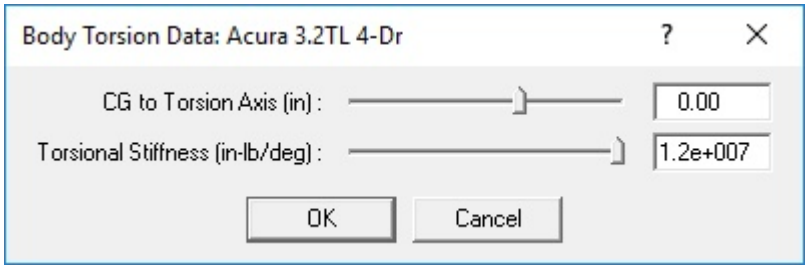


Figure 11-18: Body Torsion dialog.

To display or edit the vehicle's torsional data, perform the following steps:

1. In the Vehicle Viewer, click on the CG sphere. The Sprung Mass options for the current vehicle will be displayed.
2. Choose *Body Torsion* from the Sprung Mass option list. The Body Torsion dialog will be displayed.

3. Edit the torsional stiffness and torsional roll axis location (distance from the CG to the torsion axis).
4. Press *OK* to accept the changes.

Table 11-9: Vehicle Body Torsion Parameters

Parameter	Units (US / SI)	Description
Axis Location	in / cm	Vehicle-fixed z-distance from the CG to the torsional roll axis
Body Stiffness	ft-lb / N-m	Torsional stiffness of the body

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 11, pages 11-1..11-35 — revised for the current HVE release, 2026-07-05.

Chapter 11 — Vehicle Model Definition (Part B: Exterior)

This part covers the vehicle exterior: the exterior dimensions (CG to Front, Right Side, Back, Left Side, Top and Bottom) and the exterior structural stiffness coefficients. These properties are edited by clicking on the exterior spheres displayed on the vehicle in the Vehicle Viewer.

Vehicle Exterior Dimensions

The vehicle's exterior dimensions are defined by its overall length and overall width. The user may choose to edit these dimensions using the Vehicle Dimensions dialogs (CG to Front, Right Side, Back, Left Side, Top, Bottom).

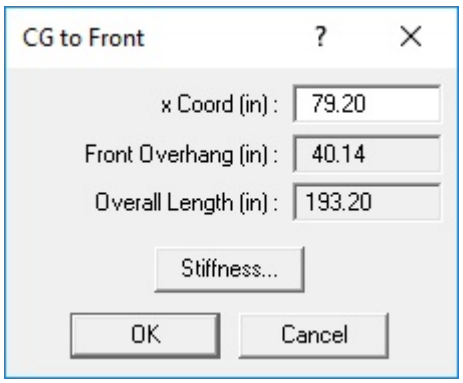


Figure 11-19: Vehicle Exterior Dimensions dialogs allow the user to view and edit the dimensions from the CG to the front, right side, back, left side, top and bottom.

> **NOTE:** The dialog also calculates and displays the overall vehicle length (or width), front/rear overhang, overall height and ground clearance based on the entered data.

(updated: the displayed distances are relative to either the sprung-mass CG or the total-mass CG, according to the current Vehicle Dimensions Basis selected in the user options. The front/rear overhang and overall length calculations account for the selected basis, converting between the sprung and total CG positions as required.)

To display or edit the current vehicle's overall dimensions, perform the following steps:

1. In the Vehicle Viewer, click on the Front, Right, Back, Left, Top or Bottom exterior sphere. The appropriate dialog (CG to Front, Back, etc.) will be displayed.
2. View and/or edit the exterior dimension.
3. Press *OK* to accept the changes. *(updated: an Apply button is also available — it applies the entered dimension without closing the dialog; Cancel reverts the vehicle to its original dimensions and stiffness.)*

The Exterior Dimension parameters are described below.

- **CG to Front** — Assigns the distance from the CG to the front of the current vehicle.
- **CG to Right Side** — Assigns the distance from the CG to the right side of the current vehicle.
- **CG to Back** — Assigns the distance from the CG to the back of the current vehicle (the value will be negative, indicating it is behind the CG).
- **CG to Left Side** — Assigns the distance from the CG to the left side of the current vehicle (in the SAE coordinate system the value will be negative, indicating it is to the left of the CG).

- **CG to Top** — Assigns the distance from the CG to the top of the current vehicle (the value will be negative, indicating the roof is above the CG).
- **CG to Bottom** — Assigns the distance from the CG to the bottom side of the current vehicle.

Table 11-10: Vehicle Exterior Dimension Parameters

Parameter	Units (US / SI)	Description
CG to Front/Back/Side/Top/Bottom	in / cm	Vehicle-fixed distance from the CG to the front, right side, back, left side, top or bottom

See also the reference pages: CG to Front, CG to Right Side, CG to Back, CG to Left Side, CG to Top, CG to Bottom.

Vehicle Exterior Stiffness Coefficients

The vehicle's structural properties are defined by its stiffness coefficients. The user may choose to edit these coefficients for the front, back or sides of the vehicle using the Vehicle Stiffness dialogs.

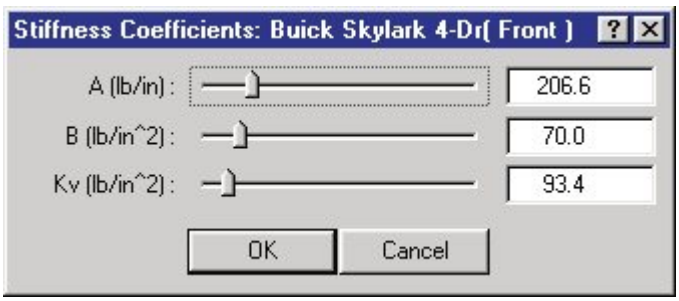


Figure 11-20: Vehicle Exterior Stiffness Coefficients dialogs allow the user to view and edit the A, B and Kv stiffness coefficients for the selected surface.

(updated: the A/B/Kv Stiffness Coefficients dialog is used when HVE is running in 2-D mode. In 3-D mode, the Stiffness button in the Vehicle Dimensions dialogs opens the Vehicle Exterior Stiffness dialog instead, which additionally supports 3-D — constant/linear/quadratic/cubic — stiffness definitions, saturation crush and maximum crush. The Stiffness button is disabled for Fixed Barrier and Movable Barrier vehicle types.)

To display or edit the current vehicle's A, B and Kv stiffness coefficients, perform the following steps:

1. In the Vehicle Viewer, click on the front, right side, back or left side exterior sphere. The appropriate dialog (CG to Front, Right Side, Back, Left Side, Top or Bottom) will be displayed.
2. Click on *Stiffness*. The Stiffness Coefficients dialog for the current surface will be displayed.
3. View and/or edit the stiffness properties.
4. Press *OK* to accept the changes.

The Exterior Stiffness parameters are described below.

- **'A' Stiffness Coefficient** — Assigns the force per unit of damage width required to initiate permanent crush (see Figures 11-20 and 11-21).
- **'B' Stiffness Coefficient** — Assigns the linear spring rate (force vs deflection constant) per unit of damage width required to produce permanent crush (see Figures 11-20 and 11-21).

- **'Kv' Stiffness Coefficient** — Assigns the linear spring rate (force vs deflection constant) per unit of damage width required to produce permanent crush. This value differs from the B constant in that it assumes the force vs deflection graph goes through zero (i.e., A = 0).

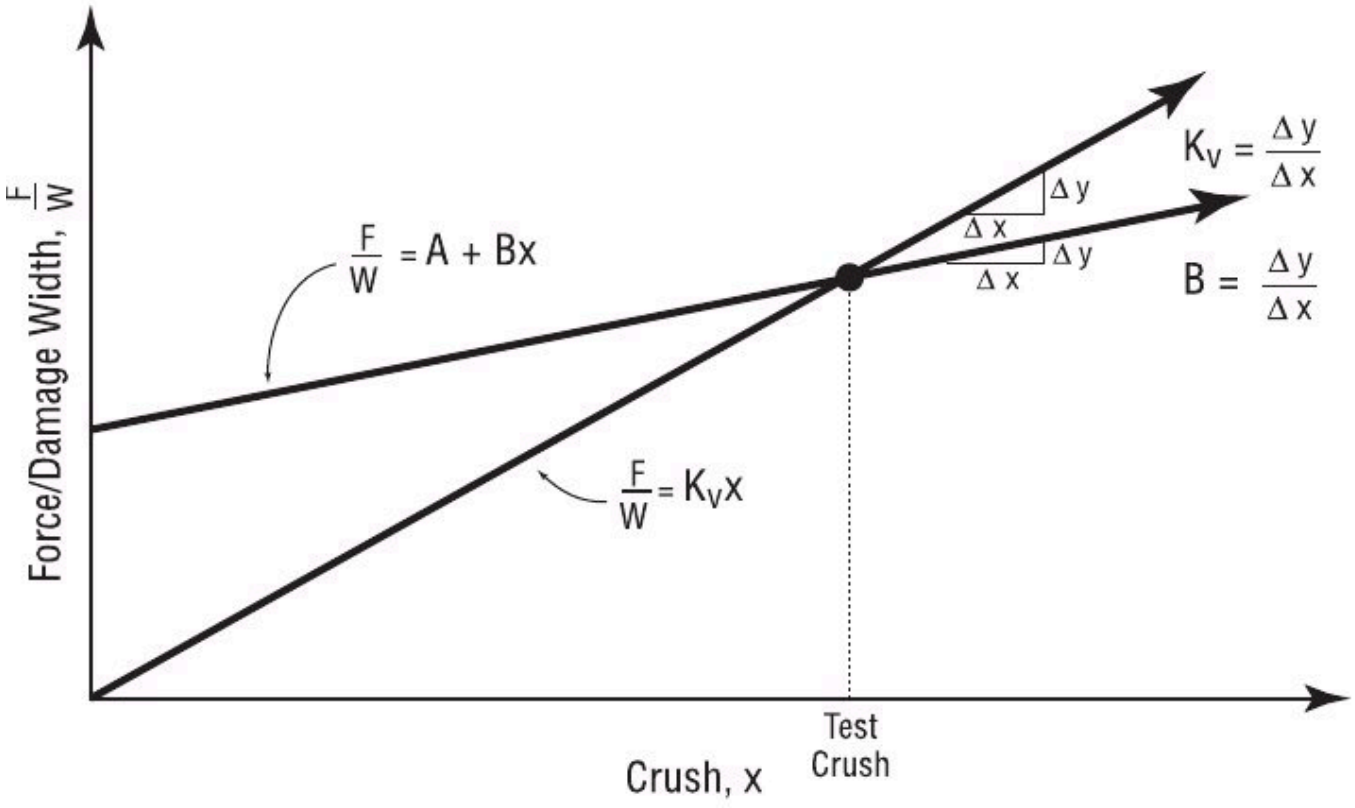


Figure 11-21: Force vs Deflection characteristics of a vehicle, and their relationship to the A, B and Kv stiffness coefficients (see also references 2.7–2.13, 4.20, 4.29).

Table 11-11: Vehicle Exterior Stiffness Coefficients Parameters

Parameter	Units (US / SI)	Description
A Stiffness	lb/in / N/cm	Vehicle 'A' stiffness coefficient
B Stiffness	lb/in² / N/cm²	Vehicle 'B' stiffness coefficient
Kv Stiffness	lb/in² / N/cm²	Vehicle 'Kv' stiffness coefficient

See also the reference pages: Stiffness Coefficients dialog (2-D), A Stiffness, B Stiffness.

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 11, pages 11-36..11-39 — revised for the current HVE release, 2026-07-05.

Chapter 11 — Vehicle Model Definition (Part C: Suspension)

Suspension Properties

The suspension parameters of the vehicle are displayed and edited using the Suspension Data dialog

(updated: the legacy manual calls this the "Suspension Information" dialog; the current dialog is titled "Suspension Data" and identifies the current vehicle, axle number and side, e.g., "Suspension Data: ... (Axle No. 1, Left Side)")

.

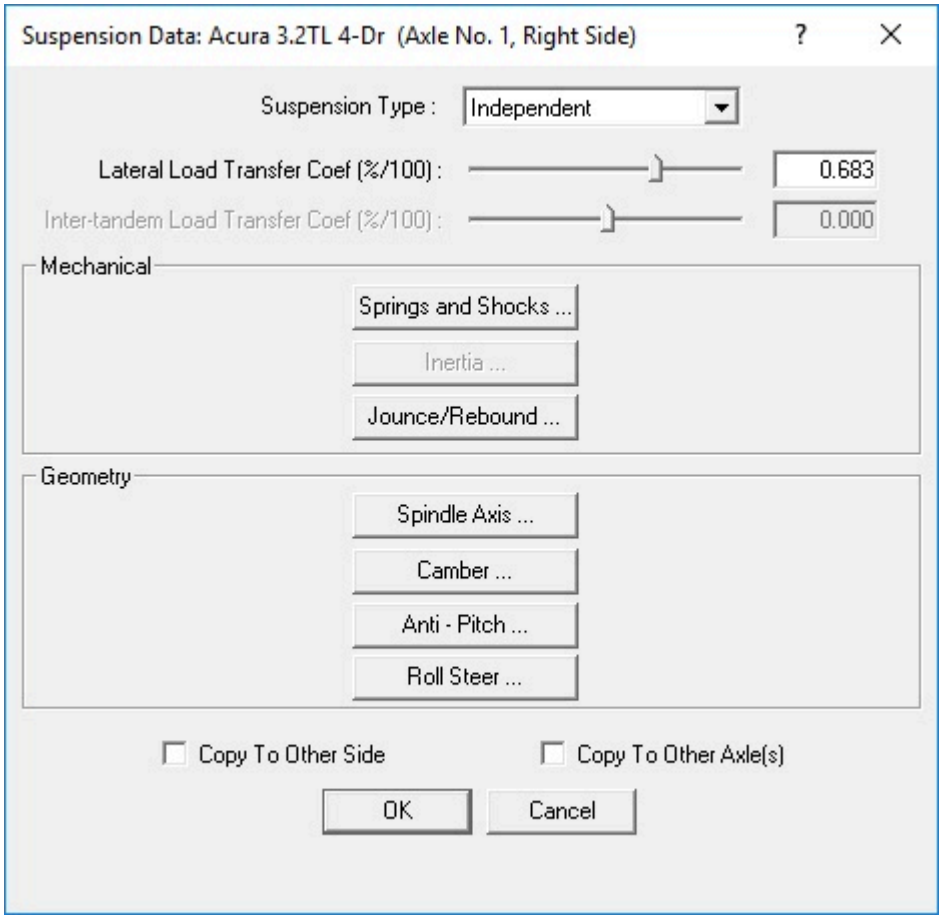


Figure 11-22: The Suspension Data dialog allows the user to choose specific suspension properties for viewing and editing.

The Suspension Data dialog includes pushbuttons allowing the user to select specific suspension properties:

- Suspension Type
- Roll Couple Distribution *(updated: added since the legacy manual — see below)*
- Inter-tandem Load Transfer (Tandem Axle suspension types only)
- Mechanical Properties (Springs and Shocks, Inertias, Jounce/Rebound Stops)
- Geometry Properties (Spindle Axis, Camber and Half-track Tables, Anti-pitch and Roll Steer)

Each of these properties is described in the following sections. See also the reference page, Suspension Data dialog.

Suspension Type

The HVE Vehicle Model includes the following suspension types:

- **Independent** — The right and left wheels move independently from each other (see Figure 11-23a).
- **Solid Axle** — The right and left wheels are rigidly connected (see Figure 11-23b).
- **Tandem, 4-Spring** — A set of two solid axle suspensions sharing an articulated spring support, such that the force on the rear of the leading spring is transferred to the front of the trailing spring (see Figure 11-23c).
- **Tandem, Walking Beam** — A set of two solid axle suspensions connected by a rigid link (see Figure 11-23d).

(updated: the tandem types are offered in the list only where a tandem axle set is possible — on rear axles of vehicles with more than two axles, and on trailers or dollies with more than one axle. Otherwise the list contains only Independent and Solid Axle.)

To choose the suspension type for any axle, perform the following steps:

1. Click the right or left wheel for the selected axle. The Suspension Data dialog for the selected wheel is displayed.
2. Click on the *Suspension Type* option button to display the Suspension Type option list, and select one of the available types.

> **NOTE:** Inertial properties are disabled for independent suspensions.

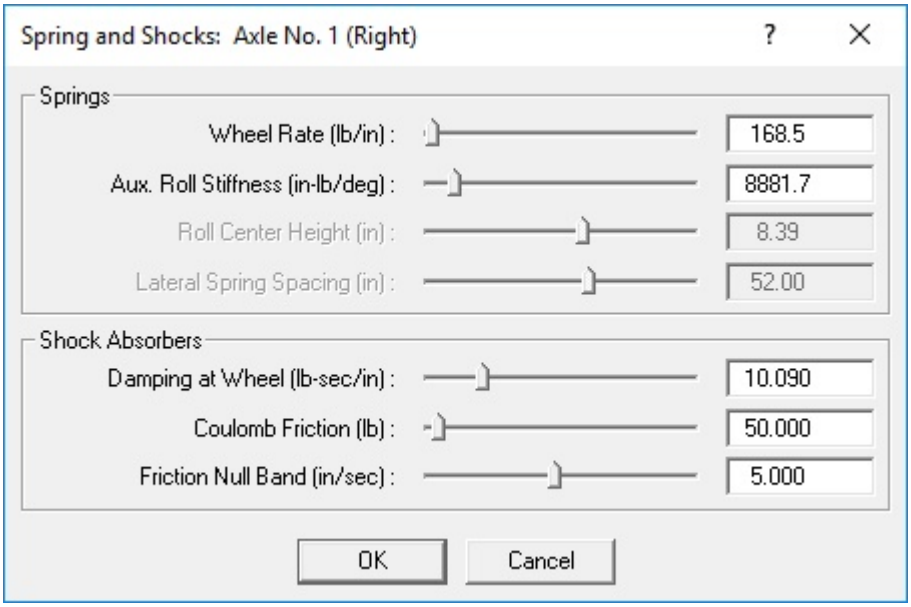


Figure 11-23: Suspension Types: Independent (a), Solid Axle (b), Tandem Axle, 4-Spring (c) and Tandem Axle, Walking Beam (d).

Roll Couple Distribution

(updated: this option was added after the legacy manual was written.)

A slider/edit field for the fraction (%/100) of the total sprung mass roll couple (lateral load transfer) reacted at the selected axle. Values at the individual axles are automatically balanced so that the distribution over all axles totals 1.0 — for example, changing the value at a rear axle adjusts the front axle share accordingly; on three-axle vehicles the remainder is split between the other axles. The controls are disabled if the vehicle has fewer than two axles, or if the vehicle has no driver location defined (except movable barriers).

Inter-tandem Load Transfer

If the selected suspension type is a tandem axle (either 4-Spring or Walking Beam), the Inter-tandem Load Transfer parameter is enabled, allowing the user to enter a value (%/100) for the inter-tandem load transfer for the selected axle set. To enter the inter-tandem load transfer for a tandem axle suspension, perform the following steps:

1. Click on *Suspension Type* and choose a tandem axle suspension type (4-Spring or Walking Beam). The inter-tandem load transfer parameter is enabled.
2. Enter the desired value.

Springs and Shocks

The Springs and Shocks properties for the selected wheel location are displayed and edited using the Springs and Shocks dialog. See also the reference page, Springs and Shocks dialog.

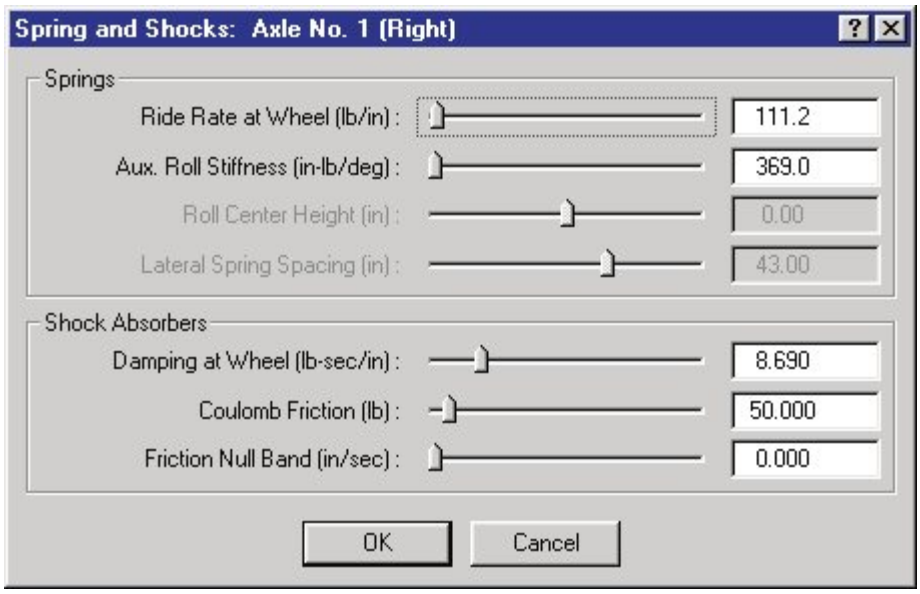
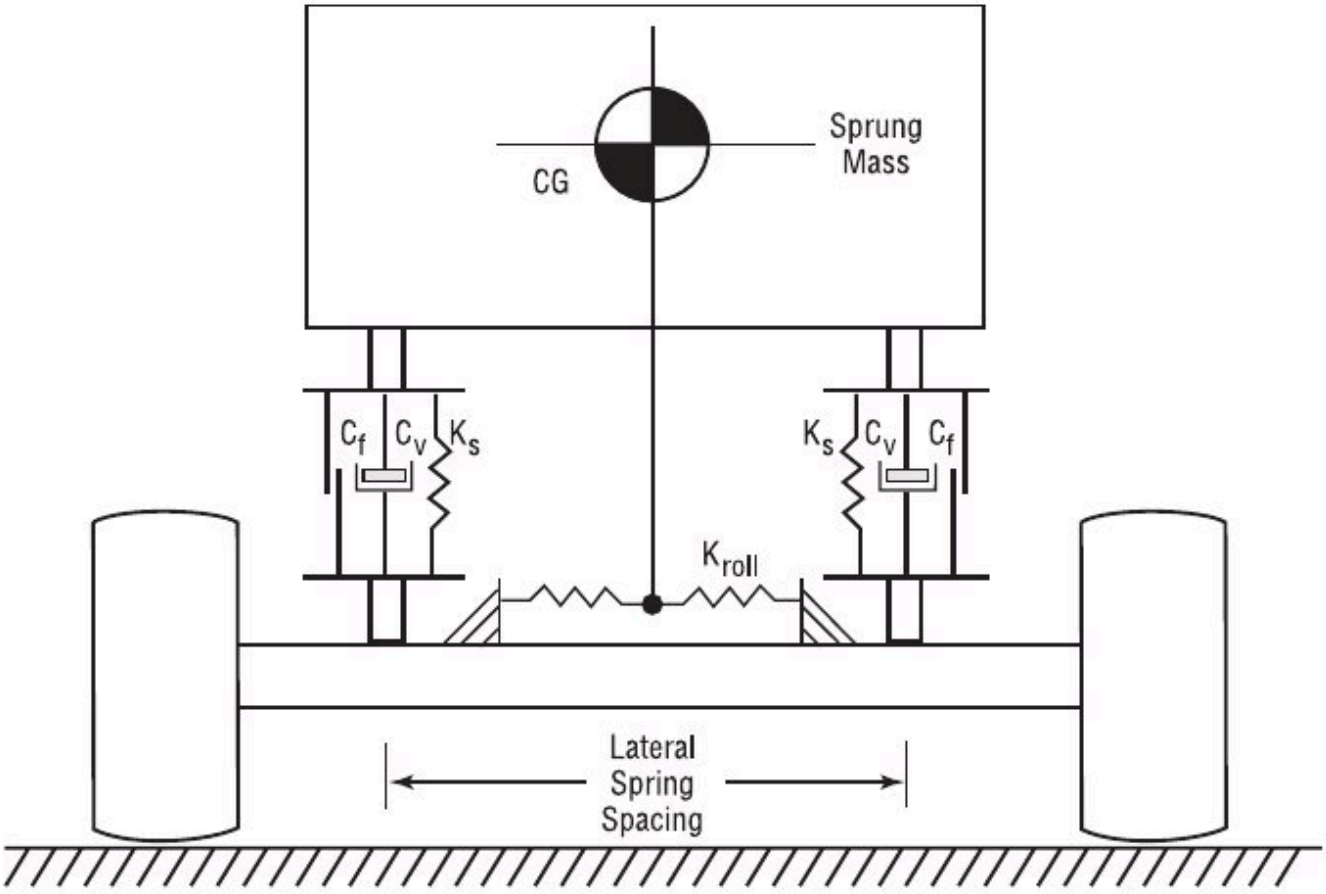


Figure 11-24: Springs and Shocks dialog for selected wheel location.

To display or edit the spring and shock parameters for the selected wheel position, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Springs and Shocks*. The Springs and Shocks dialog is displayed.
4. View and/or edit the desired properties.
5. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.
6. If desired, click *Copy To Other Side* to make the changes apply to both sides.
7. Press *OK* to accept the suspension changes to the selected wheel position(s).


CG
Sprung
Mass
C_f
C_v
K_s
K_s
C_v
C_f
K_{roll}
Lateral
Spring
Spacing

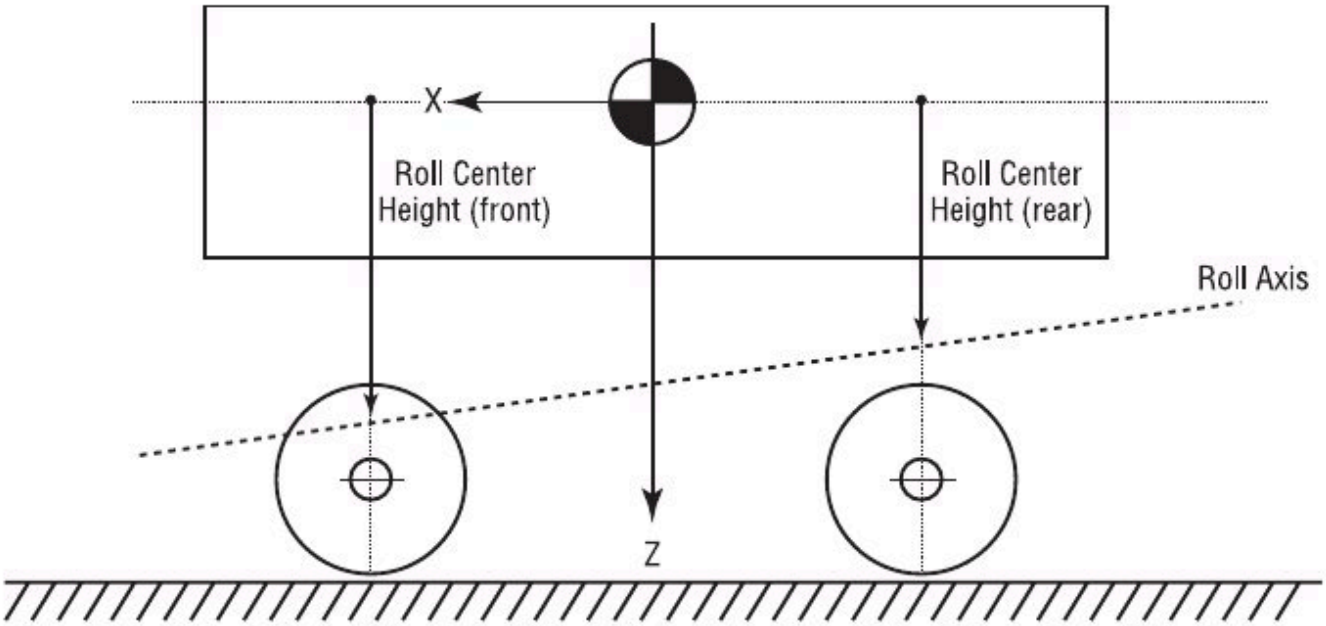

X
Roll Center
Height (front)
Roll Center
Height (rear)
Roll Axis
Z

Figure 11-25: Suspension parameters used in the HVE Vehicle Model. Ks, Cv and Cf are spring rate, damping rate and coulomb friction, respectively, for each wheel position. Kroll is the additional roll stiffness contribution due to an anti-sway bar.

The Springs and Shocks properties are described below.

- **Ride Rate at Wheel** — Spring rate (force per unit of wheel travel) at the selected wheel due to the spring.
- **Auxiliary Roll Stiffness** — Roll stiffness (torque per unit of sprung mass roll) added by the addition of an anti-sway bar.
- **Roll Center Height** — Vertical distance from the sprung mass CG down (+) to the suspension roll center.
- **Lateral Spring Spacing** — Lateral distance between springs for solid axle suspension types.
- **Damping Rate at Wheel** — Velocity-dependent suspension force due to shock absorbers.
- **Coulomb Friction** — Friction force required to initiate suspension travel.
- **Friction Null Band** — Minimum suspension velocity required for full coulomb friction force.

Table 11-12: Suspension Spring and Shock Parameters

Parameter	Units (US / SI)	Description
Ride Rate	lb / N	Linear force vs deflection characteristic effective at the wheel
Auxiliary Roll Stiffness	in-lb/deg / N-m/deg	Moment per unit of body roll produced by an anti-sway bar
Roll Center Height	in / cm	Height of the roll axis measured at the suspension
Lateral Spring Spacing	in / cm	Lateral distance between left and right springs for solid axle suspensions
Damping at Wheel	lb-sec/in / N-sec/m	Damping rate effective at the wheel
Coulomb Friction	lb / N	Force required to initiate suspension travel
Friction Null Band	in/sec / cm/sec	Suspension velocity required for full coulomb friction

Axle Inertia

The Suspension Inertial properties for the selected wheel location are displayed and edited using the Axle Inertia dialog.

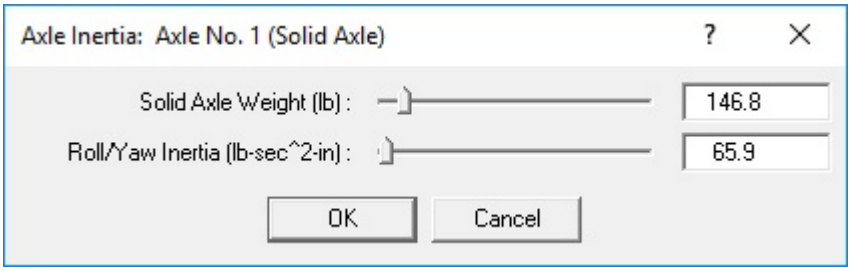


Figure 11-26: Suspension Inertia dialog for selected axle — the selected wheel shares these parameters with the wheel on the other side of the vehicle.

To display or edit the suspension inertial parameters for the selected wheel/axle, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Axle Inertias*. The Axle Inertia dialog is displayed.

> **NOTE:** The Axle Inertia option is only available for solid axle suspension systems.

1. View and/or edit the desired properties.
2. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.
3. If desired, click *Copy To Other Side* to make the changes apply to both sides.

> **NOTE:** This action is actually irrelevant for the inertias dialog because it automatically applies to the axle shared by both wheels.

1. Click *OK* to accept the suspension changes to the selected wheel position(s).

The Suspension Inertial properties are described below.

- **Weight** — Total weight of the axle and springs.

> **NOTE:** The Vehicle Editor actually divides your entered value by the current gravitational constant (see Environment Editor) and stores the resultant mass. That way, if you take the vehicle to the moon, it will behave correctly!
>
> **NOTE:** The weight of the wheels and brakes should not be included here! By convention, the weight of each wheel assembly, including the brake, is to be included in the tire weight (see Tire Physical Parameters).

- **Roll/Yaw Inertia** — Roll and yaw inertia of the axle. Per the above note, do not include wheel inertias; they are included with the tires.

> **NOTE:** The pitch inertia of the axle is assumed to be negligible.

Table 11-13: Suspension Inertial Parameters

Parameter	Units (US / SI)	Description
Weight	lb / N	Weight of the suspension, excluding wheels and brakes
Roll/Yaw Inertia	lb-sec²-in / kg-m²	Roll and yaw rotational inertias of axle, excluding wheels and brakes

Jounce/Rebound

The Jounce and Rebound properties for the selected wheel location are displayed and edited using the Jounce/Rebound dialog. Suspension stops are used to limit the travel of the suspension. See also the reference page, Jounce and Rebound dialog.

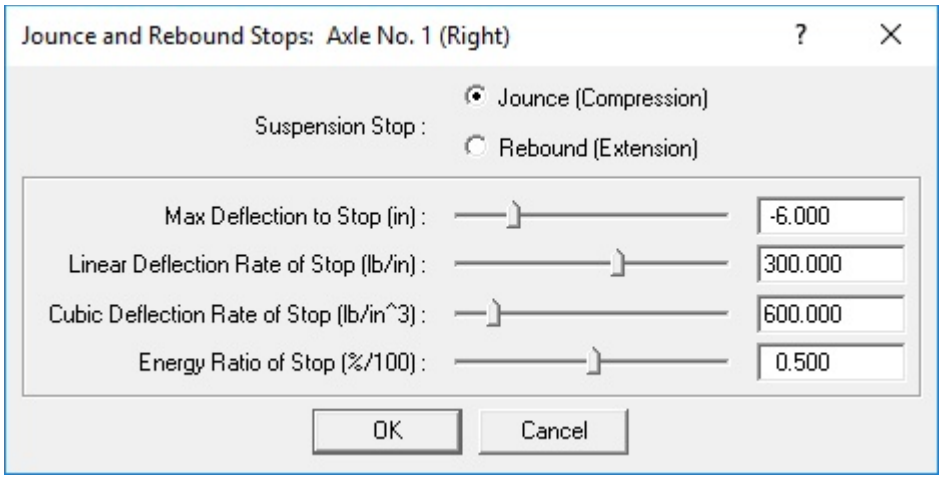
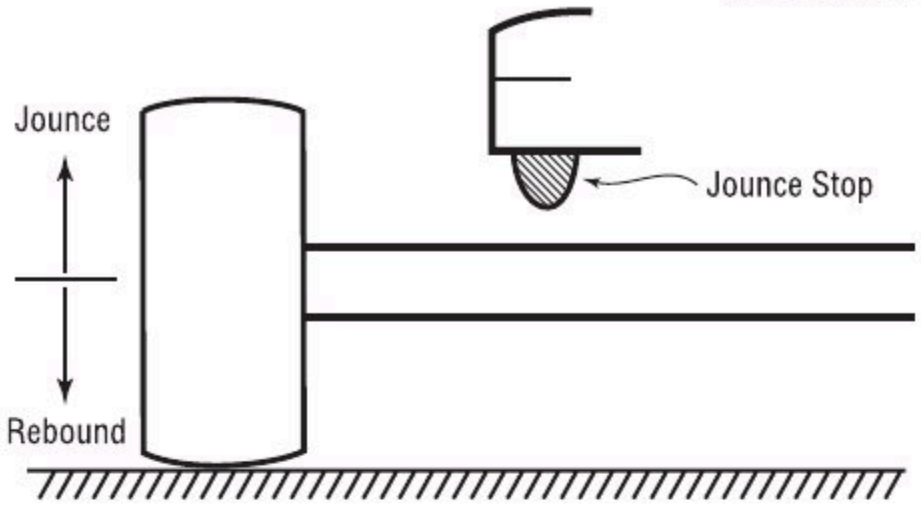
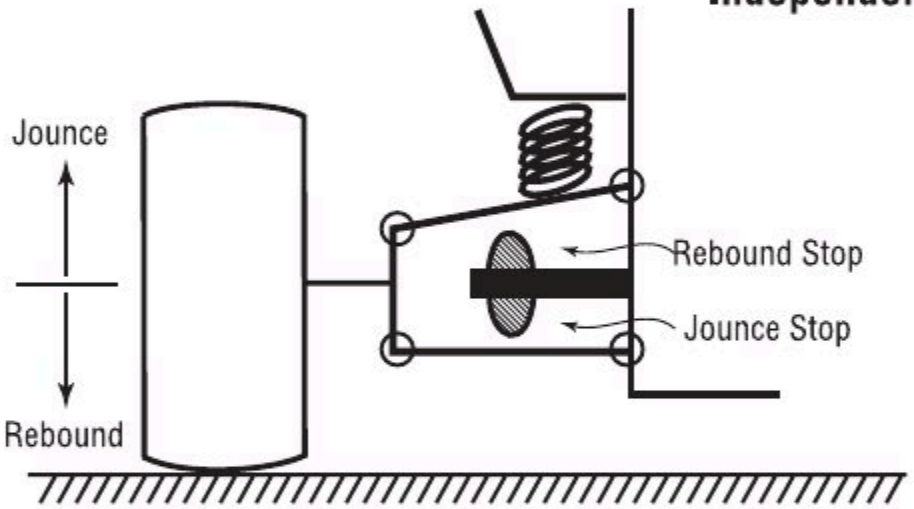


Figure 11-27: Jounce and Rebound dialog for selected wheel.

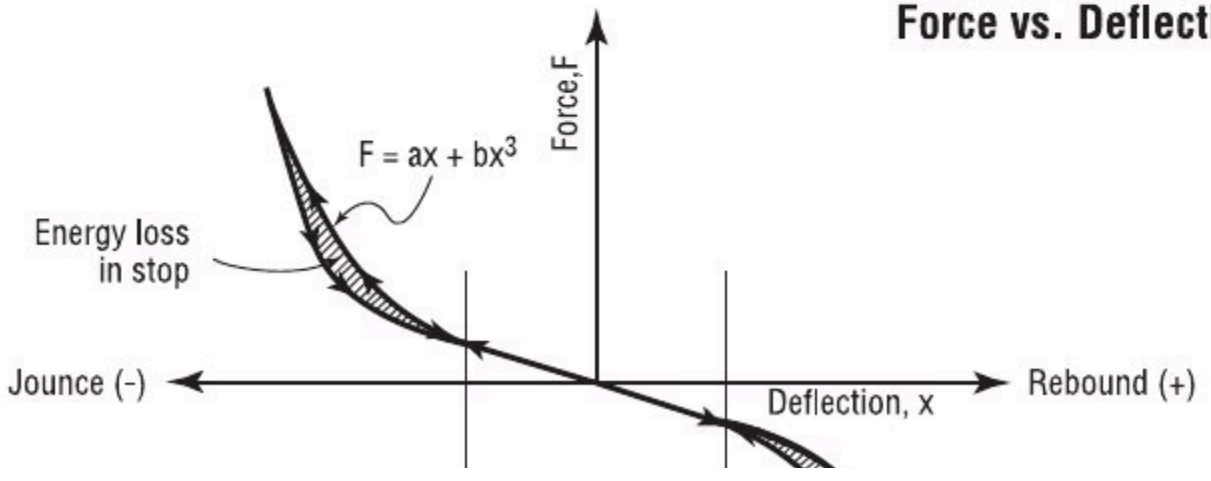
Solid Axle Suspension



Independent Suspension



Jounce/Rebound Stop Force vs. Deflection



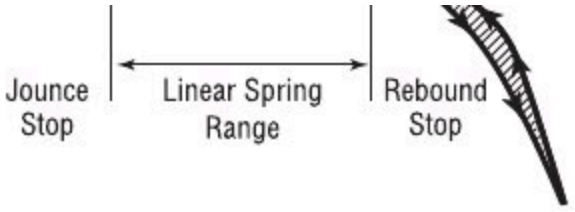


Figure 11-28: Suspension Jounce and Rebound parameters.

To display or edit the suspension stop parameters for the selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Jounce/Rebound*. The Jounce/Rebound dialog is displayed.
4. Click the *Jounce* radio button to view and/or edit the desired jounce stop properties.
5. Click the *Rebound* button to view and/or edit the desired rebound stop properties.
6. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.
7. If desired, click *Copy To Other Side* to make the changes apply to both sides.
8. Click *OK* to accept the suspension changes to the selected wheel position(s).

The Jounce/Rebound properties are described below.

- **Jounce/Rebound Button** — Type of stop: Jounce (compression travel limit) or Rebound (extension travel limit).
- **Maximum Deflection to Stop** — Suspension travel distance from equilibrium to selected stop.
- **Stop Linear Rate** — Linear constant (force per unit of wheel travel) of the selected suspension stop.
- **Stop Cubic Rate** — Cubic constant (force per unit of wheel travel) of the selected suspension stop.
- **Stop Energy Ratio** — Ratio of conserved to total energy (restitution) of the selected suspension stop.

Table 11-14: Suspension Jounce/Rebound Parameters

Parameter	Units (US / SI)	Description
Stop Type	n/a	Location of stop (Jounce or Rebound)
Maximum Deflection	in / cm	Suspension travel from equilibrium to jounce or rebound stop
Linear Stop Rate	lb/in / N/cm	Spring linear constant (force per unit of wheel travel) of the selected suspension stop *(the legacy table said "cubic" here — a typo)*
Cubic Stop Rate	lb/in^3 / N/cm^3	Spring cubic constant (force per unit of wheel travel) of the selected suspension stop
Energy Ratio	%/100	Ratio of conserved to total energy (restitution) of the selected suspension stop

Spindle Axis

The Spindle Axis geometry properties for the selected wheel location are displayed and edited using the Spindle Axis dialog. See also the reference page, Spindle Axis dialog.

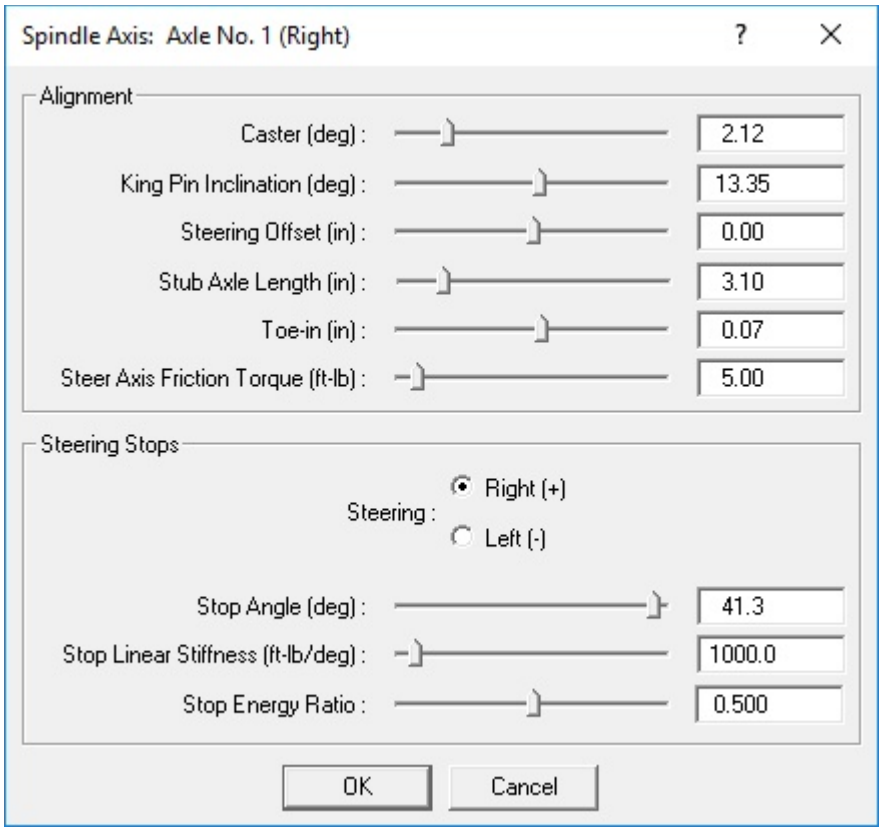


Figure 11-29: Spindle Axis dialog for selected wheel.

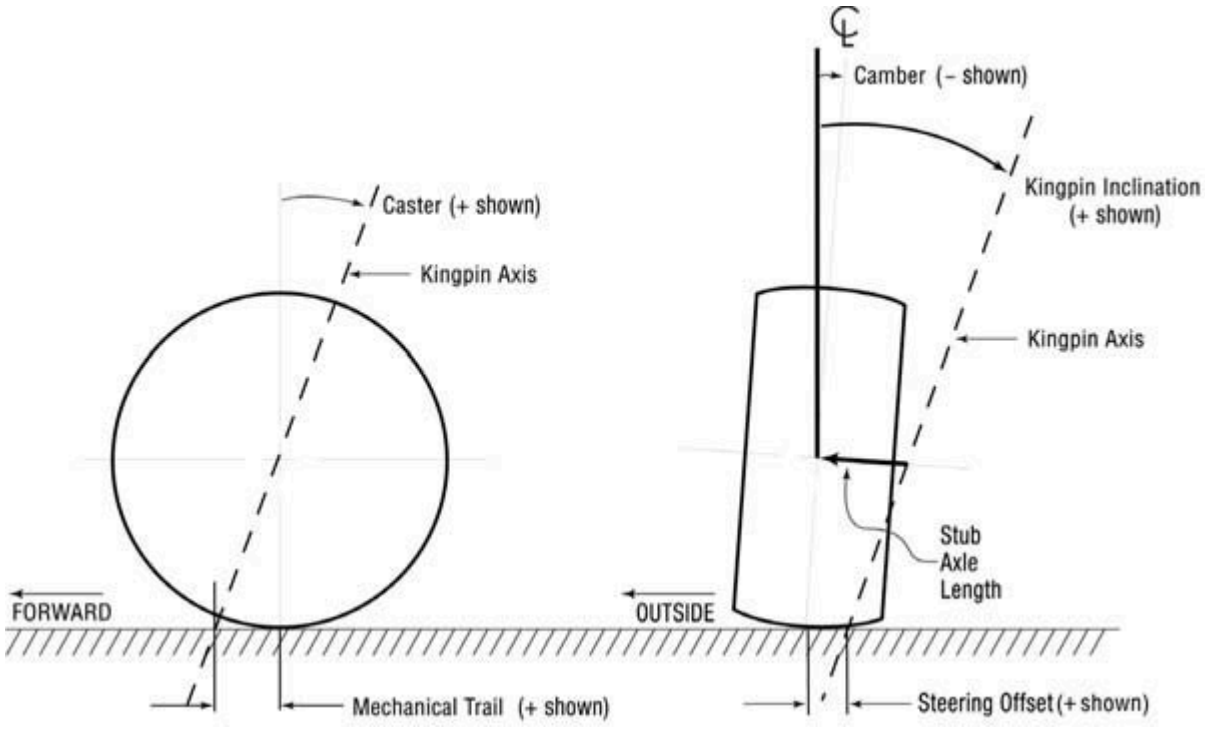


Figure 11-30: Suspension Spindle Axis geometry parameters.

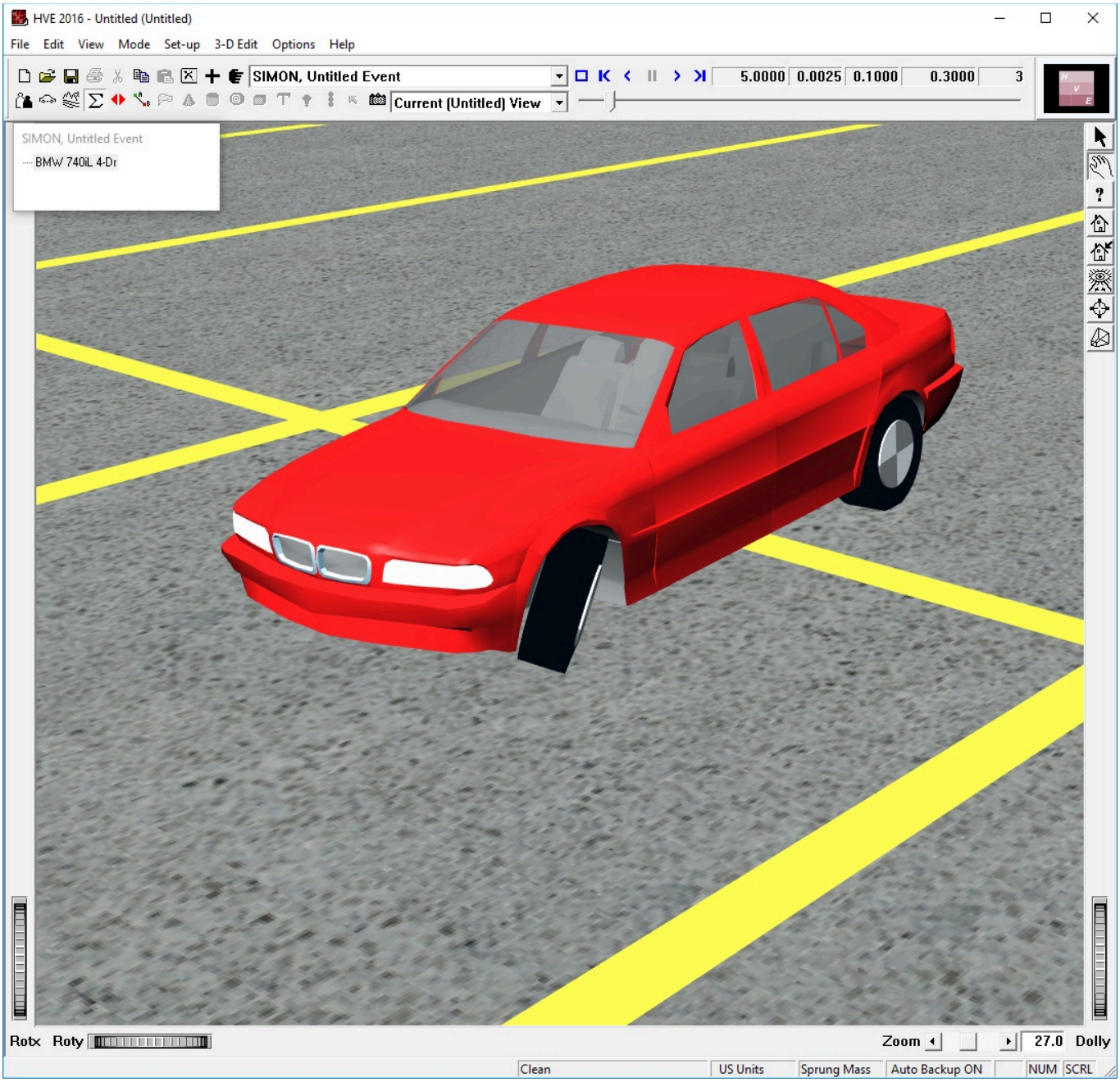


Figure 11-31: Effect of suspension alignment (in particular, caster, kingpin inclination and stub axle length) on the geometry of a steered wheel — note left front wheel.

Alignment

The Alignment parameters, especially Caster, Camber, Kingpin Inclination and Steering Offset (see Figure 11-30), play an important role in determining how tire forces on a steerable axle affect directional response.

> **NOTE:** Some simulation models assume the wheel steers about a vertical axis through the tire plane. This is a simplification. A wheel on a steerable axle actually steers about the kingpin axis. Some simulation models (e.g., SIMON) rigorously model this behavior.

To display or edit the Alignment parameters for the selected wheel position, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Spindle Axis*. The Spindle Axis dialog is displayed.
4. View and/or edit the desired Alignment properties.

> **NOTE:** Kingpin Inclination, Steering Offset and Stub Axle Length are inter-related. Editing the Kingpin Inclination or Stub Axle Length will automatically recalculate Steering Offset. Similarly, editing Steering Offset will automatically update Stub Axle Length.

Steering Stops

The Steering Stop parameters define the logical limit to steer angles (+ or −) at each wheel. They are also used by the Steer DOF model, when it is employed, to calculate the steering torque that results when a wheel is steered beyond its limit.

To display or edit the Steering Stop parameters for the selected wheel position, continue as follows:

1. Click on the *Steering* radio button and choose the Right or Left steering stop.
2. Edit the Stop Angle, Stiffness and/or Stop Energy Ratio as desired.
3. Press *OK* to accept the changes to the Alignment and Steering Stop parameters. The Suspension Data dialog is still displayed.
4. If desired, click *Copy To Other Side* to make the changes apply to both sides of the vehicle.
5. Click *OK* to accept the suspension changes to the selected wheel position(s).

> **NOTE:** The Spindle Axis dialog is displayed only for steerable axles! (See Steering System.)

The Spindle Axis Alignment properties are described below.

- **Caster** — The fore-aft angle of the spindle axis (see Figure 11-30). This angle is largely responsible for the self-aligning properties of a steerable axle.

> **NOTE:** Caster Angle is defined according to SAE J670e as positive (+) when the top of the axis tilts rearward (see Figure 11-30).

- **Kingpin Inclination** — The angle in front elevation between the steering axis and a line parallel to the vehicle-fixed z axis (see Figure 11-30). This angle, along with the caster angle, is responsible for the self-aligning properties of a steerable axle.

> **NOTE:** Kingpin Inclination Angle is defined according to SAE J670e as positive (+) when the top of the axis tilts inward, regardless of right or left side (see Figure 11-30).

- **Steering Offset (Scrub Radius)** — The distance from the center of the tire contact patch to the point where the steering axis intersects the ground at equilibrium conditions. This distance produces a steering torque (positive or negative, depending on the sign of the scrub radius) due to longitudinal tire force (either braking or driving).

> **NOTE:** This value should be kept small. Otherwise, unwanted steering effects will be introduced on acceleration or braking.

- **Stub Axle Length** — (Also called Kingpin Offset at Wheel Center.) The distance from the steering axis to the center of the wheel (see Figure 11-30). As described above, Kingpin Inclination, Steering Offset and Stub Axle Length are inter-related; changes to one of these parameters affect the others.
- **Toe-in** — The difference in distance between the center of the tire plane at the spindle and the front of the tire.
- **Steer Axis Friction Torque** — The coulomb friction in the steering axis. This friction produces torque that resists steering moments, either from the tire-terrain interface or from the steering wheel. This parameter affects the Steer DOF model.

The Steer Axis Steering Stop parameters are described below.

- **Steering Stop** — This radio button allows the user to select the desired stop (Right Steer or Left Steer).
- **Stop Angle** — The vehicle-fixed steering angle at which the stop is reached.
- **Stop Stiffness** — The mechanical stiffness of the steering stop. Additional steering will be resisted by a torque that attempts to return the wheel towards zero steer.
- **Stop Energy Ratio** — The ratio of conserved to total energy of the selected steering stop (this property is analogous to Jounce and Rebound Stops; see Figure 11-28). This parameter keeps the wheel from bouncing off the steering stop.

Table 11-15: Spindle Axis Parameters

Parameter	Units (US / SI)	Description
Caster	deg	Fore-aft angle of spindle axis
Kingpin Inclination	deg	Lateral angle of spindle axis
Steering Offset (Scrub Radius)	in / cm	Lateral distance from the steering axis to the center of the tire contact patch, measured at the road plane
Stub Axle Length	in / cm	Distance from the steering axis to the center of the wheel
Toe-in	in / cm	Difference in the lateral distance between the tire centerline, measured at the front and rear of the tire
Steering Stop	n/a	Stop selector for right or left steering stop
Stop Angle	deg	Steering angle at which the stop is reached
Stop Linear Stiffness	ft-lb/deg / N-m/deg	Mechanical stiffness of the steering stop
Stop Energy Ratio	—	Ratio of conserved to total energy in the steering stop

Camber & Half-track Table

The Camber and Half-track properties for the selected wheel location are displayed and edited using the Camber & Half-track Table dialog. See also the reference page, Camber dialog.

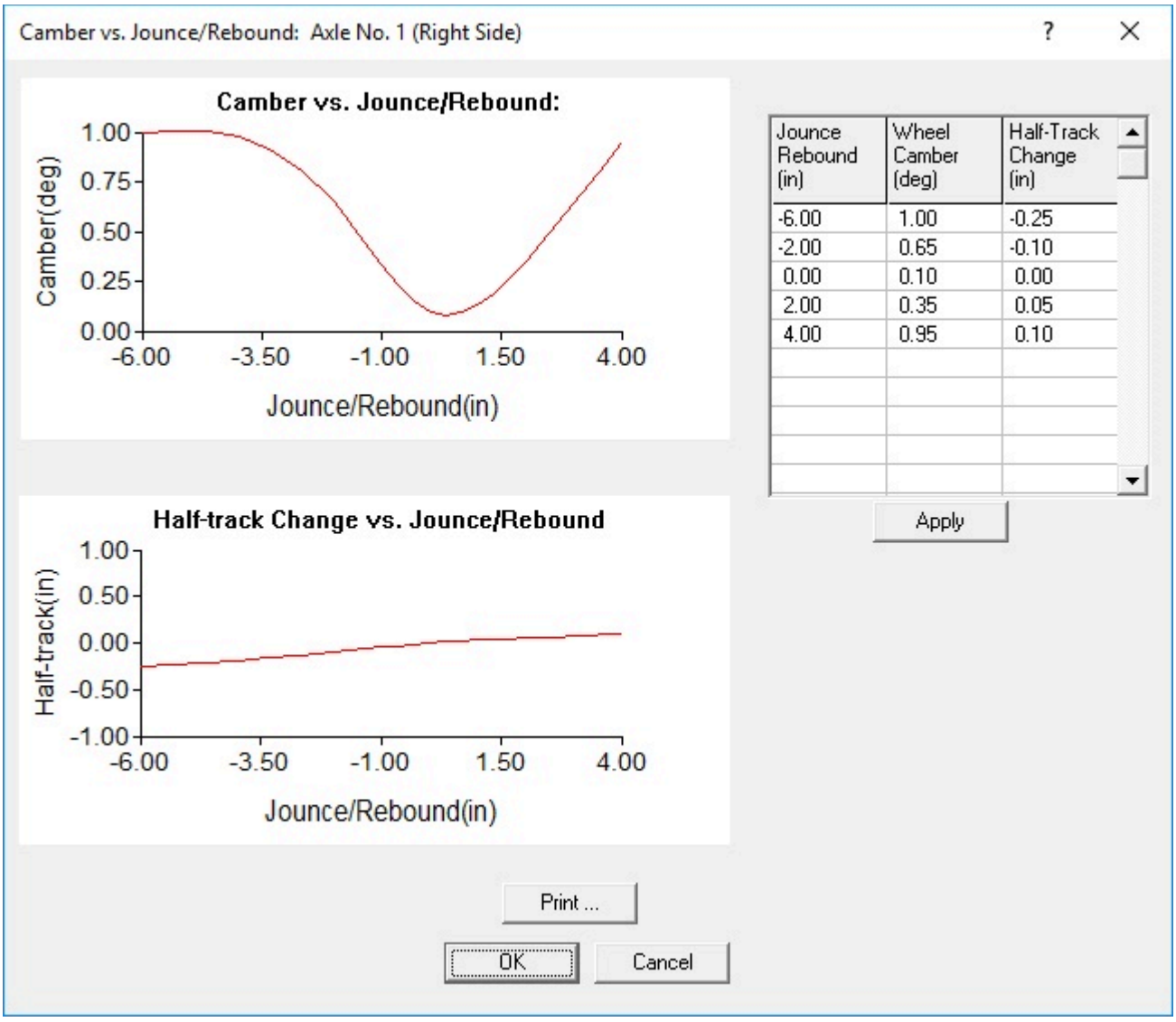


Figure 11-32: Camber and Half-track Table dialog for selected wheel.

To display or edit the camber and half-track tables for the selected wheel position, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Camber & Half-track Table*. The Camber & Half-track Table dialog is displayed.
4. View and/or edit the desired properties.
5. Press *Apply* to graph the changes to the camber and half-track tables.
6. Press *Print* to print the tables.

> **NOTE:** The tables are printed on the system printer. See Printing for more information.

1. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.
2. If desired, click *Copy To Other Side* to make the changes apply to both sides.
3. Press *OK* to accept the suspension changes to the selected wheel position(s).

The Camber and Half-track Table properties are described below.

- **Jounce/Rebound** — Suspension travel from equilibrium (see Figure 11-28). Jounce (compression) is negative; rebound (extension) is positive.
- **Camber** — Camber angle (see Figure 11-30) at the specified suspension deflection.

> **NOTE:** Camber angle is defined according to SAE J670e as positive (+) when the top of the wheel tilts outward, regardless of right or left side (see Figure 11-30).

- **Half-track Change** — Change in lateral wheel position due to suspension jounce or rebound.

Table 11-16: Camber and Half-track Table Parameters

Parameter	Units (US / SI)	Description
Jounce/Rebound	in / cm	Suspension travel from equilibrium to Jounce (−) or Rebound (+) stop
Camber	deg	Camber angle of wheel at specified suspension deflection
Half-track Change	in / cm	Change in lateral wheel position due to suspension jounce or rebound

Anti-Pitch

The Anti-Pitch properties for the selected wheel location are displayed and edited using the Anti-Pitch dialog. See also the reference page, Anti-Pitch dialog.

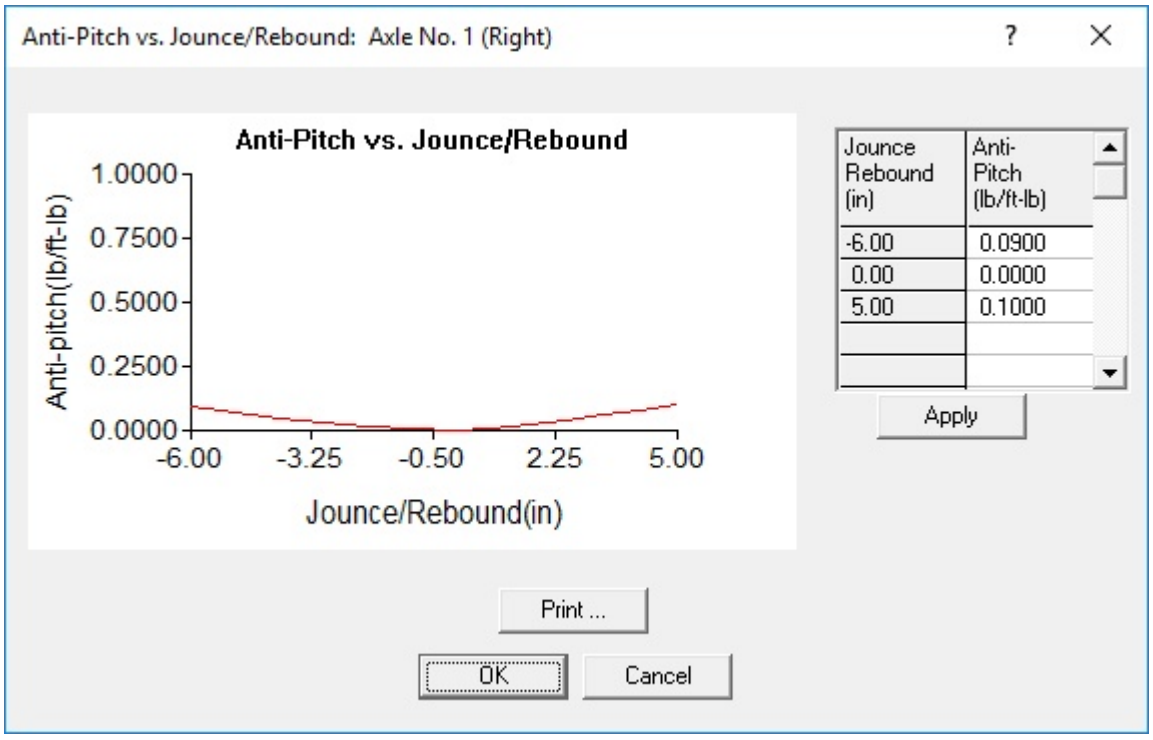


Figure 11-33: The Vehicle Anti-Pitch dialog allows the user to enter suspension anti-pitch characteristics.

To display or edit the anti-pitch table for the selected wheel position, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.

2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Anti-pitch*. The Anti-pitch Table dialog is displayed.
4. Enter jounce/rebound and a corresponding anti-pitch value. Up to 15 pairs of data may be entered.
5. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.
6. If desired, click *Copy To Other Side* to make the changes apply to both sides.
7. Click *OK* to accept the suspension changes to the selected wheel position(s).

The Anti-pitch Table properties are described below.

- **Jounce/Rebound** — Suspension travel from equilibrium (see Figure 11-28). Jounce (compression) is negative; rebound (extension) is positive.
- **Anti-pitch** — Anti-pitch force at the specified suspension deflection.

Table 11-17: Anti-pitch Table Parameters

Parameter	Units (US / SI)	Description
Jounce/Rebound	in / cm	Suspension travel from equilibrium to Jounce (−) or Rebound (+) stop
Anti-pitch	lb/ft-lb / N/N-m	Anti-pitch force at given suspension jounce or rebound

Roll Steer

The Roll Steer properties for the selected wheel location are displayed and edited using the Roll Steer dialog. The roll steer dialogs are different for independent and solid axle suspensions: independent suspensions use the polynomial-coefficient dialog with a

Roll Steer vs. Jounce/Rebound

graph; solid axle (and tandem) suspensions use a single-coefficient dialog. See also the reference pages, Roll Steer dialog (independent) and Roll Steer Properties (solid/tandem axle).

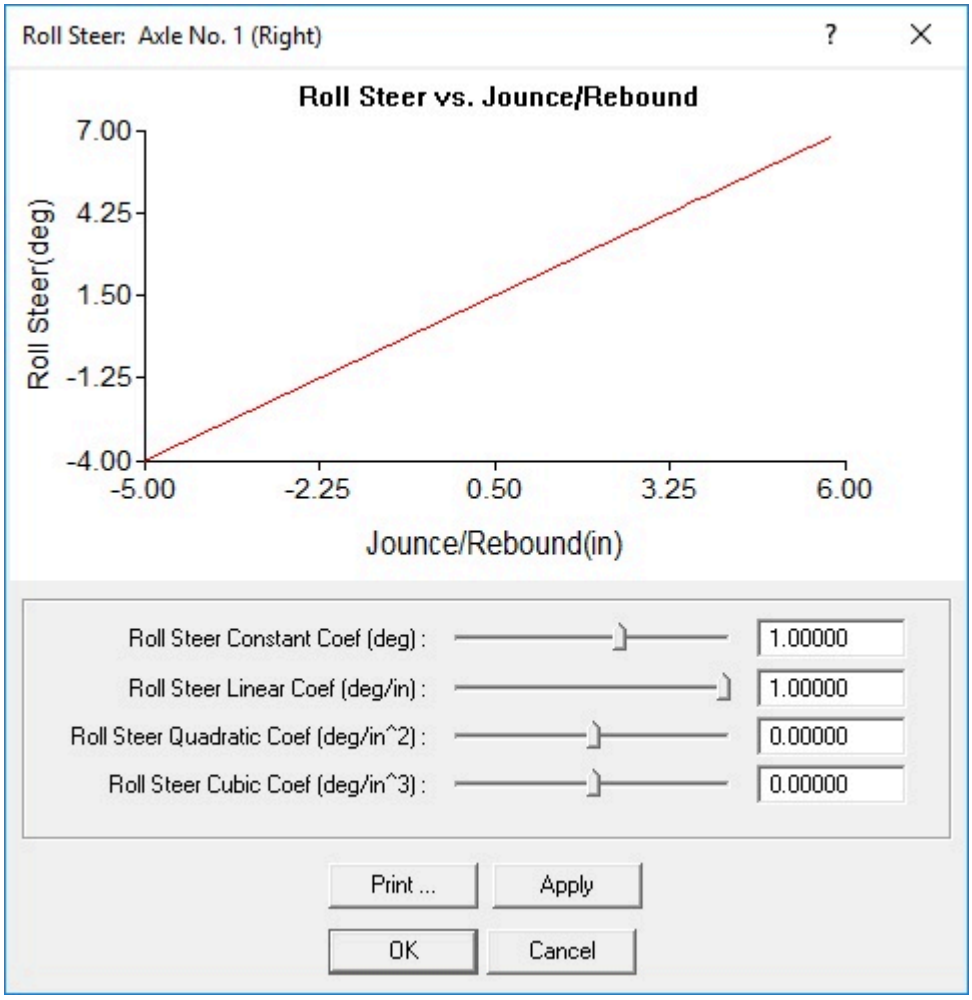


Figure 11-34: Roll Steer dialog for independent suspensions.

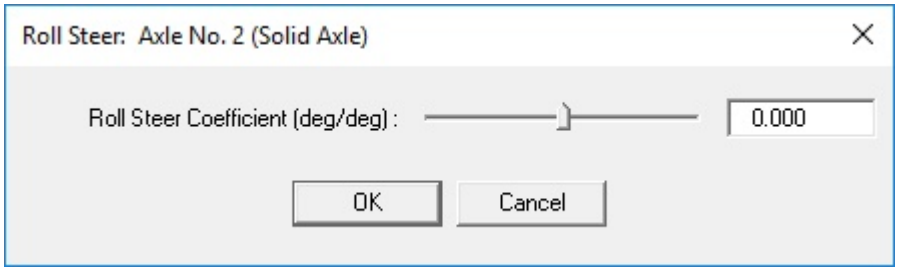


Figure 11-35: Roll Steer dialog for solid axle suspensions.

To display or edit the roll steer parameters, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Suspension* from the Unsprung Mass option list. The Suspension Data dialog is displayed.
3. Choose *Roll Steer*. The Roll Steer dialog is displayed.
4. View and/or edit the desired properties.
5. Choose *Apply* to graph the changes.
6. If desired, choose *Print* to print a graph (independent types).
7. Press *OK* to accept the changes. The Suspension Data dialog is still displayed.

8. If desired, click *Copy To Other Side* to make the changes apply to both sides.
9. Press *OK* to accept the suspension changes to the selected wheel position(s).

The Roll Steer parameters are described below.

- **Jounce/Rebound** — Suspension travel from equilibrium (see Figure 11-28). Jounce (compression) is negative; rebound (extension) is positive.
- **Roll Steer Coefficients** — For independent suspensions, the constant, linear, quadratic and cubic coefficients of the wheel steer angle vs jounce/rebound polynomial. For solid axle suspensions, a single roll steer coefficient: the ratio of wheel (axle) steer to body roll. *(The legacy manual repeated the Anti-pitch description here — a typo.)*

Table 11-18: Roll Steer Parameters

Parameter	Units (US / SI)	Description
Jounce/Rebound	in / cm	Suspension travel from equilibrium to Jounce (−) or Rebound (+) stop
Constant	deg	Initial steer angle at zero roll
Linear Coef	deg/in / deg/cm	Linear roll steer vs jounce/rebound
Quadratic Coef	deg/in² / deg/cm²	Quadratic roll steer vs jounce/rebound
Cubic Coef	deg/in³ / deg/cm³	Cubic roll steer vs jounce/rebound
Coefficient (Solid)	—	Ratio of wheel steer to body roll

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 11, pages 11-40..11-61 — revised for the current HVE release, 2026-07-05.

Chapter 11 — Vehicle Model Definition (Part D: Brakes, Tires and Wheels)

This part covers the unsprung-mass properties edited by clicking on a wheel in the Vehicle Viewer: the Brake Assembly at each wheel, the Tire properties, the Wheel Location and the Wheel Image.

Brake Assembly Parameters

The Brake Assembly properties for the selected wheel are displayed and edited using the Brake Assembly dialog. See also the reference page, Brake Assembly dialog.

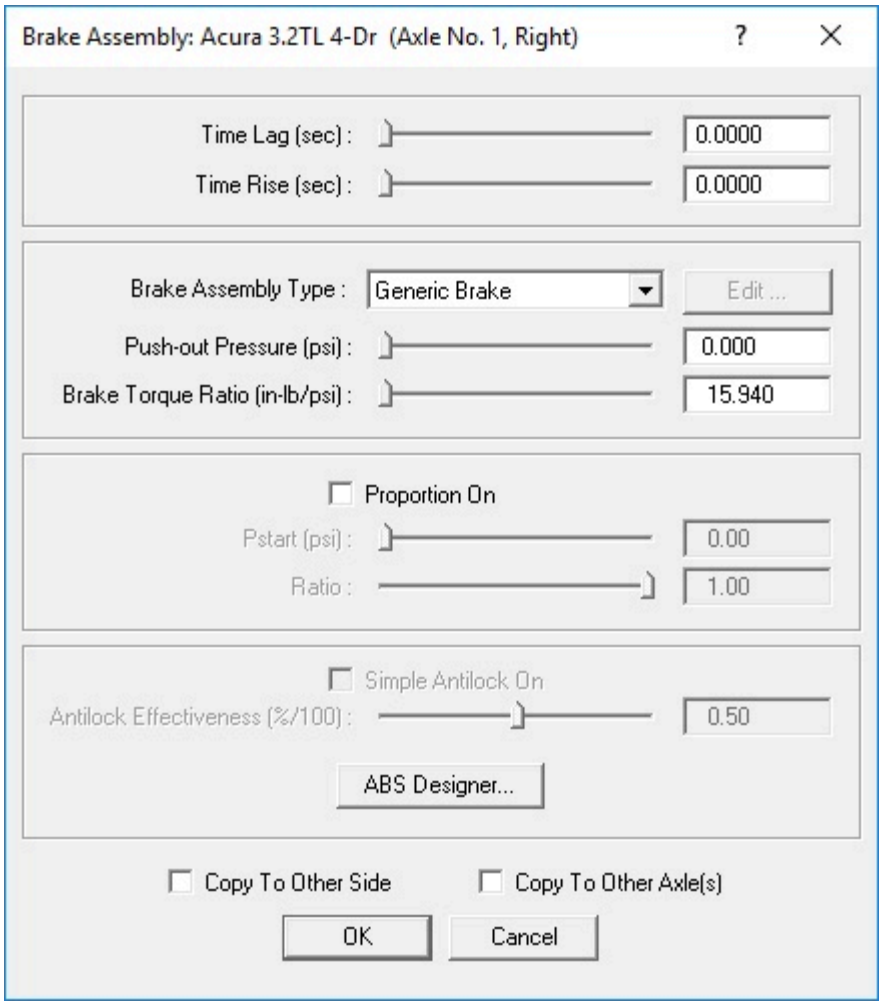


Figure 11-36: Brake Assembly dialog for editing the properties of the brake system at the selected wheel.

To display or edit the current vehicle's brake assembly parameters at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Brake* from the Unsprung Mass option list. The Brake Assembly dialog for the selected wheel is displayed.
3. View and/or edit the desired properties.
4. Press *OK* to accept the changes.

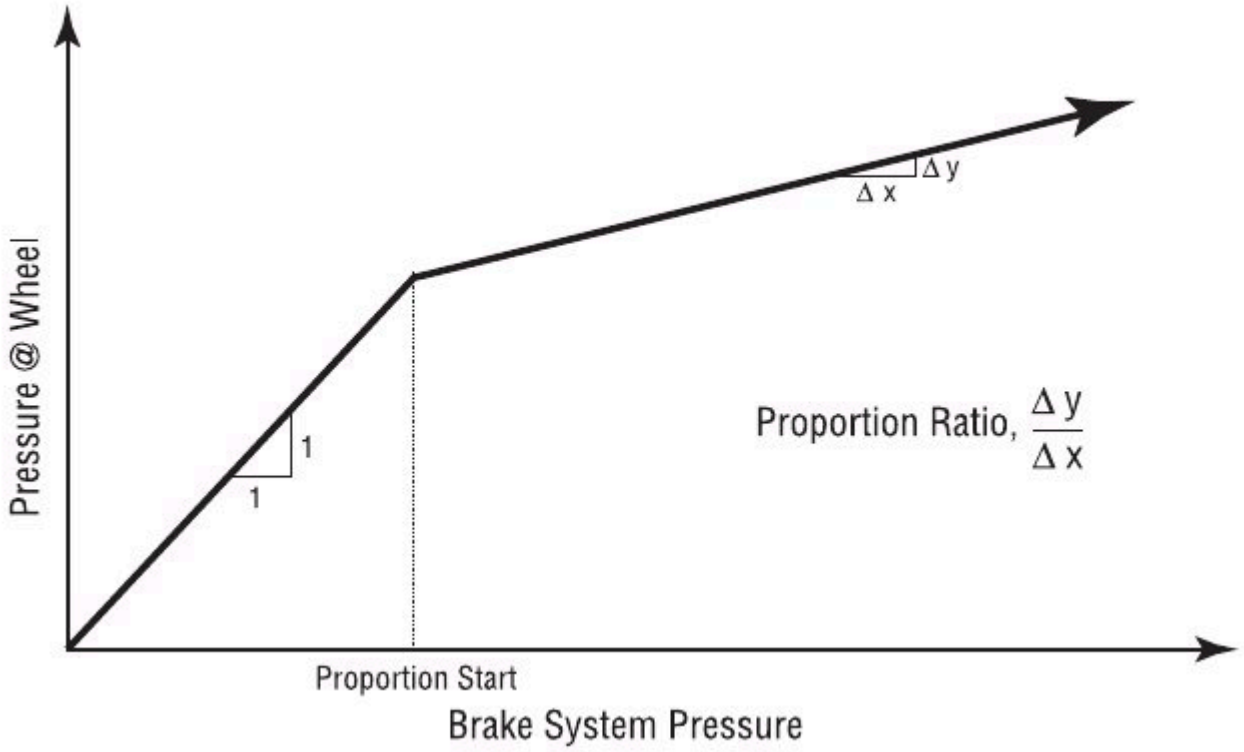


Figure 11-37: Brake System Proportioning.

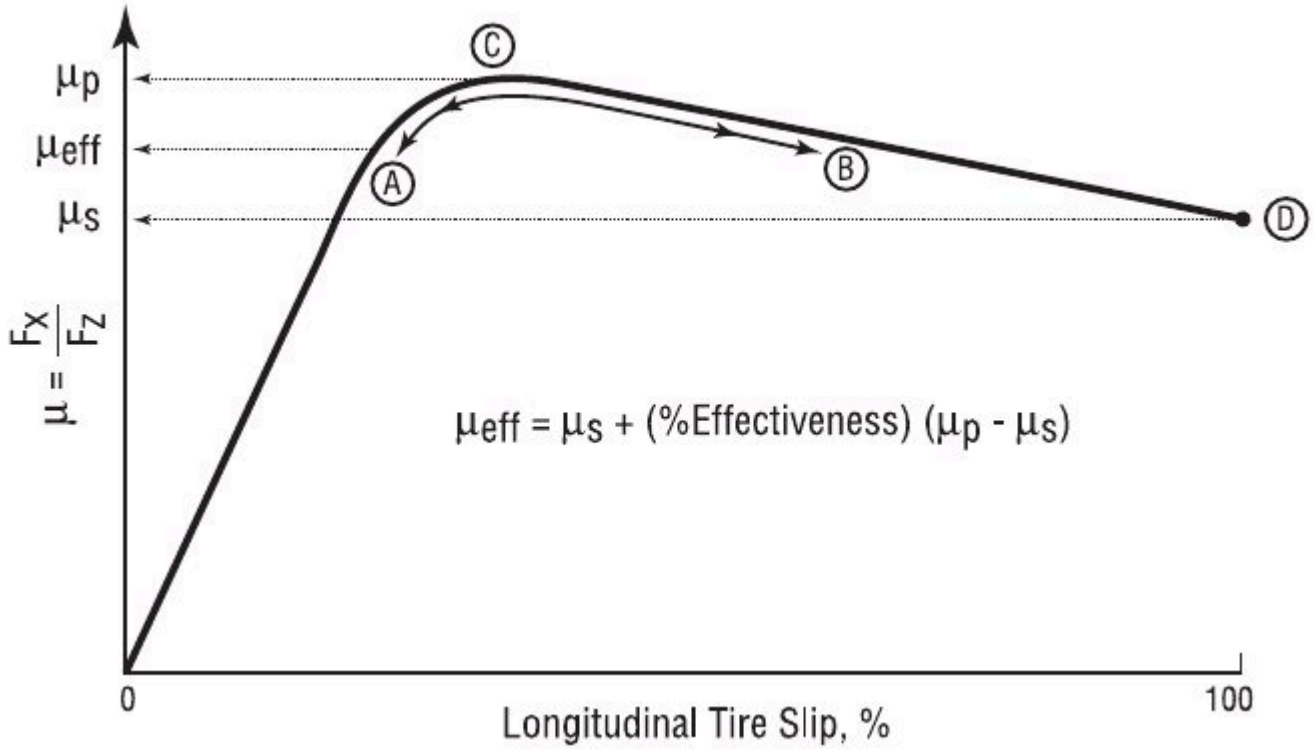


Figure 11-38: Simple Antilock Model.

The Brake properties at each wheel are described below.

- **Time Lag** — Time required before the driver's brake pedal inputs reach the wheel.
- **Time Rise** — Time required for the brake pressure at the wheel brake actuator to reach approximately 90 percent of full system pressure (pressure in the air chamber rises at an exponential rate).

> **NOTE:** If a simulation model does not use HVE's Time Lag or Time Rise parameters, the lag and rise time may be simulated in the Brake Table (see Event Editor, Driver Controls).

- **Brake Type** — An option list displaying the type of brake assembly used at the selected wheel location. The default is *Generic*; in this case, the Brake Torque Ratio (see below) is defined directly by the user. The other types are *Disc, Duo-Servo, Duplex, Single Piston, Dual Piston, S-Cam, Single Wedge, Dual Wedge* and *Air Disc (updated: Air Disc was added since the legacy manual)*.

> **NOTE:** See the HVE Brake Designer documentation for details on the use of this option. When a specific (non-Generic) brake design is selected, the *Edit Brake...* button opens the Brake Designer dialog for that assembly type, and the Push-out Pressure and Brake Torque Ratio are computed by the Brake Designer. *(updated: selecting Disc or Air Disc also sets the wheel brake assembly type for the axle to Disc, while the drum-based designs set it to Drum.)*

- **Push-out Pressure** — Brake system pressure required to begin causing brake torque (any system pressure below this point is simply taking out the slack in the system).
- **Brake Torque Ratio** — Wheel brake torque per unit of system pressure at the wheel. The actual braking torque at the wheel is the product of the current wheel system pressure and the Brake Torque Ratio.
- **Proportioning Check Box** — Click on this check box if the system pressure at the selected wheel is reduced by a proportioning valve.
- **Proportioning Starting Pressure** — Beginning at this level of system pressure, any additional master cylinder pressure will be reduced (see Figure 11-37).
- **Proportioning Ratio** — At master cylinder pressures above the Proportioning Starting Pressure, additional pressure is reduced by this ratio (see Figure 11-37).
- **Anti-lock Check Box** — Click on this check box if the selected wheel has a simple (functional) anti-lock device installed.
- **Anti-lock Efficiency** — A 100% effective anti-lock system maintains a longitudinal wheel force associated with the peak coefficient of friction. A 0% effective system maintains a longitudinal wheel force associated with the slide coefficient of friction. A 50% effective system would maintain a longitudinal force halfway between that associated with peak and slide friction (see Figure 11-38).
- **ABS Designer** — A pushbutton used to access the ABS Wheel Data dialog of the HVE Brake Designer *(updated: the ABS Designer edits detailed ABS wheel data — wheel kinematics thresholds such as minimum velocity, tire slip range and spin acceleration range, and pressure modulation parameters such as cycle rate, apply/release delays and apply/release rates)*.

> **NOTE:** The Simple Antilock option is not enabled if the vehicle is fitted with an ABS brake system. The ABS Designer pushbutton is not enabled if the brake system is not fitted with ABS (ABS is installed in the Brake System Pressure vs Pedal Force dialog). See Brake System Parameters in Part E.

Table 11-19: Wheel Brake Assembly Parameters

Parameter	Units (US / SI)	Description
Time Lag	sec	Time required for brake system pressure to reach the wheel
Time Rise	sec	Time required for the brake pressure at the wheel brake actuator to reach approximately 90 percent of full system pressure
Push-out Pressure	psi / kPa	Pressure required to begin braking
Brake Torque Ratio	—	Brake torque per unit of system pressure at the wheel
Proportioning	boolean (TRUE or FALSE)	Flag; if TRUE the wheel pressure is reduced by a proportioning valve
Pstart	psi / kPa	Brake system pressure at which proportioning begins for the selected wheel location
Proportioning Ratio	—	Ratio of pressure at the wheel to pressure at the master cylinder
Antilock	boolean (TRUE or FALSE)	Flag; TRUE if the wheel has an antilock device
Anti-lock Efficiency	%/100	Efficiency of the antilock system at the selected wheel

HVE Brake Designer

The HVE Brake Designer provides a detailed brake design capability integrated directly within the HVE simulation environment. This feature allows vehicle designers and safety researchers to develop a specific brake design for each wheel and evaluate the resulting vehicle performance using a predefined suite of maneuvers or compliance test simulations. The HVE Brake Designer also includes a detailed tool for simulating ABS.

The Brake Designer dialogs for the individual assembly types are documented in the reference pages: Disc Brake, Duo-Servo, Duplex, Single Piston, Dual Piston, Dual Wedge and Brake Material Properties.

Tire Properties

The tire parameters for the current vehicle are displayed and edited using the Tire Information dialog. See also the reference page, Tire Information dialog.

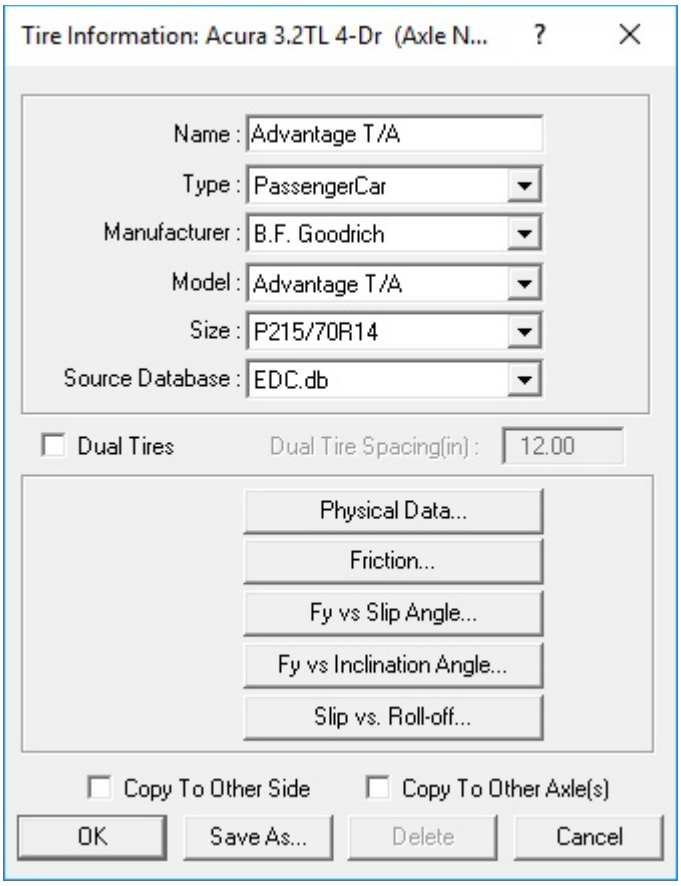


Figure 11-39: The Tire Information dialog is used for selecting and editing tires, as well as for maintaining the Tire Database.

The Tire Information dialog includes a user-extendible database that allows the user to create and edit tires according to the following database keys:

- Tire Type
- Tire Manufacturer
- Tire Model
- Tire Size
- Source Database

To display and possibly change the current tire at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed, showing the current tire's Name, Type, Manufacturer, Model and Size Designation.
3. If desired, change the tire's Name, Type, Manufacturer, Model and/or Size Designation by entering a new name or clicking on the appropriate option list and choosing a new option.

> **NOTE:** The Name is simply a user-editable field available for describing the tire. For example, you may choose a tire from the database and greatly reduce its cornering and camber stiffness to simulate a flat tire. In that case, you might enter the name as "Flat Generic Tire" to indicate its properties have been changed.

1. Press *OK* to accept the changes.

For the selected tire, the Tire Information dialog also allows the user to edit the following specific tire properties:

- Physical Properties
- Frictional Properties
- Cornering Stiffness Table (Fy vs Slip Angle)
- Camber Stiffness Table (Fy vs Inclination Angle)
- Slip vs Rolloff Tables

Each of these is described in the following sections.

> **NOTE:** For information about adding new tires to the Tire database, refer to Chapter 2 (see Creating Databases) and Appendix VIII (User Databases).
>
> **NOTE:** Contact EDC to obtain an interesting video that shows how these tire properties are determined from tests on Calspan's Flatbed Tire Testing machine. Ask for EDC Library Ref. No. 1077.

Tire Physical Data

The Tire Physical parameters for the selected wheel position are displayed and edited using the Tire Physical Data dialog. See also the reference page, Tire Physical Data dialog.

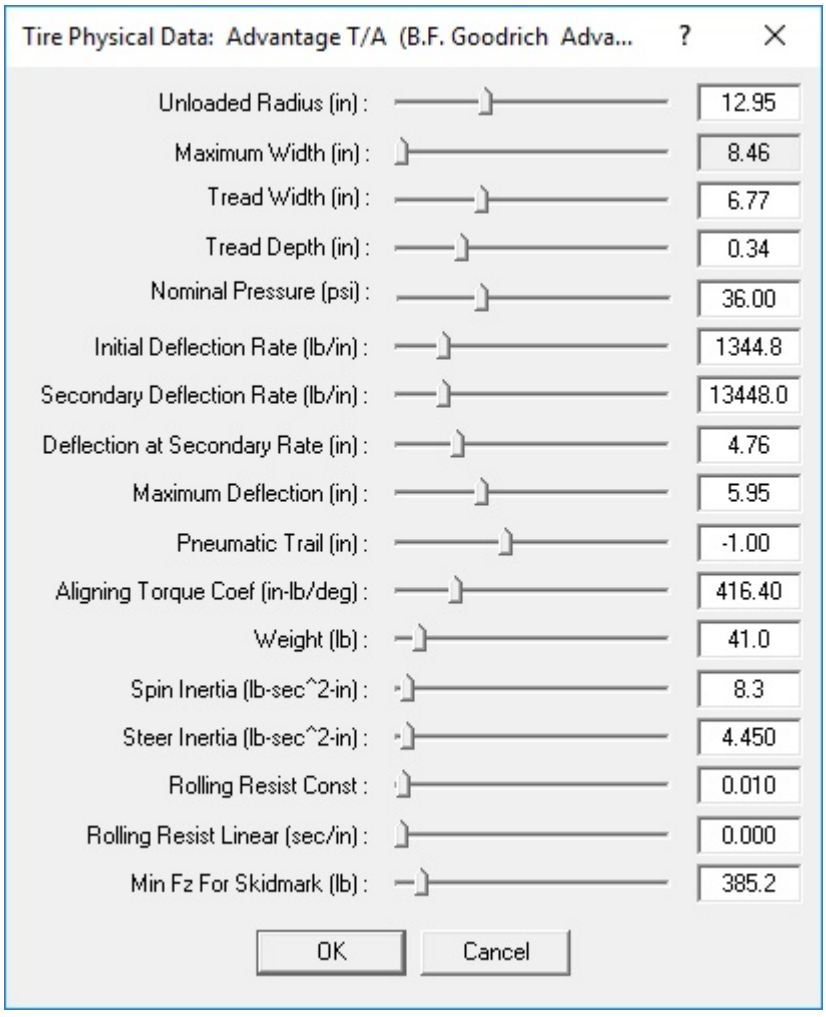


Figure 11-40: The Tire Physical Data dialog is used for editing the tire's physical parameters.

To display or edit the current vehicle's physical tire properties at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed.
3. If desired, change the tire by selecting a different Tire Type, Manufacturer, Model and/or Size.
4. Choose *Physical Data*. The Tire Physical Data dialog for the selected tire is displayed.
5. View and/or edit the desired properties.
6. Press *OK* to accept the changes. The Tire Information dialog is still displayed.
7. If desired, click *Copy To Other Side* to make the changes apply to both sides.
8. Press *OK* to accept the tire physical changes to the selected wheel position.

The Tire Physical Data parameters are described below.

- **Unloaded Radius** — Undeflected tire radius (no-load condition).
- **Maximum Width** — *(updated: added in Version 7 tire data.)* Maximum section width of the tire. This value is display-only; it is computed from the tire size designation string.
- **Tread Width** — *(updated: added in Version 7 tire data.)* Width of the tire tread contact patch.
- **Tread Depth** — *(updated: added in Version 7 tire data.)* Depth of the tire tread.
- **Nominal Pressure** — *(updated: added in Version 7 tire data.)* Nominal (rated) inflation pressure of the tire.
- **Initial Deflection Rate** — The initial tire spring rate (vertical tire load per unit of linear tire deflection). See Figure 11-41.
- **Secondary Deflection Rate** — The spring rate under conditions of excessive tire vertical load. See Figure 11-41.
- **Deflection At Secondary Rate** — Tire deflection at which the secondary deflection rate begins. See Figure 11-41.
- **Maximum Deflection** — Tire maximum deflection. This value is normally used to terminate execution, indicating the probability of a damaged or broken wheel rim. See Figure 11-41.
- **Pneumatic Trail** — Longitudinal distance from the center of the tire contact patch to the center of pressure. Pneumatic trail increases the self-aligning tendency of a steerable wheel (negative if the center of pressure is behind the center of contact).
- **Aligning Torque Stiffness** — Self-aligning torque due to mechanical and pneumatic trail.
- **Weight** — Total weight of the wheel assembly (tire plus rim). For an independent suspension, this value reflects the total unsprung mass. For a solid axle suspension system, the axle weight is added to the wheel weight to determine unsprung mass.

> **NOTE:** The entered value is divided by the current gravitational constant and stored as mass. That way, if you take the vehicle to the moon, it will behave correctly!

- **Spin Inertia** — Tire rotational inertia about its spin axis.
- **Steer Inertia** — Tire rotational inertia about its steering axis.
- **Rolling Resistance Constant** — Tire rolling resistance force coefficient.
- **Rolling Resistance Linear Coefficient** — Tire rolling resistance linear, velocity-dependent force coefficient.
- **Minimum Fz For Skid** — Minimum vertical tire load, Fz, required in order to leave a tire mark on the road surface. If the current vertical tire load is less than this value, no tire mark is displayed.

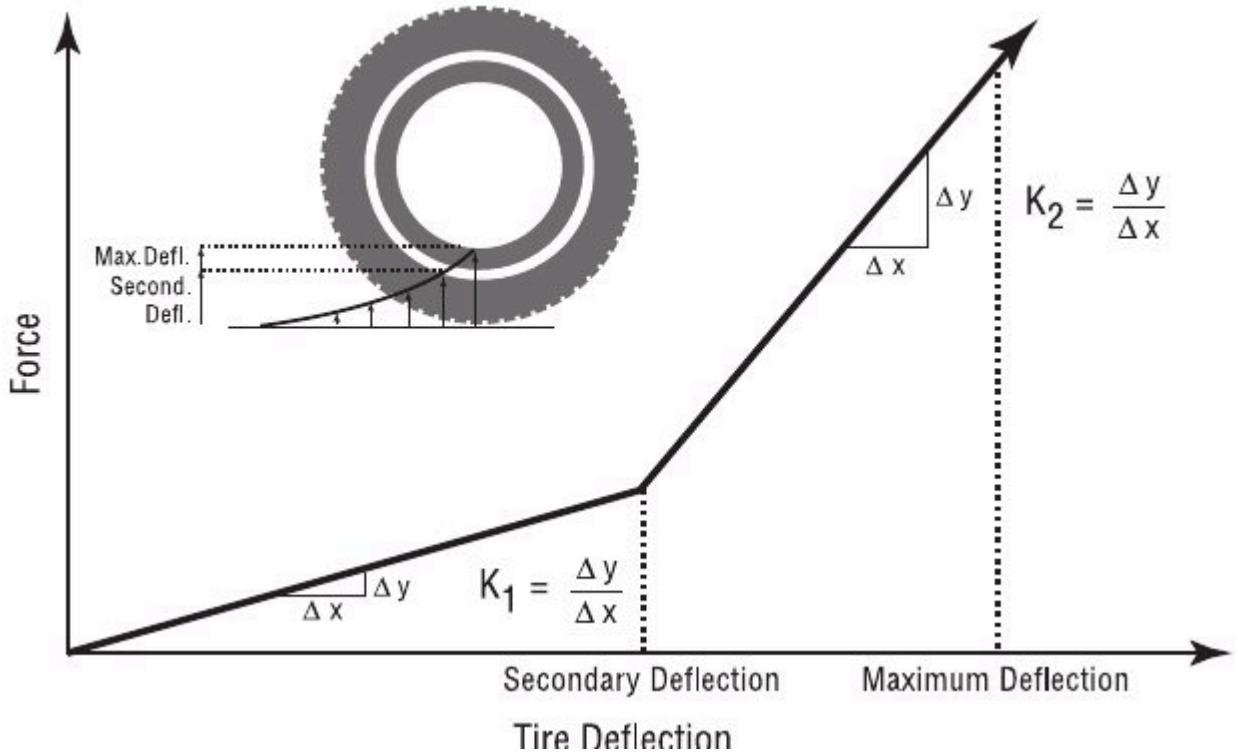


Figure 11-41: Tire Physical Parameters.

Table 11-20: Tire Physical Data Parameters

Parameter	Units (US / SI)	Description
Unloaded Radius	in / cm	Tire radius in unloaded condition
Maximum Width	in / cm	Maximum section width (computed from size designation) *(updated: Version 7)*
Tread Width	in / cm	Width of the tread contact patch *(updated: Version 7)*
Tread Depth	in / cm	Depth of the tire tread *(updated: Version 7)*
Nominal Pressure	psi / bar	Nominal inflation pressure *(updated: Version 7)*
Initial Deflection Rate	lb/in / N/cm	Deflection rate from zero deflection to secondary deflection
Secondary Deflection Rate	lb/in / N/cm	Deflection rate after preliminary deflection
Deflection At Secondary Rate	in / cm	Deflection at start of secondary deflection rate
Maximum Deflection	in / cm	Maximum allowable tire deflection
Pneumatic Trail	in / cm	Distance from center of the contact patch to the center of pressure
Aligning Torque Stiffness	in-lb/deg / N-m/deg	Steering torque produced by tire slip angle
Weight	lb / N	Total weight of wheel (tire + rim)
Spin Inertia	lb-sec²-in / kg-m²	Rotational inertia about spin axis
Steering Inertia	lb-sec²-in / kg-m²	Rotational inertia about steering axis
Rolling Constant	—	Ratio of rolling resistance force to total vertical force on tire
Rolling Linear Coef	sec/in / sec/m	Velocity-dependent tire rolling resistance force coefficient
Min Fz for Skidmark	lb / N	Minimum vertical tire force required for a tire to leave a skidmark

Tire Friction Data

The tire frictional properties for the selected wheel position are displayed and edited using the Tire Friction Data dialog. See also the reference page, Tire Friction Data dialog.

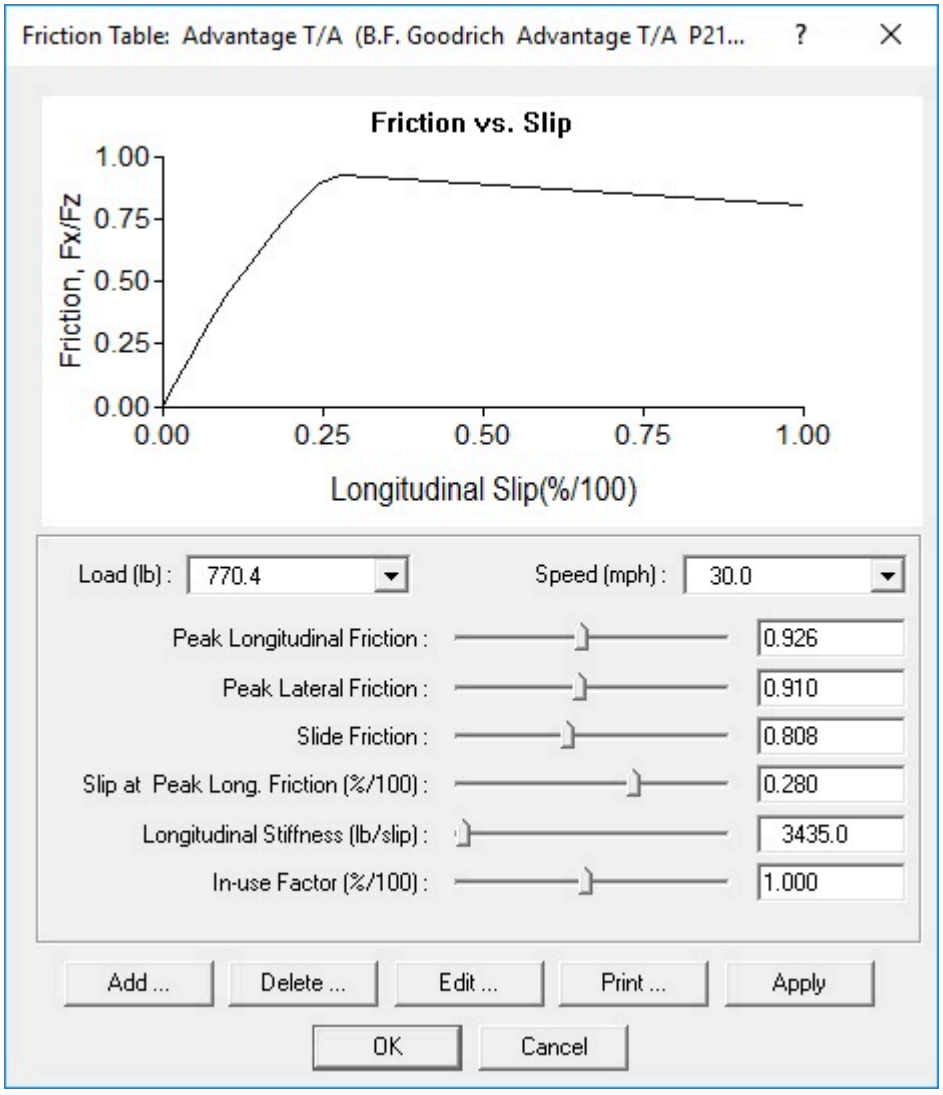


Figure 11-42: The Tire Friction Data dialog is used for editing the tire's frictional properties.

To display or edit the current vehicle's tire frictional properties at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed.
3. If desired, change the tire by selecting a different Tire Type, Manufacturer, Model and Size.
4. Click on *Friction Data*. The Tire Friction Data dialog for the selected tire is displayed.
5. Click on the *Load* combo box to view and/or edit the list of vertical loads (up to 3 loads may be supplied).
6. Click on the *Speed* combo box to view and/or edit the list of test speeds (up to 3 speeds may be supplied).
7. At the selected load and speed, view and/or edit the frictional properties and longitudinal stiffness of the tire.
8. Select an *In-use Factor* for the tire. The In-use Factor acts as a modifier that multiplies the current dependent tire data.

> **NOTE:** For example, let's say you believe a given tire's frictional properties are only 90 percent of the values displayed in the dialog. You could calculate 90 percent of the peak and slide friction and longitudinal stiffness values and edit the current data accordingly, or you could simply choose an in-use factor of 0.90.

1. Press *Apply* to update the graph of Fx/Fz vs Longitudinal Slip.
2. Press *Print* to print the graph on the system printer. Press *OK* to accept the changes. The Tire Information dialog is still displayed.
3. If desired, click *Copy To Other Side* to make the changes apply to both sides.
4. Press *OK* to accept the tire friction data changes to the selected wheel position.

The Tire Friction Data parameters are described below.

- **Load** — Vertical tire load at which the frictional values were obtained.
- **Speed** — Road speed at which the frictional values were obtained.
- **Peak Longitudinal Friction** — Maximum value of Fx/Fz achieved during the test at the given load and speed.
- **Peak Lateral Friction** — Maximum value of Fy/Fz achieved during the test at the given load and speed.
- **Slide Friction** — Value of Fx/Fz at 100% longitudinal slip achieved during the test at the given load and speed.
- **Slip at Peak Friction** — Longitudinal slip at which the peak longitudinal friction was achieved at the given load and speed.
- **Longitudinal Stiffness** — Slope of the Friction vs Slip curve (see Figure 11-42) at the graph's origin.

> **NOTE:** The graph uses the longitudinal stiffness to display the curve up to 0.5 times the Peak Longitudinal Friction.

- **In-use Factor** — Global multiplier for dependent values (Peak and Slide Friction, Longitudinal Stiffness).

Table 11-21: Tire Friction Parameters

Parameter	Units (US / SI)	Description
Load	lb / N	Vertical tire load, Fz, during test
Speed	in/sec / m/sec	Speed during test
Peak Longitudinal Friction	—	Maximum value of Fx/Fz achieved at given load and speed
Peak Lateral Friction	—	Maximum value of Fy/Fz achieved at given load and speed
Slide Friction	—	Value of Fx/Fz achieved at 100% slip
Slip at Peak Long. Friction	%/100	Longitudinal slip at which Peak Longitudinal Friction was measured
Longitudinal Stiffness	lb/slip / N/slip	Slope of Friction vs Slip graph at zero slip
In-use Factor	%/100	Multiplier for Peak and Slide Friction and Longitudinal Stiffness

Tire Fy vs Slip Angle Table

The Tire Fy vs Slip Angle Data (Cornering Stiffness) for the selected wheel position are displayed and edited using the Tire Fy vs Slip Angle Data dialog.

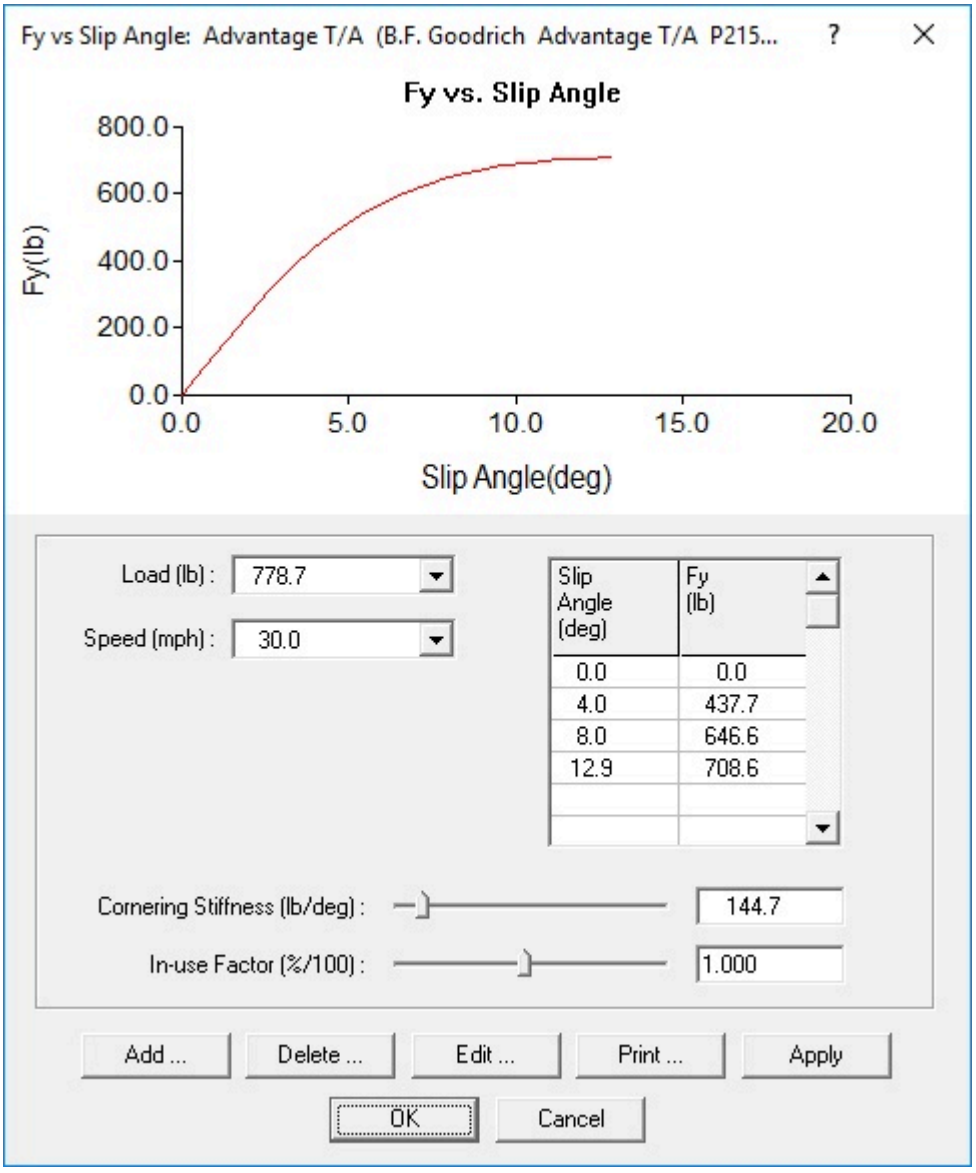


Figure 11-43: The Tire Fy vs Slip Angle Data dialog is used for editing the selected tire's lateral force producing properties.

To display or edit the current vehicle's cornering parameters at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed.
3. If desired, change the tire by selecting a different Tire Type, Manufacturer, Model and Size.
4. Choose *Fy vs Slip Angle Table*. The Fy vs Slip Angle Table dialog will be displayed.
5. Click on the *Load* combo box to view and/or edit the list of vertical loads (up to 3 loads may be supplied).
6. Click on the *Speed* combo box to view and/or edit the list of test speeds (up to 3 speeds may be supplied).
7. At the selected load and speed, view and/or edit the Fy vs Slip Angle Table using the scrollable listbox.
8. At the selected load and speed, view and/or edit the cornering stiffness of the tire.

> **NOTE:** The cornering stiffness defines the slope of the Fy vs Slip Angle curve at zero slip angle.

1. Select an *In-use Factor* for the tire. The In-use Factor acts as a modifier that multiplies the current dependent tire data.

> **NOTE:** For example, let's say you believe a given tire is partially deflated, resulting in a 50 percent reduction in cornering stiffness characteristics. You could calculate 50 percent of each Fy and Cornering Stiffness value, then edit the Fy vs Slip Angle Table and Cornering Stiffness. Alternatively, you could achieve the same result by simply choosing an in-use factor of 0.50.

1. Press *Apply* to update the graph of Fy vs Slip Angle.
2. Press *Print* to print the graph on the system printer.
3. Press *OK* to accept the changes. The Tire Information dialog is still displayed.
4. If desired, click *Copy To Other Side* to make the changes apply to both sides.
5. Press *OK* to accept the tire cornering stiffness parameter changes to the selected wheel position.

The Tire Fy vs Slip Angle Data parameters are described below.

- **Load** — Vertical tire load at which the cornering stiffness values were obtained.
- **Speed** — Road speed at which the cornering stiffness values were obtained.
- **Slip Angle** — Angle (relative to the tire axis system; see Appendix III) from the tire's forward axis to its local velocity vector.
- **Fy** — Lateral tire force (relative to the tire axis system) produced at the current slip angle for the given load and speed.
- **Cornering Stiffness** — Slope of the Fy vs Slip Angle curve (see Figure 11-43) at the graph's origin.

> **NOTE:** The graph uses the cornering stiffness to display the curve up to 0.5 times the first Slip Angle entry in the table.

- **In-use Factor** — Global multiplier for dependent values (Fy and Cornering Stiffness).

Table 11-22: Tire Fy vs Slip Angle Parameters

Parameter	Units (US / SI)	Description
Load	lb / N	Vertical tire load, Fz, during test
Speed	in/sec / m/sec	Speed during test
Slip Angle	deg	Angle (relative to the tire axis system; see Appendix III) from the tire's forward axis to its local velocity vector
Fy	lb / N	Lateral tire force (relative to the tire axis system) produced at the current slip angle for the given load and speed
Cornering Stiffness	lb/deg / N/deg	Slope of Fy vs Slip Angle graph at zero slip angle
In-use Factor	%/100	Multiplier for Fy and Cornering Stiffness

Tire Fy vs Inclination Angle Table

The Tire Fy vs Inclination Angle Data (sometimes referred to as Camber Stiffness) for the selected wheel position are displayed and edited using the Tire Fy vs Inclination Angle Data dialog. See also the reference page, Fy vs Inclination Angle dialog.

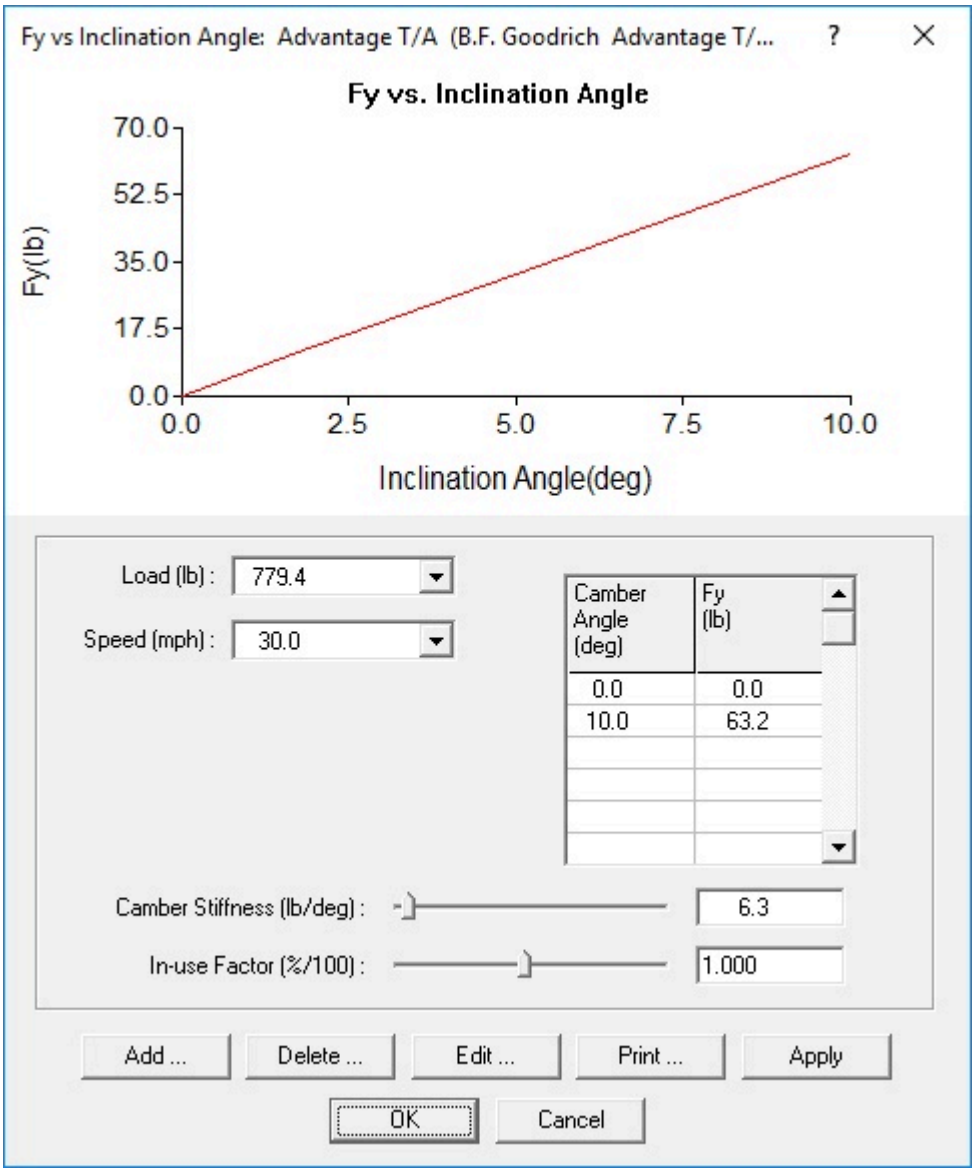


Figure 11-44: The Tire Fy vs Camber Angle dialog is used for editing the tire's camber stiffness properties.

> **NOTE:** As a rough estimate, camber stiffness is normally about 1/10 of the cornering stiffness.

To display or edit the current vehicle's Fy vs Camber Angle parameters at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed.
3. If desired, change the tire by selecting a different Tire Type, Manufacturer, Model and Size.
4. Choose *Fy vs Camber Angle Table*. The Fy vs Camber Angle dialog will be displayed.

5. Click on the *Load* combo box to view and/or edit the list of vertical loads (up to 3 loads may be supplied).
6. Click on the *Speed* combo box to view and/or edit the list of test speeds (up to 3 speeds may be supplied).
7. At the selected load and speed, view and/or edit the Fy vs Camber Angle Table using the scrollable listbox.
8. At the selected load and speed, view and/or edit the camber stiffness of the tire.

> **NOTE:** The camber stiffness defines the slope of the Fy vs Inclination Angle curve at zero inclination angle.

1. Select an *In-use Factor* for the tire. The In-use Factor acts as a modifier that multiplies the current dependent tire data.

> **NOTE:** For example, let's say you believe a given tire is partially deflated, resulting in a 50 percent reduction in camber stiffness characteristics. You could calculate 50 percent of each Fy and Inclination Angle value, then edit the Fy vs Inclination Angle Table and Camber Stiffness. Alternatively, you could achieve the same result by simply choosing an in-use factor of 0.50.

1. Press *Apply* to update the graph of Fy vs Camber Angle.
2. Press *Print* to print the graph on the system printer.
3. Press *OK* to accept the changes. The Tire Information dialog is still displayed.
4. If desired, click *Copy To Other Side* to make the changes apply to both sides.
5. Press *OK* to accept the tire parameter changes for the selected wheel position.

The Tire Fy vs Inclination Angle Data parameters are described below.

- **Load** — Vertical tire load at which the camber stiffness values were obtained.
- **Speed** — Road speed at which the camber stiffness values were obtained.
- **Inclination Angle** — Angle (relative to the tire axis system; see Appendix III) from the vertical axis of the tire plane to a vector normal to the road plane.
- **Fy** — Lateral tire force (relative to the tire axis system) produced at the current camber angle for the given load and speed.
- **Camber Stiffness** — Slope of the Fy vs Inclination Angle curve (see Figure 11-44) at the graph's origin.

> **NOTE:** The graph uses the camber stiffness to display the curve up to 0.5 times the first Inclination Angle entry in the table.

- **In-use Factor** — Global multiplier for dependent values (Fy and Camber Stiffness).

Table 11-23: Tire Fy vs Inclination Angle Parameters

Parameter	Units (US / SI)	Description
Load	lb / N	Vertical tire load, Fz, during test
Speed	in/sec / m/sec	Speed during test
Inclination Angle	deg	Angle (relative to the tire axis system; see Appendix III) from the vertical axis of the tire plane to a vector normal to the road plane
Fy	lb / N	Lateral tire force (relative to the tire axis system) produced at the current inclination angle for the given load and speed
Camber Stiffness	lb/deg / N/deg	Slope of Fy vs Inclination Angle graph at zero inclination angle
In-use Factor	%/100	Multiplier for Fy and Camber Stiffness

Tire Slip vs Rolloff Tables

The Tire Longitudinal and Lateral Slip vs Rolloff Data Tables for the selected wheel position are displayed and edited using the Tire Slip vs Rolloff Tables dialog. See also the reference page, Slip vs Rolloff dialog.

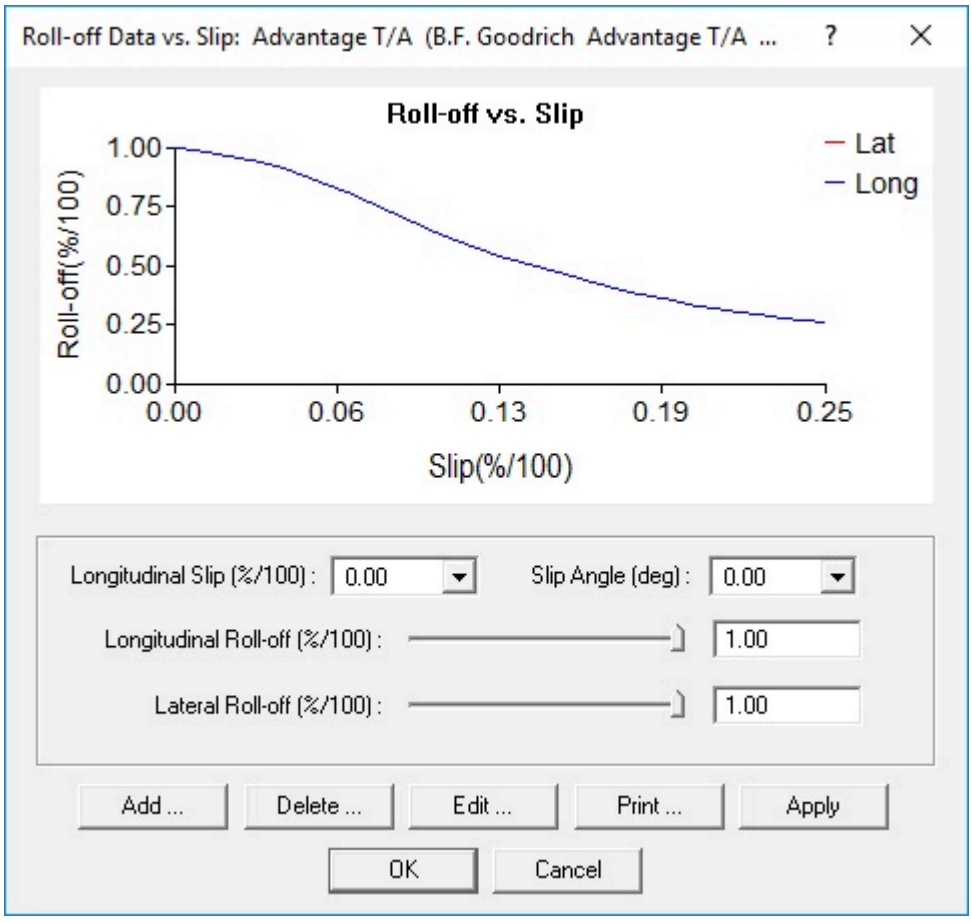


Figure 11-45: The Tire Longitudinal and Lateral Slip vs Rolloff Tables dialog is used for editing the tire's slip vs rolloff properties.

To display or edit the current vehicle's Slip vs Rolloff parameters at a selected wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Tire* from the Unsprung Mass option list. The Tire Information dialog for the selected wheel is displayed.
3. If desired, change the tire by selecting a different Tire Type, Manufacturer, Model and Size.
4. Choose *Slip vs Rolloff Table*. The Slip vs Rolloff Table dialog will be displayed.
5. Click on the *Longitudinal Slip* combo box to view and/or edit the list of longitudinal slips (up to 6 slip values may be supplied).
6. Click on the *Slip Angle* combo box to view and/or edit the list of lateral slip angles (up to 6 slip angles may be supplied).
7. At the selected longitudinal and lateral slips, view and/or edit the Longitudinal and Lateral Rolloffs.
8. Press *Apply* to update the graphs of Rolloff vs Slip.
9. Press *Print* to print the graph on the system printer.
10. Press *OK* to accept the changes. The Tire Information dialog is still displayed.
11. If desired, click *Copy To Other Side* to make the changes apply to both sides.
12. Press *OK* to accept the tire slip vs rolloff parameter changes to the selected wheel position.

The Tire Slip vs Rolloff Data parameters are described below.

- **Longitudinal Slip** — Longitudinal tire slip at which the lateral roll-off values were obtained.
- **Slip Angle** — Lateral tire slip (slip angle) at which the longitudinal roll-off values were obtained.
- **Longitudinal Rolloff** — Reduction in lateral tire force (relative to the tire axis system; see Appendix III) at the specified longitudinal slip.
- **Lateral Rolloff** — Reduction in longitudinal tire force (relative to the tire axis system) at the specified lateral slip (slip angle).

Table 11-24: Tire Slip vs Rolloff Parameters

Parameter	Units (US / SI)	Description
Longitudinal Slip	%/100	Longitudinal tire slip at which the lateral roll-off values were obtained
Slip Angle	deg	Lateral tire slip (slip angle) at which the longitudinal roll-off values were obtained
Longitudinal Rolloff	%/100	Reduction in lateral tire force (relative to the tire axis system; see Appendix III) at the specified longitudinal slip
Lateral Rolloff	%/100	Reduction in longitudinal tire force (relative to the tire axis system) at the specified lateral slip (slip angle)

Wheel Location

The wheel location coordinates relative to the vehicle-fixed coordinate system are displayed and edited using the Wheel Location dialog. See also the reference page, Wheel Location dialog.

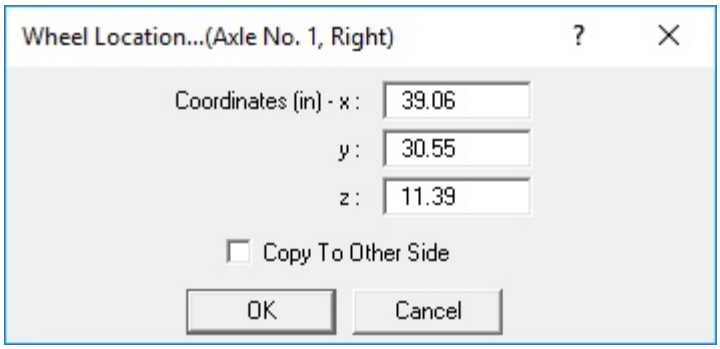


Figure 11-46: The Wheel Location dialog allows the user to view and edit the wheel's x,y,z vehicle-fixed coordinates.

To display and possibly change the wheel location coordinates, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.
2. Choose *Location* from the Unsprung Mass option list. The Wheel Location dialog for the selected wheel is displayed, showing the current wheel's x,y,z vehicle-fixed coordinates.
3. View and/or edit the wheel's x,y,z coordinates.
4. Click *Copy To Other Side* if you wish to apply the edits to the other side's wheel location, making the vehicle bilaterally symmetrical.

> **NOTE:** The resulting x- and z-coordinates for the copied wheel will be the same; the y-coordinates will have the same magnitude but the opposite sign.

1. Press *OK* to accept the changes to the wheel location data.

> **NOTE:** If you wish to move the CG, you can edit each individual wheel's x,y,z coordinates. However, an alternative (and easier) way is to use the Move CG dialog (see Sprung Mass Parameters in Part A).

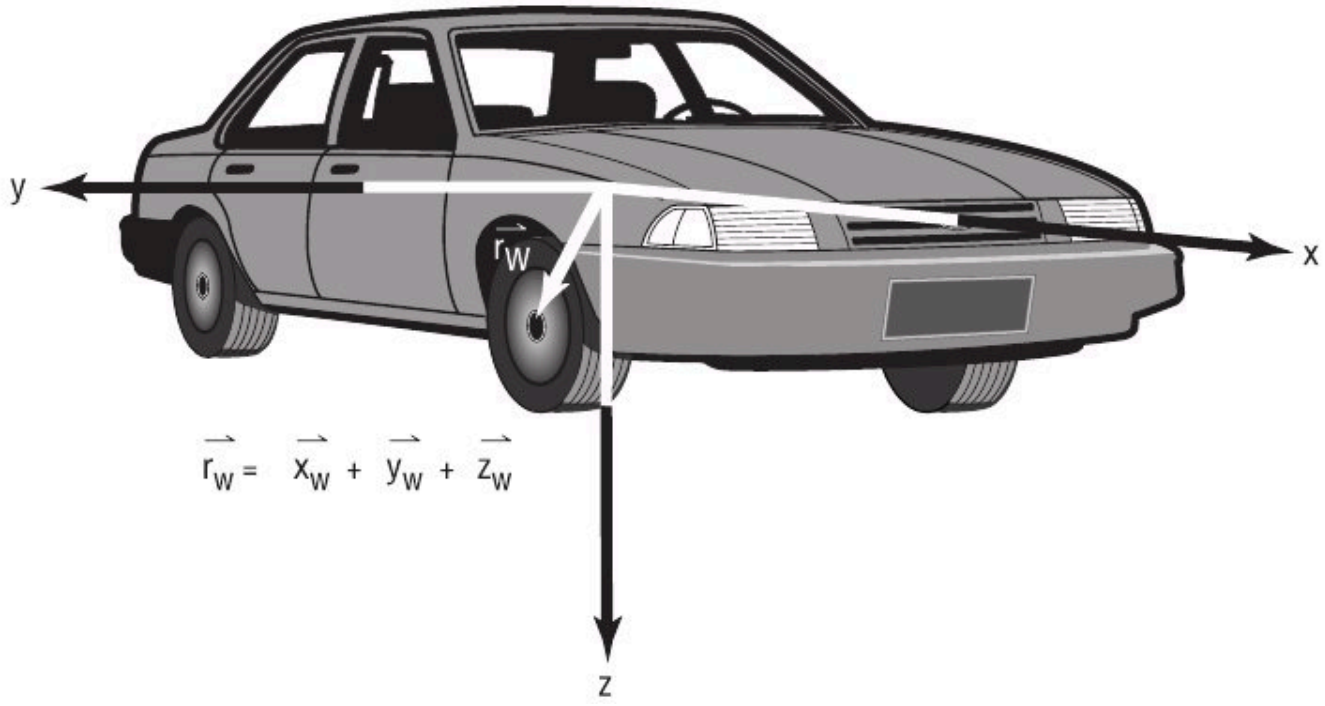


Figure 11-47: Wheel locations relative to the vehicle-fixed coordinate system.

The Wheel Location parameters are described below (see Figure 11-47).

- **Wheel x** — The longitudinal distance from the vehicle CG to the selected wheel.
- **Wheel y** — The lateral distance from the vehicle CG to the selected wheel.
- **Wheel z** — The vertical distance from the vehicle CG to the selected wheel.

Table 11-25: Wheel x,y,z Location Parameters

Parameter	Units (US / SI)	Description
x, y, z coordinates	in / cm	Vehicle-fixed x,y,z coordinates of the selected wheel relative to the sprung mass CG

Wheel Image

The vehicle's wheels may be visualized as either a gray disk or using a bitmap (texture). In the latter case, a photograph of the wheels may be taken and used to provide a very realistic representation of the vehicle's wheels.

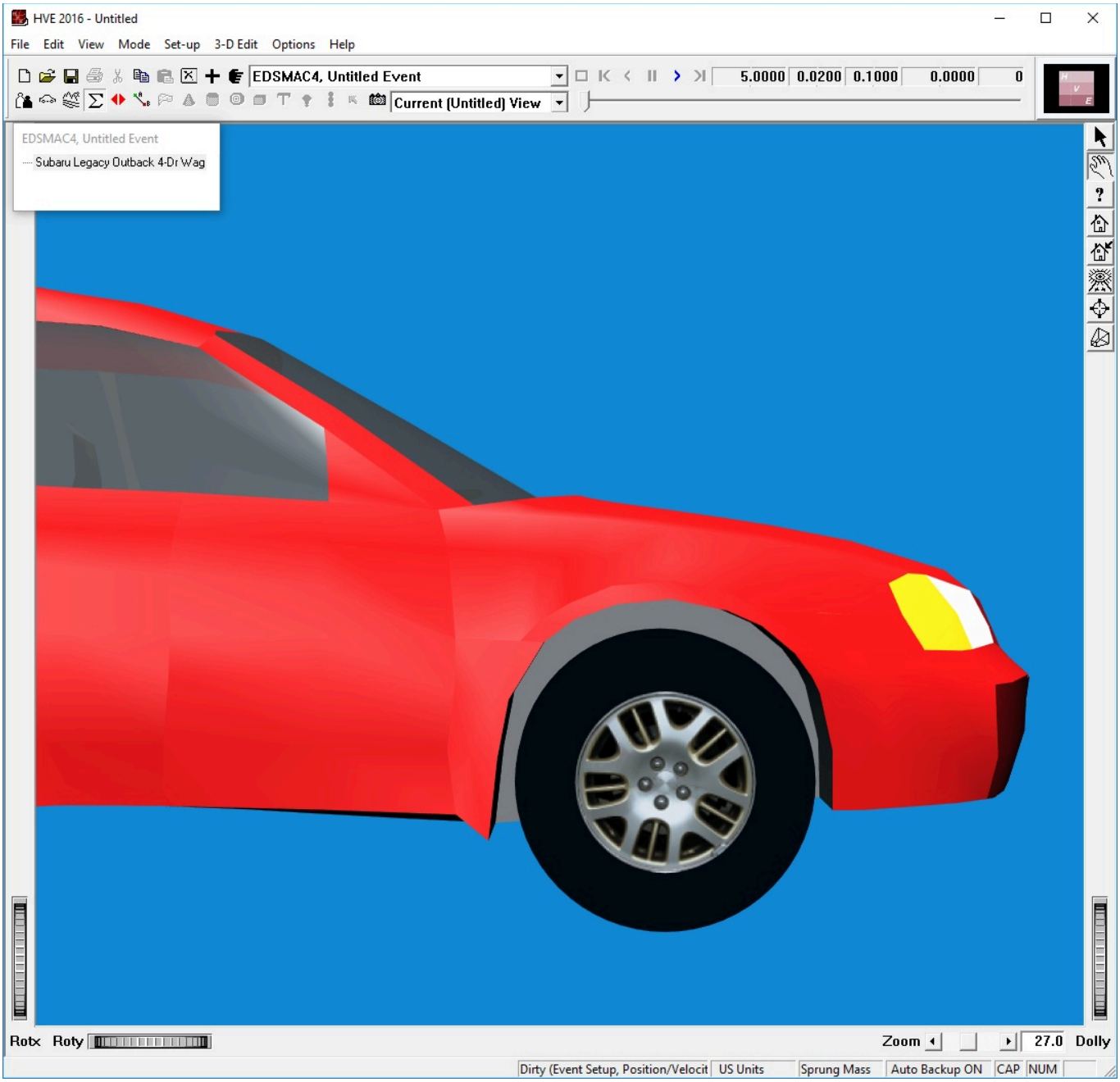


Figure 11-48: Typical wheel image.

When *Wheel Image* is selected from the Wheel option list, HVE displays a cascade menu containing two more options: *New* and *Open*.

> **NOTE:** HVE includes many wheel image files for your use. If you wish to supply your own wheel image file, you must place it in the `...\Images\Vehicles\WheelTextures` subdirectory.

New

The

New

option removes the existing wheel bitmap (if any) and replaces it with the default gray disk.

> **NOTE:** Choose this option if you accidentally select the wrong wheel image file and need to remove it.

To remove the existing wheel image file, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Wheel options for the selected wheel will be displayed.
2. Choose *Wheel Image* from the Wheel option list. The Wheel Image cascade menu will be displayed.
3. Choose *New*.

Any existing wheel image is removed from the selected wheel.

Open

The

Open

option displays the Wheel Image File Selection dialog and allows the user to assign a photographic image (texture map) of the wheel.

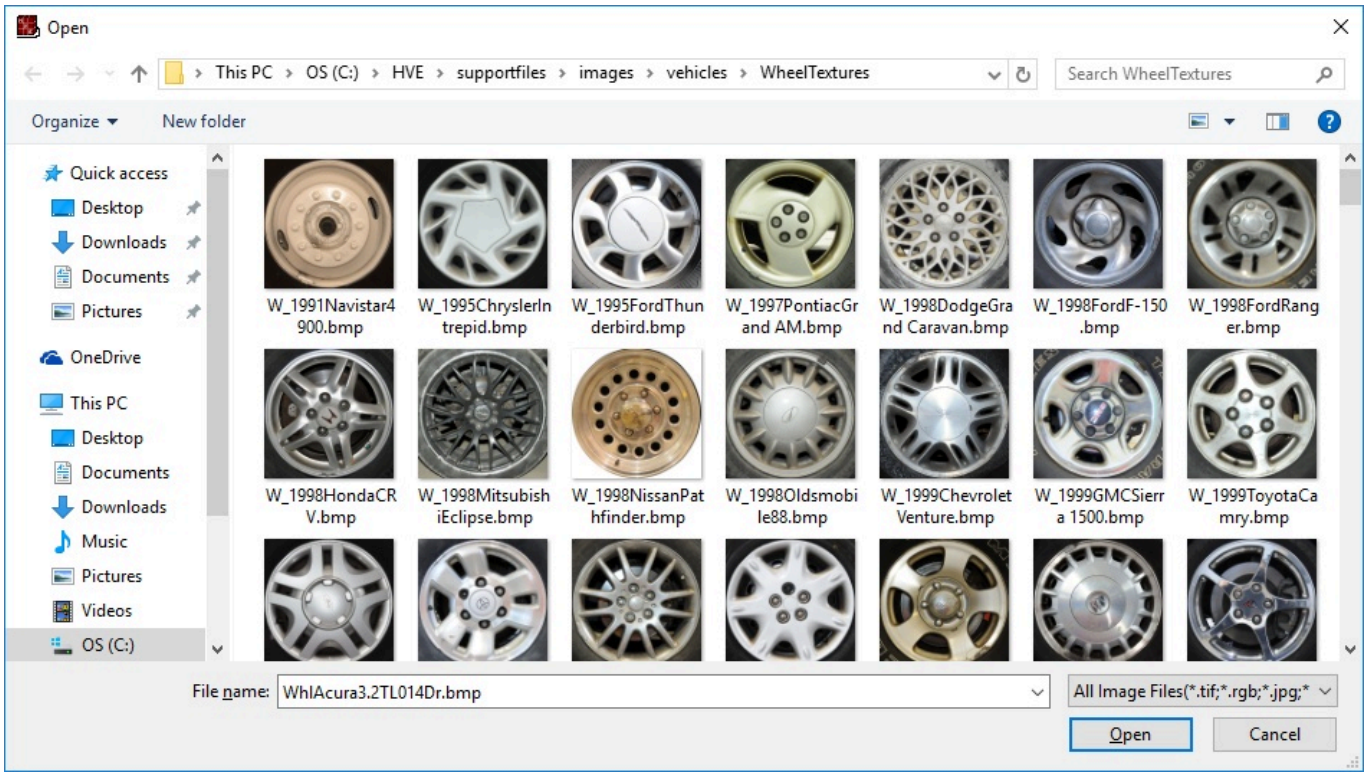


Figure 11-49: Image File Selection dialog, used for selecting wheel texture images.

To open a new wheel image and assign it to the current wheel, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Wheel options for the selected wheel will be displayed.
2. Choose *Wheel Image* from the Wheel option list. The Wheel Image cascade menu will be displayed.
3. Choose *Open*. The Wheel Image File Selection dialog will be displayed.
4. Click on the *Format* option list to select the file format of the desired wheel image file.
5. Select a wheel image file from the list.
6. Press *OK*.

The selected wheel image is displayed on the selected wheel. Perform these steps for each wheel.

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 11, pages 11-62..11-86 — revised for the current HVE release, 2026-07-05.

Chapter 11 — Vehicle Model Definition (Part E: Drivetrain, Steering and Brake System)

This part covers the vehicle-level systems edited by clicking on the Engine icon (Drivetrain), the Steering Wheel icon (Steering System) and the Brake Pedal/Master Cylinder icon (Brake System) in the Vehicle Viewer.

Drivetrain

The Drivetrain parameters for the current vehicle are displayed and edited using the Drivetrain dialog. The drivetrain is composed of three components that are edited individually:

- Engine
- Transmission
- Differential

Each of these components is described below. See also the reference page, Drivetrain dialog.

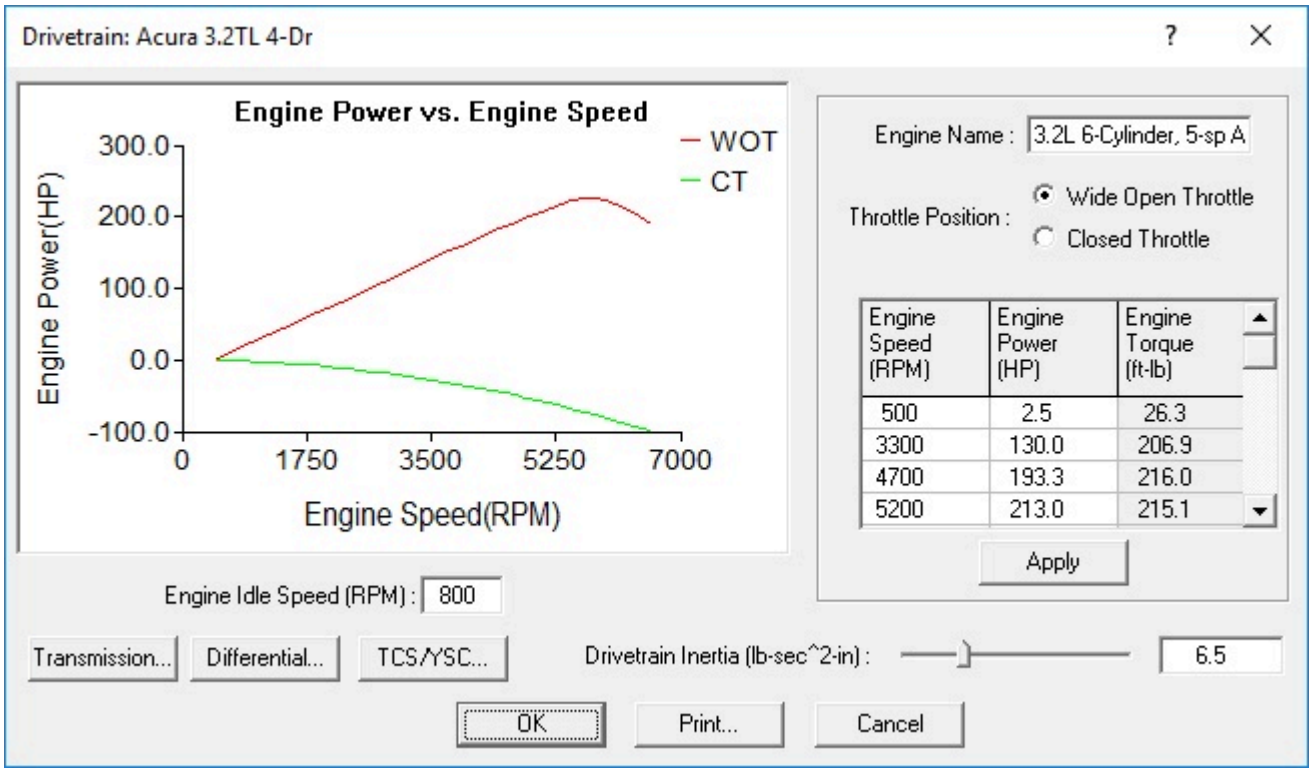


Figure 11-50: The Vehicle Drivetrain dialog, including the Engine, Transmission and Differential.

Engine

The HVE Vehicle engine is modeled by power vs engine speed table data for both a wide-open-throttle (WOT) condition and a closed-throttle (CT) condition.

To display or edit the current vehicle's engine parameters, perform the following steps:

1. In the Vehicle Viewer, click on the engine. The Drivetrain dialog will be displayed. The power vs engine speed curve is displayed, showing two curves: one for the WOT condition and the other for the closed-throttle condition.
2. To edit the WOT engine parameters, click the *WOT Throttle Position* radio button. The WOT Power vs Speed Table is displayed. View and/or edit the engine speed and power entries (up to 15 are allowed).

> **NOTE:** Torque is calculated, but not editable.

1. To edit the closed-throttle engine parameters, click the *Closed Throttle Position* radio button. The Closed Throttle Power vs Speed Table is displayed. View and/or edit the engine speed and power entries (up to 15 are allowed).
2. Press *Apply* to update the graphs of power vs engine speed.
3. Press *Print* to print the graph on the system printer.
4. Press *OK* to accept the changes to the current engine properties.

The Engine parameters are described below.

- **Engine Name** — User-editable description of drivetrain components.
- **Throttle Position** — Switch between WOT and Closed Throttle positions.
- **Engine Speed** — Rotational velocity of the crankshaft.
- **Engine Power** — Rate of work performed by the engine at the current engine speed.
- **Engine Torque** — Rotational torque produced at the current engine speed (calculated; not editable).
- **Drivetrain Inertia** — Rotational inertia of drivetrain components.
- **Engine Idle Speed (RPM)** — *(updated: new option, not documented in the legacy manual.)* Idle speed of the engine.
- **TCS/YSC...** — *(updated: new option, not documented in the legacy manual.)* Displays the Electronic Stability System dialog, used to indicate whether Traction Control (TCS) and/or Yaw Stability Control (YSC) are installed, and to edit their activation thresholds (velocity, yaw velocity error, wheel spin velocity error, wheel side-to-side difference, axle-to-axle difference, drive torque and brake pressure) and activation rates (drive torque change and brake pressure change).

Table 11-26: Engine Parameters

Parameter	Units (US / SI)	Description
Engine Name	—	Description of drivetrain components
Throttle Position	—	Switch for Wide-Open-Throttle (WOT) and Closed Throttle engine tables
Engine Speed	RPM	Engine rotational velocity
Engine Power	HP / kW	Engine power
Engine Torque	ft-lb / N-m	Engine torque (calculated; not editable)
Drivetrain Inertia	lb-sec²-in / kg-m²	Rotational inertia of drivetrain components

Transmission

The HVE Vehicle transmission is modeled by a reverse gear, neutral and up to 12 forward gears. The transmission is edited using the Transmission dialog. See also the reference page, Transmission Data dialog.

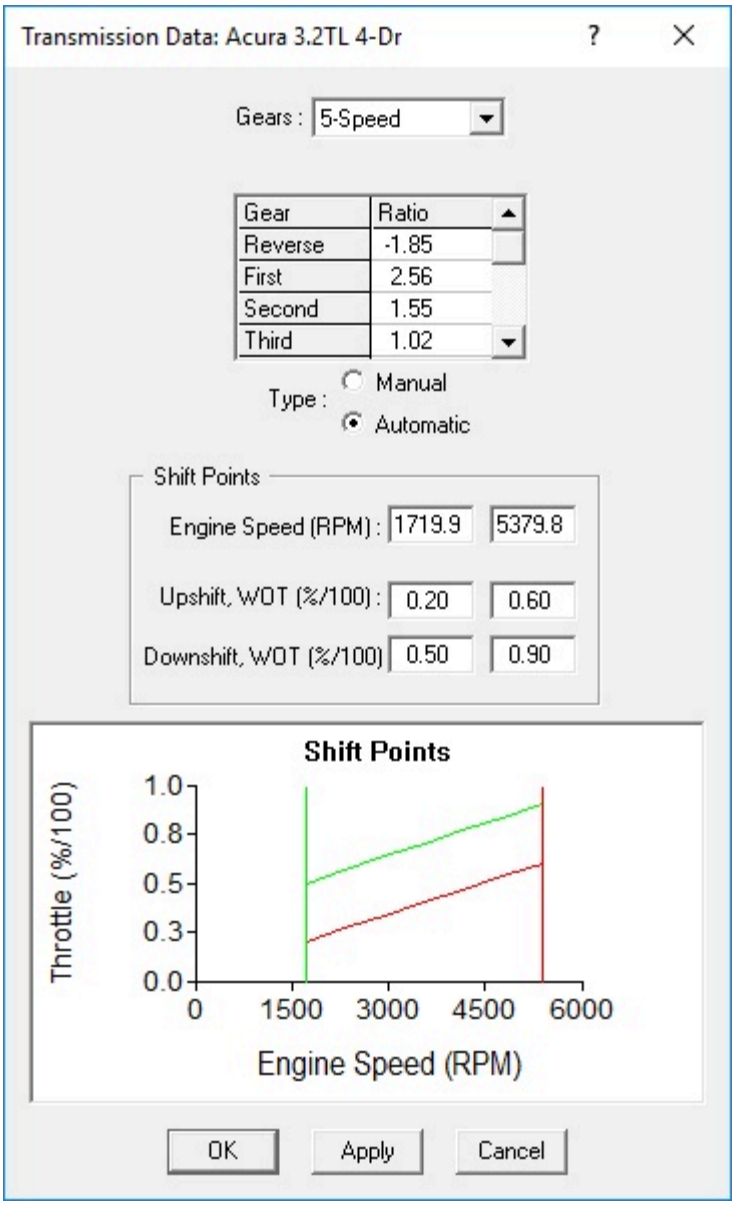


Figure 11-51: The Transmission Data dialog allows the user to edit the transmission ratios.

To display or edit the current vehicle's transmission parameters, perform the following steps:

1. In the Vehicle Viewer, click on the Engine icon. The Drivetrain dialog is displayed.
2. Click on *Transmission*. The Transmission dialog is displayed, showing the current number of forward gears and the table of gear ratios.
3. If desired, click on the *Gears* option list to change the number of forward gears.
4. View and/or edit the ratios for each gear.

> **NOTE:** A reverse gear is always supplied. A neutral gear, although not displayed, always exists and has no user-editable ratio.

1. Press *OK* to accept the changes to the current transmission. The Drivetrain dialog is still displayed.
2. Press *OK* on the Drivetrain dialog to accept the changes to the drivetrain.

The Transmission parameters are described below.

- **Forward Speeds** — Number of forward speeds in the current transmission (up to 12).
- **Gear Ratios** — Ratio for each gear.
- **Transmission Type** — *(updated: new option, not documented in the legacy manual.)* Radio buttons selecting *Standard* (manual) or *Automatic*. Selecting Automatic enables the Shift Points group, which defines the automatic transmission shift schedule (throttle position vs engine speed, with upshift and downshift lines between the low- and high-speed shift points; the shift schedule graph is redrawn after pressing Apply).

Table 11-27: Transmission Parameters

Parameter	Units (US / SI)	Description
Forward Speeds	—	Number of forward speeds in the current transmission
Transmission Gear	—	Ratio for each gear

Differential

The HVE Vehicle differential is modeled by up to 3 ratios. The differential is edited using the Differential dialog. See also the reference page, Differential Data dialog.

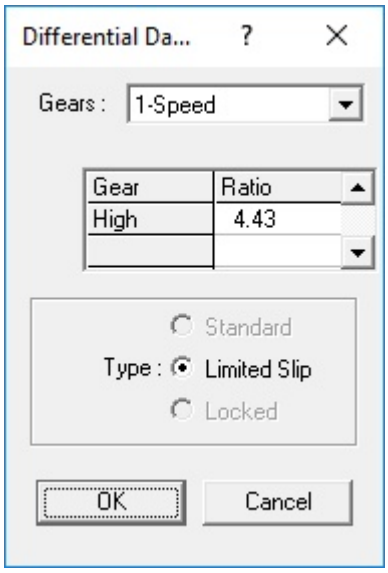


Figure 11-52: The Differential Data dialog allows the user to edit the differential ratios.

To display or edit the current vehicle's differential parameters, perform the following steps:

1. In the Vehicle Viewer, click on the Engine icon. The Drivetrain dialog is displayed.
2. Click on *Differential*. The Differential dialog is displayed, showing the current number of gears and the table of gear ratios.
3. If desired, click on the *Gears* option list to change the number of gears.
4. View and/or edit the ratios for each gear.
5. Press *OK* to accept the changes to the current differential. The Drivetrain dialog is still displayed.
6. Press *OK* on the Drivetrain dialog to accept the changes to the drivetrain.

The Differential parameters are described below.

- **Forward Speeds** — Number of speeds in the current differential (up to 3). *(The legacy manual said "transmission" here — a typo.)*
- **Gear Ratios** — Ratio for each gear.
- **Differential Type** — *(updated: the current dialog also shows Standard/Limited Slip/Locked radio buttons; these options are presently disabled — the HVE drivetrain model assumes an open (standard) differential, and Limited Slip and Locked are not yet implemented.)*

Table 11-28: Differential Parameters

Parameter	Units (US / SI)	Description
Forward Speeds	—	Number of forward speeds in the current differential
Differential Gear	—	Ratio for each gear

Steering System Parameters

The Steering System parameters for the current vehicle are displayed and edited using the Steering System dialog. See also the reference page, Steering System dialog.

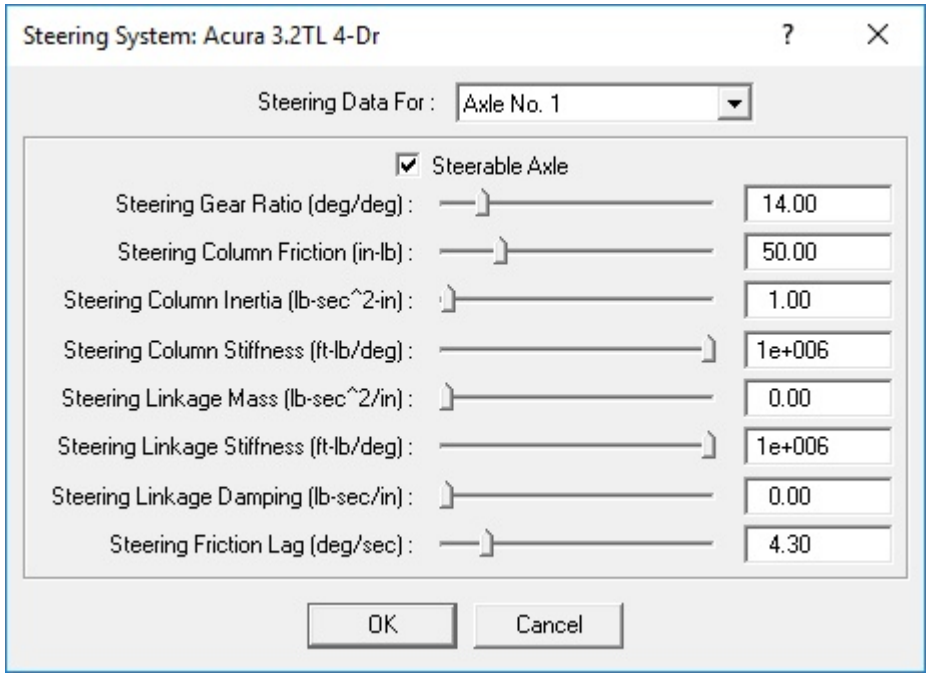


Figure 11-53: The Steering System dialog allows the user to choose which axles are steerable and to assign gear ratio(s) and mechanical properties for the selected steering linkage system.

To display or edit the current vehicle's steerable axles and associated parameters, perform the following steps:

1. In the Vehicle Viewer, click on the Steering Wheel icon. The Steering System dialog is displayed.
2. Click the *Current Axle* option list to choose the current axle.

> **NOTE:** The number of available axles is determined by the basic vehicle parameters defined in the Vehicle Information dialog.

1. If desired, click the *Is Steerable* check box to enable editing of the steering system parameters for the selected axle.
2. View and/or edit the desired properties.
3. Press *OK* to accept the changes.

The Steering System parameters are described below.

- **Axle Selector** — Axle number (1, 2 or 3, limited by the number of axles defined in the Vehicle Information dialog).
- **Is Steerable Check Box** — Switch to allow or disallow steering at the selected axle.
- **Gear Ratio** — Ratio of the steering wheel angle to the wheel angle.
- **Steering Column Friction** — Friction torque measured at the steering wheel.
- **Steering Column Inertia** — Rotational inertia of steering wheel and column.
- **Column Stiffness** — Torsional stiffness of the steering system between the steering wheel and the wheel on the driver's side.

> **NOTE:** The driver's side is defined in the Vehicle Information dialog.

- **Linkage Mass** — Mass (weight/g) of all translating linkage components.
- **Linkage Stiffness** — Torsional stiffness of the steering system between the driver's side wheel and the opposite-side wheel.
- **Steering Linkage Damping** — Linear damping of the linkage system.
- **Steering Friction Lag** — Rotational velocity required to develop steering friction torque.

Table 11-29: Steering System Parameters

Parameter	Units (US / SI)	Description
Axle Selector	n/a	Current axle number
Is Steerable	boolean (TRUE or FALSE)	Check box; if TRUE, the current axle is steerable
Gear Ratio	deg/deg	Ratio of the steering wheel angle to the wheel angle
Steering Column Friction	in-lb / N-m	Frictional torque in steering column
Steering Column Inertia	lb-sec²-in / kg-m²	Rotational inertia of steering wheel and column
Column Stiffness	ft-lb/deg / N-m/deg	Torsional stiffness of the steering system between the steering wheel and the wheel on the driver's side
Linkage Mass	lb-sec²/in / kg	Total mass of all translating linkage components
Linkage Stiffness	ft-lb/deg / N-m/deg	Torsional stiffness of the steering system between the driver's side wheel and the opposite-side wheel
Steering Linkage Damping	lb-sec/in / N-sec/m	Linear damping of linkage
Steering Friction Lag	deg/sec	Friction null band for full column friction

Brake System Parameters

The Brake System parameters for the current vehicle are displayed and edited using the Brake System dialog (Brake System Pressure vs Pedal Force). See also the reference page, Brake System Pressure vs Pedal Force dialog.

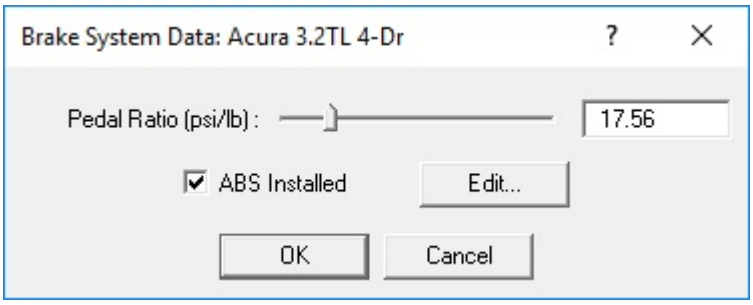


Figure 11-54: The Brake System dialog allows the user to assign the master cylinder pedal ratio (master cylinder pressure per unit of brake pedal force application) and ABS System status.

To display or edit the current vehicle's brake master cylinder pressure ratio, perform the following steps:

1. In the Vehicle Viewer, click on the Brake Pedal/Master Cylinder icon. The Brake System dialog is displayed.
2. View and/or edit the master cylinder pressure ratio.
3. If desired, click on the *ABS Installed* checkbox to activate the vehicle's antilock brake system.
4. If desired, press *Edit* to display and edit the vehicle's current ABS System parameters.
5. Press *OK* to accept the changes.

The Brake System parameters are described below.

- **Pedal Ratio** — Ratio of master cylinder pressure per unit of brake pedal force.

> **NOTE:** While using the Event Editor, Driver Controls, the user may assign a table of pedal force vs time. The entered pedal force is multiplied by the master cylinder pedal ratio to determine the current brake system pressure.
>
> **NOTE:** For airbrake systems, the Pedal Ratio is normally set equal to 1.0; thus, for an airbrake system, the pedal force is also equal to the treadle valve outlet pressure.

- **Brake System Type** — *(updated: new option, not documented in the legacy manual.)* Radio buttons selecting the type of brake system installed on the vehicle: *Hydraulic* or *Air*.
- **Brake Type on Each Axle** — *(updated: new group, not documented in the legacy manual.)* Displays and edits the brake assembly type (*Disc* or *Drum*) installed on each axle. This setting is also updated automatically when a specific brake design is chosen in the Brake Assembly dialog (Disc and Air Disc designs set it to Disc; drum-based designs set it to Drum). A warning message is issued when the dialog opens if the right and left brake types on an axle are different, as this is probably a mistake.
- **ABS Installed Checkbox** — Assigns ABS System parameters. Also enables the *Edit* pushbutton that displays the ABS System Data dialog and allows editing of the ABS System variables *(updated: system-level ABS parameters — control source, ABS algorithm, control method, cycle rate, ABS pressure threshold, ABS velocity threshold, low friction threshold, and delay parameters: delay method, apply delay and release delay).*

> **NOTE:** Clicking this checkbox also enables the ABS Designer option and the antilock options in the Wheel Brake Assembly dialog (see Part D, Figure 11-36).

Table 11-30: Brake System Parameters

Parameter	Units (US / SI)	Description
Pedal Ratio	psi/lb / kPa/N	Brake system pressure per unit of pedal force
ABS Installed	—	Flag indicating an ABS system is installed on the vehicle

HVE Brake Designer

The HVE Brake Designer provides a detailed brake design capability integrated directly within the HVE simulation environment. This feature allows vehicle designers and safety researchers to develop a specific brake design for each wheel and evaluate the resulting vehicle performance using a predefined suite of maneuvers or compliance test simulations. The HVE Brake Designer also includes a detailed tool for simulating ABS.

The Brake Designer is accessed from the Brake Assembly dialog (via the Brake Type option list and Edit Brake pushbutton — see Part D) and from the ABS Designer pushbutton (per-wheel ABS data) and the Brake System dialog's ABS Edit pushbutton (system-level ABS data). The individual Brake Designer dialogs are documented in the reference pages in Brakes and Powertrain: Disc Brake, Duo-Servo, Duplex, Single Piston, Dual Piston, Dual Wedge and Brake Material Properties. *(updated: the legacy manual described the Brake Designer in detail in its Section Eleven; the current brake types additionally include Air Disc.)*

Source: HVE User's Manual (Version 5, Seventh Edition, Jan 2006), Chapter 11, pages 11-87..11-98 — revised for the current HVE release, 2026-07-05.

Section Five — Environment Editor

The purpose of the HVE Environment Editor is to provide the physical and visual environmental attributes which may interact with humans and vehicles during the execution of a reconstruction or simulation model. This section of the HVE User's Manual describes how to create and edit these environment models. It also provides an overview of the environment model parameters.

The basic physical/mathematical model of the environment provides the following information:

- An atmosphere, that may be used for modeling aerodynamic forces on humans and vehicles.
- Fog, haze or smoke in the atmosphere, that may be used to study visibility attenuation.
- A sun that is positioned according to the local latitude, longitude and time zone, along with the time and date and angle of the earth-fixed coordinate system.
- The physical environment gravitational constant and surfaces that may be used to model interaction forces (i.e., tires and human and vehicle exteriors).

General Procedure

To use the Environment Editor, perform the following steps:

1. Select Environment Mode.
2. Add the physical environment, either by creating a new one or by selecting an environment from a previous HVE case.
3. Use the 3-D Editor to create the roads, curbs, sidewalks, trees, buildings and other objects with which the humans and vehicles may interact. These objects also provide an extremely important visual context for accident causation.
4. Import a scanned photograph of the environment, which may also be used to provide visual context.

Overview of Section Five

This section provides a detailed explanation of the process of creating and editing environments, and also presents a detailed overview of the environment model's physical properties:

- Chapter 12 — Creating & Editing Environments describes the process of creating and editing environments.
- Chapter 13 — Environment Model Definition provides a detailed overview of the HVE Environment Model.
- Chapter 14 (reserved — a RoadTools overview placeholder in the legacy edition).

Dialog references for this section are collected in the Environment Dialog Reference.

Chapter 12 — Creating & Editing Environments

The HVE Environment Editor is used for creating and editing environments for the current HVE case. These environments may also be used in other cases. This chapter describes how to create and edit environments, beginning with a description of the Environment Editor's components.

> **NOTE:** The environment is optional; none is required. If an environment is not supplied, the surface is assumed to be flat and cannot be visualized.
>
> **NOTE:** Refer to the next chapter, Environment Model Definition, for a detailed description of each Environment Model parameter.

Environment Editor Components

To use the Environment Editor, choose Environment Mode using the mode selector. This puts HVE in Environment Mode and allows access to the Environment Editor's components:

- **Environment Information Dialog** — The Environment Information dialog is used for adding an environment to the current case and for editing its parameters.
- **Environment Viewer** — The Environment Viewer is used for visualizing the current environment.

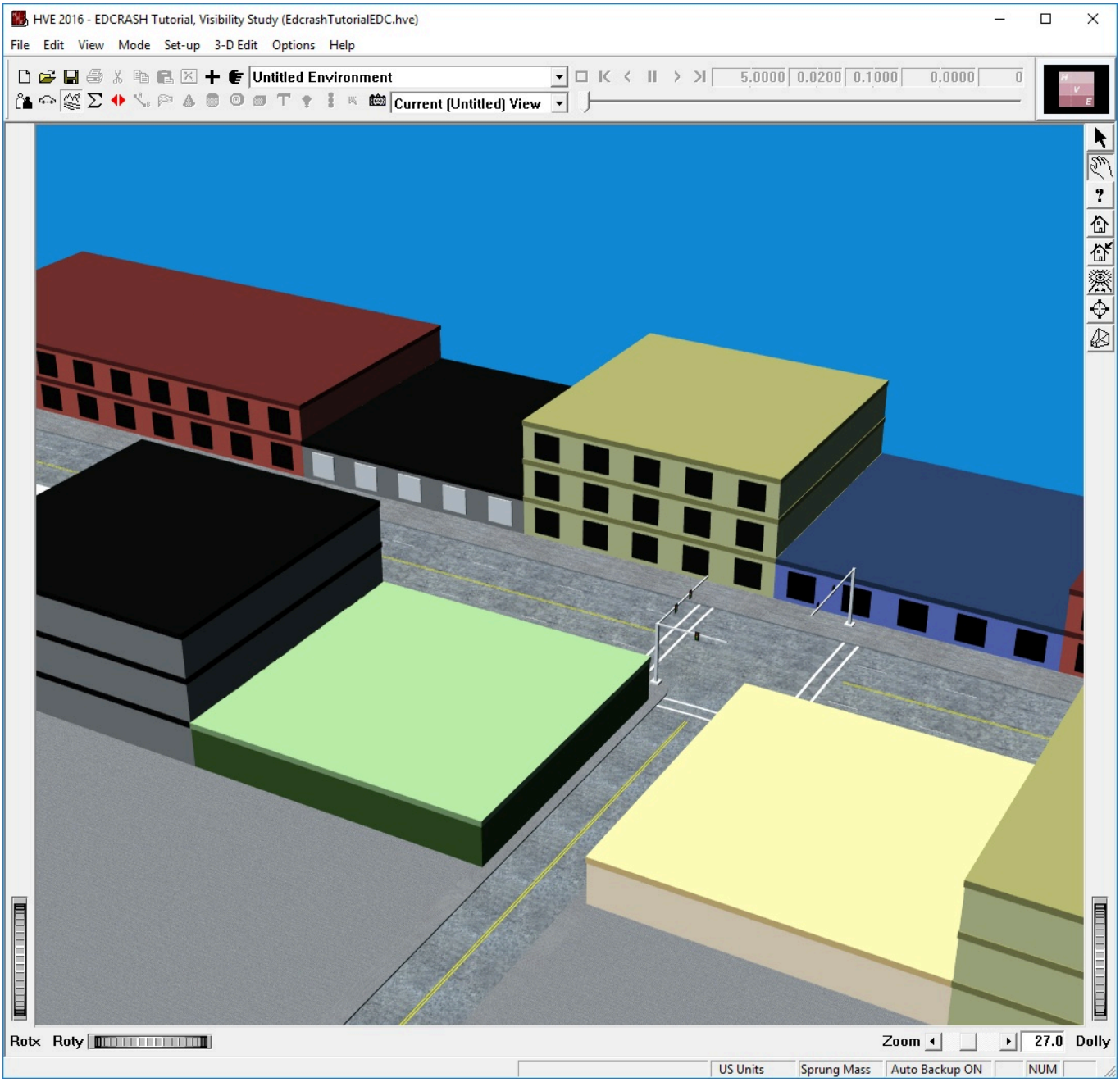


Figure 12-1: Environment Editor dialog and Viewer, showing the Blind Intersection environment model used in the EDCRASH and EDSMAC4 tutorials (EDCRASH Tutorial case shown).

Environment Editor Dialog

The Environment Editor dialog is the heart of the HVE Environment Editor. The dialog includes the following functionality:

- **Add Environment** — Allows the user to add an environment to the current case and displays two options: New and Previous. Choose *New* to create a new environment; choose *Previous* to add an environment from a previous HVE case. These options are further described below.
- **Current Environment Label** — Displays the name of the current environment. Click on the *Object Info* button on the toolbar to display the Environment Information dialog for the current environment.

> **NOTE:** The Current Environment Label serves exactly the same purpose as the Active Humans List or Active Vehicles List.

- **3-D Editor** — Loads the current environment into the 3-D Editor for editing of its physical and visual properties. To launch the 3-D Editor, select *Launch 3D Editor* from the *3D-Edit* menu option.

Environment Viewer

The Environment Viewer displays the current environment. It contains the following components:

- **Current Environment** — Allows the user to visualize the current environment.

> **NOTE:** There are no pickable objects in the Environment Viewer.

- **Viewer Controls** — Allows the user to Rotate, Pan, Zoom and set the current Pick Mode (see Chapter 2 for details about using the viewer controls).

Adding a New Environment

A new environment is added to the current case using the Environment Editor dialog. To add a new environment, perform the following steps:

1. Click on *Add New Object*. The Environment Information dialog (see Figure 12-2) will be displayed. *(updated: this dialog is a tabbed property sheet. Three top-level fields — Name (Environment Name), Date and Time — sit above the tabs, and the tabs organize the remaining parameters:* Terrain, Sky, Physical, Location *and* Traffic Signals. *In HVE-3D all five tabs are shown; in HVE-2D only the Terrain tab is present. See the Environment Information Dialog.)* The following user-definable parameters are available:
2. **Location Database** *(Location tab)* — A user-editable database of locations:
3. **City, State/Province and Country** — A user-entered location name. No formatting is required; however, the user is encouraged to maintain a consistent format to improve the usability of the database.
4. **Latitude and Longitude** — Used to define the global latitude and longitude, entered in degrees.minutes.seconds. N and S denote north and south latitudes, respectively; E and W denote east and west longitudes, respectively.
5. **GMT** — (Also called UTC offset.) The difference between the local standard time and the time at zero longitude. West longitudes are negative (e.g., New York City is -5 hours).

> **NOTE:** The location is used, along with the time and date of the crash and the orientation of the earth-fixed coordinate system (see below), to determine the location of the sun in the environment. See Chapter 13, Environment Model Definition.

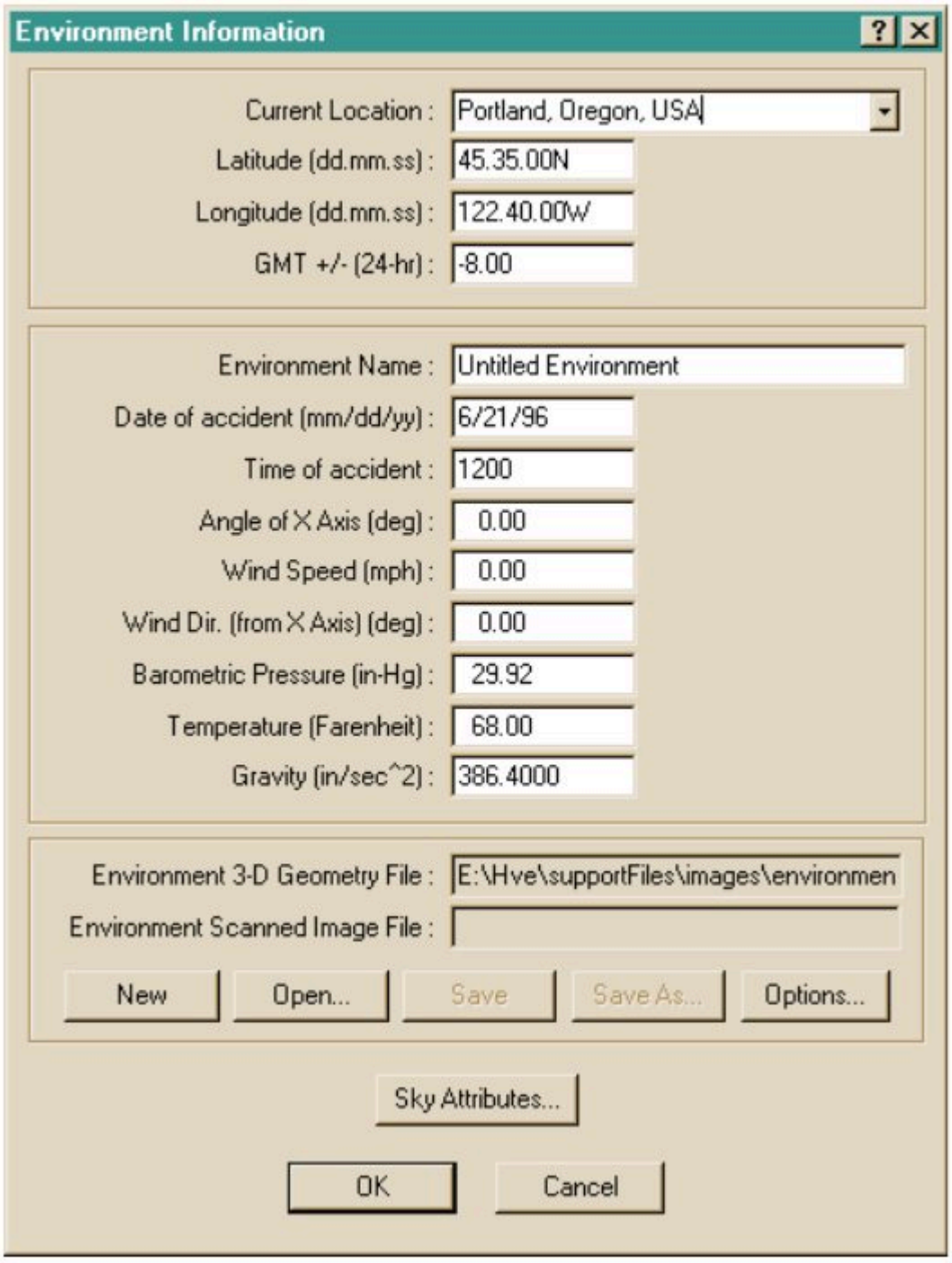


Figure 12-2: Environment Information dialog. See the field reference: Environment Information Dialog.

- **Name** *(top-level field)* — A user-editable field allowing the user to assign a name to the current environment.
- **Date** *(top-level field)* — The date of the crash or test being simulated. *(updated: the entry format follows the user's date-style preference — mm/dd/yyyy (US) or dd/mm/yyyy (European) — and the field label displays the expected format.)*
- **Time** *(top-level field)* — The standard time of day (24-hour) for the crash or test being simulated.
- **Angle of X-Axis** *(Physical tab)* — The angle of the user-defined earth-fixed coordinate system relative to true north. *(updated: valid range is -360 to +360 degrees.)*

> **NOTE:** True North and Magnetic North are normally very nearly equal; however, you should confirm the conditions in your area before making that assumption.

- **Wind Speed** *(Physical tab)* — The ambient wind velocity.
- **Wind Direction** *(Physical tab)* — The ambient wind direction relative to the earth-fixed X-axis.
- **Barometric Pressure** *(Physical tab)* — The ambient barometric pressure.
- **Temperature** *(Physical tab)* — The ambient temperature.

> **NOTE:** The wind speed and direction, barometric pressure and temperature may be used for aerodynamic calculations.
>
> **NOTE:** *(updated)* The Physical (atmospheric) fields — Wind Speed, Wind Direction, Temperature and Barometric Pressure — are not present in HVE-2D. The entire Physical tab appears only in HVE-3D, so in HVE-2D these fields are simply not shown; they are not displayed read-only.

- **Gravitational Constant** *(Physical tab)* — The local acceleration of gravity.

> **NOTE:** If you're simulating a lunar excursion module, you had better change this value!

1. Click on the Location combo box to choose the desired location, or enter a new City, State and Country, adding the Local Latitude, Longitude and Hours from GMT.

> **NOTE:** You can usually find latitude and longitude information in a pocket atlas (or from any online mapping service). Airline magazines usually contain time zone information.

1. Enter a Name, Date, Time, Angle of X-Axis, Wind Speed and Direction, Barometric Pressure, Temperature and Gravitational Constant for the current environment.

> **NOTE:** All these fields are optional.

1. If desired, select the file type and click on *Open* to select a previously created Environment 3-D Geometry file, aerial photograph and/or scanned photograph background (see 3-D Geometry, later in this chapter, for additional information).
2. If desired, use the Geometry File Options dialog (displayed when opening a geometry file) to scale and rotate the environment 3-D geometry file.
3. If desired, select the *Sky* tab to set sky color, fog and visibility for the current environment (see Sky Attributes, later in this chapter, for additional information).
4. Choose OK to add the current environment to the case.

> **NOTE:** You can click on the *Object Info* button on the toolbar to display the Environment Information dialog and attributes for the current environment.

Adding a Previous Environment

The user may also include environments from other cases in the current case. This option is useful when another case includes a study at the same or similar location (intersection, highway, etc.).

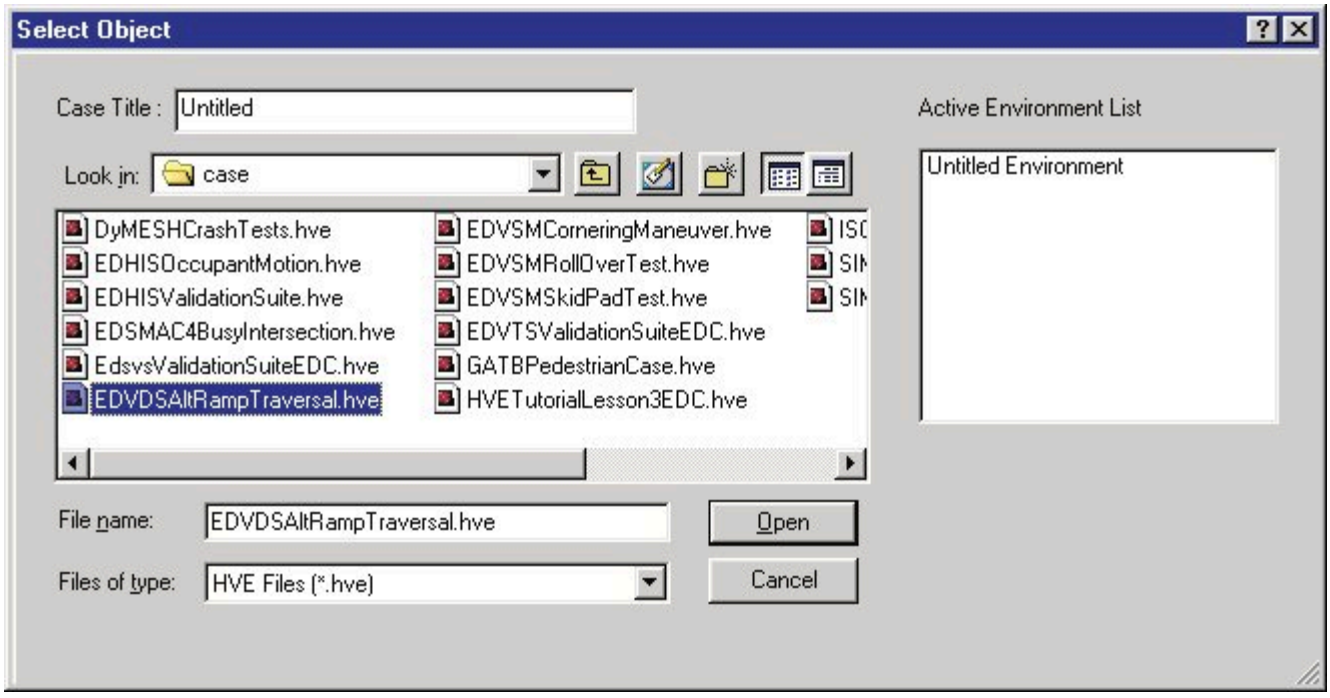


Figure 12-3: Adding Environments from previous cases.

To choose an environment from another case, perform the following steps:

1. Click on *Mode, Environment, Add* and choose *Previous*. The Previous Environment Files Selection dialog will be displayed (see Figure 12-3). This dialog displays a list of case names for all cases in the `\case` subdirectory.
2. Click on a case name. The Environment Name for the selected case will be displayed.
3. Press OK to add the selected environment to the current case.

Sky Attributes

Sky attributes determine the appearance of the background when a 3-D geometry file is displayed. *(updated: the sky attributes are now set on the* Sky *tab of the Environment Information property sheet — not through a separate "Set Sky Attributes" button and dialog. The Sky tab is shown only in 3-D run modes or with a 3-D camera viewer.)* The following attributes are available:

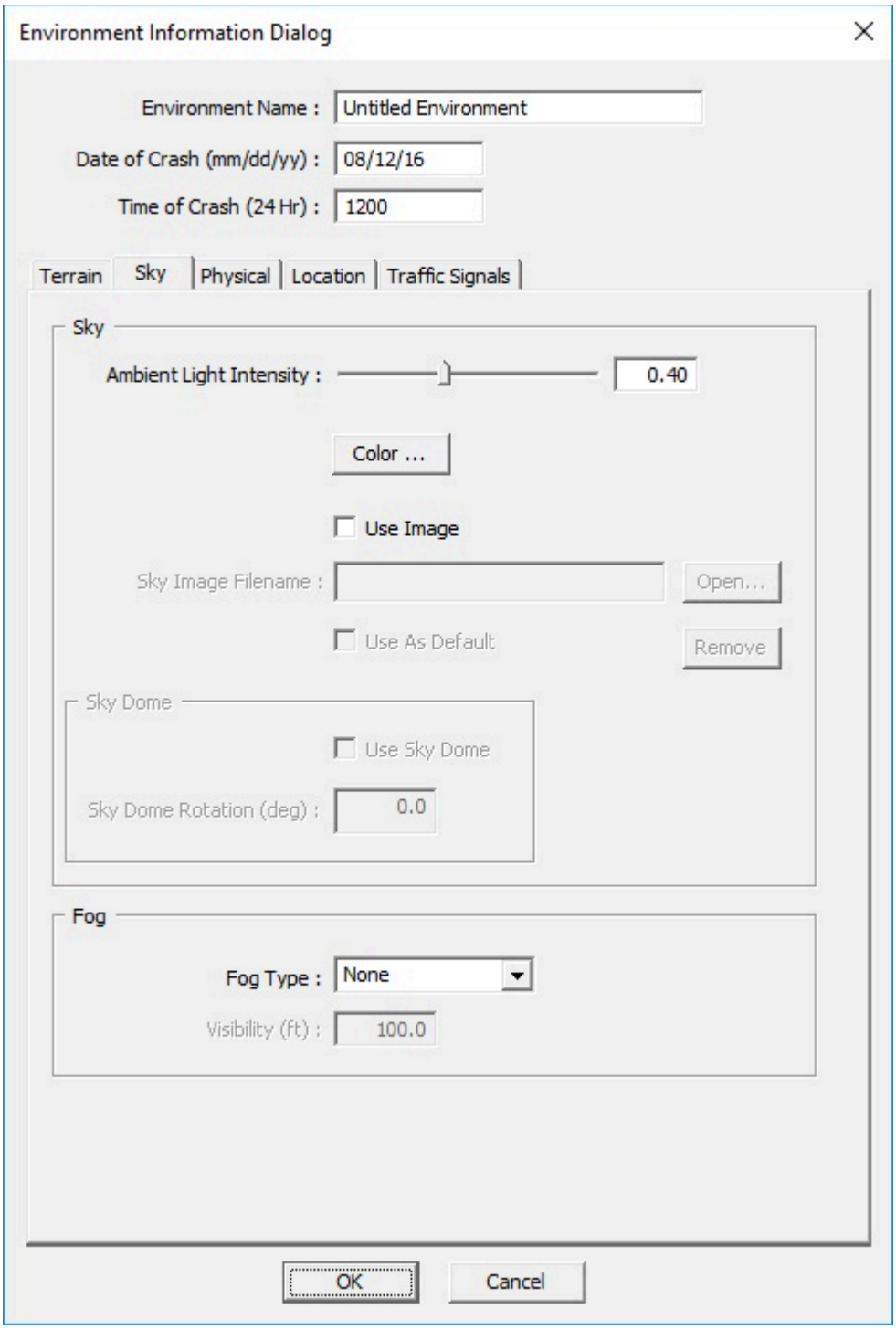


Figure 12-4: Environment Sky Attributes dialog.

- **Ambient Intensity** — Allows the user to determine the ambient lighting level in the scene. This attribute may be used for visualizing various levels of daylight, dusk/dawn and darkness.
- **Sky Color** — Allows the user to edit the current sky color, using either a color wheel or by supplying directly the red, green and blue color values.
- **Fog Type** — A user-selectable type of visual occlusion. The options are:
- None

- Fog
- Haze
- Smoke

> **NOTE:** The difference in the above fog types is in the way visibility is reduced as a function of distance. *None* implies no reduction. *Fog* has a linear reduction, *Haze* has a quadratic reduction and *Smoke* has a cubic reduction. See Chapter 13, Environment Model Definition, for additional information.

- **Maximum Visibility** — A user-editable field defining the distance (from the camera) at which objects are no longer visible.

> **NOTE:** Because the camera position represents the eye position of a person viewing the scene, the Maximum Visibility attribute is very useful for visibility studies!

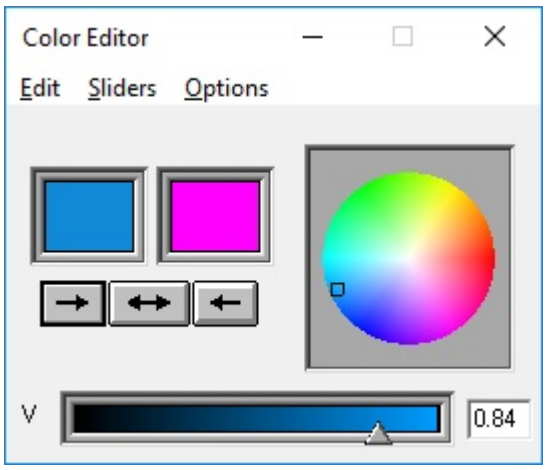


Figure 12-5: Sky Color tool for assigning the color of the sky.

To set the sky attributes, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by double-clicking on the Current Environment in the Environment Editor dialog.
2. Select the *Sky* tab (see Figure 12-4). *(updated: the Sky tab is available only when a 3-D camera viewer is in use; it is not present in HVE-2D.)*
3. Set the Ambient Light Intensity field using the slider or by typing in a number.
4. Click on *Sky Color* to display the Set Sky Color dialog (see Figure 12-5). Set the sky color using the color wheel (move the hot spot in the color wheel) or by typing in the Red, Green and Blue color values. Press *Accept* to see the result in the sample window.
5. Choose the desired fog type (None, Fog, Haze or Smoke).
6. If a Fog Type was selected, enter the desired Maximum Visibility distance.
7. Press OK to accept the current sky attributes.

Traffic Signals

(updated) The Environment Information property sheet includes a *Traffic Signals* tab, used to configure the timing of traffic signals placed in the environment. Like the Sky tab, it is shown only in 3-D run modes or with a 3-D camera viewer. Select a signal from the Traffic Signals list to edit its timing; the Type and Layout fields describe the selected signal, and the *Is Active*, *Repeating* and *Signal Timing* controls (Start Time, or *Follow* another signal with an Overlap) define its cycle. See the Environment Information Dialog for the full field reference.

3-D Environment Geometry

The user may supply a 3-D geometry file used for visualizing the environment. This geometry is also used by the current calculation model to provide detailed information about the surface elevation, slope and friction beneath each tire.

> **NOTE:** If no 3-D geometry file is supplied, the current simulation model will normally assume a flat terrain with uniform friction (associated with the vehicle's individual tires; refer to the Vehicle Editor for details regarding the tire's frictional properties).

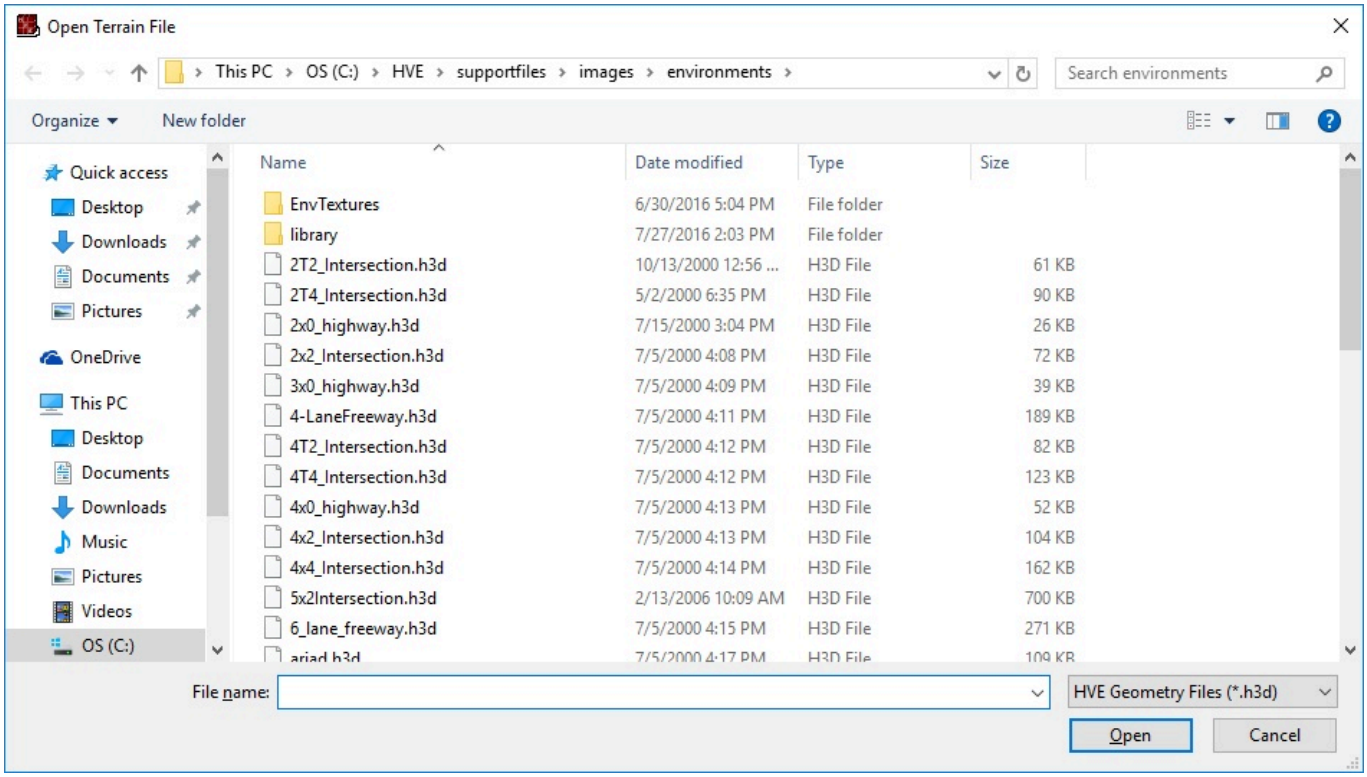


Figure 12-6: Environment File Selection dialog used for opening and saving 3-D Geometry and image files. See Table 12-1 for supported formats.

3-D Geometry information for the current case is supplied using the *Files* group of the Environment Information dialog. *(updated: the current dialog uses a single Files group with a File Type selector — Terrain, Aerial Photo or Sky Image, depending on licensed features — in place of the separate 3-D Geometry File and Scanned Image File entries in earlier versions.)* The following buttons are provided:

- **New** — Removes the current environment file of the selected type from the current case. *(updated: any terrain, aerial or backdrop file currently in use is cleared, and aerial image coordinates are reset to their defaults.)*

- **Open** — Allows the user to add a previously created 3-D geometry file (or image) to the current case.
- **Save / Save-As** — Allows the user to save the current 3-D geometry file in the HVE `.h3d` file format.

New

To remove the current 3-D geometry or image environment file, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by clicking the *Object Info* button on the toolbar.
2. Select the desired File Type and click on *New*.
3. Press OK.

The current environment geometry and/or image will be gone.

Open

To open an environment 3-D geometry file, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by clicking the *Object Info* button on the toolbar.
2. Choose *Open* to display the File Selection dialog (see Figure 12-6).
3. Choose the format of the desired file (e.g., HVE Geometry or other supported format) using the file-type filter.

> **NOTE:** The current format acts as a file filter for the files in the `images/environments` subdirectory. Table 12-1 provides a list of formats and their associated extensions. The number of supported formats increases as new translators become available.

1. Select the desired 3-D geometry file and press OK. The selected filename will appear in the Environment Information dialog.
2. Press OK.

The selected 3-D environment will be displayed in the Environment Viewer.

Table 12-1: HVE Supported File Formats *(updated to current file dialogs)*

Format	Type	Extension
HVE Geometry (internal format)	3-D Geometry	`.h3d`
Inventor	3-D Geometry	`.iv`
VRML	3-D Geometry	`.wrl`
OBJ	3-D Geometry	`.obj`
DXF (requires DXF translator feature)	2-D or 3-D Geometry	`.dxf`
DWG (requires DXF translator feature)	2-D or 3-D Geometry	`.dwg`
Bitmap	2-D Image	`.bmp`
TIFF	2-D Image	`.tif`
GIF	2-D Image	`.gif`
RGB	2-D Image	`.rgb`
JPEG	2-D Image	`.jpg`

(updated: JPEG support has been added; 3D Studio import listed in some older editions is handled via the DXF/translator features. Image formats are used for Aerial Photo and Sky Image file types.)

Save-As

To save the current environment as a new 3-D geometry file, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by clicking the *Object Info* button on the toolbar.
2. Choose *Save-As* to display the File Selection dialog.
3. Enter a name for the file.

> **NOTE:** HVE automatically appends its own `.h3d` extension. Texture pathnames are stored relative to the HVE `envtextures` directory.

1. Press OK to save the file.
2. Press OK to close the Environment Information dialog.

The selected 3-D environment will be displayed in the Environment Viewer.

3-D Geometry File Options

Most popular CAD packages use a conventional coordinate system in which the Z-axis points up. HVE uses the coordinate system adopted by the Society of Automotive Engineers (see Appendix III). In this coordinate system, the Z-axis points down (in the direction of gravity). In addition, users may have created their geometry files using meters, feet, or any other unit of length. In HVE, program units for length are always inches (user units, however, may be anything the user desires).

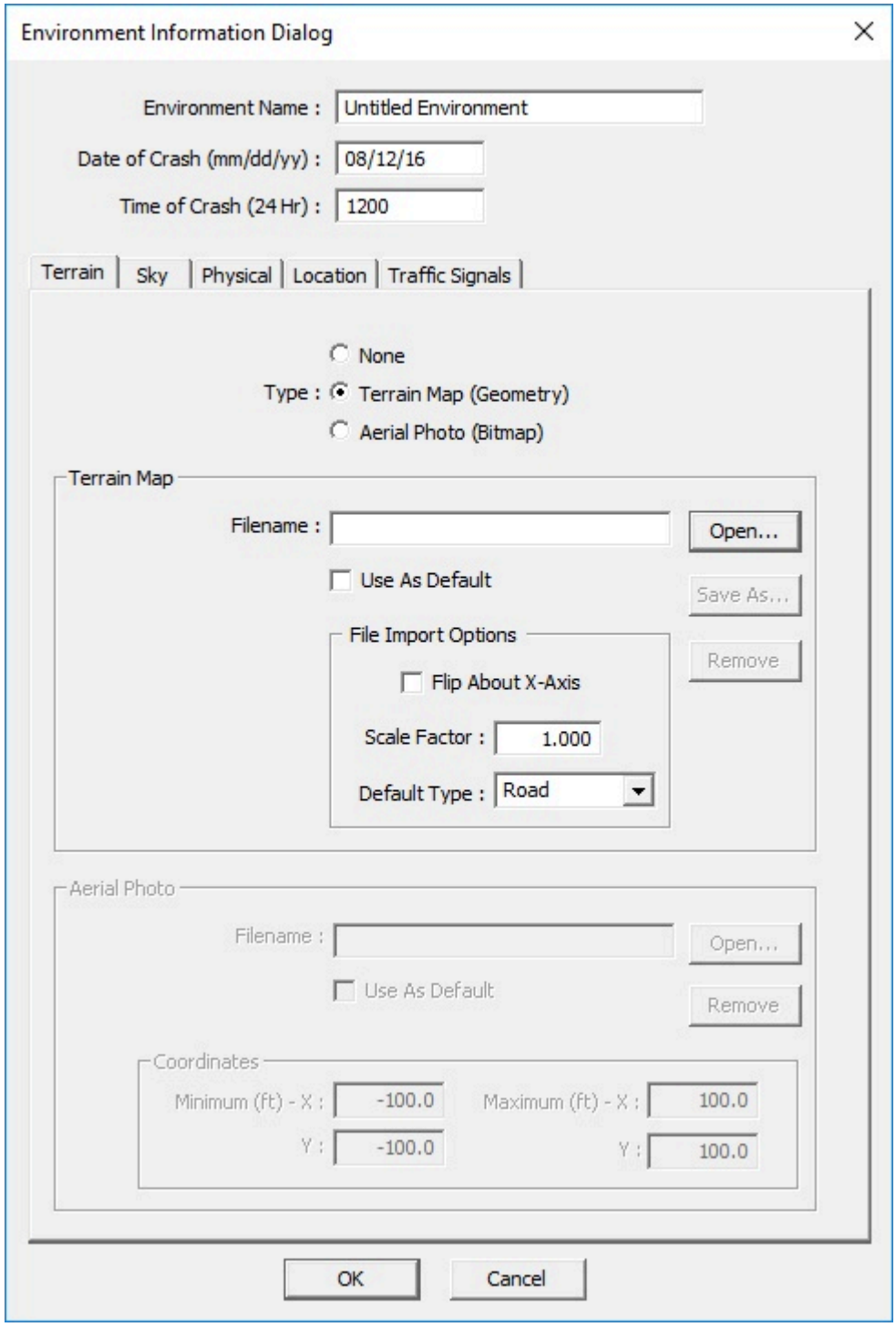


Figure 12-7: Geometry File Options dialog, used for flipping over the environment (HVE requires the Z-axis to point down) and for scaling the environment (HVE requires internal units of inches).

The Geometry File Options dialog allows the user to perform these simple conversions.

(updated: this dialog is displayed automatically when opening a terrain geometry file — the axis flip and scale entries apply chiefly to DXF files, and the dialog also offers the option to use the opened file as the default environment.)

Perform the following steps:

1. Open a 3-D Geometry file, as described on the previous pages. The Geometry Options dialog is displayed (see Figure 12-7).

2. Click *Flip About X-Axis* to make the Z-axis point down.
3. Enter the Scale Factor required to convert your drawing units to inches. For example, if your drawing is in feet, the conversion factor is 12.0; if your drawing is in meters, the conversion factor is 39.37.
4. Press OK. The Geometry File Options dialog is removed.
5. Press OK. The environment geometry is flipped and scaled, and the result is displayed in the Environment Viewer.

Environment Image File Subdirectory

All environment image files, whether 3-D geometry or 2-D images, are found in the `\images\environments` subdirectory.

> **NOTE:** If you use another tool to create a 2-D image or 3-D geometry file, you must copy the file to the `\images\environments` subdirectory in order for HVE to find it. See Appendix I for more information.

Scanned Photographs

A scanned photograph (a 2-D image, or bitmap) may be used to visualize the environment. Using a scanned photograph is a very quick and convenient way to provide a realistic environment background or sky. *(updated: the current Environment Information dialog distinguishes two image uses — an* Aerial Photo*, displayed on a ground polygon at user-entered ground coordinates, and a* Sky Image*, used as a fixed backdrop. The discussion below applies to the Sky Image/backdrop use.)*

> **NOTE:** Scanned photographs are created using a scanner or a digital camera.

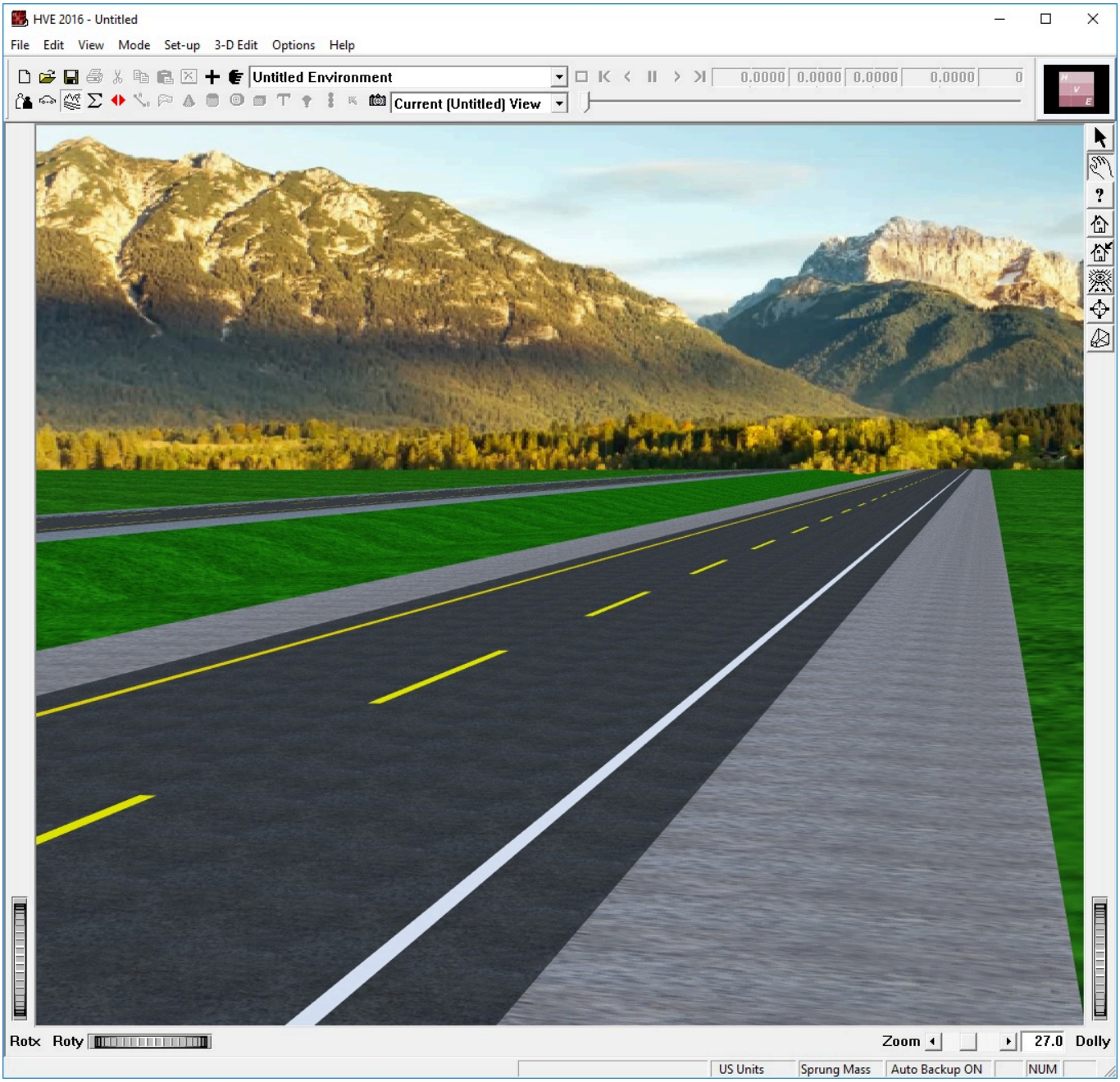


Figure 12-8: Example of a 3-D Geometry file used as the foreground and a scanned image used as the background and sky.

Scanned photographs differ in an important and fundamental way from 3-D geometry files: they are a fixed, 2-D image. The image is not affected by manipulating the viewer thumbwheels or changing the view using the Set Camera dialog.

> **NOTE:** The key to using a scanned accident site image is to use the Set Camera dialog to assign the same camera position, picture center and camera focal length as were used to take the original picture. This process links the view of the photograph to the view of the simulation.
>
> **NOTE:** If you do not know the exact location from which the photograph was taken, you can use the viewer thumbwheels (along with a little trial and error!) to figure out the correct camera location.

In addition, a scanned photographic image provides no physical information (elevation, slope, friction) to the current calculation model.

To add a scanned background image, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by clicking the *Object Info* button on the toolbar.
2. Select the *Sky Image* file type and choose *Open* to display the File Selection dialog.
3. Choose the format of the desired file (e.g., BMP, TIFF, JPEG or other supported image format). The Environment File Selection dialog is displayed, showing all the environment images of the selected file format.

> **NOTE:** HVE automatically distinguishes between 3-D geometry formats and image formats. Table 12-1 provides a list of formats, file types and their associated extensions.

1. Select the desired scanned image file from the list box and press OK. The selected filename will appear in the Environment Information dialog.
2. Press OK to display the current environment scanned image.

An example of a sky bitmap and 3-D geometry terrain is shown in Figure 12-8.

> **NOTE:** Both a 3-D Geometry file and a scanned photograph may be selected. Scanned images are always placed behind the 3-D geometry. Thus, you can provide a photograph for the visual effect and the 3-D geometry file for the physical information.
>
> **NOTE:** Scanned images make great skies. The RGB format includes several pre-defined skies for your use. See Figure 12-8 for an example of this effect.
>
> **NOTE:** If a scanned photograph is used, the view is fixed! This means that the user may attach the camera only to the environment, and not to any of the humans or vehicles.
>
> **NOTE:** If a scanned image is used for the road surface, you should confirm the scanned image was imported at the correct scale. The best way to do this is to identify some known landmarks in the viewer (you'll need to know the actual distance between these landmarks), then perform a simple simulation test to confirm that it takes the vehicle the correct time to travel between the landmarks. *(updated: the Aerial Photo file type addresses this directly — the Aerial Image dialog asks for the upper-left and lower-right ground coordinates of the image, which fixes the scale.)*

Overlays

Each 3-D geometry object (road surface, trees, delineation, etc.) has an overlay name (see the 3-D Editor documentation for additional information about overlay names). The *Overlays* option in the *View* menu displays a list containing every overlay name, as shown in Figure 12-9. By selecting or deselecting an overlay name, the user can decide whether an object is displayed in the viewer.

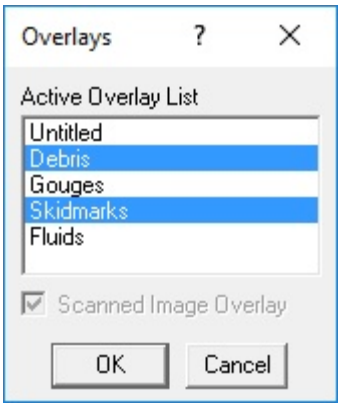


Figure 12-9: The Display Overlays dialog allows the user to select and deselect overlays.

> **NOTE:** If you deselect an overlay, its 3-D surface data (elevation, slope and friction) are still supplied to the current calculation model. One option is to include both a scanned photograph and a 3-D geometry file with all the overlays deselected. That way, the simulation will include the 3-D surface information, but the view will be from a photograph.

To display or remove selected overlays, perform the following steps:

1. Choose *Overlays* from the *View* menu. The Overlays dialog will be displayed, showing a list box with all the overlay names in the current scene. The currently selected overlays will be highlighted.
2. Select additional overlays you wish to be displayed, and deselect those overlays you wish to remove.
3. Click on the *Scanned Image Overlay* check box to display or remove any scanned environment image.
4. Press OK when the desired overlays and scanned image have been selected.

Editing the Current Environment

After a new or previous environment has been created, it may be edited. To edit the current environment, perform the following steps:

1. Click on the *Object Info* button on the toolbar. The Environment Information dialog will be displayed.
2. Edit any of the available fields to update the physical environment, as described earlier in this chapter.
3. Select the *Sky* tab to update the sky attributes, as described under Sky Attributes.
4. Click on *New, Open* or *Save-As* to update the environment 3-D Geometry, aerial photo or scanned image file, as described under 3-D Environment Geometry.
5. Press OK to update the current environment.

See also (dialog references):

Environment Information Dialog · Environment Material Properties · Surface Editor

Chapter 13 — Environment Model Definition

This chapter provides a detailed definition of the HVE Environment Model. The Environment Model's parameters define both the physical and the visual characteristics used by the current reconstruction or simulation model.

Overview

Chapter 13 is a very important chapter. One of HVE's most powerful features is the ability of humans and vehicles to interact with (drive on or slide on) the earth's surface. Another important feature is HVE's ability to view scenes obstructed by smoke or fog. These features are part of HVE's environment model. The mechanisms by which these features are implemented are described in this chapter.

The subject of the environment is divided into three basic topics:

- **Earth's Surface** — Gravitational Constant and 3-D Driving Surfaces used to calculate interaction forces.
- **Atmosphere** — Sky Color, Temperature, Pressure, Wind Velocity and Direction, Fog and Sight Distance used to help visualize the scene.
- **Sun Position** — Latitude, Longitude, GMT, Time/Date and Angle of the Earth-fixed X-axis used to determine the location of the sun and its lighting effects on the scene.

Each of these topics is addressed in this chapter.

Surface Model

The Environment Surface Model defines how humans and vehicles interact with the environment to produce forces that affect motion. The two physical factors in the Environment Surface Model are:

- 3-D Physical Surfaces
- Gravitational Constant

3-D Physical Surface Characteristics

Perhaps the single most powerful feature of HVE is the fact that HVE actually views the 3-D environment as a physical surface. Humans and vehicles may interact with this physical surface during execution to determine tire and other forces. The importance of this simple statement cannot be stressed enough! It means that, if the environment has a road, any simulated human or vehicle will roll or drive on the surface; if the environment has a hill, a simulated vehicle will naturally drive up the hill; if it has a ditch, the vehicle may roll into it; if it has a cliff, the vehicle may drive off it.

> **NOTE:** This capability of HVE must be taken advantage of by the human or vehicle simulation model, which queries HVE for the surface beneath each contact point. Refer to the individual calculation model's documentation to determine if this feature is included in the model.

How It Works

During execution, the reconstruction or simulation model may request the following information for any earth-fixed X,Y coordinates of the HVE environment (see Figure 13-1, Figure 13-2 and Table 13-1):

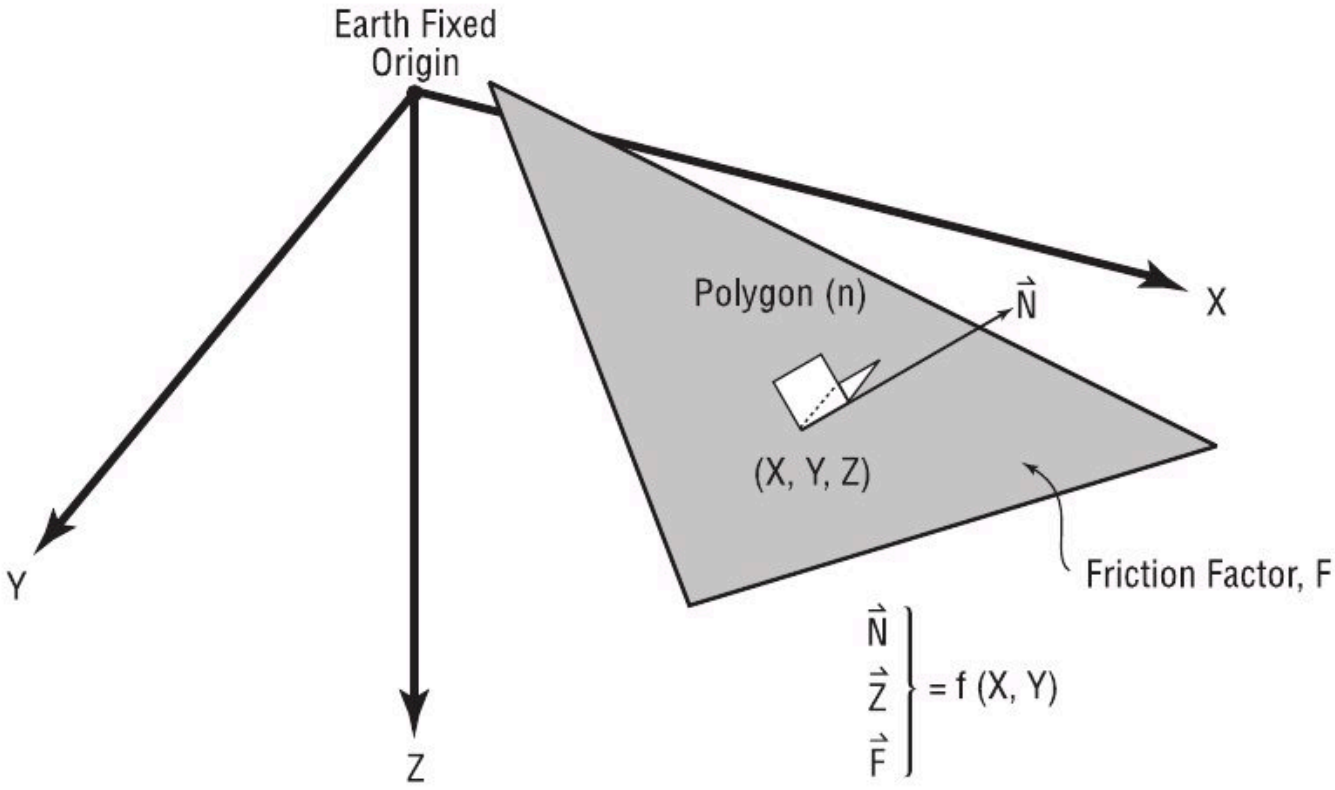


Figure 13-1: 3-D Surface Model Definition.

- **Elevation** — The Z-coordinate of the environment surface at X,Y.
- **Slope** — The surface normal vector, u_x, u_y, u_z, of the environment at X,Y.
- **Material Name** — A user-editable name describing the terrain at X,Y.
- **Force Constant** — The force required to initiate deflection of the terrain at X,Y.
- **Linear Stiffness** — The first-order force-deflection coefficient of the terrain at X,Y.
- **Quadratic Stiffness** — The second-order force-deflection coefficient of the terrain at X,Y.
- **Cubic Stiffness** — The third-order force-deflection coefficient of the terrain at X,Y.
- **Damping Constant** — The velocity-dependent force contribution of the terrain at X,Y.
- **Friction Factor** — The friction multiplier for the environment surface at X,Y.

> **NOTE:** The Friction Factor is a multiplier, not a friction coefficient! See below for details.

- **Unloading Slope** — The first-order unloading force-deflection coefficient of the terrain at X,Y.
- **Bekker Soil Exponent, N** — The exponent of deformation used in the soil mechanics equations for the terrain at X,Y (used by the Soft Soil Tire-Terrain Model).
- **Frictional Soil Modulus, Kphi** — Frictional soil constant used in soil sinkage calculations for the terrain at X,Y (used by the Soft Soil Tire-Terrain Model).
- **Cohesive Soil Modulus, Kc** — Cohesive soil constant used in soil sinkage calculations for the terrain at X,Y (used by the Soft Soil Tire-Terrain Model).
- **Moisture Content** — The amount of moisture in the soil for the terrain at X,Y.

- **Macrotexture** — The surface macrotexture depth of the terrain at X,Y, used by tire-road friction (e.g., wet-road) calculations. *(updated: the current material model carries Macrotexture in place of the Clay Content parameter listed in earlier editions; see the Environment Material Properties dialog.)*

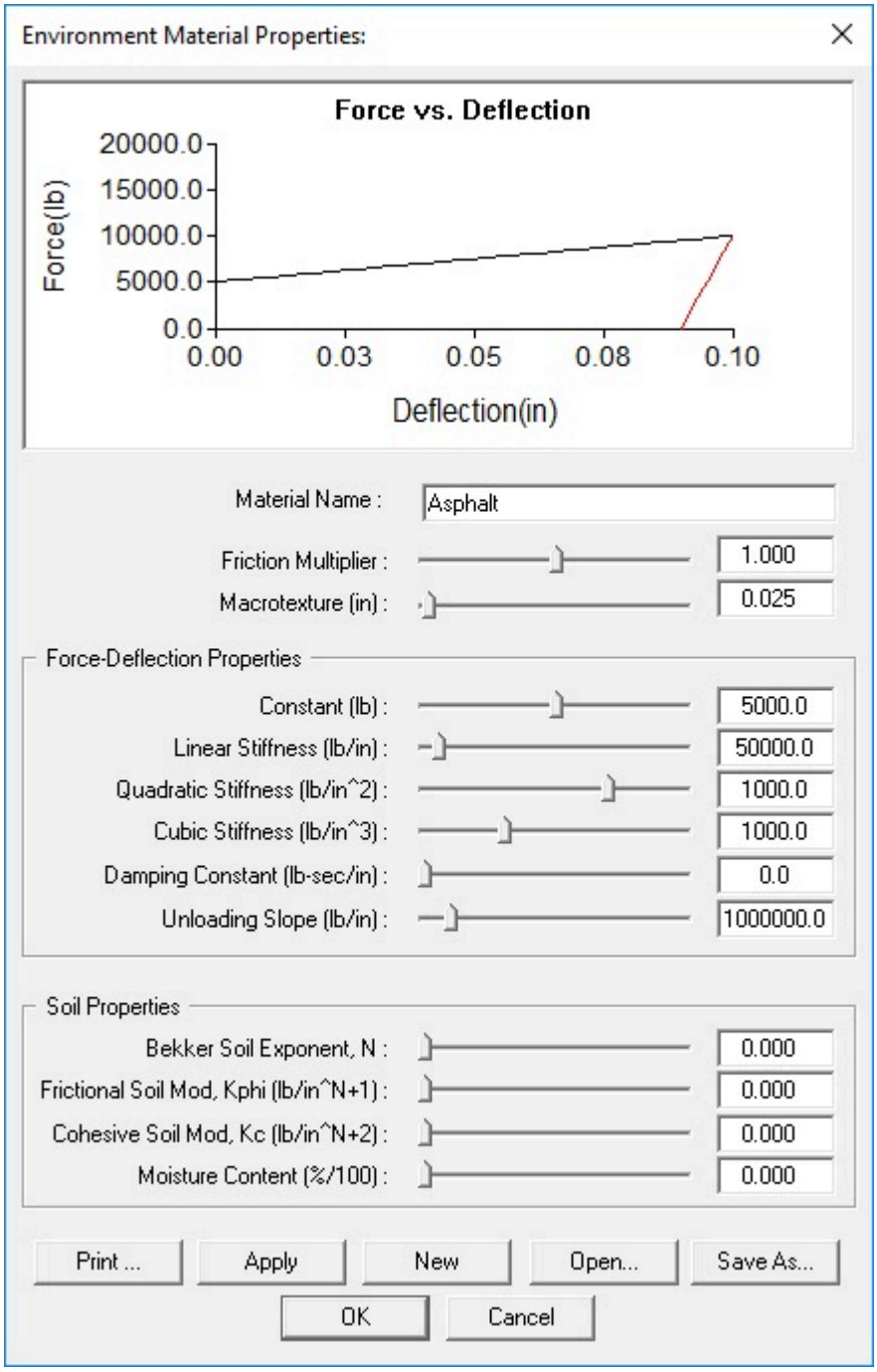


Figure 13-2: Environment Terrain Material Properties dialog. For the current field list — including sliders, valid ranges and defaults — see Environment Material Properties.

> **NOTE:** The Bekker Soil Exponent, Frictional Soil Modulus and Cohesive Soil Modulus are used by the Soft Soil Tire-Terrain Model. See the Tire-Terrain Model discussion in Chapter 4.

A vehicle simulation's tire model obviously can benefit from this information: given a tire's current X,Y (earth-fixed) coordinates, the simulation's tire model can learn all the above attributes for the terrain in the vicinity of each tire. A human simulation's contact ellipsoids can also include these effects when calculating forces on each human body segment for pedestrians struck on the roadway. The same is true of the vehicle's 3-D geometry; these data are used for rollover simulation.

Table 13-1: Environment Terrain Material Parameters

Parameter	Units (US / SI)	Description
Z-coord	ft / m	Elevation at X,Y
u_x, u_y, u_z	—	Unit normal vector defining the slope at X,Y
Material Name	—	User-editable material name for terrain at X,Y
Constant	lb / N	Force required to initiate deflection at X,Y
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient at X,Y
Quadratic Stiffness	lb/in² / N/cm²	Quadratic material deformation coefficient at X,Y
Cubic Stiffness	lb/in³ / N/cm³	Cubic material deformation coefficient at X,Y
Damping Constant	lb-sec/in / N-sec/cm	Material velocity-dependent deformation constant at X,Y
Friction Constant	—	Friction multiplier at X,Y
Unloading Slope	lb / N	Linear unloading slope beginning at maximum deflection
Bekker Soil Exponent	—	Soil deformation exponent for sinkage calculations
Frictional Soil Modulus	lb/in^N+1 / N/cm^N+1	Frictional soil constant for sinkage calculations
Cohesive Soil Modulus	lb/in^N+2 / N/cm^N+2	Cohesive soil constant for sinkage calculations
Moisture Content	%/100	Percentage of moisture in the soil
Macrotexture	ft / m	Surface macrotexture depth at X,Y

How to Define Surface Characteristics

The surface model characteristics are defined directly by the 3-D surface polygons (sometimes called 3-D geometry) used to create the environment. The 3-D geometry database often includes tens of thousands of polygons. During execution, the simulation must test the entire list of polygons to determine which polygon is currently beneath each tire or contact surface, to determine the elevation, slope and friction factor of the polygon.

To reduce the number of polygons tested by the calculation model (and thus reduce processing time), HVE's 3-D Editor allows the user to define several types of objects. The following object types are available:

- **Human** — Object type for all human polygons
- **Vehicle** — Object type for all vehicle polygons
- **Road** — Default object type for all environment polygons
- **Friction Zone** — User-assignable object type for environment polygons
- **Other** — User-assignable object type for environment polygons

(updated: the current environment model also supports Curb *and* Water *polygon types — water polygons include water depth, depth method and water friction multiplier attributes used by water-related tire models.)*

Polygons of type Human are tested only for human vs. vehicle/road/friction zone interaction. Polygons of type Vehicle are tested only for human/road/friction zone interaction. Polygons of type Other are not tested or used by the calculation model.

> **NOTE:** It makes no sense to test the red lens of a traffic signal, or the sides of a 10-story building, for interaction with a vehicle's tires!
>
> **NOTE:** If you know an environment object will not be in contact with a simulated human or vehicle, be sure to define its type as *Other* to reduce processing time.

How Surfaces Are Tested for Physical Contact

So far, this section seems pretty straightforward. Now, let's ask a few questions:

- Suppose the vehicle is on an overpass. What prevents the calculation model from using the elevation of the surface beneath the overpass?
- Alternatively, suppose the vehicle is on an underpass. What prevents the calculation model from using the elevation of the road surface on the overpass? How about the underside of the overpass?
- Suppose the user has created an oil slick and placed it on the road surface. What prevents the calculation model from using the road, instead of the oil slick surface?

Get Surface Information

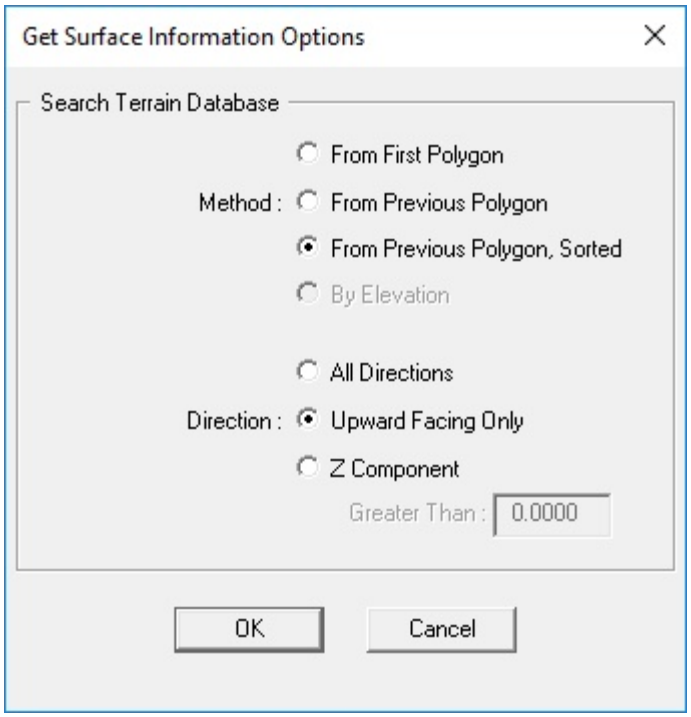


Figure 13-3: Get Surface Information dialog.

To answer these and similar questions, a strict definition is required. HVE uses two different methods to determine which polygon gets used:

- **Surface search** — Used by the Point Contact and Soft Soil Tire-Terrain Models (see the Event Set-up Wheels option; Chapter 4). This method searches in the direction of the earth-fixed gravity vector (i.e., straight down) to determine the desired terrain polygon.
- **Terrain search** — Used by the Radial Spring and Sidewall Impact Tire-Terrain Models (again, see the Event Set-up Wheels option; Chapter 4). This method searches in a programmatically defined direction (i.e., in the direction of a tire radial or sidewall spring) to determine the desired terrain polygon.

The Get Surface Information dialog has four search methods

(updated: a fourth method, From Previous Polygon, Sorted, has been added since the legacy edition and is now the default)

:

- Use First Polygon
- Use Previous Polygon
- Use Previous Polygon, Sorted *(updated: current default)*
- Use Elevation

The following rules are used by either the surface search or the terrain search, according to the selected option:

From First Polygon

- *The first surface found beneath the specified X,Y coordinates is used.* The search ends when a surface is found. This rule reduces calculation time. By returning the results for the first surface, it is not necessary to read every polygon in the database and then decide which one to use.
- *Friction Zones are used first.* This is an important rule. It means there is a hierarchy: if two polygons exist, one just above the other, the first one found in the database will be used. To ensure that the oil slick is used instead of the surface beneath it (see third bullet, above), assign its type as a *Friction Zone*. To ensure the top surface of the overpass is used instead of the underside of the overpass or the road below the underpass (see first and second bullets, above), assign its type as a *Friction Zone*.

From Previous Polygon

- *The surface used during the previous timestep is tried first.* If the wheel is still on that surface, that surface is used. This method has the potential for working much faster, because the chances are good that the tire is still on the same surface.
- *Search in both directions.* If the tire is not on the surface used during the previous timestep, try the next polygon in the database, then try the previous polygon (rather than starting over with the first polygon). This also helps to increase calculation speed because polygon databases are typically organized such that surfaces lying physically adjacent to each other are also stored next to each other in the database.

From Previous Polygon, Sorted *(updated)*

- Works like From Previous Polygon, but the polygon database is first sorted so that physically adjacent polygons are stored adjacently, making the previous-polygon search still more effective. This is the default search method in the current version.

By Elevation

- *The surfaces are searched by elevation.* The highest surface still beneath the wheel center is used. This method is the most robust, but is very time-consuming because the entire polygon database must be searched at every timestep.

(updated: this option remains in the dialog for compatibility but is not supported by the current physics models; use one of the polygon-search methods instead.)

The Get Surface Information dialog also allows the user to define the allowable direction for surface normals on each terrain polygon (see Figure 13-1). The options are:

- **All Directions** — All terrain polygons are used by the tire model, regardless of the direction of the Z-component of the terrain surface normal.

> **NOTE:** Use this option if you wish to simulate a stunt driver performing a loop-de-loop!

- **Upward Facing Only** — Only terrain polygons with a positive Z-component are included by the tire model.
- **User-Defined** — The tire model uses a range for surface Z-components that is defined by the user.

If User-Defined is selected, the user may enter the allowable range for the Z-component of the surface normal.

> **NOTE:** This method is often required for curb impact simulation in which the curb face is not exactly vertical due to rounding error. Try entering 0.001.

Assigning the Object Type Attribute

Object Type is an attribute assigned using the 3-D Editor (see Figure 13-4). The default type for all human 3-D geometry is Human; for all vehicle 3-D geometry, Vehicle; for all environment 3-D geometry, Road.

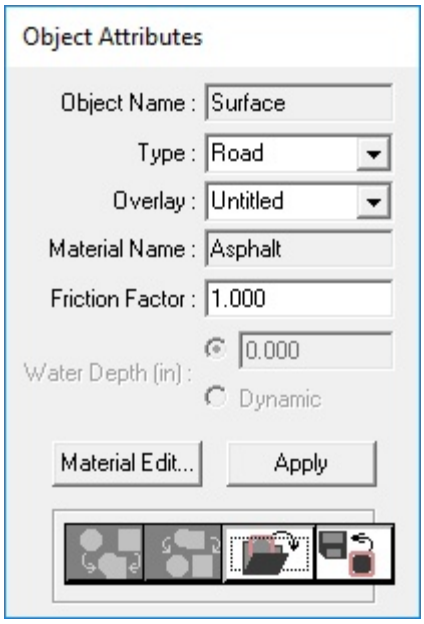


Figure 13-4: Assigning the Object Attributes using the 3-D Editor.

By default, all environment surfaces will be used by the calculation model. To ensure a particular surface is used by a calculation model, assign its type as Friction Zone.

> **NOTE:** See the 3-D Editor documentation for more information about assigning object type attributes.

Importing 3-D Files from Other Modeling Programs

HVE can import a variety of 3-D formats used by other programs. The imported geometry will be used by the current calculation model. Thus, the surface elevation and normal for each polygon will be assigned according to the rules defined earlier.

Because other modeling programs do not assign a friction factor or surface type, default values (Friction Factor = 1.0, Type = Road) will be assigned and used. The default Friction Factor and Type of objects in imported files may be edited using HVE's 3-D Editor.

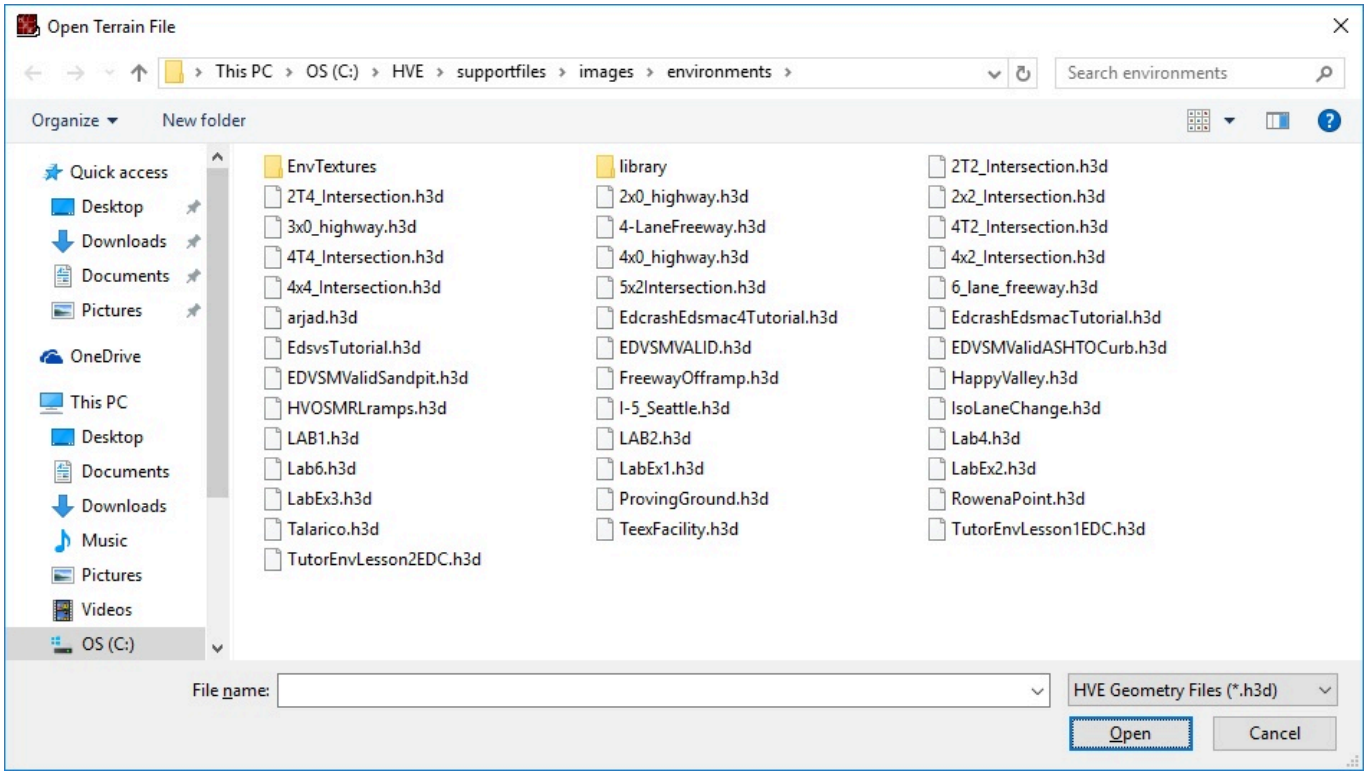


Figure 13-5: Environment File Selection dialog used for opening and saving 3-D Geometry and image files. See Table 12-1 for supported formats.

To load 3-D files from other modeling programs and edit the default surface attributes, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by clicking on the *Object Info* button on the toolbar.
2. Click *Open* to display the File Selection dialog (see Figure 13-5).
3. Choose the format of the desired file (e.g., DXF or other supported format) using the file-type filter.

> **NOTE:** The current format is a file filter for the files in the `images/environments` subdirectory. Table 12-1 provides a list of formats and their associated extensions. Formats not yet available are grayed out. The number of supported formats increases as new translators become available.

1. Select the desired 3-D geometry file from the list box and press OK. The selected filename will appear in the Environment Information dialog.
2. Press OK. The selected 3-D environment will be displayed in the Environment Viewer.

3. Choose *3-D Editor* from the Environment Editor dialog. The current environment will be loaded in the HVE 3-D Editor.
4. Click on the desired object to select it. The object's attributes (Object Name, Type, Overlay and Friction Factor) will be displayed in the 3-D Editor's Object Attributes Tool.
5. Click on the Object Type option list (the current type will be Road) and select either *Friction Zone* (to increase its priority) or *Other* (to remove it from the list of physical surface objects).
6. Edit the object's friction factor.
7. Close the 3-D Editor and return to Event Mode to execute an event using the new object type and friction surface attributes.

> **NOTE:** It is crucial when creating a 3-D environment using another editor to consider the way the individual objects will be treated when imported into the HVE 3-D Editor. This is especially important when creating object groups and surfaces. See the 3-D Editor documentation for more information.
>
> **NOTE:** While creating a 3-D environment, it is quite possible to (accidentally) create two surfaces sharing the same space. Although the visual appearance will be unaffected, the surface search may use the wrong surface. Be careful!

Gravitational Constant

The gravitational constant defines the natural acceleration of objects towards the center of the earth (in the direction of the earth-fixed Z-axis). The value of the gravitational constant varies only slightly as a function of earth-fixed coordinates. However, if the human or vehicle being simulated may be taken to the moon or Mars, it is simple (and important) to determine the effect of differences in the gravitational constant.

To edit the gravitational constant, perform the following steps:

1. Display the Environment Information dialog (if it is not currently displayed) by double-clicking on the Current Environment in the Environment Editor dialog.
2. Enter the desired gravitational constant (see Table 13-2 for selected gravitational constants).
3. Press OK.

The entered value will be assigned to the current environment.

> **NOTE:** Because all inertial objects are stored according to mass, the correct weight will be displayed in the Human or Vehicle Editor. During Event mode, the correct inertial properties of the object will be used for the current environment.

Table 13-2: Gravitational Constants for Selected Environments

Location	Gravitational Constant (in/sec²)
Earth — Common Range	386.08 – 386.4
Earth — At Equator	385.06
Earth — London (51°32')	386.29
Earth — Poles (90°)	387.09
Moon	64.4
Mars	146.8
Jupiter	1024.0

Atmospheric Model

Physical Characteristics

The physical characteristics of the atmosphere include the following parameters:

- Temperature
- Barometric Pressure
- Wind Velocity and Direction

These parameters may be used to determine the aerodynamic drag force on an object by calculating the current level of dynamic pressure.

Air Density

Air density, δ, is calculated using the current temperature and pressure according to the following formula:

```
δ = p / (R·T)
```

where

Symbol	Meaning
δ	Air Density, lb/in³
p	Atmospheric Pressure, lb/in²
R	Gas Constant, 639.6 in-lb/lb-°R
T	Absolute Temperature, °R (°F + 460)

Dynamic Pressure and Force

Given the current air density and wind velocity, the force, F_{air}, acting on a vehicle (or any object) due to aerodynamic pressure is as follows:

```
F_air = (c_a · δ · A · V_a²) / (2·g)
```

where

Symbol	Meaning
F_{air}	Air Resistance Force, lb
c_a	Air Drag Coefficient, dimensionless
δ	Air Density, lb/in³
A	Projected Area, in²
V_a	Wind Velocity relative to vehicle, in/sec
g	Gravitational Constant, in/sec²

> **NOTE:** The aerodynamic drag coefficient and projected area are vehicle parameters. Refer to Chapter 11, Vehicle Model Definition, for further details.

Visibility

The visual characteristics of the atmosphere include the following parameters:

- Fog Type
- Maximum Sight Distance (graph of attenuation vs. distance)
- Sky and Fog Color

Fog Type

HVE provides three types of atmospheric effects that limit visibility:

- **Fog** — The percentage of vision attenuation increases as a linear function of distance from the camera (see Figure 13-6).
- **Haze** — The percentage of vision attenuation increases as a quadratic function of distance from the camera (see Figure 13-6).
- **Smoke** — The percentage of vision attenuation increases as a cubic function of distance from the camera (see Figure 13-6).

(Note: the legacy edition's text transposed the quadratic/cubic assignments between its Chapter 12 and Chapter 13 discussions; the ordering above — Fog linear, Haze quadratic, Smoke cubic — matches the Chapter 12 description and the rendering behavior.)

These atmospheric effects determine how sight distance is attenuated. Visual attenuation is determined algorithmically.

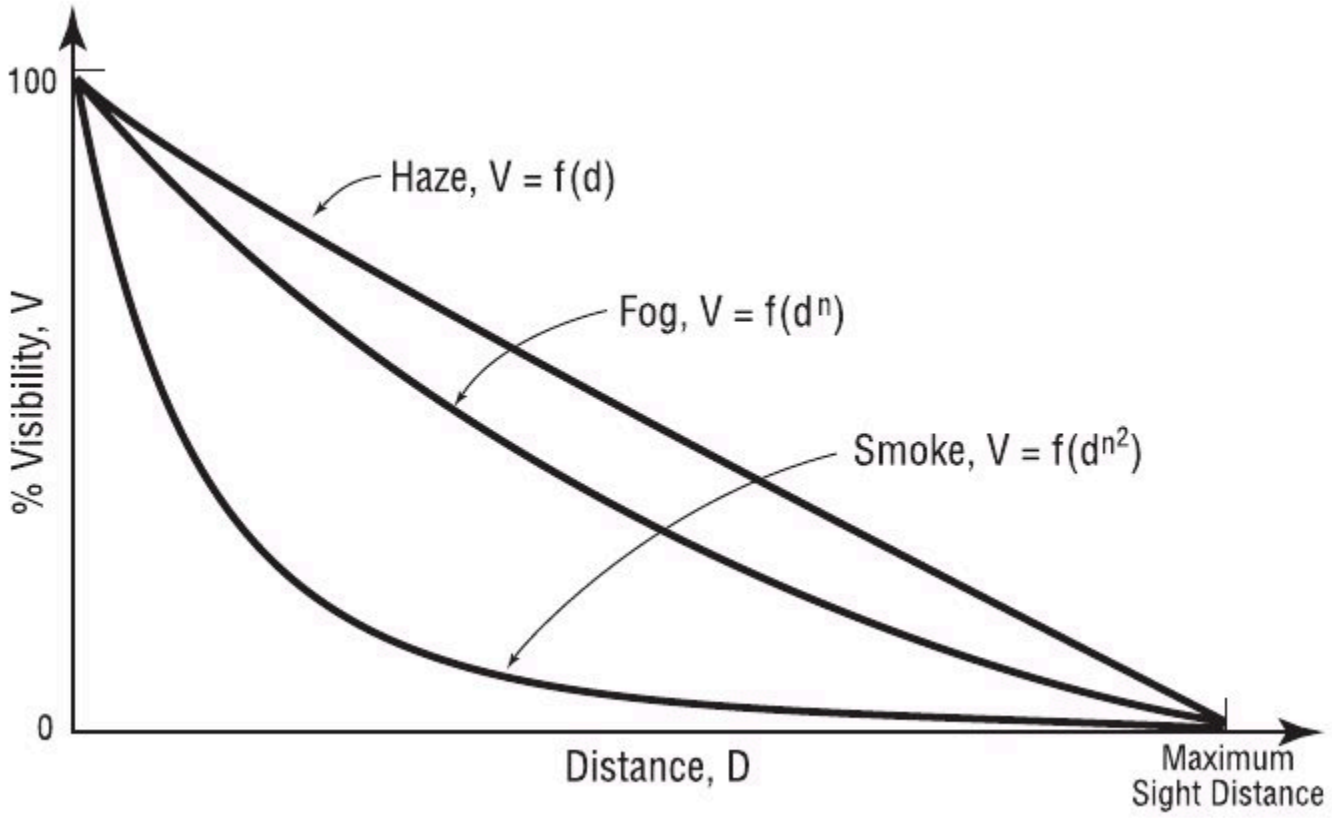


Figure 13-6: Visibility vs. Distance for Fog, Smoke and Haze.

Maximum Sight Distance

Maximum Sight Distance is used in the fog calculations to determine the distance at which vision is 100 percent attenuated. The Maximum Sight Distance is entered using the Sky Attributes dialog.

Sky and Fog Color

Sky and Fog Color are used to create a visually accurate representation of the environment. The assessment of color is qualitative, and is not subject to objective calculations. The Sky and Fog Color are entered using the Sky Attributes dialog.

Sun Position

The location of the sun is used to assign directional lighting to the environment. The sun is also displayed as a sphere at the correct astronomical location.

The following parameters are used to assign the sun position:

- **Latitude** — The angular distance from the equator, expressed in degrees, minutes and seconds north (+ or N) or south (- or S) from the equator.
- **Longitude** — The angular distance from the prime meridian, expressed in degrees, minutes and seconds west (+ or W) or east (- or E) from the prime meridian.
- **Hours from GMT** — The time, expressed in hours, from the prime meridian.

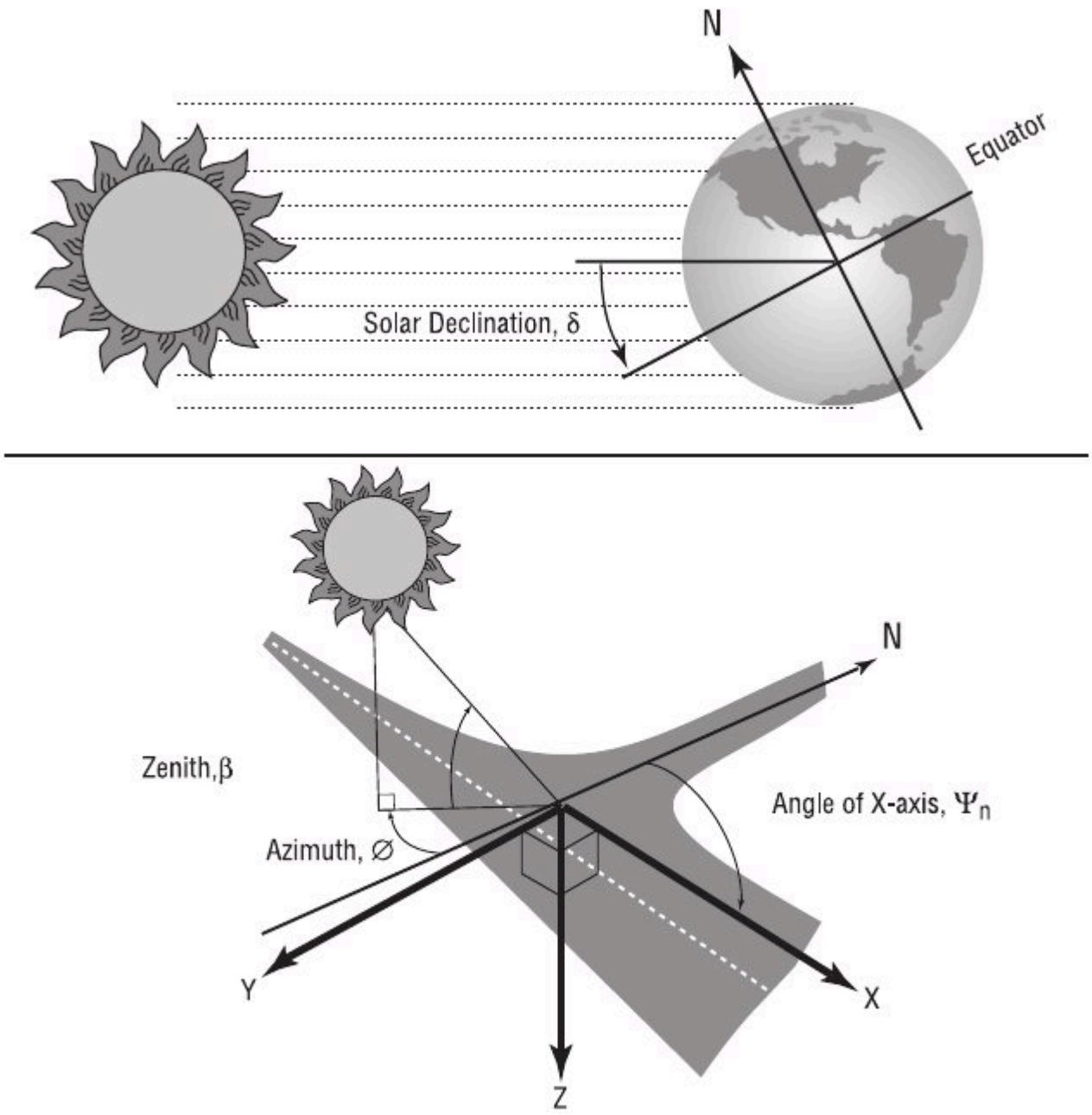
> **NOTE:** GMT stands for Greenwich Mean Time — the difference, in integer hours, from the time at Greenwich, England, to the local time zone. GMT for nearly any global location may be found in most world atlases and travel references.

- **Date** — The date of the event (month, day, year).
- **Time** — The time of the event (24-hour clock; e.g., 10:15 PM is 2215).

> **NOTE:** Daylight saving time is not considered! If an event occurred during daylight saving time, you need to subtract one hour from the stated time.

Sun Azimuth and Zenith

Given the data above, HVE calculates the location of the sun in the current environment. The sun location is used for determining the location of the solar light source, resulting in the correct shading of objects, as well as for displaying the sun in the environment. Figure 13-7 illustrates the sun position calculations.



$$\sin\beta = \cos L \cos\delta \cos H + \sin L \sin\delta$$

$$\cos\phi = \frac{\sin\beta \ \sin L - \sin\delta}{\cos\beta \cos L}$$

where L = Latitude
δ = Solar Declination
H = Hour Angle

Figure 13-7: Sun Azimuth and Zenith calculations.

> **NOTE:** By displaying the sun, you might stumble onto the notion that a driver was looking into the sun at the time of an accident.

Environment Location Database

The HVE Environment Editor includes a user-extendible database of environment locations, stored according to the Location Name, Latitude, Longitude and GMT, as shown in Figure 13-8.

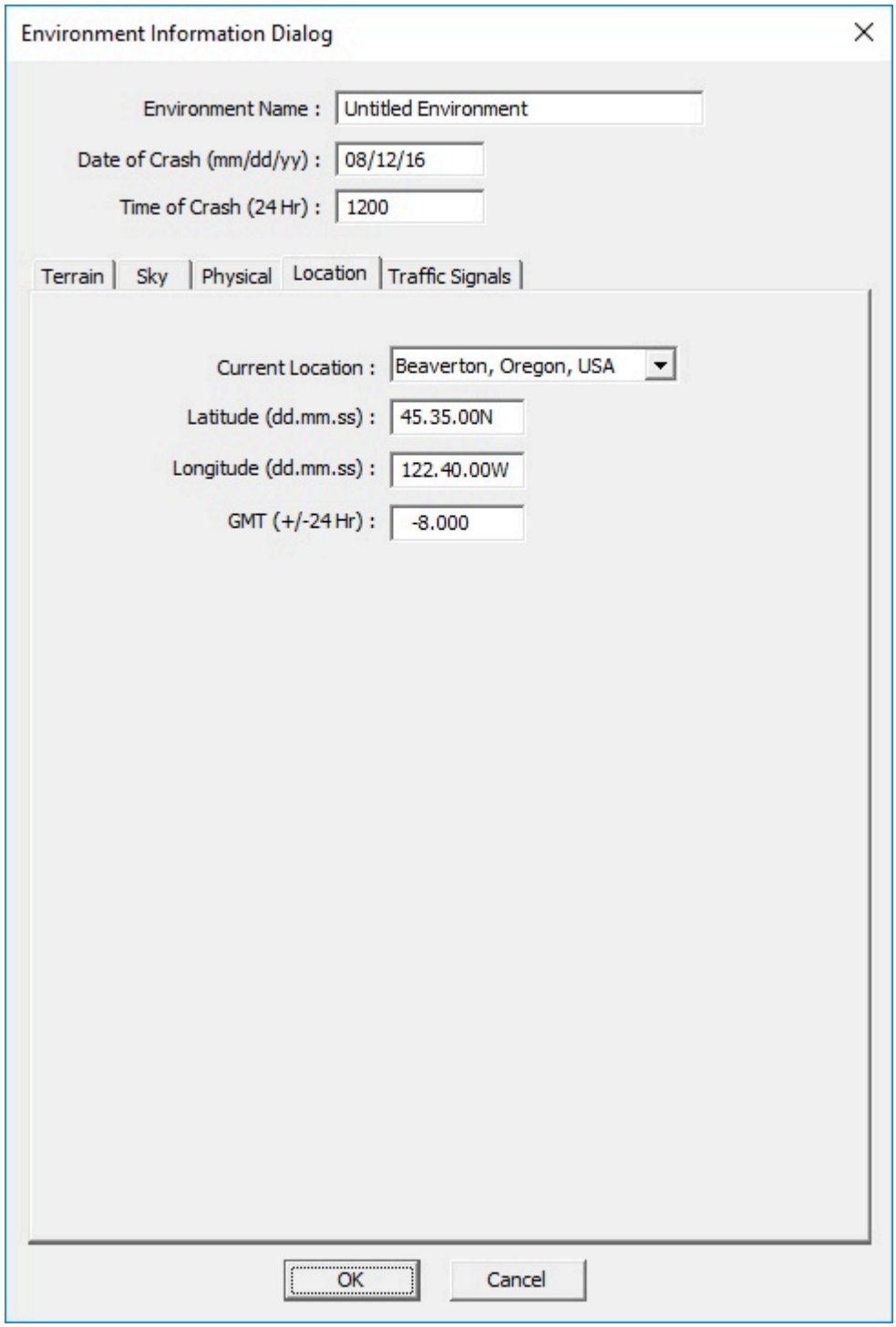


Figure 13-8: The Location Database allows the user to store Latitude, Longitude and GMT for any location, normally entered as City/State/Country.

Table 13-3: Environment Model Parameters

Parameter	Units (US / SI)	Description
Current Location	30-character text string	City, State (or Province), Country (Location Database)
Latitude	deg	Latitude for the current location. The user-specified value is stored in the location database for future use.
Longitude	deg	Longitude for the current location. The user-specified value is stored in the location database for future use.
GMT	sec	The number of hours from Greenwich, England, to the specified location (conventionally +/- 24 hours; west of Greenwich is negative)
Environment Name	30-character text string	User-entered name for the current environment
Date of Crash	Text string	Entered value is parsed according to month/day/year (or day/month/year, per the user's date-style preference) to determine the date relative to the solar calendar.
Time of Crash	Text string	Entered value is parsed according to a 24-hour clock (i.e., 2230 is 10:30 PM) to determine the time relative to midnight (standard time is used; daylight saving time is not considered).
Angle of X-Axis	deg	Angle of the user-specified earth-fixed X-axis relative to true north. The user must make any adjustments from magnetic north if a compass was used to identify direction.
Wind Speed	mph / km/h	Wind velocity
Wind Direction	deg	Wind direction relative to true north. The user must make any adjustments from magnetic north if a compass was used to identify direction.
Barometric Pressure	in-Hg / kPa	Ambient barometric pressure
Temperature	deg F / deg C	Ambient temperature
Gravitational Constant	in/sec² / m/sec²	Local gravitational constant

See also (dialog references):

Environment Information Dialog · Environment Material Properties · Linear Friction Zone Properties · Surface Editor

Section Six — Event Editor

An event is defined as a single analysis involving the interactions between human(s), vehicle(s) and their environment. The purpose of the HVE Event Editor is to study these interactions through the use of reconstruction and simulation models. This section of the HVE User's Manual describes how to set up and execute these reconstruction and simulation models. This section also provides a detailed overview of all the event-related options available to the user.

The basic physical/mathematical model of an event provides the following information:

- **Event Inputs** (the humans, vehicles and environment)
- **Event Set-up Parameters** (the positions, velocities, driver controls and other event-related parameters for each human and vehicle)
- **Event Calculation Model** (the reconstruction or simulation model used for studying the interactions between the humans, vehicles and environment)
- **Event Outputs** (the results of execution)

General Procedure

To use the Event Editor, perform the following steps:

1. Select Event mode.
2. Select the desired humans and/or vehicles.
3. Choose a reconstruction or simulation model to study the interactions between the selected humans and/or vehicles.
4. Set up the event:
5. Position the human(s) and/or vehicle(s) in the environment at one or more desired locations. For simulations, supply an initial velocity.
6. Provide any other set-up options (Payload, Driver Controls, Damage Profiles, Collision Pulse, Restraint Systems, Contacts).
7. Execute the event.
8. Edit the event (by adjusting positions, velocities, driver controls and other set-up options) and re-execute the event.

Repeat the above steps for any number of additional events.

Overview of Section Six

This section provides a detailed explanation of the process of setting up and executing events, and also presents detailed definitions for the available event-related options:

- Chapter 15 — Creating & Editing Events describes the process of setting up and executing events.
- Chapter 16 — Event Model Definition provides detailed definitions for the Event set-up options and outputs.

Detailed dialog references for this section — including the current Driver Controls dialogs (Pedal Force brake tables, Wide-open Throttle tables, Ackermann steering, neuro-muscular filter and the HVE Driver path/speed follower) — are collected in the Events and Driver Controls reference.

Chapter 15 — Creating & Editing Events

The HVE Event Editor is used for creating and editing events for the current HVE case. These events may also be viewed and combined with other events in the Playback Editor, as described in Section Seven (Playback Editor).

This chapter describes how to set up and execute events. This process is central to the user's goals when using HVE.

> **NOTE:** Refer to the next chapter, Event Model Definition, for a detailed description of the Event Model parameters.

Event Editor Components

To use the Event Editor, choose Event Mode using the mode selector. This puts HVE into Event Mode and displays the Event Editor's components:

- **Event Information Dialog** — The Event Information dialog is used for adding events to the current case and to select the current event.
- **Event Viewer** — The Event Viewer displays the current environment and is used for visualizing the current event and editing human and vehicle positions.
- **Event Controller** — The Event Controller is part of the Event Editor dialog and includes a VCR-like panel of buttons allowing the user to start, stop, reset and play events forwards and backwards.

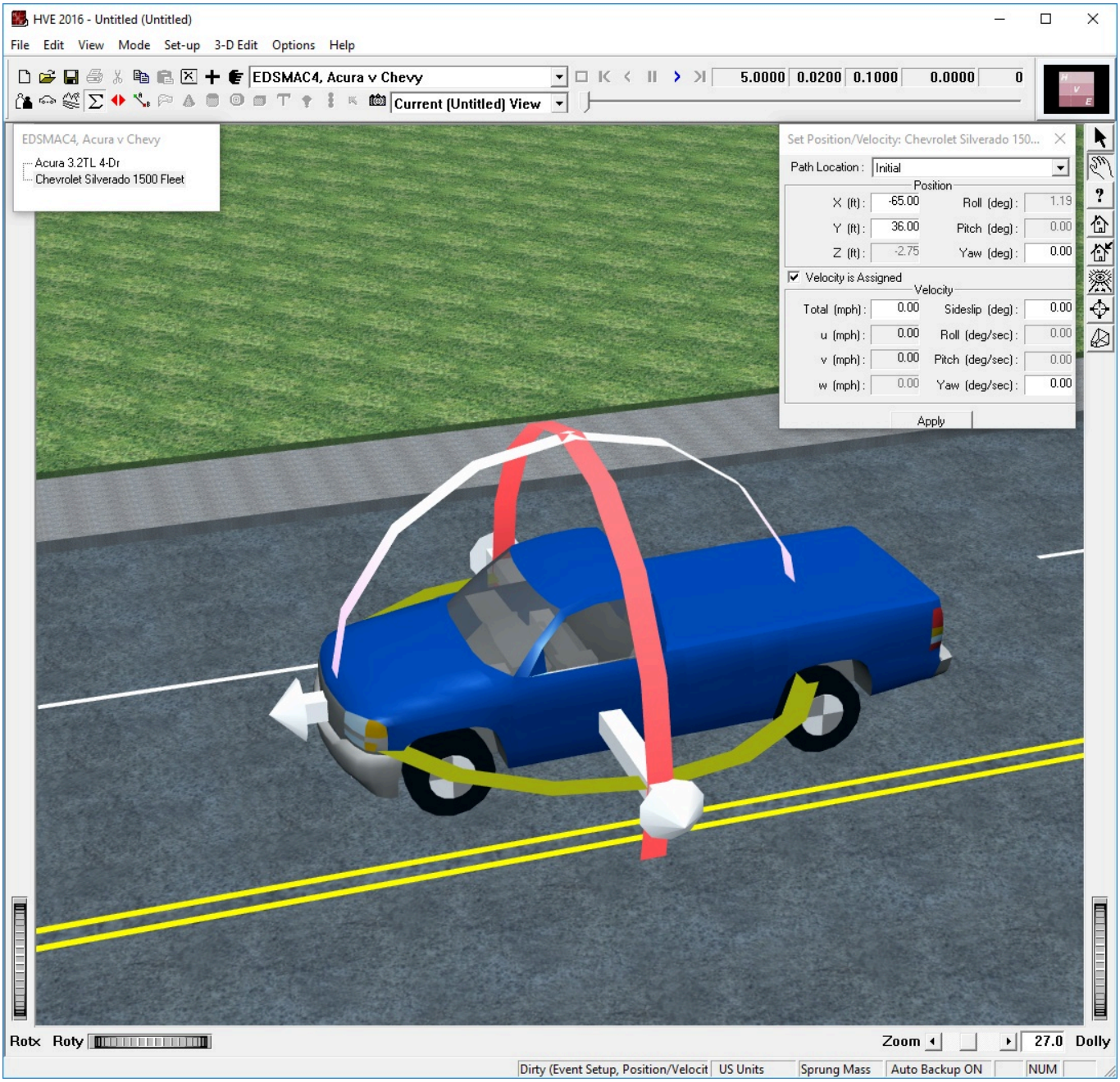


Figure 15-1: Event Editor, Viewer, Position/Velocity dialog and Event Controller.

Event Editor Dialog

The Event Editor dialog is the heart of the HVE Event Editor. The dialog includes the following functionality:

- **Add Event** — Allows the user to add an event to the current case.
- **Active Events List** — Displays a list of the names for all events in the current case. The name of the current event is displayed in the Current Event label. Click on the drop-down list to select a different event. Click on the *Object Info* button to display its Event Information dialog.
- **Copy Event** — Creates an exact copy of the current event.
- **Delete Event** — Removes the current event from the case.

> **NOTE:** Delete is not selectable if the current event is used in a Playback Window (see Section Seven: Playback Editor).

Event Viewer

The Event Viewer (see Figure 15-1) displays the current event. The Event Viewer contains the following components:

- **Current Event** — Allows the user to visualize the humans, vehicles and environment used in the current event.
- **Event Humans** — Displays each of the humans used in the current event.
- **Event Vehicles** — Displays each of the vehicles used in the current event.
- **Viewer Controls** — Allow the user to Rotate, Pan, Zoom and set the current Pick Mode (see the Viewers documentation for details about using the viewer controls).

Position/Velocity Dialog

The Position/Velocity dialog (see Figure 15-2) displays the position and orientation of the selected human or vehicle. It includes the following components (for the current field reference see the Position/Velocity dialog page):

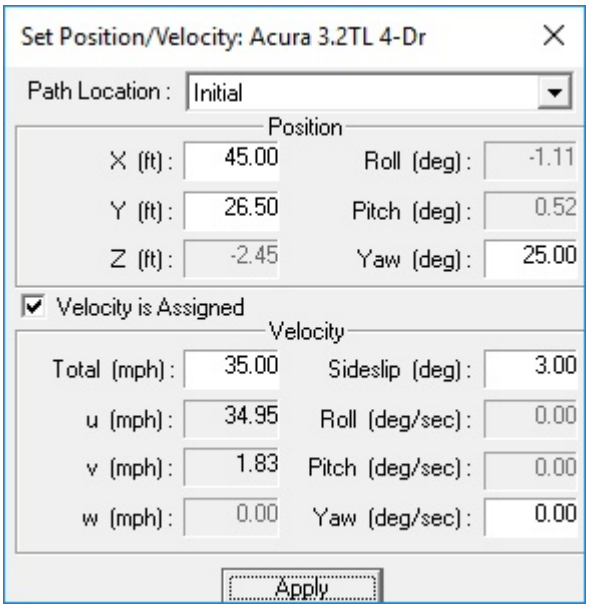


Figure 15-2: Position/Velocity dialog, used for assigning positions and velocities to humans and vehicles in the current event.

- **Path Location Listbox** — Allows the user to choose a path location at which to place the vehicle. The available path locations are Initial, Begin Perception, Begin Braking, Impact, Separation, Point on Curve, End of Rotation and Final/Rest.
- **X, Y, Z Position Fields** — Allow the user to enter or edit the selected object's coordinates.

> **NOTE:** If the object is a human occupant, the X, Y, Z fields are relative to the vehicle's CG. Otherwise, the X, Y, Z fields are relative to the earth-fixed origin.
>
> **NOTE:** If the current calculation model is 2-dimensional, the Z coordinate is assigned using AutoPosition, and is not user-editable.

- **Roll, Pitch, Yaw Orientation Fields** — Allow the user to enter or edit the selected object's orientation.

> **NOTE:** If the object is a human occupant, the Roll, Pitch, Yaw fields are relative to the vehicle's coordinate axis system. Otherwise, the fields are relative to the earth-fixed axis system.
>
> **NOTE:** The order of rotations is Yaw, Pitch, Roll. Rotation is not commutative in 3-space!
>
> **NOTE:** If the current calculation model is 2-dimensional, the roll and pitch orientations are assigned using AutoPosition, and are not user-editable.

- **Velocity Is Assigned Check Box** — Clicking on this check box allows the user to assign a velocity to the object at the selected position.
- **Total Velocity, Vertical Velocity and Sideslip Fields** — Allow the user to assign a linear velocity to the object at the selected position.

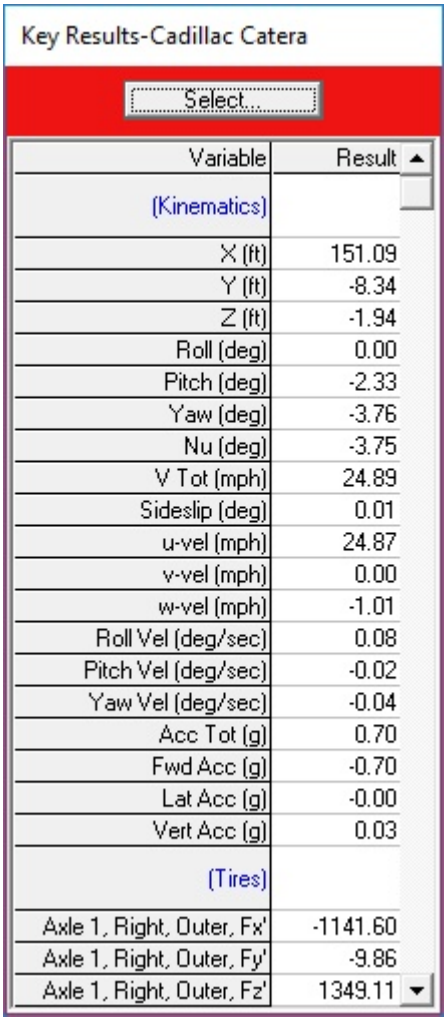
> **NOTE:** If the object is a human occupant, the velocity is relative to the vehicle-fixed coordinate system. For vehicles and human pedestrians, the velocity is relative to the earth-fixed coordinate system.
>
> **NOTE:** If the current calculation model is 2-dimensional, the vertical velocity field is not available.
>
> **NOTE:** The forward velocity (u) and lateral velocity (v) are auto-computed from the total velocity, V_t, and sideslip, β ($u = V_t \cos \beta$ and $v = V_t \sin \beta$), but remain user-editable. Editing U, V or the Sideslip angle recomputes the other components accordingly.

- **Roll, Pitch, Yaw Velocity Fields** — Allow the user to enter or edit the selected object's angular velocities.

> **NOTE:** If the object is a human occupant, the Roll, Pitch, Yaw velocities are relative to the vehicle's coordinate axis system. For vehicles and human pedestrians, the velocities are relative to the earth-fixed axis system.
>
> **NOTE:** If the current calculation model is 2-dimensional, the roll and pitch velocities are ignored and are not user-editable.

Key Results Windows

Key Results windows (see Figure 15-3) display the current value of one or more user-specified variables during execution.



Variable	Result
(Kinematics)	
X (ft)	151.09
Y (ft)	-8.34
Z (ft)	-1.94
Roll (deg)	0.00
Pitch (deg)	-2.33
Yaw (deg)	-3.76
Nu (deg)	-3.75
V Tot (mph)	24.89
Sideslip (deg)	0.01
u-vel (mph)	24.87
v-vel (mph)	0.00
w-vel (mph)	-1.01
Roll Vel (deg/sec)	0.08
Pitch Vel (deg/sec)	-0.02
Yaw Vel (deg/sec)	-0.04
Acc Tot (g)	0.70
Fwd Acc (g)	-0.70
Lat Acc (g)	-0.00
Vert Acc (g)	0.03
(Tires)	
Axle 1, Right, Outer, Fx'	-1141.60
Axle 1, Right, Outer, Fy'	-9.86
Axle 1, Right, Outer, Fz'	1349.11

Figure 15-3: Key Results Window, used for displaying important, real-time results during events. Simulation models display a Simulation Key Results window; reconstruction models have a similar window.

> **NOTE:** The Kinematics variables are displayed by default.

For more information about Key Results windows, see Menu Reference, Options Menu.

Event Controller

The current event is executed using the Event Controller, shown in Figure 15-4. The Event Controller has buttons that function much like a VCR controller:

- **Reset** — Reinitializes the current calculation model for re-execution.
- **Rewind to Start** — Returns the current simulation to t_0.
- **Reverse** — Plays the current simulation in reverse.
- **Stop/Pause** — Pauses the current simulation.
- **Forward (Play)** — Plays the current calculation model.
- **Advance to End** — Advances the current simulation to the end of the run.



Figure 15-4: Event Controller, used for starting and stopping event execution. The Event Controller can also be used for replaying previously executed events, both forward and backward.

Adding New Events

New events are added to the current case using the Event Editor dialog. To add a new event, perform the following steps:

1. Click on

   Add New Object

   . The Event Information dialog (see Figure 15-5) will be displayed, allowing the user to enter or select the following options (see also the Event Information dialog page):

2. **Event Name** — A user-editable field allowing the user to assign a name to the current event.

3. **Active Humans List** — A list of all the active humans created for the current HVE case. Click on one or more humans to be included in the current event.
4. **Active Vehicles List** — A list of all the active vehicles created for the current HVE case. Click on one or more vehicles to be included in the current event.
5. **Event Humans & Vehicles List** — A list of the humans and vehicles to be used in the current event.

> **NOTE:** The order of selection may be important. For example, for a tractor-trailer simulation, the first vehicle is assumed to be the tow vehicle, the second vehicle is towed by the first vehicle, and so on.

- **Calculation Model Option Button** — An option button displaying the current calculation model (e.g., EDSMAC4, EDVSM, SIMON). Press this button to display the list of calculation models available for selection.

> **NOTE:** Calculation methods are licensed. If you choose a method for which you have no license, HVE will issue a message and your selection will be ignored.

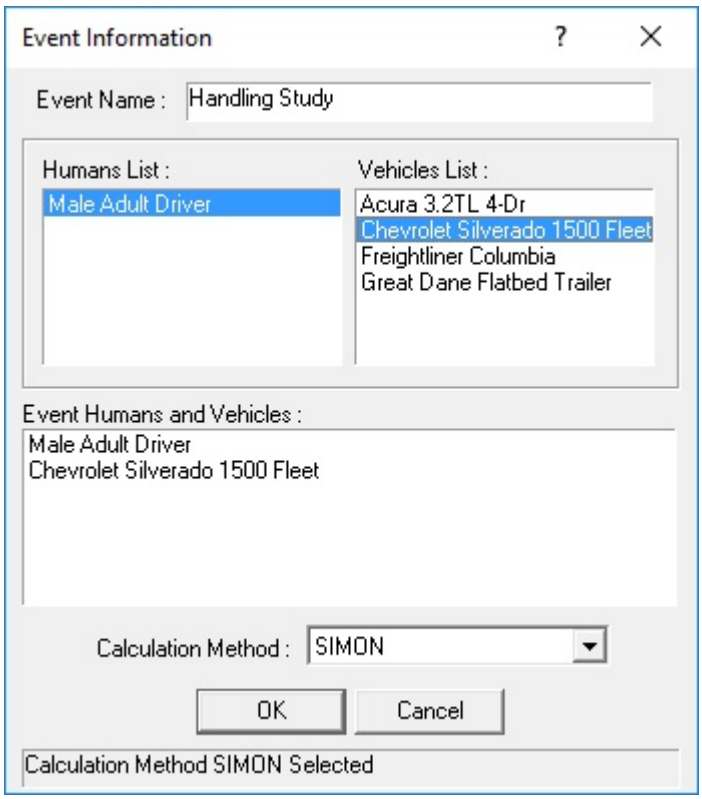


Figure 15-5: Event Information dialog.

1. Enter a name for the current event.
2. Click on the names of the desired human(s) and vehicle(s) in the Active Humans and Active Vehicles Lists, respectively.
3. Click on the Calculation Model options list and choose the desired calculation model.
4. Press OK.

Event Verify

When OK is pressed in the Event Information dialog, HVE automatically confirms the selected humans, vehicles and environment are compatible with the selected event. This process is called Event Verification. HVE tests for such things as:

- Are the selected object types compatible with the selected event? For example, you cannot load a human into a single vehicle simulator.
- Are the object's attributes compatible with the selected event? For example, you cannot load a tandem-axle vehicle into a simulator which does not accept tandem axles.
- Are the inter-vehicle connections compatible? For example, you cannot connect a tow vehicle with a fifth wheel to a trailer with a hitch (a kingpin is required).

The event cannot be set up and executed if the event verify returns an error condition.

> **NOTE:** Technically, it is up to the developer of the calculation model to perform these tests and return an error condition so HVE can tell the user what is wrong.

If an error condition is displayed, the user must reselect from the available humans, vehicles and calculation models, or press Cancel.

After compatibility is successfully verified, the calculation model communicates back to HVE, telling HVE which Event Set-up options to make available in the Set-up menu (e.g., Driver Controls, Collision Pulse; see below), as well as which output types it produces (e.g., Messages, Trajectory Simulation, etc.). For simulations, the calculation model also tells HVE which simulation variables (e.g., Sprung Mass X, Y, Z; Tire Fx, Fy, Fz; Suspension dz; Driver Throttle Position; Steer Angle; etc.) it produces. Refer to the following chapter, Event Model Definition, for a complete description of all HVE outputs.

Event Set-up

After selecting the humans, vehicles and calculation method, the next step is to set up the event. This process involves positioning the humans and vehicles in the environment, providing velocities, and entering other event-related parameters, such as damage profiles, payloads and collision pulses.

To set up the event, perform the following steps:

1. Click on a human or vehicle in the Event Editor dialog's Selected Objects list box. The selected object will be highlighted, indicating it is the current human or vehicle.
2. Click on

   Set-up

   for the current human or vehicle. The following event set-up options are available:
3. Position/Velocity
4. Driver Controls (Throttle, Brakes, Steering, Gear Selection, Path Follower, Wheel Data)
5. Damage Profiles
6. Payload
7. Collision Pulse
8. Mesh
9. Wheels (Blow-out, Displacement, Brakes)
10. Accelerometers
11. Contacts
12. Restraints

> **NOTE:** Normally, the first option you'll choose is Position/Velocity, in order to assign positions and velocities to the selected human or vehicle.
>
> **NOTE:** Only those options meaningful for the selected human or vehicle will be available in the menu.
>
> **NOTE:** For studies involving human occupants, you must position the vehicle first.
>
> **NOTE:** Each of the event set-up options is described in detail in the following chapter, Event Model Definition.

1. Enter or edit the data requested by the selected event set-up option, and press OK to accept the data.
2. Choose and edit additional event set-up options, as desired.
3. Repeat the above steps to set up each human and/or vehicle in the event.

Event Execution

After the event is set up, the next step is to execute it. The current event is executed using the Event Controller (see Figure 15-4). To execute the current event, perform the following steps:

1. Press *Play* to begin execution of the event.

> **NOTE:** If the current calculation model is a reconstruction, it will finish almost instantly. For reconstruction programs, only the Play and Reset buttons are used.

1. Press *Pause* to stop the event momentarily, perhaps to view the current tire forces or other important parameters in the Key Results windows.
2. Press *Play* to continue execution of the event.

> **NOTE:** If you do not intervene, the event will continue to completion.

After the event runs to completion, the buttons will reset, allowing the user to replay the event, forwards and backwards, without performing the calculations. To replay the event:

1. Press *Rewind* to return the humans and vehicles to their initial position.
2. Press *Play*, *Pause* and/or *Reverse*, as desired, to view the simulation.

> **NOTE:** If you press Reset, all your current results will be erased! If you wish to save the results for viewing during Playback mode, do NOT press Reset! Use the Pause/Stop pushbutton instead.

After viewing the reconstruction or simulation, the user often wishes to make adjustments to the event and re-execute it. To make these adjustments:

1. Press *Reset*.

> **NOTE:** If you don't press Reset, any event set-up changes will have no effect!

1. Choose a human or vehicle, either by clicking on it in the Event Viewer or by selecting it from the Event Humans & Vehicles List in the Event Editor dialog.
2. Click on *Set-up* in the menu bar and choose the desired event set-up option (Position/Velocity, Driver Controls, etc.).
3. Make the desired changes.
4. Re-execute the event, as described above.

> **NOTE:** For more information about execution, refer to the documentation manual for the selected calculation model.

Selecting and Editing the Current Event

After an event is created (i.e., its humans, vehicles and calculation model have been selected and verified by the Event Information dialog), it is added to the Active Events List. The last event added to the case becomes the current event. The current event is displayed in the Event Viewer and may be edited.

To select and edit any event in the Active Events List, perform the following steps:

1. Click on the desired event name in the Active Events List. The selected event becomes the current event and will be displayed in the Event Viewer.
2. Optionally, click on the *Object Info* button on the toolbar to display its Event Information dialog (described earlier; see Figure 15-5). This allows the user to change the basic attributes of the current event (Name, Selected Humans and Vehicles, Calculation Model and Calculation Options). Press OK to update the current event.

> **NOTE:** Updating the current event removes any previous results and reinitializes the event.

1. If desired, choose a human or vehicle and edit its event set-up parameters.
2. Press *Play* to re-execute the event.

> **NOTE:** For more information about setting up and executing events, refer to the documentation manual for the selected calculation model.

Copying an Event

After an event is created, set up and executed, a complete copy of the event can be created. This is quite useful for multiple studies of the same event with only a single parameter change (e.g., different steering inputs).

To copy an event, perform the following steps:

1. Click on the desired event name in the Active Events List. The selected event becomes the current event and will be displayed in the Event Viewer.
2. Click on *Edit* in the main menu bar and choose *Copy*. The Copy Event dialog is displayed (see Figure 15-6) with the default event name for the new event (the default name is "Copy Of " + the selected event name).

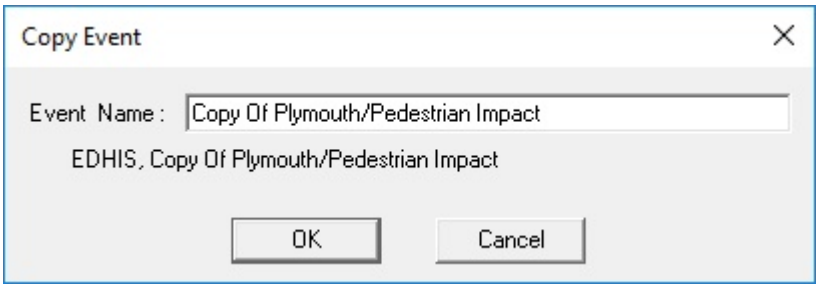


Figure 15-6: Copy Event dialog.

1. Edit the event name as desired.
2. Press OK.

A copy of the selected event is created. The new copy becomes the current event and is displayed in the Event Viewer.

Deleting an Event

An event may also be deleted. To delete the current event, perform the following steps:

1. Choose *Edit, Delete*. A message box will ask you if you're sure you want to delete the current event.

2. Press OK. The current event will be deleted.

Deleting a Vehicle Position

A vehicle position can be removed from the current event. To remove a vehicle position, perform the following steps:

1. Select the vehicle position, either by clicking on the unwanted vehicle position or by clicking on any vehicle position and choosing the unwanted position from the Path Location listbox.
2. Choose *Edit, Delete*. A message box will ask you if you're sure you want to delete the vehicle position.
3. Press Yes. The selected vehicle position will be deleted.

See also (dialog references):

Event Information · Event Setup menu · Position/Velocity · Driver Controls

Chapter 16 — Event Model Definition

The HVE Event Editor is a fully 3-dimensional environment for executing human and vehicle dynamics models. Both reconstruction-type (i.e., backwards calculating) and simulation-type (i.e., forwards calculating or predictive) models are accommodated. The HVE Event Editor inherently uses 3-D visualization to provide feedback to the user; that is, all the motion calculated by the model is displayed in HVE's 3-D Event Viewer and may be visualized from any perspective. Numeric results are displayed in viewers called Key Results windows.

The HVE execution environment has an open architecture: users may choose to execute commercially available reconstructions and simulations or develop their own. Any HVE-compatible reconstruction or simulation model may be executed. The details of writing HVE-compatible reconstruction and simulation models are provided in the HVE Developer's Toolkit references.

Overview

The HVE Event Editor is used for setting up and executing reconstructions and simulations involving the selected humans, vehicles and environment. The Event Model thus requires two basic definitions:

- Event set-up definitions for all inputs required by an HVE-compatible reconstruction or simulation model
- Event output track definitions for all outputs produced by the reconstruction or simulation model

This chapter provides these definitions, beginning with a general overview of the HVE execution environment.

General Overview

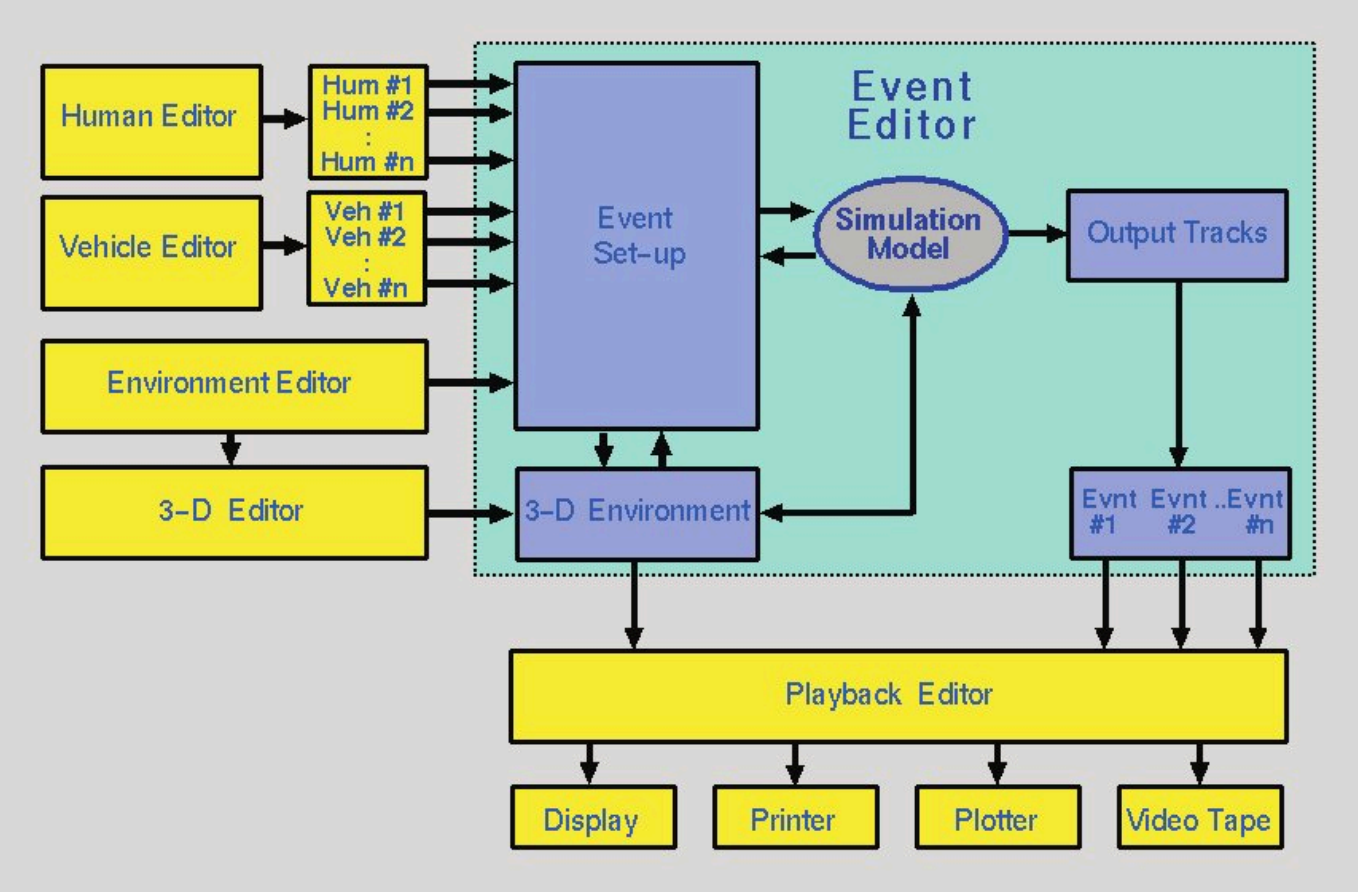


Figure 16-1: HVE system environment, showing the Human, Vehicle, Environment, Event and Playback Editors.

A block diagram for the HVE execution environment is shown in Figure 16-1. The basic components of this environment are:

- **Human Editor** — Creates and manages all the humans used in the current case.
- **Vehicle Editor** — Creates and manages all the vehicles used in the current case.
- **Environment Editor** — Creates and manages the current environment.
- **Event Editor** — Creates and manages all the events in the current case.
- **Playback Editor** — Manages all the output from each event and allows the user to specify various forms of output.

A more detailed review of the Event Editor reveals it performs the following functions:

- Loads the selected objects (humans and/or vehicles) into the current event
- Adds the requested event set-up parameters (position, velocity, vehicle damage, etc.)
- Executes the event
- Stores the program output

The event set-up definition provides the exact format for the parameters passed to the calculation model as input. The output track definition provides the exact format for the parameters produced by the calculation model and stored by the event as output. The remainder of this chapter provides the details of the event set-up and output track definitions.

Event Set-up Parameters

Event set-up parameters are selected from the HVE Set-up menu (see Chapter 4). The following options are available:

- Position/Velocity
- Driver Controls (Throttle, Brakes, Steering, Gear Selection, HVE Driver path follower, Wheel Data and — where supported — Lights)
- Vehicle Damage (Damage Profile)
- Vehicle Payload
- Collision Pulse
- Mesh
- Wheels (Blow-out, Damage, Tire-Terrain and Brakes)
- Accelerometers
- Human vs. Vehicle Contacts
- Belt and Airbag Restraint Systems Usage

Position/Velocity

The Position/Velocity Event Set-up parameters are defined in Table 16-1.

Table 16-1: Position/Velocity Event Set-up Parameters

Parameter	Units (US / SI)	Description
X or x, Y or y, Z or z	in / cm or ft / m *	Linear distance from earth-fixed origin (vehicles and human pedestrians) or vehicle-fixed origin (human occupants)
Roll, Pitch, Yaw	deg	Angular displacement about the vehicle-fixed z, y and x axes (in that order) or human segment-fixed z, y and x axes (again, the order is important). For trailers and human segments other than the pelvis, these angles are articulation angles relative to the upstream vehicle or human segment.
Fwd, Lat, Vert velocity	in/sec, mph / km/h *	Linear velocity components in the vehicle-fixed or human segment-fixed forward, lateral and vertical directions
Roll, Pitch, Yaw velocity	deg/sec	Angular velocities about the vehicle-fixed z, y and x axes or human segment-fixed z, y and x axes. For trailers and human segments other than the pelvis, these are articulation velocities relative to the upstream segment.

* Vehicle units (in / cm; mph / km/h) for human occupants; environment units (ft / m; mph / km/h) for human pedestrians and vehicles.

Position and velocity may be assigned for the current human or vehicle. The following positions and velocities may be assigned:

- Initial
- Begin Perception
- Begin Braking
- Impact
- Separation

- Point-on-curve
- End-of-rotation
- Final/Rest

The position(s) and velocities required by a reconstruction or simulation model vary from model to model. In general, simulation-type models require only initial positions and velocities. Reconstruction-type models normally require positions at impact and rest. However, all the above positions may be entered for any model. The reconstruction or simulation model knows which positions are actually used by the model. Positions which are entered but not used are called

target positions

because they provide visual feedback to the user regarding how close the simulated path matches the desired path.

> **NOTE:** Target humans and vehicles are translucent, to distinguish them from positions which are used by the model.

When a human or vehicle is selected for positioning, its Position/Velocity dialog is displayed. The Position/Velocity dialog allows the user to enter the position and velocity according to the following procedures.

Positioning Unit Vehicles

Single vehicles, and the tow vehicle of vehicle-trailer combinations, are positioned relative to the earth-fixed coordinate system. X, Y, Z linear positions and roll, pitch, yaw orientations may be entered. Total velocity and sideslip angle are entered. Vertical velocity may be entered for 3-D models. The forward (U) and lateral (V) velocity are auto-computed from the total velocity, sideslip angle and vertical velocity, but remain editable; editing U, V or the sideslip angle recomputes the other components.

(updated)

Positioning Trailers

Trailers and dollies are positioned relative to the vehicle pulling them; thus only relative roll, pitch and yaw positions and velocities are required.

> **NOTE:** Trailer and dolly names are not displayed in the selected objects list. To select a trailer or dolly for event set-up (positioning, driver controls, etc.), click on the desired object in the Event Viewer.

Positioning Human Occupants

Human occupants are positioned using the CG of the pelvis segment, and are positioned relative to the vehicle-fixed coordinate system. x, y, z linear positions and roll, pitch, yaw orientations of the pelvis may be entered. i, j, k linear velocities and roll, pitch, yaw angular velocities are entered relative to the pelvis segment's local coordinate system.

Positioning Human Pedestrians

Human pedestrians are positioned using the CG of the pelvis segment, and are positioned relative to the earth-fixed coordinate system. X, Y, Z linear positions and roll, pitch, yaw orientations of the pelvis may be entered. i, j, k linear velocities and roll, pitch, yaw angular velocities are entered relative to the pelvis segment's local coordinate system.

For both human occupants and pedestrians, segments other than the pelvis are positioned using joint articulation angles relative to the upstream (i.e., closer to the pelvis) adjoining segment; thus only relative roll, pitch and yaw positions and

velocities are required.

> **NOTE:** 2-D, yaw-plane models do not use Z, roll or pitch information. HVE will automatically make these fields non-user-editable and calculate the values using AutoPosition.

Deleting Human and Vehicle Positions

Human and vehicle positions may be deleted from an event. This is done simply by selecting the undesired human or vehicle in the viewer, then choosing *Delete* from the *Edit* menu.

> **NOTE:** If no object is selected before choosing Delete, you will be asked if you want to delete the entire event! Therefore, you must first select a human or vehicle before choosing Delete from the Edit menu.

Rotation Direction

According to the SAE coordinate system, rotation direction is defined using the right-hand rule. Thus, positive rotation is clockwise about a given axis. For example, looking down on the X-Y plane from above, a vehicle turning right will have a positive rotation direction.

It follows that the coordinate system convention has implications when entering a sequence of positions (e.g., Impact, Separation, Rest). When two successive positions are entered, the direction of rotation is defined by the difference in the heading angles. For example, if the heading angle at impact, Ψ_{imp}, is 35 degrees and the heading angle at rest, Ψ_{rest}, is 80 degrees, then the difference from impact to rest is $\Psi_{rest} - \Psi_{imp} = 80 - 35 = +45$ degrees. The positive sign indicates the direction of rotation from impact to rest is positive (i.e., clockwise). In general, any rotation direction is defined as follows:

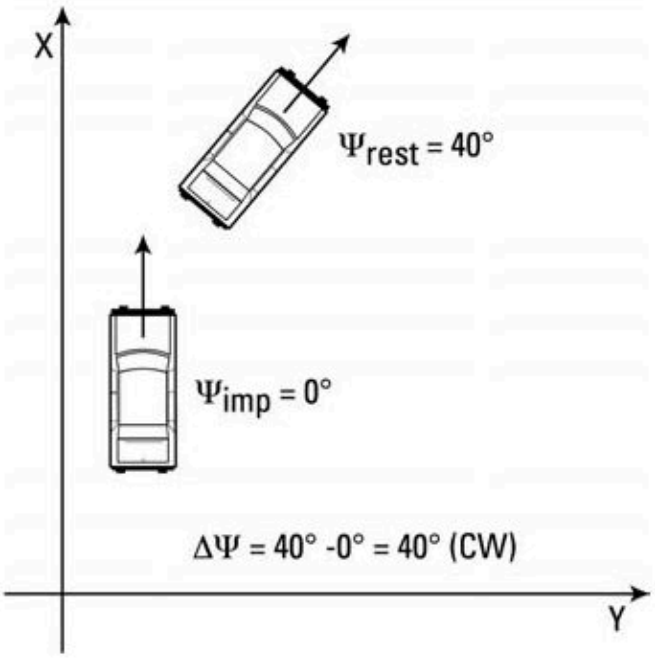
```
ΔΨ = Ψ(n) – Ψ(n–1)
```

where

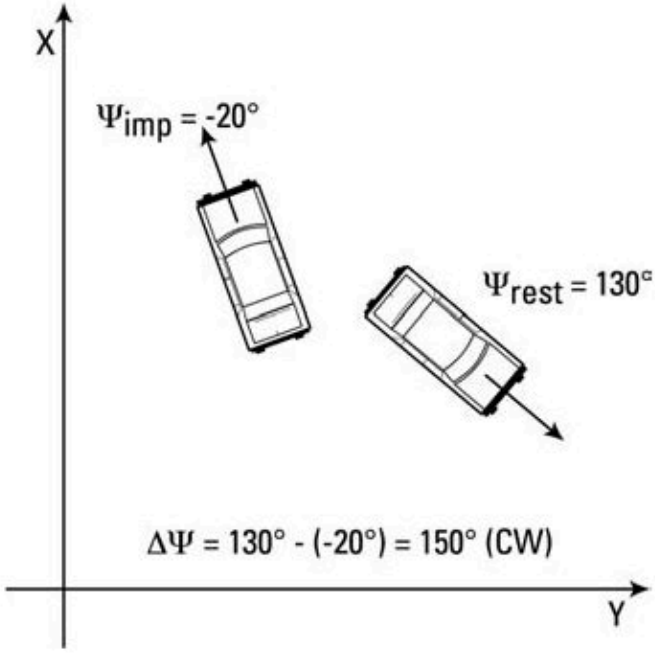
Symbol	Meaning
n	a position in a sequence of positions (e.g., rest)
n−1	the previous position in a sequence of positions (e.g., impact)

If ΔΨ is positive, the direction is clockwise; if ΔΨ is negative, the direction is counter-clockwise. The above applies to roll and pitch rotations as well as yaw (heading angle) rotations.

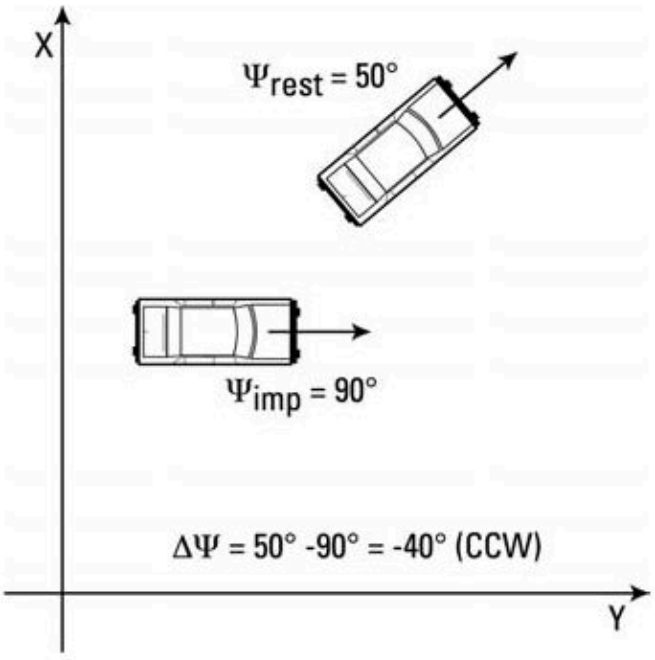
> **NOTE:** This definition is especially important to EDVAP/EDCRASH users. In those earlier programs, the direction of rotation was entered directly by the user as Clockwise, Counter-clockwise or None. In HVE, the direction of rotation is not entered directly; it is determined as shown above.


X
Ψrest = 40°
Ψimp = 0°
ΔΨ = 40° -0° = 40° (CW)
Y

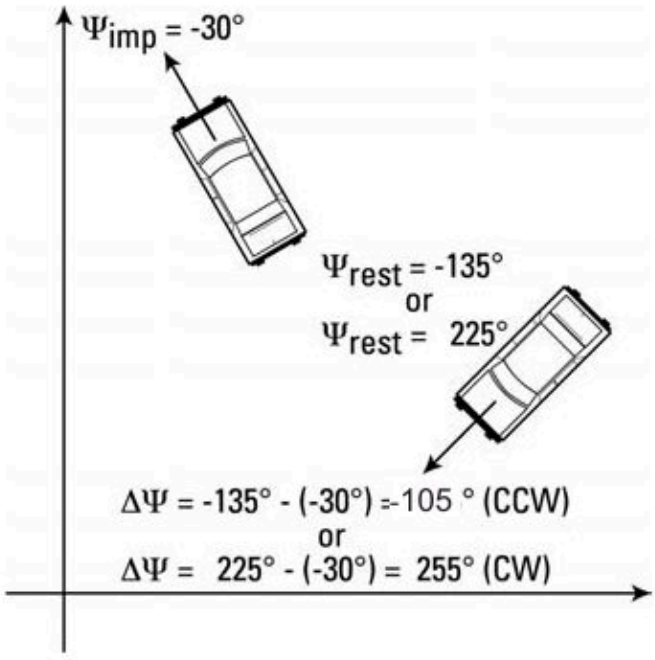
(a)


X
Ψimp = -20°
Ψrest = 130°
ΔΨ = 130° - (-20°) = 150° (CW)
Y

(b)


X
Ψrest = 50°
Ψimp = 90°
ΔΨ = 50° -90° = -40° (CCW)
Y

(c)


Ψimp = -30°
Ψrest = -135°
or
Ψrest = 225°
ΔΨ = -135° - (-30°) = -105 ° (CCW)
or
ΔΨ = 225° - (-30°) = 255° (CW)

(d)

Figure 16-2: Examples of rotation direction for various position sequences.

Driver Controls

The Driver Control Event Set-up parameters are defined in Tables 16-2 through 16-6. Driver controls may be selected for the current vehicle. The available driver controls are:

- Throttle
- Brakes
- Steering
- Gear Selection
- HVE Driver (Path Follower)
- Wheel Data
- Lights *(updated: a Lights page is available when the calculation method supports light systems and the vehicle has lights defined)*

The Driver Controls dialog is a tabbed property sheet; the pages that appear depend on the capabilities of the current event's calculation method and on the selected vehicle. For the current page-by-page reference, see Driver Controls. The Driver Controls options are described below.

Throttle

The Throttle Table Event Set-up parameters are defined in Table 16-2.

Table 16-2: Throttle Table Event Set-up Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of throttle input
Throttle Input	%/100, lb / N or %/100	Current value for throttle input

Throttle controls are available for simulations, and are used to accelerate the vehicle. Three methods of throttle control tables are available (see also the Throttle page):

- **Wide-open Throttle (Percent WOT)** — This method allows the user to enter a table of throttle position versus time. The resulting table determines how much engine power is applied. An entry of 100 percent applies a drive torque associated with 100 percent of the current vehicle's engine power at the current engine speed. *(updated: this option engages the vehicle's engine and drivetrain data.)*

> **NOTE:** This method is available only if the current simulation includes an engine model with the ability to calculate drive torque according to the current engine speed and throttle position. A Gear Table must also be supplied.

- **Tractive Effort** — This method allows the user to enter a table of tractive effort (total force accelerating the vehicle) versus time. The resulting table determines how much accelerating force is applied at each drive wheel.

> **NOTE:** The Tractive Effort Table only allows entries at drive wheels, as specified by the Vehicle Information dialog.

- **Percent Available Friction** — This method allows the user to enter a table of available tractive effort (percentage of total available frictional force accelerating the vehicle) versus time. The resulting table determines how much accelerating force is applied at each drive wheel. The calculations are performed by the simulation model, which multiplies the entered value by the currently available tire friction and vertical tire load.

> **NOTE:** The Percent Available Friction Table only allows entries at drive wheels, as specified by the Vehicle Information dialog.

Brakes

The Brake Table Event Set-up parameters are defined in Table 16-3.

Table 16-3: Brake Table Event Set-up Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of brake input
Brake Input	lb / N or %/100	Current value for brake input

Brake controls are available for simulations. Three methods are available (see also the Brake page):

- **Pedal Force** *(called "At Pedal" in the legacy edition)* — This method allows the user to enter a table of brake pedal force versus time. The resulting table determines how much brake torque is applied at each wheel. The brake torque at each wheel is the product of the entered table value, master cylinder ratio, proportioning rate and wheel torque ratio. *(updated: this is the only option that engages the vehicle's brake system — and Brake Designer — data.)*

> **NOTE:** The Pedal Force option is available only if the current simulation includes a brake model with the ability to calculate brake torque according to the brake system parameters and brake pedal force.

- **Brake Force** — This method allows the user to enter a table of brake force (total force decelerating the vehicle) versus time. The resulting table determines how much decelerating force is applied at each wheel.
- **Percent Available Friction** — This method allows the user to enter a table of available braking force (percentage of total available frictional force braking the vehicle) versus time. The resulting table determines how much brake force is applied at each wheel. The calculations are performed by the simulation model, which multiplies the entered value by the currently available tire friction and vertical tire load.

Steering

The Steer Table Event Set-up parameters are defined in Table 16-4.

Table 16-4: Steer Table Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current level of steer input
Steer Input	deg	Current value for steer input

Steering controls are available for simulations. Two methods are available:

- **At Steering Wheel** — This method allows the user to enter a table of steering wheel angle versus time. The entered value is divided by the axle's steering gear ratio to determine the nominal steer angle at the tire.

> **NOTE:** The actual steer angle of an individual tire may be affected by roll steer and/or toe-in. The inclusion of these parameters is simulation-dependent.

- **At Axle** — This method allows the user to enter a table of steer angle at each tire of a steerable axle. The entered value is applied directly to the tire, and is unaffected by any roll steer or toe-in.

(updated: the current Steer page also provides a Use Ackermann Steering *check box — when checked, the simulation applies Ackermann geometry to the steer angles, so that the inside and outside tires of a steerable axle are steered through slightly different angles. See the Steer page.)*

Gear Selection

The Gear Selection Table Event Set-up parameters are defined in Table 16-5.

Table 16-5: Gear Selection Table Event Set-up Parameters

Parameter	Units (US / SI)	Description
Time	sec	Time associated with the current gear shift
Gear Selection	n/a	Current gear number for transmission or differential

The Gear Selection dialog is available for simulations. This dialog allows the user to enter a table of gear selection versus time. Separate tables may be entered for the transmission and differential.

> **NOTE:** This option is available only if the current Throttle Method is Wide-open Throttle.

The dialog has a user-selectable option for transmission and differential.

> **NOTE:** The differential option is available only if the selected vehicle has more than one differential ratio.

HVE Driver (Path Follower)

The HVE Path Follower — presented in the current version as the **HVE Driver** page — allows the user to define a 3-D path using target positions. Using this path, the simulation model determines the steering, throttle and braking inputs required to make the vehicle follow the path. The HVE Path Follower Event Set-up parameters are defined in Table 16-6.

(updated: the current HVE Driver page offers two driver-model versions, Ver. 1 *and* Ver. 2*, selected by radio buttons enabled when* Use Path Follower *is checked, and a Path Source selector — Position/Velocity dialog positions, or a path table (reserved). See the HVE Driver page.)*

The HVE Path Follower includes several features, some required and some optional. These features are:

- **Path Generator (required)** — user-defined path positions (e.g., Initial, Begin Braking, etc.) to define the attempted path
- **General Parameters (required)** — user-entered parameters required for HVE Path Follower operation
- **Method (required)** — user-entered parameters defining how the correction is accomplished

- **Speed Follower (optional)** — user-entered velocities at each path position are used to define an attempted speed at each point on the path
- **Neuro-muscular Filter (optional)** — user-entered parameters that define driver physiological capabilities

Path Generator

A primary input to a path follower is the procedure used to define the attempted path. The HVE Path Follower uses up to eight user-defined path positions (see Event Set-up, Assigning Positions). These positions and orientations are used as nodes in a 3-D spline path (see Figure 16-3). It is this spline path that the vehicle attempts to follow.

General Parameters

The basic parameters required for use of the HVE Path Follower are the path (described above), Start Time and Sample Interval, Preview Distance (the point ahead of the vehicle where the driver is actually looking and presumably wants to go), the Allowable Path Error and the level of Lateral Acceleration acceptable to the driver. *(updated: the current Driver tab specifies the preview point by a* Driver Preview Time *(sec) together with a* Driver Minimum Preview Distance *(ft) used at low speeds, plus a* Path Error Null Distance, *a* Max Speed Error *and a* Driver Comfort Level *(g); see the HVE Driver page.)*

Method

Two methods are available in the HVE Path Follower. These are:

- **Variable Steering** — The path correction is accomplished by means of a steering correction. In this case, the user supplies an Initial Steer Angle, Maximum Steering Wheel Velocity, Steering Correction Factor and Steer Damping.
- **Variable Torque** — The path correction is accomplished by means of a torque application at the steerable wheels. In this case, the user supplies an Initial Steering Wheel Torque, Maximum Steering Wheel Torque, Torque Correction Factor and Torque Damping.

> **NOTE:** *(updated)* The Variable Torque option is currently disabled in the user interface; Variable Steering is the available path follower method.

Table 16-6: HVE Path Follower Parameters

Parameter	Units (US / SI)	Description
Start Time	sec	Simulation time for start of path follower
Sample Time	sec	Time increment for path sampling
Driver Preview Distance	ft / m	Distance ahead of vehicle where driver is looking *(updated: entered via Driver Preview Time and Minimum Preview Distance in the current dialog)*
Max Path Error	ft / m	Allowable distance from desired path to point projected ahead of vehicle at the Driver Preview Distance
Max Lat Accel	g	Driver's limit for comfortable lateral acceleration
Path Follower Method	—	Method used to accomplish path following (Variable Steer or Torque)
Initial Steer Angle	deg	Steering wheel angle at Driver Start Time
Max Steer Velocity	deg/sec	Limit on steering wheel angular velocity
Steer Correction Factor	deg/ft / deg/m	Amount of steering correction per unit of path error
Steer Correction Damping	deg-sec/ft / deg-sec/m	Damping, used to limit steering activity
Initial Steering Torque	in-lb / N-m	Steering wheel torque at Driver Start Time
Max Steering Torque	in-lb / N-m	Limit on steering wheel torque
Torque Correction Factor	in-lb/ft / N-m/m	Amount of steering torque correction per unit of path error
Torque Damping	in-lb-sec/ft / N-m-sec/m	Damping, used to limit steering activity
Max Speed Error	mph / km/h	Allowable difference between current speed and desired speed
Max Throttle Pedal	%/100	Maximum allowable pedal position
Max Brake Pedal	lb / N	Maximum allowable pedal force
Driver Lag Time	sec	Neuro-muscular filter lag time
Driver Lead Time	sec	Neuro-muscular filter lead time
Driver Time Delay	sec	Neuro-muscular filter time delay

Speed Follower

The speed follower option attempts to maintain the required speed, as established by the user-entered velocities at each path position (if a velocity for any path position is not entered, an error message will be issued and the method will abort). In addition, the user supplies an Allowable Speed Error, Maximum Throttle Application and Maximum Brake Pedal Force.

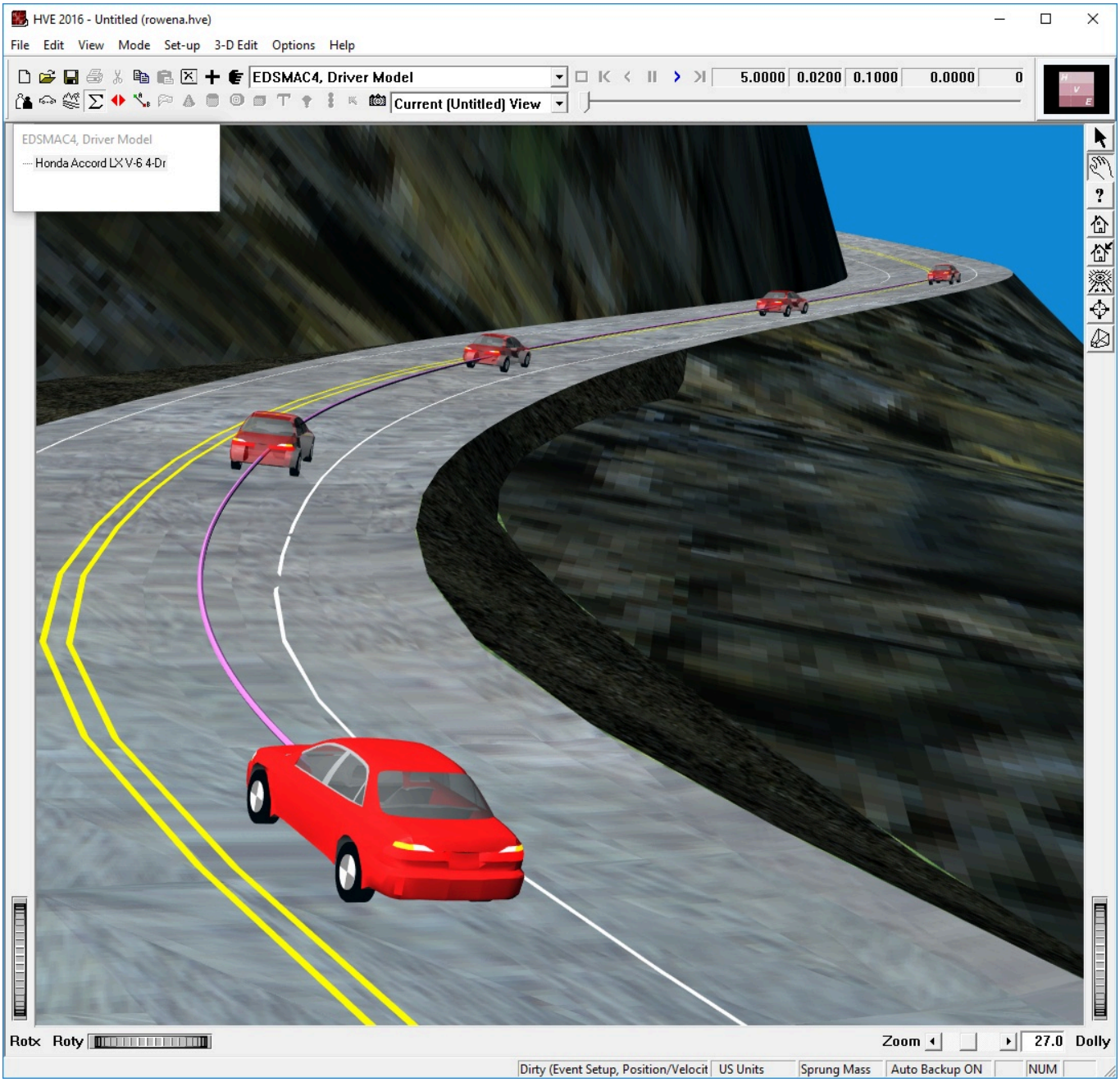


Figure 16-3: The HVE Path Follower uses up to eight user-assigned positions to define a 3-D path. Intermediate path positions are determined using a 3-D spline.

> **NOTE:** *(updated)* The legacy edition stated that the Speed Follower option was not implemented. The Speed follower is implemented in the current version: user-entered velocities at each path position define an attempted speed at each point on the path, and the driver model applies throttle and brake pedal inputs to achieve those speeds. See the Speed Follower page.

Neuro-muscular Filter

The neuro-muscular filter from the original HVOSM VD-2 model has been included in the HVE Path Follower. Originally developed for NASA, the neuro-muscular filter represents a simplified model of the physiological operator

which incorporates a Time Delay, Lead Time and Lag Time. These parameters correspond to the first-order effects of the neurological and muscular systems of a human driver.

(See the Filter page.)

To use the HVE Path Follower, first you must assign at least two path positions for the vehicle, then perform the following steps:

1. Choose *Driver Controls* from the Set-up menu.
2. Choose the *HVE Driver* (Path Follower) page. The HVE Path Follower parameters will be displayed.
3. Assign the desired HVE Path Follower parameters.
4. Press OK when the desired parameters are assigned.

Wheel Data

The Wheel Data Event Set-up parameters are defined in Table 16-7.

Table 16-7: Wheel Data Event Set-up Parameters

Parameter	Units (US / SI)	Description
Pre-impact Drag Factor	%/100	Linear deceleration rate from Begin Braking to Impact
Percent Friction	%/100	Constant percent of available friction at each wheel used to decelerate the vehicle between impact and rest
Steer Angle	deg	Constant steer angle at each wheel

The Wheel Data dialog is available for reconstruction models. This dialog allows the user to enter the following information:

- **Pre-impact Deceleration Rate (Drag Factor)** — The entered value is multiplied by the gravitational constant to determine the total vehicle deceleration between the user-entered Begin Braking and Impact positions. This value is relevant only if the user has supplied a Begin Braking position for the current vehicle.
- **Rotation/Lateral Skidding Checkbox** — This checkbox may be used by reconstruction models to trigger special calculations for spinning vehicles.

> **NOTE:** Spinning vehicles have tire forces which vary, resulting in a variable deceleration rate between impact and rest!

- **Percent Available Friction used at each wheel** — The entered value represents the percentage of the available friction force used in longitudinal braking at each wheel during the post-impact phase.
- **Steer Angle at each wheel** — The entered value is used to determine the steer angle at each wheel. This angle is assumed to remain constant, and applies only during the post-impact phase.

Damage Profile

The Damage Profile Event Set-up parameters are defined in Table 16-8.

Table 16-8: Damage Profile Event Set-up Parameters

Parameter	Units (US / SI)	Description
CDC	(none)	Collision Deformation Classification, 7-character damage code describing location and character of damage
PDOF	deg	Principal Direction Of Force, the direction of the collision impulse (same as direction of delta-V)
Impulse Center	in / cm	Vehicle-fixed x,y coordinates of the impulse center
EES	mph / km/h	Equivalent Energy Speed, the barrier impact speed required to cause the observed vehicle damage
Width	in / cm	Total width of damage
Offset	in / cm	Longitudinal or lateral distance from the CG to the center of the damage profile
Crush Depth	in / cm	Measured depth of crush at up to 10 points along the damage profile
A Stiffness	lb/in / N/cm	Force per unit of damage width required to initiate measurable damage (for each crush zone)
B Stiffness	lb/in² / N/cm²	Force per unit of crush depth per unit of damage width required to cause the measured damage (for each crush zone)

The Damage Profile dialog allows the user to enter a damage profile for the selected vehicle. The damage profile is obtained by post-crash inspection of the vehicle, and provides a significant amount of useful information about the collision, including:

- Damage Energy
- Peak Force
- Linear and angular velocity change
- Principal Direction of Force (PDOF)

This information, in turn, can be used to help estimate impact speed and collision severity.

CDC

HVE's Damage Profile dialog asks the user for a Collision Deformation Classification (CDC), a seven-character damage code describing the vehicle damage. The CDC is an SAE Recommended Practice, and is defined in SAE J224, portions of which are included in the HVE Help Index.

The entered CDC is used to define a default PDOF and damage profile, which includes damage width, depth and location. The default data provided by the CDC are used to fill in the fields in the Damage Profile dialog. The resulting delta-V, damage energy and peak force are calculated and displayed in the dialog.

> **NOTE:** The calculated values define what would happen if the current vehicle struck a rigid barrier, not another vehicle! To determine what would happen during a collision with another vehicle requires information about the second vehicle (in particular, its mass and damage energy).

The specific damage profile parameters are defined below.

PDOF

Simply stated, the PDOF is the direction of the impulse. This, of course, has physical significance, because the direction of the impulse is the same as the direction of the delta-V. In fact, because the damage analysis has no way of computing the forward and lateral components of the delta-V, the user-entered PDOF is used for this purpose.

> **NOTE:** If scene data are entered, the PDOF can often be calculated. This serves as an excellent cross-check on the user-entered value.

The PDOF is assigned by the CDC as the closest hour angle (or clock direction), as shown in Figure 16-5. Each hour equals 30 degrees. The Damage Profile dialog displays the resulting PDOF in degrees. It may be edited by the user.

Use Newton's 3rd Law

Some reconstruction programs (e.g., EDCRASH) are able to calculate the PDOF of one vehicle in a two-vehicle collision based on the impact heading angles and the PDOF of the other vehicle. Clicking on the

Use Newton's 3rd Law

check box executes that option.

> **NOTE:** You can only choose this option for one of the vehicles; if you choose this option for both vehicles (or the other vehicle is a barrier), the program will normally display an error message.

Impulse Center

The Impulse Center X,Y coordinates define the vehicle-fixed location through which the PDOF acts. The default impulse center is calculated by summing moments and forces for each zone (see Damage Profile Zones, below) in the damage profile. These coordinates are user-editable.

> **NOTE:** Once edited by the user, further changes to the damage profile data (width, crush depths, offset) will no longer affect the displayed values for the impulse center.

Basis

After entering a CDC, HVE calculates a default damage profile, delta-V and other damage-related information. This provides the starting point for damage-based calculations. The results may be further refined by editing either the EES or the Damage Profile.

> **NOTE:** Whether the reconstruction program uses the damage profile or EES is program-dependent.

EES

If the Basis is EES (Equivalent Energy Speed), the default value for EES is assigned from the total delta-V (initially calculated from the CDC). The EES may then be user-edited.

Damage Profile Zones

When the Basis is Damage Profile, the damage data may be edited, causing the delta-V, Damage Energy and Peak Force to be updated. This method also allows the user to edit the A and B stiffness coefficients that are used in the calculations

(see below). The Damage Profiles drop-down list allows the user to select the number of Crush Zones (and therefore, the number of Crush Depths) in the damage profile.

> **NOTE:** See also Crush Zones and Crush Depths, later in this section.

Damage Width

The Damage Width is the width of the damaged region (see Figure 16-4). The default value assigned by the CDC does not include induced damage (non-contact damage adjacent to the area of actual contact with another vehicle). In most cases, the user should include induced damage if the goal is to estimate delta-V.

Damage Offset

The Damage Offset is simply the longitudinal or lateral distance from the CG to the midpoint of the damage profile (see Figure 16-4).

Crush Depths

A table of crush depths defines the shape of the damage profile. Up to 10 equally spaced crush depths may be provided along the total damage width, as shown in Figure 16-4.

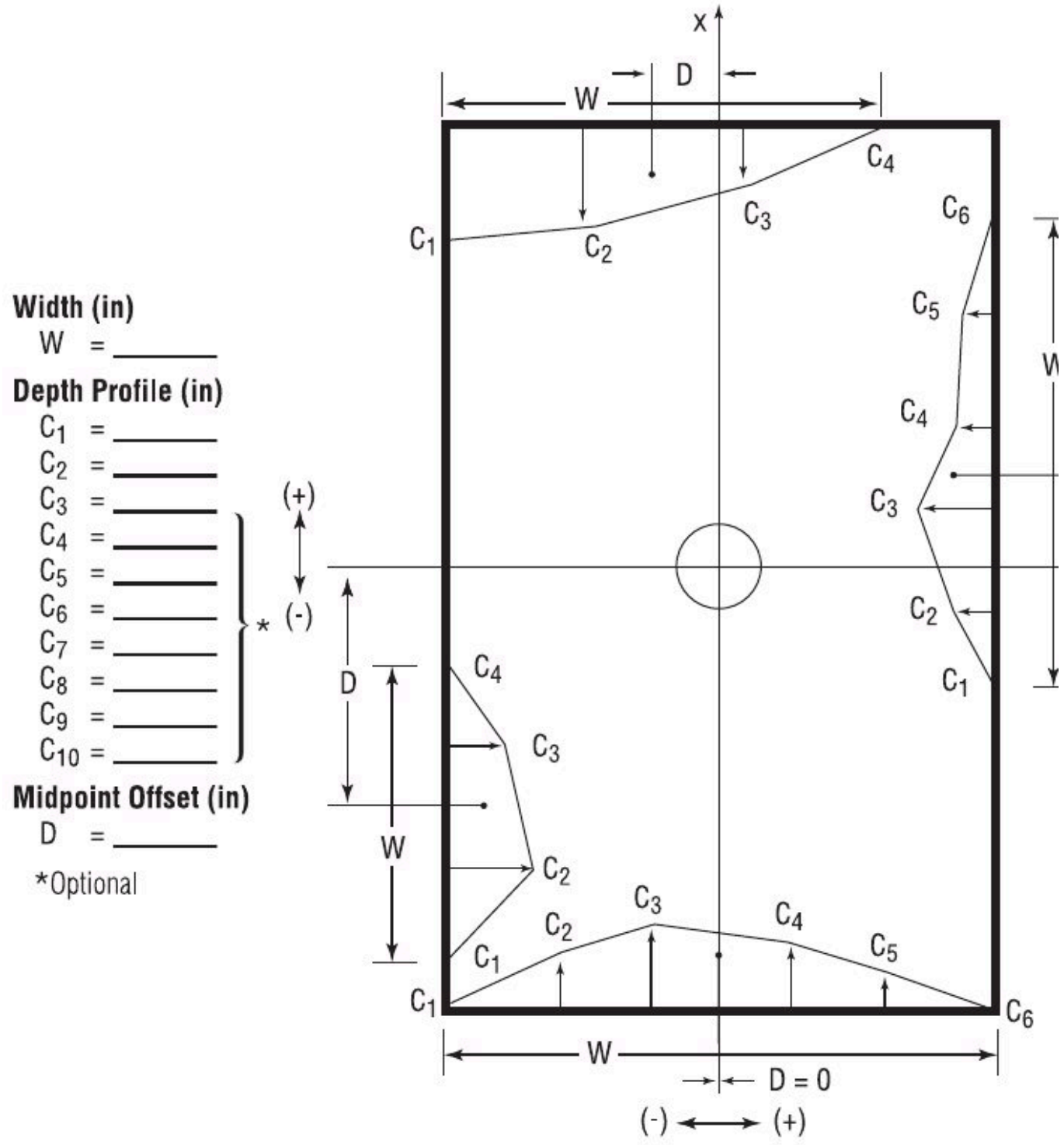


Figure 16-4: Damage Profile examples.

Stiffness Coefficients

The damage analysis requires empirical coefficients determined by crash tests. These coefficients are called stiffness coefficients, because they define the structural stiffness of the vehicle. Two coefficients are provided:

- **A Coefficient** — The A coefficient has units of force per unit of damage width, and defines the force necessary to begin crushing the vehicle's exterior.
- **B Coefficient** — The B coefficient has units of force per unit of crush depth per unit of damage width, and defines the linear spring rate of the vehicle's exterior.

Given values for A and B, the crush force per unit of damage width is assumed to have a linear relationship with crush depth. Figure 16-6 shows this relationship. The A and B coefficients are provided in the A Stiffness and B Stiffness dialogs, respectively (see Menu Reference, Set-up menu). The A Stiffness and B Stiffness dialogs allow the user to assign equal coefficients for every zone or vary the coefficients from zone to zone.

Crush Zones

The area between any two crush depth measurements is called a crush zone; thus the number of crush zones will always be one less than the number of entered crush depths, as shown in Figure 16-4.

Obviously, the actual stiffness of a vehicle varies according to the structure absorbing the crush, especially on the side of a vehicle, where one area, such as the quarter panel, contains unsupported sheet metal, while another area, such as a wheel, might be quite rigid. A separate set of A and B coefficients may be assigned to each crush zone.

> **NOTE:** The A and B coefficients are not independent from each other; one should never be adjusted without adjusting the other. In addition, the percentage of adjustment should be the same for both the A and B stiffness coefficients.

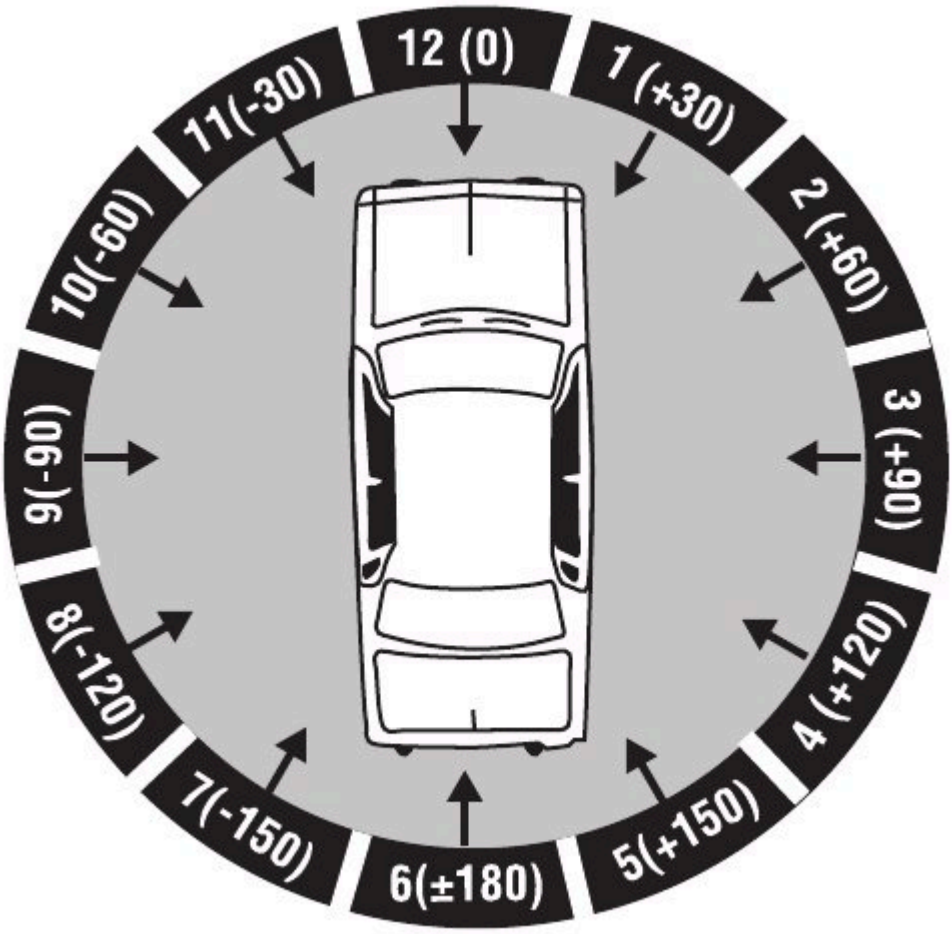


Figure 16-5: The default PDOF is assigned by its clock direction (12, 01, 02, etc.) in the first two characters of the CDC.

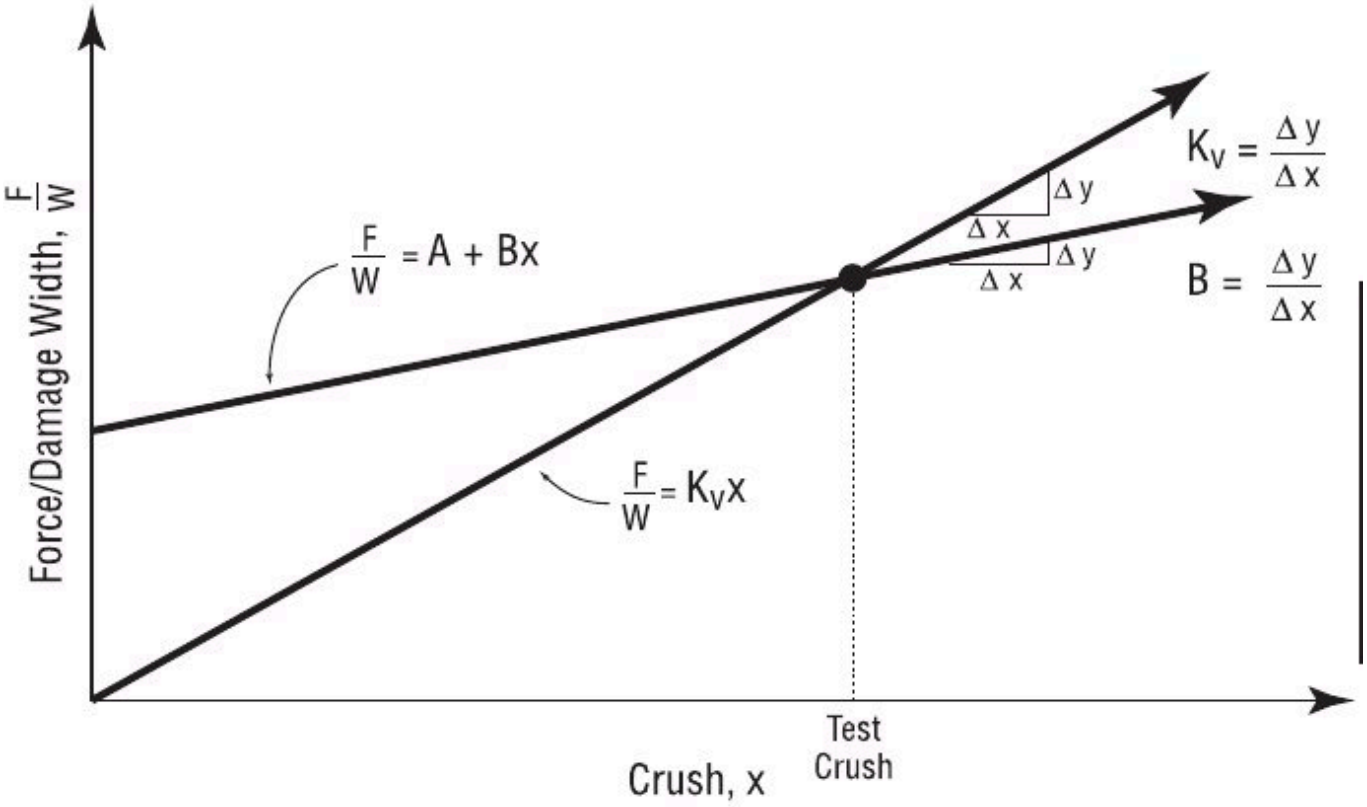


Figure 16-6: Force vs. Crush characteristics.

Mesh

The Vehicle Mesh Event Set-up parameters are defined in Table 16-9.

Table 16-9: Mesh Event Set-up Parameters

Parameter	Units (US / SI)	Description
Maximum Side Length	in / cm	Determines the longest side of a polygon
Inter-vehicle Friction	—	Inter-vehicle friction coefficient for the selected pair of vehicles
Relaxation Length	%/100	Determines the amount of reduction in displacement vector

The Vehicle Mesh Event Set-up options are:

- **Tessellate** — The user enters a value for the maximum length of a side for any triangle in the vehicle mesh. Each triangle is then re-evaluated and any side longer than the specified value is again halved until all sides of every triangle are no larger than the specified Maximum Side Length.
- **Inter-Vehicle Friction** — The user specifies the inter-vehicle friction for the selected pair of vehicles (or vehicle body and environment) in a vehicle simulation.
- **Relaxation Length** — This value represents the amount (as a percentage of deformation) of reduction in vertex displacement.

Payload

The Payload Event Set-up parameters are defined in Table 16-10.

Table 16-10: Payload Event Set-up Parameters

Parameter	Units (US / SI)	Description
x, y, z coordinates	in / cm	x, y and z distances from the vehicle CG to the Payload CG
Weight	lb / N	Payload weight
Rotational Inertia	lb-sec²-in / kg-m²	Payload rotational inertia about its x, y and z axes

The Payload dialog allows the user to assign a payload to the selected vehicle. The payload may be any object which adds inertia to a vehicle that is not accounted for in the vehicle's inertial properties. Examples include cargo and occupants.

The availability of these fields is dependent on the reconstruction or simulation model. For example, a 2-D model will not ask the user for roll or pitch inertia because they aren't required. Some models do not allow a lateral payload offset, so the y-coordinate is not requested.

The vehicle's center of gravity shifts when a payload is added. Therefore, HVE will calculate and update the earth-fixed coordinates of the vehicle's CG. The new X, Y, Z coordinates are displayed in the Position/Velocity dialog. This same updating of the vehicle CG location occurs when occupants are added. Refer to the User's Manual for the specific calculation model to see if it supports automatic repositioning of the vehicle CG when occupants are added to the vehicle.

Collision Pulse

The Collision Pulse Event Set-up parameters are defined in Table 16-11.

Table 16-11: Collision Pulse Event Set-up Parameters

Parameter	Units (US / SI)	Description
Pulse Type	—	Type of collision pulse (Position, Velocity, Acceleration, or Force/Moment)
Longitudinal, Lateral and Vertical linear motion	in / cm, mph / km/h, g or lb / N	Vehicle-fixed components of the linear motion
Roll, Pitch, Yaw angular motion	deg, deg/sec, deg/sec² or ft-lb / N-m	Angular motion about the vehicle's roll, pitch and yaw axes
Pulse Factors	%/100	Multipliers for individual directions
Threshold Acceleration	g	Minimum total linear acceleration required for inclusion in the collision pulse
Tstart	sec	Event time at which the collision pulse begins
Impulse Center x, y, z	in / cm	Vehicle-fixed coordinates of the impulse center (Force/Moment pulse type only)

The Collision Pulse parameters define the acceleration vs. time response of a vehicle. The collision pulse is used for occupant simulations.

The Pulse Factors are a convenient way to evaluate the outcome by reducing or increasing the acceleration(s) by a user-defined percentage.

Setting the Threshold Acceleration allows the user to specify a minimum acceleration for the collision pulse. The table begins only after the acceleration reaches this minimum value and ends after the acceleration drops below this value.

Tstart specifies the time at which the collision pulse begins. Setting Tstart to a non-zero value allows the user to choose only a portion of the collision pulse, if desired.

Impulse Center x, y, z specifies the vehicle-fixed coordinates for the point of application of the force in a Force/Moment collision pulse table.

Wheels

The Wheels Event Set-up options are:

- **Tire Blow-out** — Parameters that activate the HVE Tire Blow-out Model, a time-dependent variation in tire properties.
- **Wheel Damage** — Parameters that allow the time-dependent relocation of wheels and wheel lock-up, as from a collision.
- **Brakes** — Parameters that model the slack adjusters and lining and drum temperatures.
- **Tire-Terrain** — Parameters that model tire-terrain interaction.

The Wheel Event Set-up parameters are defined in Table 16-12.

Table 16-12: Wheel Editing Event Set-up Parameters

Parameter	Units (US / SI)	Description
Axle Number	—	Axle index (1, 2 or 3)
Side	—	Side index (Right or Left)
Location	—	Tire location index (Inner or Outer)
Tire Is Blown	—	Flag indicating the tire is blown
AutoStart	—	Flag indicating the start time is set by the simulation
Start Time	sec	Simulation time at which blow-out or wheel displacement starts
Duration	sec	Duration over which air loss or wheel displacement occurs
Stiffness Factor	—	Multiplier for tire stiffness parameters
Rolling Resistance Factor	—	Multiplier for tire rolling resistance
Change in Coordinates	in / cm	Vehicle-fixed wheel displacement distance
Change in Camber	deg	Vehicle-fixed wheel camber change angle
Percent Peak Lock-up Torque	%/100	Percentage of nominal torque required to lock the wheel
Initial Lining Temp	deg F / deg C	Temperature of the brake lining at the start of the simulation
Initial Drum Temp	deg F / deg C	Temperature of the brake drum at the start of the simulation
Adjuster Slack	in / cm	Slack adjuster stroke before brake torque begins

Table 16-12: Wheel Editing Event Set-up Parameters (continued — Tire-Terrain)

Parameter	Units (US / SI)	Description
Axle Number	—	Axle index (1, 2 or 3)
Side	—	Side index (Right or Left)
Location	—	Tire location index (Inner or Outer)
Tire-Terrain Model	—	Tire-Terrain Model option
Sidewall Impact	—	Flag indicating the Sidewall Impact option is to be used
Radial Adjustment Increment	in / cm	Incremental change in radial spring length used in the spring force calculation
Angular Span	deg	Sweep angle for radial springs
Angular Increment	deg	Angular increment between radial springs
Sidewall Slide Friction Coefficient	—	Tire sidewall slide friction coefficient
Number of Sidewall Springs	—	Number of sidewall springs attached to each radial spring
Spring Number	—	Spring index
Spring Radial Dist from Wheel Center	in / cm	Radial distance from the wheel center to the current lateral spring
Spring Lateral Dist to Wheel Center	in / cm	Distance from the tire centerline plane to the tire sidewall for the current spring
Sidewall Spring Stiffness Ratio	—	Ratio of sidewall spring stiffness to radial spring stiffness
Spring Saturation Deflection	in / cm	Maximum allowable deflection for the current sidewall spring

Accelerometers

The Accelerometers Event Set-up parameters are defined in Table 16-13.

Table 16-13: Accelerometers Event Set-up Parameters

Parameter	Units (US / SI)	Description
Device Number	—	Device selector number
Device Is Used	—	Flag indicating the selected device is in use
x-coord, y-coord, z-coord	in / cm	Vehicle-fixed coordinates of the accelerometer

Simulations can report the local velocity and acceleration for each of up to five vehicle-fixed accelerometer locations.

- **Device Number** is the index for the specified accelerometer.
- The **Device Is Used** check box is used to activate the specified accelerometer.
- **x, y, z-coordinates** are the vehicle-fixed coordinates for the specified accelerometer.

Contacts

The Contacts Event Set-up parameters are defined in Table 16-14.

Table 16-14: Contact Event Set-up Parameters

Parameter	Units (US / SI)	Description
Source Segment	n/a	Name of the source segment (may be either a human ellipsoid or a vehicle contact surface)
Target Segment	n/a	Name of the target segment (may be either a human ellipsoid or a vehicle contact surface)
Material Name	—	User-editable material name
Constant	lb / N	Force required to initiate deflection
Linear Stiffness	lb/in / N/cm	Linear material deformation coefficient
Quadratic Stiffness	lb/in^2 / N/cm^2	Quadratic material deformation coefficient
Cubic Stiffness	lb/in^3 / N/cm^3	Cubic material deformation coefficient
Damping Constant	lb-sec/in / N-sec/m	Material velocity-dependent deformation constant
Friction Constant	—	Inter-segment friction coefficient
Maximum Force	lb / N	Force at which the 3rd-order force-deflection relationship is abandoned
Maximum Deflection	in / cm	Deflection at which the 3rd-order force-deflection relationship is abandoned
Unloading Slope	lb/in / N/cm	Linear unloading slope beginning at maximum deflection
Max Defl/Timestep	in / cm	Logical distance defining the amount of deformation allowed during one timestep (used to define front/back)
Edge Constant	—	Force reduction coefficient for edges

The Contacts parameters are used by occupant and pedestrian simulations. The Contacts dialog allows the user to display all human ellipsoid vs. vehicle contact surface pairs and deselect them, if desired.

The purpose of deselecting the interaction of a particular ellipsoid vs. contact surface pair is to reduce calculation time by removing irrelevant ellipsoid vs. contact force calculations. For example, it is clear that a pedestrian will not strike the seat bottom cushion, so by removing this (and similar) ellipsoid/contact pairs, the simulation will skip the calculations required to determine their interaction.

Some human simulations use the human ellipsoid material properties and the vehicle contact surface material properties together to define the force vs. deflection relationship for a particular ellipsoid/surface pair. Other human simulations do not use these individual properties; instead they use a single Combined Material property. In that case, the Combined Material Properties dialog is used to assign these properties. The material properties for an ellipsoid/surface pair are defined by first selecting the pair, and then choosing *Combined Material* in the Contacts dialog. A standard materials dialog is displayed for the selected pair. *(See the Contacts dialog page.)*

Restraints

The Restraints Event Set-up parameters are defined in Table 16-15.

Table 16-15: Restraint Event Set-up Parameters

Parameter	Units (US / SI)	Description
x, y and z Attachment Coordinates	in / cm	Segment-based x, y, z coordinates for the attachment point of the ends of the right and left belt segments
Belt Slack	in / cm	Belt slack. The amount of displacement between the ends of the belt allowed before creating belt tension (a negative value implies pre-load)
Airbag Begin Fill Time	sec	Simulation time at which the airbag is deployed and begins to fill
Airbag Fill Duration	sec	Length of time during which airbag filling occurs

The Restraints parameters allow the user to provide in-use restraint system parameters for occupant events. For Belt Restraint systems, this means assigning the segment-fixed belt attachment point on the human (the other end of the belt is attached to the vehicle, and is therefore a vehicle parameter assigned in the Vehicle Editor) and belt slack.

> **NOTE:** Most occupant simulators will view negative belt slack as pretensioning.

For Airbag Restraint systems, the in-use factors are deployment time and fill duration.

> **NOTE:** To assign in-use parameters for either the belt or airbag restraint systems requires that a restraint exist at the seated position of the occupant.
>
> **NOTE:** The seated position of the occupant is assigned using the Human Editor when the human is created.
>
> **NOTE:** The seat position of each restraint system is assigned using the Vehicle Editor.

Output Track Parameters

The Event output track definition is dependent on object type, Human or Vehicle. Output track results are displayed during Event Mode in Key Results windows and, during Playback Mode, in the Variable Output Table.

Human and Vehicle Output Track Parameters are provided below.

Human Output Tracks

The output tracks for HVE humans are divided into five categories:

- Kinematics
- Joints
- Occupant Contacts
- Belts
- Airbags

The output track parameter definitions for each of these groups are described below. Specific definitions are provided in Tables 16-16 through 16-20.

Kinematics

The Human Segment Kinematics output groups are defined in Table 16-16.

Table 16-16: Human Segment Kinematics Output Track Group Parameters

Parameter	Units (US / SI)	Description
x or X, y or Y, z or Z	in / cm for occupants, ft / m for pedestrians	Coordinates for the center of gravity of each segment, relative to the vehicle-fixed coordinate system (occupants) or earth-fixed coordinate system (pedestrians)
Roll, Pitch, Yaw	deg for occupants, deg for pedestrians	Orientation of each segment, relative to the vehicle-fixed coordinate system (occupants) or earth-fixed coordinate system (pedestrians)
Vtotal, i vel, j vel, k vel	in/sec / km/h	Total, i, j and k velocity of each segment, relative to the vehicle-fixed coordinate system (occupants) or earth-fixed coordinate system (pedestrians)
Roll Vel, Pitch Vel, Yaw Vel	deg/sec	Roll, pitch and yaw velocities of each segment relative to the segment's axis system
Atotal, i acc, j acc, k acc	deg/sec^2	Roll, pitch and yaw acceleration of each segment relative to the segment's axis system

The Human Segment Kinematics output tracks define the current position, velocity and acceleration for each human segment.

Joints

The Human Joints output groups are defined in Table 16-17.

Table 16-17: Human Inter-segment Joint Output Track Group Parameters

Parameter	Units (US / SI)	Description
Roll, Pitch, Yaw	deg	Angular orientation of segment relative to the upstream segment's coordinate axis system
FxElastic, FyElastic, FzElastic	lb / N	Segment-fixed joint force components from elasticity
FxDamping, FyDamping, FzDamping	lb / N	Segment-fixed joint force components from damping
FxTotal, FyTotal, FzTotal	lb / N	Segment-fixed total joint force components
MxElastic, MyElastic, MzElastic	in-lb / N-m	Moment about the upstream joint's i, j, k axis system from joint elasticity
MxDamping, MyDamping, MzDamping	in-lb / N-m	Moment about the upstream joint's i, j, k axis system from joint damping
MxTotal, MyTotal, MzTotal	in-lb / N-m	Total moment about the upstream joint's i, j, k axis system from joint torques

The Human Joint output tracks define the current conditions at each joint.

Contacts

The Human Contacts output groups are defined in Table 16-18.

Table 16-18: Human Contacts Output Track Group Parameters

Parameter	Units (US / SI)	Description
x, y, z	in / cm	Contact surface contact coordinates
Deflection	in / cm	Contact surface deflection at x, y, z
F (total)	lb / N	Total contact force
F (normal)	lb / N	Contact surface normal force
F (friction)	lb / N	Contact surface frictional force

The Human Contacts output tracks define the current interaction between each selected human ellipsoid and vehicle contact surface.

> **NOTE:** These are the same values reported in the Vehicle Contacts output tracks.

Belts

The Human Belt Restraints output groups are defined in Table 16-19.

Table 16-19: Human Belt Restraints Output Track Group Parameters

Parameter	Units (US / SI)	Description
Belt Attachment i-coord, j-coord, k-coord	in / cm	The i, j, k segment-fixed coordinates for the belt attachment point
Belt Strain	%/100	Simulated belt strain
Belt Tension	lb / N	Simulated belt tension
Belt Pressure	lb / N	Simulated belt force against the segment

The Human Belt output tracks define the current belt outputs.

Airbags

The Human Airbag Restraints output groups are defined in Table 16-20.

Table 16-20: Human Airbag Restraints Output Track Group Parameters

Parameter	Units (US / SI)	Description
Pressure	psi / kPa	Airbag internal pressure
Radius	in / cm	Airbag radius
Bag Fn	lb / N	Normal contact force between human and airbag
Bag Fmu	lb / N	Friction contact force between airbag and vehicle
Bag x, Bag y, Bag z	in / cm	Human contact point with airbag

The Human Airbag Restraints output tracks define the current interaction between the human occupant and the airbag.

> **NOTE:** These are similar to the values reported in the Vehicle Airbag Restraints output tracks.

Vehicle Output Tracks

The output tracks for HVE vehicles are divided into the following categories:

- Kinematics
- Kinetics
- Accelerometers
- Damage
- Tires
- Wheels
- Connections
- Drivetrain
- Driver Inputs
- Occupant Contacts
- Belts
- Airbags

The output track parameter definitions for each of these groups are described below. Specific definitions are provided in Tables 16-21 through 16-32.

Sprung Mass Kinematics

The Vehicle Sprung Mass Kinematics output groups are defined in Table 16-21.

Table 16-21: Vehicle Sprung Mass Kinematics Output Track Group Parameters

Parameter	Units (US / SI)	Description
X, Y, Z	ft / m	Coordinates of the vehicle center of gravity relative to the earth-fixed coordinate system
Roll, Pitch, Yaw	deg	Orientation of the vehicle-fixed coordinate system relative to the earth-fixed coordinate system (order of rotations is Yaw, Pitch, Roll)
Radius	ft / m	Path radius of the vehicle CG
Nu (Course Angle)	deg	Angle of the total velocity vector relative to the earth-fixed coordinate system
Vtotal, u vel, v vel, w vel	mph / km/h	Total, u (longitudinal), v (side) and w (normal) velocity of the vehicle (see SAE J670e for definitions)
Beta (Sideslip Angle)	deg	arctan(v vel / u vel), angle from the vehicle heading vector to the velocity vector
Roll Velocity, Pitch Velocity, Yaw Velocity	deg/sec	p (roll), q (pitch) and r (yaw) velocity about the vehicle's x, y, z axis system
Atotal, u dot, v dot, w dot, Long Acc, Lat Acc, Vert Acc	g	Total, u, v and w accelerations, and Longitudinal, Lateral and Vertical accelerations (see SAE J670e for definitions)
Roll Acc, Pitch Acc, Yaw Acc	deg/sec²	Roll, pitch and yaw accelerations about the vehicle's x, y, z axis system
Translation KE, Rotation KE, Total KE	ft-lb / Joule	Kinetic energy associated with translation, rotation and their sum

The Vehicle Kinematics output tracks define the current position, velocity and acceleration, and other related parameters, at the current timestep.

Sprung Mass Kinetics

The Vehicle Sprung Mass Kinetics output groups are defined in Table 16-22.

Table 16-22: Vehicle Sprung Mass Kinetics Output Track Group Parameters

Parameter	Units (US / SI)	Description
FxSusp, FySusp, FzSusp	lb / N	Summation of suspension forces at the vehicle CG relative to the vehicle-fixed axis system
MxSusp, MySusp, MzSusp	ft-lb / N-m	Summation of suspension moments about the vehicle CG relative to the vehicle-fixed axis system
FxColl, FyColl, FzColl	lb / N	Summation of collision forces at the vehicle CG relative to the vehicle-fixed axis system
MxColl, MyColl, MzColl	ft-lb / N-m	Summation of collision moments about the vehicle CG relative to the vehicle-fixed axis system
FxAero, FyAero, FzAero	lb / N	Summation of aerodynamic forces at the vehicle CG relative to the vehicle-fixed axis system
MxAero, MyAero, MzAero	ft-lb / N-m	Summation of aerodynamic moments about the vehicle CG relative to the vehicle-fixed axis system
FxConn, FyConn, FzConn	lb / N	Summation of inter-vehicle connection forces at the vehicle CG relative to the vehicle-fixed axis system
MxConn, MyConn, MzConn	ft-lb / N-m	Summation of connection moments about the vehicle CG relative to the vehicle-fixed axis system

The Vehicle Sprung Mass Kinetics output tracks define the forces and moments at the sprung mass CG due to the various sources listed above.

Accelerometers

The Accelerometers Output Group parameters are defined in Table 16-23.

Table 16-23: Vehicle Accelerometers Output Track Group Parameters

Parameter	Units (US / SI)	Description
x, y, z	in / cm	Vehicle-fixed accelerometer coordinates
V-tot, u-vel, v-vel, w-vel	mph / km/h	Accelerometer velocity at x, y, z, resolved according to the vehicle-fixed coordinate system
X-dot, Y-dot, Z-dot	mph / km/h	Accelerometer velocity at x, y, z, resolved according to the earth-fixed coordinate system
Acc-tot, Long Acc, Lat Acc, Vert Acc	g	Accelerometer acceleration at x, y, z
u-dot, v-dot, w-dot	g	Accelerometer acceleration at x, y, z; time derivative of u-vel, v-vel, w-vel

The Accelerometers output tracks define the position, velocity (earth-fixed and vehicle-fixed) and acceleration (earth-fixed and vehicle-fixed) for each installed accelerometer. Up to five accelerometers may be defined (see Event Set-up, Accelerometers).

Damage

The Damage Output Group parameters are defined in Table 16-24.

Table 16-24: Vehicle Damage Output Track Group Parameters

Parameter	Units (US / SI)	Description
Is Damaged	—	Flag indicating the vertex sustained damage since the previous timestep
Vertex ID	—	Vertex number, used for indexing
x, y, z	in / cm	Vertex vehicle-fixed coordinates
Fx, Fy, Fz	lb / N	Vehicle-fixed force components at x, y, z
F0 (surf), F1 (surf), F2 (surf), F3 (surf)	lb / N, lb/in / N/cm, lb/in² / N/cm², lb/in³ / N/cm³	Force-deflection characteristics of the vertex
Mu (surf)	—	Vertex friction multiplier
Fmax (surf)	lb / N	Vertex maximum force characteristic
Fu (surf)	lb/in / N/cm	Vertex unloading linear stiffness

Each vehicle vertex has associated with it the above data. These data are currently not available to the user through the variable output interface.

> **NOTE:** The x, y, z coordinates are used to visualize the damage done to the vehicle exterior.

Tires

The Vehicle Tires output groups are defined in Table 16-25.

Table 16-25: Vehicle Tires Output Track Group Parameters

Parameter	Units (US / SI)	Description
x, y, z	in / cm	Coordinates of the tire contact patch relative to the vehicle-fixed coordinate system (see SAE J670e)
X, Y, Z	ft / m	Coordinates of the tire contact patch relative to the earth-fixed coordinate system (used for drawing tire marks)
Fx', Fy', Fz'	lb / N	Longitudinal, lateral and normal force at the tire contact patch relative to the tire's coordinate axis system
Mx', My', Mz'	in-lb / N-m	Rolling resistance moment, overturning moment and aligning torque relative to the tire's coordinate axis system
Trad	in / cm	Tire loaded radius
LongSlip	%/100	Longitudinal tire slip
LatSlip	deg	Tire slip angle
Inclination Angle	deg	Angle from the tire z-axis to the surface normal
SkidMark	—	Opacity of skidmark (0 = clear; 1 = opaque)
Mu (surf)	—	Friction multiplier of the surface
Nx, Ny, Nz	—	Earth-fixed unit normal of the terrain at X,Y
F0 (surf), F1 (surf), F2 (surf), F3 (surf)	lb / N, lb/in / N/cm, lb/in² / N/cm², lb/in³ / N/cm³	Force-deflection characteristics of the terrain at X,Y
N, Kphi, Kc	none, lb/in^N+1 / N/cm^N+1, lb/in^N+2 / N/cm^N+2	Bekker soil constants for the terrain at X,Y
Moisture	%/100	Moisture content of the terrain at X,Y
Clay	%/100	Clay content of the terrain at X,Y *(updated: the Clay track is retained; the terrain Macrotexture parameter was added as an additional track, not as a replacement — see Chapter 13)*
Sinkage	ft / m	Soil deformation at X,Y
Fx' plow, Fy' plow	lb / N	Wheel-fixed tire plow force at X,Y
Fx Sidewall, Fy Sidewall, Fz Sidewall	lb / N	Longitudinal, lateral and normal tire sidewall force (sidewall-impact model)
Water Depth	in / cm	Depth of water on the terrain at X,Y
Macrotexture	in / cm	Terrain macrotexture depth at X,Y
Water Mu Mult	—	Friction multiplier applied for the water depth at X,Y
Water RR	—	Rolling-resistance contribution from the water depth at X,Y

Tire-related vehicle output tracks define the conditions at the tire contact patch for each tire.

Wheels

The Vehicle Wheels output groups are defined in Table 16-26.

Table 16-26: Vehicle Wheel Output Track Group Parameters

Parameter	Units (US / SI)	Description
x (wheel), y (wheel), z (wheel)	in / cm	Coordinates of the wheel center with respect to the vehicle-fixed axis system
dz (axle)	in / cm	Vertical axle displacement (solid axles)
phi (axle)	deg	Solid axle vehicle-fixed roll angle
gamma (wheel)	deg	Wheel camber relative to the vehicle-fixed axis system (+ clockwise about an axis parallel to, and in the direction of, the x-axis)
spin (wheel)	deg	Wheel rotation angle about the spin axis
delta (wheel)	deg	Vehicle-fixed wheel steer angle
Vtot (wheel)	mph / km/h	Total earth-fixed velocity at the wheel center
Xvel (wheel)	mph / km/h	Earth-fixed X velocity at the wheel center
Yvel (wheel)	mph / km/h	Earth-fixed Y velocity at the wheel center
Zvel (wheel)	mph / km/h	Earth-fixed Z velocity at the wheel center
vtot (wheel)	mph / km/h	Vehicle-fixed total velocity at the wheel center
u-vel (wheel)	mph / km/h	Vehicle-fixed forward velocity at the wheel center
v-vel (wheel)	mph / km/h	Vehicle-fixed lateral velocity at the wheel center
w-vel (wheel)	mph / km/h	Vehicle-fixed vertical velocity at the wheel center
zdot (axle)	mph / km/h	Solid axle vehicle-fixed vertical velocity
phid (axle)	deg/sec	Solid axle roll velocity
gamma-dot	deg/sec	Wheel camber change rate
Spin Vel (wheel)	deg/sec	Wheel rotational speed about the spin axis
Delta-dot	deg/sec	Wheel steer velocity
A tot, X acc, Y acc, Z acc	g	Wheel center acceleration relative to the earth-fixed coordinate system
Along, Alat, Avert	g	Wheel center acceleration relative to the vehicle-fixed coordinate system
zddot (axle)	g	Solid axle vehicle-fixed vertical acceleration
phidd (axle)	deg/sec²	Solid axle vehicle-fixed angular acceleration
gamma-ddot	deg/sec²	Wheel camber acceleration
Spin Acc	deg/sec²	Wheel spin angular acceleration
Delta-ddot	deg/sec²	Wheel steer angular acceleration
zddot	g	Wheel jounce/rebound acceleration
Fx, Fy, Fz	lb / N	Force at the wheel center with respect to the vehicle-fixed axis system
Mx, Mz	ft-lb / N-m	Wheel moments about the camber and steer axes (see axle Drive/Brake Torque)
dz (susp)	in / cm	Suspension vertical displacement (+ down)

Parameter	Units (US / SI)	Description
zvel (susp)	mph / km/h	Suspension vertical velocity (+ down)
Fz Susp, Fz Damp, Fz A-P	lb / N	Suspension force components from suspension spring, damping (i.e., shock absorber) and anti-pitch
RollStf	ft-lb / N-m	Auxiliary roll stiffness from anti-sway bar
WheelBrkPress	psi / kPa	Brake pressure at the wheel cylinder
Drive Tq, Brake Tq	ft-lb / N-m	Torque applied about the wheel spin axis
FxDrv (Att), FxBrk (Att)	lb / N	Attempted drive or brake force (from throttle and/or braking inputs)
Brk Stroke	in / cm	Stroke of the wheel cylinder or slack adjuster before brake torque is applied
Brk Press	psi / kPa	Brake fluid pressure (air or hydraulic) at the wheel cylinder or brake chamber
F Piston	lb / N	Force from the piston or slack adjuster on the braking mechanism
T Interface	deg F / deg C	Temperature at the interface between the brake lining and drum
T Drum	deg F / deg C	Brake drum temperature
T Lining	deg F / deg C	Brake lining temperature

The above table defines output tracks for each wheel location. Some are axle-related (choosing the right or left side for these parameters will display the same results). Some parameters are effective at the wheel center, some are at the suspension and some are at the brake.

Inter-vehicle Connections

The Vehicle Inter-vehicle Connections output groups are defined in Table 16-27.

Table 16-27: Vehicle Connections Output Track Group Parameters

Parameter	Units (US / SI)	Description
Roll, Pitch, Yaw	deg	Roll, pitch and yaw orientation relative to the towing vehicle's coordinate system
Roll Vel, Pitch Vel, Yaw Vel	deg/sec	Roll, pitch and yaw articulation velocities relative to the towing vehicle's coordinate system
Rollddot, Pitchddot, Yawddot	deg/sec²	Roll, pitch and yaw articulation accelerations relative to the towing vehicle's coordinate system
Fx, Fy, Fz	lb / N	Connection forces in the vehicle-fixed coordinate system
Mx, My, Mz	ft-lb / N-m	Connection moments in the vehicle-fixed coordinate system

The Inter-vehicle Connections output tracks define the conditions at the connection point between the towing and towed vehicle.

Drivetrain

The Vehicle Drivetrain output groups are defined in Table 16-28.

Table 16-28: Vehicle Drivetrain Output Track Group Parameters

Parameter	Units (US / SI)	Description
Engine speed	RPM	Engine rotational velocity (i.e., RPM)
Power	HP / kW	Engine power
Torque	ft-lb / N-m	Engine torque
Transmission Ratio	—	The current numeric ratio for the transmission
Differential Ratio	—	The current numeric ratio for the differential

The Vehicle Drivetrain output tracks define the current drivetrain conditions.

Driver Controls

The Vehicle Driver Controls output groups are defined in Table 16-29.

Table 16-29: Vehicle Driver Controls Output Track Group Parameters

Parameter	Units (US / SI)	Description
% WOT	%/100	Throttle position, percent of WOT
Brake Force	lb / N	Brake pedal force
Brake Press	psi / kPa	Brake system master cylinder pressure
Trans Gear	—	Transmission gear selection
Diff Gear	—	Differential gear selection
Steer Angle	deg	Steering wheel angle
Steer Ang Vel	deg/sec	Steering wheel angular velocity
Steer Ang Acc	deg/sec²	Steering wheel angular acceleration
Steer Tq	in-lb / N-m	Steering wheel torque
Path X, Path Y	ft / m	Earth-fixed X,Y coordinates of the path used by the HVE Path Follower
Path X Att, Path Y Att	ft / m	Earth-fixed X,Y coordinates of the path at the preview distance
Path Err	ft / m	Distance from the attempted path to the desired path
DX Preview, DY Preview	ft / m	Earth-fixed coordinates of the preview point

The Vehicle Driver Controls output tracks define the current driver inputs. Results from the steer degree of freedom and HVE Path Follower are also included in the Vehicle Driver output group.

Contacts

The Vehicle Contacts output groups are defined in Table 16-30.

Table 16-30: Vehicle Contacts Output Track Group Parameters

Parameter	Units (US / SI)	Description
x, y, z	in / cm	Contact surface contact coordinates
Deflection	in / cm	Contact surface deflection at x, y, z
Ftotal	lb / N	Total contact force
Fnormal	lb / N	Contact surface normal force
Ffriction	lb / N	Contact surface frictional force

The Vehicle Contacts output tracks define the current interaction between each selected human ellipsoid and vehicle contact surface.

> **NOTE:** These are the same values reported in the Human Contacts output tracks.

Belt Restraints

The Vehicle Belt Restraints output groups are defined in Table 16-31.

Table 16-31: Vehicle Belt Restraints Output Track Group Parameters

Parameter	Units (US / SI)	Description
Belt x, Belt y, Belt z	in / cm	Vehicle-fixed belt anchor point coordinates
Belt Azimuth	deg	The angle of the belt in the vehicle-fixed x-y plane
Belt Zenith	deg	The elevation angle of the belt in the vehicle-fixed coordinate system
Belt Strain	%/100	Simulated belt strain
Belt Stretch	in / cm	Simulated belt stretch
Belt Tension	lb / N	Simulated belt tension

The Vehicle Belt output tracks define the current belt forces and stretch.

Airbag Restraints

The Vehicle Airbag Restraints output groups are defined in Table 16-32.

Table 16-32: Vehicle Airbag Restraints Output Track Group Parameters

Parameter	Units (US / SI)	Description
Pressure	psi / kPa	Airbag internal pressure
Radius	in / cm	Airbag radius
Bag Defl	in / cm	Airbag deflection caused by the back-side contact surface
Bag Fn	lb / N	Airbag normal force from the back-side contact surface
Bag Fmu	lb / N	Frictional force between the back-side contact surface and the airbag
Deploy x, Deploy y, Deploy z	in / cm	Airbag vehicle-fixed deployment coordinates

The Vehicle Airbag Restraints output tracks define the current interaction between the human occupant and the airbag.

> **NOTE:** These are similar to the values reported in the Human Airbag Restraints output tracks (see Human Output Tracks, Airbags).

Static Reports

In addition to the time-dependent output track parameters described above, additional output parameters related to the HVE human or vehicle may be monitored. These results are stored in the form of static reports and include:

- Accident History
- Damage Data
- Damage Profile
- Driver Data
- Environment Data
- Event Data
- Human Data
- Injury Data
- Messages
- Momentum Diagram
- Program Data
- Site Drawing
- Vehicle Data

Whereas the event produces human and vehicle output tracks at each output interval, the above static reports are produced only once at the end of the run.

> **NOTE:** There is an important and fundamental difference between the output track definitions and the information displayed in the Static Reports: each available output track parameter is strictly defined by the HVE Event Model. However, the static reports may contain any information the developer of the reconstruction or simulation model chooses to report.

See also (dialog references):

Event Information · Event Setup · Position/Velocity · Driver Controls · Contacts · Inter-vehicle Connections · Simulation Controls

Section Seven: Playback Editor

A motor vehicle crash sequence often includes several events involving many different vehicles, and any number of occupants and pedestrians. The purpose of the Playback Editor is to combine simulations from several events into a single, coherent sequence involving multiple humans and vehicles. The Playback Editor's video interface allows the user to produce professional quality video output. In addition, the Playback Editor provides the ability to view and print the complete output reports for each event. This section of the HVE User's Manual describes how to use the Playback Editor to perform these tasks.

(updated: the video interface now records directly to AVI movie files using a user-selected compressor, rather than to video tape; see the Video Setup notes in Chapter 17.)

General Procedure

To use the Playback Editor, perform the following steps:

1. Select Playback mode.
2. Create one or more Report Windows to view the results for each event.
3. If desired, print the results on the system printer.
4. Create a Playback Window to combine several trajectory simulations into a single, coherent sequence involving multiple humans and vehicles.
5. Use HVE's video interface to produce a simulation movie file.

Overview of Section Seven

This section provides a detailed explanation of the process of viewing, combining and editing the results of events:

- Chapter 17 describes the process of creating Report and Playback windows.
- Chapter 18 provides a detailed overview of the process of combining and editing multiple events.

Related Reference Pages

- Playback Controls (Toolbar)
- Report Window Dialog Box
- Key Results Dialog Box
- Output report reference: Section 11 — Reports and Output

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Section Seven; revised to match the current version of HVE, 2026-07-05.

Chapter 17 — Creating Report and Playback Windows

The purpose of the Playback Editor is to allow the user to view and manage all program outputs by performing the following operations:

- View all program outputs (numeric, graphic and simulation).
- Print all program outputs.
- Edit the accident sequence.
- Combine multiple simulation events into a single Playback Window.
- Record the Playback Window to a movie file. *(updated: output is recorded to an AVI movie file using the built-in video interface; direct routing to a video tape recorder is no longer the normal workflow.)*

This chapter describes how to use the Playback Editor to perform these functions, beginning with a description of the Playback Editor's components.

Playback Editor Components

To use the Playback Editor, choose Playback Mode using the mode selector

(Mode menu, Playback, or Ctrl+5)

. This puts HVE into Playback Mode and displays the Playback Editor's components:

- **Playback Editor Dialog** — The Playback Editor dialog is used for adding report and playback windows to the current case and to select the current window.
- **Report Window(s)** — A Report Window displays the selected output and is used for viewing various program outputs (these are described later in this chapter).
- **Playback Window** — The Playback Window displays one or more trajectory simulations and is used for visualizing sequences involving multiple events. The Playback Window is also used for recording output to the current video destination (movie file). It also includes video input (source) and output (destination) selection options.
- **Playback Controller** — The Playback Controller displays a VCR-like panel of buttons allowing the user to start, stop, reset and play events forwards and backwards. *(updated: the Playback Controller is implemented as the playback controls on the main toolbar; see Playback Controls (Toolbar).)*

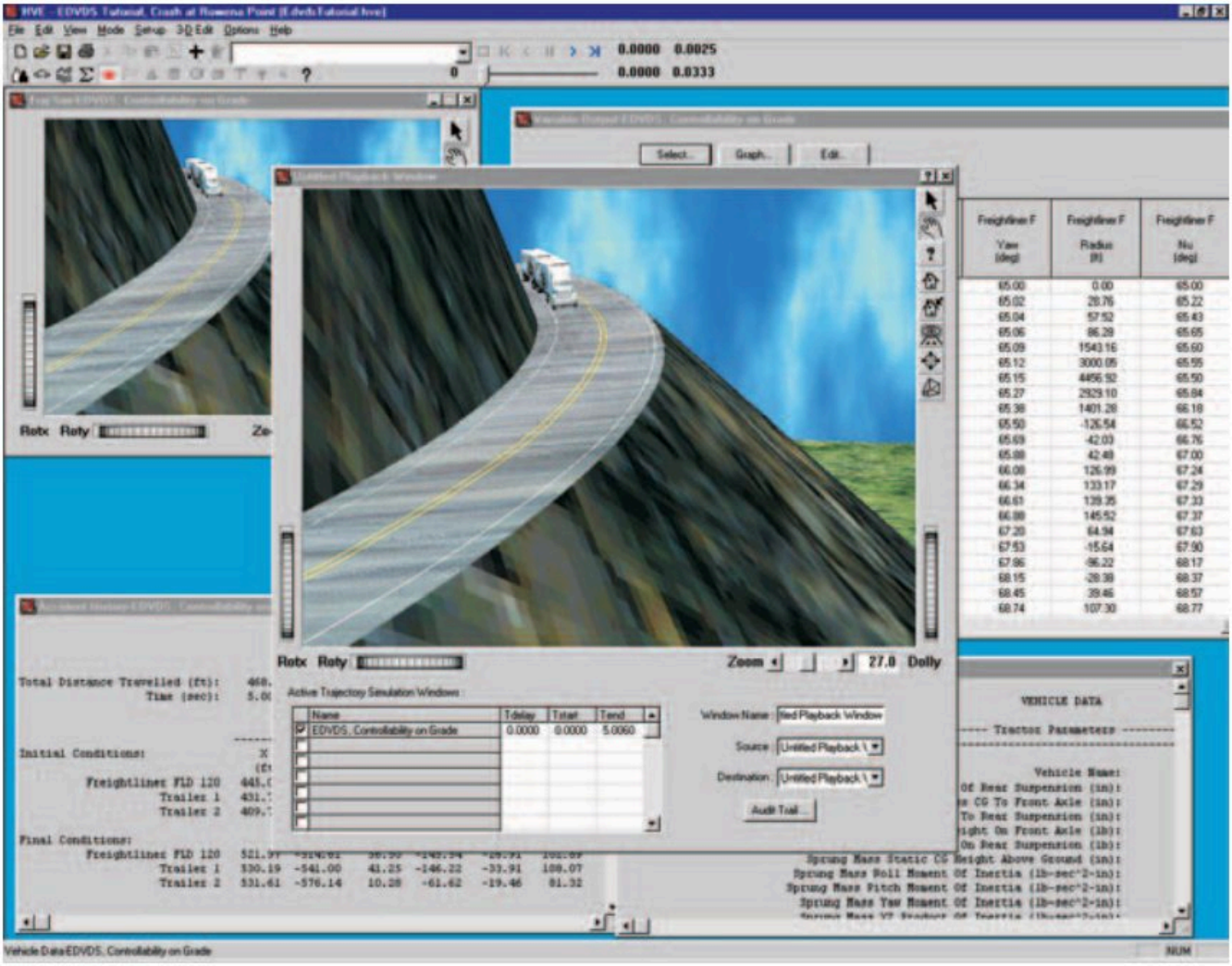


Figure 17-1 — Playback Editor, Report Windows and Playback Controller.

Playback Editor Dialog

The Playback Editor dialog is the heart of the HVE Playback Editor. The dialog includes the following functionality:

- **Add Report Window** — Allows the user to add a Report Window to the current case.
- **Add Playback Window** — Allows the user to add a Playback Window to the current case.
- **Delete Window** — Removes the current Report or Playback Window from the case.

(updated: in the current version the Playback Editor dialog is a docked window list; each Report and Playback window in the case appears in the list, and the current window is chosen by selecting it in the list or in the window-name combo box on the toolbar. Report windows are added with the Add New Object toolbar button, and windows are removed with the Delete Object toolbar button.)

> **NOTE:** The Delete pushbutton is not selectable if the current window is a Traj Sim Window that is used in a Playback Window.

Report Windows

Report Windows are used for displaying the results of each event. The results are displayed in several forms:

- **Numeric Output Windows** — Allow the user to view numeric reports produced by any event (see Figure 17-2 for an example). The following output reports are available:
- **Accident History** — A table of important positions and velocities during the event (see Accident History).
- **Damage Data** — Vehicle damage sustained during the event (see Damage Data).
- **Driver Data** — Vehicle driver inputs (Throttle, Steering, Brake and Gear Selection) during the event.
- **Environment Data** — Environment parameters used by the event.
- **Event Data** — Event-related parameters.
- **Injury Data** — Human injuries sustained during the event.
- **Messages** — Program diagnostics produced by the event.
- **Program Data** — General program parameters used by the event.
- **Vehicle Data** — Vehicle properties used by the event.

Numeric reports are displayed in a scrollable Common Reports window (see Common Reports Dialog).

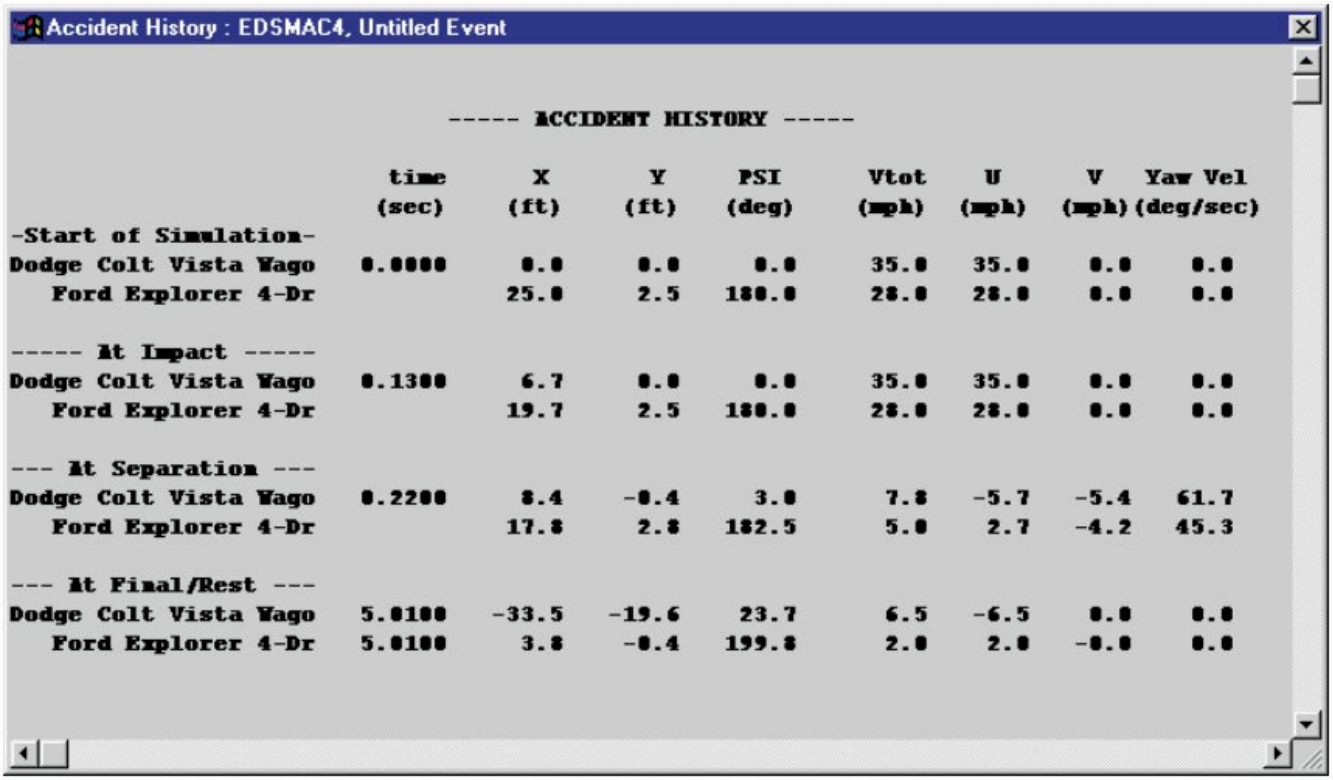


Figure 17-2 — Report Window with Numeric Results.

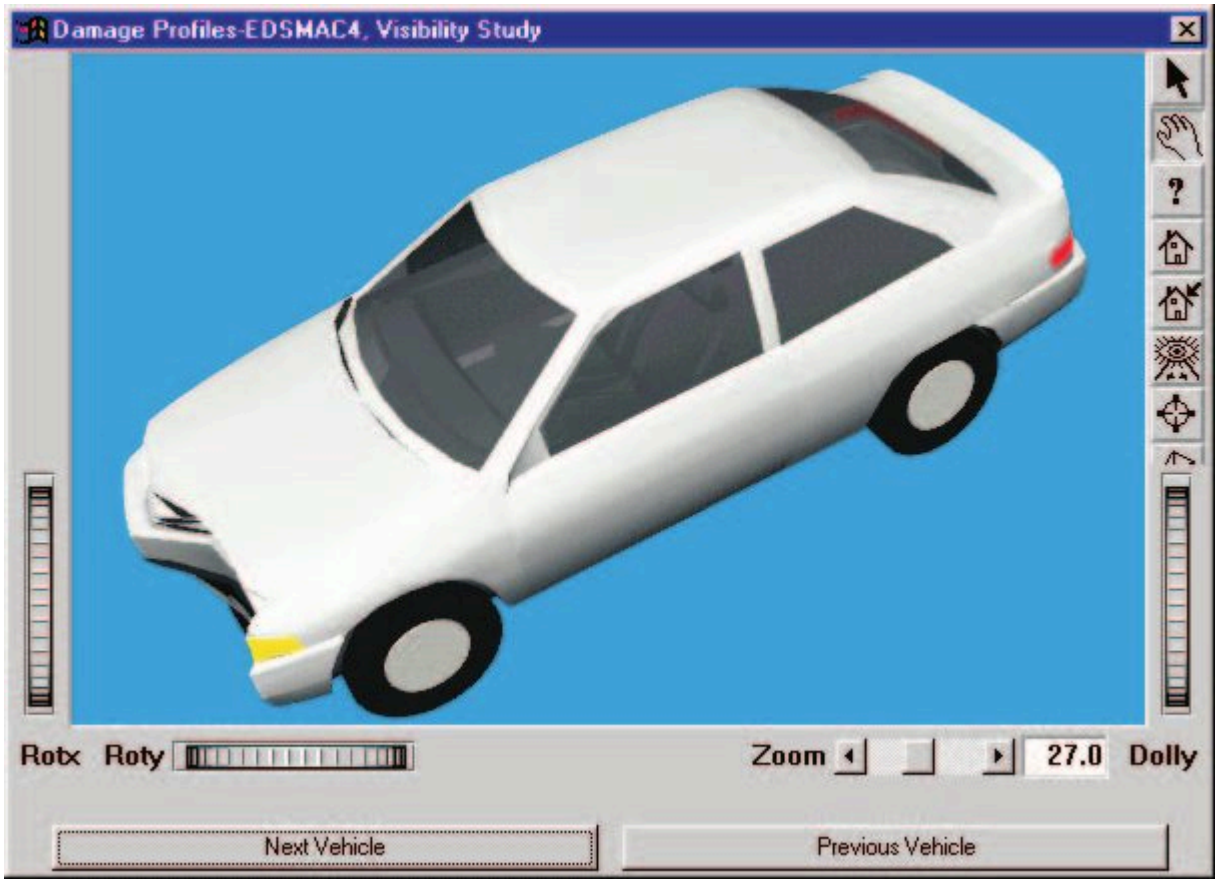
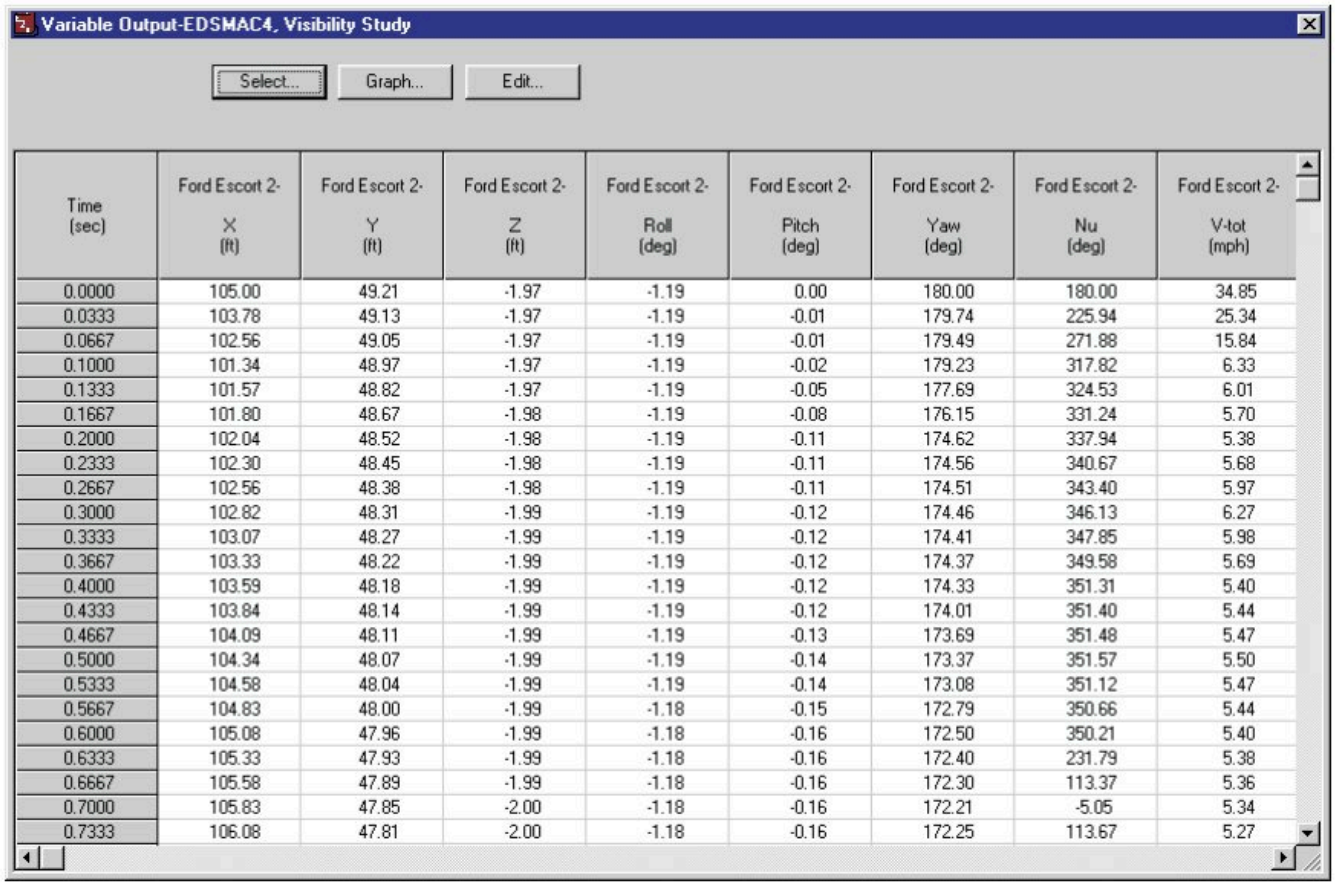


Figure 17-3 — Report Window with a Graphic Output; an example of a Damage Profile report from a simulation.

- **Graphic Output Windows** — Allow the user to view graphic reports produced by an event (see Figure 17-3 for an example). The following Graphic Output reports are available:
- **Damage Profile** — A visual showing the damage sustained by each vehicle during impact. The visual is static for reconstruction-type programs; it is dynamic (i.e., time-dependent) for simulations.
- **Momentum Diagram** — A vector diagram produced by collision models showing the pre- and post-impact momentum.
- **Site Drawing** — A static visual showing each vehicle at its user-entered positions (Impact, Separation, Rest, etc.) (see Common Scene Reports).
- **Variable Output Table** — A table of simulation results as a function of time, reported at the current Playback time increment (see Figure 17-4 for an example, and Variable Output Report). Refer to Chapter 16, Event Model Definition, for information about the available output parameters.
- **Trajectory Simulation Viewers** — Allow the user to visualize the selected event (see Figure 17-5 for an example, and Trajectory Simulation Report). Trajectory simulation viewers use the same viewer controls (Pan, Zoom and Pick Mode) as other HVE viewers.



Time (sec)	Ford Escort 2- X (ft)	Ford Escort 2- Y (ft)	Ford Escort 2- Z (ft)	Ford Escort 2- Roll (deg)	Ford Escort 2- Pitch (deg)	Ford Escort 2- Yaw (deg)	Ford Escort 2- Nu (deg)	Ford Escort 2- V-tot (mph)
0.0000	105.00	49.21	-1.97	-1.19	0.00	180.00	180.00	34.85
0.0333	103.78	49.13	-1.97	-1.19	-0.01	179.74	225.94	25.34
0.0667	102.56	49.05	-1.97	-1.19	-0.01	179.49	271.88	15.84
0.1000	101.34	48.97	-1.97	-1.19	-0.02	179.23	317.82	6.33
0.1333	101.57	48.82	-1.97	-1.19	-0.05	177.69	324.53	6.01
0.1667	101.80	48.67	-1.98	-1.19	-0.08	176.15	331.24	5.70
0.2000	102.04	48.52	-1.98	-1.19	-0.11	174.62	337.94	5.38
0.2333	102.30	48.45	-1.98	-1.19	-0.11	174.56	340.67	5.68
0.2667	102.56	48.38	-1.98	-1.19	-0.11	174.51	343.40	5.97
0.3000	102.82	48.31	-1.99	-1.19	-0.12	174.46	346.13	6.27
0.3333	103.07	48.27	-1.99	-1.19	-0.12	174.41	347.85	5.98
0.3667	103.33	48.22	-1.99	-1.19	-0.12	174.37	349.58	5.69
0.4000	103.59	48.18	-1.99	-1.19	-0.12	174.33	351.31	5.40
0.4333	103.84	48.14	-1.99	-1.19	-0.12	174.01	351.40	5.44
0.4667	104.09	48.11	-1.99	-1.19	-0.13	173.69	351.48	5.47
0.5000	104.34	48.07	-1.99	-1.19	-0.14	173.37	351.57	5.50
0.5333	104.58	48.04	-1.99	-1.19	-0.14	173.08	351.12	5.47
0.5667	104.83	48.00	-1.99	-1.18	-0.15	172.79	350.66	5.44
0.6000	105.08	47.96	-1.99	-1.18	-0.16	172.50	350.21	5.40
0.6333	105.33	47.93	-1.99	-1.18	-0.16	172.40	231.79	5.38
0.6667	105.58	47.89	-1.99	-1.18	-0.16	172.30	113.37	5.36
0.7000	105.83	47.85	-2.00	-1.18	-0.16	172.21	-5.05	5.34
0.7333	106.08	47.81	-2.00	-1.18	-0.16	172.25	113.67	5.27

Figure 17-4 — Report Window with a Variable Output Table.

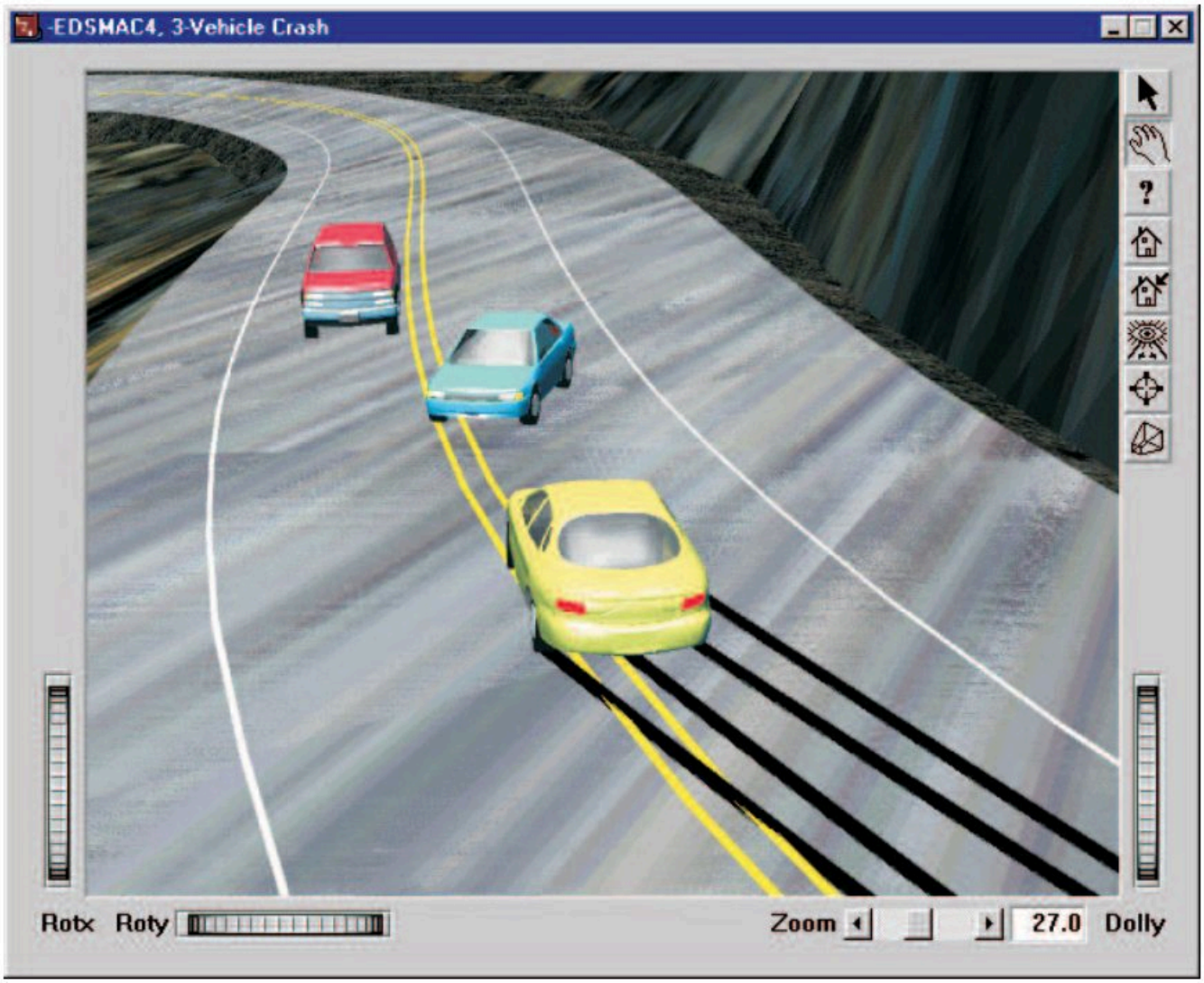


Figure 17-5 — Report Window with the Trajectory Simulation results from a single event.

Playback Window

The Playback Window allows the user to visualize multiple events in a single viewer and record the results to a movie file. The Playback Window viewer uses the same viewer controls (Pan, Zoom and Pick Mode) as other HVE viewers. The Playback Window also includes the following components:

Audit Trail

Each Playback Window may contain the results from several events. The merging of these events requires a considerable amount of logic. For example, the pre-impact phase of an accident sequence may use one simulation model, the impact phase may use a second, and one of the vehicles might use a third model for the post-impact phase.

The HVE Playback Window has an internal table to determine which objects are driven by which events during which time intervals. This table is called the Audit Trail (see Figure 17-6); it may be displayed and printed. The Audit Trail is a scrollable text box containing the following information about each event in the Playback Window:

- Report Window Name

- Event Name
- Event Objects used in the Playback Window

> **NOTE:** Even though an event may include two objects, the motion of one of its objects may be controlled by a different event. For example, after impact, a vehicle might roll over. You can use a different (3-D) model to control this portion of the object's motion in the Playback Window. See Chapter 18, Editing the Event Sequence.

- The starting and ending times during which the object's motion is controlled by that event.

The Audit Trail is a convenient way to document the simulation models that control the motion of each object in the Playback Window.

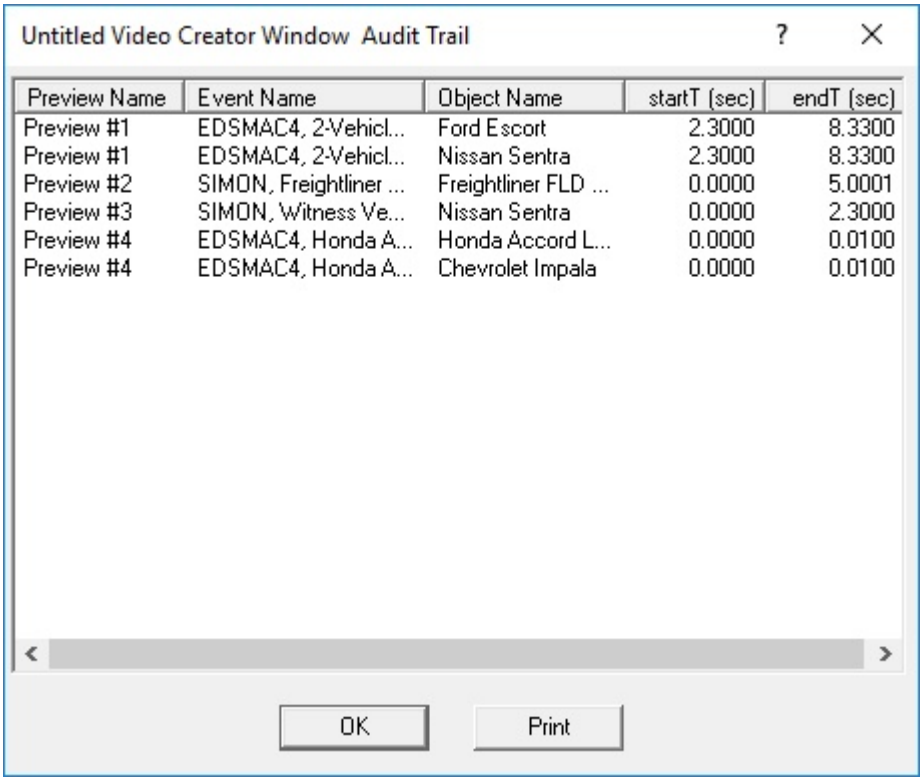


Figure 17-6 — The Audit Trail displays a table of each event included in the current Playback Window, including the objects and the starting and ending times for each object.

Playback Controller

The Playback Controller (see Figure 17-7) is used for controlling the motion in Trajectory Simulation, Damage Profile and Playback output windows. Like the Event Controller described earlier (see Chapter 15, Creating and Editing Events), the Playback Controller works much like a VCR in both form and function. For example, it has the same buttons:

- **Stop/Reset** — Stop the simulations displayed in Trajectory Simulation and/or Playback Windows and initialize them to their starting conditions.
- **Rewind** — Return to the start of the simulation.
- **Reverse** — Run the simulation backwards.
- **Pause** — Pause the simulation(s).
- **Play** — Run the simulation forwards or execute a reconstruction model.

- **Advance To End** — Advance to the end of the simulation.

> **NOTE:** The only difference between Pause and Stop has to do with video recording devices: Pause leaves the recording engaged, while Stop disengages the recording mechanism.

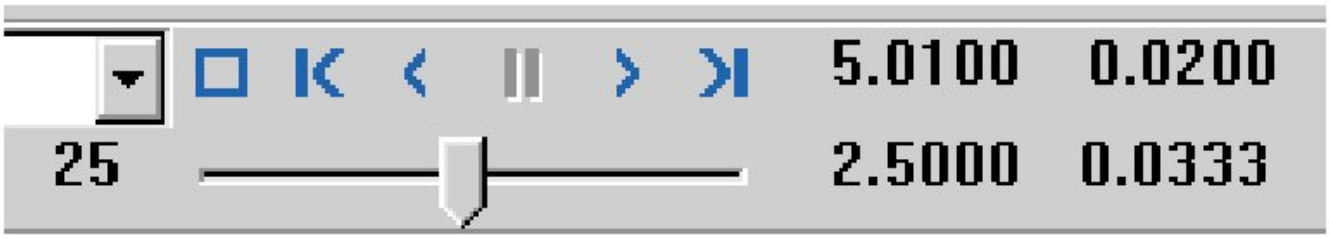


Figure 17-7 — Playback Controller, used for displaying trajectory simulations and playback windows.

In addition to the above trajectory simulation control features, the Playback Controller also includes the following components:

- **Time Display** — Displays the current simulation time.
- **Frame Control Slider** — Allows the user to visually choose a frame within the sequence. To move to the first frame, move the slider all the way to the left; to choose a frame near the middle of the sequence, move the slider to the middle, and so forth.
- **Frame Counter** — Displays the number of the frame currently being displayed. *(updated: the toolbar also displays the Output Time Interval used for playback frames; see Playback Controls for the full list of controls.)*

Adding Report Windows

New Report Windows are added to the current case using the Playback Editor. To add a new Report window, perform the following steps:

1. Click on **Add New Object**. The Report Window dialog (see Figure 17-8, and Report Window Dialog Box) will be displayed, containing the following components:
2. **Active Events List** — A list box containing all the events in the current case.
3. **Event Status** — A non-editable field describing the status of the selected event, Executed or Not Executed.

> **NOTE:** If the event status is Not Executed, no output reports will be available.

- **Name** — A user-editable text box allowing the user to assign a name to the Report Window.
- **Report Type** — A list of all the available output types for the selected event. More than one may be selected; the *Select All* and *Clear All* buttons select or clear every entry.
- Select an event from the Active Events List. A default Report Window name will be assigned according to the selected event.
- If desired, edit the default Report Window Name.
- In the Report Type list, select the desired output report(s). *Select All* selects every report; each selected report opens in its own Report Window.
- Press OK to display the selected Report Window.

The selected output will be displayed in a Report Window.

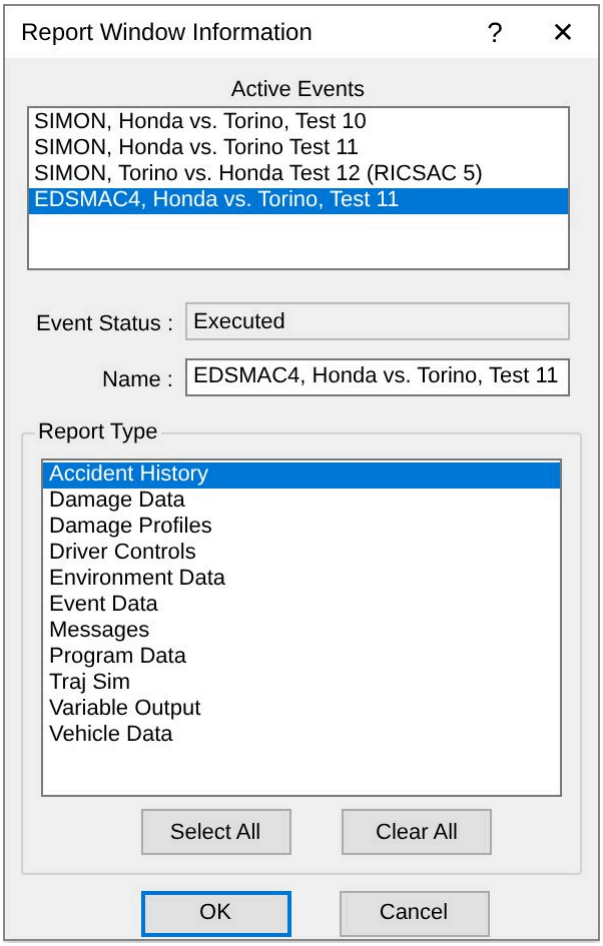


Figure 17-8 — Report Window Information dialog.

Adding Playback Windows

A Playback Window is added to the current case using the Playback Editor. To add a Playback window, perform the following steps:

1. Add a Playback Window. *(updated: the legacy "Options, Add Playback Window" menu command no longer exists; the Playback Window and its Playback Information dialog are created from Playback mode using the Playback Editor.)* The Playback Information dialog (see Figure 17-9) will be displayed, containing the following components:
2. **Active Trajectory Simulations List** — A table containing all the Trajectory Simulation Report Windows in the current case. Each row includes a check box used to select the event for playback, the event name, and the Tdelay, Tstart and Tend timing fields (see Chapter 18).

> **NOTE:** Only Trajectory Simulations are displayed in the table, because only trajectory simulations may be added to the Playback Window, which, by definition, is a visual simulation of all the combined events.

- **Window Name** — A user-editable text box allowing the user to assign a name to the Playback Window.
- **Source** — An option list which allows the user to choose the source of the information displayed in the Playback Window (the live playback, or a previously recorded movie file).

- **Destination** — An option list which allows the user to route the Playback Window to a user-selectable destination. *(updated: the destination is now the playback window itself or an AVI movie file; tape-deck destinations such as "S-Video/Composite Video" are obsolete.)*
- **Video Setup pushbutton** — Displays the Video Setup dialog, used to select the movie format, compressor, recording size and recording speed.
- **Recording Information** — A read-only summary of the current video settings (Format, e.g. AVI; Compressor; Recording Size, e.g. HDTV 1080p (1920x1080); Recording Speed, e.g. frames/sec). *(updated: new in the current version.)*
- **Audit Trail pushbutton** — Displays the Audit Trail for the current Playback Window.
- **Key Results pushbutton** — Displays Key Results windows showing user-selected variables during playback. *(updated: new in the current version.)*
- **Traffic Signals pushbutton** — Displays the traffic signal setup for signals placed in the environment with the 3-D Editor's Signal Tool. *(updated: new in the current version.)*
- **Reorder Events pushbutton** — Allows the user to change the order of the events in the Active Trajectory Simulations table (important for object precedence; see Chapter 18). *(updated: new in the current version.)*

> **NOTE:** For more information about Source and Destination, see Creating A Video, later in this chapter. Also refer to Section Nine: Video Output, as well as the Video Setup option.

1. Check one or more Traj Sims in the Active Trajectory Simulations table.
2. If desired, edit the default Playback Window Name.
3. If desired, click on Audit Trail to display the Audit Trail for the current Playback Window.
4. Press Apply.

The selected Trajectory Simulations will be displayed in the Playback Window.

> **NOTE:** There is only one Playback Window.

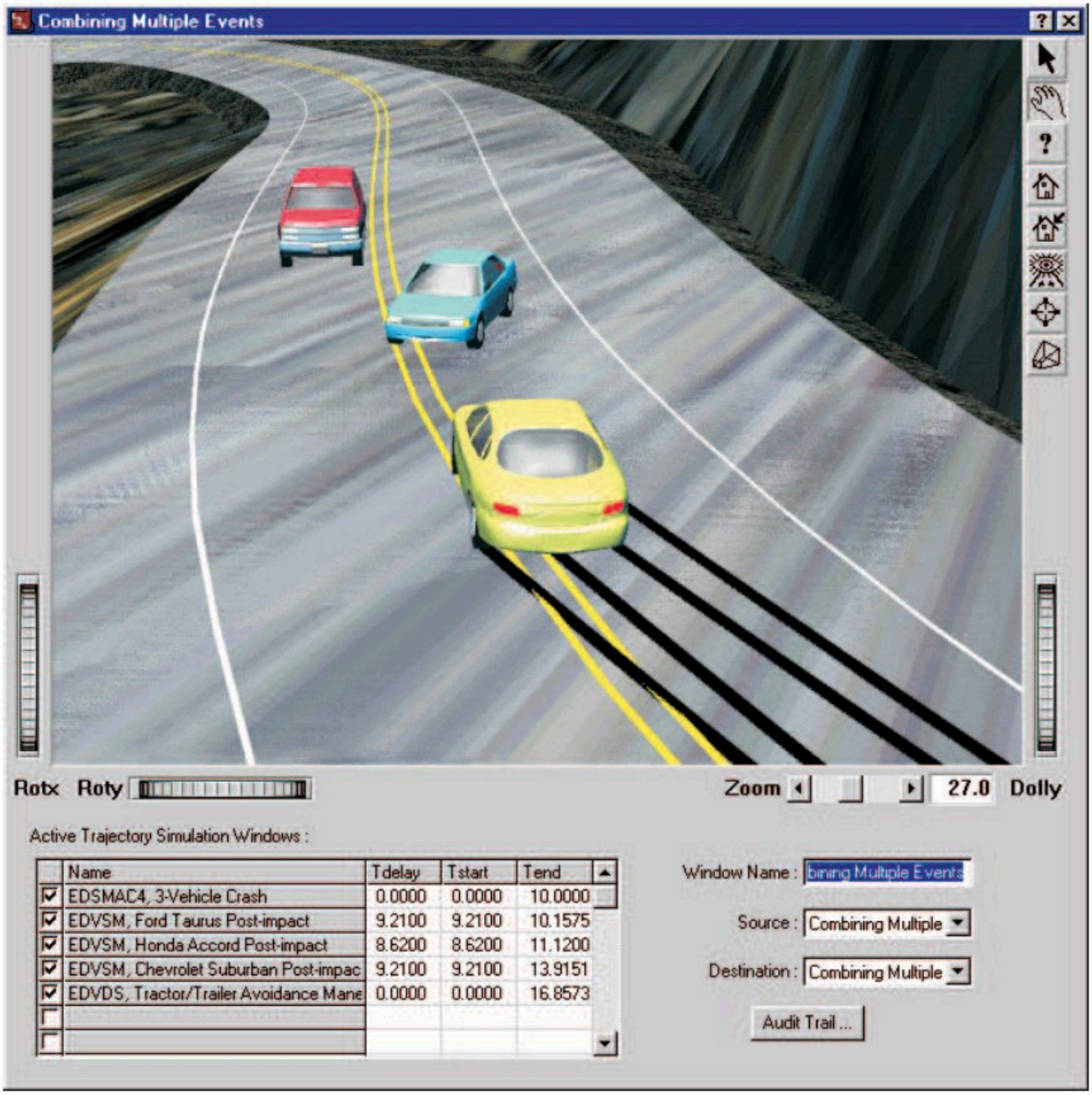


Figure 17-9 — Playback Information dialog.

Displaying Simulations

Simulations in both the Report Windows and Playback Window are displayed and controlled using the Playback Controller.

To display one or more simulations using the Playback Controller, perform the following steps (the following steps assume one or more Trajectory Simulation Report Windows and a Playback Window have been added using the steps described in the previous sections):

1. If necessary, drag the windows to convenient locations on the screen.

2. Note the initial Source and Destination are assigned the Playback Window Name, e.g., Untitled Playback Window.

> **NOTE:** If no Playback Window exists, these options would be unavailable.

1. Note the Time field displays the current simulation time, 0.0000 seconds.
2. Click **Play**. The simulations in each Report Window and the Playback Window will begin. The Frame Slider thumb control will advance and the Frame Counter will display the current frame number.

> **NOTE:** You can iconify (minimize) any window if it is not necessary to watch it. Because iconified windows do not have to be rendered, the simulations will play more quickly.

1. Press **Pause**. The simulations will stop.
2. Press **Reverse**. The simulations will play in reverse. Press **Stop**.
3. Press **Rewind**. The simulations will return to the start.
4. Press **End**. The simulations will advance to the end of the longest event.
5. Click on the Frame Slider thumb control and drag it towards the middle of the range. On releasing the thumb control, the simulations will display the selected frame and the Time field will display the current simulation time.

Creating A Video

One of the most powerful features of HVE is its ability to create professional quality video output using its built-in video interface. Once a Playback Window has been created, producing a video is quick and easy.

To create a video of the current Playback Window, use the Playback Controller to perform the following steps:

1. If necessary, return the simulation to the start using any of these methods:
2. Slide the Frame Slider thumb control all the way to the left, or
3. Press Rewind.
4. Click on the Source option button and choose the current playback window name.
5. Click on the Destination option button and choose the video output destination. *(updated: choose the AVI movie file destination; use the Video Setup dialog to set the movie filename, compressor, resolution and frame rate.)*
6. Press Play.

The simulation in the Playback Window will be recorded to the movie file as it plays. For more information about using HVE's built-in video interface, refer to Section Nine: Video Output.

Printing

Any of the Playback windows may be printed on the system printer. To print the desired window, perform the following steps:

1. Select the desired window.
2. Choose Print from the File menu.

The selected window will be printed on the system printer.

> **NOTE:** The system printer is selected using your computer's Print Manager, selected using the Windows control panel. Contact your system administrator if no printer has been installed on your system.

For more information about using the Playback Controller, refer to the next chapter, Editing the Event Sequence, and to Section Nine: Video Output.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 17; revised to match the current version of HVE, 2026-07-05.

Chapter 18 — Editing the Event Sequence

Overview

Using the Playback Editor, simulations from several events may be edited into a single, coherent sequence involving multiple humans and vehicles. For example, a multiple vehicle collision with several occupants and pedestrians may be created using the Playback Editor. The purpose of this chapter is to explain how to edit multi-event sequences.

Editing multi-event sequences involves two basic steps:

- Combining the desired events
- Editing the sequence

These steps are described below.

Combining Events

The first step in creating an accident sequence that includes multiple events is to select the events (each Traj Sim Window displays exactly one event). The order of events in the Playback Editor is important, and is discussed in detail in the Object Precedence section of this chapter.

> **NOTE:** The following example assumes several Traj Sim Windows containing simulation events have already been created, as explained in Chapter 17.

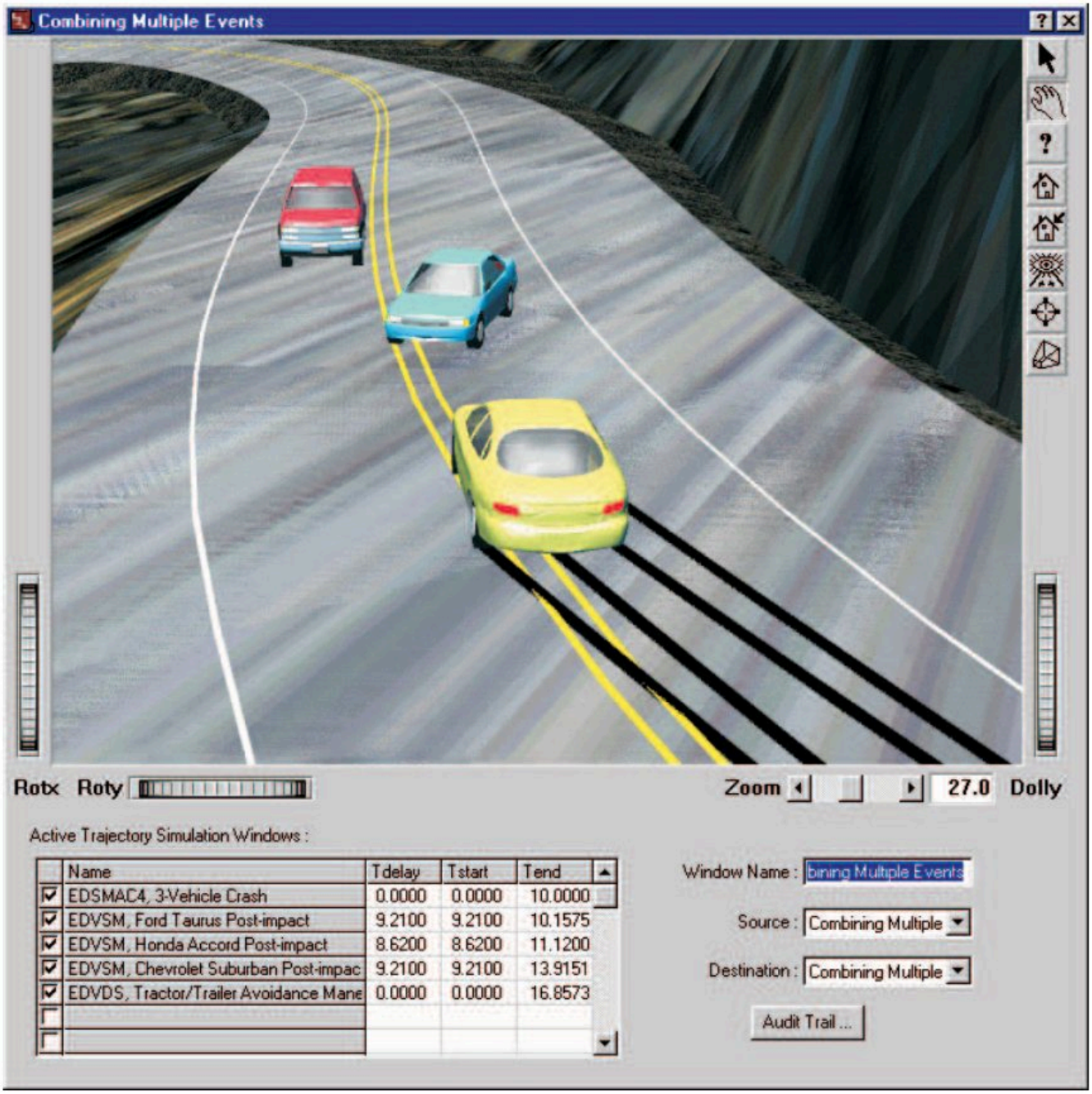


Figure 18-1 — The Playback Window allows the user to combine multiple events.

To select the events, use the Playback Editor to perform the following steps:

1. Display the Playback Information dialog (see Figure 18-1). *(updated: the legacy "File, Options, Add Playback Window" command has been removed; the Playback Information dialog is opened from Playback mode as described in Chapter 17.)* This dialog includes the Active Trajectory Simulations table.
2. Choose the desired events from the Active Trajectory Simulations table by setting the check box next to each event.

> **NOTE:** The order of events displayed in the Active Trajectory Simulations table is important; see Playback Object Table, later in this chapter. *(updated: the current dialog includes a Reorder Events pushbutton, so the order can be changed after the windows have been created.)*

1. Enter a name for the Playback Window (the name is optional).

The Playback Window will include all the humans and vehicles used in each of the selected events.

Editing Events

After the events are selected, they normally must be edited so each event occurs at the proper time and sequence. In addition, certain non-calculated information may be edited. While performing these operations, the user assumes a role much like a movie editor: the Output Tracks produced by each event are the film, and each event's Active Trajectory Simulations listing becomes the cutter and splicer. These editing operations are described below.

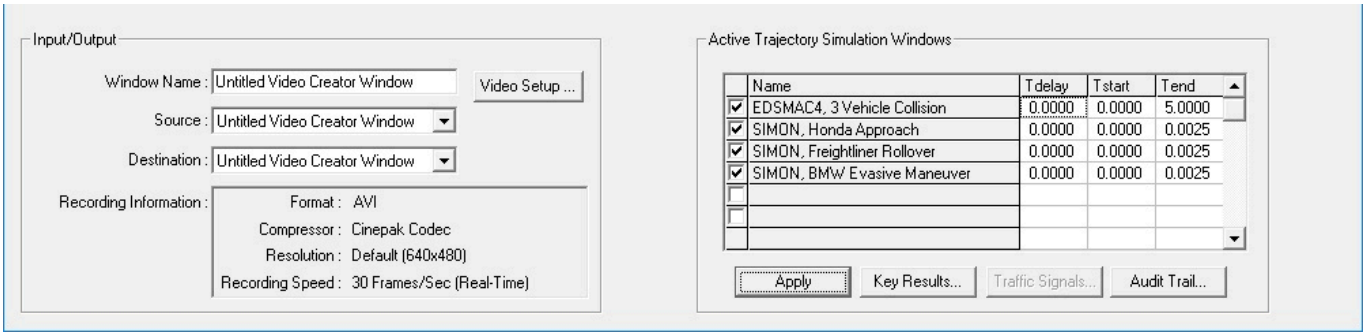


Figure 18-2 — Active Trajectory Simulations table found in the lower portion of the Playback Information dialog. The editing parameters, Tdelay, Tstart and Tend, are located in this table.

Timing The Sequence

The timing of each event is accomplished using the Active Trajectory Simulations table. As shown in Figure 18-2, each event's listing includes fields that allow the user to set the following timing parameters:

- **Editing Tdelay** — Set Tdelay to delay the onset of an event by the user-entered time value.
- **Editing Tstart and Tend** — Set Tstart and Tend to display a portion of the entire event, defined from Tstart to Tend.

> **NOTE:** Tstart and Tend allow you to select a specific portion of the event. For example, an occupant simulation may contain 50 milliseconds of irrelevant motion while the human settles into the seat and reaches equilibrium. By selecting Tstart as 50 milliseconds after the output begins, you can remove this undesired motion.

To edit the timing of an event, perform the following steps:

1. Choose the desired event and enter the delay time, start time and end time. The Active Trajectory Simulations table will be updated when you press Enter in each field.
2. Using the Playback Controller, press Rewind, then press Play. You will notice that objects for that event will not begin moving until the current simulation time reaches Tstart in either the Traj Sim or Playback Window.
3. Repeat the above steps for each event you wish to edit.

The motion for each event is displayed separately in its Traj Sim Window. Use the Traj Sim Window to confirm the event(s) are correctly synchronized. The Playback Window shows all the events correctly synchronized in a single window.

> **NOTE:** After a Playback Window has been created, you may wish to iconify (minimize) all the Report Windows to reduce rendering time.

Object Precedence

It seems like a fairly simple task: create a window, put all the events in it and start the simulation running. However, there are some technical issues to be concerned with. For example:

- What if two events in the window have the same object?
- What if an object is changed (for example, suppose a vehicle is damaged in the middle of the sequence)?

The solution to these problems is based on a simple observation: the number of objects included in the Playback Window remains fixed. If you think about it, this makes sense! Cars and humans cannot disappear and/or reappear in the middle of an accident sequence (contrary to some of the stories you hear!).

Rules

To implement the above solution requires a few important rules that describe the hierarchy of objects:

- **Rule 1:** The number of objects (humans and vehicles) in the Playback Window remains fixed.
- **Rule 2:** Object Names, assigned when the Human or Vehicle Editor was used to create the object, are used to distinguish different humans and vehicles.
- **Rule 3:** Parentheses in object names are used to distinguish between different instances of the same object. For example, the user may create two vehicles: *Ford F-150* and *Ford F-150 (Damaged)*. The Playback Editor views these as the same vehicle. This allows the user to model changes to a vehicle that occurred during the sequence.
- **Rule 4:** If more than one instance of an object exists at the same time, the last one in wins. Thus, the order that events are selected in the Active Trajectory Simulations list is important. New instances replace existing instances. For example, if *Ford F-150* appears in an event that starts at t=0 seconds and also in a second event that starts at t=2.1 seconds, the motion of the Ford will be driven by the first event from 0 to 2.1 seconds and by the second event thereafter (or until a third event takes over).
- **Rule 5:** Vehicles containing humans have the lowest precedence. Thus, if an occupant event includes *Ford F-150* and, simultaneously, a vehicle event also includes *Ford F-150*, the F-150 in the occupant event will be ignored. The F-150's motion will be driven by the vehicle simulation event.

Playback Object Table (Audit Trail)

The above rules are used to create an internal table, called the Playback Object Table. This table determines exactly which events are driving which objects. If you are curious about this table, take a look at the Audit Trail: the Audit Trail (see Figure 18-3) is actually a printable dialog that displays the Playback Object Table!

Refer to the Tutorial section of this manual for examples of editing an accident sequence.

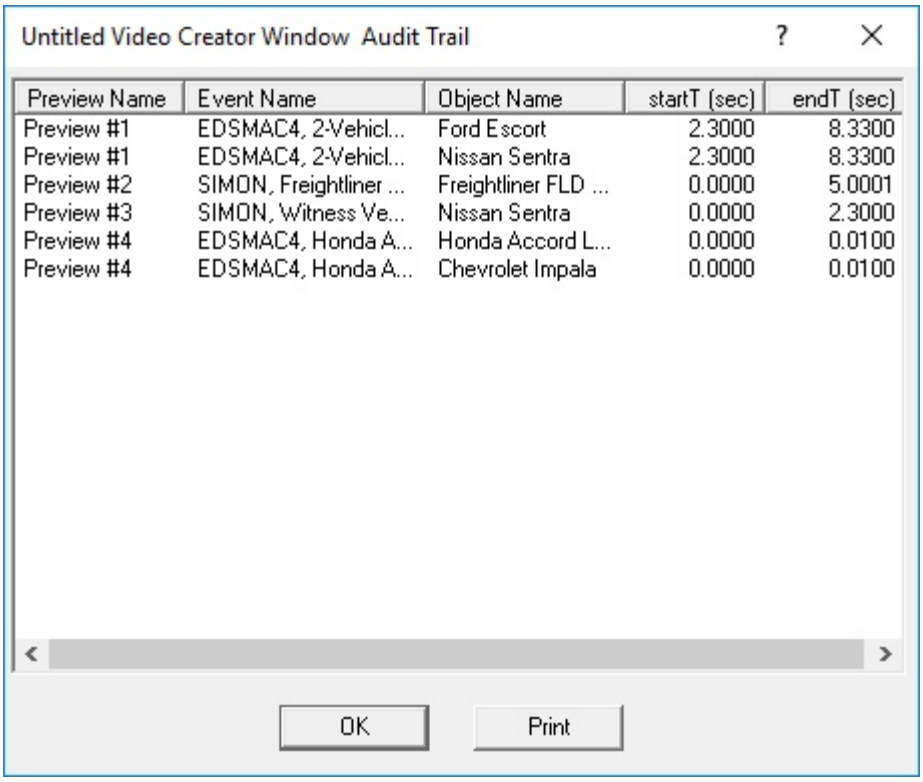


Figure 18-3 — Audit Trail.

Editing Non-calculated Results

The developer of a human or vehicle simulation model may choose to make certain simulation results editable by the HVE user. For example, a 2-D simulation may allow the user to edit the sprung mass Z position and roll and pitch orientations. By editing these non-calculated fields, the user can create a 3-D sequence using a 2-D program.

> **NOTE:** The term non-calculated is slightly misleading. Actually, the Z, roll and pitch values will be calculated automatically by HVE using AutoPosition.

Non-calculated results are edited using the Variable Output Window (see Variable Output Report). To edit these results, perform the following steps:

1. Choose **Add New Object**. The Report Window dialog will be displayed.
2. Select a simulation event (only simulations produce a Variable Output Window).
3. Click on Report Type and choose Variable Output.
4. Optionally, enter a name for the Variable Output window.
5. Press OK.

The Variable Output Table for the selected event will be displayed. Repeat the above procedure for each event you wish to edit.

The next step is to select the non-calculated results for each event:

1. Click on **Select** in the Variable Output window for the desired event. The Variable Selection dialog for the event will be displayed.

2. Click the Object Name option button and select the desired human or vehicle.
3. Select the desired non-calculated results for the human or vehicle.

> **NOTE:** Refer to the Simulation Model's documentation to determine which results are non-calculated.

1. Press OK.

The selected variables will be displayed in the Variable Output Window.

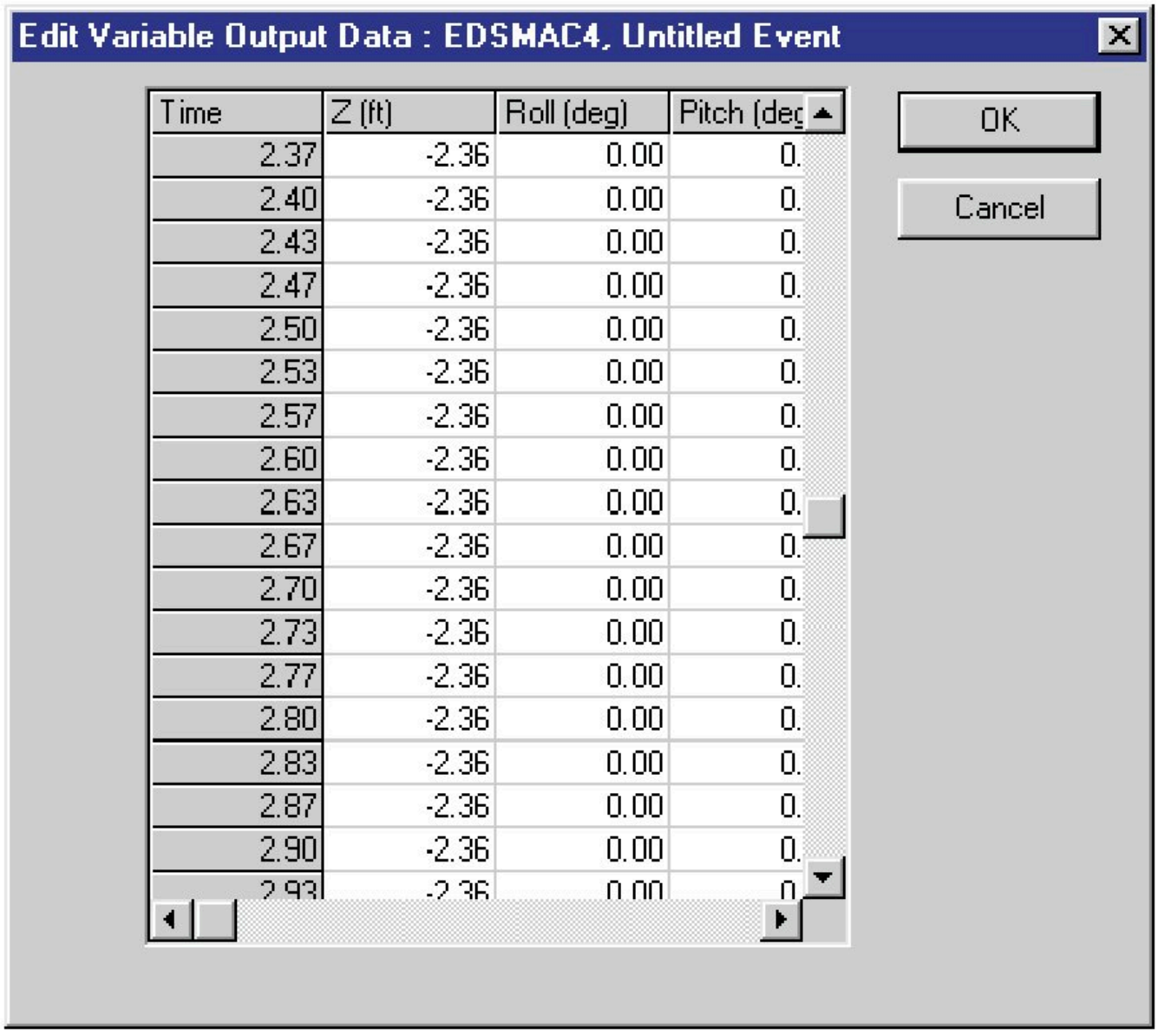


Figure 18-4 — Edit Variables dialog.

The next step is to edit the selected variables:

1. Click **Edit**. The Variable Edit dialog will be displayed, containing a table of the selected variables.
2. Edit the table.
3. Press OK.

The edited values are displayed in the Variable Output Table. Now, use the Playback Controller to visualize the changes:

1. Press Rewind.
2. Press Play.

The Trajectory Simulation window (and possibly the Playback Window, if one is present and it includes the edited Traj Sim Window) will display the edited results.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 18; revised to match the current version of HVE, 2026-07-05.

Section Eight: 3-D Editor

The 3-D Editor is used for creating and editing the objects used by HVE. These objects are broadly categorized as human, vehicle or environment objects. The user creates and edits these objects using the 3-D Editor's selection of creation and editing tools. The 3-D Editor can also edit objects created using other programs. This important capability allows the user to create named overlays and add friction properties to objects used by HVE.

General Procedure

To use the 3-D Editor, perform the following steps:

1. Select Environment mode.
2. Add an environment, if one does not exist.
3. Start the 3-D Editor (3-D Edit menu, Launch 3-D Editor, or Ctrl+L). The current 3-D environment will be automatically loaded and displayed in the viewers.

> **NOTE:** If no 3-D geometry file is selected, the viewers will be empty.

1. Create or edit the current 3-D geometry.
2. Close the 3-D Editor to return to Environment mode. The current 3-D environment geometry is displayed in the Environment Viewer.

Overview of Section Eight

This section provides a detailed explanation of the process of creating and editing 3-D objects (humans, vehicles and environments):

> **NOTE:** Only environments may currently be edited in the 3-D Editor.

- Chapter 19 provides an overview of the 3-D Editor.
- Chapter 20 describes each of the 3-D Editor Object Tools.
- Chapter 21 describes the positioning of objects using manipulators.
- Chapter 22 (reserved for future use)
- Chapter 23 provides a tutorial for using the basic components of the 3-D Editor.
- Chapter 24 (reserved for future use)
- Chapter 25 (reserved for future use)

Related Reference Pages

- 3-D Editor Menu
- Object Attributes Dialog Box
- Overlays Dialog Box
- Object editor dialogs: Surface, Cone, Box, Cylinder, Sphere, Text, Light

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Section Eight; revised to match the current version of HVE, 2026-07-05.

Chapter 19 — Using the HVE 3-D Editor

The purpose of the 3-D Editor is to provide the user a tool to create and edit 3-D objects used to visualize humans, vehicles and environments. In addition to providing visual information, the surfaces are also physical surfaces, with physical properties, including surface normal and friction properties, that simulate a physical environment for human and vehicle dynamics models. This chapter provides an overview of the 3-D Editor's basic components.

Description

The 3-D Editor consists of the following basic components (see Figure 19-1):

- Viewers
- Object Attributes Dialog
- Object Editor Dialog

This chapter provides detailed descriptions for each of these components.

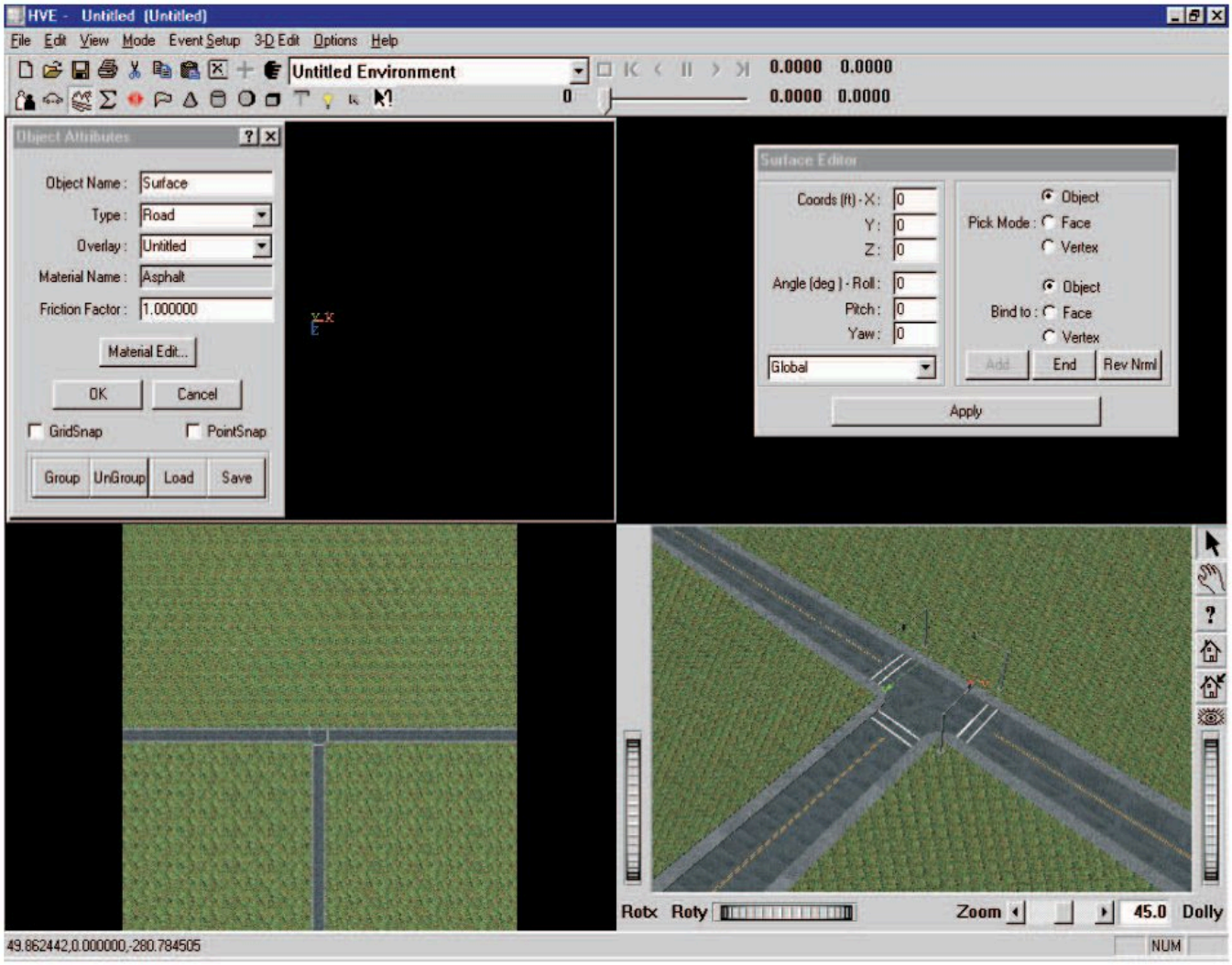


Figure 19-1 — 3-D Editor, including four viewers, Object Attributes dialog and current object (in this case, Surface) dialog.

Viewers

As shown in Figure 19-1, the 3-D Editor has four viewers:

- **Perspective Viewer** — Displays the object at any user-specified perspective.
- **X-Y Orthographic Viewer** — Displays the object as viewed along an axis normal to the X-Y plane (i.e., along the Z axis) — the plan view.
- **Y-Z Orthographic Viewer** — Displays the object as viewed along an axis normal to the Y-Z plane (i.e., along the X axis) — the front view.
- **X-Z Orthographic Viewer** — Displays the object as viewed along an axis normal to the X-Z plane (i.e., along the Y axis) — the side view.

The user may select and edit any object displayed in any of these viewers.

(updated: the 3-D Edit menu now includes a Viewer option that displays the 3-D Editor Viewer Options dialog. Its Show Viewer(s) drop-down selects which viewer layout is displayed while editing: the XZ (side), YZ (front) or XY (plan) orthographic viewer alone, the Perspective viewer alone, or All (all four viewers at once). See 3-D Editor Menu.)

3-D Editor Dialog

The 3-D Editor dialog is the heart of the 3-D Editor. As shown in Figure 19-2, the 3-D Editor dialog contains the following basic components:

- **Object Attributes Tool** — Allows the user to assign the Object Type, Overlay Name and Friction Factor attributes for the current object. The Object Name is assigned when the object is created (according to the tool used to create it; see Current Object Tool, below), and the Material Name is assigned according to the current object's material attributes.
- **Group Tool** — Allows the user to combine and disassemble previously created objects, as well as to load and save previously created objects for future use in other 3-D geometry files.

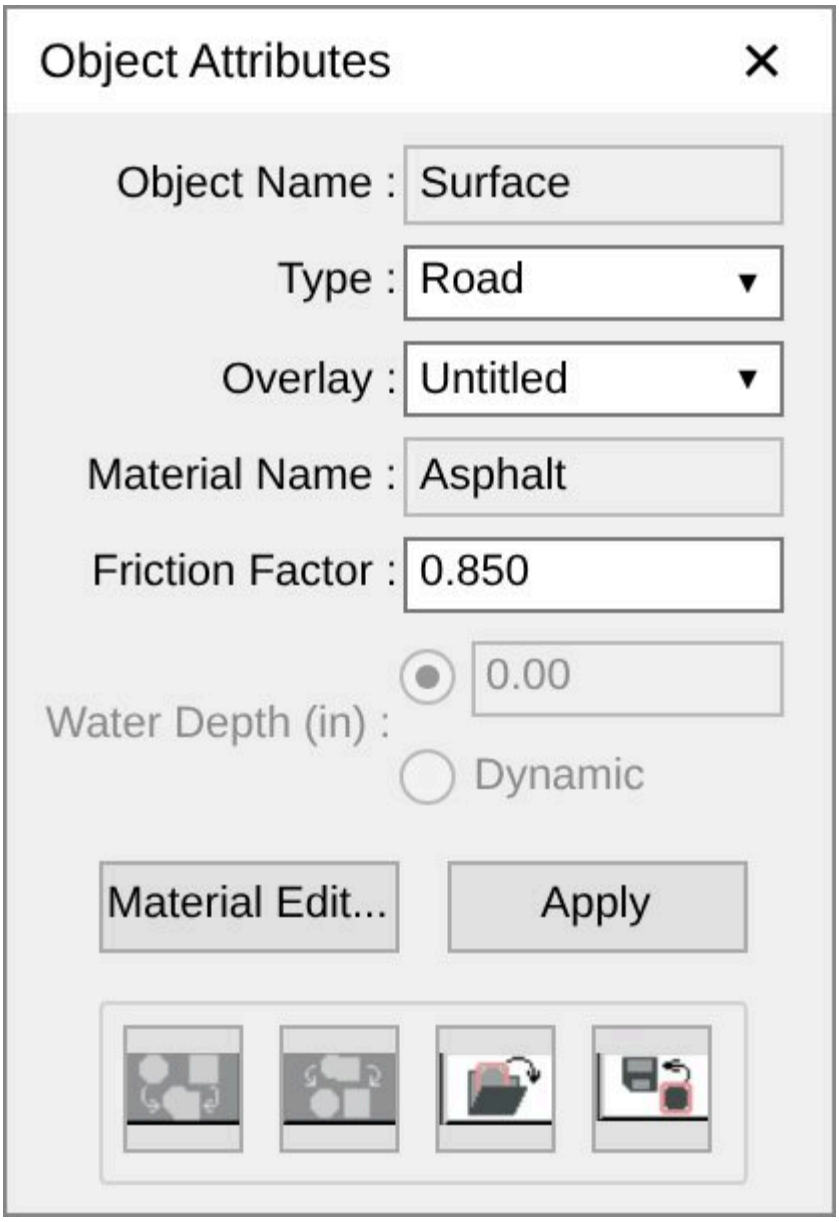


Figure 19-2 — The 3-D Editor dialog includes the Current Object Tool, Object Attributes Tool and Group Tool.

Current Object Tool

An object, such as a surface or sphere, is always created by first selecting an object tool.

(updated: the object tools are now selected from the toolbar rather than from the 3-D Editor dialog.)

The following Object Tools are available (see Figure 19-3):

- **Surface** — Used for creating surfaces. This tool allows the user to create drivable surfaces (roads, medians and adjacent surfaces).
- **Cone** — Used for creating basic cone-shaped objects (great for highway delineators).
- **Box** — Used for creating basic box-shaped objects (for example, buildings and curbs).
- **Sphere** — Used for creating spherical- and ellipsoidal-shaped objects.
- **Cylinder** — Used for creating cylindrical-shaped objects (man-hole covers, speed bumps).

- **Text** — Used for adding text to objects (for example, street signs, building signs and highway delineation).
- **Light** — Used for adding light sources (such as street lights).
- **Signal** — Used for creating traffic signal lights in the environment. *(updated: new tool, not present in the legacy manual.)*
- **Edit** — Used for selecting objects for editing.

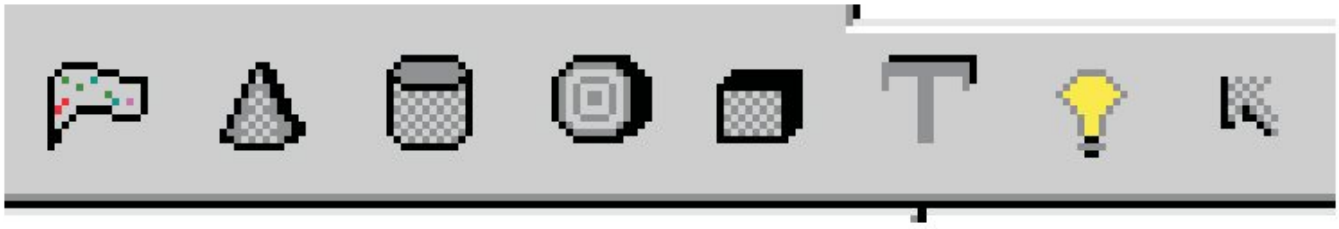


Figure 19-3 — The Current Object Tool allows the user to create and edit objects.

To use the Object Tool to create an object, perform the following steps:

1. Click on the desired Object Tool (e.g., Box). The Box Editor dialog will be displayed.
2. Place the mouse cursor in any viewer and press/drag mouse button 1. The object will be placed in the viewer at the selected cross hairs location.
3. Drag the mouse to position the object at the desired location.

> **NOTE:** You can also use the Box Editor's position fields to directly enter the desired location.

1. Release the mouse button. The Edit Tool becomes the current tool and the object will remain selected so its attributes may be edited.

Object Dialog

The Object Dialog (see Figure 19-4) displays 3-D geometry data for the current object. The contents of this dialog are dependent on the currently selected Object Tool (e.g., Surface or Cone). Generally speaking, each object dialog displays information about the current object, including its size and position/orientation.

> **NOTE:** See the next chapter, Object Tools, for a complete description of each object dialog.

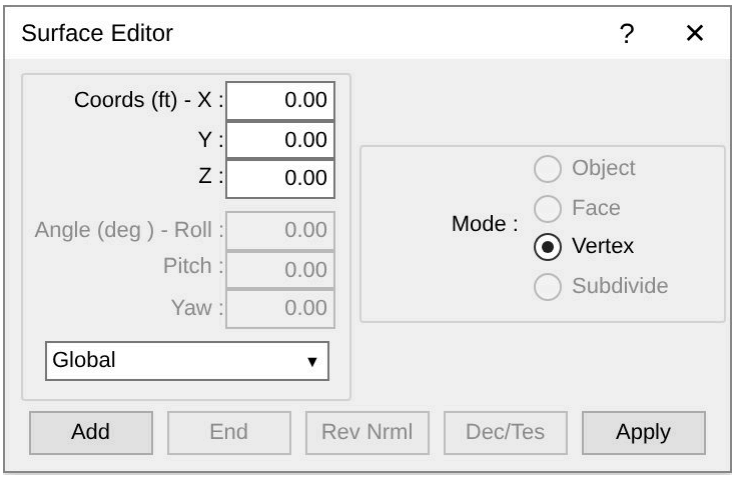


Figure 19-4 — The Current Object Dialog allows the user to edit the geometric properties of the current object. Each Object Tool has a special Object Dialog to meet the needs of the object; the Surface Object dialog is shown.

Object Attributes Tool

Every object created using the 3-D Editor has the following attributes (see also the Object Attributes Dialog Box reference):

- **Object Name** — A non-editable text field that displays the name of the tool used to create the object (e.g., Box).
- **Object Type** — An option list that allows the user to define the type of object (Human, Vehicle, Road, Friction Zone, Curb, Water or Other). *(updated: the Curb and Water types have been added since the legacy manual; selecting Water enables Water Depth controls.)*
- **Overlay Name** — A combo box that allows the user to assign an overlay name for the current object.
- **Material Name** — A non-editable text field that displays the material name of the current object.
- **Friction Factor** — A user-editable field allowing the user to enter the friction multiplier for the current object.
- **Material Pushbutton** — A button that allows the user to select or edit material attributes for the current object.

The current attributes for the selected object are displayed and edited using the Object Attributes Tool (see Figure 19-5).

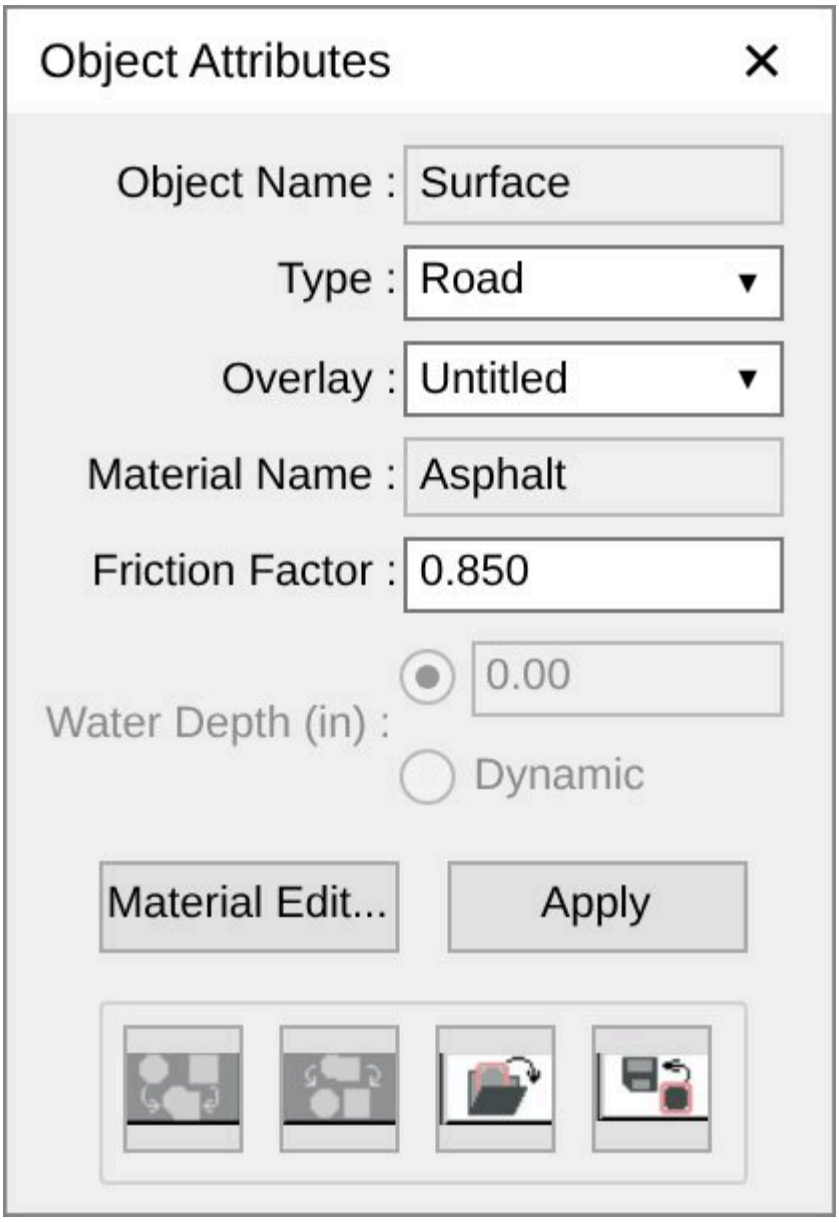


Figure 19-5 — The Object Attributes Tool allows the user to view and edit the attributes for the current object.

Object Name

The Object Name is simply the name of the object tool used to create the object (e.g., Surface, Box, Cylinder). If the selected object is a group composed of several individual objects, the Object Name is Group. If the grouped object was selected from the Highway Furnishings Library, the Object Name is the object's filename.

> **NOTE:** Groups are further discussed later in this chapter.

Object Type

Every object created by the 3-D Editor has an Object Type attribute. The available types are:

- **Human** — The default object type attribute for humans.

- **Vehicle** — The default object type for vehicles.
- **Road** — The default object type for environments. Road objects have physical properties (surface elevation, normal and friction factor) used by the current reconstruction or simulation model.
- **Friction Zone** — An optional object type. Friction Zone objects have physical properties (surface elevation, normal and friction factor) used by the current reconstruction or simulation model.
- **Curb** — An optional object type identifying the object as a curb for calculation models that include curb impacts. *(updated: new type.)*
- **Water** — An optional object type identifying the surface as water; selecting Water enables the Water Depth controls. *(updated: new type.)*
- **Other** — An optional object type for purely visual objects.

> **NOTE:** An object type of Road or Friction Zone must be selected if the object is part of the physical environment with which a human or vehicle may interact.
>
> **NOTE:** If an environment object will probably not interact with humans or vehicles (for example, the top of a tree or the roof of a building), you should select Other. Setting the object type to Other removes the object from the list of surfaces the physics needs to evaluate, thus reducing the computation time.

To assign the object type to the selected object, perform the following steps:

1. If necessary, select an object (a bounding box is displayed around the selected object). The attributes (Name, Type, Overlay and Friction Factor) for the selected object are displayed.
2. Click on the current Object Type. The Object Type option list is displayed.
3. Choose the desired object type from the list.

The current object type is updated.

Overlay Name

Every object created using the 3-D Editor belongs to a named overlay. Examples of named overlays are Accident Debris and Measured Skids.

> **NOTE:** The default Overlay Name is "Untitled".

HVE maintains a list of current overlay names. This list is available in the Overlays dialog by choosing Overlays from the View menu (see Overlays Dialog Box). The user may display or remove one or more overlays using the Overlays dialog.

Overlays serve several very useful purposes. Some examples are:

- Overlays may be used to show an accident scene as it appeared before and after the accident by creating overlays named Accident Debris and Skidmarks. Turning off these overlays shows the scene as it appeared before the accident; turning them on shows the scene after the accident.
- An overlay may be used to compare simulated and measured skidmarks.
- A scanned image (bitmap) overlay may be used to compare a 3-D model of the accident site with a photograph to confirm the model's accuracy.

To assign an overlay name to the selected object, perform the following steps:

1. If necessary, select an object (a bounding box is displayed around the selected object). The attributes (Name, Type, Overlay and Friction Factor) for the selected object are displayed.
2. Modify the current overlay name using either of the following procedures:
3. Click on the Overlay Name combo box button to display a list of previously created overlay names and choose the desired name, or
4. Place the mouse cursor in the Overlay Name combo box text field (it will become a text cursor) and edit the current name to create a new overlay name. Press Enter to update the list of overlays.

The current overlay name is updated.

Friction Factor

Objects created using the 3-D Editor have friction properties. The Friction Factor is a user-editable object attribute used by the current calculation model to determine the frictional properties of all objects.

Use Friction Factors to create regions with different frictional properties from the base surface. Examples include gravel shoulders, oil slicks and other regions.

To edit the friction factor for the selected object, perform the following steps:

1. If necessary, select an object (a bounding box is displayed around the selected object). The attributes (Name, Type, Overlay and Friction Factor) for the selected object are displayed.

> **NOTE:** If the current object type is Other, the Friction Factor field is disabled. To enable the Friction Factor field, change the object type to Road or Friction Zone.
>
> **NOTE:** If the current object type is Other, the object's friction and other geometrical properties will not be used by the current calculation model.

1. Place the mouse cursor in the Friction Factor field and edit the value.
2. Press Enter to update the value.

The friction multiplier for the selected object is updated.

> **NOTE:** For more information about how the friction attribute is used, see Environment Model Definition.

Materials Editor

The Materials Editor (see Figures 19-6 and 19-7) allows the user to apply material attributes (color and texture) to an object's surface, and to determine how that surface reflects light from its surroundings. The object's basic color is determined using a color wheel, located in the upper right quadrant of the dialog, and a color intensity slider, located just below the color wheel. The remaining attributes are assigned using a series of sliders located below the color wheel. A sample illustrating the results of the current color attributes is displayed on a sphere located in the upper left quadrant of the dialog.

The surface appearance is determined by six attributes:

- **Ambient** — The intensity of the color that is reflected in response to the ambient lighting in the environment. Because the lighting in the environment is normally white (or nearly white) light, increasing this attribute normally does not change the color of the object, but makes it brighter. However, if, for example, the object is a vehicle passing beneath a yellow street light, the vehicle's color will become more yellow.
- **Diffuse** — The intensity of the object's basic color. Increasing this attribute also makes the object brighter.
- **Specular** — The intensity of an object's highlights. Increasing this attribute creates a "sparkling" effect. Normally this field is kept at a low level.
- **Emissive** — The intensity of light produced by the object. Increasing this attribute causes the object to glow. An example of an emissive object is a vehicle's taillight. By increasing its emissivity, the taillight appears to be on; however, it does not cast light on other objects.
- **Shininess** — Adds reflectivity to an object. Increasing this attribute creates the appearance of a polished surface.
- **Transparency** — Makes an object invisible. Increasing this attribute increases the object's transparency.

To modify an object's color and appearance, perform the following steps:

1. Click on the object to select it. A bounding box is displayed about the object indicating it has been selected for editing.
2. Using the HVE menu bar, click on 3-D Edit and choose Material Color or Material Texture. The Material Color or Material Texture Editor dialog is displayed, showing the current color or texture attributes.
3. To change the object's basic color, click the Edit radio button next to the attribute to be changed (e.g., Ambient), then click the circle in the color wheel and drag it to the desired color location. The sample sphere reflects the new color.
4. To lighten the object, drag the intensity slider to the right; to darken it, drag it to the left.

> **NOTE:** To create a black object, drag the intensity slider all the way to the left.

1. To modify the object's appearance, drag the appearance sliders, noting the effect on the current appearance, as displayed in the sample sphere.
2. To apply the current color and attributes to the selected object, you do not need to do anything else. The object's color and appearance are automatically updated.

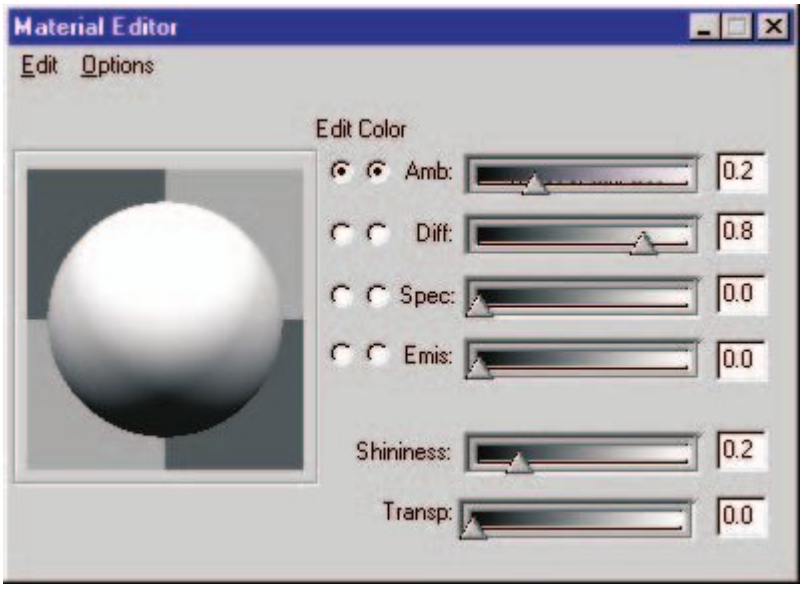


Figure 19-6 — The Materials Editor allows the user to assign color and other material attributes for the current object.

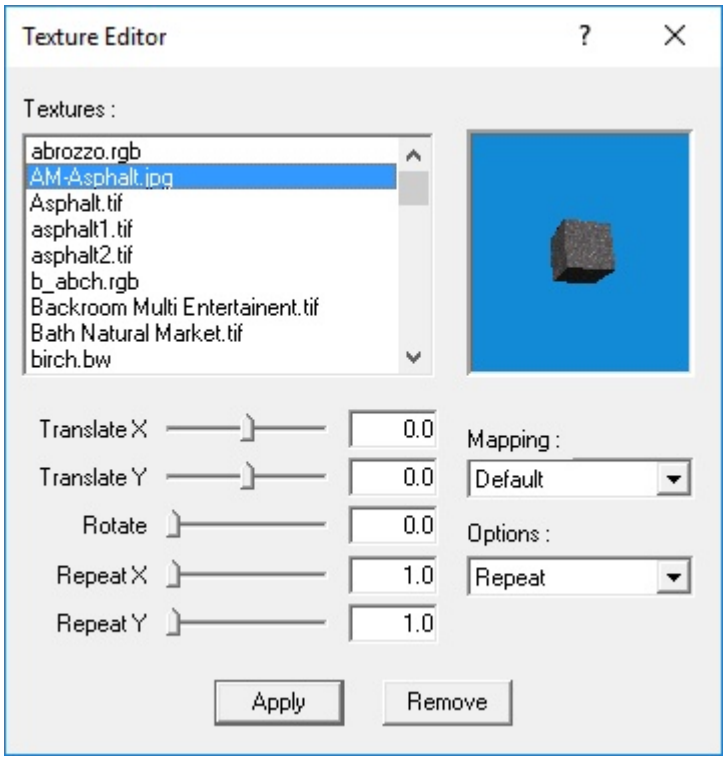


Figure 19-7 — The HVE Texture Editor is used for applying textures to objects.

Texture Editor

HVE includes a powerful tool called the Texture Editor. Textures (more formally referred to as Texture Maps) are basically photographs you can attach to 3-D objects. The texture mapping engine applies the photo to the surface by "mapping" coordinates from the photograph to coordinates on the polygon surface.

Some examples of the use of Texture Maps are:

- **Road Surface** — You can create a road surface and apply a concrete or asphalt texture to it. The result is a road surface that looks like concrete or asphalt.
- **Buildings** — Create the exterior of a building and attach a photo of a building to it.
- **Signs** — Create realistic signs of any type (street signs, billboards, etc.).
- **Dirt/Grass** — Create the roadside adjacent to the road and apply a grass (or dirt) texture to it.
- **Water** — Create a pond, make the bottom surface of type Road, and the water surface of type Other. Apply a water texture to the top surface, increase its transparency slightly, and you can drive into a lake (hydrodynamics are currently ignored!). *(updated: the current Object Type list also includes a dedicated Water type with a Water Depth setting; see the Object Type section above.)*

Using the HVE Texture Editor

Texture editing is performed using the 3-D Editor. Textures are only available in Environment mode. Textures are most often applied to surface objects, although textures may also be attached to cones, spheres and other objects. The Material Editor and the Texture Editor may be used in conjunction with each other to achieve the desired appearance.

The Texture Editor includes the following components:

- **Available Textures List** — A collection of textures placed in a single group. HVE includes several default textures, including buildings, road materials, grass and signs. You can also add your own textures to this list (see Adding New Textures, below).
- **Appearance Attributes** — A collection of controls that allow you to modify the texture's appearance. The available controls are:
- **Translate** — A percentage of the texture (-1 to 1) that is offset before the texture is applied in the X or Y direction. A translation of 0 means the texture is applied starting from the center. A Translate X of -0.5 means that the start of the texture is moved 50 percent to the right. Note that the sliders are positive left, negative right; this arrangement reinforces that concept.
- **Rotate** — Degrees of rotation of the texture.
- **Repeat** — The number of times a texture is repeated or "wrapped" across a face. A repeat of 1 means the texture is stretched from one edge of the face to the other. A repeat of 10 means the texture is repeated 10 times across the face in that direction. The Repeat control is typically used to "scale" the texture to the desired surface.
- **Mapping** — Not yet implemented.
- **Options** — Repeat or Clamp. Note that the Repeat sliders change to Scale when the Options selection is changed from Repeat to Clamp. Options affects how the texture is applied to a face. Repeat is explained above. Clamping allows the user to directly scale the texture, with the last pixel of the picture (texture) being repeated to the edge of the face for scaling less than 1.
- **Accept (Apply) Button** — Causes the current Appearance Attributes to be applied to the object.

> **NOTE:** Textures are applied per FACE, not per object. This has a noticeable effect on objects such as roads (see the example below).

Let's say you want to texture a 200 ft x 40 ft piece of road using a texture called Cement.rgb. To apply the concrete texture to the road, perform the following steps:

1. Find Cement.rgb in the list and click on it. The cube in the render area to the right of the palette should have the concrete texture applied to it, and all fields should have 0's for Translate X and Y and Rotate, and 1's for Repeat X

and Y.

2. Press Accept. The texture is applied to the road, but it doesn't look quite right if you zoom in on it. This is because the picture is being stretched to each edge of the face, therefore making it blurry when you look closely at it. When you zoom out, the texture looks magnified.

To fix this you must increase the Repeat (or wrapping) of the texture:

1. Try changing the Repeat X to 54 and the Repeat Y to 45.5 and press Accept. This produces a "rutted" texture across the road.
2. The ruttedness comes from one edge of the picture not blending with the opposing edge. You can fix this by editing the image in a paint program and matching the edges using a clone tool from one side to the other; we leave this as an exercise for the reader. Note that rutting can be used in a positive way to create the appearance of tire ruts in a road.

Adding New Textures

To add a new texture (for example, a new Cement.rgb) to the existing texture list, perform the following steps:

1. Be sure the Texture Editor is not displayed on your desktop. Now, using your file browser, copy the texture file into the `\HVE\supportFiles\images\environments\EnvTextures` directory.
2. Select 3-D Edit, Material Texture from the main menu.
3. Scroll down the list of textures until you see the file you just copied into the directory.
4. Press Accept. The texture is applied to the road.

Group Tool

A group is a collection of objects that are manipulated as a single object. Groups are created by combining individually created objects (Surfaces, Boxes, etc.).

HVE's 3-D Editor includes the following group tools (see Figure 19-8):

- **Group** — Combines the selected objects into a group (single object).
- **Ungroup** — Decomposes the selected group into its individual objects or groups.
- **Load (Open)** — Displays a file selection dialog allowing the user to add a previously created group to the current human, vehicle or environment.
- **Save** — Displays a file selection dialog allowing the user to save the currently selected group for use in other cases.

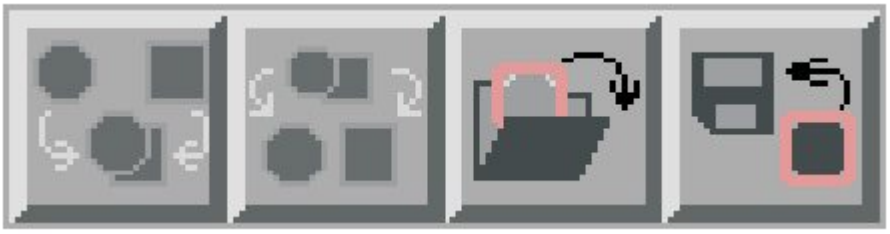


Figure 19-8 — The Group Tool allows the user to create and disassemble grouped objects, to save grouped objects and to insert grouped objects into other scenes.

Group

To create a group, perform the following steps:

1. Use the Object Tool to create two or more individual objects.
2. Select the individual objects using multiple selection (hold the Shift key down while clicking on the desired objects).

> **NOTE:** Groups can be nested (placed inside other groups).

1. Choose Group.

The desired group will be created.

Ungroup

To disassemble an existing group, perform the following steps:

1. Click on the grouped object to select it.
2. Choose Ungroup.

The grouped object will be disassembled into its individual objects.

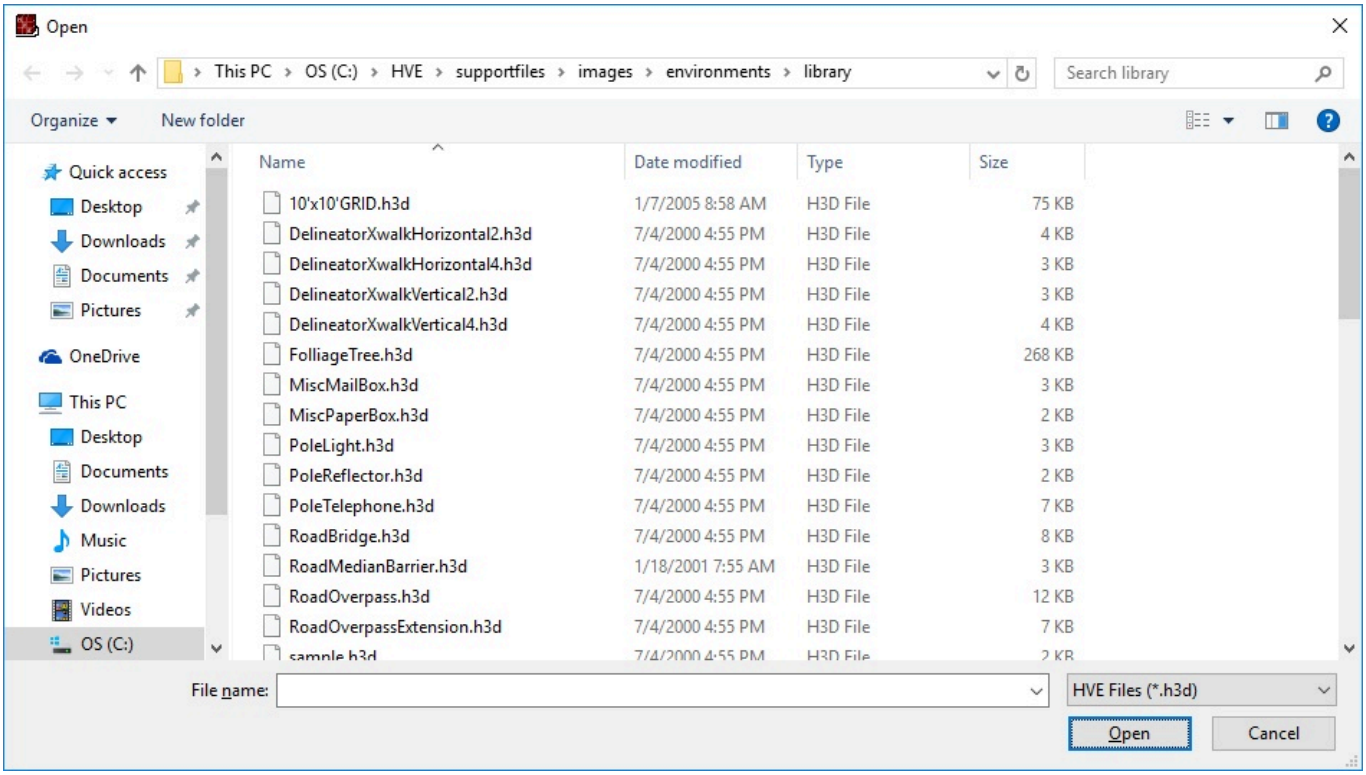


Figure 19-9 — The Group Object File Selection dialog allows the user to load and save grouped objects.

Open

To add a previously created group to the current scene, perform the following steps:

1. Click on the 3-D Editor's Open icon. The Highway Furnishings Library file selection dialog will be displayed (see Figure 19-9).
2. Select the desired filename and press OK.
3. Click the mouse in the viewer to add the group to the current scene (just like adding a box or any other object).

The grouped object will be displayed in the scene.

> **NOTE:** The grouped object will be located according to the earth-fixed coordinates where it was saved. If you want the default location of the grouped object to be the origin, you must move it there before saving it in the library.

After adding the grouped object to the scene, it may be positioned and edited like any other object.

Save

To save the current group for future use, perform the following steps:

1. Click on the grouped object to select it.
2. Click on the 3-D Editor's Save icon. The Highway Furnishings Library file selection dialog will be displayed (see Figure 19-9).
3. Select or enter a filename and press OK.

The named group will be added to the library.

> **NOTE:** Objects are saved according to their current location relative to the origin. This fact is important when the object is later selected from the library and positioned in the environment. If the object was located 50 ft from the origin when it was saved, it will be located 50 ft from the current coordinates when it is pasted back into the environment. Thus, you will normally want to move the object to the origin before saving it in the library.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 19; revised to match the current version of HVE, 2026-07-05.

Chapter 20 — 3-D Editor Object Tools

The 3-D Editor is used for creating and editing the objects used by HVE. These objects are broadly categorized as human, vehicle or environment objects. The user creates and edits these objects using the 3-D Editor's selection of creation and editing tools.

Overview

An object, such as a surface or sphere, is always created by first selecting an object tool from the toolbar. The following Object Tools are available:

- **Surface Tool** — Used for creating surfaces, such as roads, shoulders and adjacent highway environment.
- **Cone Tool** — Used for creating cone-shaped objects such as traffic cones.
- **Box Tool** — Used for creating box-shaped objects, including buildings, sidewalks and bridges.
- **Sphere Tool** — Used for creating spherical- and ellipsoidal-shaped objects, such as commercial building signs.
- **Cylinder Tool** — Used for creating cylindrical-shaped objects, such as signs, sidewalk corners and tanks.
- **Text Tool** — Used for labeling objects such as signs, buildings and vehicles.
- **Light Tool** — Used for creating light sources in the environment.
- **Signal Tool** — Used for creating traffic signal lights in the environment. *(updated: new tool; signal timing is coordinated with the Traffic Signals option in the Playback Information dialog.)*
- **Edit Tool** — Used for editing existing objects.

Each of these object tools is described in this chapter. The field-by-field dialog descriptions are kept in the reference pages linked from each section.

Surface Tool

The Surface Object Tool allows the user to create and edit 3-D surfaces, such as roads, medians and any other object which may be defined by a series of polygons. The Surface Editor dialog, shown in Figure 20-1, displays information about the current surface object.

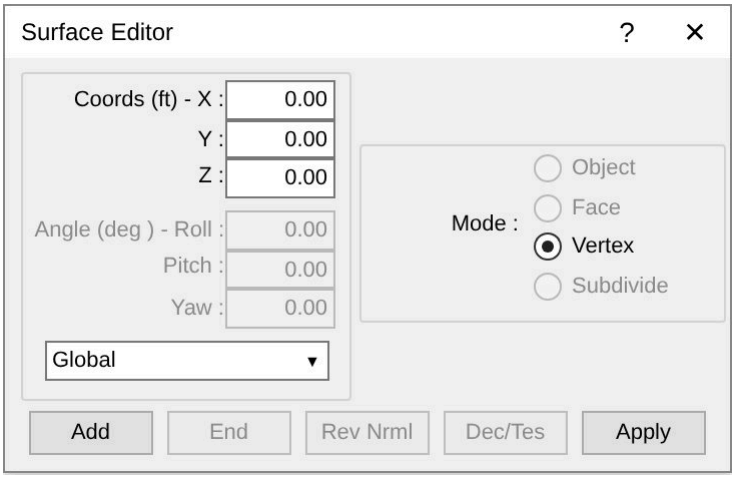


Figure 20-1 — The Surface Editor dialog allows the user to edit the geometric properties of the current surface.

Surface Editor Dialog

See the Surface Editor Dialog reference for full details. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation (active only in Object mode).
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Pick Mode Switch** — Allows switching between picking of surface objects, faces or vertices. *(updated: a fourth mode, Subdivide, has been added — a scissors mode used to split a surface; the split is performed when Apply is clicked.)*
- **Add Face Pushbutton** — Allows adding additional faces (usually, a face is an individual triangle) to the current surface object.
- **End Pushbutton** — Closes the current face or face set (a face set is a connected set of triangles).
- **Reverse Normal (Rev Nrml)** — Reverses the direction of the selected face's surface normal (faces with normals pointing away from the sun will not be lit by the sun).
- **Dec/Tes Pushbutton** — Displays the Decimate/Tessellate dialog, used to reduce (decimate) or refine (tessellate) the surface mesh. *(updated: new in the current version.)*
- **Apply** — Accepts the entered parameters; Surface Editor actions applied this way are registered with the Undo/Redo manager. *(updated.)*

(updated: the "Bind To" radio buttons that appeared on this dialog in some earlier versions are no longer part of the Surface Editor; they remain in the Cone and Cylinder editors. Also, when the Surface tool is selected the editor now starts in Vertex mode so the vertices of a new surface can be entered immediately.)

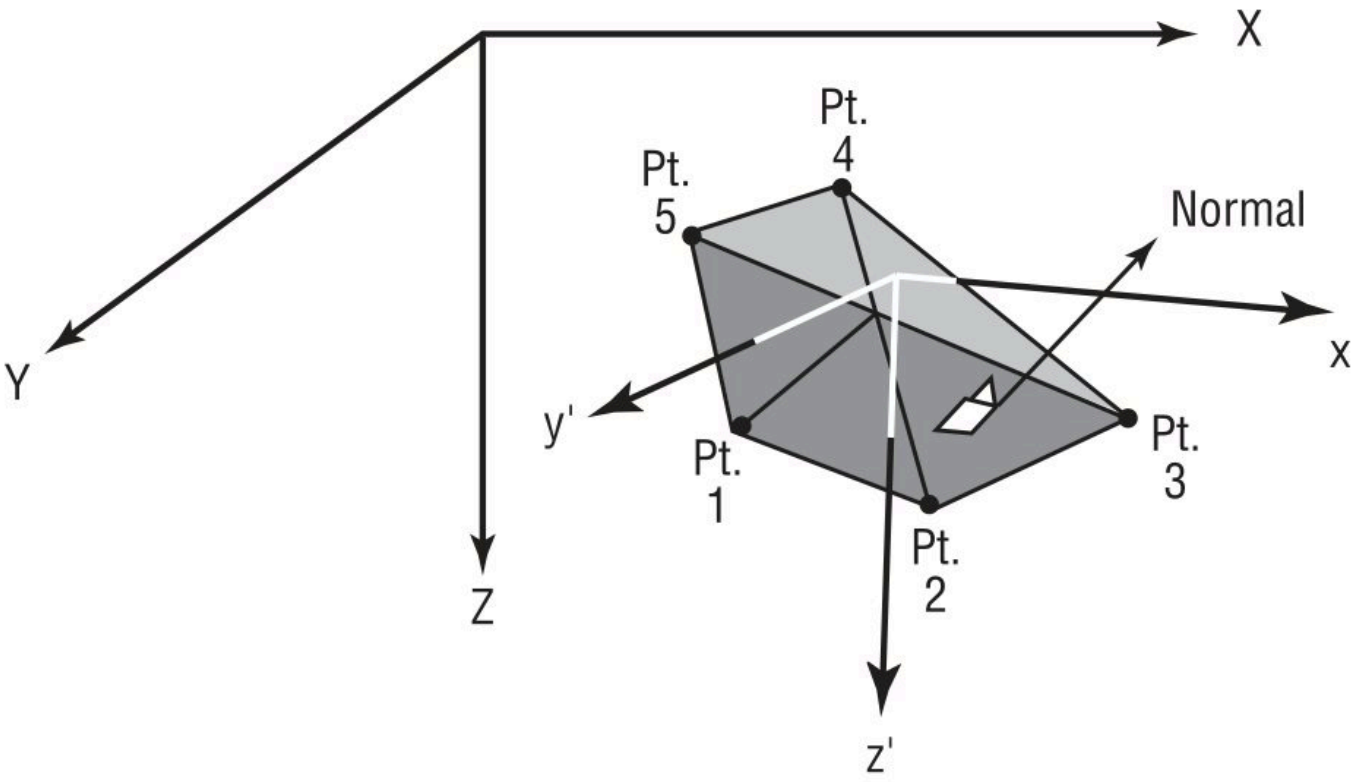


Figure 20-2 — Surface geometrical properties. The example shows a single surface object composed of a single face set containing five triangles, created by clicking points 1 through 5 in counter-clockwise order.

Creating A New Surface

To create a new surface, perform the following steps:

1. Choose the Surface Object tool.
2. Enter the X,Y,Z coordinates of at least three vertices (a minimum of three points is necessary to create a single face).
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates of each vertex.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the coordinate fields in the Surface Editor dialog. Each vertex is created when you press Apply.
5. Close the face.
6. **Using the Mouse:** Reselect the first vertex you entered (the first vertex is highlighted in a different color), or press End in the Surface Editor dialog.
7. **Using the Keyboard:** Instead of pressing Enter, press Ctrl+Enter after entering the coordinates for the last vertex.
8. Repeat steps 2 and 3 for each desired face.

When finished creating the surface object, choose the Edit tool. After creating the surface it becomes the current object and its materials and other properties may be edited.

Editing an Existing Surface

To edit an existing surface, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing surface object (the Attributes Tool will confirm the current object name is Surface). The Surface Editor dialog is displayed.

> **NOTE:** If you click on a surface and the current object name is Group or a user-entered name, the surface you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Choose the Pick Mode according to the desired editing operation:
2. Choose **Object** (the default) to reposition the surface object or apply different attributes (Type, Overlay, Friction Factor or Material).
3. Choose **Face** to edit a single face (usually to reverse its normal).
4. Choose **Vertex** to edit an existing X,Y,Z coordinate location.
5. Choose **Subdivide** to split the surface with a scissors line; press Apply to perform the split. *(updated.)*
6. Perform the desired editing operation.

> **NOTE:** The Surface object remains selected for additional editing operations.

Cone Tool

The Cone Object Tool allows the user to create and edit 3-D cones. Cones are useful for creating highway delineator cones and simple trees. The Cone Editor dialog, shown in Figure 20-3, displays information about the current cone object.

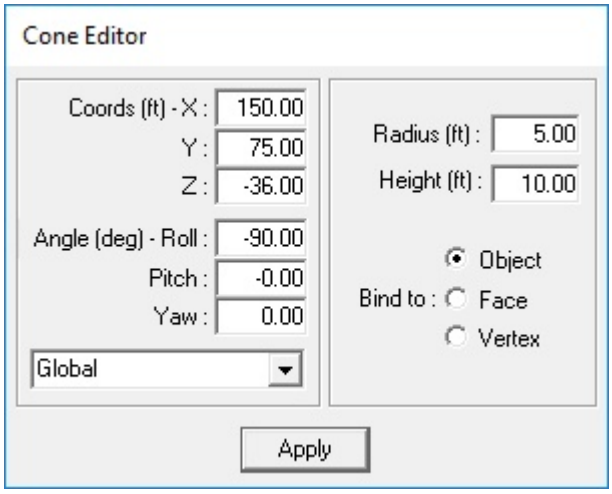


Figure 20-3 — The Cone Editor dialog allows the user to edit the geometric properties of the current cone.

Cone Editor Dialog

See the Cone Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation.
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Base Radius** — Allows editing the cone's base radius.
- **Height** — Allows editing the cone's height.

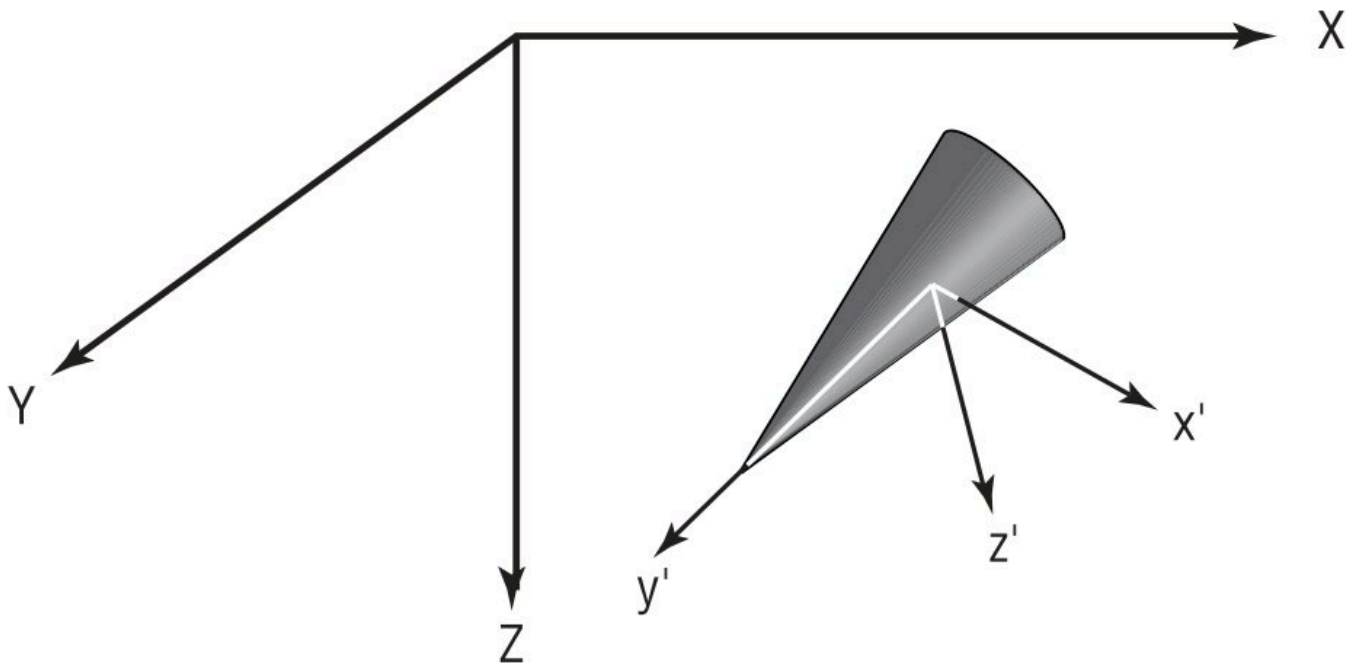


Figure 20-4 — Cone geometrical properties.

Creating a New Cone

To create a new cone, perform the following steps:

1. Choose the Cone Object tool.
2. Enter the X,Y,Z coordinates where you wish the cone to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the base of the cone.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the coordinate fields in the Cone Editor dialog and press Apply.
5. Enter the orientation angles about the X,Y,Z axes.
6. **Using the Mouse:** Click on the 3-D Edit menu and choose Manipulators. A cascade menu displays several manipulator types. Choose the appropriate manipulator (see Chapter 21, Manipulators, to decide which manipulator suits your needs). Then pick the manipulator handles and position the cone as desired.
7. **Using the Keyboard:** Enter the desired angles for the cone in the orientation fields in the Cone Editor dialog and press Apply.

After the cone is positioned, the 3-D Editor will return to Edit mode. The cone becomes the current object and its materials and other properties may be edited.

Editing an Existing Cone

To edit an existing cone object, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing Cone object to select it for editing (the Attributes Tool will confirm the current object name is Cone). The Cone Editor dialog is displayed.

> **NOTE:** If you click on a Cone and the current object name is Group or a user-entered name, the cone you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous section.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The Cone object remains selected for additional editing operations.

Box Tool

The Box Object Tool allows the user to create and edit 3-D boxes. Boxes are useful for creating buildings, sidewalks, curbs and other box-shaped objects. The Box Editor dialog, shown in Figure 20-5, displays information about the current Box object.

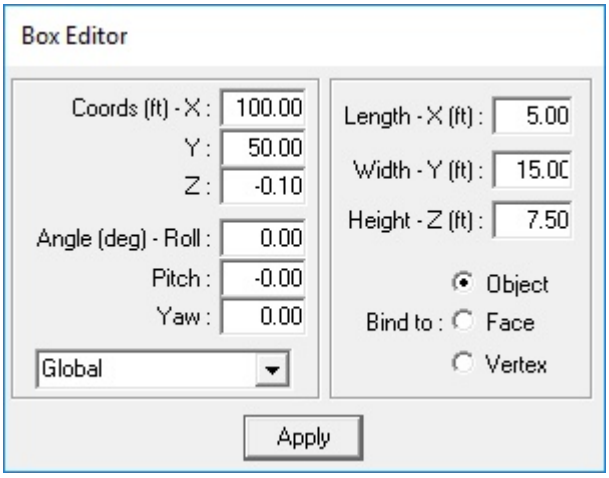


Figure 20-5 — The Box Editor dialog allows the user to edit the geometric properties of the current box.

Box Editor Dialog

See the Box Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation.
- **Local/Global Option Switch** — Allows switching between local (object-based) and global coordinates and angles.
- **Length** — Allows the user to edit the box's length in the local x' direction.
- **Width** — Allows the user to edit the box's width in the local y' direction.
- **Height** — Allows the user to edit the box's height in the local z' direction.

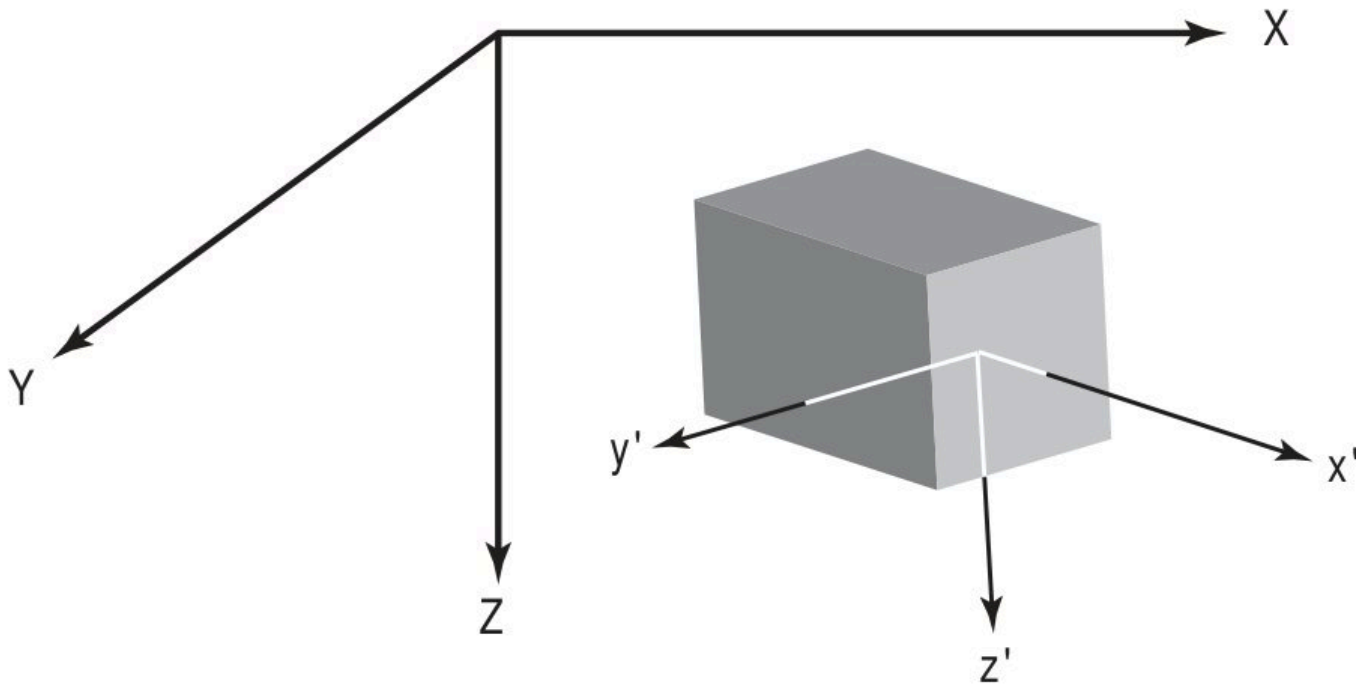


Figure 20-6 — Box geometrical properties. Boxes are useful for creating a variety of objects, such as buildings and sidewalks.

Creating a New Box

To create a new box, perform the following steps:

1. Choose the Box Object tool.
2. Enter the X,Y,Z coordinates where you wish the box to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the center of the box.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the coordinate fields in the Box Editor dialog and press Enter (or click Apply).
5. Enter the orientation angles about the X,Y,Z axes.
6. **Using the Mouse:** Click on the 3-D Edit menu and choose Manipulators. A cascade menu displays several manipulator types. Choose the appropriate manipulator (see Chapter 21, Manipulators). Then pick the manipulator handles and position the box as desired.
7. **Using the Keyboard:** Enter the desired angles for the box in the orientation fields in the Box Editor dialog and press Enter.

After the box is positioned, the 3-D Editor will return to Edit mode. The box becomes the current object and its materials and other properties may be edited.

Editing an Existing Box

To edit an existing box object, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing box object to select it for editing (the Attributes Tool will confirm the current object name is Box). The Box Editor dialog is displayed.

> **NOTE:** If you click on a Box and the current object name is Group or a user-entered name, the Box you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous section.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The Box object remains selected for additional editing operations.

Cylinder Tool

The Cylinder Object Tool allows the user to create and edit 3-D cylinders. Cylinders are useful for creating telephone poles, sign posts and numerous other objects. The Cylinder Editor dialog, shown in Figure 20-7, displays information about the current cylinder object.

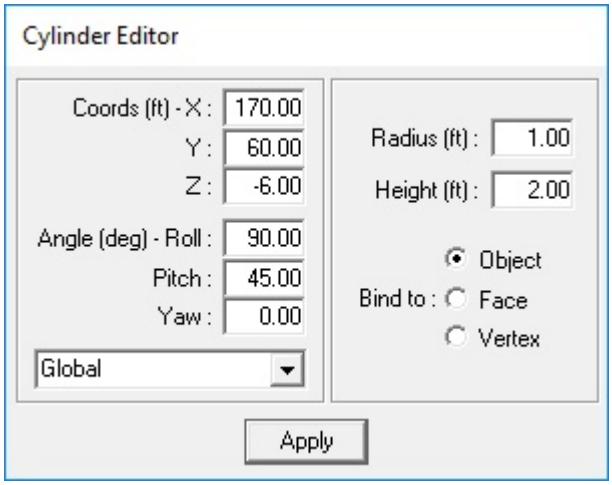


Figure 20-7 — The Cylinder Editor dialog allows the user to edit the geometric properties of the current cylinder.

Cylinder Editor Dialog

See the Cylinder Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation.
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Radius** — Allows the user to edit the cylinder's radius.
- **Length** — Allows the user to edit the cylinder's length.

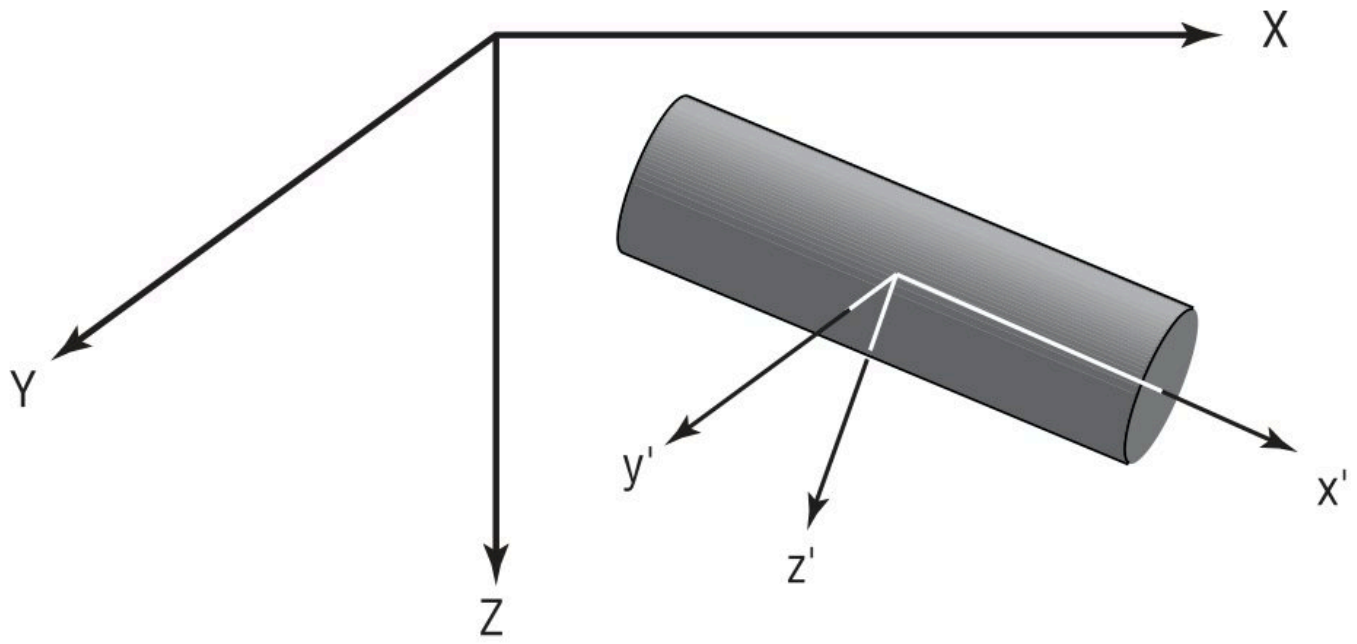


Figure 20-8 — Cylinder geometrical properties. Cylinders are useful for modeling poles, trees and a variety of other cylinder-shaped objects.

Creating a New Cylinder

To create a new cylinder, perform the following steps:

1. Choose the Cylinder Object tool.
2. Enter the X,Y,Z coordinates where you wish the cylinder to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the center of the cylinder.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the fields in the Cylinder Editor dialog and press Enter.
5. Enter the orientation angles about the X,Y,Z axes.
6. **Using the Mouse:** Click on the 3-D Edit menu and choose Manipulators. A cascade menu displays several manipulator types. Choose the appropriate manipulator (see Chapter 21, Manipulators). Then click and drag the manipulator handles and position the cylinder as desired.
7. **Using the Keyboard:** Enter the desired angles for the cylinder in the orientation fields in the Cylinder Editor dialog and press Enter.

After the cylinder is positioned, the 3-D Editor will return to Edit mode. The cylinder becomes the current object and its materials and other properties may be edited.

Editing an Existing Cylinder

To edit an existing cylinder object, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing cylinder object to select it for editing (the Attributes Tool will confirm the current object name is Cylinder). The Cylinder Editor dialog is displayed.

> **NOTE:** If you click on a Cylinder and the current object name is Group or a user-entered name, the Cylinder you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous section.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The Cylinder object remains selected for additional editing operations.

Sphere Tool

The Sphere Object Tool allows the user to create and edit 3-D spheres. Spheres are useful for creating spherical- or ellipsoidal-shaped objects. The Sphere Editor dialog, shown in Figure 20-9, displays information about the current sphere object.

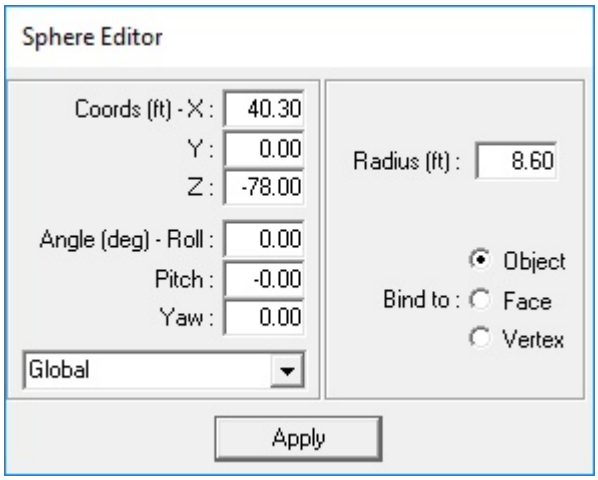


Figure 20-9 — The Sphere Editor dialog allows the user to edit the geometric properties of the current sphere.

Sphere Editor Dialog

See the Sphere Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation.
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Radius** — Allows editing the sphere's radius.

> **NOTE:** The sphere can be elongated in each direction using a manipulator (see Chapter 21, Manipulators).

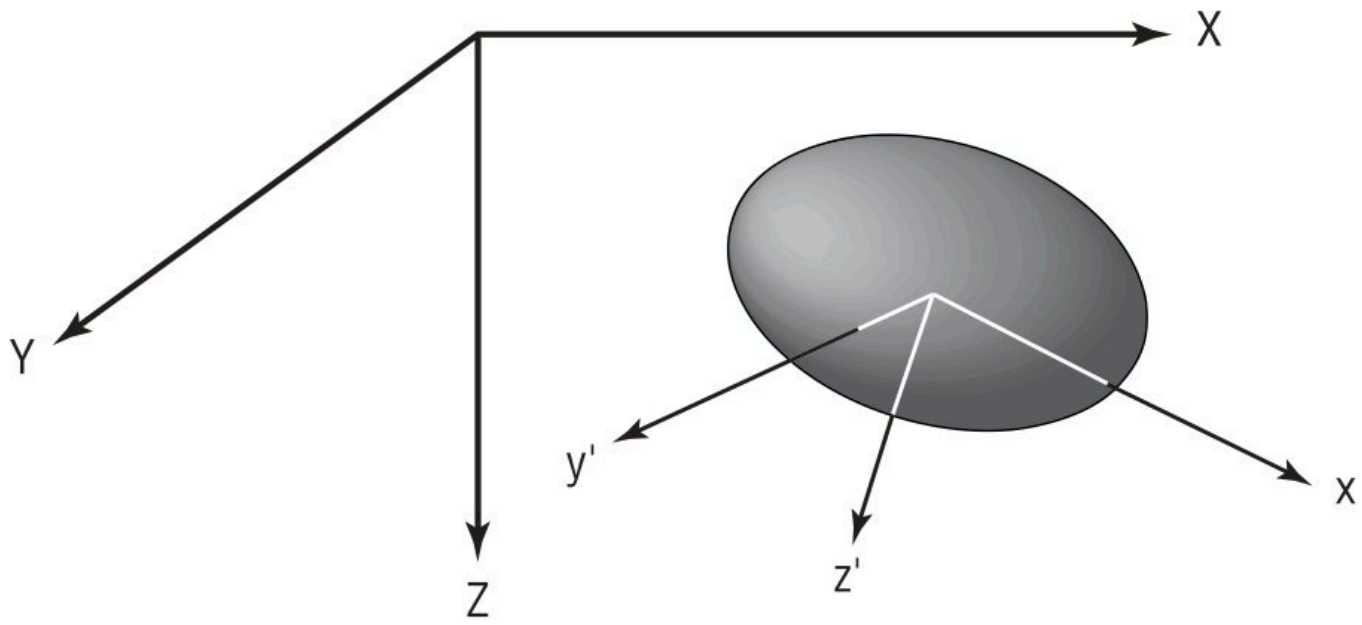


Figure 20-10 — Sphere geometrical properties.

Creating a New Sphere

To create a new sphere, perform the following steps:

1. Choose the Sphere Object tool.
2. Enter the X,Y,Z coordinates where you wish the sphere to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the center of the sphere.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the fields in the Sphere Editor dialog and press Enter.
5. Enter the orientation angles about the X,Y,Z axes.
6. **Using the Mouse:** Click on the 3-D Edit menu and choose Manipulators. A cascade menu displays several manipulator types. Choose the appropriate manipulator (see Chapter 21, Manipulators). Then click and drag the manipulator handles and position the sphere as desired.
7. **Using the Keyboard:** Enter the desired angles for the sphere in the orientation fields in the Sphere Editor dialog and press Enter.

After the sphere is positioned, the 3-D Editor will return to Edit mode. The sphere becomes the current object and its materials and other properties may be edited.

Editing an Existing Sphere

To edit an existing sphere object, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing sphere object to select it for editing (the Attributes Tool will confirm the current object name is Sphere). The Sphere Editor dialog is displayed.

> **NOTE:** If you click on a Sphere and the current object name is Group or a user-entered name, the Sphere you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous section.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The Sphere object remains selected for additional editing operations.

Text Tool

The Text Object Tool allows the user to create and edit 3-D text. Text may be used to label signs, buildings and other objects. The Text Editor dialog, shown in Figure 20-11, displays information about the current text object.

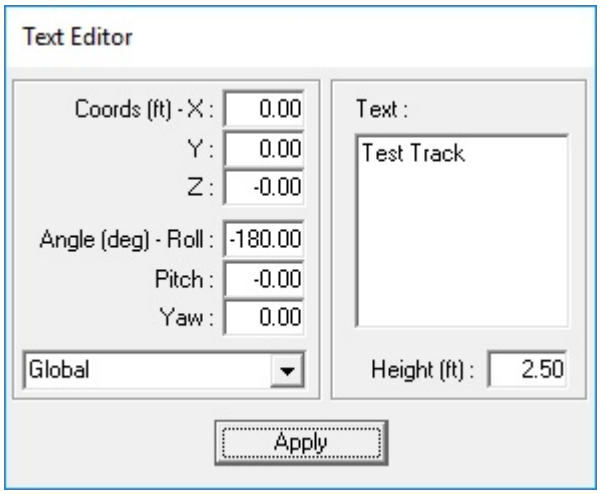


Figure 20-11 — The Text Editor dialog allows the user to create and place text strings in the scene.

Text Editor Dialog

See the Text Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the selected object's local or global coordinates.
- **Roll, Pitch, Yaw Orientation Fields** — Display and allow editing of the selected object's local or global orientation.
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Height** — Allows editing the text height.
- **Text** — Allows the user to enter or edit the desired text string.

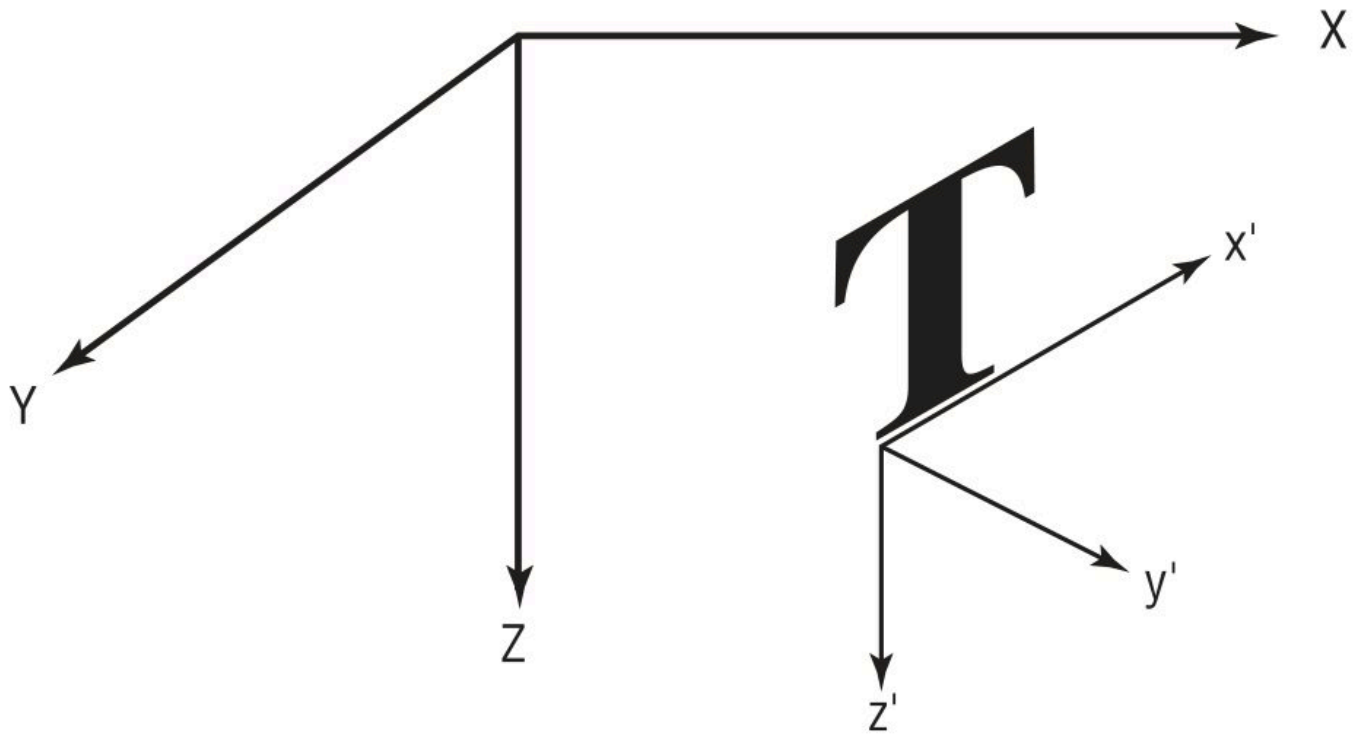


Figure 20-12 — Text properties. Text strings are used for road signs and other labels.

Creating a New Text String

To create a new text string, perform the following steps:

1. Choose the Text Object tool.
2. Enter the X,Y,Z coordinates where you wish the text to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the base of the text.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the fields in the Text Editor dialog and press Enter.
5. Enter the orientation angles about the X,Y,Z axes.
6. **Using the Mouse:** Click on the 3-D Edit menu and choose Manipulators. A cascade menu displays several manipulator types. Choose the appropriate manipulator (see Chapter 21, Manipulators). Then click and drag the manipulator handles and position the text as desired.
7. **Using the Keyboard:** Enter the desired angles for the text string in the orientation fields in the Text Editor dialog and press Enter.

After the text string is positioned, the 3-D Editor will return to Edit mode. The text string becomes the current object and its materials and other properties may be edited.

Editing an Existing Text String

To edit an existing text string, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing text string to select it for editing (the Attributes Tool will confirm the current object name is Text). The Text Editor dialog is displayed.

> **NOTE:** Text is frequently grouped with other objects, such as signs.
>
> **NOTE:** If you click on a Text string and the current object name is Group or a user-entered name, the Text object you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous section.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The Text object remains selected for additional editing operations.

Light Tool

The Light Object Tool allows the user to create and edit light sources. Lights are objects that cast light on other objects, making them visible. An example is shown in Figure 20-13.

Lights are unique in several ways:

- You cannot see them, only the effects caused by them (i.e., other objects are lit up).
- Lights cannot be grouped like other objects can. This means you cannot create a streetlight that includes a light source that moves when you move the streetlight; instead, you must re-attach the light after you have positioned the pole.

HVE supports three types of lights:

- **Directional** — The light source illuminates uniformly along a specified direction. Its location is not used. An example is the sun.
- **Point** — The light source is located at user-specified coordinates and radiates light in all directions. An example is a light bulb.
- **Spot** — The light source is located at user-specified coordinates and radiates a beam of light. An example is a headlight on a car.

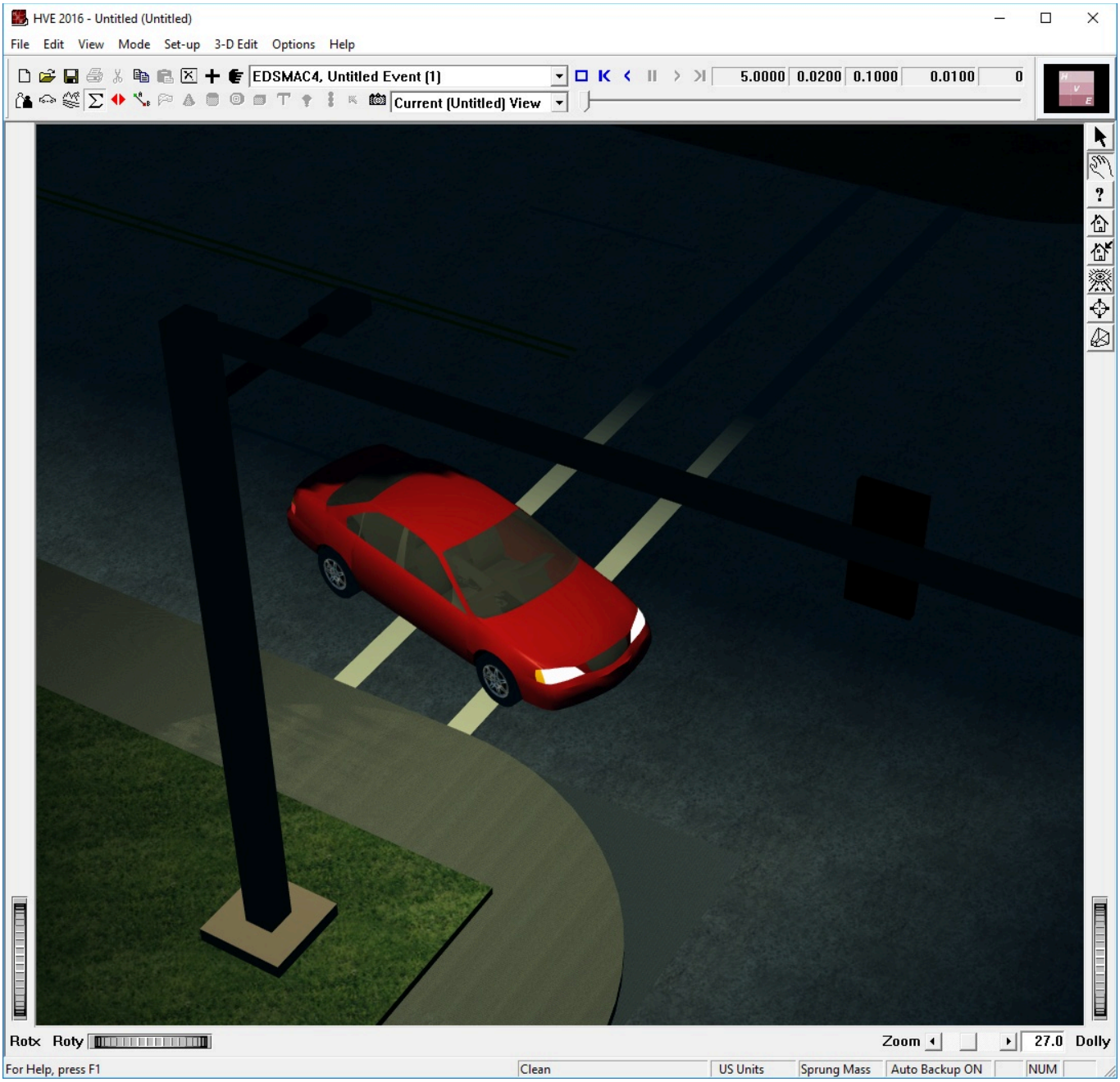


Figure 20-13 — An example of a light source; in this case, a street light illuminating a vehicle traveling beneath the light.

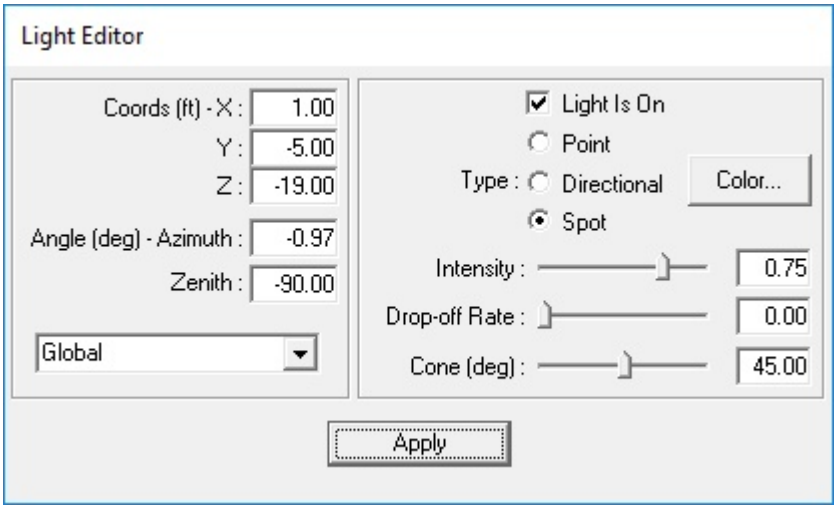


Figure 20-14 — The Light Editor dialog allows the user to create light sources and place them in the scene.

Light Editor Dialog

See the Light Editor Dialog reference. In summary, the dialog provides:

- **X,Y,Z Position Fields** — Display and allow editing of the light's coordinates. Used for Point and Spot lights; for Directional lights the fields remain enabled but the position values are simply ignored (a directional light illuminates from a direction, not a location).
- **Azimuth, Zenith Angle Fields** — The direction in which the light points, entered as an azimuth (heading) angle and a zenith (angle from vertical). These fields are used for Directional and Spot lights. *(updated: earlier versions of this dialog used Roll/Pitch/Yaw orientation fields; the current dialog uses Azimuth/Zenith angles.)*
- **Local/Global Option Switch** — Allows switching between local and global coordinates and angles.
- **Color pushbutton** — Displays the Color dialog, used to edit the light color.
- **Type** — Selects from the three available light types described earlier (Point, Directional and Spot).
- **Intensity** — The brightness of the light.
- **Drop-off Rate** — For Spot lights only, the light attenuation rate from the center of the beam to its edge.
- **Cone (deg)** — For Spot lights only, the included angle of the cone of light. *(updated: formerly labeled "Cut-off Angle".)*
- **Light Is On** — Check box that activates and deactivates the light source.

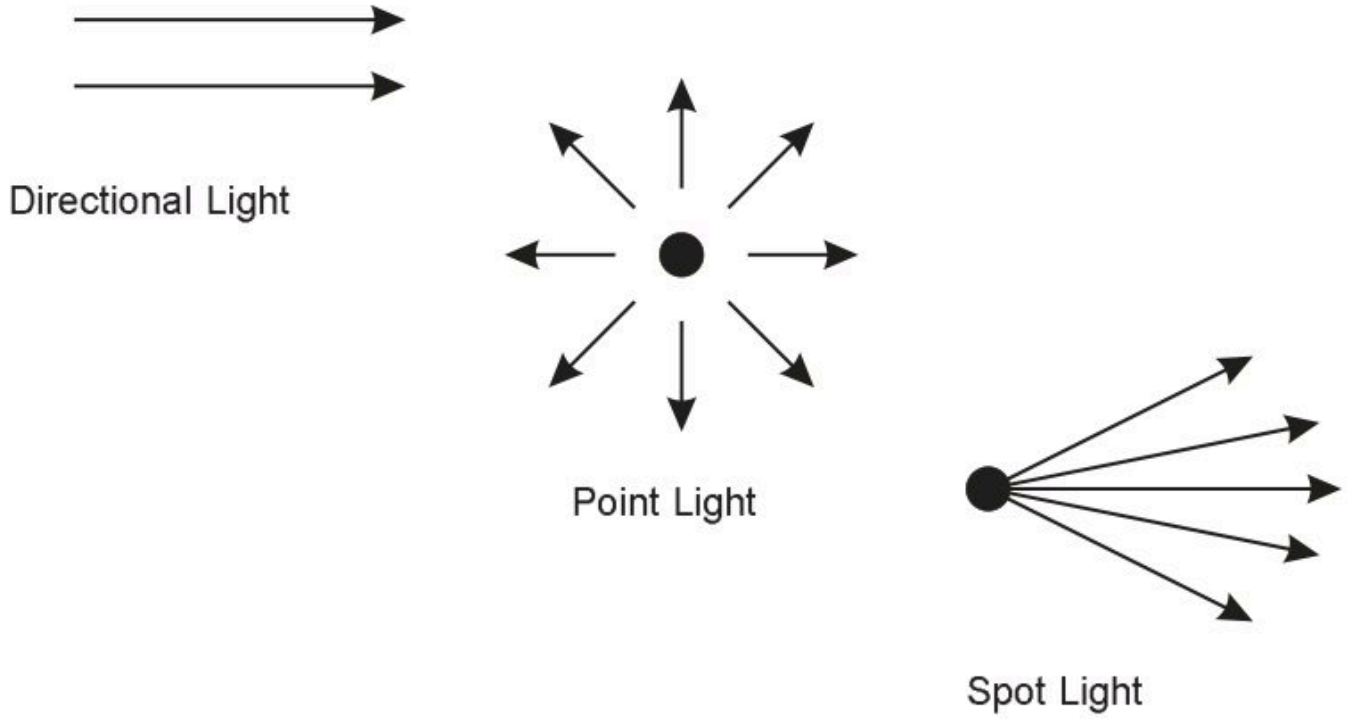


Figure 20-15 — Illustration of the light types: Directional Light (top), Point Light (middle) and Spot Light (bottom).

Creating a New Light

To create a new light, perform the following steps:

1. Click on the Light Object tool.
2. Enter the X,Y,Z coordinates where you wish the light to be placed.
3. **Using the Mouse:** Click the mouse at the desired X,Y,Z coordinates for the light.
4. **Using the Keyboard:** Enter the X,Y,Z coordinates in the fields in the Light Editor dialog and press Enter.
5. Enter the light direction.
6. **Using the Mouse:** Click and drag the manipulator handles and orient the light as desired.
7. **Using the Keyboard:** Enter the desired light direction using the Azimuth and Zenith fields in the Light Editor dialog and press Enter. *(updated.)*

After the light is positioned, the 3-D Editor will return to Edit mode and a manipulator is displayed at the selected location (remember, you cannot see the light, only its effects on the scene; therefore the manipulator is required to tell you the light exists and where it is located). The light becomes the current object and its attributes may be edited.

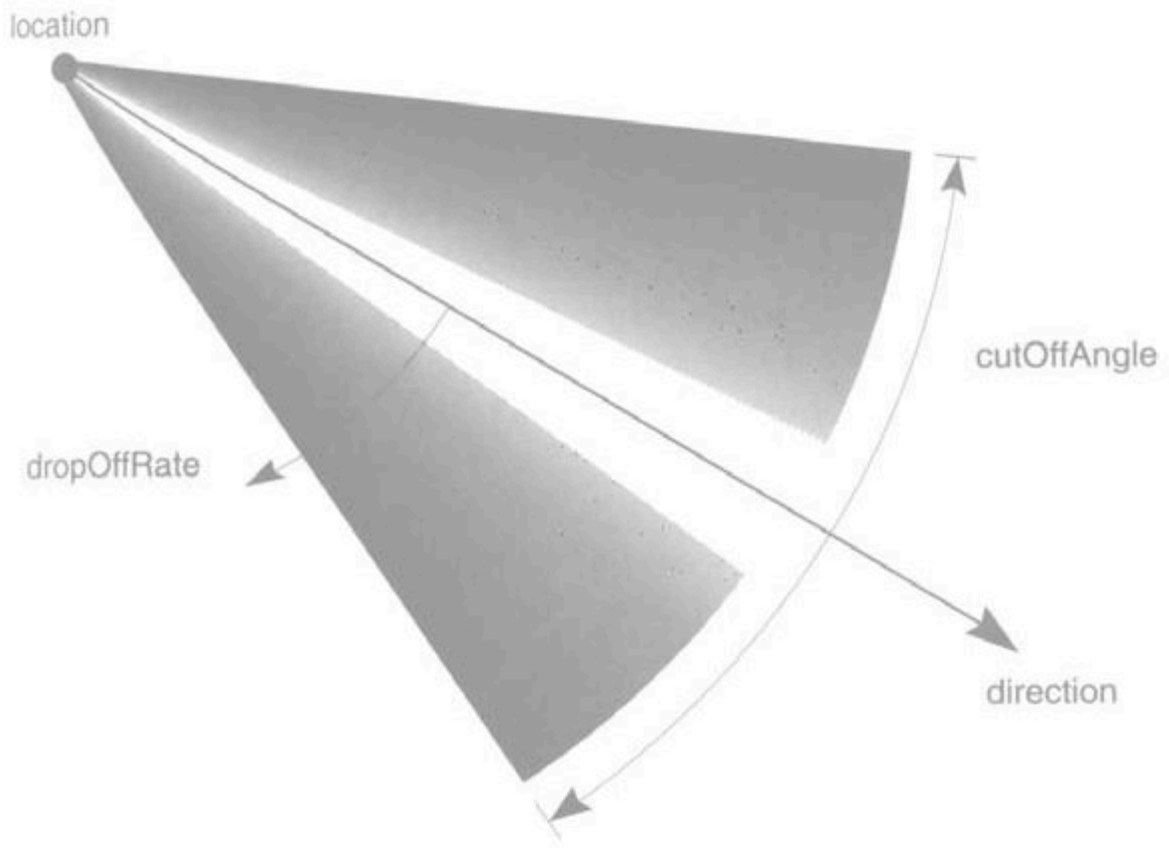


Figure 20-16 — Spot Light attributes: Drop-off Rate and Cone (Cut-off) Angle.

Editing an Existing Light

To edit an existing light, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on an existing light to select it for editing (the Attributes Tool will confirm the current object name is Light). The Light Editor dialog is displayed.
3. Perform the desired editing operations:
4. Edit its position and direction using either the mouse or the keyboard, as described in the previous section.
5. Edit its Type, Color and/or Intensity attributes using the Light Editor dialog.
6. If the light type is Spot, edit its Drop-off Rate and/or Cone angle using the Light Editor dialog.
7. Alternatively, if the light is a Spot light, click and drag the cone manipulator to edit the cone angle.

> **NOTE:** Editing of light attributes instantly updates the light properties.

Edit Tool

The Edit Tool allows the user to deselect any currently selected object and select a different object for editing. The Edit tool is also automatically selected after each object is created to allow for its immediate editing.

To edit an existing object, perform the following steps:

1. Choose the Edit tool (if necessary).
2. Click on the desired object to select it for editing (the Attributes Tool will confirm the correct object has been selected; for example, if you attempt to click a box and the current object name is Text, you have selected the wrong object, so try again). The Object Editor dialog for the selected object is also displayed.

> **NOTE:** If you click on an object and the current object name is Group or a user-entered name, the object you clicked on is part of a grouped object that must be disassembled before it can be edited.

1. Perform the desired editing operations:
2. Edit its position and orientation using either the mouse or the keyboard, as described in the previous sections.
3. Edit its Object Type, Overlay and Friction Factor attributes using the Attributes Tool.
4. Edit its materials attributes by choosing Material Color or Material Texture from the 3-D Edit menu.

> **NOTE:** The object remains selected for additional editing operations.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 20; revised to match the current version of HVE, 2026-07-05.

Chapter 21 — Manipulators

Overview

A selected object may be manipulated, that is, it may be positioned and oriented relative to other objects. A manipulator is a tool used for visually positioning an object. Manipulators have handles, which are the places the user clicks on in order to modify the selected object.

The types of handles and resulting object modifications depend on the current manipulator. In general, three types of modifications are possible:

- **Translation** — The entire object is translated, that is, moved in the X, Y and/or Z direction(s). Most translation manipulators allow translation in one or two directions. Three-dimensional translation manipulators are not used by HVE.

> **NOTE:** To move an object in three dimensions, use two different viewers. For example, use the X-Y viewer to move the object to the desired X,Y coordinate, then use the X-Z viewer to move it in the Z direction.
>
> **NOTE:** 3-dimensional manipulation in the perspective viewer is not recommended.

- **Rotation** — The entire object is rotated, normally about a single axis of rotation. The specific rotation axis depends on the manipulator.
- **Scale** — The object is scaled (stretched or shortened) in a selected direction.

The specific manipulators available in the 3-D Editor are as follows (see also the 3-D Editor Menu reference):

- **Direct** — No manipulator is used. Instead, the user clicks directly on the object and moves it.
- **Trackball** — 3-D rotation of the selected object; no translation.
- **HandleBox** — 3-D translation and scaling; no rotation.
- **Jack** — Scales the selected object in all directions. *(updated: the current 3-D Edit menu documentation describes the Jack manipulator as scale-only — no translation or rotation; the legacy manual additionally described 3-D translation and rotation via its axes, which remains available through the Jack's axis handles in the viewer.)*
- **Centerball** — Translates the object center to new X,Y,Z coordinates; rotates the object about the new center.
- **TransformBox** — Scales the object in all directions; translates and rotates the object in the selected direction.
- **TabBox** — Translates and scales the object in the selected plane.

To use a manipulator, perform the following steps:

1. If necessary, start the 3-D Editor with an object (or add a new object).
2. Click on an object (Surface, Box, Cylinder, etc.) to select it. A red bounding box will be displayed around the object to indicate it is selected.
3. Choose Manipulators from the 3-D Edit menu. A cascade menu displays the list of available manipulators.
4. Choose a manipulator (e.g., Trackball) from the list. The selected manipulator will be displayed about the selected object.

> **NOTE:** The Direct manipulator does not display any handles or pick points; the object itself is used.

1. Manipulate the object using the methods available for the selected manipulator.

Each manipulator performs a specific function or set of functions (translation, rotation and scaling). Each manipulator is described below.

Direct Manipulator

The Direct manipulator provides 2-dimensional translation in the viewer plane. The Direct manipulator is shown in Figure 21-1.

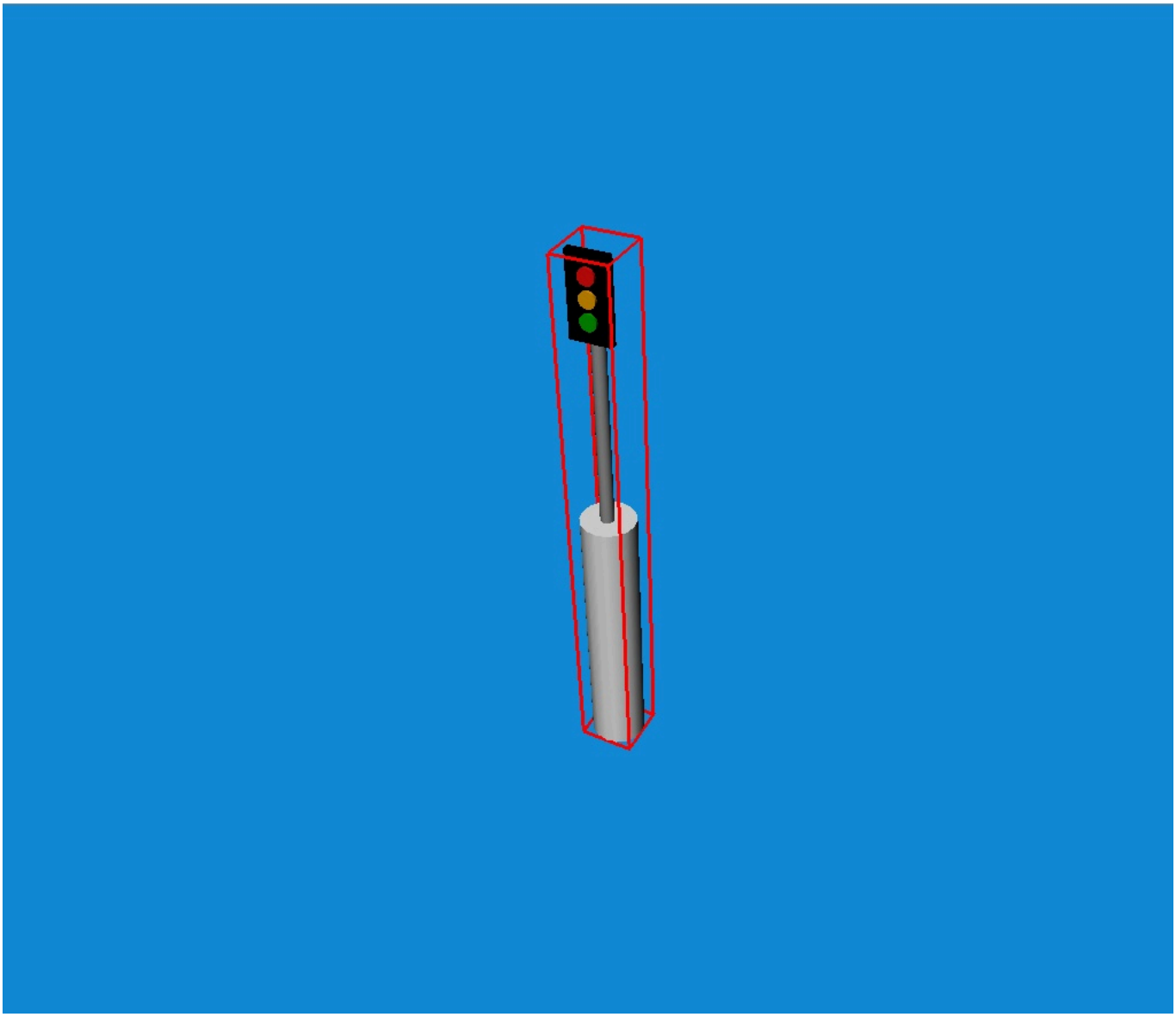


Figure 21-1 — Direct Manipulator.

To use the Direct manipulator, perform the following steps:

1. Click on the object to select it (if necessary). A red bounding box will be displayed around the entire object.
2. Click anywhere on the object and drag the object to a new location. The object is dragged in the viewer plane.

> **NOTE:** Dragging an object in the perspective viewer often yields unpredictable results because the viewer plane is not explicitly defined by the user.

1. When the object is positioned correctly, release the mouse button.

The object position is updated.

Trackball Manipulator

The Trackball manipulator provides 3-dimensional rotation of the selected object. The Trackball manipulator is shown in Figure 21-2.

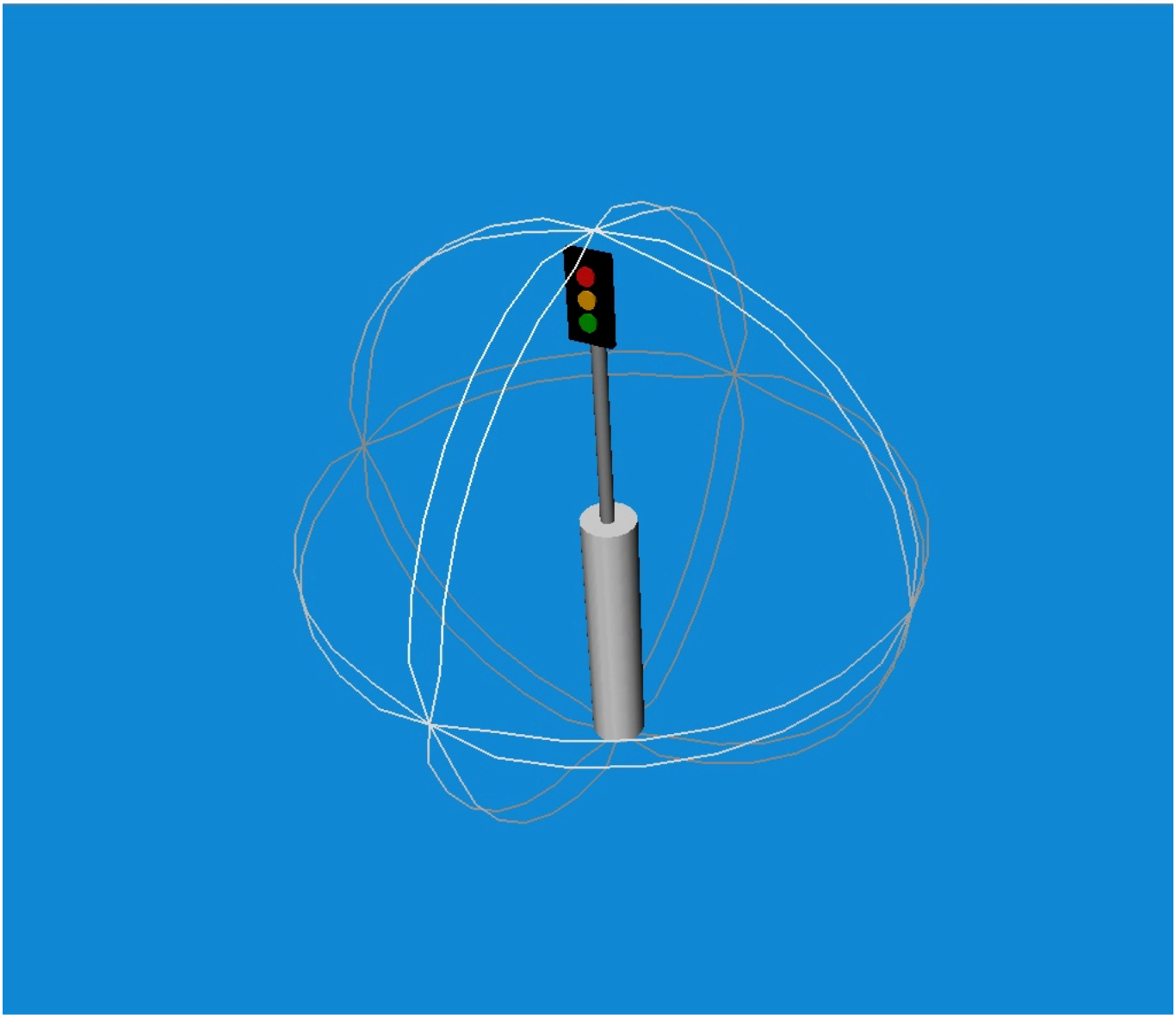


Figure 21-2 — Trackball Manipulator.

To use the Trackball manipulator, perform the following steps:

1. Click on the object to select it (if necessary). Three rotation bands will be displayed around the entire object.

2. Click on any of the bands and drag in a tangential direction. The object rotates about an axis normal to the selected band.
3. Click at the intersection of two bands for general rotation.
4. When the object is positioned correctly, release the mouse button.
5. Click and drag on another band to rotate the object about a different axis.

The object orientation is updated.

HandleBox Manipulator

The HandleBox manipulator provides 3-dimensional translation and scaling of the selected object. The HandleBox manipulator is shown in Figure 21-3.

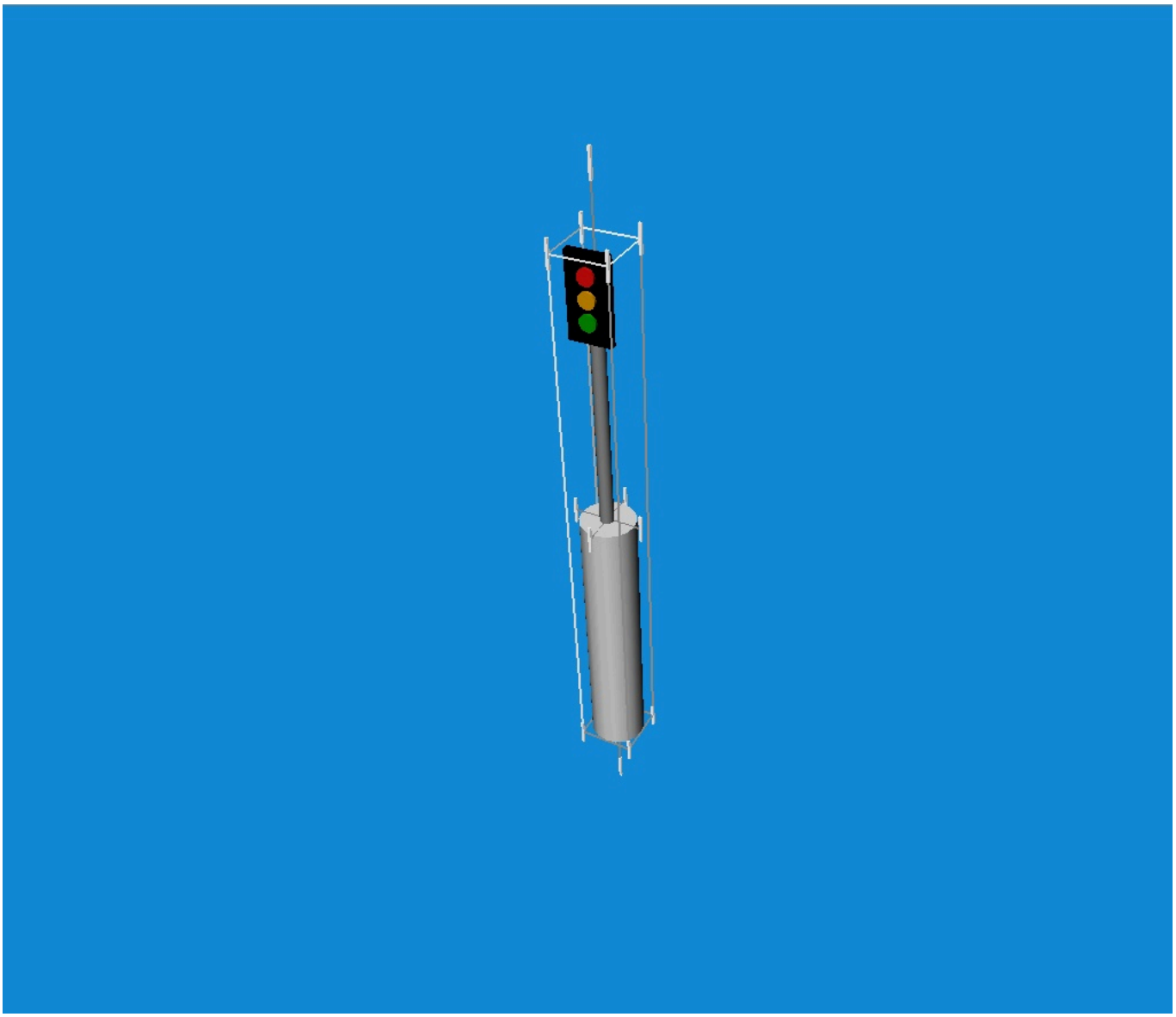


Figure 21-3 — HandleBox Manipulator.

To use the HandleBox manipulator, perform the following steps:

1. Click on the object to select it (if necessary). A 3-dimensional bounding box with several pick points (also called handles) will be displayed around the entire object.
2. Click and drag any of the handles to stretch the object in that direction. When the object is scaled correctly, release the mouse button.

The object dimensions are updated.

Jack Manipulator

The Jack manipulator provides scaling of the selected object, along with translation and rotation using its axes. The Jack manipulator is shown in Figure 21-4.

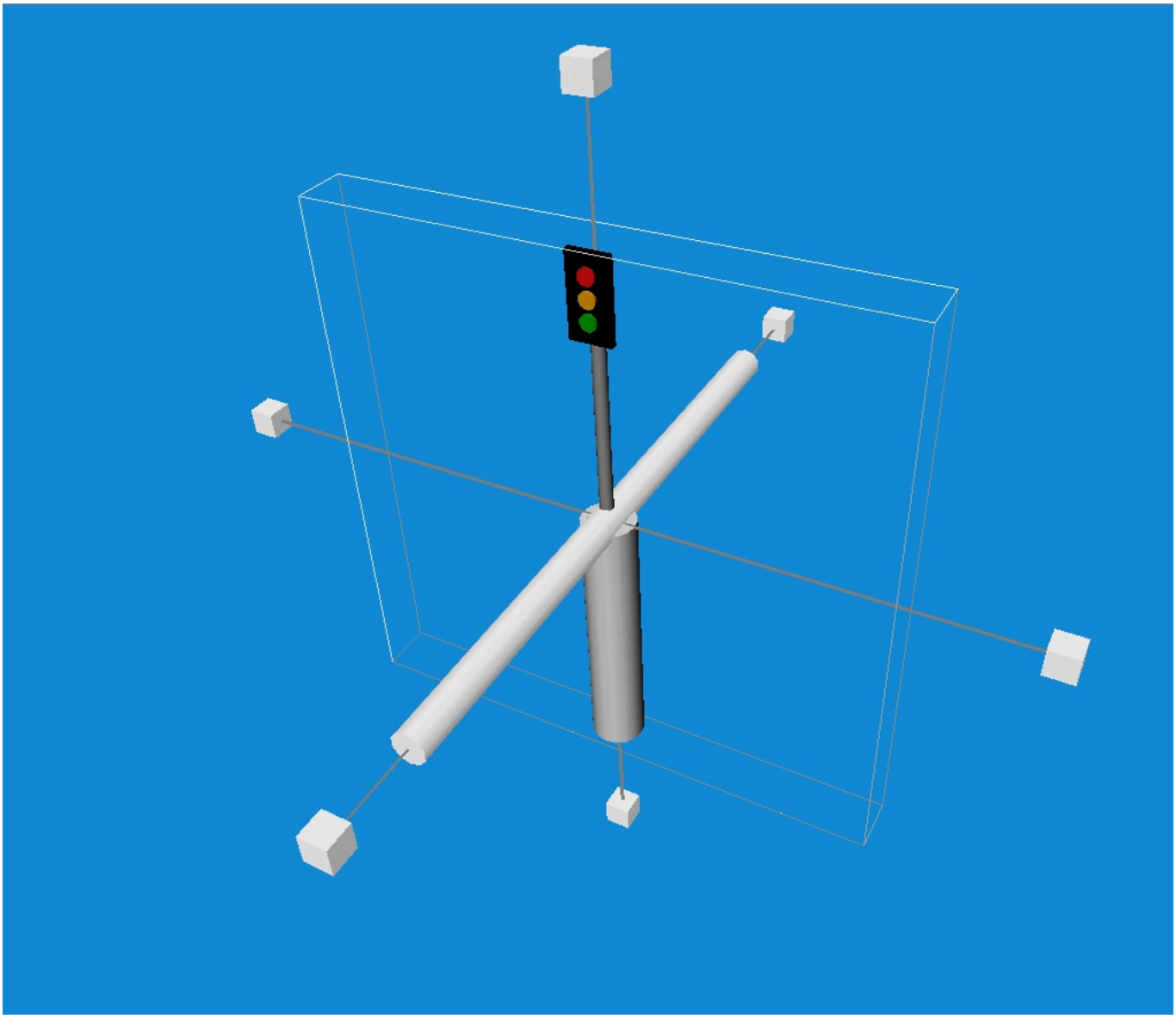


Figure 21-4 — Jack Manipulator.

To use the Jack manipulator, perform the following steps:

1. Click on the object to select it (if necessary). A set of orthogonal axes (note that one axis has a thick portion) will be displayed with the object at its origin. Pick points (handles) are displayed at each end of the axes.
2. Click and drag any of the handles to stretch (scale) the object in the direction of the axis.
3. Click and drag any of the thin axes to rotate about either of the other axes.
4. Click and drag on the thick portion of the axis to translate the object in the direction of the axis. Release the mouse button when the object has been correctly positioned and scaled.

The object position and scale are updated.

Centerball Manipulator

The Centerball manipulator provides 3-dimensional rotation of the selected object about a user-defined origin. The Centerball manipulator is shown in Figure 21-5.

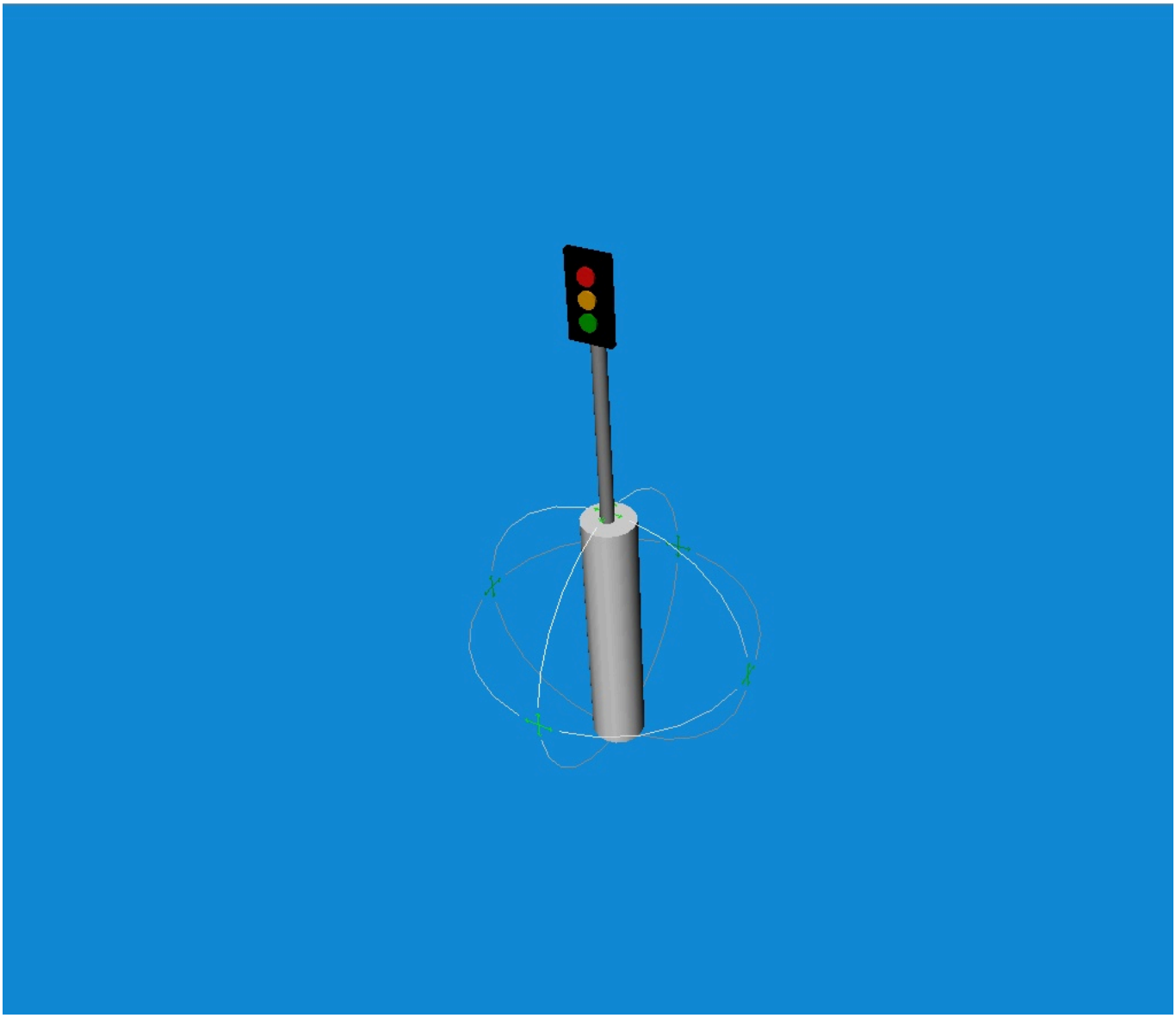


Figure 21-5 — Centerball Manipulator.

To use the Centerball manipulator, perform the following steps:

1. Click on the object to select it (if necessary). Three rotation bands are displayed with the object at its origin. A set of arrows is displayed where the bands intersect.
2. Click and drag any of the arrows to translate the origin of the object to a new location.
3. Click and drag any of the bands to rotate the object about an axis normal to the band. The object rotates about the new origin. When the object is positioned correctly, release the mouse button.

The object position and orientation are updated.

TransformBox Manipulator

The TransformBox manipulator provides 3-dimensional translation, rotation and scaling of the selected object. The TransformBox manipulator is shown in Figure 21-6.

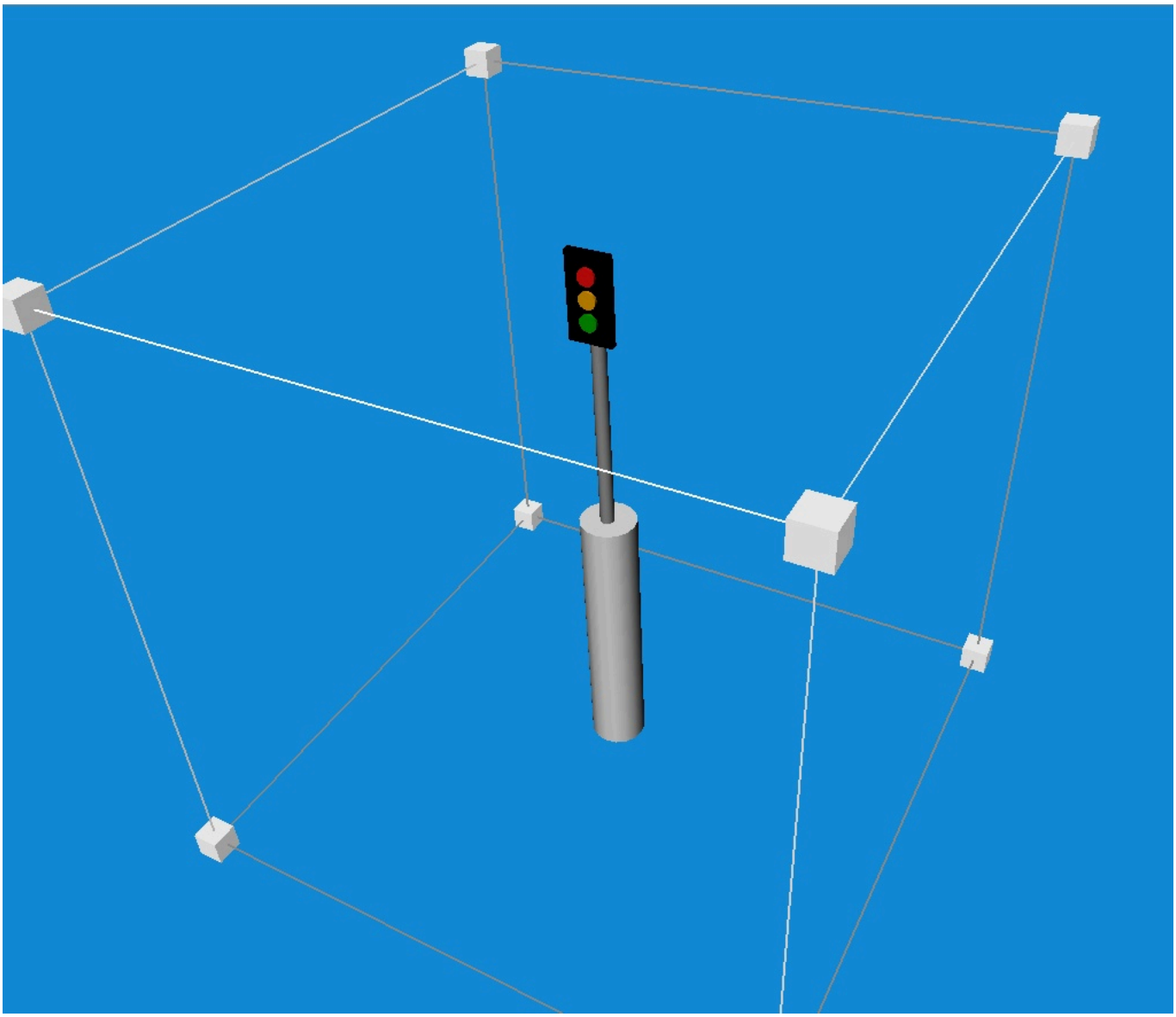


Figure 21-6 — TransformBox Manipulator.

To use the TransformBox manipulator, perform the following steps:

1. Click on the object to select it (if necessary). A bounding box with handles will surround the object.
2. Click and drag any of the handles to scale the entire object uniformly in all directions.
3. Click and drag any of the edges of the bounding box to rotate the object about an axis parallel to the selected edge.
4. Click and drag any of the sides of the box to translate the object in a plane parallel to the selected side. When the object is positioned correctly, release the mouse button.

The object position, orientation and scale are updated.

TabBox Manipulator

The TabBox manipulator provides 3-dimensional translation and scaling in the selected plane. The TabBox manipulator is shown in Figure 21-7.

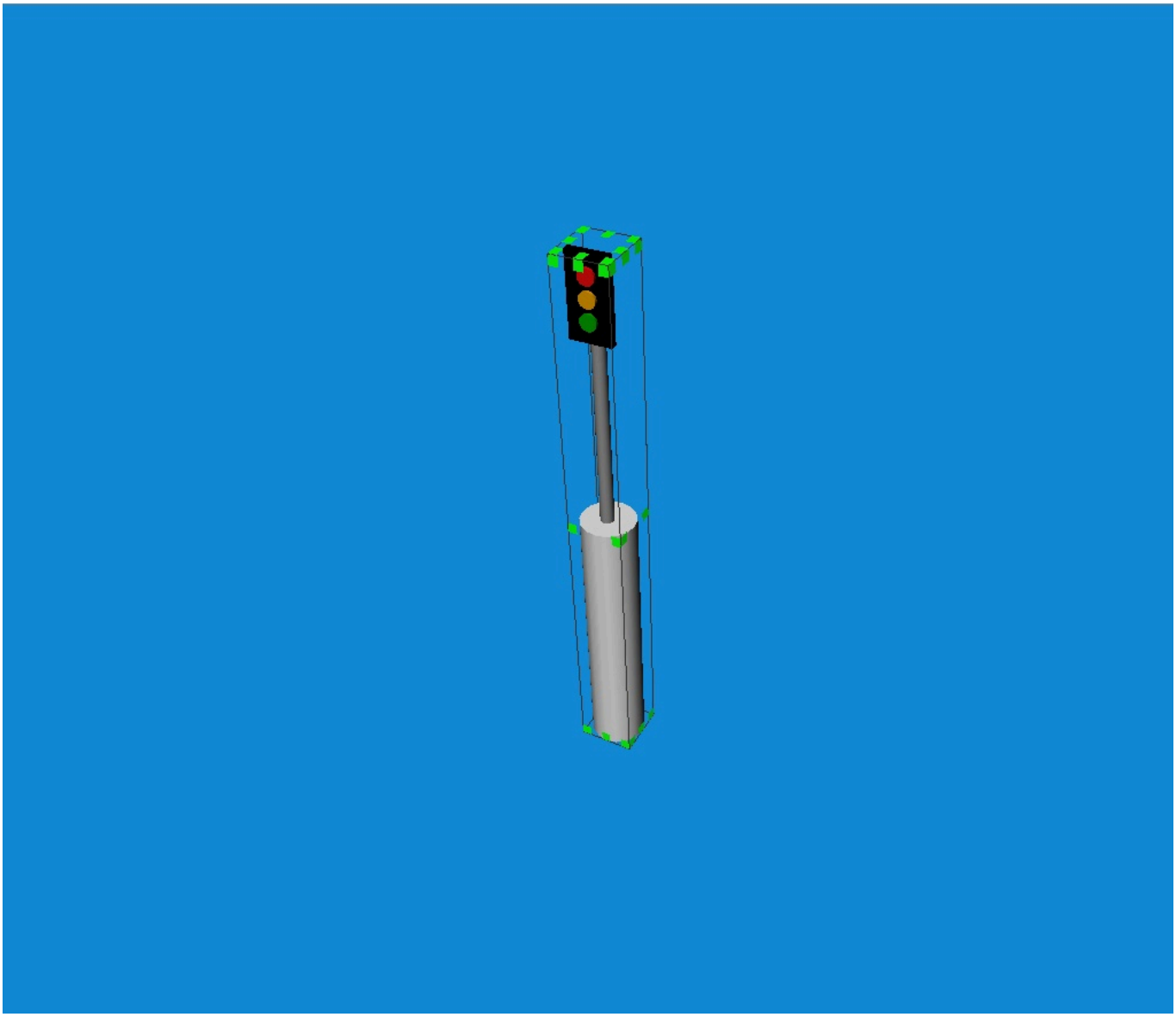


Figure 21-7 — TabBox Manipulator.

To use the TabBox manipulator, perform the following steps:

1. Click on the object to select it (if necessary). A bounding box will be displayed around the object with tabs at the corners and middle.
2. Click on any of the tabs to scale in one, two or three directions, depending on the number of sides on the selected tab.

> **NOTE:** A corner tab has three sides and, therefore, scales in three directions. An edge tab has two sides and scales in two directions. A side tab has one side and scales in one direction.

1. When the object is scaled correctly, release the mouse button.

The object position and scale are updated.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 21; revised to match the current version of HVE, 2026-07-05.

Chapter 23 — Basic 3-D Editor Tutorial

Overview

This tutorial contains three basic lessons that describe how to use the HVE 3-D Editor:

- Lesson 1 — Creating a Simple Road Surface
- Lesson 2 — Editing a Surface
- Lesson 3 — Using Groups

> **NOTE:** The Advanced Tutorial chapter referenced by the legacy manual (Chapter 24) is reserved for future use.

Even though HVE's interface uses standard techniques for input and editing, all graphics editors are somewhat different, and HVE's 3-D Editor is no exception. Don't be discouraged if you don't understand something right away. The HVE 3-D Editor is a well-designed program and everything happens for a reason. As these reasons become clear to you, you will learn how things work and predict how something is going to work before you try it; it simply takes a little practice.

> **NOTE:** We assume that HVE is up and running and that you are familiar with HVE's basic features, such as using HVE's dialogs and 3-D viewers, as well as the HVE Human, Vehicle, Environment, Event and Playback Editors. The purpose of this tutorial is to use those basic features to learn how to use the HVE 3-D Editor. For a refresher on these topics, the user is referred to the Using HVE section of this manual.

Lesson 1 — Creating a Simple Road Surface

This lesson is designed to make you generally familiar with how the 3-D Editor works. In this lesson, you will learn the following:

- Starting the 3-D Editor
- Creating a Simple Road Surface
- Assigning friction characteristics to the surface
- Assigning material and texture attributes to the surface
- Assigning overlay names
- Saving the surface in the case file
- Saving the surface as a unique file

> **NOTE:** We assume HVE is already running and you're ready to begin a new case. If necessary, refer to Chapter 2, Using HVE, for information about starting HVE. Also, check out Chapter 32, HVE Tutorial.

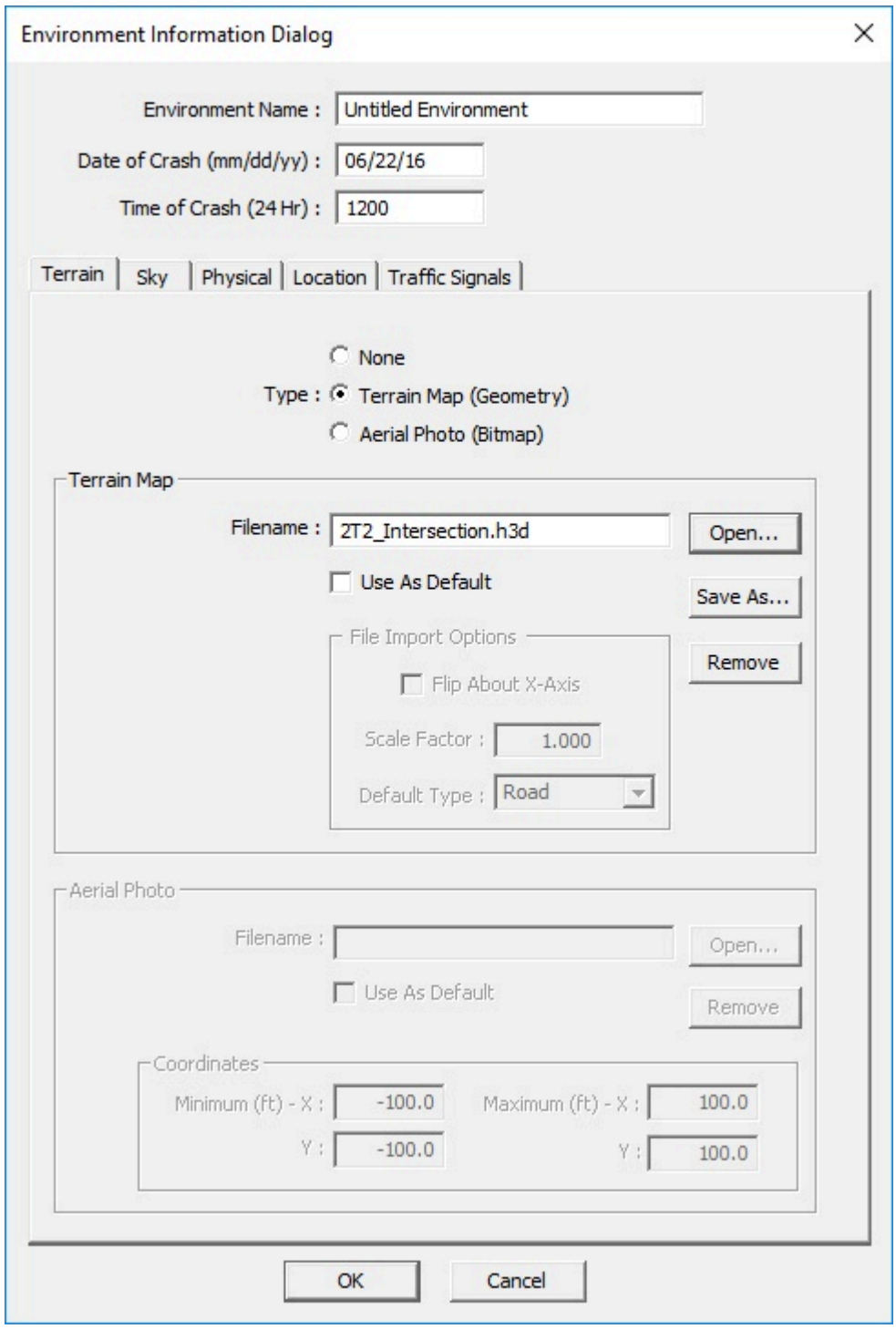


Figure 23-1 — Environment Information dialog, used to assign physical environment parameters.

Starting the 3-D Editor

To start the 3-D Editor, perform the following steps:

1. Choose Environment mode. The Environment Editor is displayed.
2. Click on Add New Object. The Environment Information dialog is displayed (see Figure 23-1, and the Environment Information reference), allowing you to assign basic environment attributes. Because these attributes are not

relevant to building our 3-D environment, let's accept the current values.

3. Press OK. The default environment information is assigned and the Environment Viewer and Editor are displayed, as shown in Figure 23-2.

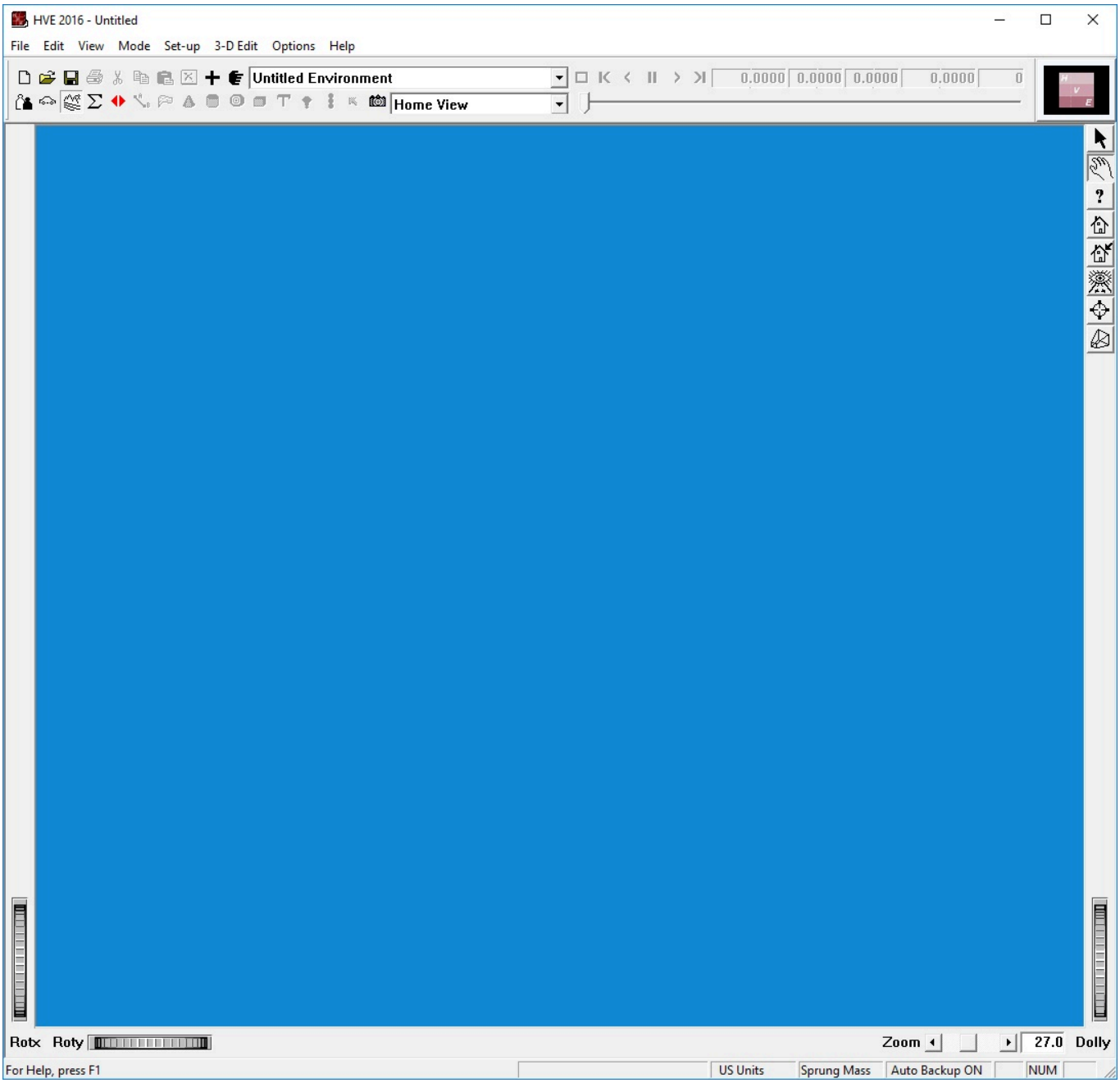


Figure 23-2 — Environment Editor and 3-D Viewer. No 3-D geometry has been created yet, so the 3-D viewer is empty.

At this point, the 3-D Viewer is empty because no geometry file was specified. In our first tutorial, we'll create one.

To create a geometry file, we need to start the HVE 3-D Editor:

1. Select 3-D Edit on the main menu and then select Launch 3-D Editor on the cascade menu *(or press Ctrl+L)*.

The 3-D Editor is displayed. Its basic components are the four viewers (X-Y, X-Z, Y-Z and Perspective), the 3-D Editor dialog and the Current Object Tool dialog (when first started, the Edit Tool is displayed). These components are shown in

Figure 23-3.

(updated: you can also choose 3-D Edit, Viewer to display the Viewer Options dialog and select which viewer layout is shown — a single XZ, YZ, XY or Perspective viewer, or All four at once.)

Typically, you will want to arrange these components to suit your personal needs. To move a dialog or viewer, click on its title bar and drag it to the desired location.

> **NOTE:** Because most of your work will probably be done using the X-Y viewer (lower left) and Perspective viewer (lower right), you may find it convenient to keep the lower area of the screen clear. Therefore, you should position the Object Attributes dialog near the top of the screen.

We're now ready to begin working on our single-surface environment.

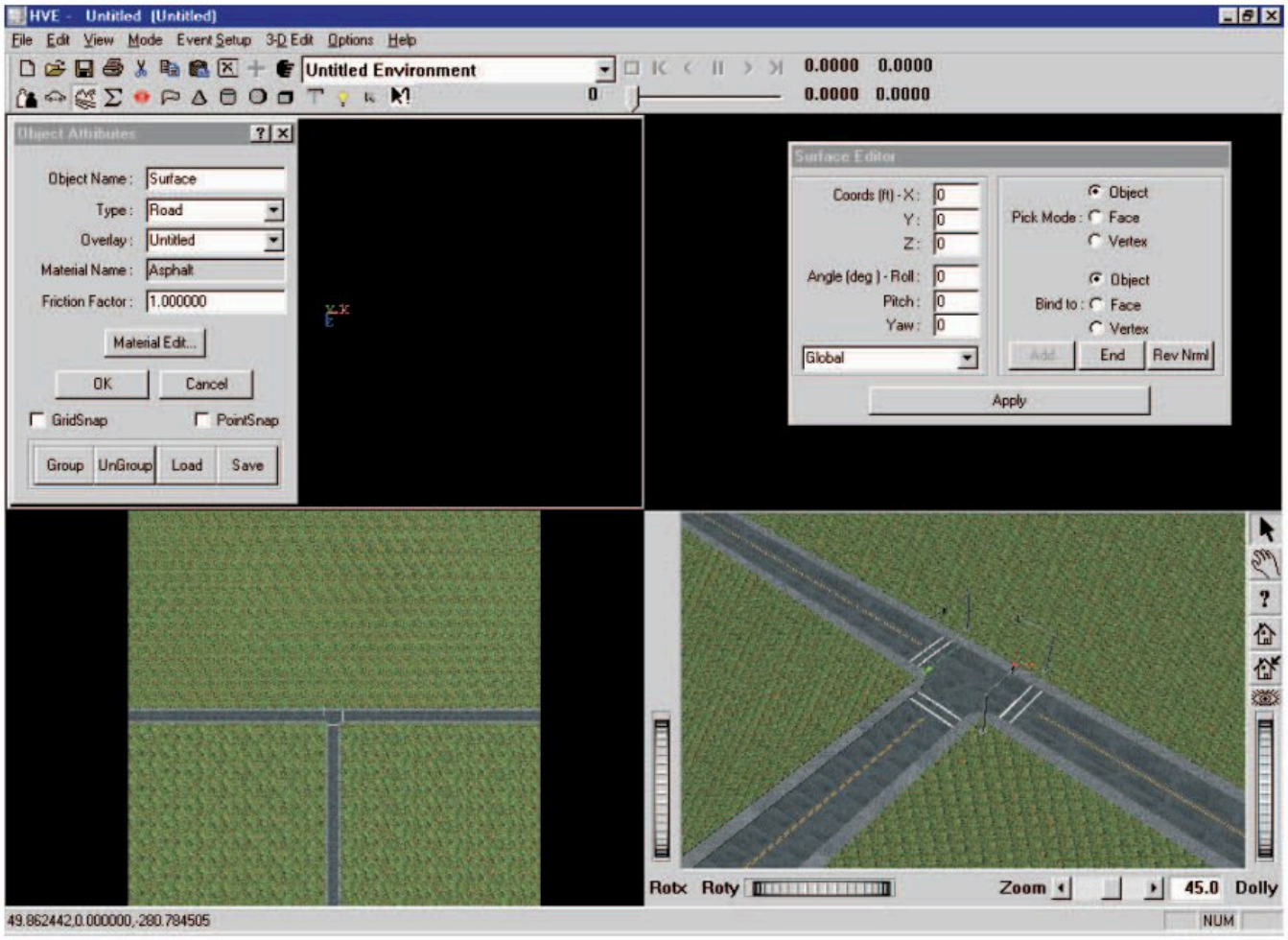


Figure 23-3a — 3-D Editor, including 4 viewers (X-Y, X-Z, Y-Z and Perspective).

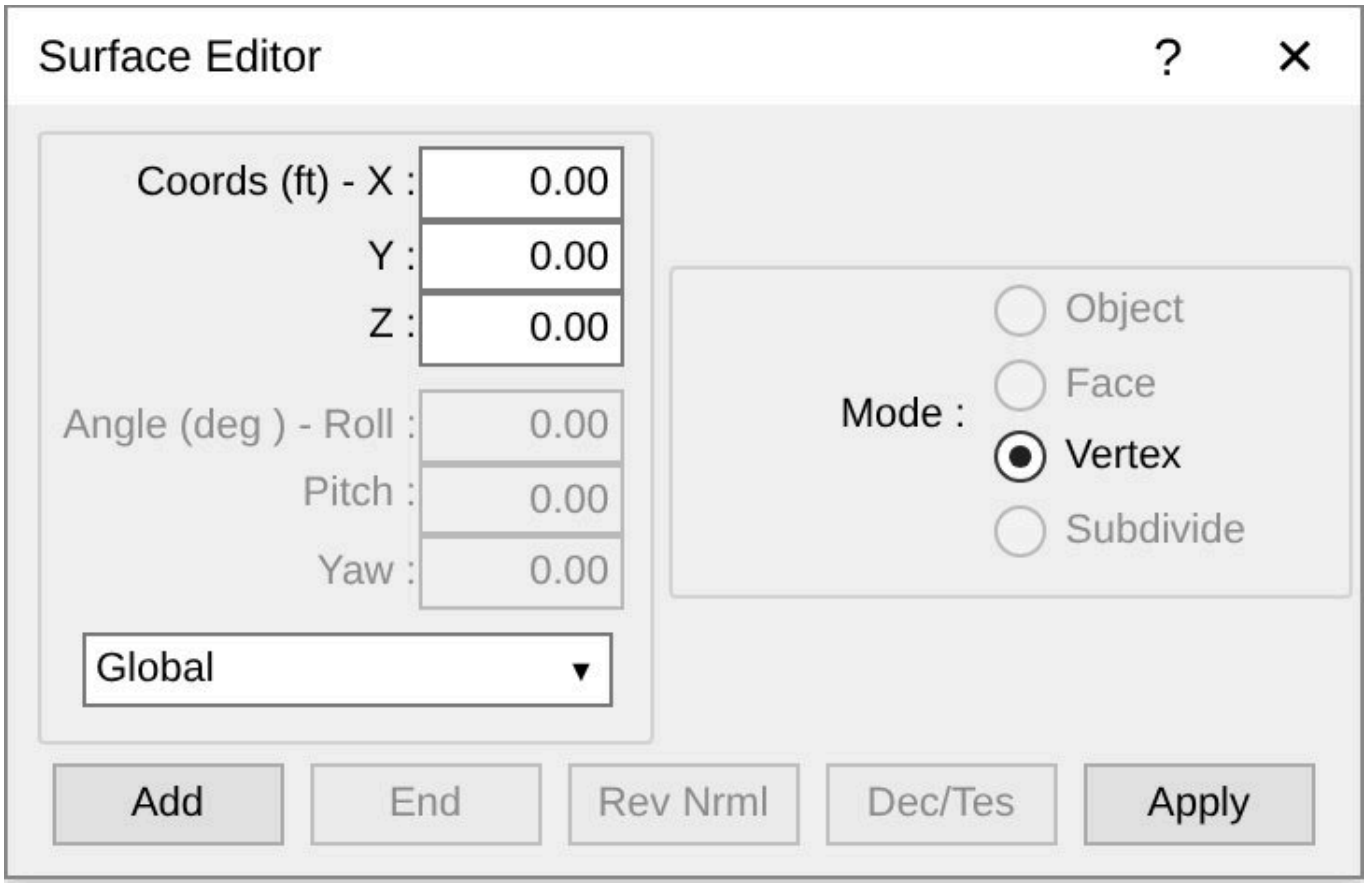


Figure 23-3b — 3-D Editor, Current Object Editor.

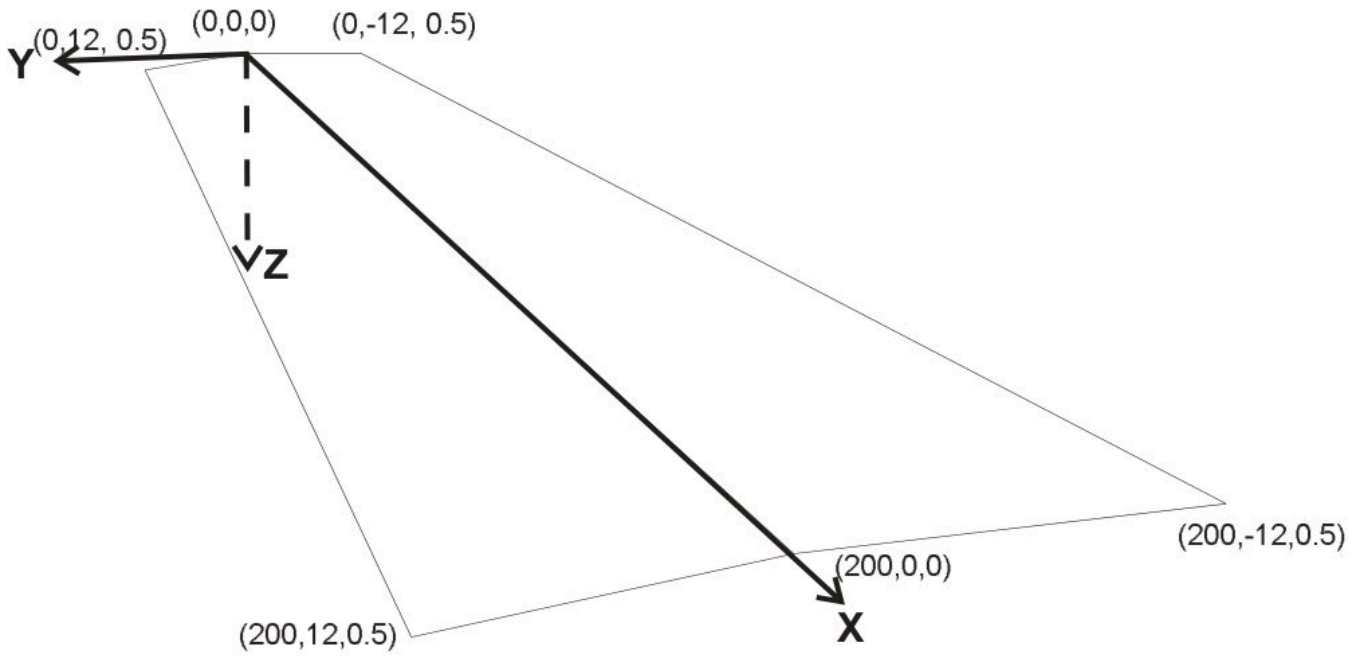


Figure 23-4 — Line drawing showing the crowned road surface we wish to create using the 3-D Editor. Each lane drops 0.5 feet over its 12-foot width. Vertex coordinates: (0,0,0), (0,12,0.5), (200,12,0.5), (200,0,0) for one lane, and (0,-12,0.5), (200,-12,0.5) for the other.

Creating a Simple Road Surface

The diagram of the road surface we wish to create is shown in Figure 23-4. It consists of two individual surfaces, each with four vertices. Like most roads, the surface is crowned slightly. The coordinates are also shown in Figure 23-4.

> **NOTE:** At a minimum, you should always create a crowned road surface, if one exists, to take advantage of HVE's core 3-D functionality!

To create the surface, perform the following steps:

1. Click on the Surface Tool icon on the toolbar. The Surface Editor is now enabled (see Figure 23-5), ready to begin creating a surface.

The Surface Tool dialog includes X,Y,Z coordinate fields and Roll, Pitch, Yaw orientation fields. Beneath these on the left side is the Local/Global option switch. On the right side, the Surface Tool dialog includes the Pick Mode radio buttons and the Add, End, Rev Nrml and Dec/Tes buttons. See Chapter 20 and the Surface Editor reference for a detailed description of each of these features. *(updated: the "Bind To" radio buttons mentioned in the legacy manual are no longer part of the Surface Editor; a Subdivide pick mode and a Dec/Tes (Decimate/Tessellate) button have been added. The editor starts in Vertex mode, ready for vertex entry.)*

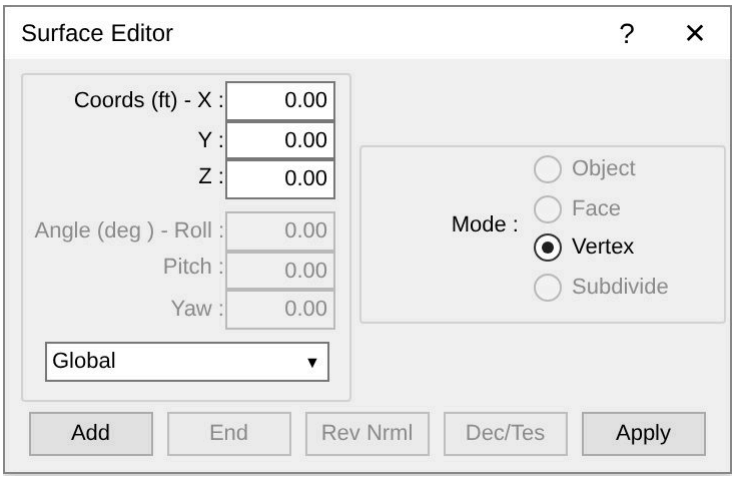


Figure 23-5 — Surface Tool dialog, used for entering and editing 3-D surfaces.

Let's create the surface. To enter the coordinates for the first surface, perform the following steps:

1. Place the cursor in the Surface Tool's X-coordinate field and press Enter.

Pressing Enter causes the X,Y,Z coordinates to be accepted, and the first vertex will appear in the viewers at 0,0,0. A small, bright marker is displayed at the location of the first vertex.

> **NOTE:** The X,Y,Z coordinates default to 0,0,0. Because these are the desired values, they need not be entered. Simply pressing Enter creates the vertex at the current location.

Let's continue entering the other three vertices. To enter the second vertex, perform the following steps:

1. Enter 12.0 (do not press Enter yet) in the Y coordinate field, and 0.5 in the Z coordinate field.
2. Press Enter to create the vertex at 0, 12, 0.5.

A vertex will appear in the viewer at the desired location of the second vertex.

> **NOTE:** The default X value remains at 0.0, from our previous vertex.

1. Enter 200.0 in the X coordinate field, followed by Enter.

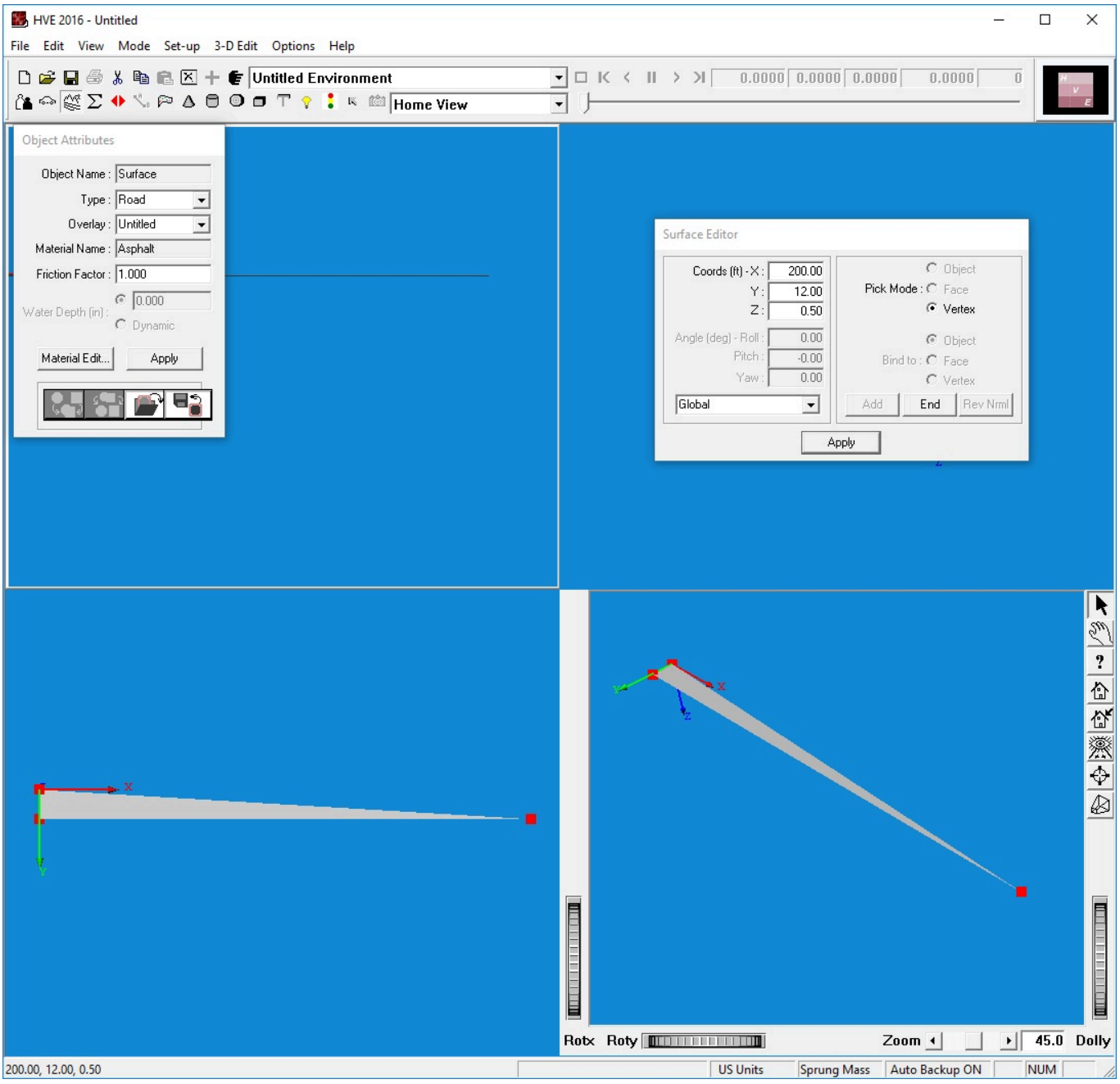


Figure 23-6 — Creating the first surface, shown after entering the first three vertices.

The vertex is displayed for the third coordinate at 200, 12, 0.5. See Figure 23-6 for an in-progress view of our first surface.

> **NOTE:** Again, note the Y and Z coordinates remain unaltered from their previous values.

1. Enter 0.0 in the Y coordinate field and 0.0 in the Z coordinate field, followed by Enter.

All four vertices for the first surface have been entered. To create the surface, click the Surface Tool's End button. The surface is displayed, as shown in Figure 23-7.

> **NOTE:** The vertices were entered in counter-clockwise order. Thus, according to the right-hand rule, the positive side of the surface is facing up. This is important! The simulation's tire model will generally not recognize any surfaces with the positive side facing down.

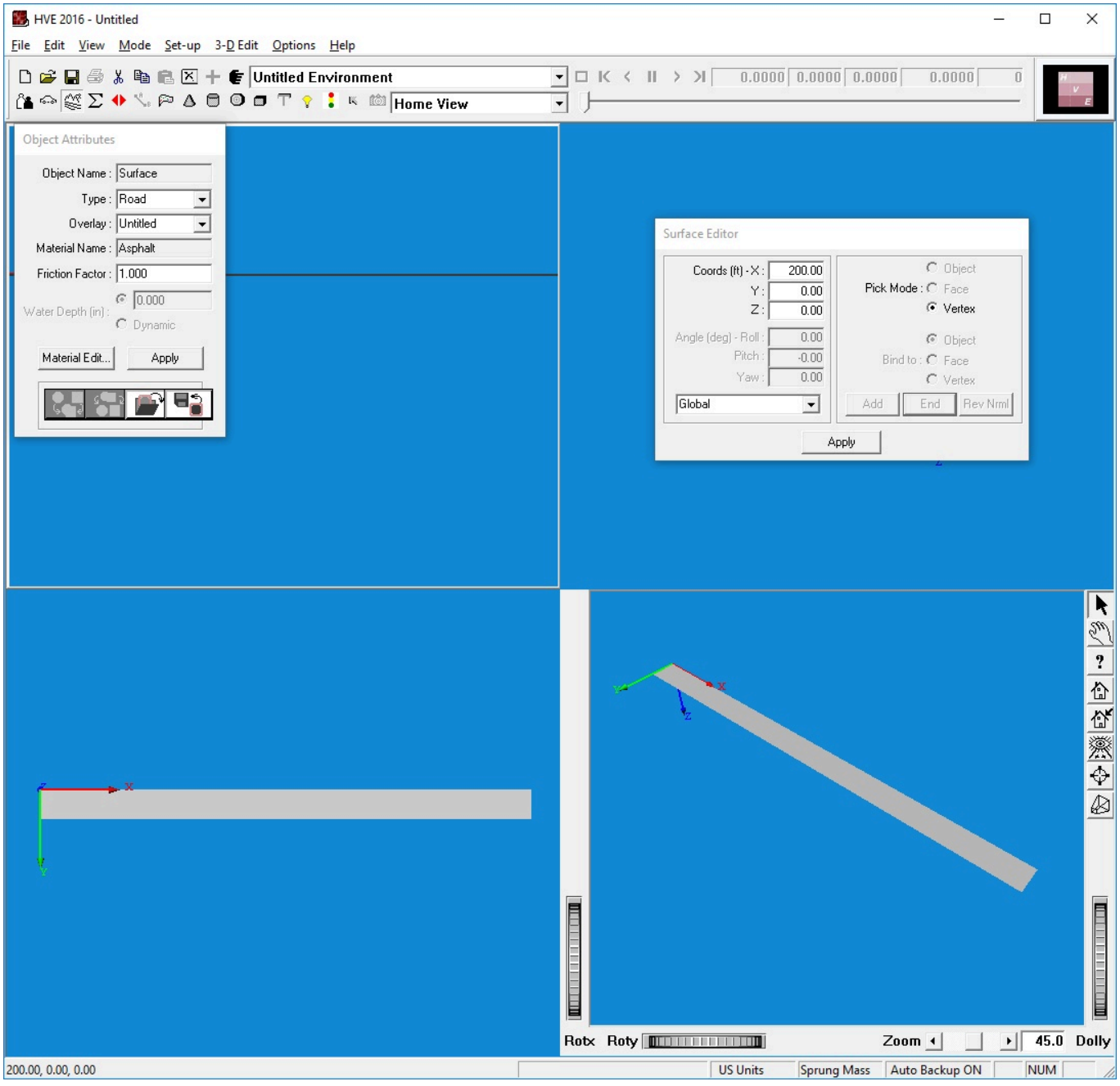


Figure 23-7 — 3-D Editor Perspective viewer after creating the first surface.

Let's create a second surface next to the first surface:

1. Place the mouse cursor in the X-coordinate field and press Enter to accept and display the first vertex at 200,0,0.

> **NOTE:** The starting coordinate for the second surface is the same as the last-entered coordinate for the first surface. This is often the case, and always saves a little time.

1. Enter -12.0 in the Y coordinate field, and 0.5 in the Z coordinate field, followed by Enter to accept and display the next vertex at 200,-12,0.5.
2. Enter 0.0 in the X coordinate field, followed by Enter to accept and display the next vertex at 0,-12,0.5.

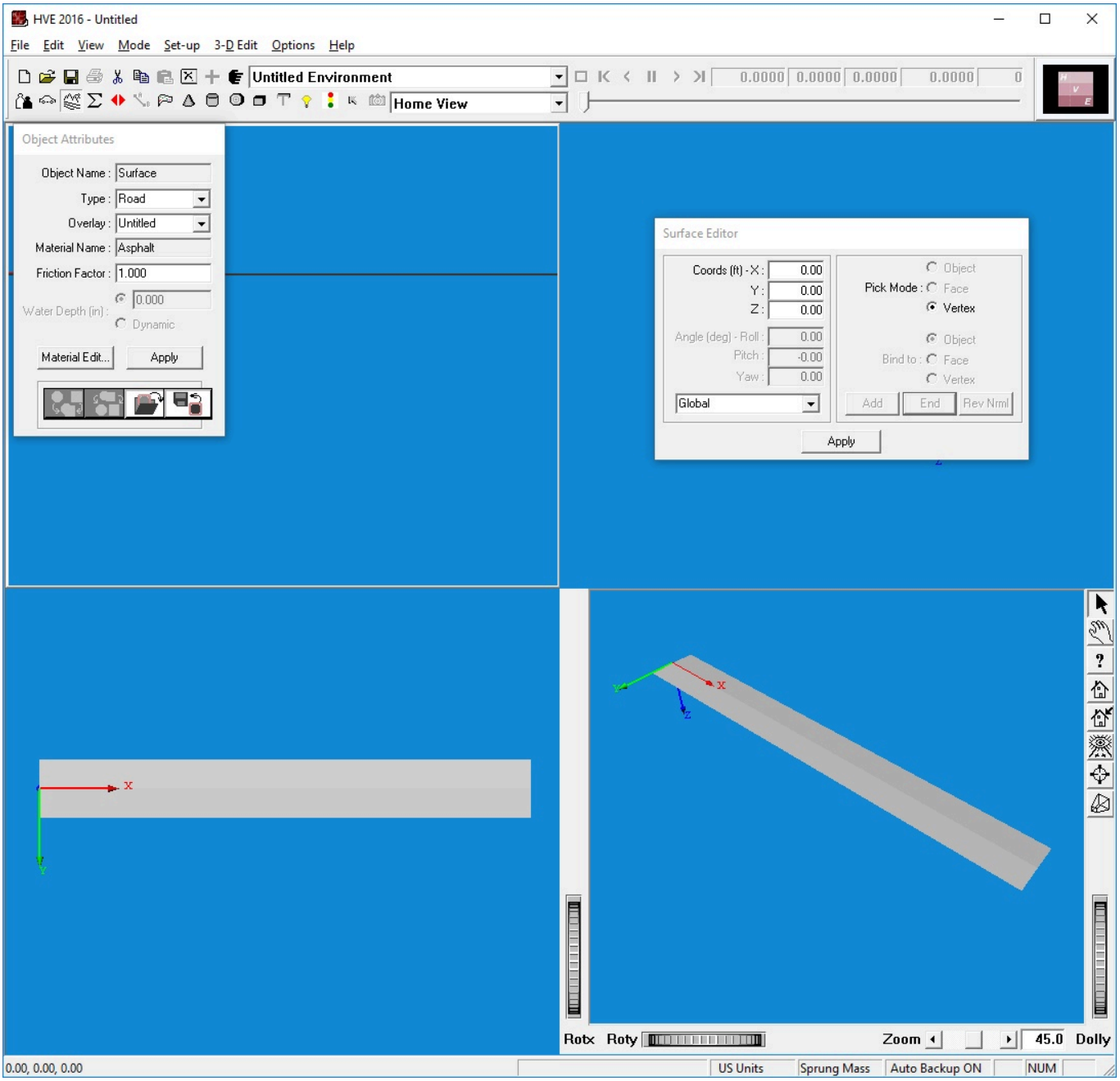


Figure 23-8 — 3-D Editor Perspective viewer after creating the second surface.

1. Enter 0.0 in the Y-coordinate field and 0.0 in the Z-coordinate field, followed by Enter, to display the fourth vertex at 0,0,0.

2. Press the End button in the Surface Editor dialog.

The red vertex icons disappear and the second surface is displayed in the viewers, as shown in Figure 23-8.

Our surface object is complete, so let's edit the surface:

1. Click on the Edit tool icon on the toolbar.

The Surface Tool is closed and the object is selected (as indicated by the red box surrounding it) and ready for editing.

Let's edit the surface by assigning friction characteristics and material attributes.

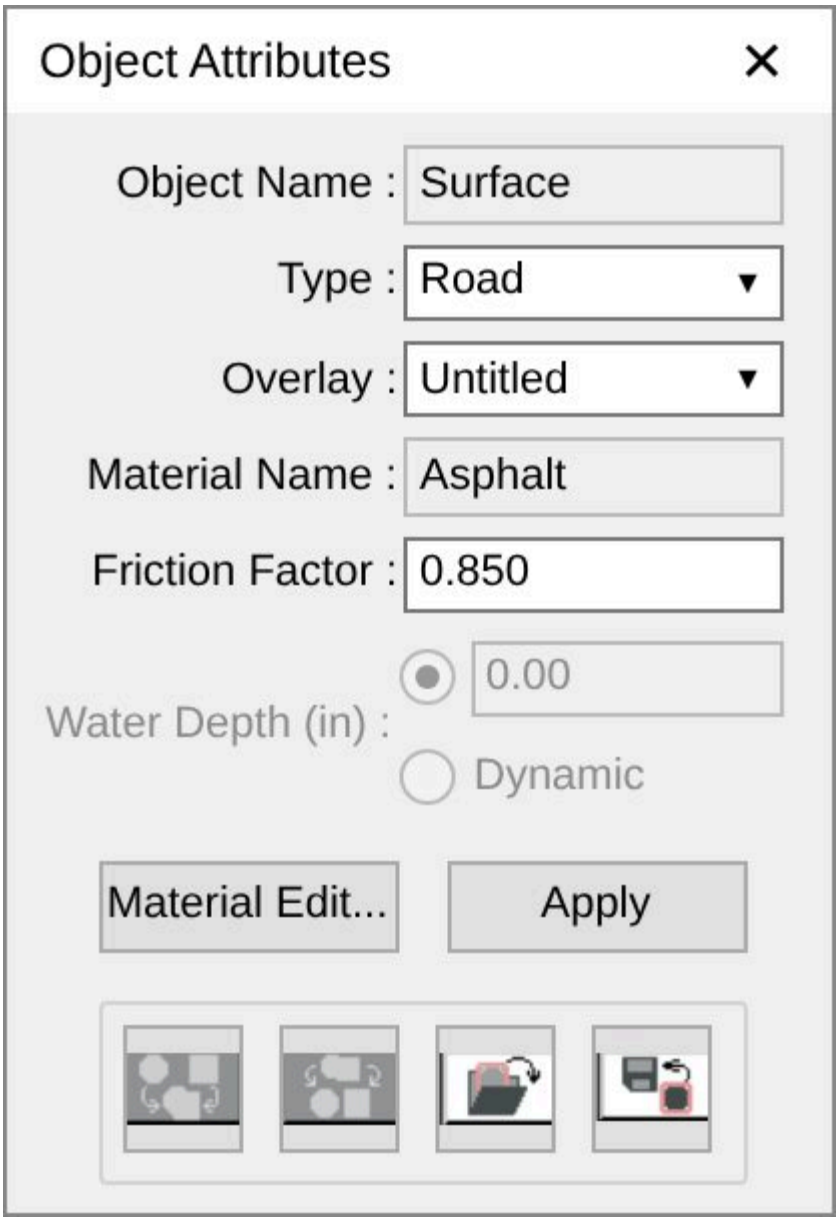


Figure 23-9 — The Object Attributes dialog is used for assigning surface friction properties.

Surface Friction

The friction characteristics are defined by two parameters:

- Object Type
- Friction Multiplier

These parameters are assigned using the 3-D Editor.

Object Type

The main object types available for environment objects are Road, Friction Zone and Other
(updated: Curb and Water types are also available in the current version; see the Object Attributes reference)
. The default type is Road, and that's what we want, so no action is required.

> **NOTE:** The selection of Road and Friction Zone object types establishes a hierarchy used by simulations: by default, all Friction Zone objects are searched first, then Road objects. Thus, if both a Friction Zone and a Road object exist beneath a tire, the Friction Zone will be used.
>
> **NOTE:** Simulations ignore objects of type Other.

Lesson 2 in this tutorial discusses object hierarchy. See also Chapter 19, Using the HVE 3-D Editor, for more information about object types.

Friction Multiplier

The friction multiplier is a user-entered value that modifies the dependent tire friction properties for every tire driving on a particular polygon. Thus, it may be used to increase or decrease the peak and slide friction properties for each tire that travels on that surface. In our example, let's assume the surface is well-traveled asphalt having 95 percent of the friction properties that were measured for the tires on a flat-bed tire tester. To change the friction properties for the surface, perform the following steps:

1. First, confirm the object is selected (as indicated by the red bounding box surrounding the object). If it is not selected, you can select the road surface by clicking on it.
2. Place the mouse cursor in the Friction Factor field and change the default Friction Factor to 0.950 from 1.000, followed by Enter. Then click on the Apply button (see Figure 23-9).

> **NOTE:** Be sure to press Enter and click the Apply button; otherwise the new value will not take effect.

The surface friction is now reduced to 95 percent of the original tire's peak and slide friction, as shown in Table 23-1.

Table 23-1 — Tire friction properties, as modified by the current Friction Factor. As each tire travels onto a new surface, its peak and slide friction properties are modified according to the surface's Friction Factor.

Surface	mu-peak (used by tire model)	mu-slide (used by tire model)
Normal (Factor = 1.0)	0.930	0.732
Modified (Factor = 0.95)	0.884	0.695

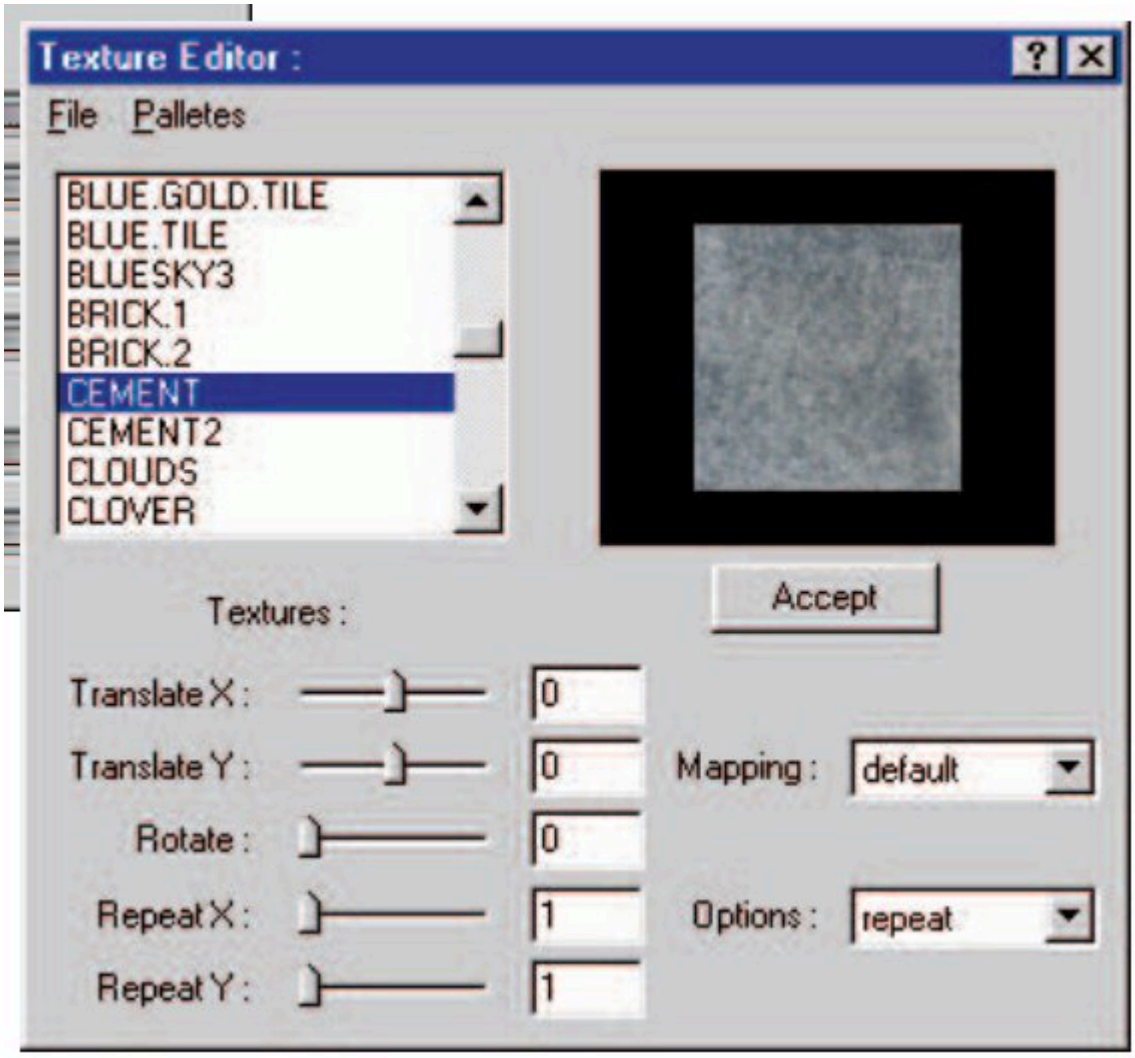


Figure 23-10 — Material Color Editor (left) and Texture Editor (right).

Assigning Material and Texture Attributes

An object's material attributes (color, intensity, and other factors affecting its appearance) are edited using the Material and Texture Editors. The default material is light gray in color. Let's change its appearance by assigning a texture to the surface.

To change the material attributes, perform the following steps:

1. First, confirm the object is selected (as indicated by the red bounding box surrounding the object). If it is not selected, you can select the road surface by clicking on it.
2. Choose the 3-D Edit menu option in the main menu bar, and click on Material Color or Material Texture. The Material Editor and Texture Editor dialogs are shown in Figure 23-10.

Note the Material Editor includes sliders for the Material Appearance attributes and an example viewer demonstrating the present appearance of a selected object. To change the color of an object, click on the Color radio button for the attribute (typically Diffuse) and a color wheel dialog will be displayed. The present color is indicated by the small box located on the color wheel. The Texture Editor includes a list of available textures and also attributes that affect the texture's appearance.

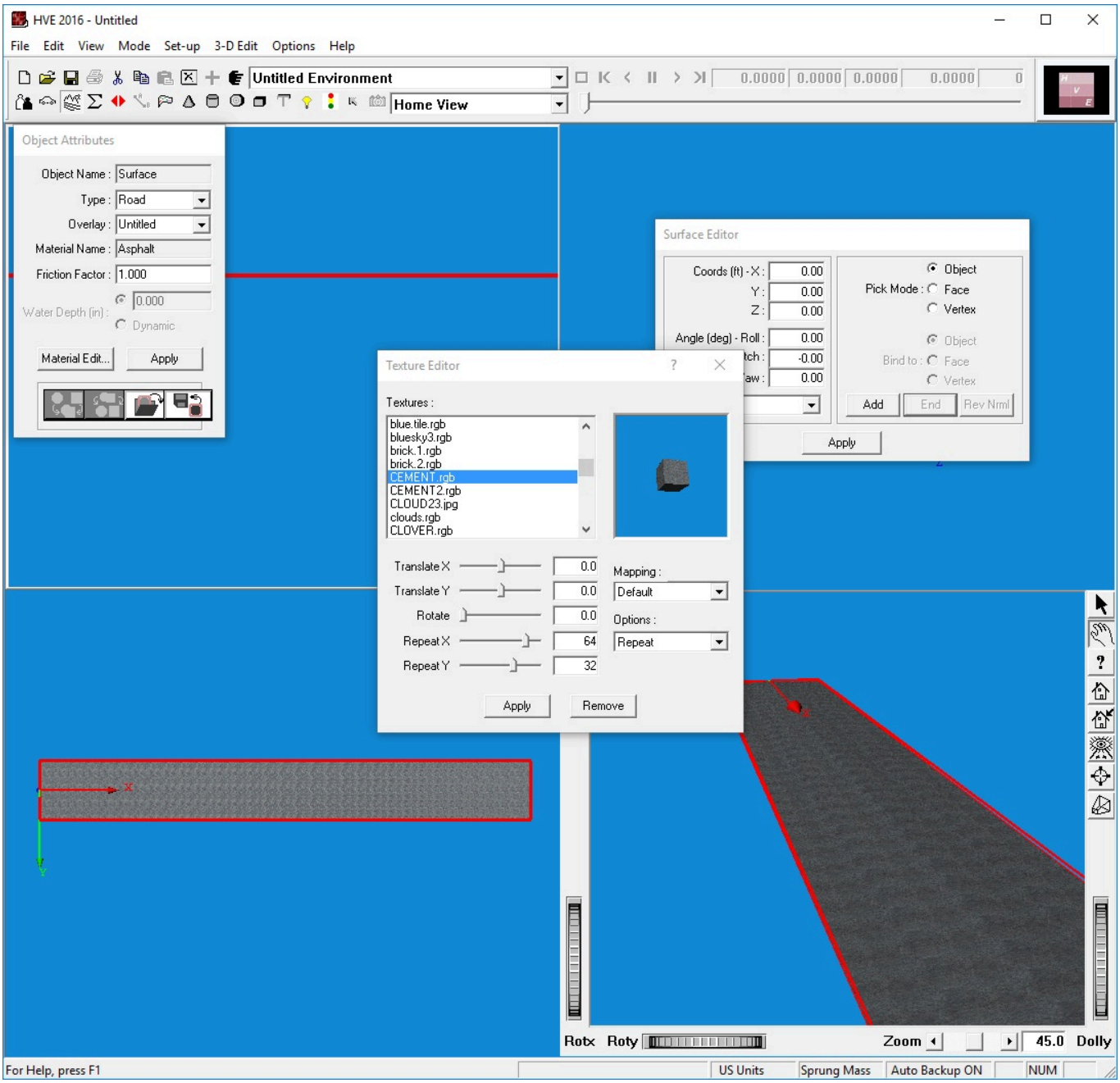


Figure 23-11 — Road surface after applying a texture.

Let's add a texture to the surface:

1. Select the object and then choose 3-D Edit and Material Texture. The Texture Editor dialog will be displayed.
2. Choose Cement from the list of available textures. Cement is displayed in the Texture Window.
3. Press Accept. The concrete texture is applied to our road surface.

Although the texture has been applied, it is blurry. The texture is out of focus because it has been stretched over a very large surface. To improve the appearance, we need to edit the Texture Attributes using the Texture dialog:

1. Change Repeat X to 100 from 1.0.
2. Change Repeat Y to 40 from 1.0.

3. Press Accept. The texture now looks much better.

> **NOTE:** The Advanced Tutorial chapter with an in-depth discussion of textures is reserved for future use; see the Texture Editor section of Chapter 19 in the meantime.

The resulting road surface is displayed in Figure 23-11.

1. Click on the Close dialog control button (upper right corner) to close the Texture Editor.

Assigning an Overlay Name

Overlays are used to improve your efficiency as an HVE user. Because all 3-D objects take time to render, it makes sense to allow the user to select certain objects for rendering, and deselect other objects that are not important for the current task.

> **NOTE:** The most obvious time to use overlays is during Event mode. Because you are executing a simulation that relies on physical surfaces, it makes sense to select those surfaces that are involved in the physical interaction with humans and vehicles, and deselect all other objects to reduce rendering time. This can greatly speed up your work during Event mode. Then, during Playback mode, while creating your video, you can turn these overlays back on before rendering your final simulation sequence.
>
> **NOTE:** Even though an object is not rendered if its overlay is turned off, its surface geometry is still used by the tire model. In other words, your vehicle would be driving up on the embankment, even though you could not see the embankment.

Overlays are created using the Object Attributes dialog. When you create any object, it is assigned to the current overlay, according to the current overlay name.

> **NOTE:** The default overlay is named Untitled. The Untitled overlay is just like any other overlay.

Overlay names are assigned by editing the current Overlay Name. Let's change the overlay name of our road surface from Untitled to Road:

1. Place the mouse cursor in the Overlay field and replace the existing overlay name, Untitled. Enter: Road, followed by Enter.

> **NOTE:** You must press Enter; otherwise the name will not be assigned. See Chapter 32, HVE Tutorial, for more information about entering data in modeless dialogs.

The new name is displayed in the Overlays combo box, as shown in Figure 23-12.

In order to turn overlays on or off, we will need to first deselect the objects displayed in the 3-D Editor.

1. Move the cursor to a location not over any objects and click the left mouse button. The red bounding box disappears.

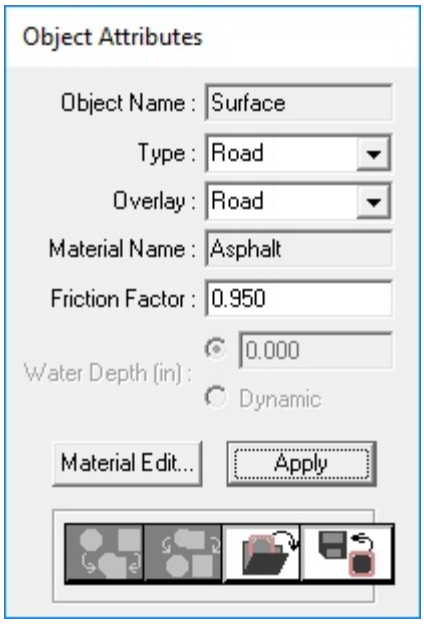


Figure 23-12 — The 3-D Editor dialog after adding our new overlay, named Road. Clicking on the combo box's button displays all the overlays in the current environment.

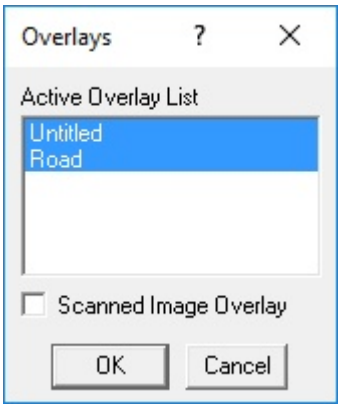


Figure 23-13 — Overlays dialog, showing all the overlays selected in our current case.

Because we assigned a unique overlay name, we can choose to render, or not render, the Road object we just created using the Overlays dialog, selectable from the View menu (see the Overlays reference). Let's illustrate this point:

1. Click on the View option in the HVE menu bar, and choose Overlays. The Overlays dialog is displayed, as shown in Figure 23-13. Note that Road is in the Active Overlays list, and it is selected (highlighted). Let's turn off the Road overlay:
2. In the Active Overlays list, click Road to deselect it.
3. Press OK.

The Overlays dialog is removed, and our 3-D Editor viewers are empty! Let's not leave it this way:

1. Click View in the HVE menu bar, and choose Overlays. The Overlays dialog is displayed.
2. Select Road.
3. Press OK.

The road surface is again displayed in the 3-D viewers.

> **NOTE:** This is a rather trivial example of the use of overlays. This single surface is rendered faster than we can comprehend. However, if we were rendering a scene with an overlay containing a million polygons, turning that overlay on and off would make a huge difference in rendering time (and therefore, productivity!).

Saving Your Work

Now, let's save our work. Two methods may be used:

- Saving it in the case
- Saving it as a 3-D geometry file

The benefits of these two methods are described below.

Saving The Object In The Case File

Saving the object in the case actually embeds the geometry in the case file. The next time you open the case, the geometry is automatically included in that case. Also, if you create a copy of the case and give it to another HVE user, that person's case file also includes the embedded geometry. This, of course, is essential because the geometry is a physical surface used during event execution.

To save the object in the case file, perform the following steps:

1. Choose the File menu option in the main menu bar, and choose Save As. The File Selection dialog is displayed, prompting you to enter a case title and filename.
2. First, enter a case title. This title appears as a heading on all your printed output reports. Enter: My First Road.
3. Next, enter a filename. This is the name of the case file that stores all the case information. Enter: 3dEdTutor1.
4. Click on Save.

The case is now saved, and the 3-D geometry is part of the case.

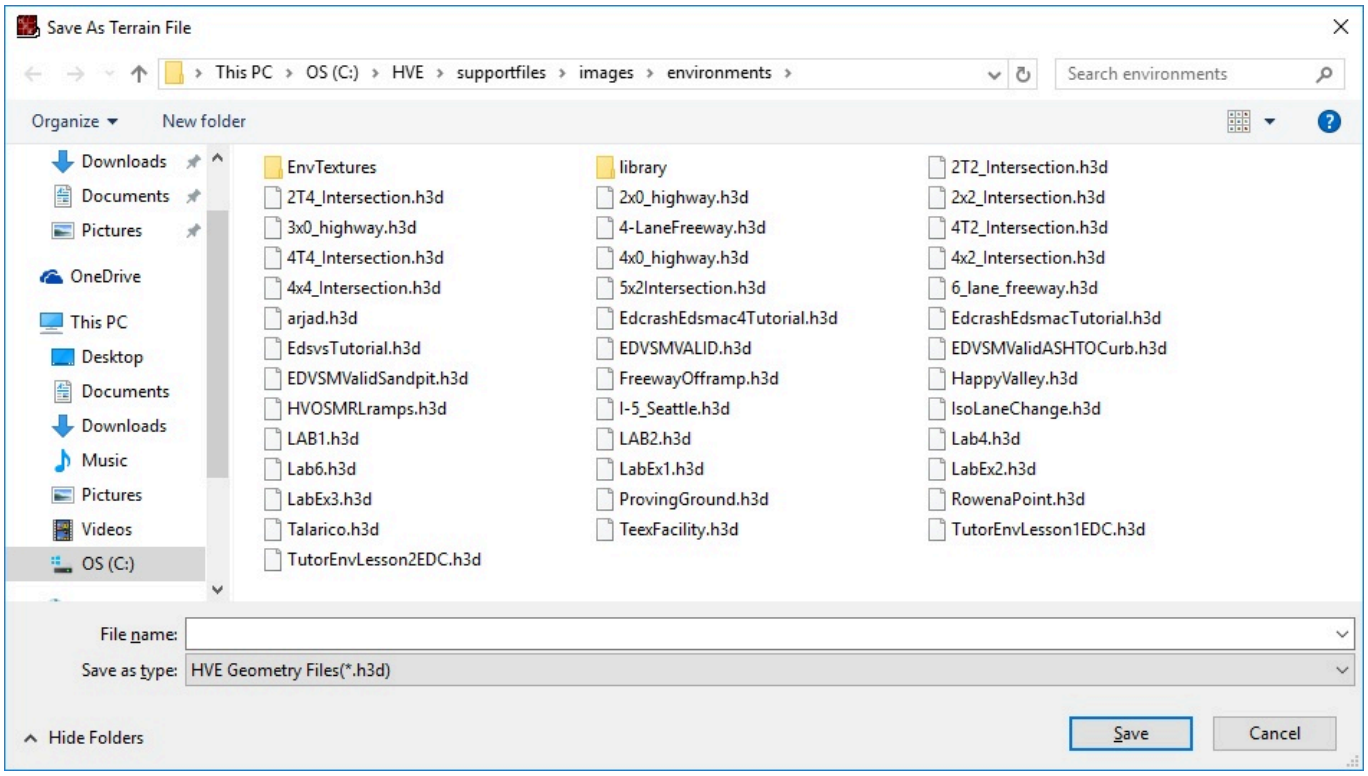


Figure 23-14 — File Selection dialog, used for loading and saving 3-D geometry files.

Saving The Object As A Unique File

What if you wanted to use your newly created environment in other cases? The best way would be to simply give it a unique filename. To save your environment by a unique filename, perform the following steps:

1. First, close the 3-D Editor by selecting 3-D Edit on the main menu and choosing Close 3-D Editor on the cascade menu. The 3-D Editor is closed, revealing the Environment Editor and Viewer with your road surface.
2. Click on the Object Info button on the toolbar. The Environment Information dialog is displayed (shown earlier in Figure 23-1), allowing us to view and edit the current environment attributes. Note we also use this dialog for opening and saving 3-D geometry files.
3. Click on Save As. The File Selection dialog is displayed, allowing us to save our geometry using the desired file format and filename (see Figure 23-14).
4. Click on the Format option list to display the available file formats, and choose HVE.

> **NOTE:** Only HVE is currently supported for saving files.

1. Enter a filename for the 3-D geometry file. Enter: TutorEnvLesson1.

> **NOTE:** There is no need to supply an extension, because HVE will automatically supply the .h3d extension.

1. Click OK in the File Selection dialog. The file is now saved in the `...\hve\supportFiles\images\environments` subdirectory.
2. Click OK in the Environment Information dialog.

We are now ready to continue.

Ending the Session

The last thing to do is end the session. This is done using the File menu:

1. Choose the File menu option in the main menu bar, and click on Exit. You are asked if you would like to save your changes.
2. Choose Yes.

> **NOTE:** It is always safest to save your file, unless you are certain you will not need it again.

HVE has now shut down.

Summary

This concludes Lesson 1 — Learning the Basics. You should now be familiar with starting the 3-D Editor, creating simple surfaces, applying surface friction attributes and material textures, and saving your 3-D geometry.

Lesson 2 — Editing a Surface

This lesson is designed to teach you how to edit existing surfaces using the 3-D Editor. In this lesson, you will learn the following:

- Moving vertices
- Reversing surface normals
- Changing object hierarchy

> **NOTE:** We assume HVE is already running and you have completed Lesson 1 — Creating a Simple Road Surface. If you have not finished Lesson 1, you should do that before continuing with this lesson.

Surface editing tasks are performed for a variety of reasons. Examples include correcting errors that occurred when the surface was created and modifying an imported geometry file. In any case, the surface editing procedures are important to learn and use.

In Lesson 2, we will start with the 3-D geometry file we created in Lesson 1, so let's begin by loading that file:

1. If necessary, switch to Environment mode. The Environment Editor is displayed.
2. Click Add New Object. The Environment Information dialog is displayed.
3. Choose Open. The Environment Geometry File Selection dialog is displayed.
4. Select TutorEnvLesson1.h3d and press OK. The File Selection dialog disappears.
5. Press OK again. The Environment Information dialog disappears and the selected environment is displayed in the Environment 3-D viewer.

Spend a few moments manipulating the viewer to look at various views of the simple road surface we created in Lesson 1.

Now we're ready to edit the surface geometry. Editing 3-D geometry is always performed using the 3-D Editor, so let's start the editor:

1. Select 3-D Edit and choose Launch 3-D Editor. The 3-D Editor is started with our simple road surface displayed in each of the four viewers.
2. Arrange your desktop to your liking.

> **NOTE:** Remember that most of the editing is usually performed using the perspective viewer in the lower right and X-Y plane viewer in the lower left. With this in mind, you'll probably want to position your dialogs in the top portion of the desktop.

Now, we're ready to begin editing our simple surface.

Moving Vertices

In Lesson 1, we created a simple road surface. Now, we'll edit that road to create a larger road. Our original road was 200 feet long and 24 feet (2 lanes) wide. Let's make it 2000 feet long and 4 lanes wide, plus a 4-foot wide median (making the total width 52 feet).

> **NOTE:** It is common for beginners to make the environment too small. It is normally just as easy, especially for straight roads, to make the road as long as possible; there's really no reason to artificially shorten it!

Let's begin editing the surface:

1. Confirm the 3-D Editor is in Edit mode (see Figure 23-15).
2. Confirm the perspective viewer is in Pick mode (i.e., the crosshairs cursor is displayed when the mouse is in the perspective viewer).
3. Click on the surface in the perspective viewer. A red bounding box surrounds the object, indicating it is now selected for editing.

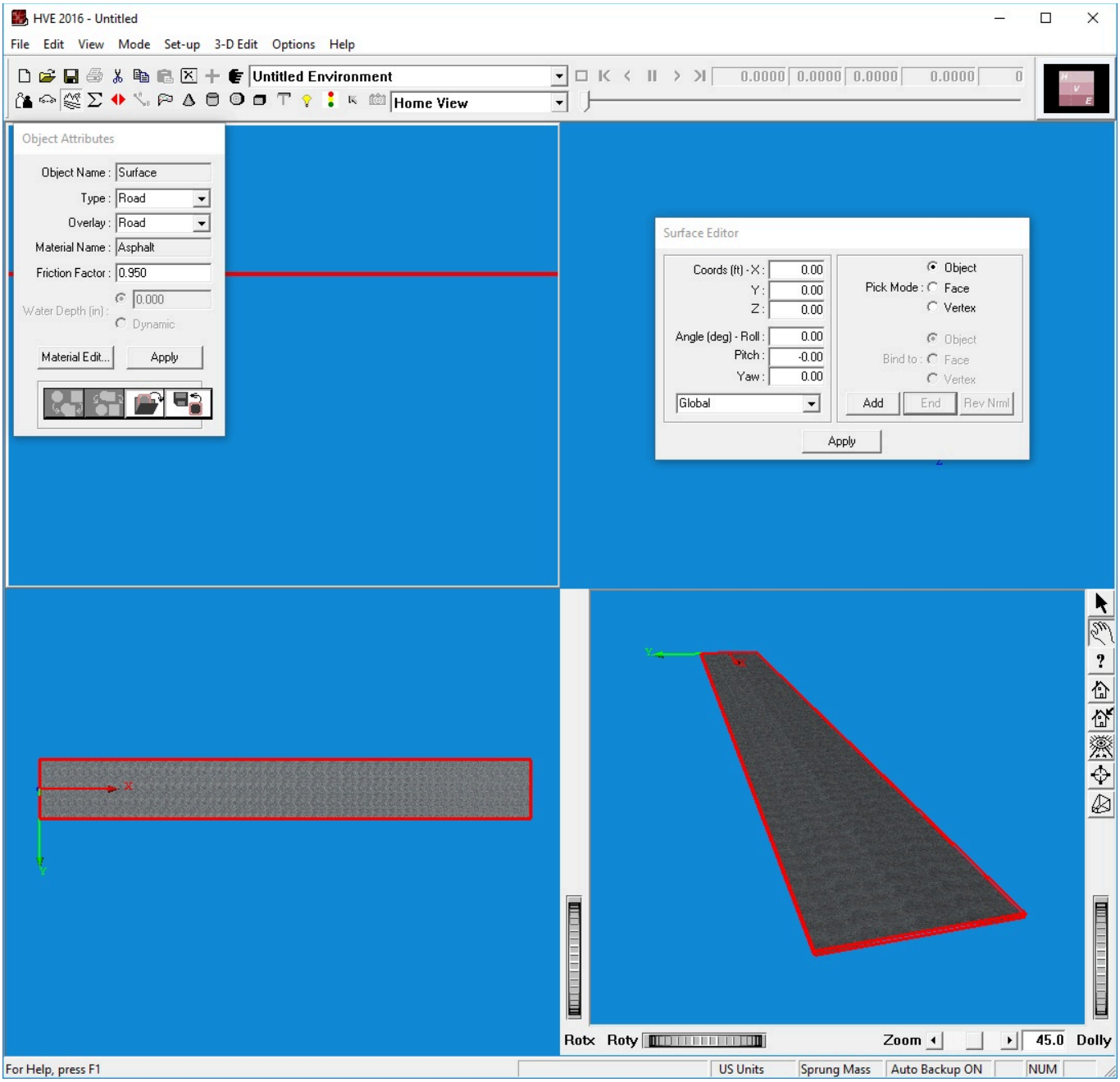


Figure 23-15 — 3-D Editor, ready to edit our simple surface.

The Surface Editor dialog is displayed, showing the current coordinates for the surface object.

> **NOTE:** The current coordinates and orientation are both 0,0,0. This is initially the case for all objects. When the object is moved from its original position, the coordinates and/or orientation will change.

The Surface Editor dialog also displays the current Pick mode. The default is Object. To edit the vertices, the Surface Editor must be in Vertex Pick mode:

1. Click the Vertex radio button.

Now we're ready to edit vertices. Let's start by editing the vertex at the origin:

1. Click near the origin inside the red bounding box. A vertex manipulator is displayed for the selected vertex and the vertex coordinates, 0,0,0, are displayed in the Surface Editor dialog's coordinate fields. See Figure 23-16.

> **NOTE:** If you click outside the bounding box, you'll deselect the surface.

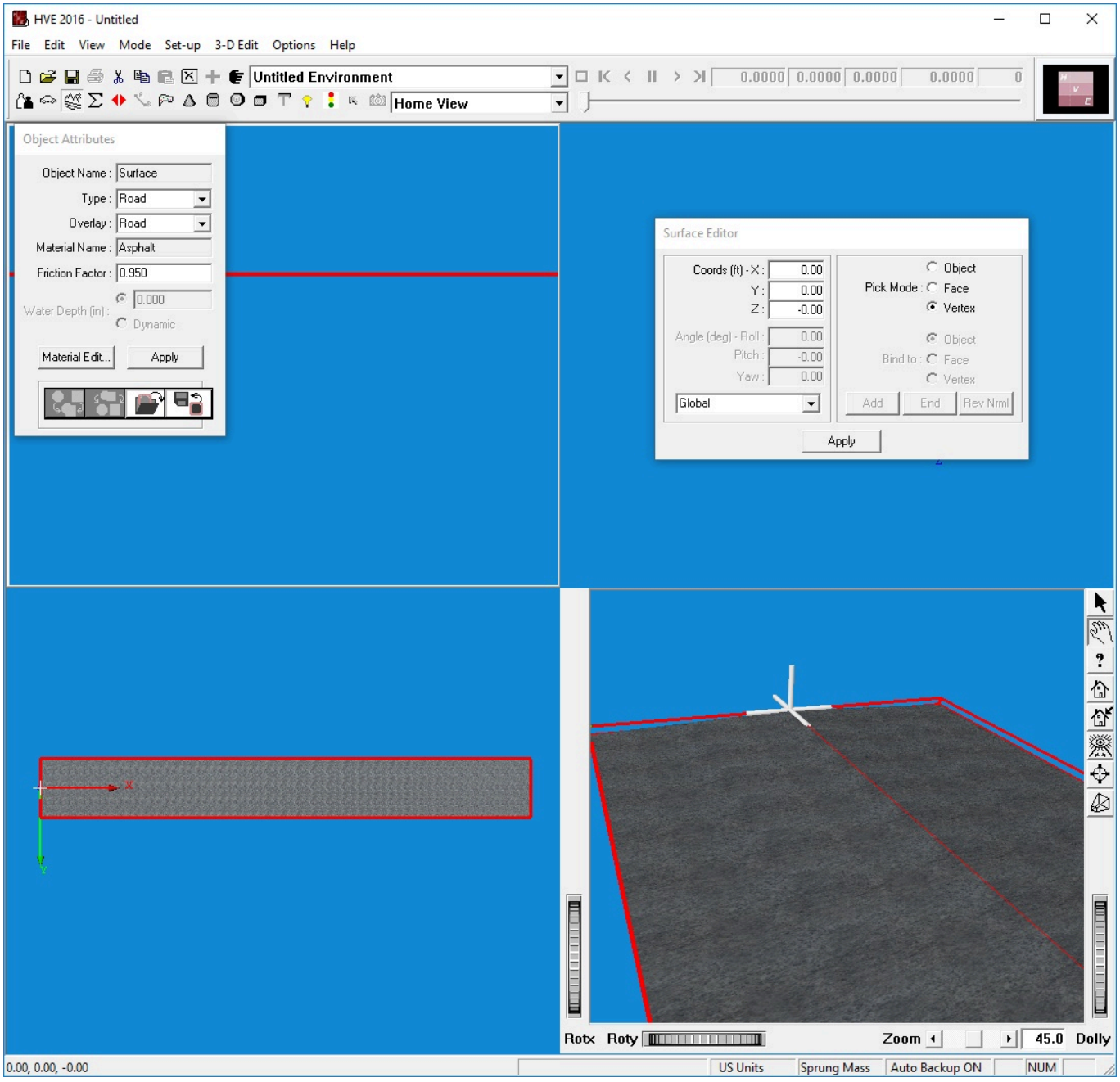


Figure 23-16 — Simple surface, after zooming in and selecting the vertex near the origin.

Let's edit the vertex coordinates:

1. Enter -1000.0 ft in the X field, followed by Enter. The vertex is repositioned at -1000,0,0 (see Figure 23-17).
2. Click the vertex inside the red bounding box near 0.0, 12.0, 0.5. A vertex manipulator is displayed for the selected vertex and the vertex coordinates are displayed.

> **NOTE:** Again, remember to click inside the bounding box; otherwise you'll deselect the surface.

Edit the vertex coordinates:

1. Enter -1000 in the X field and 26 in the Y field, followed by Enter. The vertex is repositioned at -1000, 26, 0.5.

Edit the remaining vertices using the same procedures:

1. Click the vertex near 200,12,0.5 and change its X coordinate to 1000 and its Y coordinate to 26 (remember to press Enter).
2. Click the vertex near 200,0,0 and change its X coordinate to 1000.

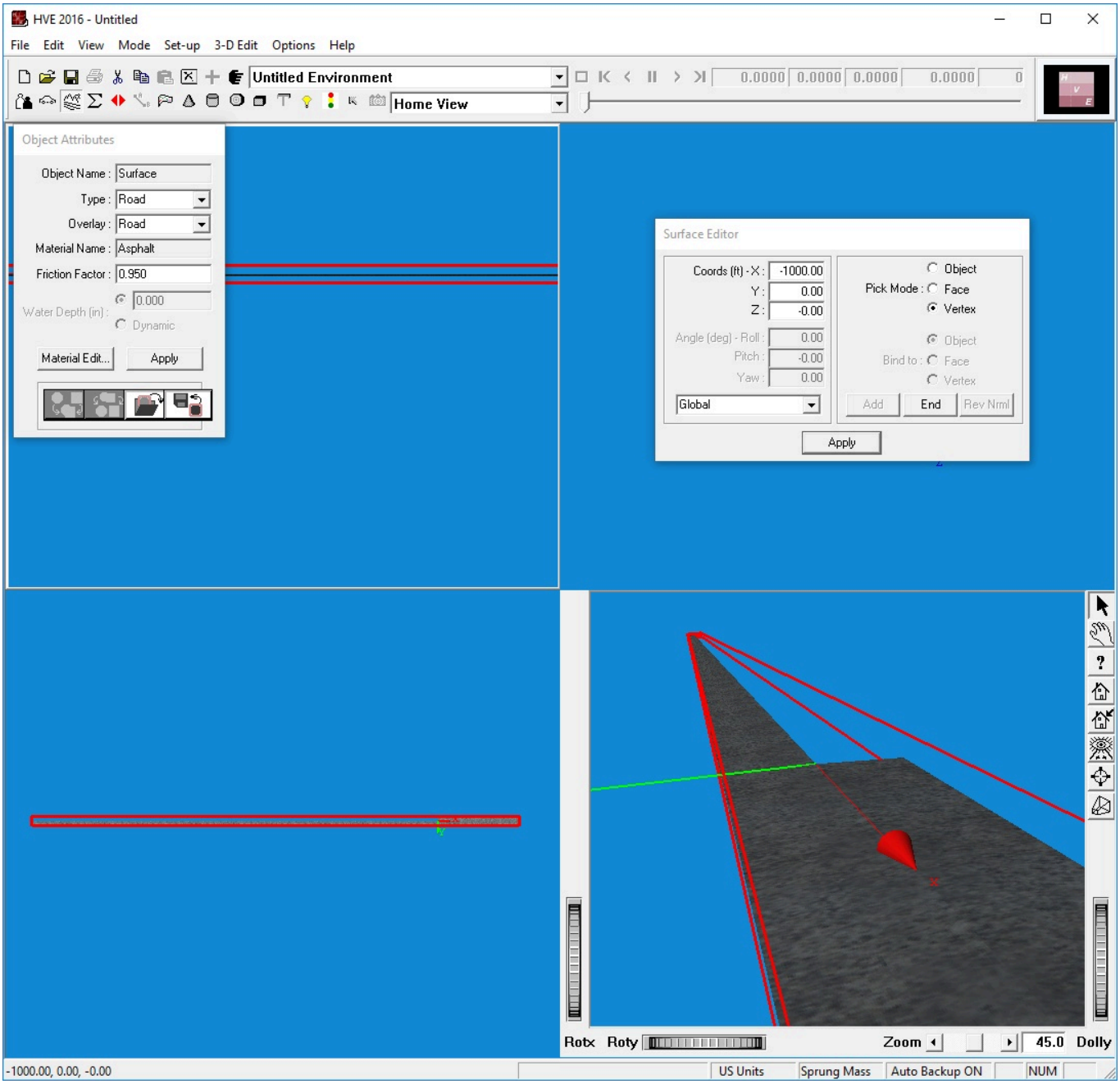


Figure 23-17 — Simple surface after moving the first vertex from 0,0,0 to -1000,0,0.

Half the surface has been edited. Let's repeat the steps to edit the other side. If necessary, adjust the view so it appears somewhat like Figure 23-18. Then edit as follows:

1. Click near the origin. The vertex at 0,0,0 is selected.

> **NOTE:** Since the vertex is not displayed until it is selected, you need to estimate where the vertex is located and click in that vicinity.

1. Edit the X value, changing it to -1000, followed by Enter.
2. Click near the vertex near 200,0,0 (see Figure 23-19) and change its X coordinate to 1000.
3. Click near the vertex at 200,-12,0.5 and change its X coordinate to 1000 and its Y coordinate to -26.
4. Click near the vertex at 0,-12,0.5 and change its X coordinate to -1000 and its Y coordinate to -26.
5. Click outside of the bounding box. The editing operation is completed and the surface is deselected.

The resulting surface is displayed in Figure 23-20.

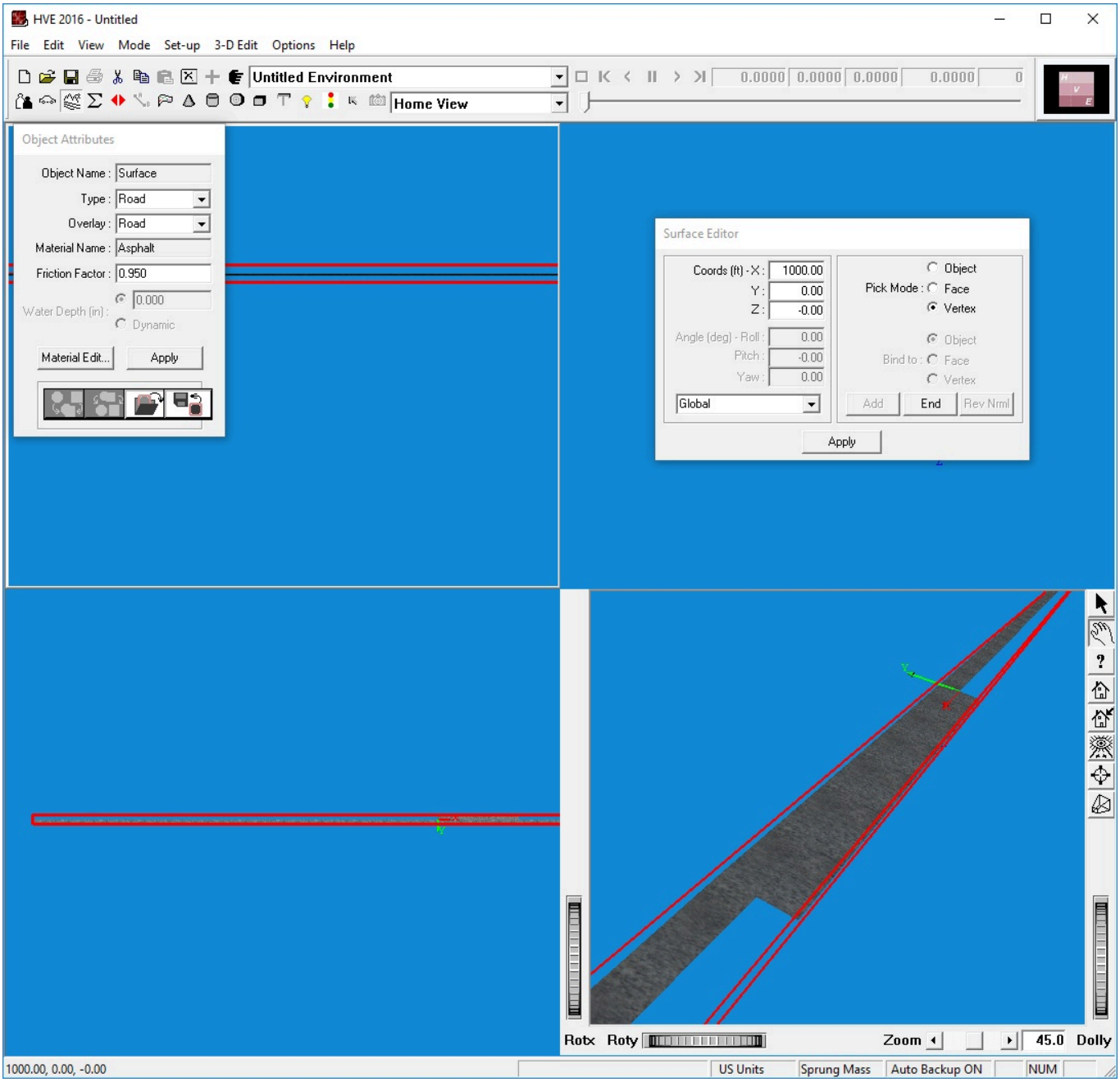


Figure 23-18 — Simple surface after editing half the surface; the view has been adjusted to get better access to the remaining surface.

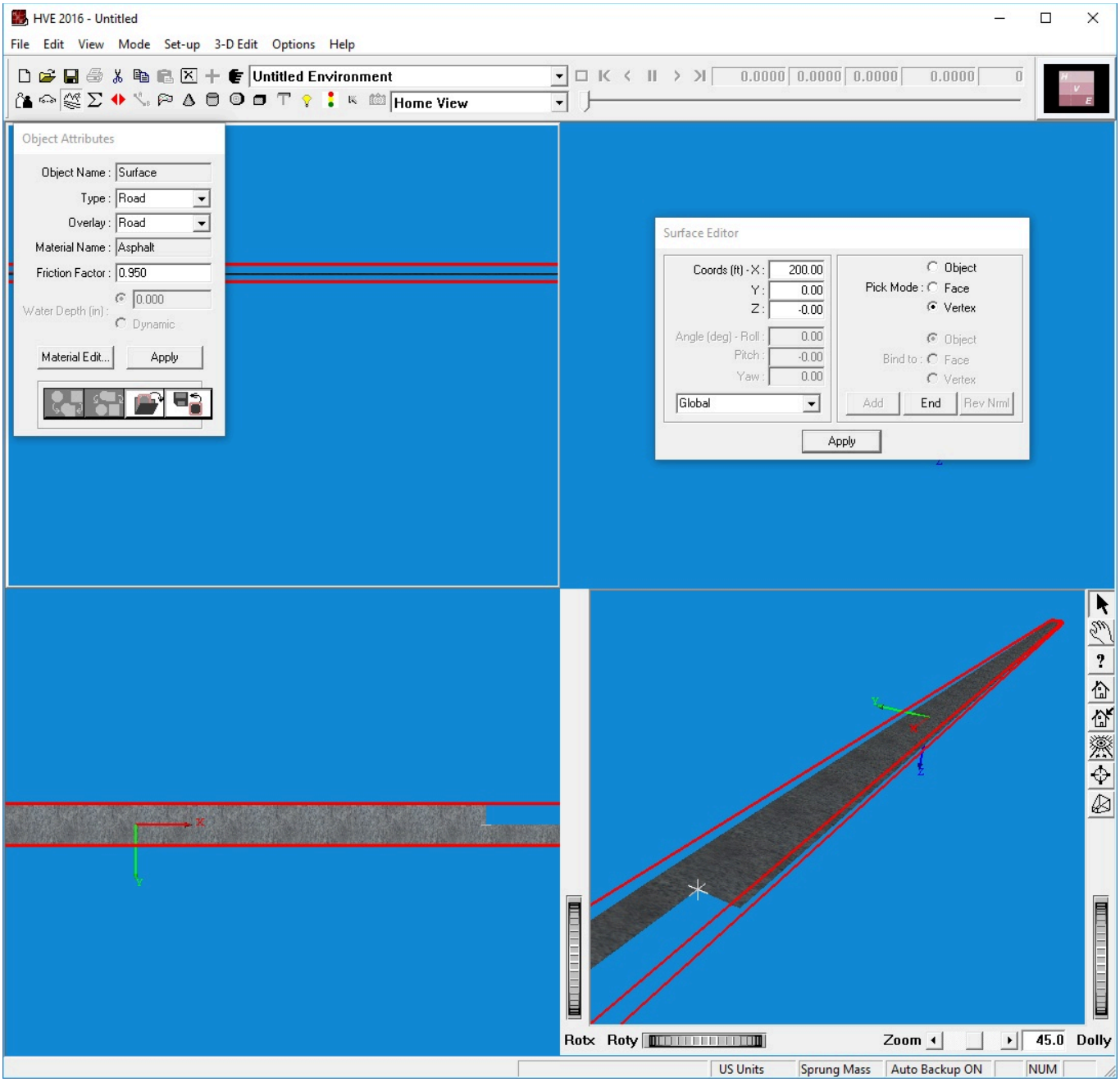


Figure 23-19 — Simple surface with vertex at 200,0,0 selected, ready to move to 1000,0,0.

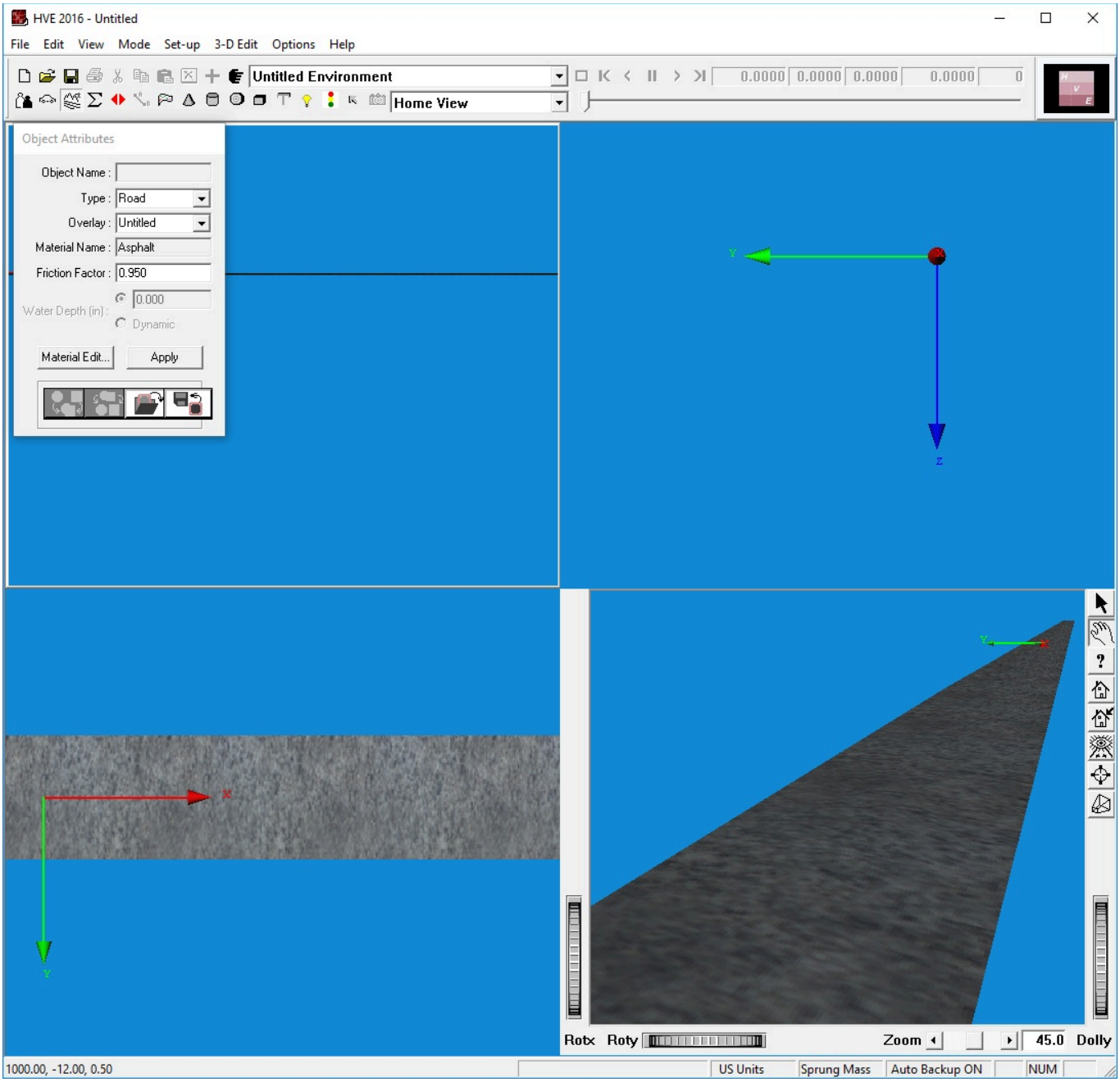


Figure 23-20 — Simple surface after editing is complete.

Reversing Surface Normals

Each 3-D surface has a (+) side and a (-) side. The surface's (+) or (-) side is determined by the direction of its surface normal. As shown in Figure 23-21, a surface normal is a vector drawn perpendicular to a specific location on the surface. The direction is determined using the right-hand rule according to the order in which the vertices were entered. Vertices must be entered in counter-clockwise order to establish that the positive side is facing you as you view the surface.

If you accidentally enter the vertices in the wrong order, the wrong side of the surface will be rendered. In addition, a tire will not see the surface and will fall through it.

> **NOTE:** Vehicles can drive only on the (+) side of a surface.
>
> **NOTE:** Some CAD programs (namely AutoCAD Release 13 and earlier) do not assign surface normals in a consistent manner. Thus, you may experience difficulty with the direction of the surface normals after importing files from these programs.

The HVE Surface Editor allows you to change the direction of the surface normals for every face. Let's illustrate this process using the surface we're currently working on. Start by adding a new surface:

1. Choose the Surface tool. The Surface Editor dialog is displayed.

Let's create a Tee intersection by adding another road surface:

1. Enter X = -20 ft, Y = 26 ft, Z = 0.5 ft. Press Enter to create the first vertex.
2. Enter X = 20 ft, Y = 26 ft, Z = 0.5 ft. Press Enter to create the second vertex.
3. Enter X = 20 ft, Y = 500 ft, Z = 25 ft. Press Enter to create the third vertex.
4. Enter X = -20 ft, Y = 500 ft, Z = 25 ft. Press Enter to create the fourth vertex.
5. Press End to create the surface.

Where is our surface? It's right in front of us — it's just upside down, so we can't see it! Change the view as required to look on the under side of the surface, as shown in Figure 23-22. You will find it is indeed there. We need to turn it over:

1. Choose Edit mode by clicking the Edit Tool button on the toolbar. A red bounding box is displayed around the object, indicating it is selected.

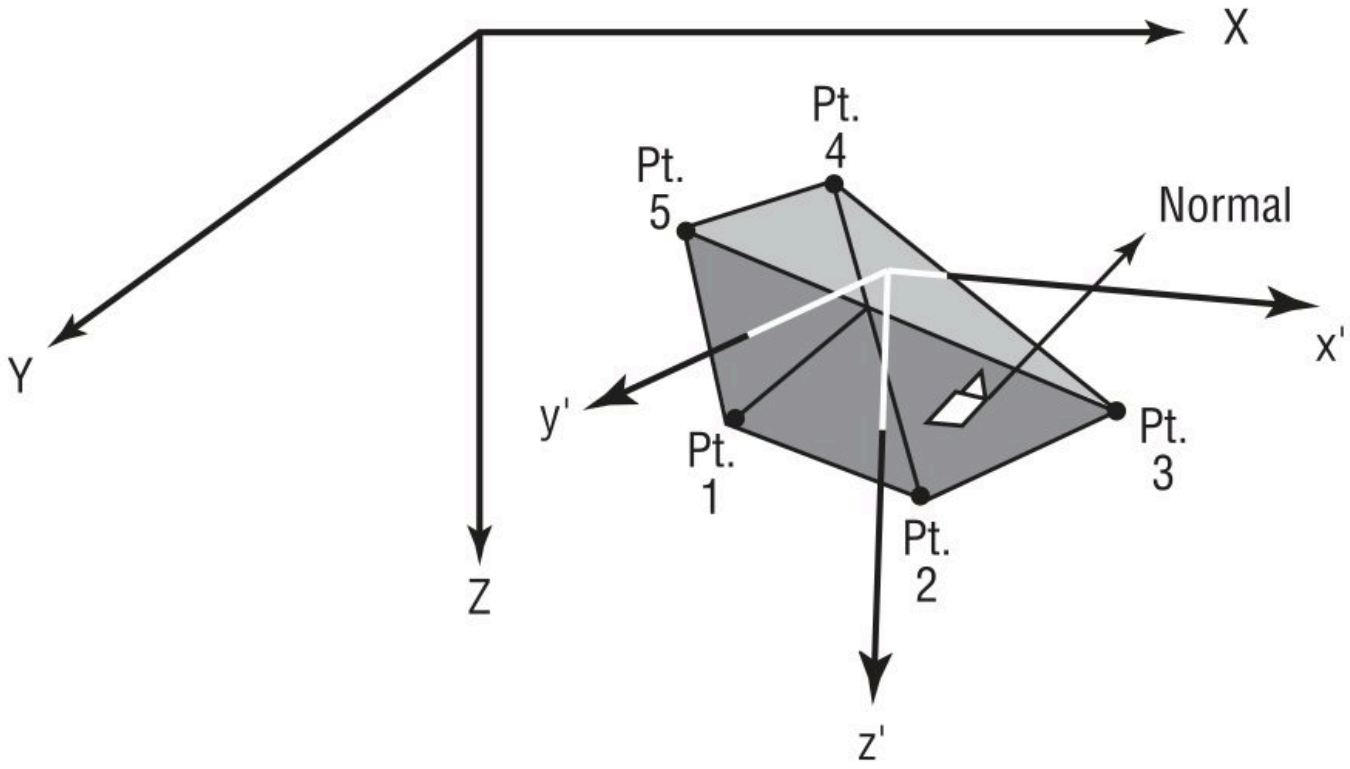


Figure 23-21 — Surface normal. The example shows a single surface object composed of a single face set containing five triangles, created by clicking points 1 through 5 in counter-clockwise order. The normal is perpendicular to the triangle composed of points 2, 3 and the computed center point.

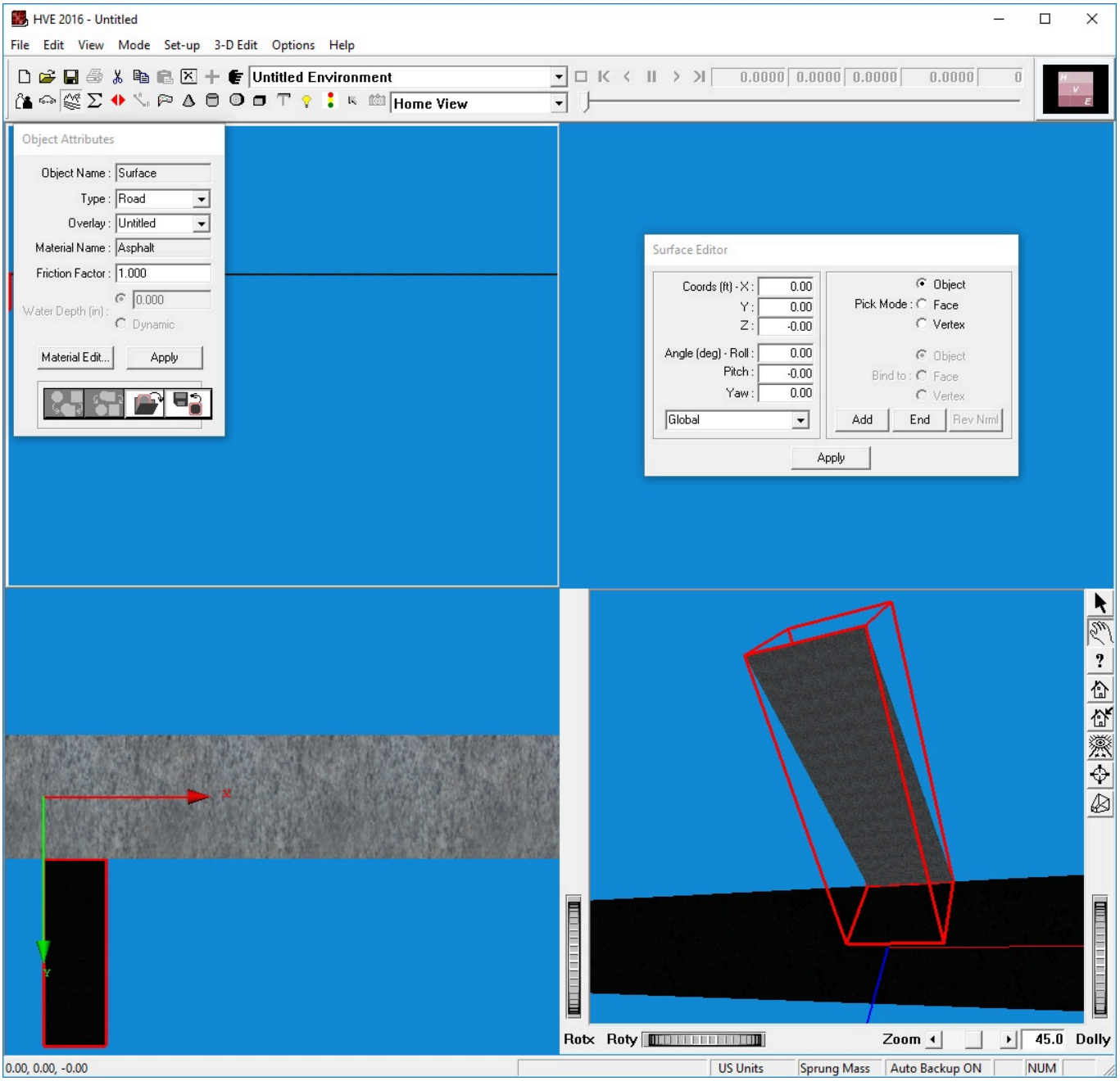


Figure 23-22 — Upside-down surface, revealed by turning the view over and finding the texture attached to the under side.

The Surface Editor's Pick Mode is currently set to Object. To edit a face, we need to pick a face:

1. Click the Face Pick Mode radio button. Nothing happens yet, because we need to pick a face.
2. Click on the surface somewhere inside the bounding box. A face is selected and a vector is displayed where we clicked. This vector is the surface normal (see Figure 23-23).
3. Click Rev Nrml. The surface is reversed, as shown in Figure 23-24.

> **NOTE:** In some imported DXF files, you may have thousands of polygon faces that need to be edited. In addition, the DXF file information has the vector reversed, so it always shows the NEGATIVE direction! EDC has some advanced tricks that will solve these problems, so please contact EDC Technical Support if you need assistance.

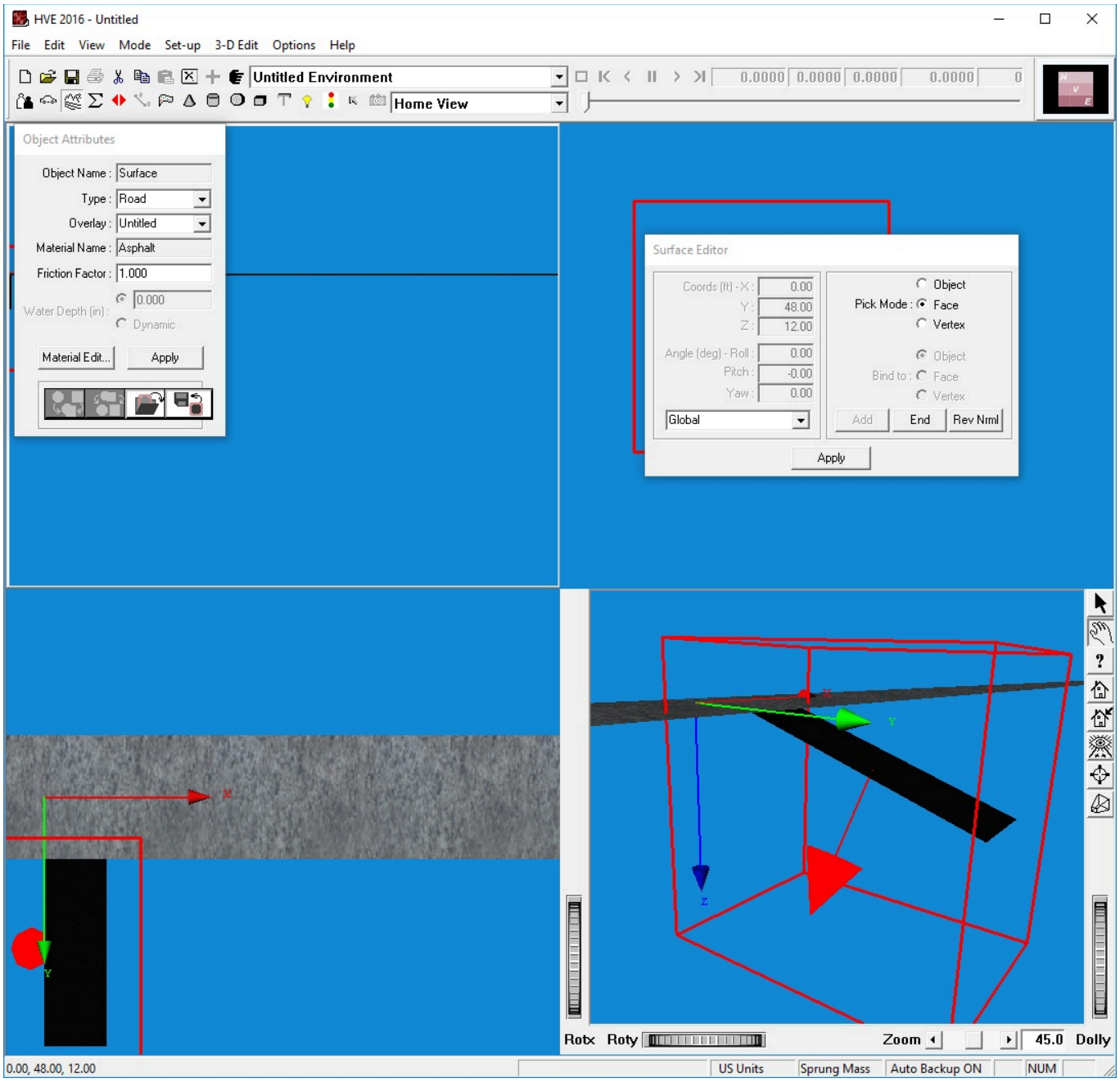


Figure 23-23 — Upside-down surface with normal vector displayed. Note the vector is pointing down (in the wrong direction).

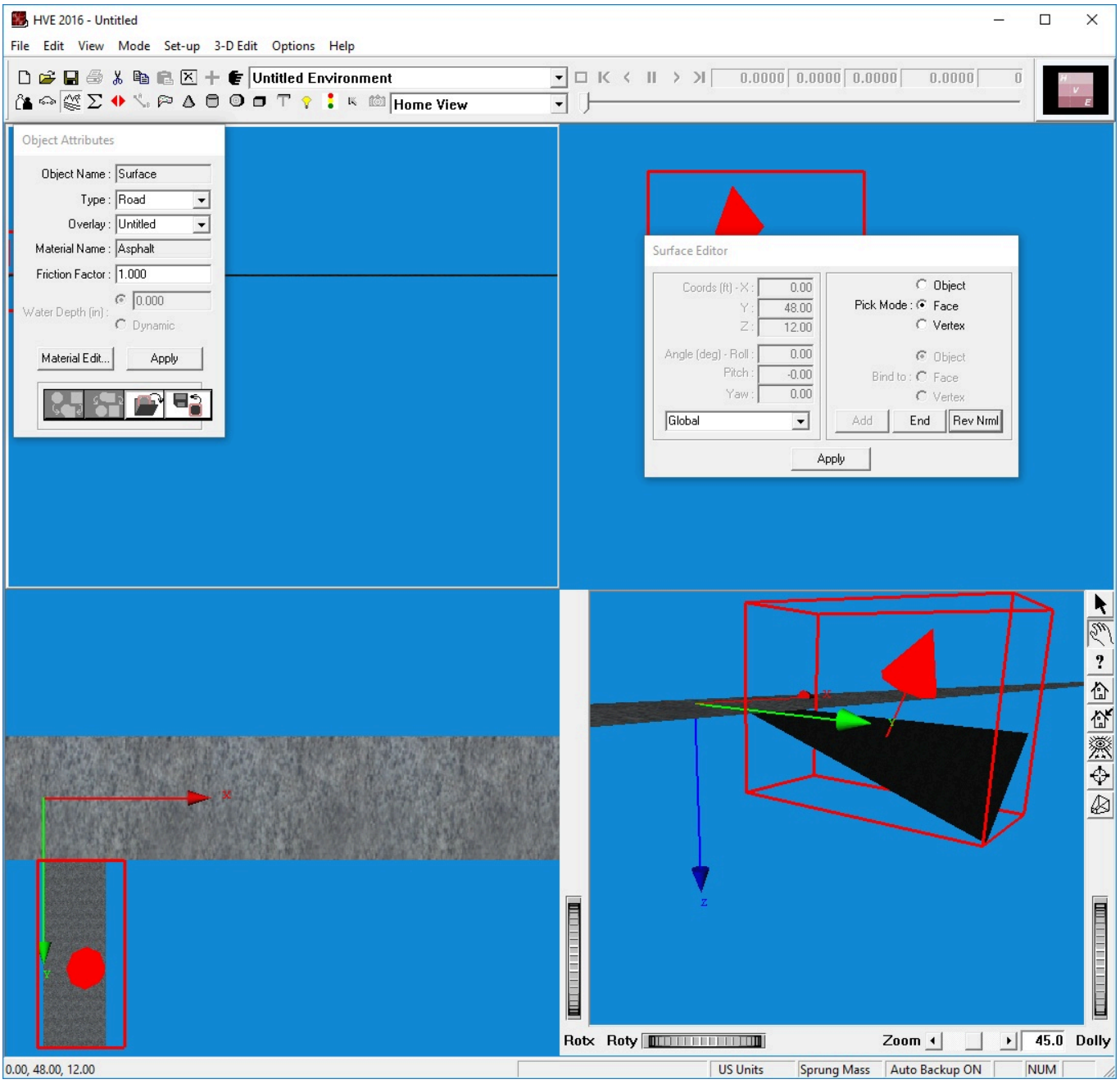


Figure 23-24 — Surface after reversing the direction of the surface normal.

Changing Object Hierarchy

Objects (surfaces, spheres, etc.) created using the 3-D Editor have a hierarchy; that is, they have an ordered relationship to each other (kind of like the relationship between a computer file system's directory, that directory's sub-directories, and each sub-directory's subdirectories, and so on, and so on...).

As a user of HVE, you don't need to be aware of object hierarchy, with two exceptions:

- Creating grouped objects (this is the subject of the next lesson in this tutorial), and
- Understanding how a vehicle's tire chooses which surface to drive on.

When an HVE-compatible simulation is executed, the tire model queries HVE to determine the elevation, surface normal and material attributes (e.g., friction) of the surface beneath the tire. If two or more surfaces exist beneath the tire, HVE uses the hierarchy to determine which surface to use. The subject of hierarchy is described in detail in Chapter 20, and will not be described further. However, we'll illustrate the issue with a simple example.

Let's create a raised median for our 4-lane highway:

1. Click on the Box Tool icon on the toolbar. The Box Editor is displayed, as shown in Figure 23-25.

A 10x10x10 box is placed at the earth-fixed origin. It is covered with the current Concrete texture, which is just fine.

1. Using the Box Editor dialog, change the current length to 1996 ft, the width to 4 ft, and the height to 1 ft.
2. Press Enter or click on Apply to implement your new dimensions.
3. Click on the 3-D Edit menu option and choose Material Color. The Material Editor dialog is displayed.

Let's change the color of the median to yellow (we're going to do some more with this median in Lesson 3):

1. Using the Material Editor dialog, click on the radio button for Diffuse Color. The color wheel dialog for Diffuse Color is now displayed.
2. Click in the bright yellow area of the color wheel. Close the color wheel dialog by clicking on the close dialog button.

Edit the color attributes:

1. Change the Diffuse Color to 1.00, the Specular Color to 0.50 and the Shininess to 0.50. The median becomes yellow, as shown in Figure 23-26.

There's only one problem with what we've done: if you execute an event on this road, you'll find that a vehicle drives right through the median; its tires don't respond to it. This happens because the road was created first, so HVE's surface lookup finds it first, and the tire never sees the median at all. Let's quickly solve this problem:

1. Using the Object Attributes dialog, click on the Type option list (see Figure 23-27) and change the Type from Road to Friction Zone.

Now the tire will respond to the median.

> **NOTE:** When the simulation is executed, it loads all Friction Zone objects before loading Road objects. Therefore, HVE's surface lookup returns the information about the median.

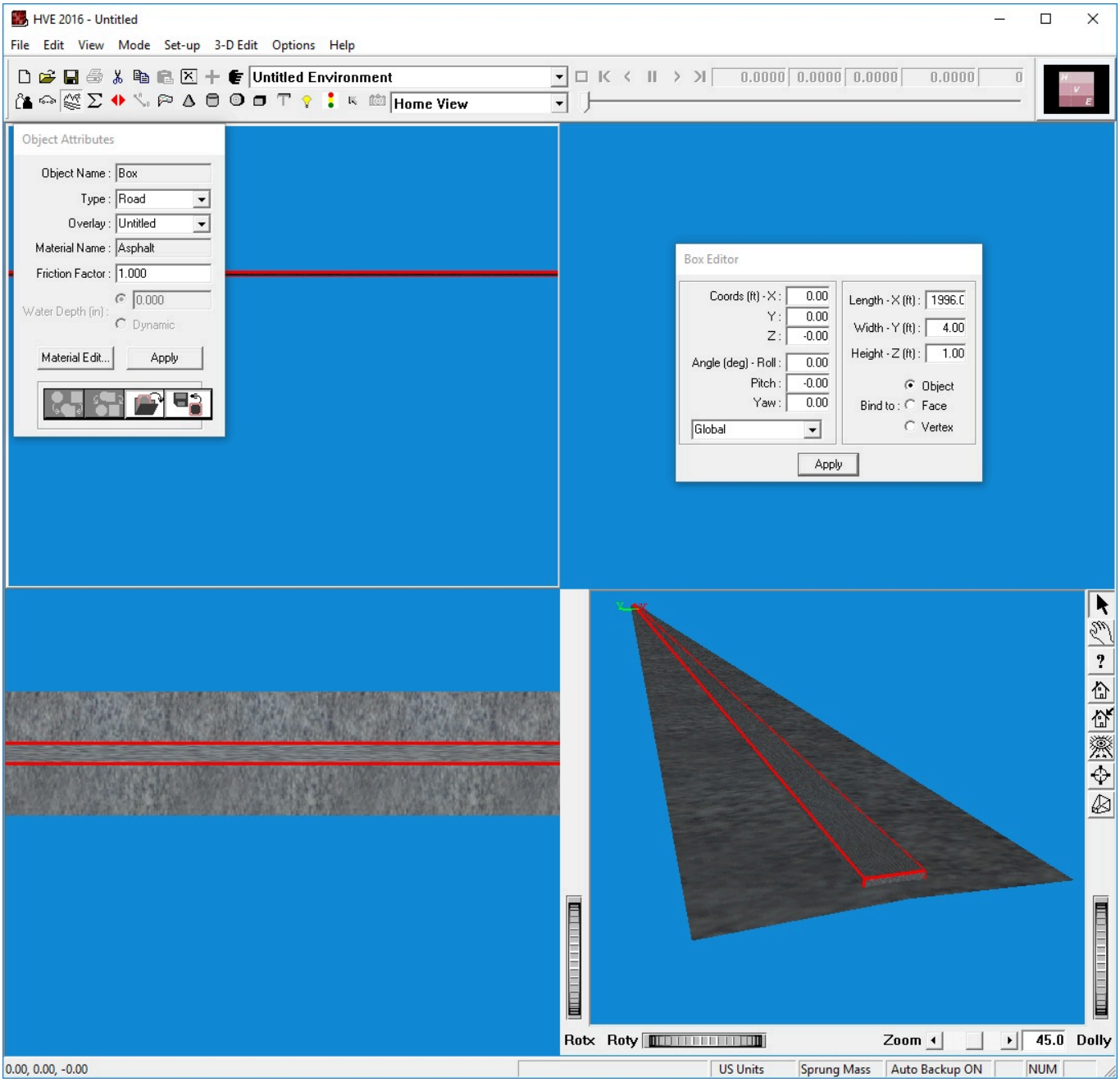


Figure 23-25 — Creating a raised median using the Box Tool.

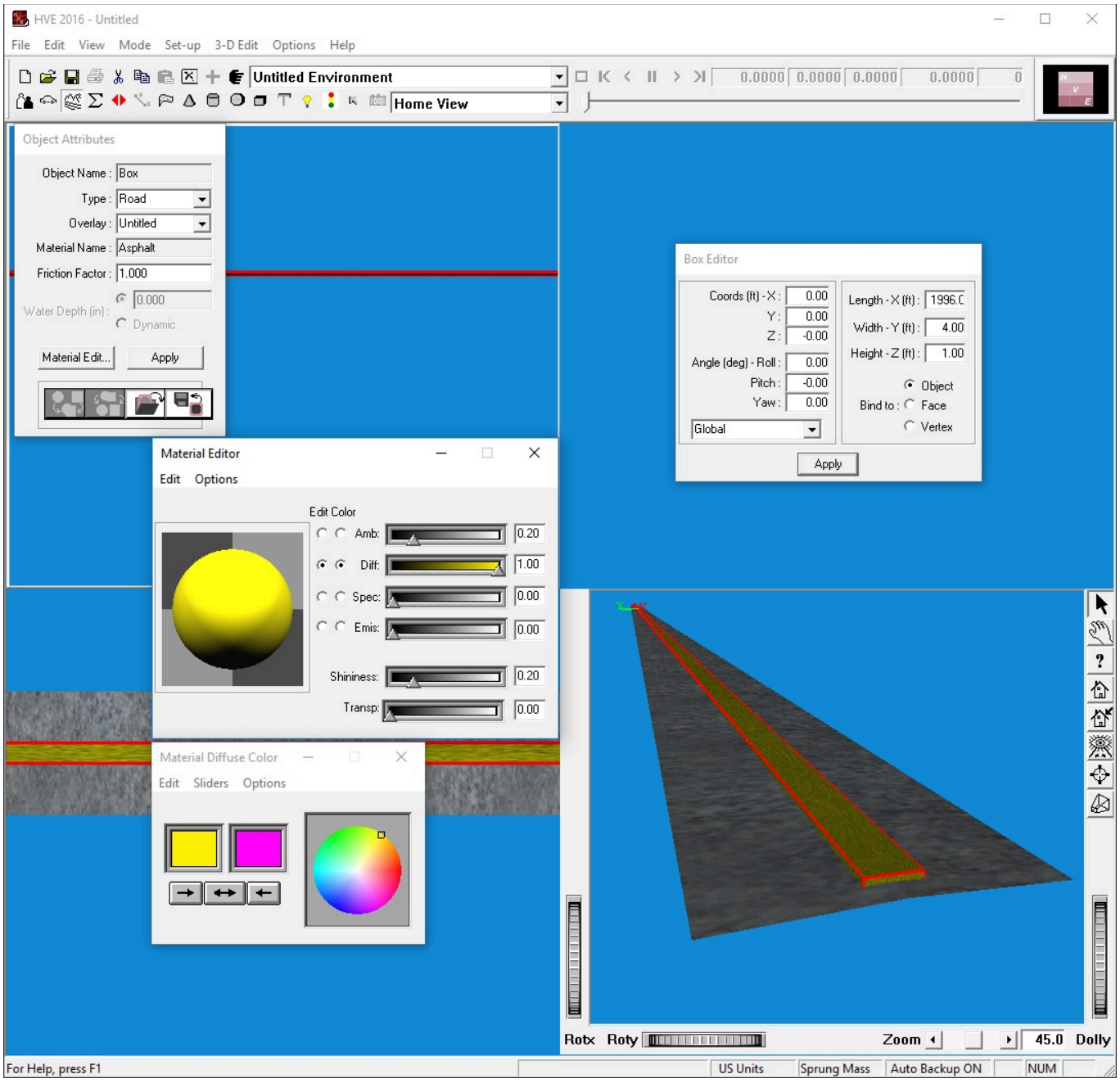


Figure 23-26 — Environment after adding a raised median and assigning yellow concrete material attributes.

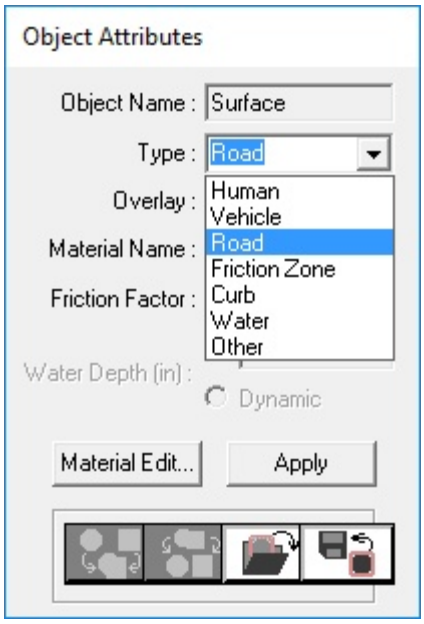


Figure 23-27 — 3-D Editor dialog with Object Type option list displayed.

Let's save our work for use in Lesson 3. Using the method described in Lesson 1 (refer back to Lesson 1 if you need a quick refresher), save your 3-D geometry file as a unique file named TutorEnvLesson2.

Summary

This concludes Lesson 2 — Editing a Surface. You should now be familiar with vertex editing, reversing surface normals and changing object hierarchy. You may either shut down HVE or continue with Lesson 3 — Using Groups.

Lesson 3 — Using Groups

This lesson describes the use of grouped objects in the 3-D environment. In this lesson, you will learn the following:

- Creating a Grouped Object
- Ungrouping a Grouped Object
- Saving a Grouped Object in a Library
- Using a Grouped Object from a Library

> **NOTE:** We assume HVE is up and running and that you have already completed Lesson 1 — Creating a Simple Road Surface, and Lesson 2 — Editing a Surface. If you have not finished those lessons, you should do that before continuing.

Grouped objects, or simply groups, are objects that are assembled from two or more individual objects. When manipulated, they act as a single unit. Examples of grouped objects include:

- Telephone Pole (vertical pole with one or more horizontal crossbars)
- Tree (vertical pole with branches and leaves)
- Sign (pole(s) with a rectangular placard)
- Traffic Cone (cone with rectangular, horizontal base)

- Fire Hydrant (vertical cylinder, several lateral cylinders, topped with a sphere)

One of the greatest benefits of using groups is that the entire object is positioned at once. For example, while placing a sign at an intersection, it is not necessary to position pole and placard separately. This is a great time saver, especially for complex objects composed of many parts. The use of grouped objects should become a common and frequent task while using the 3-D Editor.

In Lesson 3, we will start with the environment we created in Lesson 2:

1. If necessary, switch to Environment mode. The Environment Editor is displayed.
2. Click Add New Object. The Environment Information dialog is displayed.
3. Choose Open. The Environment Geometry File Selection dialog is displayed.
4. Select TutorEnvLesson2.h3d and press OK. The File Selection dialog disappears.
5. Press OK again. The Environment Information dialog disappears and the selected environment is displayed in the Environment 3-D viewer.

Spend a few moments manipulating the viewer to look at various views of the environment.

Now we're ready to edit the geometry. Editing 3-D geometry is always performed using the 3-D Editor, so let's start the editor:

1. Select 3-D Edit on the menu and Launch 3-D Editor. The 3-D Editor is started with our simple road surface displayed in each of the four viewers.
2. Arrange your desktop to your liking.

> **NOTE:** Remember that most of the editing is usually performed using the perspective viewer in the lower right and X-Y plane viewer in the lower left. With this in mind, you'll probably want to position your dialogs in the top portion of the desktop.

Now, we're ready to begin.

Creating a Grouped Object

In Lesson 2, we created a simple raised median. Let's extend that object by adding a raised grass section in the center and circular concrete end caps at both ends. We'll also make it shorter and copy it to both sides of the Tee intersection.

Start by making the median shorter:

1. Arrange your viewer so the median is easily selectable, as shown in Figure 23-28.
2. Click on the median. A bounding box is displayed around it, indicating it has been selected. The Box Editor dialog is also displayed.
3. Change the length to 496 from 1996 ft (remember to press Enter). The median length is updated.

Now, let's put the concrete end caps on both ends:

1. Click on the Cylinder Tool icon on the toolbar. The Cylinder Editor dialog is displayed.
2. Change the default radius to the desired value, 2 ft, and the default length to 1 ft (remember to press Enter).
3. Edit the cylinder's position by changing its X coordinate to 248 ft. The cylinder is displayed at one end of the median, but its orientation is incorrect.

4. Change the default roll angle, 0 degrees (about the X axis), to 90 degrees.

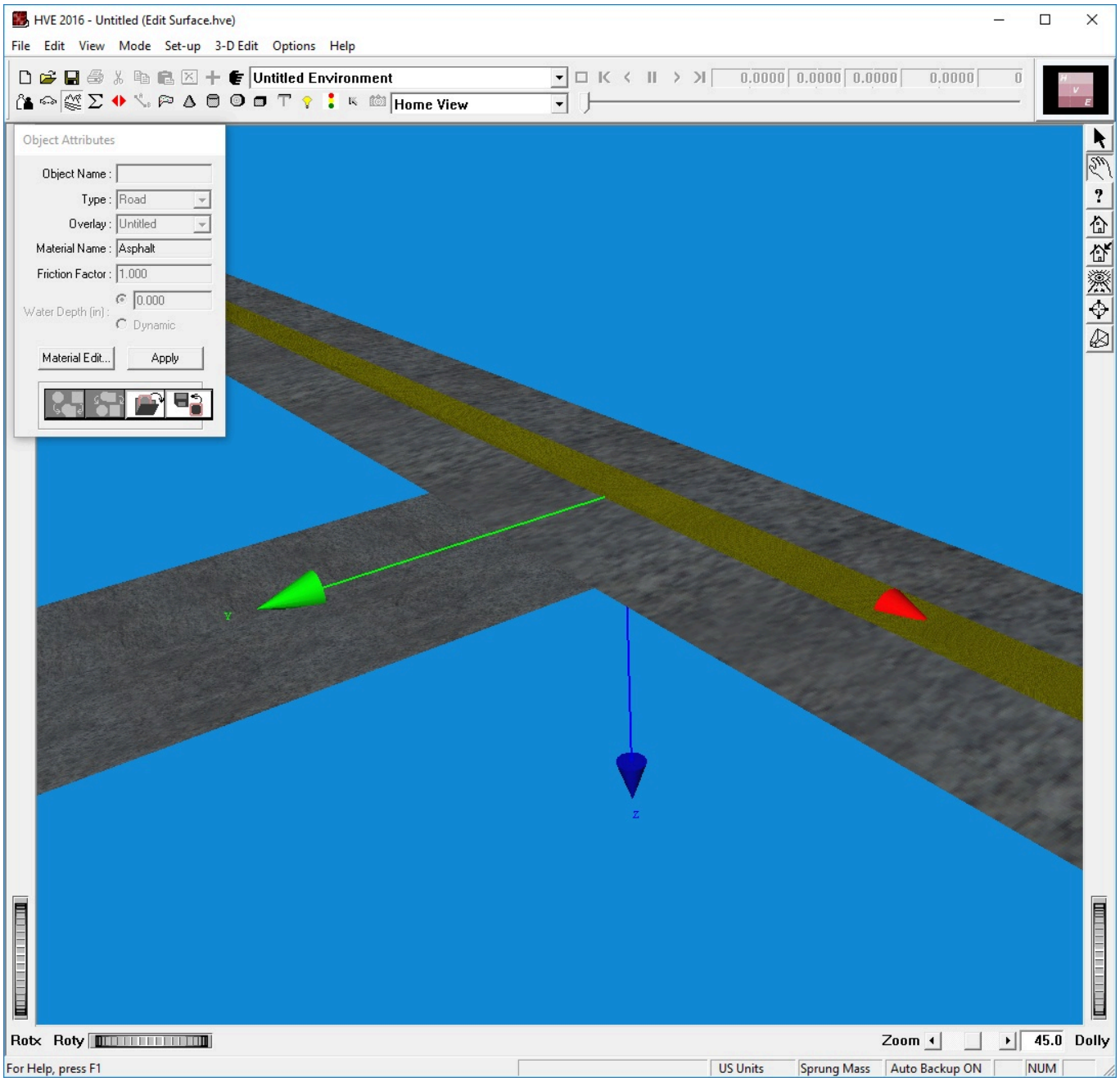


Figure 23-28 — Environment from Lesson 2.

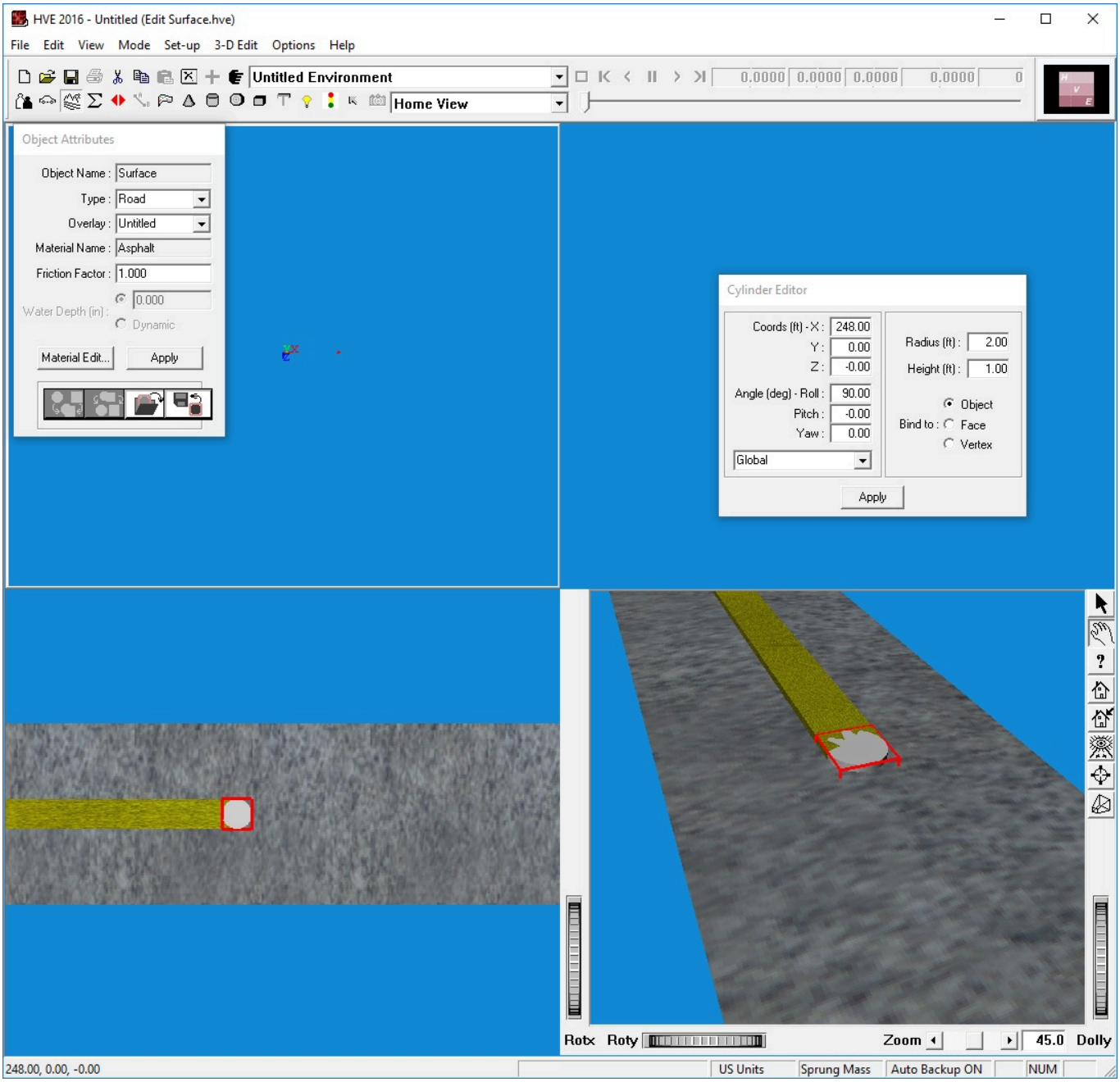


Figure 23-29 — Cylinder at end of median, before changing the material to Concrete.

If the cylinder does not have Concrete material attributes, perform the following steps:

1. Select 3-D Edit and choose Material Texture. The Texture Editor is displayed.
2. Select Cement from the list of available textures. A sample of the concrete texture is displayed in the Texture window.
3. Press Accept. The concrete texture is applied to the object.

Change the color of the concrete using the Material Editor dialog:

1. Select 3-D Edit and choose Material Color. The Material Editor is displayed.

2. Click on the radio button for Diffuse Color. Click in the bright yellow area of the color wheel dialog that appears. Close the color wheel dialog.
3. Change the attributes as follows: set the Diffuse color to 1.0, the Specular color to 0.5 and the Shininess to 0.5. The color on the end of the median is updated.

> **NOTE:** These values are simply the result of finding attributes that are pleasing to the user. They need not be exact.

The concrete end cap now matches the concrete median.

Let's put a concrete end cap on the other end:

1. Click on the Cylinder Tool icon on the toolbar. The Cylinder Editor dialog is displayed.
2. Change the default radius to the desired value, 2 ft, and the default length to 1 ft, followed by Enter.
3. Edit the cylinder's position by changing its X coordinate to -248 ft. The cylinder is displayed at the other end of the median. It also has the current texture attribute (Concrete).
4. Change the default roll orientation to 90 degrees from 0 degrees.

The median now includes caps at both ends. Now, let's add the grass:

1. Click on the Box Tool icon on the toolbar. The Box Editor dialog is displayed.
2. Change the default length to the desired value, 496 ft, the default width to 3 ft, and the default height to 1.5 ft.
3. Press Enter to display the new box.

Now, let's make it grass:

1. Click on 3-D Edit on the menu and choose Material Texture. The Texture Editor is displayed.
2. Select Grass2 from the available textures list. A sample of the Grass texture is displayed in the Texture window.
3. Change Repeat X to 200 from 1.0; the Repeat Y value does not need to be changed.
4. Press Accept. The texture is applied to the grass median. The median now has a grass strip down the middle. However, the median looks slightly yellow.
5. Using the Material Editor's Diffuse color wheel as discussed in previous steps, click in the green area. The color is updated to the selected color, as shown in Figure 23-30.

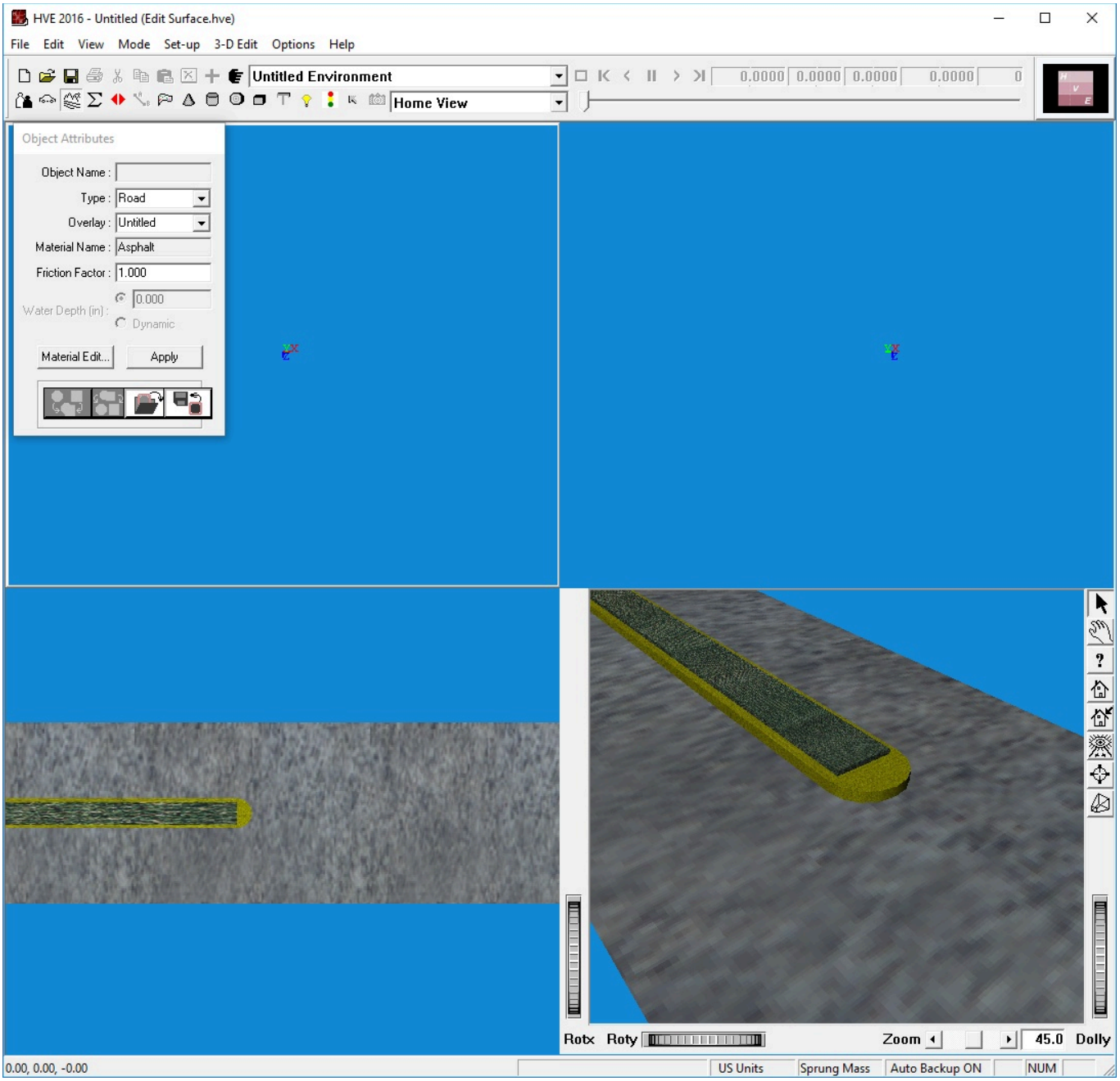


Figure 23-30 — Concrete median with grass.

We now have a good looking grass median, but it's in the wrong place; we want two medians on either side of the intersection. We have two options: we could move each individual part of the median to the desired location, or (much better) we could make these individual parts into a group, and then move the whole thing at once.

Let's create a group:

1. Hold down the Shift key and click on the grass, the concrete median (only the curb portion is visible after adding the grass) and both end caps. As you click on each part, a bounding box is displayed around it, indicating it has been selected.

> **NOTE:** You will need to manipulate the viewer to be able to click on all the objects.

After selecting the grass, the concrete median and both concrete end caps, the viewer should appear as shown in Figure 23-31.

Now let's create the group:

1. Using the 3-D Editor's Group Tools (see Figure 23-32), click on the Group icon. The individual bounding boxes are removed and a single bounding box is displayed around the entire object, indicating it is now a group.

Ungrouping a Grouped Object

Occasionally you will need to disassemble a grouped object into its individual parts. Possible reasons include:

- You wish to change the color or texture of a single part.
- You wish to edit the size or shape of the object.
- You wish to remove a part from the grouped object.

To ungroup the grouped median into its individual parts:

1. Click the Ungroup icon (see Figure 23-32).

The bounding box disappears, indicating the object has been deselected. In fact, it is no longer even a single object. Let's make it a grouped object again:

1. Hold down the Shift key and click on the grass, concrete median, and both end caps. As you click on each part, a bounding box is displayed around it, indicating it has been selected.
2. Click on the Group icon (see Figure 23-32). The individual bounding boxes are removed and a single bounding box is displayed around the entire object, indicating it is now a group.

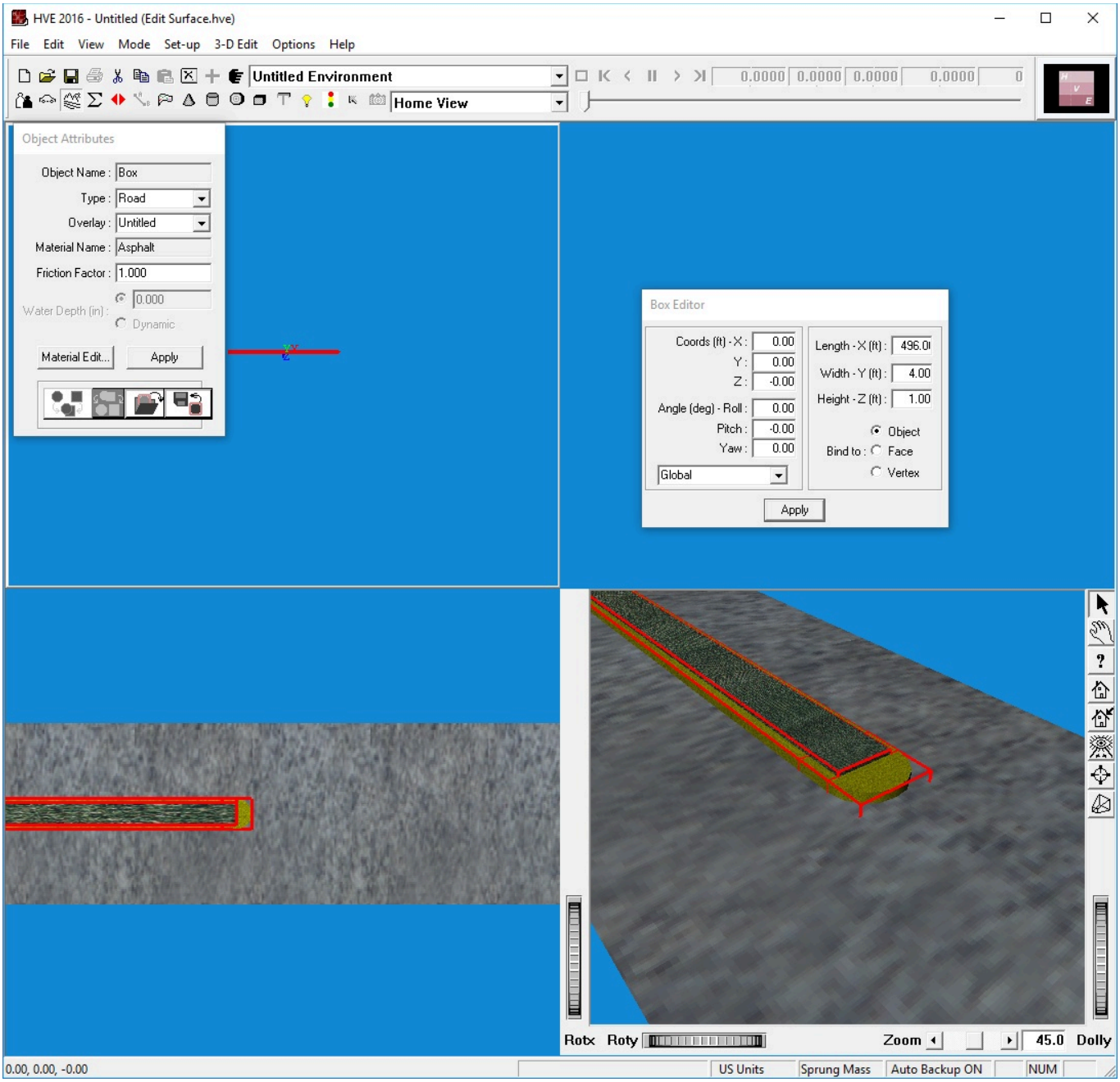


Figure 23-31 — Selection of the individual objects making up the concrete median. Note a red bounding box surrounds each individual object (the view is zoomed in on the end of the median).

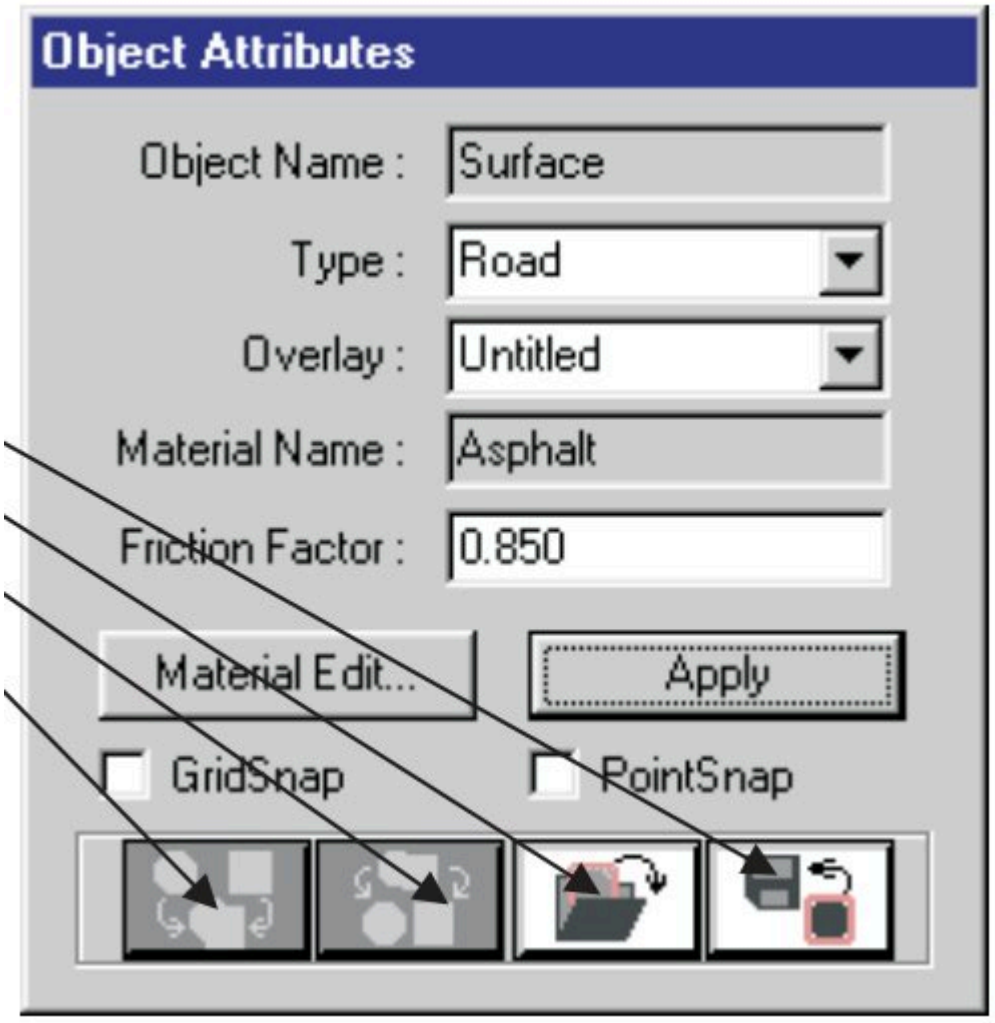


Figure 23-32 — 3-D Editor Group Tools (Group, Ungroup, Open and Save).

Saving Grouped Objects in a Library

This median is a great object! We might even want to use it again in other environments. We can make it available for future use in any environment by adding it to our Highway Furnishings Library. To add the median to our library, perform the following steps:

1. Confirm the object is selected by the presence of a bounding box. If necessary, click on the median to select it.
2. Click on the 3-D Editor's Save Group icon (see Figure 23-32). The Highway Furnishings File Selection dialog is displayed (see Figure 23-33).
3. Click on the Save as type option list and choose HVE.

> **NOTE:** HVE is probably already displayed; although several formats are available for opening (i.e., importing), HVE is the only currently supported format for saving.

1. Enter a filename for the median: enter Median.
2. Press OK.

The median has been saved in our Highway Furnishings Library for use in future environments.

> **NOTE:** Objects are saved according to their current location relative to the origin. This fact is important when the object is later selected from the library and positioned in the environment: if the object was located 50 ft from the origin when it was saved, it will be pasted 50 feet from the selected coordinates when it is selected from the library and positioned! Thus, you normally want to move the object to the origin before saving it in the library.

Let's finally place the median at the desired location:

1. Confirm the object is selected by the presence of a bounding box. If necessary, click on the median to select it.
2. The Editor dialog displays the coordinates of its local origin relative to the earth-fixed origin. Its current coordinates are still 0,0,0 (where we created it). Let's move it:
3. Change the current X coordinate to -270. After pressing Enter, the median moves to the desired location (see Figure 23-34).

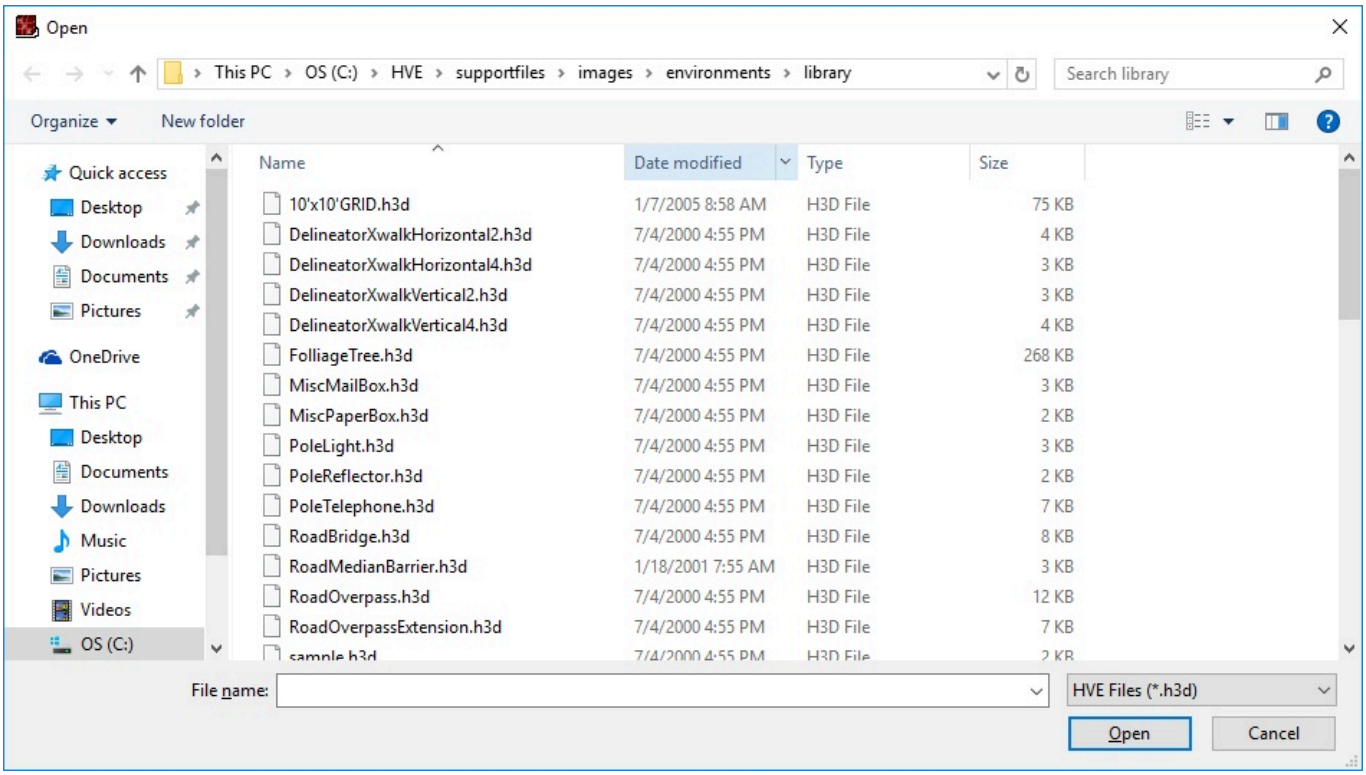


Figure 23-33 — Object Selection dialog, used for saving and opening objects in the Highway Furnishings Library.

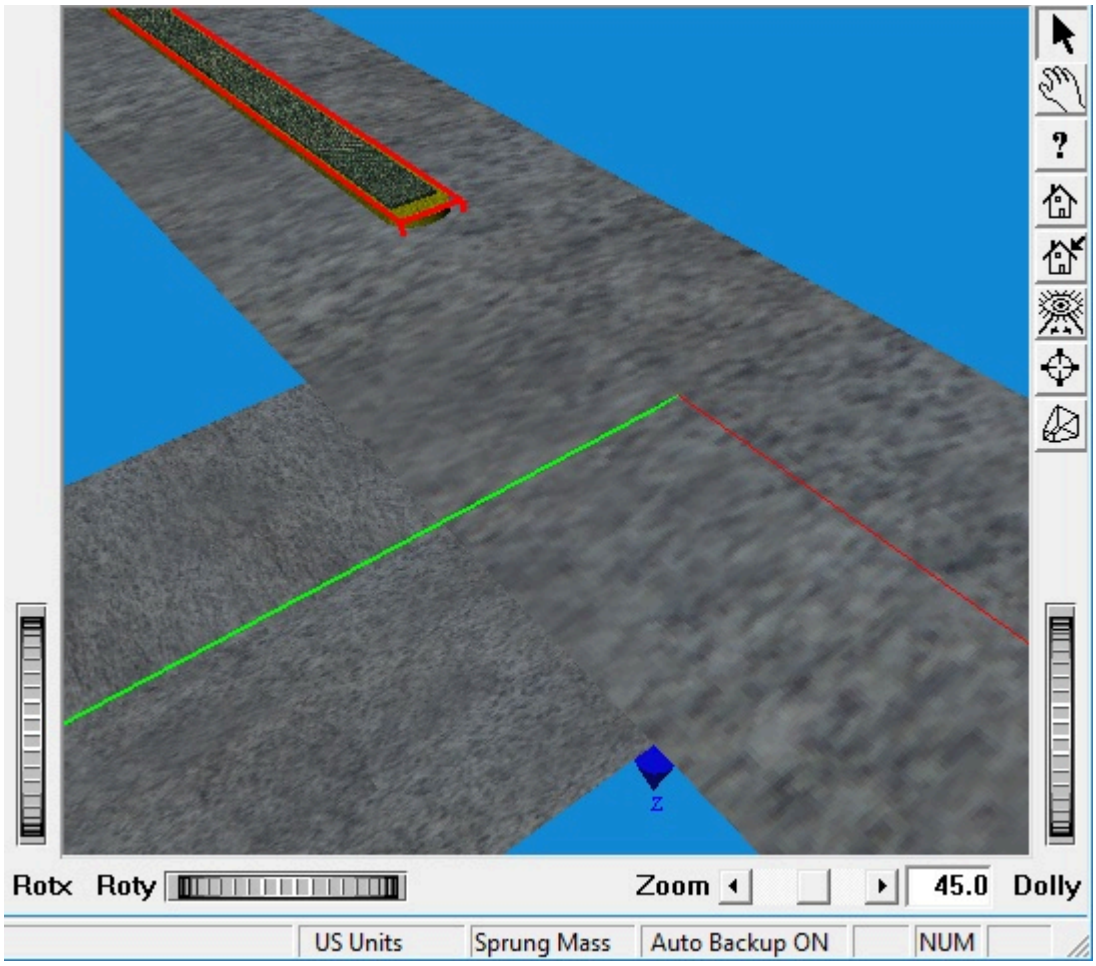


Figure 23-34 — Our environment after positioning the first median section.

Using Library Objects

Now we need an additional copy of the median for the other end of our environment. We could create it from scratch, but it's obviously easier to grab it from our Highway Furnishings Library and simply place it where we want it. Let's proceed:

1. Click on the 3-D Editor's Library Open icon (see Figure 23-32). The Highway Furnishings File Selection dialog is displayed (see Figure 23-33).
2. Click on the file type option list and choose HVE.

> **NOTE:** HVE is probably already displayed. You can use almost any popular modeling package to create your library objects.

1. Select Median.h3d from the file selection list.
2. Press OK.

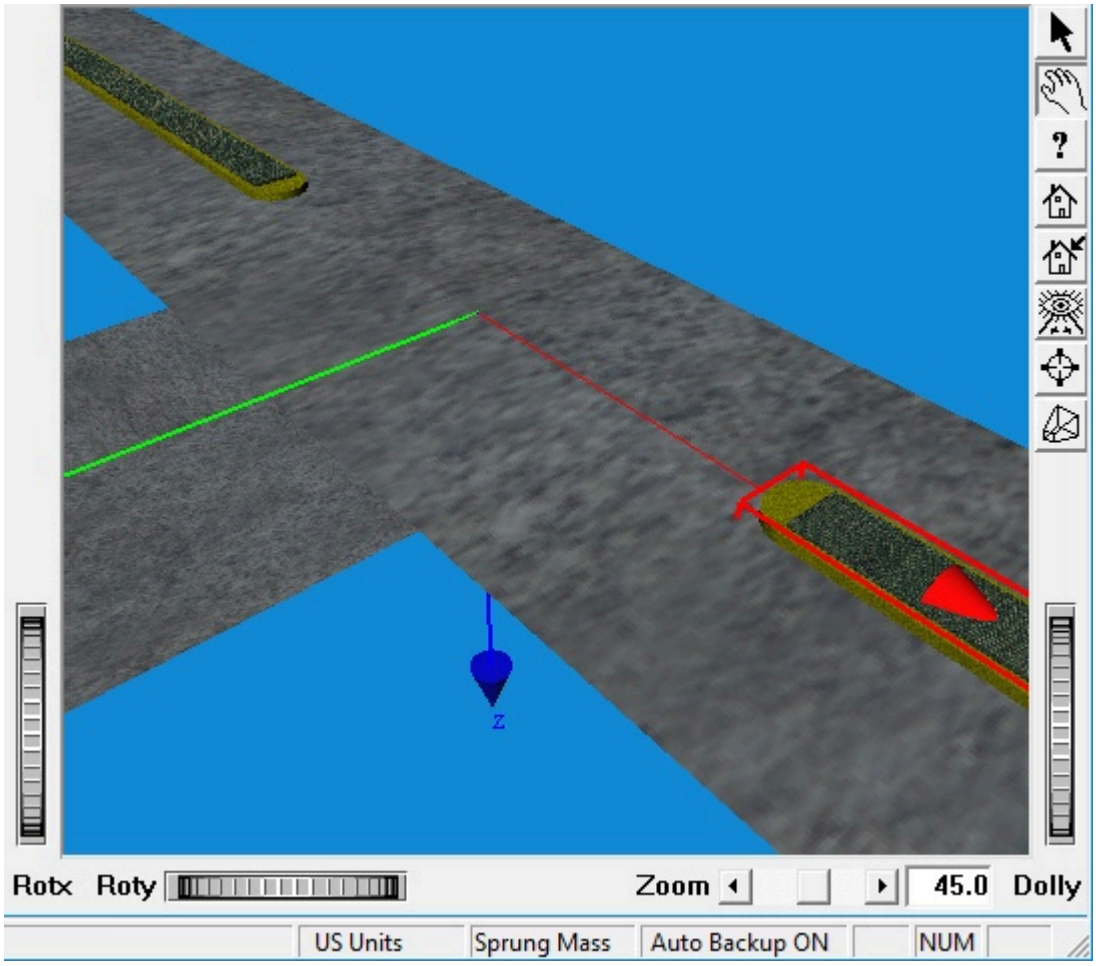


Figure 23-35 — Our environment after positioning the second median section.

The median is displayed at the earth-fixed origin.

> **NOTE:** Remember the object was saved according to its current location relative to the origin. Thus, it is initially redisplayed according to that same earth-fixed location.
>
> **NOTE:** This is why you almost always want to move an object to the origin before saving it in the Highway Furnishings Library!

Let's position the median at the correct location:

1. Confirm the object is selected by the presence of a bounding box. If necessary, click on the median to select it.
2. The Box Editor dialog displays the coordinates of its local origin relative to the earth-fixed origin. Its current coordinates are still 0,0,0 (where we created it). Let's move it:
3. Change the current X coordinate to 270 and press Enter.

The median moves to the desired location (see Figure 23-35).

Ending the Session

The last thing to do is end the session. To save your environment by a unique filename, perform the following steps:

1. First, close the 3-D Editor by selecting 3-D Edit and choosing Close 3-D Editor. The 3-D Editor is closed, revealing the Environment Editor and Viewer with your road surface.
2. Click on Object Info. The Environment Information dialog is displayed (shown earlier in Figure 23-1), allowing us to view and edit the current environment attributes. Note we also use this dialog for opening and saving 3-D geometry files.
3. Click on Save As. The File Selection dialog is displayed, allowing us to save our geometry using the desired file format and filename (shown earlier in Figure 23-14).
4. Click on the Format option list to display the available file formats, and choose HVE.
5. Enter a filename for the 3-D geometry file. Enter: TutorEnvLesson3.

> **NOTE:** There is no need to supply an extension, because HVE will automatically supply the .h3d extension.

1. Click OK in the File Selection dialog. The file is now saved in the `...\hve\supportFiles\images\environments` subdirectory.
2. Click OK in the Environment Information dialog.

We are now ready to shut down HVE. This is done using the File menu:

1. Choose the File menu option in the main menu bar, and click on Exit. You are asked if you would like to save your changes.
2. Choose No.

> **NOTE:** You saved your work in a unique 3-D geometry file, named TutorEnvLesson3.h3d. There is no need to save your case.

HVE has now shut down.

Summary

This concludes Lesson 3 — Using Groups. You should now be familiar with creating and disassembling grouped objects, saving groups in the Highway Furnishings Library, and selecting objects from the Library and placing them in the scene.

Converted and updated from the legacy HVE User's Manual (Seventh Edition, Jan 2006), Chapter 23; revised to match the current version of HVE, 2026-07-05.

Section Nine: Video Output

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Section Nine "Video Output", revised to describe the current HVE video interface.

HVE has an integrated video output interface. This interface makes it quick, simple and convenient to record a simulation as a video. This section of the manual describes HVE's video interface.

*(updated: The original edition of this section was written when HVE drove external video tape recorders (VTRs) over a serial connection and through add-in video-out hardware. Current HVE records simulation movies directly to compressed **AVI** (or MPEG) files on disk using standard Windows codecs, and can also write individual frame image files. The tape-deck-specific material is retained below where it is still informative, but is clearly flagged as legacy.)*

Users new to the subject of video output will find this section both informative and important. Producing high-quality video requires knowledge and experience. This section includes several tips aimed at improving the resulting output.

General Procedure

To use the HVE Video Interface, perform the following steps:

1. Use the Video Options (Video Set-up) dialog to select the video format (movie and/or single frames), movie size, frame rate and an optional video compressor (codec).
2. Use the HVE Event Editor to create one or more simulation events.
3. Use the HVE Playback Editor (Video Creator) to combine the events into a single Playback Window.
4. Use the Playback Controller to choose the Video Source (Playback Window or a previously recorded movie file) and Video Destination (Video File or Playback Window).
5. Use the Playback Controller to record the results from the selected Video Source to the selected Video Destination.

Overview of Section Nine

This section provides a detailed explanation of the process of creating and editing 3-D videos produced directly from HVE simulations:

- Chapter 26 — Video Basics provides an overview of video basics. This is an informative chapter for those new to the subject of computer-generated video. Much of its broadcast/tape material is legacy reference, but the tips on composing and recording simulation video remain useful.
- Chapter 27 — Using The HVE Video Interface provides a detailed overview of the HVE Video Interface.

See also Lesson 4 (Creating a Video) of the HVE Tutorial.

Chapter 26 — Video Basics

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 26, pages 26-1 through 26-14, revised to describe the current HVE software.

This chapter provides a general overview of the process of producing video output from HVE (many sections of this chapter apply to any video device). Beginners will learn a lot about the subject, while users with advanced knowledge will find much of this chapter unnecessary. However, all users should understand the information contained in this chapter before attempting to produce their first video.

*(updated: Current HVE records video directly to compressed AVI/MPEG movie files (or single-frame image files) using standard Windows codecs — it no longer drives external video tape recorders. The sections below on tape formats, broadcast standards, color encoding and recording media are retained as **legacy reference** for users producing physical video media from their rendered movies; the sections on recording methods, compression and the Tips and Suggestions remain directly applicable.)*

Overview

This chapter contains the following sections:

- Why Video
- Computer Graphics vs Video
- Typical System Layout *(legacy)*
- Basic Procedures
- Recording Methods
- Video Compression
- Video Communication Set-up *(legacy)*
- Preparation of Recording Media *(legacy)*
- Broadcast Standards *(legacy)*
- Color Encoding *(legacy)*
- Video Formats and Recording Quality *(legacy)*
- Tips and Suggestions

Why Video?

Video is different from photographs or diagrams in that it shows motion. The benefits from this fact are numerous:

- **Realism** — The life-like appearance and motion are easily understood by non-technical individuals, and also illustrate how the sequence appeared from any perspective.
- **Time vs Distance Studies** — The ability to visualize the motion allows the researcher to illustrate relative position vs time.
- **Visibility and Avoidability Studies** — The ability to visualize position vs time allows the user to understand more about a driver's opportunity to avoid a crash.

The fact that video motion is displayed at a fixed frame rate also makes it possible to produce and confirm real-time output. Thus, video is useful as an analytical tool as well.

Video is perhaps most useful because of its universal viewability: a movie file sent to your clients may be viewed easily and conveniently.

(updated: the original text emphasized the widespread availability of VCRs; today the equivalent is that an AVI or MPEG file plays on any computer, and is easily converted for DVD, web or courtroom presentation.)

Computer Graphics Vs Video

Computer-generated graphics differs from broadcast video in a number of ways. Understanding these differences will help users produce better results using the HVE Video Interface. Some of the differences include:

- **Interlacing** — In most video standards, each row of pixels is drawn every other time the monitor is refreshed. This is called *interlacing*. In contrast, computer graphics monitors typically draw every row of pixels every time the screen is refreshed. This is called *non-interlaced* (progressive).
- **Broadcast Standards** — The world historically employed three different analog video standards: NTSC, PAL and SECAM. These standards are described later in this chapter (see Broadcast Standards).
- **Tape Formats** — Several video tape recording formats existed, such as VHS, S-VHS and Betacam. These formats are described later in this chapter (see Video Formats).
- **Color Encoding** — Several color-encoding methods are used, including RGB (component), YUV (component), YIQ (component), YC (several types) and composite video. These methods are described later in this chapter (see Color Encoding).

Typical System Layout *(legacy)*

A typical (legacy) video system layout included the computer, video hardware, video interface software, simulation software, animation controller, video recorder, VHS deck and television monitor.

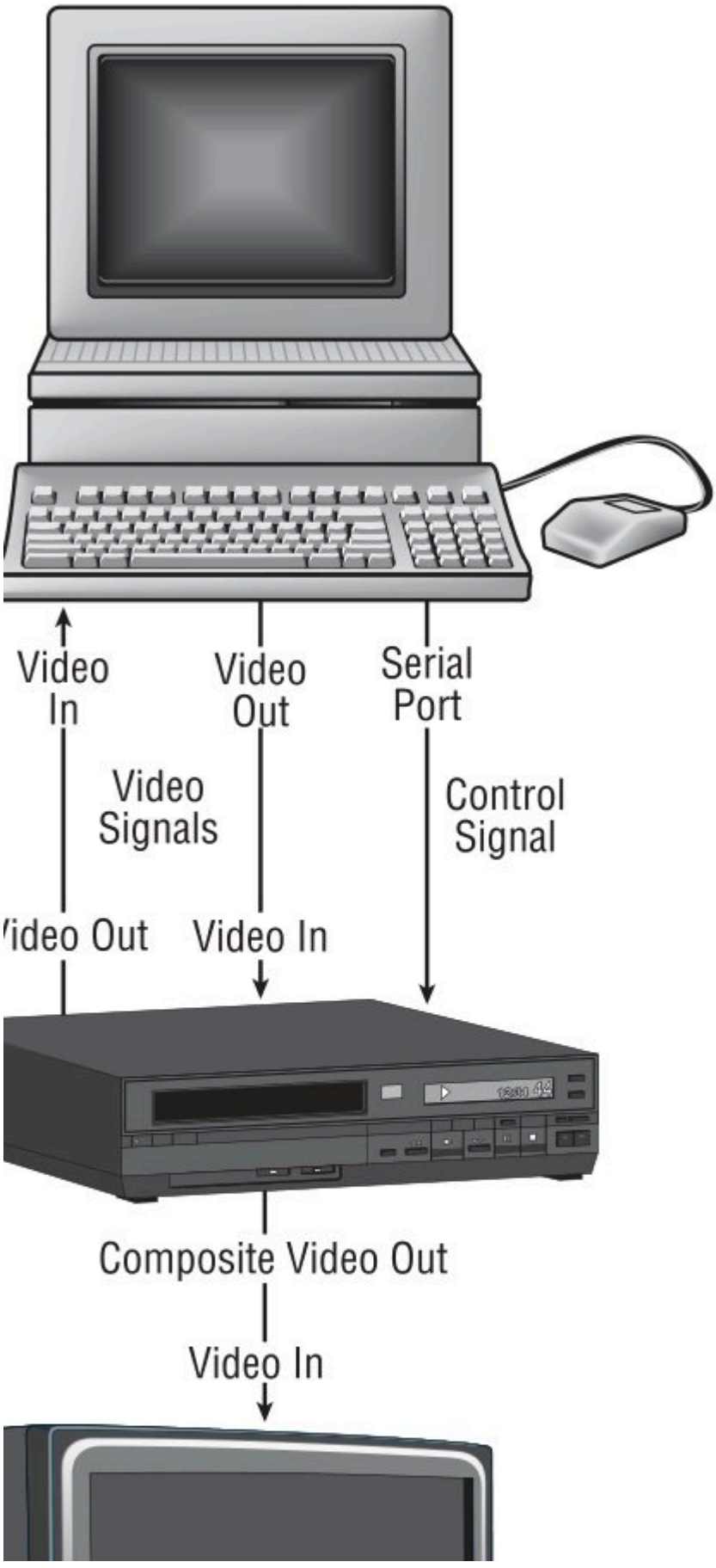

Video In
Video Out
Serial Port
Video Signals
Control Signal
ideo Out
Video In
Composite Video Out
Video In



Figure 26-1 — Typical System Layout. The control signal from the serial port is used for single-frame recorders and is not required for systems using a video compressor.

(updated: a current HVE video "system" is simply the computer itself — HVE renders the playback window to a movie file on disk. No external recorder or serial control connection is used.)

Basic Procedures

No matter what system configuration is used, recording a simulation sequence involves the same basic procedures:

1. Set up the video options (format, size, frame rate, compressor).

> **NOTE:** You should only need to do this once; the settings are stored in HVE's configuration file.

1. Using HVE, create the simulation sequence to be recorded.
2. Record the video (to a movie file in current HVE; to tape on legacy systems).
3. Replay the movie to confirm it is properly recorded.
4. Distribute copies to your clients (on legacy systems this meant copying the master tape from the professional video system to VHS).

Refer to Chapter 27, Using the HVE Video Interface, for detailed, step-by-step procedures and examples.

Recording Methods

The three basic methods of recording computer graphics on video are:

- Recording Live From Screen
- Single-frame Recording
- Real-time Recording

Each of these methods has advantages.

Recording Live From Screen

Recording live from the computer screen is the simplest form of recording (e.g., a screen-capture utility, or on legacy systems a VCR wired to the display output). It requires no special recording interface: anything displayed on the computer screen is transferred to the recording. However, there is no control over frame rate; thus, there is no constant

relationship between the time domain in the recorded sequence and real time. Because of this major limitation, recording live from screen is not generally useful for recording real-time simulations.

Single-frame Recording

Single-frame recording involves the controlled transfer of individual computer-generated images to the recording medium one frame at a time. The advantage of single-frame recording is that the frame rate is carefully controlled (normally 25 or 30 frames per second, depending on the video standard used for recording). Thus, when results recorded using single-frame recording are played back, the sequence is shown in real time (i.e., 1.0 second of recorded simulation is exactly one second on a stopwatch).

(legacy)

On tape-based systems this process required special video hardware and software and a computer-controlled, frame-accurate video tape recorder (VTR), and owing to the required pre-roll and post-roll it took about 10 seconds to record each frame (about 25 minutes to record a 5-second simulation sequence).

*(updated: current HVE offers a **Single Frames** video format option in the Video Options dialog which writes each rendered frame to an individual image file — useful for producing frame-accurate stills or for assembling video in an external editing package.)*

Real-time Recording

Real-time recording also involves the controlled transfer of individual computer-generated images, but the individually rendered frames are first transferred to the computer's hard disk, where they are stored in a movie file. After all the frames are rendered and stored, the file plays back in real time. This is the method used by current HVE: each frame of the playback sequence is rendered and appended to a compressed AVI (or MPEG) file at the selected frame rate; the resulting movie is then played back in real time in the Playback Window (or any media player). The sequence may be previewed on the computer's monitor before it is distributed.

To achieve real-time frame rates for complex scenes, the frames are compressed. A video compressor reduces the amount of storage required for each frame by eliminating or reducing the number of pixels having the same color and intensity. The trade-off is a slight loss of quality, although modern video compression algorithms produce excellent results. Both hardware and software compression are available; current HVE uses the software codecs installed in Windows.

Video Compression

Rendering a single frame of video takes a while. Even a powerful computer graphics system cannot render a complex scene fast enough to display in real time (i.e., 25–30 frames per second). One solution to this limitation is to compress the data in each frame of video and store the result on the computer's hard disk. Once this is done, the frames may be played back in real time from the disk to the screen.

Several types of video compression exist. Two of the most popular families are:

- **MPEG** (Moving Picture Experts Group)
- **JPEG** (Joint Photographic Experts Group)

> **NOTE:** An AVI (Video for Windows) type video compression system is standard equipment on Windows computers. *(updated: HVE enumerates the codecs installed in Windows through the standard compressor-selection dialog; "Full Frames (Uncompressed)" is used if no compressor is chosen. The legacy manual recommended the Cinepak codec, which is obsolete — use any modern codec installed on your system.)*

The selected compressor is chosen using HVE's Video Options dialog (see Chapter 27).

Video Communication Set-up *(legacy)*

Before any video input or output could be performed on tape-based systems, the computer's video hardware had to be configured to communicate with the video device (e.g., video tape recorder). This process was called *video communication set-up*, and the procedures were specific to the video output device installed in the computer. *(updated: no such set-up is required in current HVE — output goes directly to a disk file.)*

Preparation of Recording Media *(legacy)*

The following procedures were recommended when recording to video tape:

- Use only high-quality tape. Be sure the correct type of medium is used. Never use regular VHS tape for S-VHS recording.
- Avoid the use of thin-based tape, such as tape longer than T-120.

> **NOTE:** A typical simulation analysis requires less than 5 minutes of tape; thus, it is convenient to use the shortest tape available.

- Prior to using the tape, pack the tape by running the VTR in fast forward to the end of the tape, then rewinding it.
- Start recording several minutes into the tape. Avoid recording in the leader area.
- Avoid reusing tapes that have already been used for recording. Use the tape only for limited playback, using it as a master copy for duplication onto regular VHS tapes for distribution to your clients.

> **NOTE:** Legacy HVE video systems used special video formats requiring professional broadcast video hardware. Because clients generally did not possess such hardware, a consumer-quality VHS duplication of the master tape was normally produced for distribution.

- Keep the VTR's mechanism clean and well maintained, according to the manufacturer's recommendations. If you suspect a tape is worn, do not insert it into the VTR; it may damage the mechanism.

Broadcast Standards *(legacy)*

Broadcast standards, or video timing formats, are ways of encoding video information for broadcast to television receivers. The three recognized analog broadcast standards were:

- **NTSC (National Television Systems Committee)** — Used in all of North and South America, except Brazil, and much of East Asia.

- **PAL (Phase Alternated by Line)** — Used in the United Kingdom and Western Europe, except France, and in parts of East Asia, including Australia.
- **SECAM (Séquentiel Couleur avec Mémoire)** — Used in France, Eastern Europe, the Near East and Mideast, and in parts of Africa and the Caribbean.

NTSC uses a total of 525 horizontal lines per frame, with two fields of 262.5 lines each. Each field refreshes at 60 Hz (actually 59.94). NTSC encodes brightness, color and synchronizing information in the same signal.

PAL uses a total of 625 horizontal lines per frame, with two fields of 312.5 lines each. Each field refreshes at 50 Hz. PAL encodes brightness, color and synchronizing information in the same signal, but in a different way from NTSC.

SECAM transmits the same number of lines at the same rate as PAL, but transmits each color difference signal on alternating lines, using frequency modulation for the subcarrier.

> **NOTE:** The actual lines of active video are fewer than the total lines noted above. Typically, NTSC displays 485 lines and PAL displays 575 lines. The remainder are used for synchronizing and other information.

(updated: analog broadcasting has been replaced by digital television; the 30 fps (NTSC) and 25 fps (PAL) frame-rate conventions survive in current video work. HVE's Video Options dialog offers movie sizes from the default window size through DVD, 480p, 720i, 1080i, 1080p and 4K, with a selectable frame rate.)

Color Encoding *(legacy)*

Color encoding refers to how the video signal is generated and conducted between devices (e.g., computer and recorder). The common methods are:

- RGB (3-component signal)
- YUV (3-component signal)
- YIQ (3-component signal)
- YC (2-component signal)
- Composite Video (1-component, single signal)

RGB

RGB is the color encoding method used by most computers, as well as some professional-quality video cameras. The three colors red, green and blue are generated separately and carried on three separate wires.

YUV

YUV is a 3-signal (and 3-wire) method used by PAL and some video cameras and VCRs. Like RGB, YUV is also a component color-encoding method, but instead of conducting individual colors, YUV carries the brightness, or *luminance*, on the Y signal, and color, or *chrominance*, on the U and V signals. YUV color encoding can be recorded digitally, according to the CCIR 601 standard. This recording technique is referred to as D1.

YIQ

YIQ is a 3-signal (and 3-wire) method typically used by the NTSC format. Like YUV, it encodes luminance onto one signal, Y. It encodes color onto two signals, called I and Q (intermodulation and quadrature, respectively), that differ

from the method used by YUV.

YC (YC-358, YC-443, or S-Video)

YC is a two-wire signal that uses luminance (Y) and chrominance (C). YC results when I and Q are combined into a single color signal, C. YC-358 is the most common NTSC version of this format; YC-443 is the most common PAL version. These formats are also known as S-Video, and are used in S-VHS recorders.

Composite Video

The composite color-encoding methods encode the brightness and color into a single signal. There are NTSC and PAL versions of this format.

Video Formats and Recording Quality *(legacy)*

Video recorders were available for analog and digital recording in a number of formats. They were also informally classified according to Consumer, Industrial/Professional and Broadcast market types, a measure of recording quality (Consumer being the lowest quality and Broadcast being the highest quality).

Analog Formats

The following were the available analog videotape formats:

- **VHS** — Consumer quality format using a composite signal
- **S-VHS** — Consumer quality format using a YC or composite signal
- **S-Video** — Consumer and Industrial/Professional quality format using a YC-358 or YC-443 signal
- **Beta** — Consumer quality format using a composite signal
- **8mm** — Consumer quality format using a composite signal
- **Hi-8mm** — Consumer and Industrial/Professional quality format using a composite or YC signal
- **U-Matic** — Industrial/Professional quality format (SP) cassette, 3/4 inch, using a composite signal
- **Type C** — Broadcast quality format, reel-to-reel, 1 inch tape, using a composite signal
- **Type B** — Broadcast quality format (Europe) using a composite signal
- **Betacam** — Broadcast quality format using a component (RGB) signal
- **Betacam SP** — Broadcast quality format using YUV, YIQ or composite signal
- **MII** — Broadcast quality format using YUV, YIQ or composite signal

Digital Formats

The available digital tape formats were D1 525 (YUV), D1 625 (YUV), D2 525 (NTSC) and D2 625 (PAL). All digital formats were used in post-production and were typically not used for recording original information.

Tips and Suggestions

Video differs from computer graphics in many ways. By being aware of these differences, the video quality may be greatly improved. Some important issues are addressed below.

Color Saturation

Video recording devices do NOT record saturated colors well.

> **NOTE:** A saturated color is 100 percent red, blue or green (as defined by its RGB values).

Objects with saturated colors seem to have a halo and tend to glow. When assigning any color value using an HVE color wheel editor or slider, choose values less than 256 (move the hot spot slightly away from the edge of a color wheel and the right end of the brightness slider).

Anti-aliasing

Anti-aliasing significantly increases rendering time. There is really no benefit to anti-aliasing until you want to produce a video. At this point, use the Rendering Options dialog to increase the Anti-aliasing value to about 3 (on a scale of 1 to 10; the improvement becomes less and less visible as anti-aliasing is increased).

Keep Objects Close

This suggestion simply comes from experience. Closer objects look better and tend to command your attention. Objects farther away lose their definition and become lost in the environment.

"Camera Car"

Pretend you are directing an action movie. As the vehicle moves through the scene, you want to follow it! A single view from a distance would not focus on the items of importance and would not connect with the viewer. The solution is to move the camera along with the vehicle.

Using HVE, you can create a

Camera Car

. A camera car is a vehicle to which you have attached a camera.

> **NOTE:** See View Menu, Camera Set-up (Object-based Cameras).

Set up an event that moves this camera through the scene and points towards the desired action. This takes a little planning and practice, but has a huge payback. You'll also know the exact position of the view at each timestep (this information is available from the camera car's Variable Output Window).

Sizing Issue (TV will crop)

Nearly all video display devices crop some information (in addition to issues related to saturated colors, described earlier). View your results on the actual presentation display (television, projector, courtroom monitor) before you send anything to a client. By viewing the results on the target display, you will confirm the quality as well as the actual information displayed on the screen.

Flicker

Horizontal white lines cause flicker! This is true of even the very best broadcast video systems. (Have you ever seen the TV weatherman wear a tie with horizontal white stripes? Probably not very often.)

Think about your environment when composing your view. If the viewer is filled with pavement stripes or other information running horizontally across the screen, adjust the viewer to a slightly oblique view. The result will be significantly better.

Computer to S-VHS to VHS, Not Computer to VHS *(legacy)*

On tape-based systems, although creating composite video directly on the VTR might have seemed more efficient, the quality was not as good as creating an S-VHS master first and using it for composite copies.

(updated: the modern equivalent of this advice is to keep a high-quality, lightly compressed master movie file, and produce more heavily compressed copies for distribution from that master rather than re-recording.)

Tape Problems *(legacy)*

Use the tips listed under

Preparation of Recording Media

, earlier in this chapter, to avoid problems that may occur during recording of real-time simulations to video tape.

Screen Saver, Other Windows on top of Playback Window

HVE's video sub-system reads the rendered contents of the playback window region of the computer's frame buffer. As it turns out, a lot of objects can draw into this area! To help prevent unexpected results, do the following:

- Turn off the screen saver. Otherwise, your video may turn black when the screen saver comes on!

> **NOTE:** Remember, it can take up to 25 minutes or more to record a 20-second simulation movie if you have used a highly detailed environment model, a moving camera position and increased settings for anti-aliasing and render quality. If you walk away and return when it's done, you might be surprised about half-way through the sequence!

- Be sure there are no windows or dialogs lying within the limits of the Playback Window. These will appear in your video!
- Make sure the mouse cursor isn't sitting within the window!

Chapter 27 — Using The HVE Video Interface

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 27, pages 27-1 through 27-12, revised to describe the current HVE video interface.

This chapter describes the process of producing high-quality video directly from simulation output using the HVE video interface.

*(updated: In the original edition this chapter described routing playback to external video tape recorders. Current HVE records simulation movies directly to compressed **AVI** files (optionally MPEG, or individual frame image files) using the codecs installed in Windows. The Playback window in current HVE is opened with **Video Creator...** on the Files menu. The chapter below reflects the current workflow; legacy tape-recorder material is flagged.)*

The video interface includes two basic tools:

- **Video Options (Video Set-up) Dialog** — Opened with the Video Setup button on the Playback (Video Creator) window; used for selecting the video format (movie and/or single frames), movie size, frame rate, video compressor (codec) and output directory.
- **Playback Window (Video Creator)** — Available in the Playback Editor, used for recording and playing back video results.

Whether recording a new movie or replaying a saved one, these two tools are used. They are described below.

Video Setup Dialog

HVE has a dialog for configuring video output before recording. In the original edition this was described as installing a "video device" (loading device drivers and setting up communication between the computer and an external recorder), much like installing a printer. In current HVE no device installation is required: the Video Options dialog simply configures how the movie file is written.

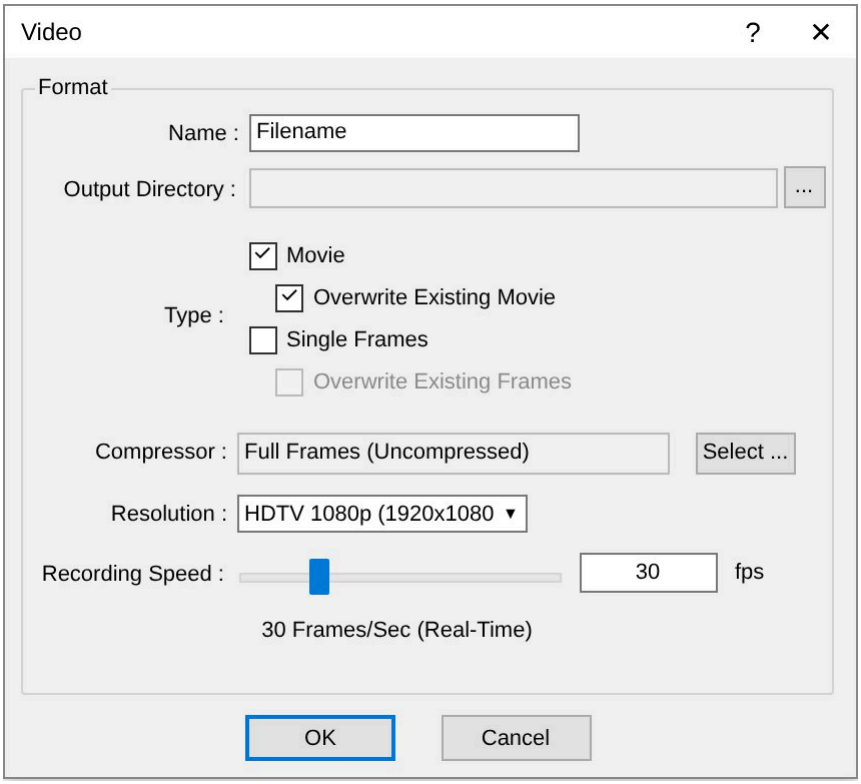


Figure 27-1 — Video dialog.

(updated: the current Video Options dialog contains the following options:)

- **Video Format** — Two check boxes select the output format(s):
- **Movie** — Records the playback to a movie file (AVI). An *Overwrite* check box controls whether an existing movie file is replaced.
- **Single Frames** — Writes each rendered frame to an individual image file. A *Filename* field sets the base frame filename and an *Overwrite* check box controls replacement of existing frames.
- **Compressor** — Displays the current video compressor. The **Select...** button opens the standard Windows compressor-selection dialog listing all codecs installed on the system. If no compressor has been selected, movies are written as "Full Frames (Uncompressed)".
- **Video Size** — Selects the movie frame size: Default (playback window size), DVD, 480p, 720i, 1080i, 1080p or 4K.
- **Recording Speed / FPS** — Slider and text field selecting the recording frame rate (frames of simulation output per second of video).
- **Output Directory** — Selects the directory where movies and frames are written (by default the `movies` support-files directory; see Appendix I).

> **NOTE:** The Video Setup options are stored in HVE's configuration file. Therefore, you will not need to set up the options again unless you re-install HVE or wish to change them.

Video Devices *(legacy)*

The original procedure installed a video output device as follows:

1. Choose Video from the Files menu. The Video Setup dialog is displayed, showing the current video setup options.
2. If *S-Video/Composite Video* is not displayed in the Device Name field, click on the Device Name option list.
3. Select the desired video device. For real-time movie file recording, choose *S-Video/Composite Video*. This allowed use of HVE's integrated video output display, sized to match the video format.
4. Press OK to activate the selected video device.

(updated: the legacy Video Setup dialog is still present, but its Device Name, Connect To (Serial 1/Serial 2/Video Out), Test Medium and Time Codes controls are permanently disabled — external video devices are no longer supported. Video output is always written to a file.)

Video Compressor

If you wish to compress the recorded movie (strongly recommended — an uncompressed AVI grows very large very quickly), you must select a video compressor. Several video compressors may be present, depending on the software installed on your system.

To select a video compressor, perform the following steps:

1. Open the Video Options dialog (Video Setup button on the Playback window).
2. Press the **Select...** button next to the Compressor field. The standard Windows compression dialog is displayed, listing all available codecs. Select the desired compressor. *(updated: the original manual instructed users to select the Cinepak Codec, which is long obsolete; choose any modern codec installed on your system.)*
3. Press OK to activate the selected video compressor.

> **NOTE:** The selected compressor is stored in HVE's configuration file. Therefore, you will not need to set up the compressor again unless you re-install HVE or wish to use a different video compressor.
>
> **NOTE:** Compressed video sequences are often referred to as "movies".

Playback Controller

Video output is recorded and played back using the Playback Editor's Playback Controller. While using the video interface, there is always a *video source* and a *video destination*. The Playback (Video Creator) window allows the user to choose the current video source and destination from two option lists.

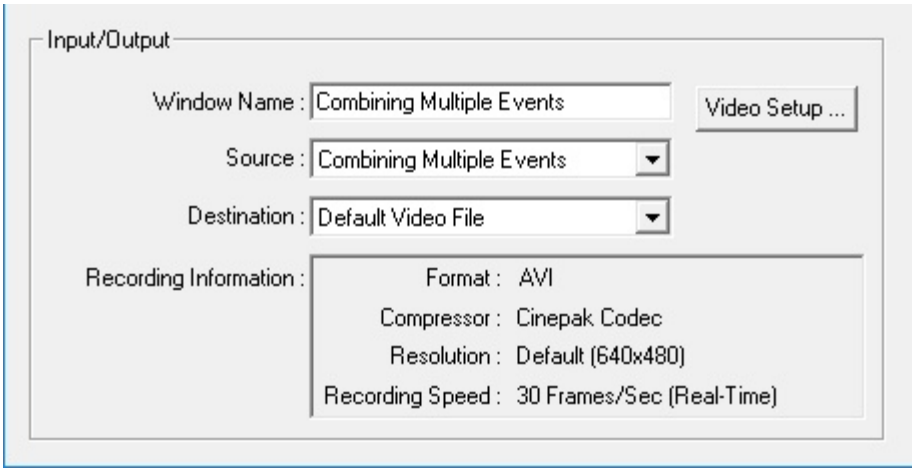


Figure 27-2 — Close-up of the portion of the Playback Window used for selecting the video source and destination.

Video Source

The video source is, as the name implies, the source of the video information. The two possible video sources are:

- **Playback Window Name** — If a Playback Window has been created, it contains a sequence of frames that may be viewed. The Playback Window normally displays an accident sequence that includes one or more HVE simulation events.
- **Movie Filename** — If a sequence has been saved as a movie, that sequence may be played back in the Playback Window. *(updated: the source list includes the default video file (`Default.avi`) if one has been recorded, plus every other `.avi` file found in the movies directory, so a previously recorded movie may be chosen directly.)*

Video Destination

For any video source, there must be a destination. The possible video destinations are:

- **Playback Window Name** — By default, all sources are routed to the current Playback Window for viewing.
- **Video File** — If the source is the Playback Window, the rendered sequence is recorded to the movie file (and/or single-frame files) configured in the Video Options dialog.

*(updated: the original manual listed a third destination, **Video Device Name** (e.g., a VTR), for routing output to a video recorder. External video devices are no longer supported; the corresponding "S-Video / Composite Video" destination entry is no longer available in the current version.)*

Recording A Movie

The above video sources and destinations provide the user significant flexibility when using HVE's video tools. To record a real-time simulation movie, perform the following steps:

1. Configure the video options (format, size, frame rate, compressor), as described earlier in this chapter.
2. Execute the desired HVE simulations (e.g., EDSMAC4, SIMON, EDVSM...) using the HVE Event Editor.
3. Choose **Video Creator...** from the Files menu and create a Playback Window that includes the desired simulations.
4. Set the view using either the viewer's thumbwheels or the Set Camera dialog.
5. Click on the **Source** option list and choose the playback window name (e.g., *Untitled Playback Window*) as the source.
6. Click on the **Destination** option list and choose the video file (AVI) as the destination.
7. Press **Play**.

HVE will record the simulation displayed in the playback window (the Source) to the `Default.avi` compressed disk file (the Destination) one frame at a time until the end of the simulation or until the user presses Pause or Stop.

To view the sequence in real time, perform the following steps:

1. Click on the **Source** option list and choose the movie file as the source.
2. Click on the **Destination** option list and choose the Playback Window (e.g., *Untitled Playback Window*) as the destination.
3. Press **Play** on the HVE Playback Controller.

The movie is displayed in the Playback Window in real time.

Video File Selection

The video recorder creates a file of the video sequence. These files may be saved and opened using a file selection dialog, similar to other files.

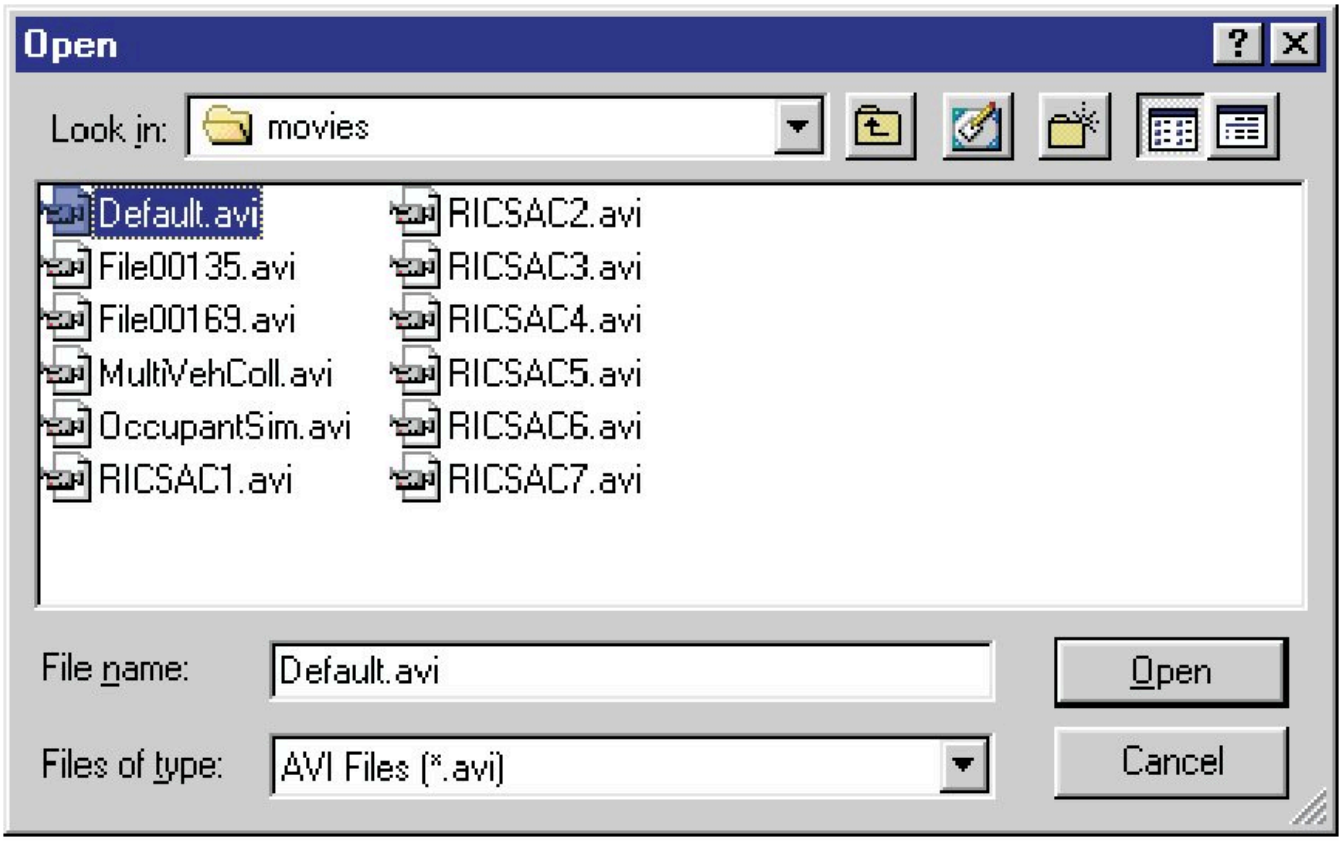


Figure 27-3 — Video File Selection dialog, used for opening and saving compressed video output.

Saving Compressed Video Files

After creating the video sequence, the user can save the video file by a unique filename for future viewing or editing. To save the current video file, perform the following steps:

> **NOTE:** At this point, it is assumed you are in playback mode, have just executed a simulation and the current video destination is the video file.

1. Open the Video Setup dialog.
2. Choose **Save-As**. The Video File Selection dialog is displayed.
3. Select a filename from the list of existing video files, or enter a new filename.

> **NOTE:** HVE will append the appropriate extension to the name you enter. For example, the filename *MyVideo* will be saved as `MyVideo.avi`. You do not need to explicitly specify a filename extension.
>
> **NOTE:** All video files are stored in the `movies` subdirectory; see Appendix I.

1. Press OK.

The compressed video file will be saved by the selected filename.

Viewing Previously Saved Video Files

To replay a previously saved video file, perform the following steps:

1. Open the Video Setup dialog.
2. Choose **Open**. The Video File Selection dialog is displayed (current HVE lists AVI and MPEG files).
3. Select a filename from the list of existing video files.
4. Press OK. The selected filename will be displayed in the Source File field.
5. Press OK in the Video Setup dialog.

The next step is to play and view the simulation. Perform the following steps:

1. If necessary, choose Playback Mode and add a Playback Window. (You must have a Playback Window in order to play a simulation movie within HVE.)
2. Click on the **Source** option list and choose the movie filename (e.g., `MyVideo.avi`). *(updated: saved movies in the movies directory also appear directly in the Source list, so the Open step above is often unnecessary.)*
3. Click the **Destination** option list and choose the Playback Window.
4. Using the Playback Controller, press **Play** to play the simulation movie.

The simulation sequence will be displayed in real time in the playback window.

> **NOTE:** Because this is a pre-recorded sequence, you cannot change the view. In addition, the simulation time in the Playback Controller is not displayed.
>
> **NOTE:** You can save this video file using the same procedure as for saving the default video, described earlier.

Creating Multiple Video Sequences

The user will often require a series of simulations recorded back-to-back. Examples include:

- Viewing the same crash sequence from several different perspectives (i.e., driver(s), witness(es), fly-by, general perspective)
- Creating alternative scenarios of a crash sequence (driver's version, witness's version, your version)

To produce a video containing multiple sequences recorded back-to-back, record each sequence to a movie file, then either:

- Record each successive sequence to the same movie file. When you save to an existing filename, HVE asks whether you wish to **replace** the existing file or **append** the current video sequence to the existing sequence — choose append to build up a back-to-back movie; or
- Record each sequence to its own movie file and assemble them in a video editing package.

(legacy)

On tape-based systems the equivalent procedure was:

1. Record the first sequence.
2. At the end of the first sequence, pause the VCR (leave the VCR in Record mode).

3. Using the Playback Controller, choose the second sequence as the Source.
4. Release the VCR's pause button, allowing the VCR to run for a few seconds.
5. Using the Playback Controller, press Play to record the second sequence.
6. Repeat the above steps for each sequence to be recorded.

Step-by-step Examples

The following examples illustrate the use of the HVE Video Interface to perform a complete and inter-related set of video output tasks.

Example 1: Recording the current simulation in real time in the Playback Window

Video compression allows the user to view a sequence in real time in the current HVE Playback Window. The obvious advantage of this approach is time savings; the recording may be previewed before it is distributed.

To create a real-time simulation movie, perform the following steps:

1. Create a Playback Window (Files menu, Video Creator...) that includes the desired simulations. Set the view using either the viewer's thumbwheels or the Set Camera dialog.
2. Choose the Playback Window name (e.g., *Untitled Playback Window*) as the source.
3. Choose the video file (AVI) as the destination.
4. Press **Play**.

HVE will route the simulation in the Playback Window (i.e., the source) to the computer hard disk using the video compressor (i.e., the destination) one frame at a time until the end of the simulation or until the user presses Pause or Stop.

> **NOTE:** HVE automatically sends the video to a default filename (`Default.avi`) in the movies subdirectory of your HVE support files. For more information, see the next example.

Example 2: Save the current compressed video file

The compressed video created in Example 1 (above) is saved in a default file. If the user wishes to save the file for later viewing and editing, the file may be saved using the HVE Video Set-up dialog. To save the current video by a user-defined filename, perform the following steps:

1. Create a compressed video as described in Example 1, above.
2. Open the Video Set-up dialog.
3. Choose **Save-As**. The Video File Selection dialog is displayed.
4. Enter or select a filename from the Files list and press OK.

> **NOTE:** If you choose an existing filename, HVE will ask if you wish to replace the existing file or append the current video sequence to the existing sequence.

1. Press OK to return to the Playback Editor.

The resulting video file is now saved and is available for later viewing, as described in the next example.

Example 3: Play a previously saved compressed video file

After saving a compressed video file as described above, the video file may be viewed in the HVE Playback Editor. To view the previously saved compressed video file, perform the following steps:

1. Create a compressed video as described in Example 1, above, and save it as described in Example 2, above.
2. Using the Playback Window, choose the saved video filename as the source (select it directly from the Source list, or open it via the Video Set-up dialog's **Open** button), and choose the playback window as the destination.
3. Press **Play**.

The sequence in the selected video file will be displayed in real time in the current playback window.

Example 4: Record the current compressed video file to tape *(legacy)*

On legacy systems with video-out hardware, a saved compressed video file could be recorded to a VTR as follows:

1. Create, save and select a compressed video as described in Examples 1–3.
2. Optionally, press Play to play the sequence to confirm it is exactly what you want to record.
3. Activate the video-out capabilities of your computer (these steps are hardware-dependent).
4. Confirm the VTR settings are correct.
5. Insert a tape into the VTR.
6. Using the Playback Window, choose a video device name (e.g., S-Video/Composite Video) as the destination.
7. Press Play. HVE routes the movie (the source) to the selected video device (the destination), displaying it in a specially formatted full-frame viewer designed to work with the computer's video sub-system.
8. Start the recording process (i.e., press the Record and Play buttons on the VCR).
9. Press the Play button on the viewer to play the movie.

The simulation movie is displayed on the screen as well as recorded on the VTR at real-time speed.

(updated: current HVE has no video-device destination. To put a simulation movie on physical media or another device, record it to an AVI file and use standard authoring/conversion tools.)

For more information about video concepts, see Chapter 26, Video Basics.

Problem Determination

If you are experiencing problems while trying to create a video, perform the following steps:

1. Confirm the video options (format, compressor, output directory) are properly set in the Video Options dialog. If a movie fails to record with a particular codec, try a different compressor or Full Frames (Uncompressed). *(legacy: on tape-based systems, confirm the video cable(s) connecting the computer's video hardware to the video recorder are properly connected to both devices — in general, the computer's video out is connected to the recorder's video in, and vice-versa — and confirm the video device controls are properly set, per the specification sheets supplied with the computer and video subsystem.)*
2. Confirm there is sufficient disk space in the output directory — uncompressed movies are very large.
3. Refer to Tips and Suggestions (see Chapter 26, Video Basics) for techniques and suggestions on improving the quality of your videos.

Section Eleven: HVE Brake Designer

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Section Eleven "HVE Brake Designer" (Section Ten / Chapter 28 of the original manual is reserved for future use), revised to describe the current HVE Brake Designer and ABS Designer.

The HVE Brake Designer is a time-domain simulation model of user-defined brake system components. The purpose of the HVE Brake Designer is to provide a detailed brake design capability integrated directly within the HVE simulation environment. The capability includes the design and evaluation of wheel brake assemblies and ABS systems by engineers charged with fitting new vehicles with properly designed brake system components, as well as the evaluation of in-use service conditions. This section of the HVE User's Manual describes how to create and edit various brake system designs. It also provides an overview of the brake and ABS model parameters.

The HVE Brake Designer was developed by Engineering Dynamics Company, LLC, and was based, in part, on previous work by Limpert [4.30], MacAdam [4.2], Flick [4.31] and Robert Bosch [4.41].

Although the HVE Brake Designer is a very powerful tool based on widely accepted scientific principles, brake design engineers know that brake torque calculations are not exact, mainly because the precise friction value is never known; it is always estimated. Therefore, brake design engineers and safety researchers should use the HVE Brake Designer as they use other modeling tools. The results are useful for matching measured dynamometer data, extending existing designs, providing parametric comparisons and estimating the behavior of a proposed design.

General Procedure

To use the HVE Brake Designer, perform the following steps:

1. Select Vehicle Mode.
2. Select one or more vehicles from the Vehicle Database or from a previous HVE case.
3. Click on the Brake System icon and confirm or edit the Brake Pedal Ratio.
4. If the vehicle is fitted with ABS, click on the ABS Installed checkbox. Press Edit to view and edit the desired ABS system properties.
5. Click on a wheel and choose Brakes from the pop-up menu.
6. Click on the Brake Assembly Type option list and choose the desired brake design (e.g., Disc Brake, Duo-servo Drum Brake). The HVE Brake Designer dialog for the selected brake type is displayed.
7. Edit the desired properties.
8. Press OK to accept the current design.
9. Edit the remaining wheels as desired.
10. Execute the desired suite of simulations to test the effectiveness of the design.
11. Edit the design as required to obtain optimum braking performance.

Overview of Section Eleven

This section provides a detailed explanation of the modeling techniques used by the HVE Brake Designer, and also presents an overview of the Brake Designer's interface.

- Chapter 29 — HVE Brake Designer describes in detail the process of using the HVE Brake Designer.
- Chapter 30 — Brake Temperature Model provides a detailed overview of the Brake Temperature Model.
- Chapter 31 — Antilock Braking Systems provides a detailed overview of the HVE Antilock Brake System Model.

Related Dialog Reference Pages

The individual brake dialogs are documented in detail in the Brakes and Powertrain reference:

- Brake Assembly dialog
- Disc Brake designer dialog
- Duo-Servo Drum Brake designer dialog
- Duplex Drum Brake designer dialog
- Single Piston Drum Brake designer dialog
- Dual Piston Drum Brake designer dialog
- Dual Wedge Drum Brake designer dialog
- Brake Designer Material Properties dialog
- Brake System Pressure vs Pedal Force dialog

Chapter 29 — HVE Brake Designer

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 29, pages 29-1 through 29-30, revised to describe the current HVE Brake Designer.

Overview

The HVE Brake Designer allows brake design engineers and safety researchers to design wheel brake assemblies and to evaluate how a given brake design affects vehicle braking performance. A major benefit of the HVE Brake Designer is that it is integrated directly into the HVE simulation environment; thus, designers can generate a brake design and then use simulation to directly evaluate its effectiveness for a specified suite of vehicle performance tests or maneuvers.

The HVE Brake Designer may be used to design and evaluate the following brake types:

- Disc Brake
- Duo-Servo Drum Brake
- Duplex Drum Brake
- Single Piston Drum Brake
- Dual Piston Drum Brake
- S-Cam Drum Brake
- Single Wedge Drum Brake
- Dual Wedge Drum Brake
- Air Disc Brake *(updated: added since the original edition — an air-actuated caliper disc brake for heavy vehicles, combining the caliper disc geometry with air-chamber actuation, slack adjuster and stroke-based actuation factor)*

The HVE Brake Designer incorporates advanced features, such as the effect of sliding speed and temperature on lining friction. The model also includes the capability to study brake fade characteristics associated with reduced braking capacity of heavy trucks on long downhill grades.

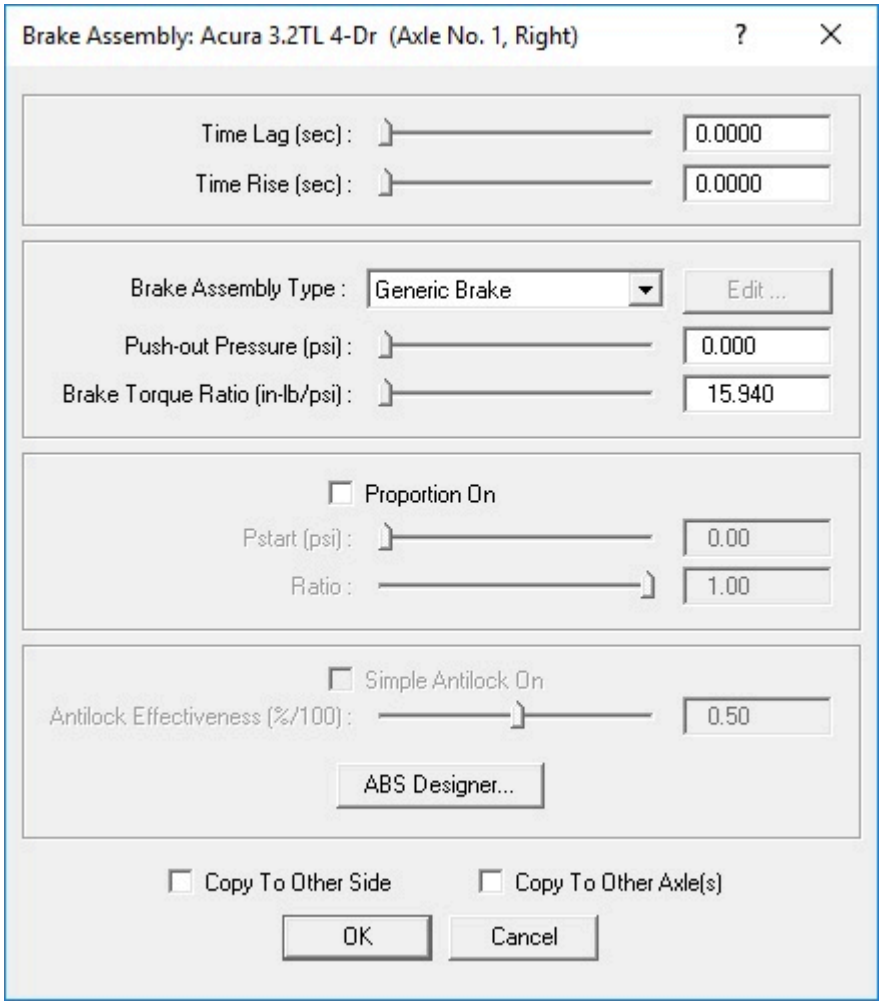


Figure 29-1 — Brake Information dialog.

To use the HVE Brake Designer, perform the following steps:

1. In the Vehicle Viewer, click on the desired wheel. The Unsprung Mass options for the selected wheel are displayed.

> **NOTE:** Refer to Chapters 10 and 11 for details about using the HVE Vehicle Editor.

1. Choose Brakes from the Unsprung Mass option list. The Brake Information (Brake Assembly) dialog for the selected wheel is displayed (see Figure 29-1). The default brake type is Generic and the default brake torque ratio is displayed.
2. Click on the Brake Assembly Type option list and choose the desired brake type (e.g., S-Cam).
3. Press Edit. The HVE Brake Designer dialog for the selected brake type is displayed, as shown in Figure 29-2. Default values are assigned for each parameter, and the calculated brake factor, actuation factor and brake torque ratio are displayed.

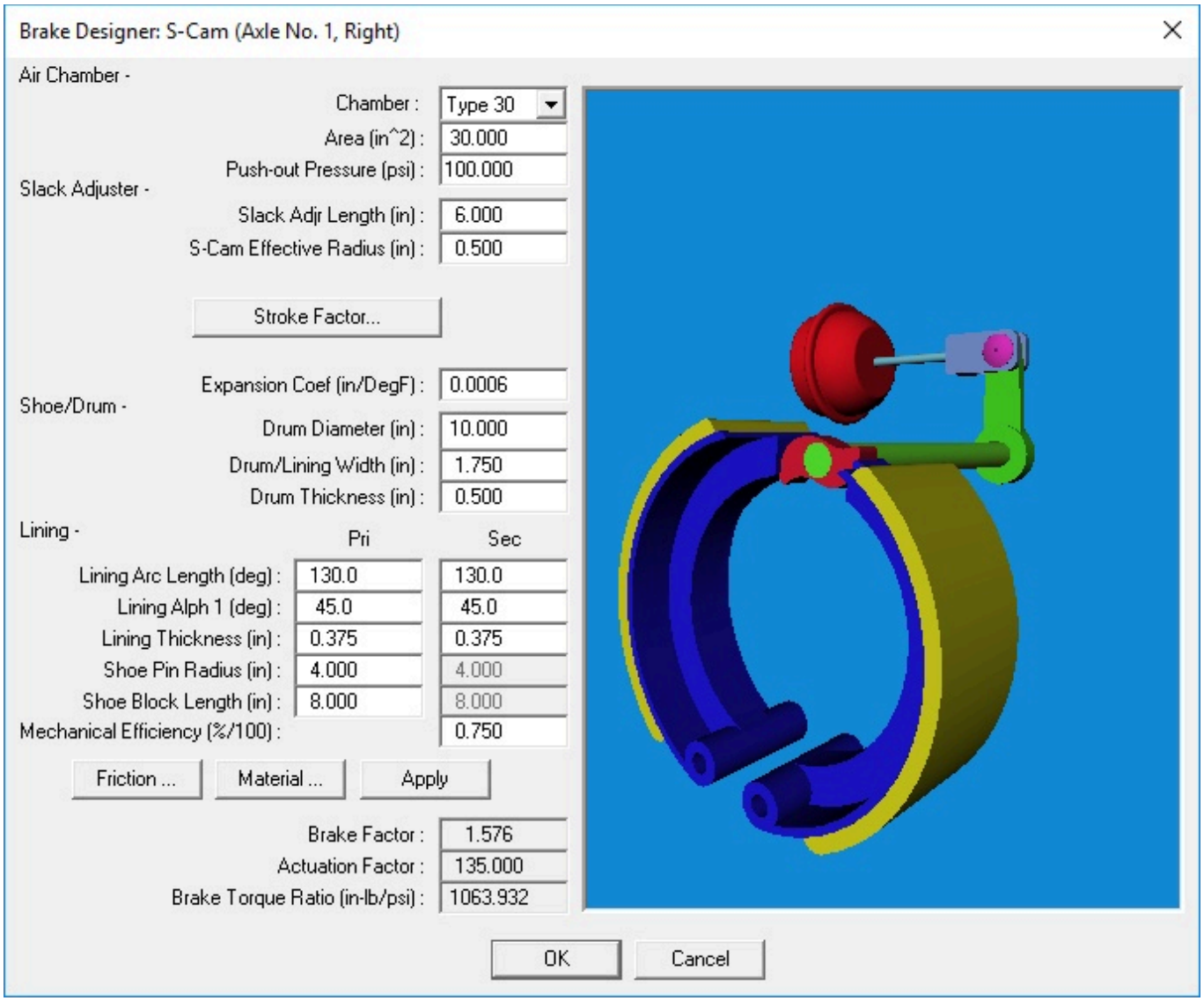


Figure 29-2 — HVE Brake Designer for an S-cam brake. Each Brake Designer dialog includes an embedded schematic preview drawing of the selected brake, updated to reflect the current dimensions.

> **NOTE:** When used in this manner, the calculated brake factor and brake torque ratio are based on the environment's ambient temperature and zero sliding velocity at the pad/rotor (or lining/drum) interface. While a simulation is executing, the brake factor and brake torque ratio may be calculated using the current brake internal temperature and sliding velocity calculated by the simulation.

1. Edit any of the parameters, then press Apply. The brake factor and brake torque ratio are calculated and displayed.
2. When finished editing the brake parameters, press OK.

The HVE Brake Designer dialog is removed. Note that the calculated brake torque ratio and pushout pressure associated with the selected design are now displayed in the Brake Information dialog.

> **NOTE:** If the Brake Designer has been used to calculate the brake torque ratio, the calculated value cannot be edited directly. If you wish to enter the brake torque ratio directly, you must first change the Brake Assembly Type to Generic.

Brake Torque

The HVE Brake Designer uses the following fundamental approach in the calculation of brake torque for a given wheel:

$$BT = BF \times TR \times AF \times Factor \qquad (29\text{-}1)$$

where

Symbol	Description
BT	Brake torque at wheel (in-lb)
BF	Brake Factor, the ratio of friction force to actuation force produced in the braking device (dimensionless)
TR	Radius of the friction force (in)
AF	Actuation Force produced by the brake actuation device (lb), (i.e., caliper piston, wheel cylinder, S-cam or wedge)
$Factor$	Percentage of reduction in actuation force (%/100), normally resulting from excessive air chamber stroke

Thus, it can be seen that to calculate brake torque under any given set of conditions, one must be able to calculate the brake factor, brake torque radius, actuation force and actuation factor. These calculations are described in detail below. For an excellent treatise on brake design, see Limpert [4.30], as well as references 4.2 and 4.31 through 4.37.

Brake Factor

Brake factor is defined as the ratio of the lining friction force to the brake actuation force. As a ratio, it is a dimensionless quantity. The lining friction force is produced at the pad/rotor interface (disc brake) or the shoe/drum interface (drum brake). The actuation force is produced by the caliper piston, wheel cylinder, S-cam or wedge under the influence of brake system pressure.

Lining Friction

Brake factor is heavily influenced by the friction coefficient between the lining material and the rotor or drum. Lining friction, in turn, is influenced by temperature and sliding velocity at the interface between the lining and rotor or drum. The HVE Brake Designer allows the user to directly study these influences by allowing the user and/or simulation model to determine the correct value of lining friction as a function of brake temperature and sliding velocity.

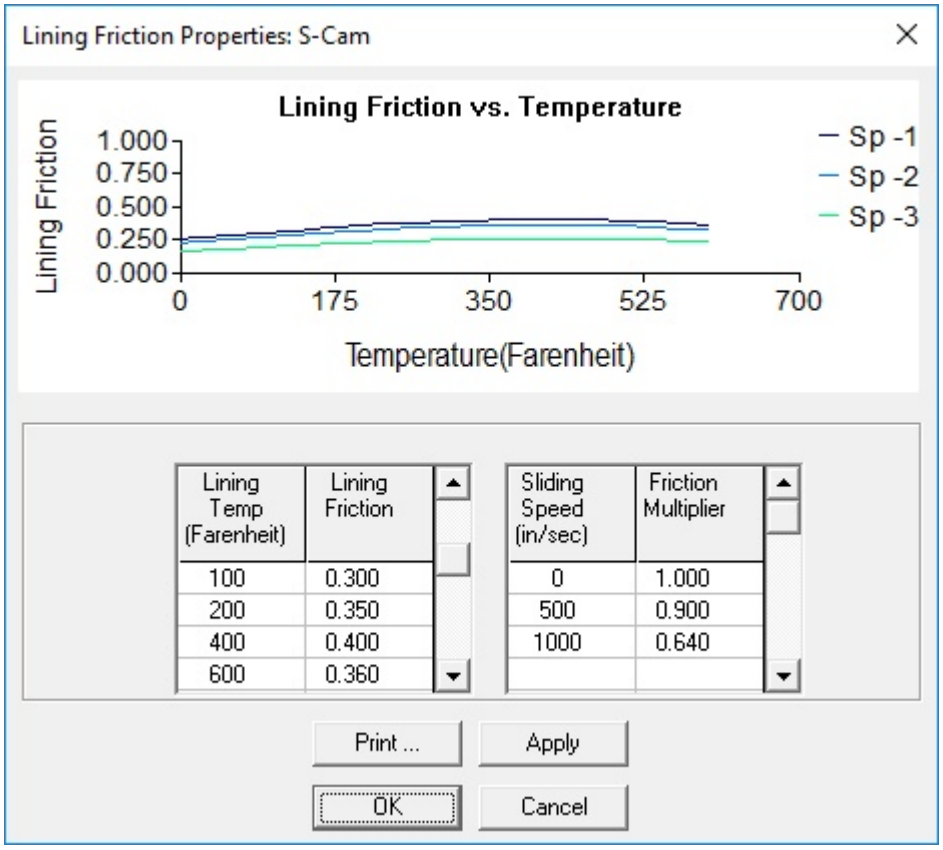


Figure 29-3 — Lining Friction dialog, used for entering lining friction vs temperature (left) and lining friction factor vs sliding speed (right) data tables. (The original caption is misnumbered "Figure 30-3".)

Table 29-1 — Lining Friction Designation (from SAE J866 MAR85 [6.5])

Code Letter	Friction Coefficient
C	Not over 0.15
D	Over 0.15 but not over 0.25
E	Over 0.25 but not over 0.35
F	Over 0.35 but not over 0.45
G	Over 0.45 but not over 0.55
H	Over 0.55
Z	Unclassified

Lining friction is determined using linear interpolation of the friction vs temperature table (see Figure 29-3). The default values are assigned according to SAE J866 MAR85, Friction Coefficient Identification System for Brake Linings [6.5] (see Table 29-1), and are user-editable.

> **NOTE:** The nominal friction for a brake lining can be identified by a 2-character code stamped on the edge of the lining. The first of the two characters defines the nominal friction at low temperature, and the second character represents the friction at high temperature.

Lining friction is then modified using linear interpolation of the friction factor vs sliding speed table (see Figure 29-3). The default values are derived from brake dynamometer test data in references 4.36 and 4.37. Values are user-editable.

The calculation of brake factor lies at the heart of a brake design. Each type of brake works in a unique manner and has a governing set of brake factor equations. These equations are described in the following sections.

Disc Brake

A free body analysis of a disc brake yields the following simple calculation for brake factor:

$$BrakeFactor = 2.0 \times \mu_L \qquad (29\text{-}2)$$

where

Symbol	Description
μ_L	Lining friction at temperature and sliding speed

Drum Brake

Two basic types of brake shoe designs are used in drum brakes: pinned shoes and abutted shoes. In both designs, the direction of drum rotation influences the brake factor. This influence results from the fact that the friction force produced between the drum and shoe produces an increased actuation force for a leading shoe and a decreased actuation force for a trailing shoe. These influences are seen in the following calculations for pinned and abutted shoes.

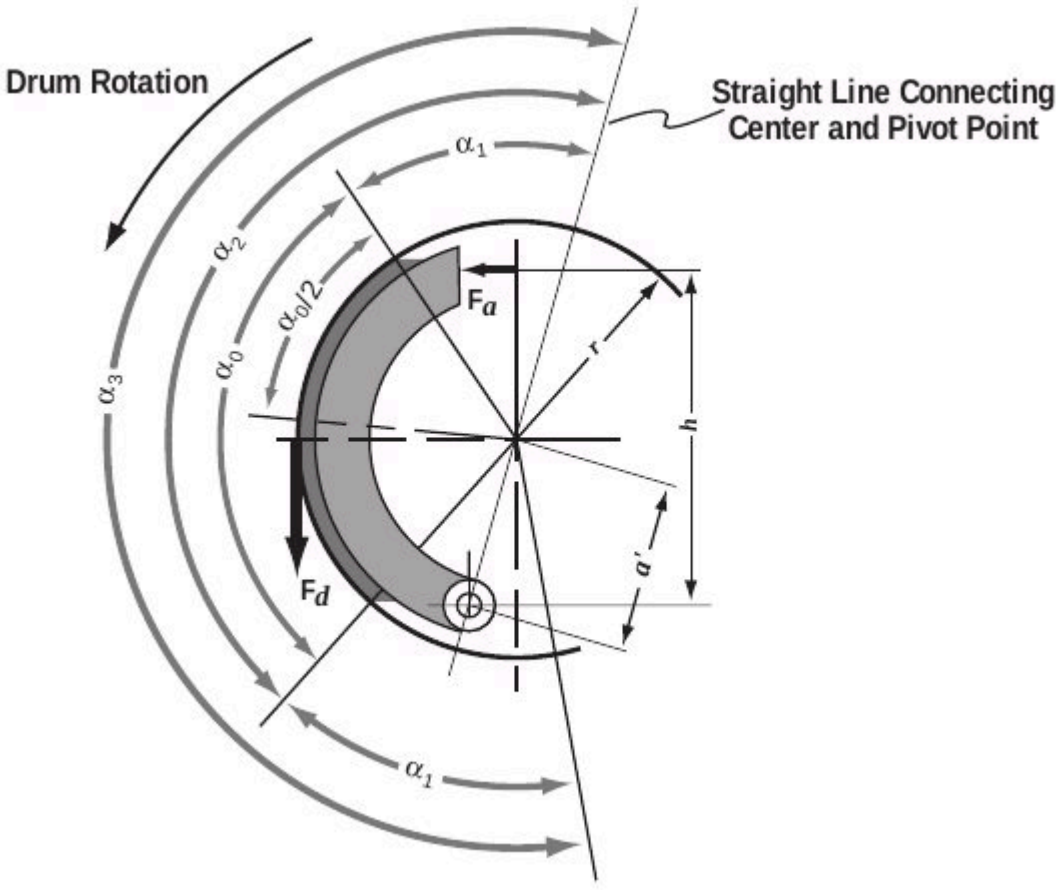


Figure 29-4 — Analysis of a pinned brake shoe [4.30].

Pinned Shoe

A free body analysis of a pinned shoe (see Figure 29-4) results in the following equation for brake factor:

$$BF = \frac{\mu_L \frac{h}{r}}{\frac{a'}{r}\left(\frac{\alpha_0 - \sin\alpha_0 \cos\alpha_3}{4\sin\left(\frac{\alpha_0}{2}\right)\sin\left(\frac{\alpha_3}{2}\right)}\right) \pm \mu_L \left(1 + \frac{a'}{r}\cos\left(\frac{\alpha_0}{2}\right)\cos\left(\frac{\alpha_3}{2}\right)\right)} \qquad (29\text{-}3)$$

where (see Figure 29-4)

Symbol	Description
F_a	Actuation force (lb)
F_d	Friction force produced by brake linings (lb)
μ_L	Lining friction coefficient at current lining temperature
h	Brake block width (in)
r	Drum inner radius (in)
a'	Shoe pin radius (in)
α_0	Lining material arc length (rad)
α_1	Angular dimension (deg)
α_3	$2\alpha_1 + \alpha_0$

Eq. 29-3 is applicable to both leading and trailing shoes; for a leading shoe, the negative sign is employed in the denominator and for a trailing shoe the positive sign is employed. The negative sign results in a larger brake factor and represents the

self-energizing

effect of the brake leading shoe.

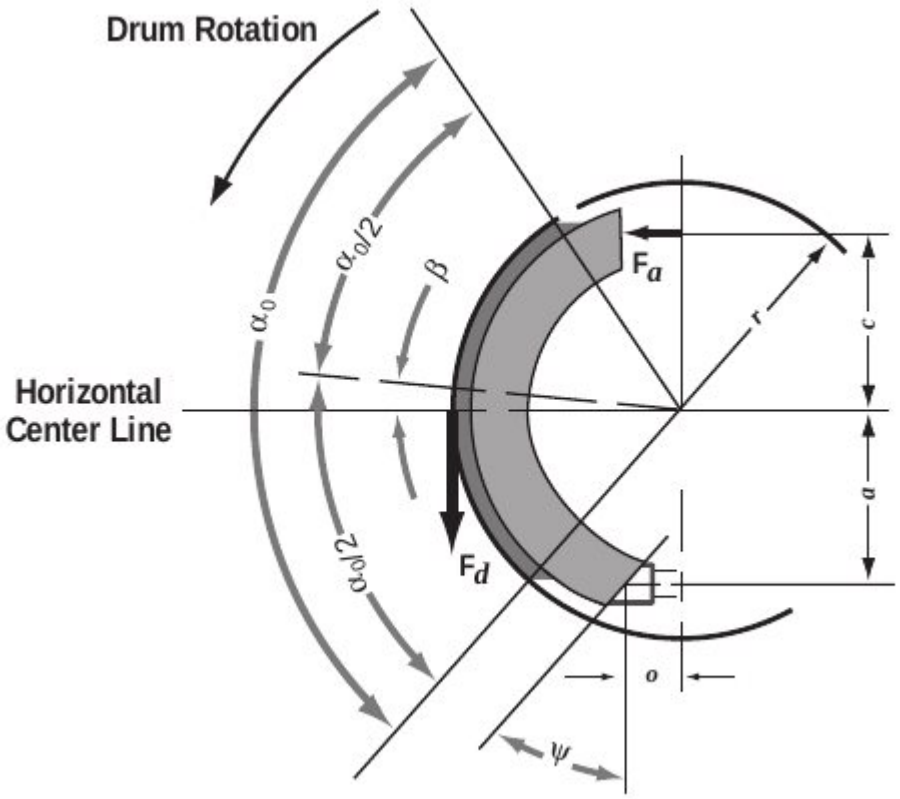


Figure 29-5 — Analysis of an abutted brake shoe [4.30].

Abutted Shoe

A free body analysis of an abutted shoe (see Figure 29-5) results in the following calculation for brake factor for a leading shoe:

$$BF = \frac{\mu_L D_B + \mu_L^2 E_B}{F_B - \mu_L G_B + \mu_L^2 H_B} \qquad (29\text{-}4)$$

and for a trailing shoe:

$$BF = \frac{\mu_L D_B - \mu_L^2 E_B}{F_B + \mu_L G_B + \mu_L^2 H_B} \qquad (29\text{-}5)$$

where (see Figure 29-5)

Symbol	Description
F_a	Actuation force (lb)
F_d	Friction force produced by brake linings (lb)
μ_L	Lining friction coefficient at current lining temperature
c	Center to actuation force distance (in)
a	Center to abutment vertical distance (in)
o	Center to abutment horizontal distance (in)
r	Drum inner radius (in)
α_0	Lining material arc length (rad)
β	Arc center angle (rad), shown + in Figure 29-5
Ψ	Abutment angle (rad)
μ_s	Abutment sliding friction coefficient

and

$$D_B = (\frac{c}{r} + \frac{a}{r} + (\mu_s + \tan\Psi)(\frac{o}{r}))\cos\beta + (\mu_s + \tan\Psi)(\frac{c}{r})\sin\beta$$

$$E_B = (\mu_s + \tan\Psi)(\frac{c}{r})\cos\beta - (\frac{c}{r} + \frac{a}{r} + (\mu_s + \tan\Psi)(\frac{o}{r}))\sin\beta$$

$$F_B = (\frac{\alpha_0 + \sin\alpha_0}{4\sin(\frac{\alpha_0}{2})})(\frac{a}{r} + (\mu_s + \tan\Psi)(\frac{o}{r}))$$

$$G_B = \cos\beta + (\mu_s + \tan\Psi)\sin\beta$$

$$H_B = F_B - ((\mu_s + \tan\Psi)\cos\beta - \sin\beta)$$

Like the pinned shoe equations, the + and − signs in equations 29-4 and 29-5 represent the

self-energizing

effect on the leading and trailing shoes.

Application of these basic equations is shown later in this section for each of the brake types supported by the HVE Brake Designer.

Brake Torque Radius

Brake torque radius is simply the moment arm at which the lining friction force is applied. For disc brakes:

$$TorqueRadius = (R_o + R_i) / 2 \qquad (29\text{-}6)$$

where

Symbol	Description
R_o	Rotor swept area outside radius (in)
R_i	Rotor swept area inside radius (in)

For drum brakes:

$$TorqueRadius = R \qquad (29\text{-}7)$$

where

Symbol	Description
R	Drum inner radius (in)

(updated: the original text printed eq. 29-7 as "R/2"; the drum torque radius is the drum inner radius, i.e. half the drum diameter. Similarly, the disc torque radius is one quarter of the sum of the outer and inner diameters, which equals $(R_o + R_i) / 2$*.)*

Actuation Force

An actuation force causes the pads to contact the rotor (or shoes to contact the drum). Various brake system types use different methods to produce an actuation force. Hydraulic brake systems use caliper pistons (disc brakes) or wheel cylinders (drum brakes). Air brake systems use air chambers, slack adjusters and an S-cam or wedge — or, for air disc brakes, an air chamber acting on a caliper. Actuation force calculations for each of these actuator types are shown below.

Hydraulic Disc Brake

The actuation force for a disc brake is simply the combined product of the hydraulic pressure, piston area, number of pistons and mechanical efficiency:

$$ActuationForce = (P_{Line} - P_{Pushout}) \times A_{Piston} \times N_{Pistons} \times \eta_{Mech} \qquad (29\text{-}8)$$

where

Symbol	Description
P_{Line}	Hydraulic pressure entering the piston chamber (psi), after proportioning
$P_{Pushout}$	Hydraulic pressure required to cause the brake pads to contact the rotor (psi), normally nil for disc brake systems
A_{Piston}	Surface area of one piston (in²)
$N_{Pistons}$	Number of pistons in a brake caliper
η_{Mech}	Mechanical efficiency of the caliper assembly (%/100), approximately 1.0 for a properly maintained disc brake caliper

Hydraulic Drum Brake

The actuation force for a hydraulic drum brake system is produced by hydraulic pressure pushing against one or more pistons. The pistons, in turn, push directly against the brake shoe or against a pushrod that pushes the brake shoe.

The actuation force for a hydraulic drum brake is:

$$ActuationForce = (P_{Line} - P_{Pushout}) \times A_{Piston} \times N_{Piston} \times \eta_{Mech} \qquad (29\text{-}9)$$

where

Symbol	Description
P_{Line}	Hydraulic pressure entering the piston chamber (psi), after proportioning
$P_{Pushout}$	Hydraulic pressure required to cause the brake linings to contact the drum (psi), normally 5 to 100 psi for hydraulic drum brake systems
A_{Piston}	Surface area of one piston (in²)
N_{Piston}	Number of pistons
η_{Mech}	Mechanical efficiency of the wheel cylinder assembly (%/100), estimated between 0.90 and 0.95 for a properly maintained hydraulic drum brake

Drum S-cam Air Brake

The actuation force for a drum S-cam air brake system is produced by an air chamber pushing a pushrod. The pushrod, in turn, pushes a lever called a slack adjuster. The slack adjuster turns an S-cam (see Figures 29-6 and 29-7). The S-cam pushes against the brake shoes causing the actuation force. This system is obviously much more complicated than a hydraulic brake system. It can even be further complicated by the loss of pushrod force when the system is out of adjustment (see Actuation Factor, later in this chapter).

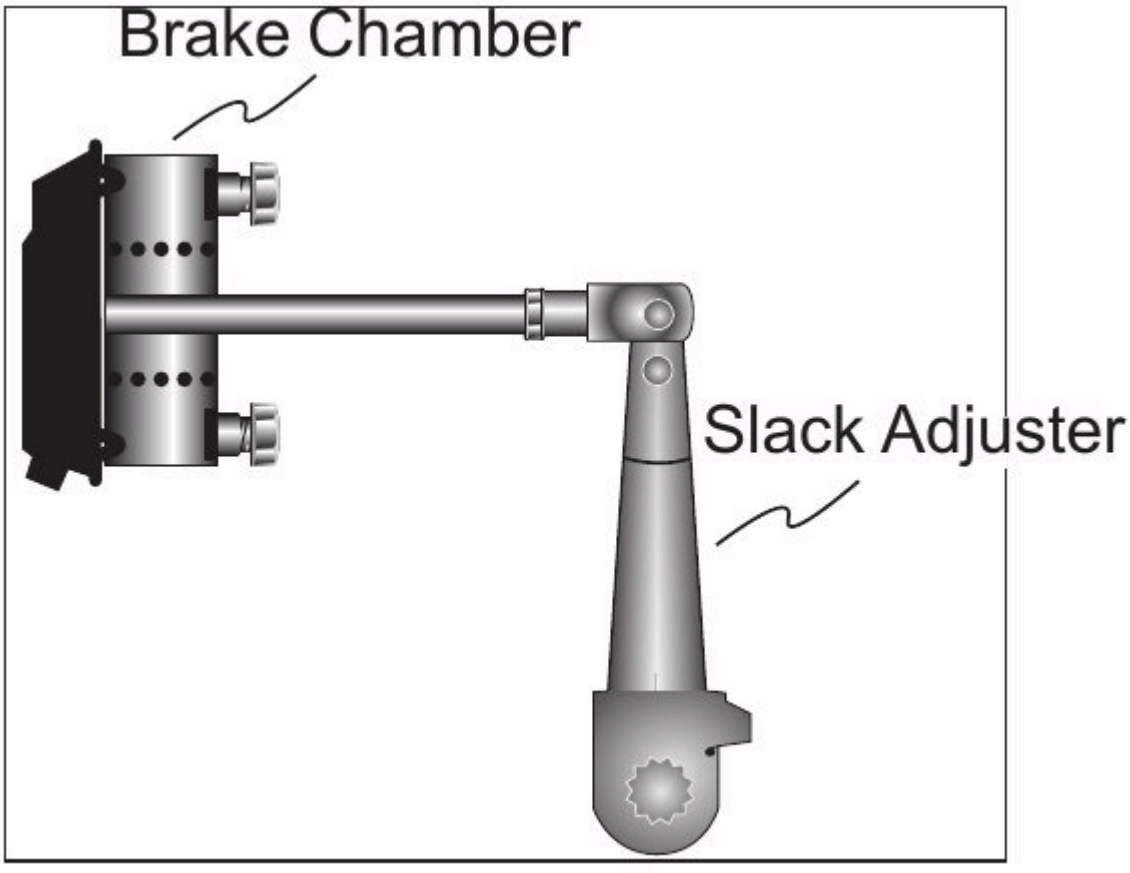


Figure 29-6 — Components of an S-cam brake assembly (brake chamber, slack adjuster, S-cam).

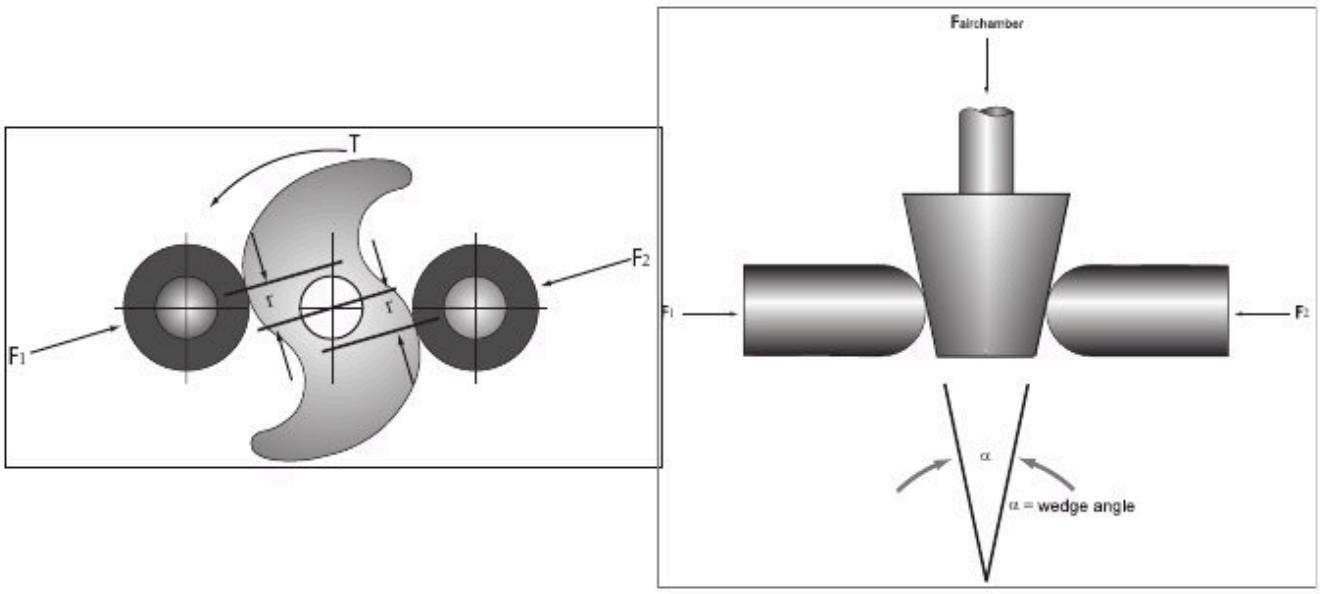


Figure 29-7 — Brake actuators: S-cam (left) and wedge (right).

The actuation force for an S-cam air brake is:

$$ActuationForce = (P_{Line} - P_{Pushout}) \times A_{AirChamber} \times (\frac{L_{Slack}}{2R_{S\text{-}cam}}) \times \eta_{Mech} \qquad (29\text{-}10)$$

where

Symbol	Description
P_{Line}	Air pressure entering the air chamber (psi), after proportioning
$P_{Pushout}$	Air pressure required to cause the brake linings to contact the drum (psi), normally about 5 psi for air drum brake systems [4.37]
$A_{AirChamber}$	Surface area of one air chamber (in²)
L_{Slack}	Slack adjuster lever arm (in)
$R_{S\text{-}cam}$	Effective radius of S-cam (in), normally 0.50 in
η_{Mech}	Mechanical efficiency of the S-cam assembly (%/100), normally between 0.70 and 0.75 for properly maintained air brake systems [4.2]

Drum Wedge Air Brake

The actuation force for a drum wedge air brake system is produced by an air chamber pushing a tapered wedge between two pushrods. The pushrods push directly on the brake shoes causing the actuation force (see Figure 29-7). The actuation force for a wedge air brake is:

$$ActuationForce = (P_{Line} - P_{Pushout}) \times A_{AirChamber} \times (\frac{1}{2\tan(\frac{\alpha}{2})}) \times \eta_{Mech} \qquad (29\text{-}11)$$

where

Symbol	Description
P_{Line}	Air pressure entering the air chamber (psi), after proportioning
$P_{Pushout}$	Air pressure required to cause the brake linings to contact the drum (psi), normally about 5 psi for air drum brake systems [4.37]
$A_{AirChamber}$	Surface area of one air chamber (in²)
α	Included wedge angle (rad)
η_{Mech}	Mechanical efficiency of the wedge assembly (%/100), normally between 0.80 and 0.88 for properly maintained air brake systems [4.2]

Actuation Factor

The actuation force for a drum air brake with an S-cam or wedge actuator (or an air disc brake) is heavily influenced by its state of adjustment (see Figure 29-8). To account for the loss in actuation force at increased levels of stroke, an

actuation factor

is employed.

The actuation factor is an empirically derived relationship between actuation force and piston stroke. The actuation factor is the percentage of the theoretical pushrod force available as stroke increases. Figure 29-8 reveals a characteristic "knee" in the force vs stroke relationship. When the stroke extends beyond the knee, pushrod force drops off significantly. The knee normally occurs at about 2.00 inches. The maximum stroke for most air chambers is 2.5 inches, although some "long stroke" chambers have a 3.00-inch (or longer) maximum stroke. By definition, no pushrod force is produced at or beyond maximum stroke.

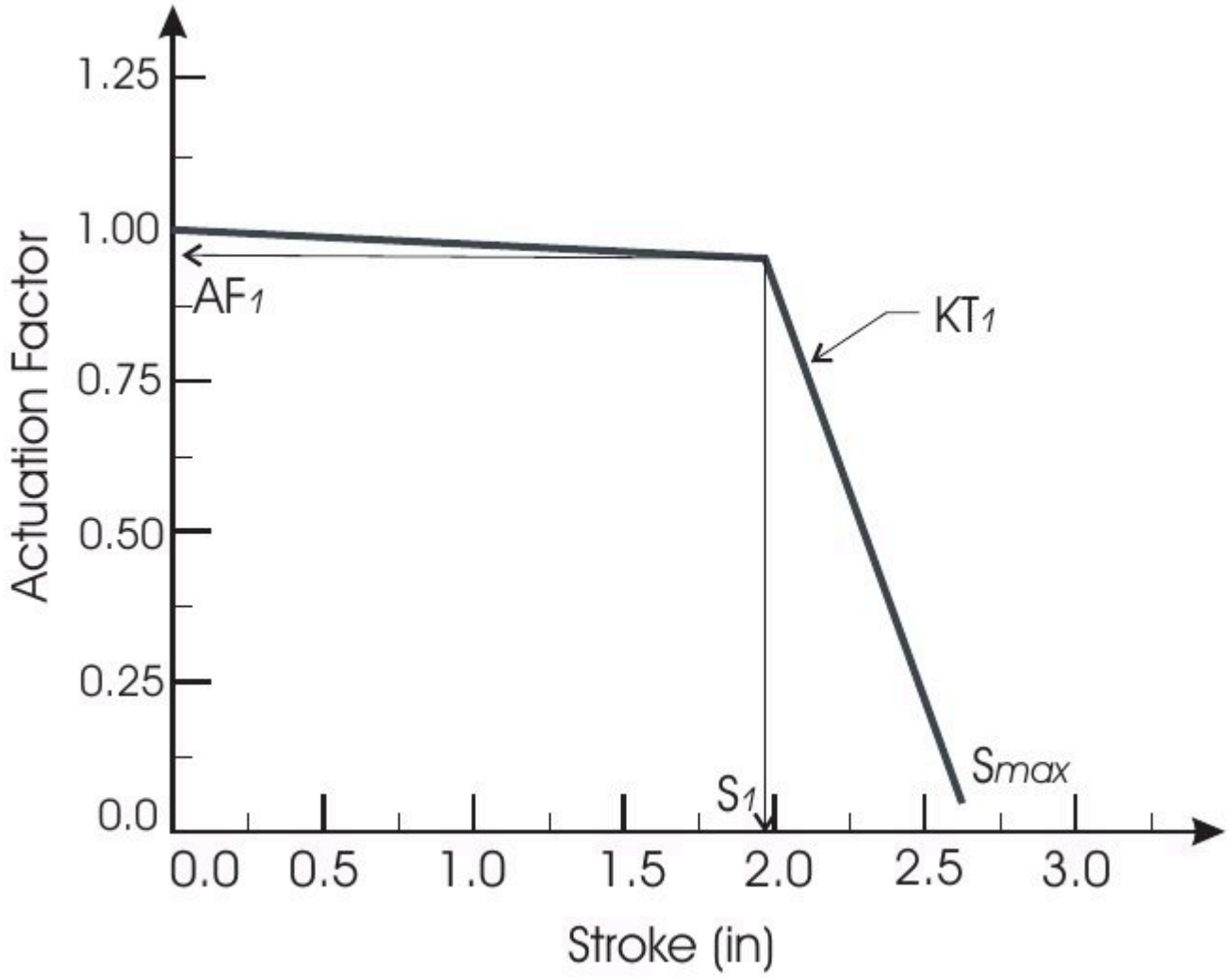


Figure 29-8 — Actuation Factor vs stroke relationship for a typical Type 30 S-cam brake, showing the knee at (S_1, AF_1), the expansion coefficient KT_1 and the maximum stroke Smax.

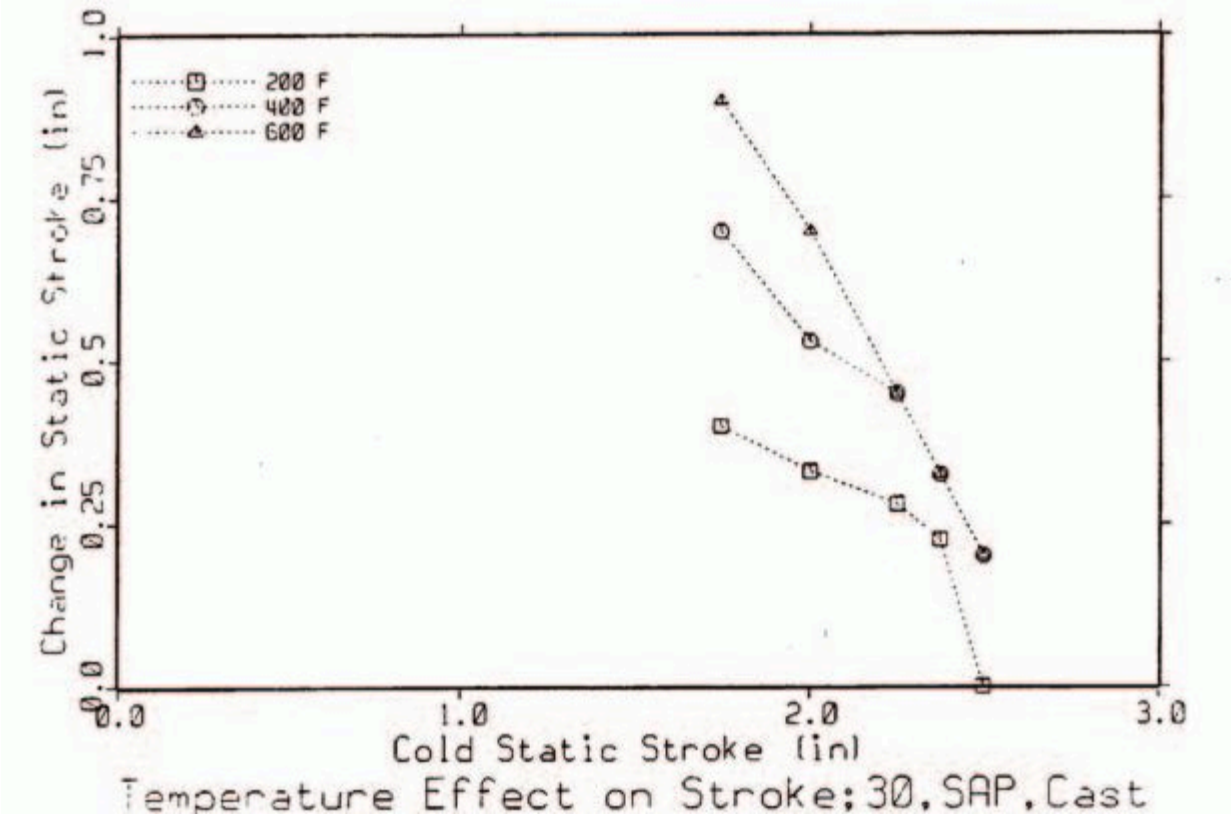


Figure 29-9 — Effect of temperature on brake stroke for a Type 30 cast iron drum S-cam brake [4.31].

Increased piston stroke due to increased brake temperature is also empirically derived. Testing [4.2, 4.31] has revealed that stroke increases approximately linearly with brake temperature. Figure 29-9 illustrates this relationship. Values of 0.0004 to 0.0008 in/deg F are commonly found in the literature; 0.0006 in/deg F is used as the default in the HVE Brake Designer.

The brake actuation factor is calculated using the following logic:

Inputs

Symbol	Description
S_{100}	Measured stroke at 100 psi (in), measured cold
S_1	Stroke at "knee" in force vs stroke relationship (in), (see Figure 29-8)
AF_1	Actuation Factor at S_1 (%/100)
S_{max}	Maximum stroke (in)
KT_1	Expansion Coefficient at S_1 (in stroke/deg F)
T_i	Current temperature (calculated by simulation) at the lining/drum interface (deg F)
T_a	Ambient temperature (deg F)

Using these inputs, the following steps are used to calculate the actuation factor at the current brake temperature:

Step 1 — Calculate T_i (this step is performed by the simulation model for the current simulation time).

Step 2 — Calculate δ_s, the increase in stroke at T_i:

$$\delta_s = KT_1(T_i - T_a) \qquad (29\text{-}12)$$

Step 3 — Calculate current stroke at T_i:

$$S = S_{100} + \delta_s \qquad (29\text{-}13)$$

Step 4 — Calculate current actuation factor, AF. If $S < S_1$:

$$AF = AF_1 + \left(\frac{S_1 - S}{S_1}\right) \times (1.0 - AF_1) \qquad (29\text{-}14)$$

otherwise,

$$AF = AF_1 - \left(\frac{S - S_1}{S_{max} - S_1}\right) \times AF_1 \qquad (29\text{-}15)$$

Brake Fade

An important issue when dealing with brake systems, from both a design as well as a safety standpoint, is the subject of brake fade. Brake fade is the in-use loss of braking capability (when compared to the design capability). Brake fade is normally associated with two factors:

- Reduction in lining friction at extremely high lining temperatures
- Reduction in actuation force due to drum expansion at high drum temperatures

The HVE Brake Designer allows users to study brake fade resulting from both of these sources. Lining temperature calculations are accomplished in the Brake Temperature Model, described in Chapter 30; loss in actuation force was discussed earlier in this chapter (see Actuation Factor).

Table 29-2 — HVE Brake Model Parameters (see Figures 29-4 to 29-8)

Parameter	Units (US / SI)	Description
BrakeType	—	Brake assembly type (Generic, Disc, Duo-servo, Duplex, Single Piston, Dual Piston, S-Cam, Single Wedge, Dual Wedge, Air Disc)
LiningAngle, α_0	deg	Included angle spanned by brake shoe lining
Alpha1, α_1	deg	Angular dimension used in pinned brake shoes (see Figure 29-4)
MuLining, μ_L	—	Friction coefficient between the lining and drum or pad and rotor
BrakeBlockWidth, h	in / cm	Distance between supports for pinned brake shoes (see Figure 29-4)
DrumRadius, R	in / cm	1/2 the diameter of the brake drum
DiscOuterRadius, R_o	in / cm	1/2 the outer diameter of the brake rotor
DiscInnerRadius, R_i	in / cm	1/2 the inner diameter of the brake rotor
LiningWidth	in / cm	Width of lining material
LiningThickness	in / cm	Thickness of lining material
PadIncludedAngle	deg	Included angle formed by pad
ShoePinRadius, a'	in / cm	Radial distance from the center of the brake shoe to the anchor pin for pinned brake shoes
ActuationDist, c	in / cm	Vertical distance from the center of the brake shoe to the actuation piston for abutted brake shoes
AbutmentDist, a	in / cm	Vertical distance from the center of the brake shoe to the lower abutment for abutted brake shoes
AbutmentWidth, o	in / cm	1/2 the horizontal distance from the center of the brake shoe to the lower abutment for abutted brake shoes
ArcCenterAngle, β	deg	Angle from the horizontal centerline to the midpoint of the span defined by α_0 (shown positive in Figure 29-5)
AbutmentAngle, Ψ	deg	Angle of abutment surface relative to brake shoe support surface
AbutmentFriction, μ_s	—	Coefficient of sliding friction between the abutment surface and the brake shoe support surface
LinePress	psi / kPa	Current brake system pressure at the inlet to the piston or air chamber (after proportioning)
PushoutPress	psi / kPa	Pressure required to overcome force associated with return springs and begin developing brake torque
PistonArea	in² / cm²	Effective area for the wheel cylinder (hydraulic systems) or air chamber (air systems)
NPistons	—	Number of pistons in a disc brake caliper (applies only to disc brakes)
η_{Mech}	%/100	Mechanical efficiency of the actuation system (caliper, wheel cylinder, air chamber/slack adjuster/S-cam, wedge)
LSlack	in / cm	Lever arm length of slack adjuster
RS-cam	in / cm	Effective radius of the S-cam (normally 0.5 inches)

Parameter	Units (US / SI)	Description
WedgeAngle, α	deg	Included angle of actuation wedge
Initial Slack, S_{100}	in / cm	Pushrod stroke at 100 psi, measured cold
Stroke knee, S_1	in / cm	Pushrod stroke at knee in force vs stroke relationship (see Figure 29-8)
AF_1	%/100	Actuation factor at S_1
S_{max}	in / cm	Maximum pushrod stroke
KT_1	in/deg F / cm/deg C	Empirical expansion coefficient defining the increase in stroke due to temperature-related brake expansion

*(updated: the original table also listed **NRotors** (number of rotors in a disc brake assembly). This parameter has been removed from the current Disc Brake designer dialog — the model uses a single rotor per wheel. See the Disc Brake dialog reference.)*

Disc Brake

The brake torque for a disc brake is:

$$BrakeTorque = BF \times TR \times AF$$

where the brake factor, BF, is calculated using equation 29-2, the torque radius, TR, is calculated using equation 29-6, and the actuation force, AF, is calculated using equation 29-8. *(The original text cited equations "29-1" and "30-6" here; the correct references are 29-2 and 29-6.)*

The HVE Brake Designer dialog for disc brakes is shown in Figure 29-10.

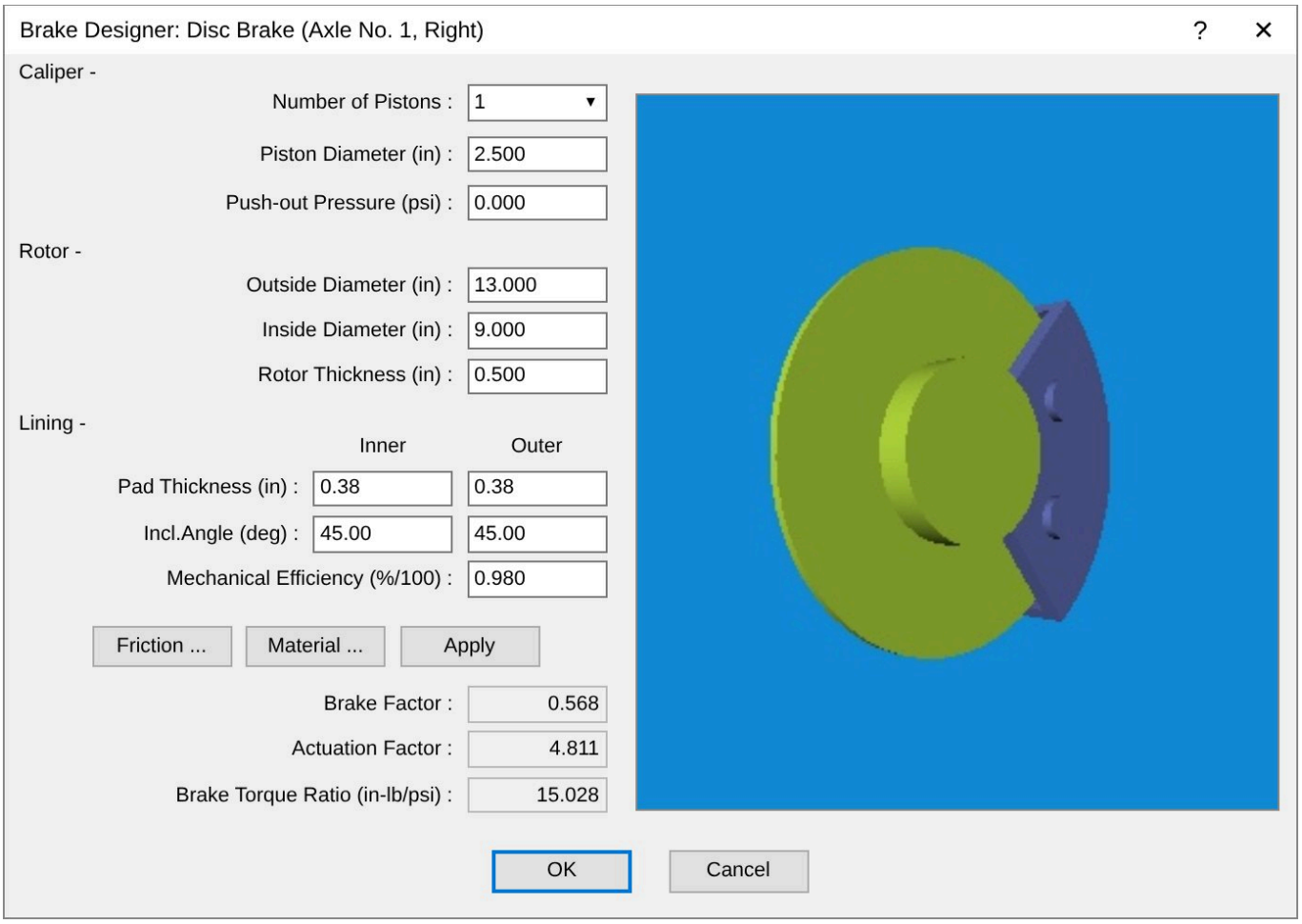


Figure 29-10 — HVE Brake Designer for disc brake types.

The lining friction and material attributes are selectable from the dialog's pushbuttons. The lining friction attributes are shown earlier in this chapter (see Lining Friction). The material attributes, used for thermal analysis of the brake assembly, are described in Chapter 30.

See also the detailed dialog reference: Disc Brake dialog.

Duo-Servo Drum Brake

The brake factor for a duo-servo drum brake is calculated by assuming the primary shoe is an abutted, leading shoe and the secondary shoe is a pinned leading shoe. A duo-servo brake is a unique type of brake in that both shoes are self-energizing, and the secondary shoe's brake factor is increased by the first shoe's brake factor. Thus its brake factor is quite high.

The brake factor for the primary shoe is calculated using equation 29-4 for an abutted leading shoe. The brake factor for the secondary shoe is calculated using equation 29-3 for a pinned leading shoe. The brake factor for the secondary shoe is multiplied by the following factor, which accounts for abutment force from the primary shoe:

$$BF_{MOD} = \frac{c + BF_1 \, r}{a} \qquad (29\text{-}16)$$

where c, a and r are as defined earlier in equations 29-4/29-5, and BF_1 is the brake factor for the abutted leading shoe.

Thus, the total brake factor for a duo-servo brake is:

$$BF = BF_1 + BF_{MOD}\, BF_2 \qquad (29\text{-}17)$$

The torque radius, TR, is calculated using equation 29-7, and the actuation force is calculated using equation 29-9. The resulting brake torque for a duo-servo drum brake is:

$$BrakeTorque = BF \times TR \times AF$$

The HVE Brake Designer dialog for duo-servo brakes is shown in Figure 29-11.

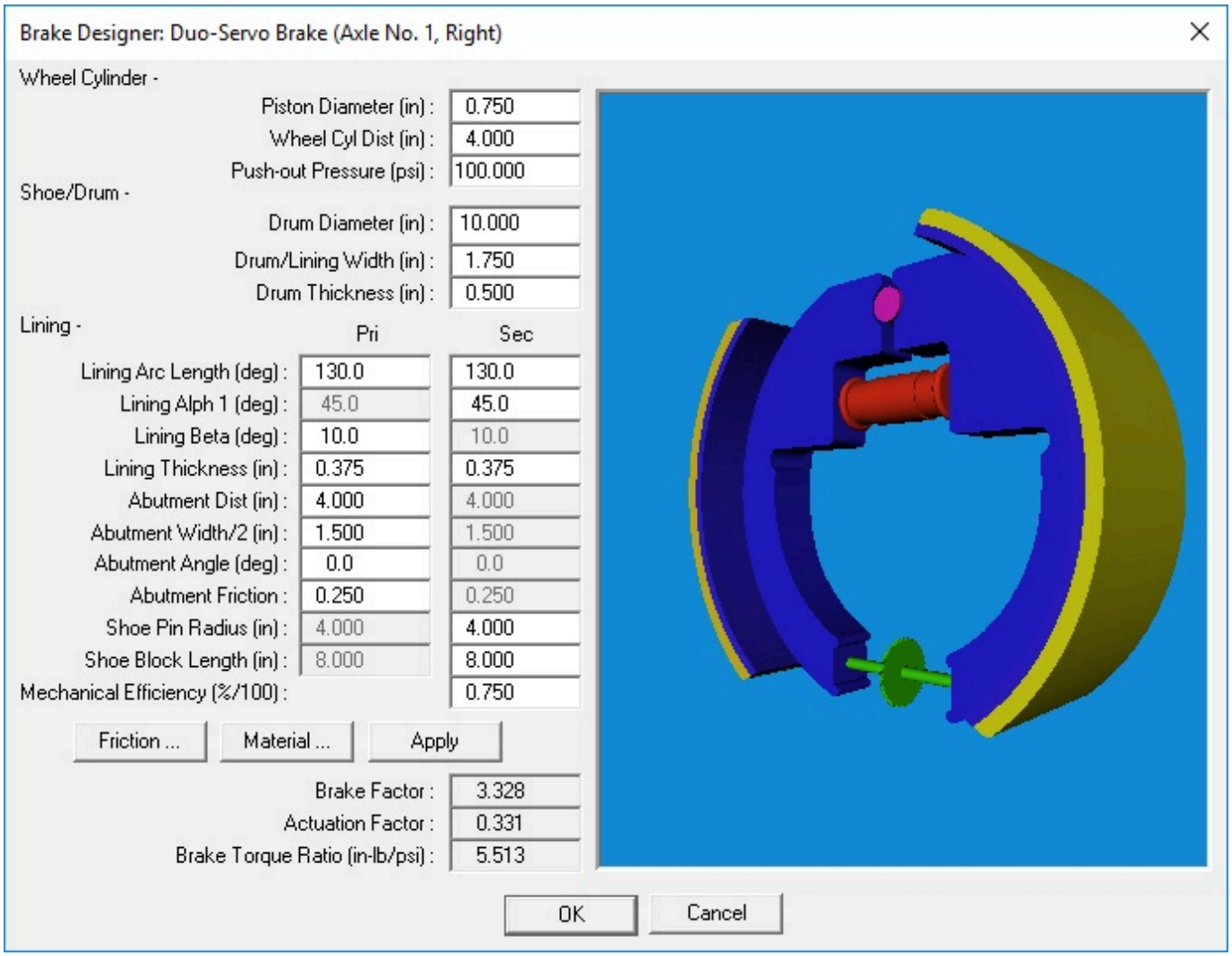


Figure 29-11 — HVE Brake Designer for duo-servo drum brake types.

See also the detailed dialog reference: Duo-Servo Drum Brake dialog.

Duplex Drum Brake

The brake factor for a duplex drum brake is calculated by assuming the primary and secondary shoes are each abutted, leading shoes.

The brake factor for both shoes is calculated using equation 29-4 for an abutted leading shoe. Thus, the total brake factor for a duplex brake is:

$$BF = BF_1 + BF_2$$

(updated: the original text printed this as $BF = 2 \times BF_1$; HVE calculates the two shoes' brake factors separately — allowing different front and rear lining angles — and sums them, which reduces to $2 \times BF_1$ when the shoes are identical.)

The torque radius is calculated using equation 29-7, and the actuation force is calculated using equation 29-9. The resulting brake torque for a duplex drum brake is:

$$BrakeTorque = BF \times TR \times AF$$

The HVE Brake Designer dialog for duplex drum brakes is shown in Figure 29-12.

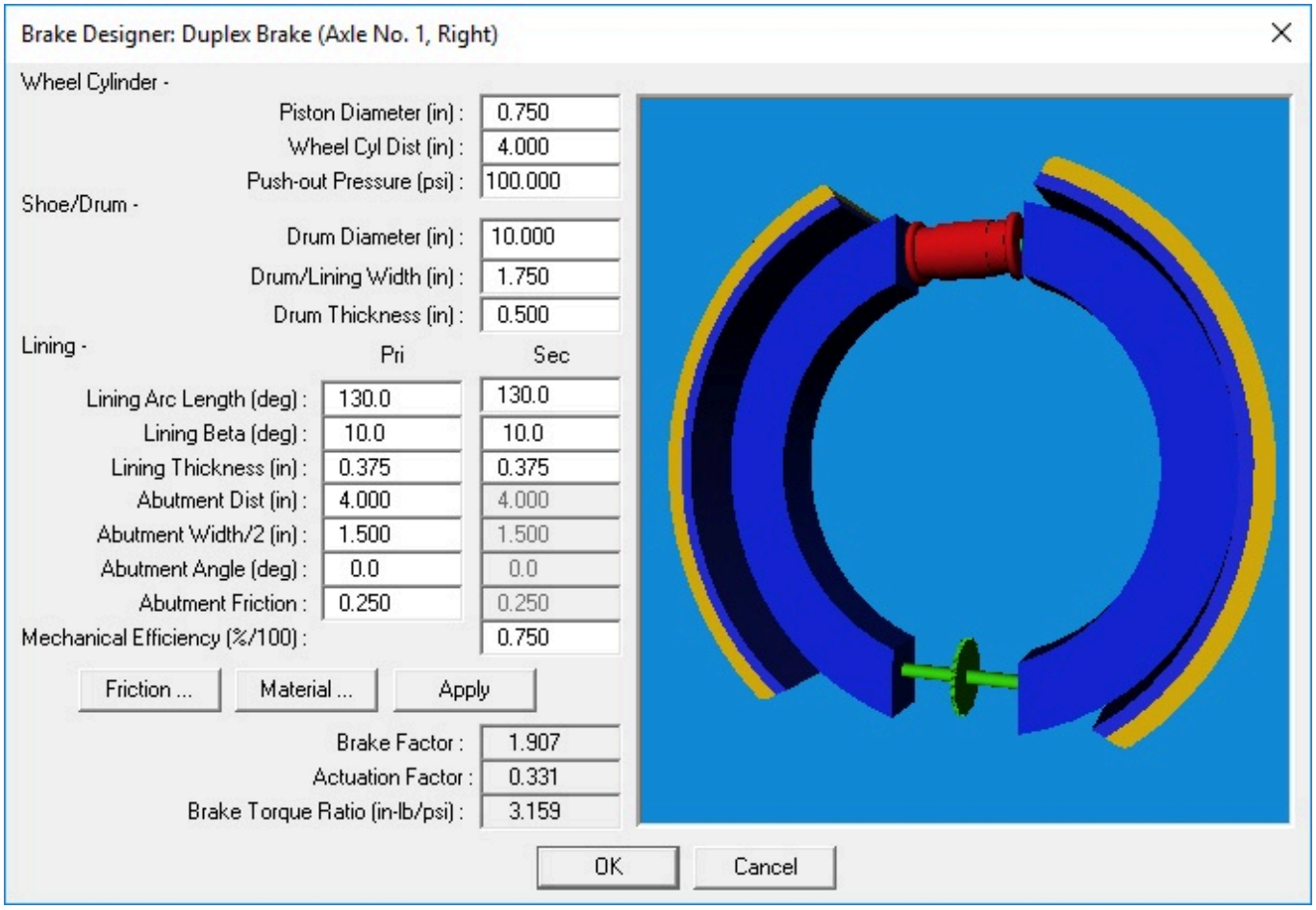


Figure 29-12 — HVE Brake Designer for duplex drum brake types.

See also the detailed dialog reference: Duplex Drum Brake dialog.

Single Piston Drum Brake

The brake factor for a single piston drum brake is calculated by assuming the primary shoe is a pinned leading shoe and the secondary shoe is a pinned trailing shoe. The brake factor for the primary shoe is calculated using equation 29-3 for a pinned leading shoe using the negative sign in the denominator. The brake factor for the secondary shoe is again calculated using equation 29-3 for a pinned trailing shoe, except the positive sign is used in the denominator. The torque radius is calculated using equation 29-7.

Some single piston drum brakes use a different piston area for the primary and secondary shoes. Therefore, the actuation force is calculated separately for the primary and secondary shoes. Equation 29-9 is used for both shoes. For the primary shoe, the actuation force is:

$$AF_{Primary\,Shoe} = (P_{Line} - P_{Pushout}) \times A_{Primary\,Piston}$$

and for the secondary shoe, the actuation force is:

$$AF_{Secondary\,Shoe} = (P_{Line} - P_{Pushout}) \times A_{Secondary\,Piston}$$

Although only one dual-acting wheel cylinder is used, it may have different sized pistons on each end. Therefore, the primary and secondary pistons may have a different size.

The resulting brake torque for a single piston drum brake is:

$$BrakeTorque = (BF_1\ AF_{Primary\,Shoe} + BF_2\ AF_{Secondary\,Shoe}) \times TR \times \eta_{Mech}$$

(updated: the original text printed the torque as $(AF_{Primary} + AF_{Secondary}) \times TR \times \eta_{Mech}$, omitting the brake factors; HVE applies each shoe's brake factor to its own actuation force before summing.)

The HVE Brake Designer dialog for single piston drum brakes is shown in Figure 29-13.

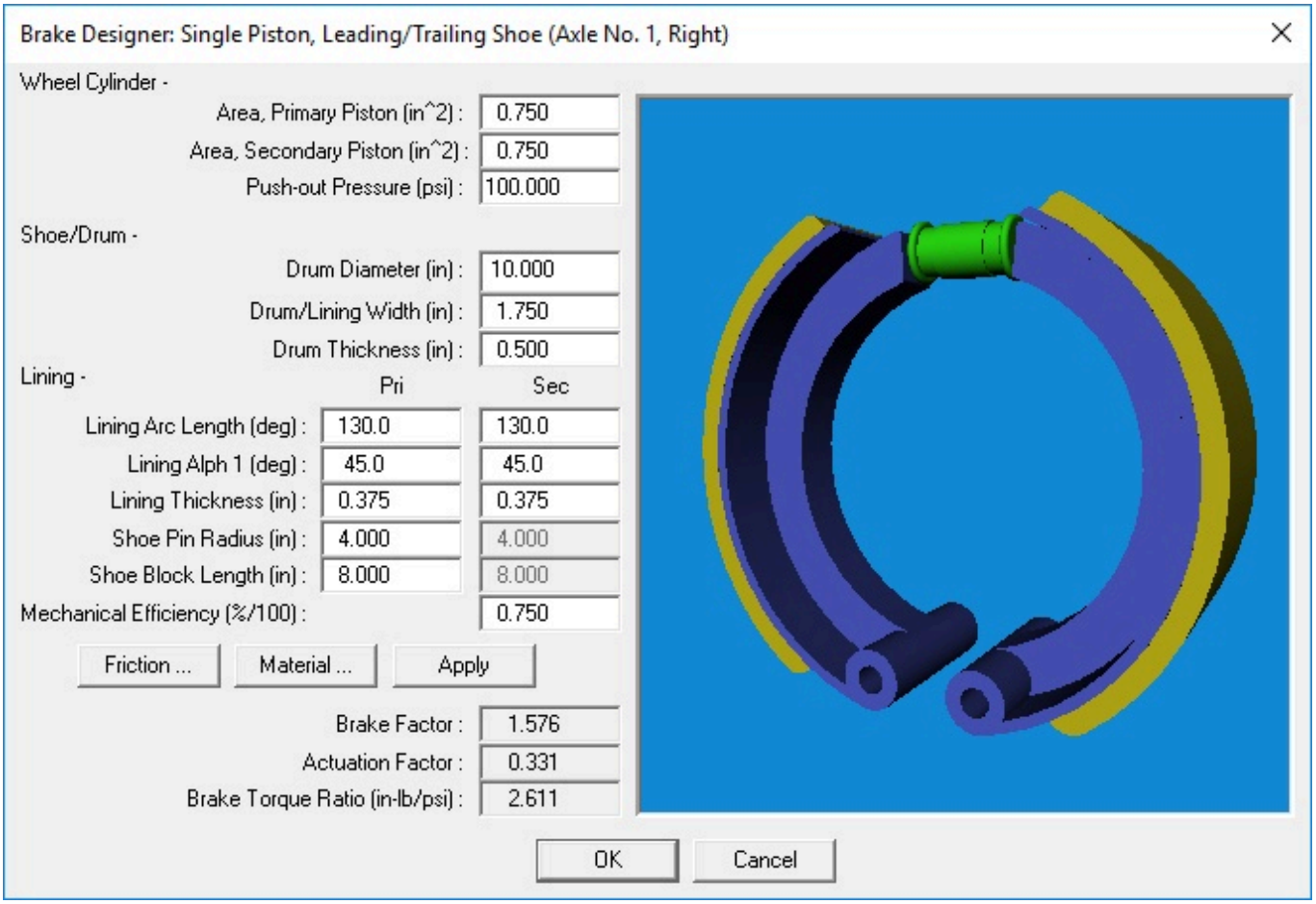


Figure 29-13 — HVE Brake Designer for single piston drum brake types.

See also the detailed dialog reference: Single Piston Drum Brake dialog.

Dual Piston Drum Brake

The brake factor for a dual piston drum brake is calculated by assuming the primary and secondary shoes are each a pinned leading shoe, using equation 29-3. The torque radius is calculated using equation 29-7.

Like a single piston brake, some dual piston drum brakes use a different piston area for the primary and secondary shoes. Therefore, the actuation force is calculated separately for the primary and secondary shoes. Equation 29-9 is used for both shoes. For the primary shoe, the actuation force is:

$$AF_{Primary\,Shoe} = (P_{Line} - P_{Pushout}) \times A_{Primary\,Piston}$$

and for the secondary shoe, the actuation force is:

$$AF_{Secondary\,Shoe} = (P_{Line} - P_{Pushout}) \times A_{Secondary\,Piston}$$

The resulting brake torque for a dual piston drum brake is:

$$BrakeTorque = (BF_1\ AF_{Primary\,Shoe} + BF_2\ AF_{Secondary\,Shoe}) \times TR \times \eta_{Mech}$$

(updated: as for the single piston brake, HVE applies each shoe's brake factor to its own actuation force.)

The HVE Brake Designer dialog for dual piston drum brakes is shown in Figure 29-14.

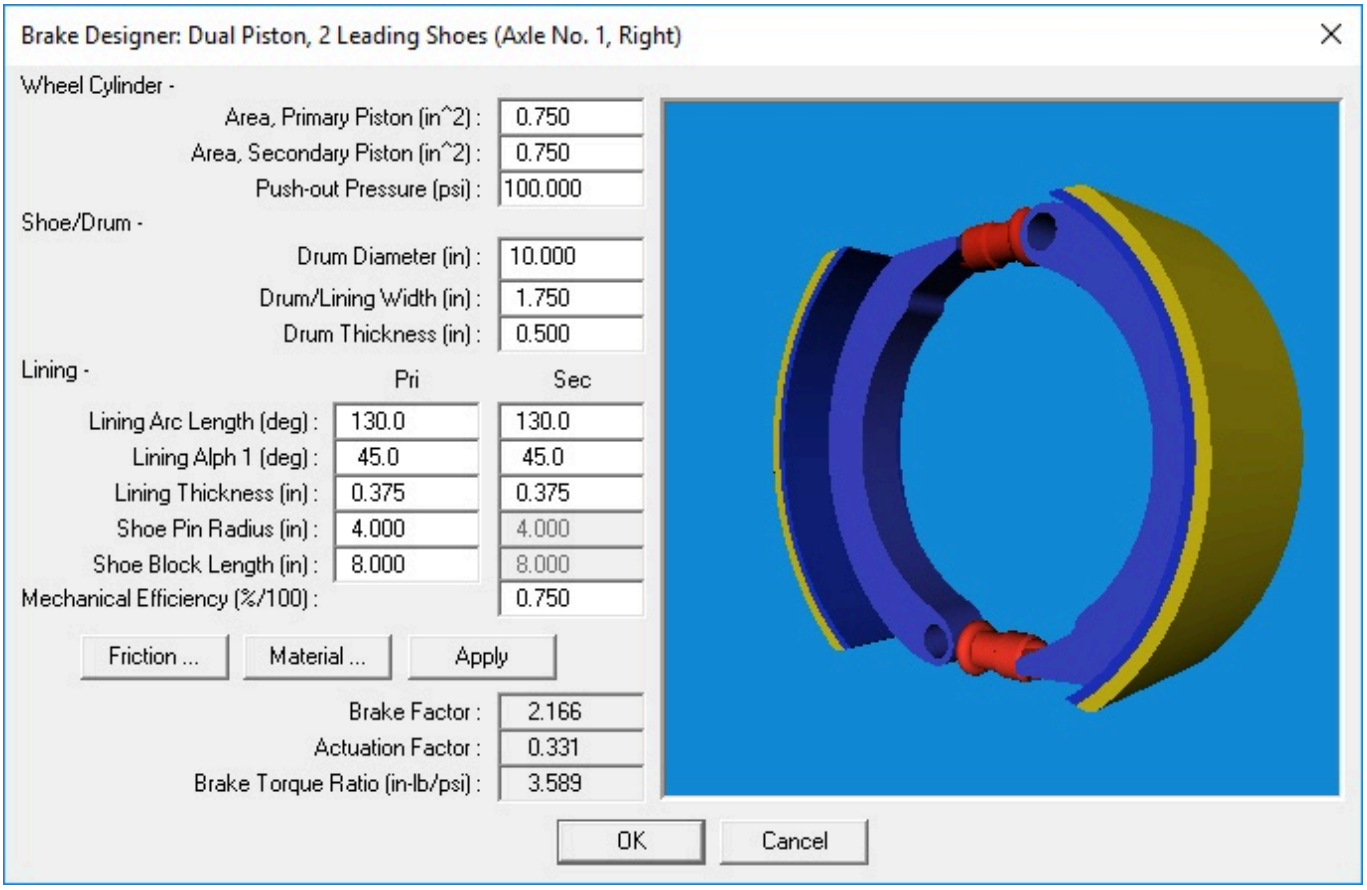


Figure 29-14 — HVE Brake Designer for dual piston drum brake types.

See also the detailed dialog reference: Dual Piston Drum Brake dialog.

S-Cam Drum Brake

The brake factor for an S-cam drum brake is calculated by assuming the primary shoe is a pinned leading shoe and the secondary shoe is a pinned trailing shoe. The brake factor for the leading shoe is calculated using equation 29-3, with the negative sign in the denominator. The brake factor for the secondary shoe is again calculated using equation 29-3 for a pinned trailing shoe, except the positive sign is used in the denominator. Because the S-cam applies equal

displacement

(rather than equal force) to the two shoes, the combined brake factor is:

$$BF = \frac{4\,BF_1\,BF_2}{BF_1 + BF_2}$$

(updated: the original edition summed the leading- and trailing-shoe brake factors, $BF = BF_1 + BF_2$. The current version of HVE uses the equal-displacement relationship above, per Limpert, Eq. 2-22, which correctly accounts for the fixed-cam force redistribution between the shoes.)

Actuation force is calculated using equation 29-10. The torque radius is calculated using equation 29-7.

Unlike the hydraulic brake types, S-cam brakes use an actuation factor to account for the level of adjustment. The actuation factor is calculated using equations 29-12 through 29-15.

The resulting brake torque for an S-cam drum brake is:

$$BrakeTorque = BF \times TR \times AF \times Factor$$

The HVE Brake Designer dialog for S-cam drum brakes is shown in Figure 29-15.

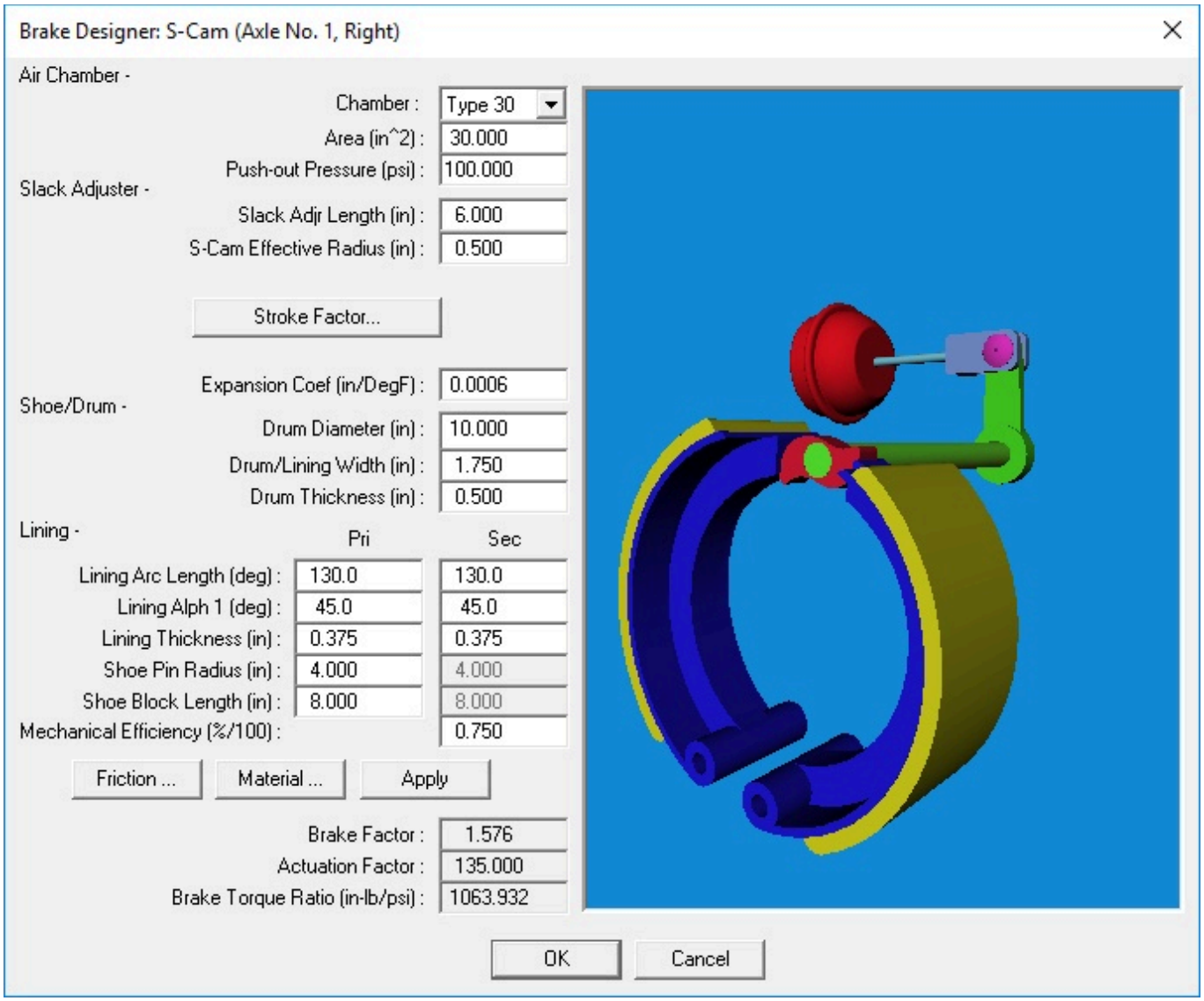


Figure 29-15 — HVE Brake Designer for S-cam drum brake types.

Single Wedge Drum Brake

The brake factor for a single wedge drum brake is calculated by assuming the primary shoe is a pinned leading shoe and the secondary shoe is a pinned trailing shoe. The brake factor for the leading shoe is calculated using equation 29-3, with the negative sign in the denominator. The brake factor for the secondary shoe is again calculated using equation 29-3 for a pinned trailing shoe, except the positive sign is used in the denominator. The total brake factor is the sum of the two shoe brake factors. The torque radius is calculated using equation 29-7. The actuation force is calculated using equation 29-11 and the actuation factor is calculated using equations 29-12 through 29-15.

The resulting brake torque for a single wedge drum brake is:

$$BrakeTorque = BF \times TR \times AF \times Factor$$

The HVE Brake Designer dialog for single wedge drum brakes is shown in Figure 29-16.

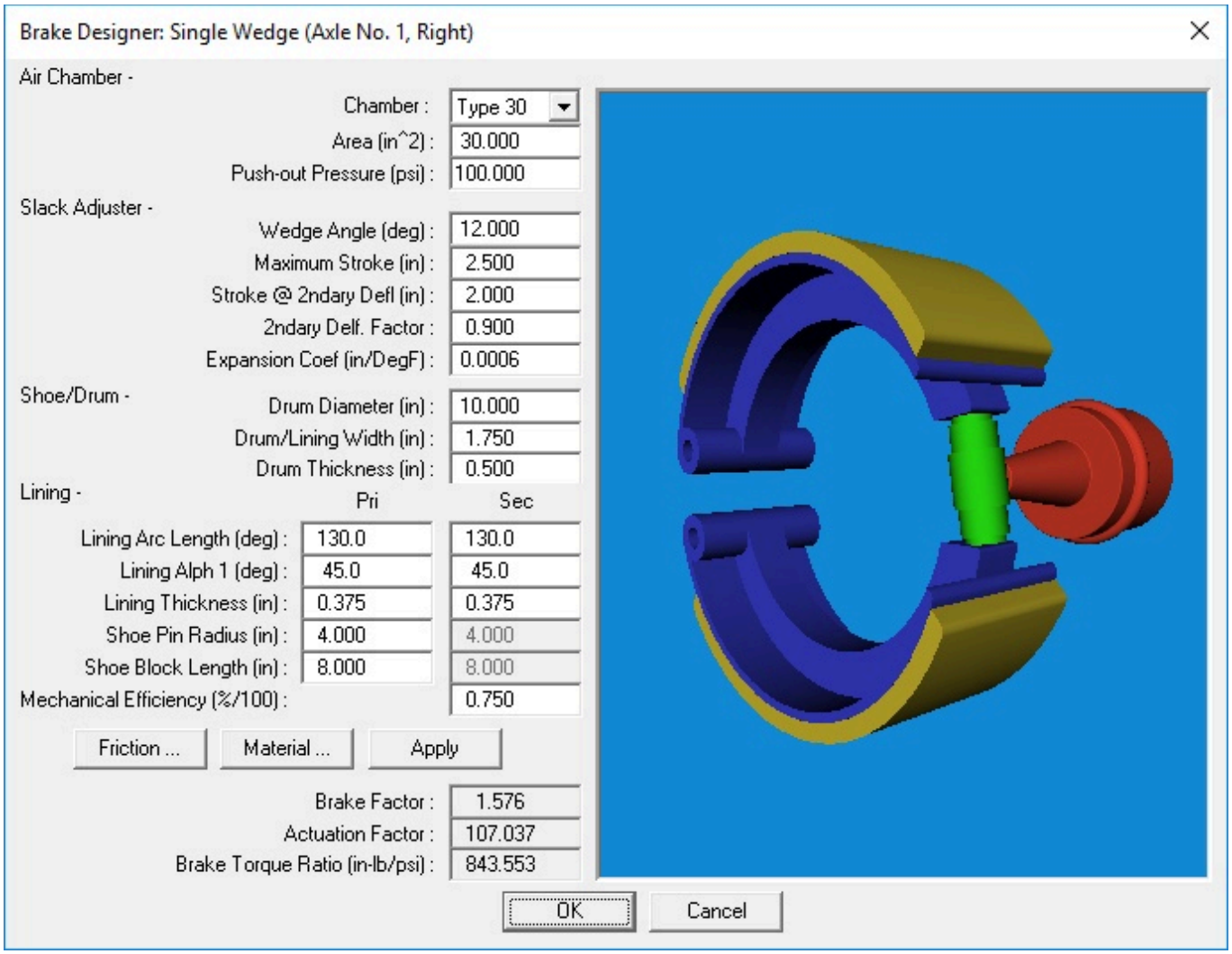


Figure 29-16 — HVE Brake Designer for single wedge drum brake types.

Dual Wedge Drum Brake

The brake factor for a dual wedge drum brake is calculated by assuming the primary and secondary shoes are both abutted leading shoes, using equation 29-4 for each shoe. The total brake factor is the sum of the two shoe brake factors. The torque radius is calculated using equation 29-7. The actuation force is calculated using equation 29-11 and the actuation factor is calculated using equations 29-12 through 29-15.

The resulting brake torque for a dual wedge drum brake is:

$$BrakeTorque = BF \times TR \times AF \times Factor$$

The HVE Brake Designer dialog for dual wedge drum brakes is shown in Figure 29-17.

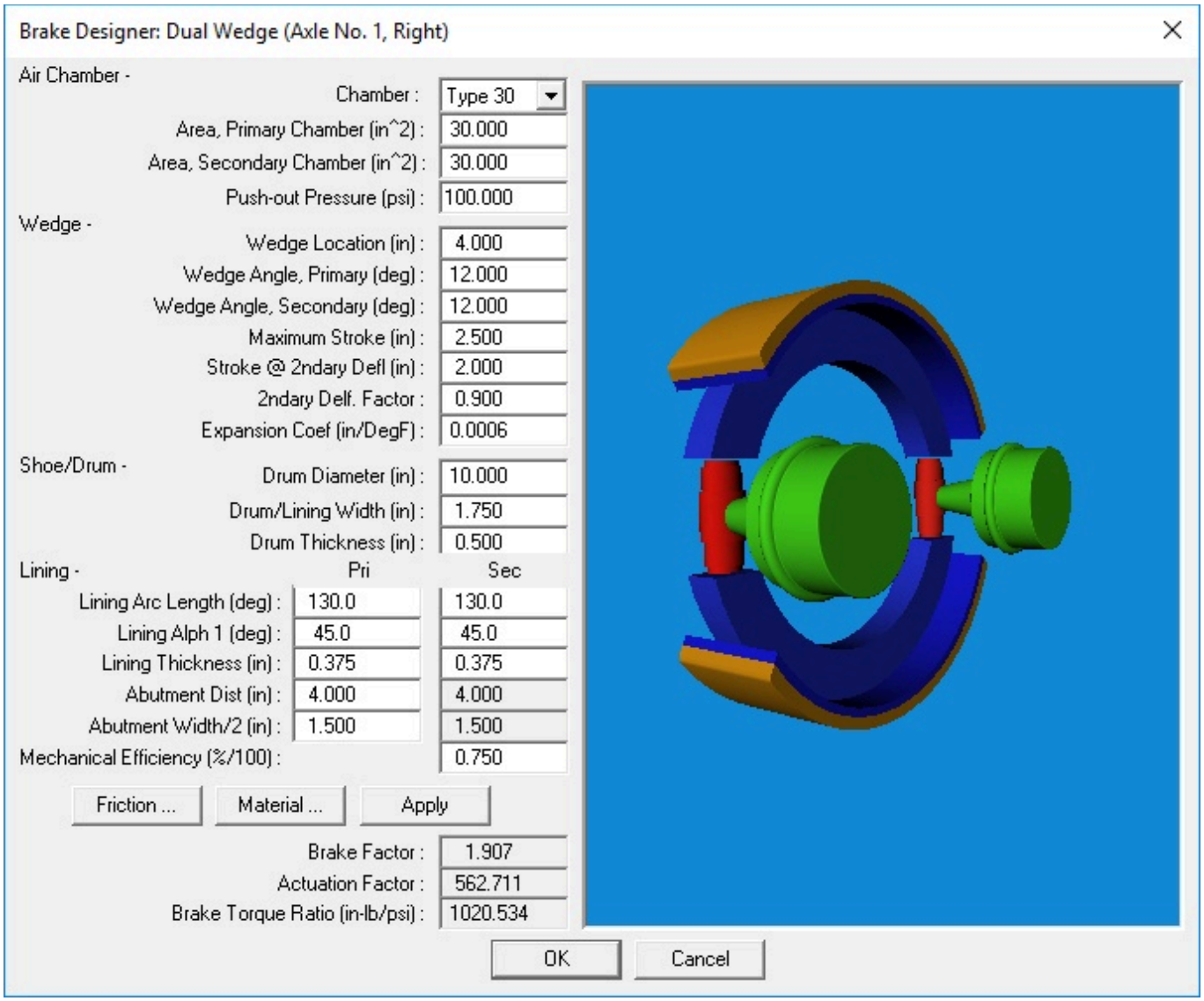


Figure 29-17 — HVE Brake Designer for dual wedge drum brake types.

See also the detailed dialog reference: Dual Wedge Drum Brake dialog.

Air Disc Brake

(updated: this brake type was added after the original edition of the manual and is documented here from the current version of HVE.)

The air disc brake combines the caliper disc brake friction model with air-chamber actuation. The brake factor is calculated using equation 29-2 (caliper brake, $BF = 2\mu_L$), and the torque radius using equation 29-6. The actuation force is produced by an air chamber acting through an internal lever (eccentric) mechanism:

$$ActuationForce = (P_{Line} - P_{Pushout}) \times A_{AirChamber} \times \eta_{Mech} \times L_{Ratio}$$

where L_{Ratio} is the internal lever (slack adjuster) ratio of the caliper mechanism.

Like the S-cam and wedge brakes, the air disc brake actuation force is multiplied by the stroke-based actuation factor calculated using equations 29-12 through 29-15, allowing out-of-adjustment and thermal-expansion effects to be studied.

The resulting brake torque for an air disc brake is:

$$BrakeTorque = BF \times TR \times AF \times Factor$$

Notes on the Brake Designer Dialogs

- Each per-type Brake Designer dialog displays an embedded schematic preview drawing of the brake, updated to reflect the current dimensions. *(The legacy dialogs included a separate Preview button; it has been removed — the preview is now always displayed.)*
- The **Friction ...** button opens the Lining Friction Properties dialog (friction vs temperature and friction factor vs sliding speed tables; see Figure 29-3).
- The **Material ...** button opens the Brake Designer Material Properties dialog, used for the thermal analysis described in Chapter 30.
- The **Apply** button recomputes the Brake Factor, Actuation Factor and Brake Torque Ratio from the current physical parameters.

Chapter 30 — Brake Temperature Model

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 30, pages 30-1 through 30-16, revised to describe the current HVE brake temperature model; see also the Brake Designer Material Properties dialog reference.

Overview

During braking, the vehicle's kinetic energy is converted into heat energy by the wheel brake assemblies. This energy causes a temperature rise in the brake assembly components (linings and rotor or drum) that is dissipated into the atmosphere through conduction and convection.

The modeling procedure used in the HVE Brake Temperature Model was adapted from the lumped mass method proposed by MacAdam, et al [4.2]. This chapter describes how brake temperatures are calculated. This description involves four subjects:

- Modeling Technique
- Model Inputs
- Model Outputs
- Equations

Temperature change affects the performance of the wheel brake assemblies by expansion of the components and by altering the friction of the lining material. Therefore, the HVE Brake Designer (see Chapter 29) includes the temperatures calculated by this model as an independent input to the equations for brake torque.

Modeling Technique

Calculating the temperature of brake system components is a classic heat flow problem. $\dot{H}$, the rate of work (heat energy) into the system is:

$$\dot{H} = Work \times Rate \qquad (30\text{-}1)$$

In the case of a wheel brake assembly, the rate of work at any instant is:

$$\dot{H} = Brake\ Torque \times Wheel\ Spin\ Velocity \qquad (30\text{-}2)$$

This energy is absorbed into the brake linings and drum/rotor through conduction, stored through capacitance and removed through convection.

> **NOTE:** Radiant heat transfer is considered negligible for the range of temperatures encountered and, thus, is not considered by the model.

Using a lumped mass model, the rotor/drum is divided into 10 nodes across its thickness, the lining is one node and the lining and rotor or drum exterior surfaces each represent a node. Thus, the model includes a total of 13 nodes at which the temperature may be calculated (see Figure 30-1). Associated with each node are its thermal parameters: conductivity, convectivity and heat capacitance. At any instant the First Law of Thermodynamics (i.e., an energy balance) may be expressed as:

$$[M][\frac{dT}{dt}] = [S][T] + [C]\,T_{L/D} + [D]\,T_A \qquad (30\text{-}3)$$

where

Symbol	Description
$[M]$	Capacitance matrix
$[\frac{dT}{dt}]$	Thermal rate matrix
$[S]$	Internal Temperature Coefficient matrix
$[T]$	Temperature matrix
$[C]$	Boundary condition conductivity matrix
$T_{L/D}$	Interface temperature
$[D]$	Boundary condition convectivity matrix
T_A	Ambient temperature

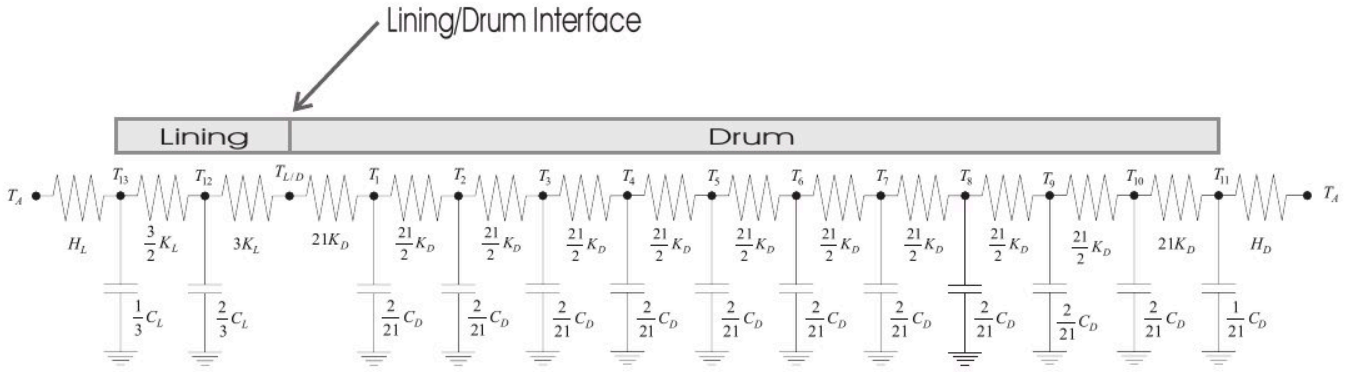


Figure 30-1 — Representation of the 13-node lumped parameter brake temperature model [4.2]: ambient air — lining exterior surface node T_{13} — lining node T_{12} — lining/drum interface T_L/D — drum/rotor nodes T_1 … T_{10} — drum/rotor exterior surface node T_{11} — ambient air, with conductances (3/2 K_L, 3K_L on the lining side; 21K_D, 21/2 K_D between drum nodes), convection coefficients H_L and H_D at the exposed surfaces, and nodal capacitances (C_L/3, 2C_L/3 for the lining; 2C_D/21 for interior drum nodes and C_D/21 at the drum surface).

This matrix equation is solved for $[T]$, the temperature at each node, at each integration timestep. The outputs of interest are T_5 (the temperature at the interior of the drum or rotor), $T_{L/D}$ (the temperature at the interface between the lining and drum or rotor), and T_{12} (the temperature at the interior of the lining). The specific model inputs, outputs and solution procedure are described below.

Model Inputs

The inputs to the lumped mass model are:

- **Drum Diameter (Drum Brake),** D_D — Inside diameter of the brake drum. The torque radius (half the drum diameter) is the lever arm of the force applied by the brake lining.
- **Rotor Inner and Outer Diameter (Disc Brake),** D_i, D_o — Rotor outer and inner diameter. The torque radius is defined as the midpoint between the inner and outer radii.
- **Lining Width (Drum Brake),** W_L — Width of the rubbing surface in the inside diameter of the drum.

- **Included Angle (Disc Brake),** γ_L — Pad included angle. The included angle and inner and outer diameter define the pad contact surface area with the rotor.
- **Drum/Rotor Specific Heat,** C_{pD} — Specific heat of the drum or rotor material.
- **Drum/Rotor Material Density,** ρ_D — Density of the drum or rotor.
- **Drum/Rotor Conduction Coefficient,** K_D — Coefficient of thermal conduction for the drum or rotor material.
- **Drum/Rotor Convection Coefficients,** H_{0D}, H_{1D} — Coefficients of thermal convection at the drum or rotor interface with the surrounding air. The first coefficient, H_{0D}, is the static convection coefficient; the second coefficient, H_{1D}, is the velocity-dependent convection coefficient to account for the effect of vehicle velocity on convective heat transfer.
- **Lining Specific Heat,** C_{pL} — Specific heat of the brake shoe or pad lining material.
- **Lining Mass,** M_L — Mass of the brake shoe or pad lining material, calculated from the product of the volume and material density.
- **Lining Conduction Coefficient,** K_L — Coefficient of thermal conduction for the brake shoe or pad lining material.
- **Lining Convection Coefficients,** H_{0L}, H_{1L} — Coefficients of thermal convection at the brake lining material interface with the surrounding air. The first coefficient, H_{0L}, is the static convection coefficient; the second coefficient, H_{1L}, is the velocity-dependent convection coefficient to account for the effect of vehicle velocity on convective heat transfer.

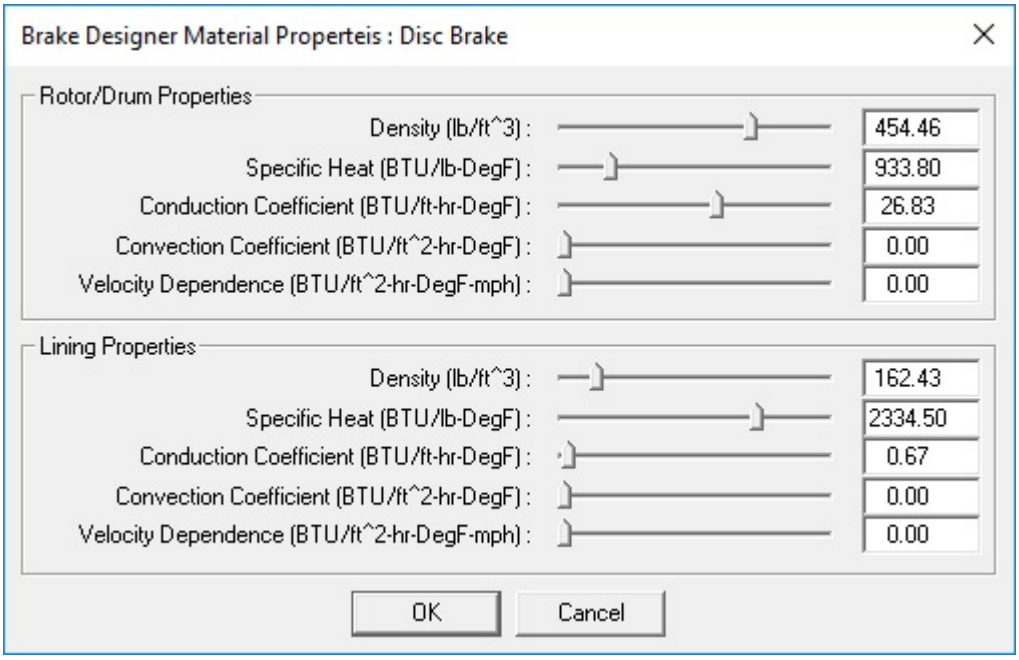


Figure 30-2 — Brake Material Properties dialog. Default material properties are assigned for cast iron drums and rotors, and common lining friction materials. (See the dialog reference page.)

These parameters, shown in Table 30-1, are assigned using the HVE Brake Designer dialogs (see Figures 29-10 through 29-17) and the Brake Material Properties dialog (see Figure 30-2).

The remaining independent parameters for the brake temperature model are assigned by the user during event set-up:

- **Environment Ambient Temperature,** T_A — Ambient temperature, assigned using the Environment Information dialog (see Section Five).

- **Initial Rotor/Drum Temperature,** T_{0D} — Temperature of the drum or rotor at the start of the simulation, assigned using the Event Editor.
- **Initial Lining Temperature,** T_{0L} — Temperature of the lining at the start of the simulation, assigned using the Event Editor (see Section Six).

> **NOTE:** See Table 29-2 for parameter definitions related to shoe and rotor/drum geometry.

Table 30-1 — Brake Material Properties

Parameter	Units (US / SI)	Description
Rotor/Drum Material Density	lb/ft³ / kg/m³	Density of rotor or drum material (used for calculating the mass of the rotor or drum)
Rotor/Drum Specific Heat	BTU/lb-deg F / kJ/kg	Change in enthalpy of the brake rotor or drum material with respect to temperature at constant pressure
Rotor/Drum Conduction Coefficient	BTU/ft-hr-deg F / Joule/cm-sec-DegR	Energy transfer coefficient through the brake rotor or drum material
Rotor/Drum Convection Coefficient	BTU/ft²-hr-deg F / Joule/cm²-sec-DegR	Static convective energy transfer coefficient between the brake rotor or drum surface and adjacent air
Rotor/Drum Velocity-Dependent Convection Coefficient	BTU/ft²-hr-deg F-mph / Joule-sec/cm³-sec-DegR	Velocity-dependent energy transfer coefficient between the brake rotor or drum surface and adjacent air (i.e., this coefficient is multiplied by vehicle linear velocity to determine the convection rate)
Lining Material Density	lb/ft³ / kg/m³	Density of lining material (used for calculating the mass of the pads or shoe linings)
Lining Specific Heat	BTU/lb-deg F / kJ/kg	Change in enthalpy of the brake lining material with respect to temperature at constant pressure
Lining Conduction Coefficient	BTU/ft-hr-deg F / Joule/cm-sec-DegR	Energy transfer coefficient through the brake lining material
Lining Convection Coefficient	BTU/ft²-hr-deg F / Joule/cm²-sec-DegR	Static convective energy transfer coefficient between the brake lining surface and adjacent air
Lining Velocity-Dependent Convection Coefficient	BTU/ft²-hr-deg F-mph / Joule-sec/cm³-sec-DegR	Velocity-dependent energy transfer coefficient between the brake lining surface and adjacent air (i.e., this coefficient is multiplied by vehicle linear velocity to determine the convection rate)

> **NOTE:** Drum Diameter, Rotor Inner and Outer Diameter, Lining Width and Included Angle are defined in Chapter 29, HVE Brake Designer.

Model Outputs

The outputs from the model are:

- **Lining Exterior Surface Temperature,** T_{LS} — Temperature at the interface between the lining and the outside air (node 13).

- **Lining Internal Temperature,** T_L — Internal lining temperature 1/3 of the distance from the interface to the lining exterior (node 12).
- **Interface Temperature,** $T_{L/D}$ — Temperature at the interface between the lining and drum or rotor. This point is important as it represents the location where heat energy is introduced into the system (see Figure 30-1).
- **Rotor/Drum Internal Temperature,** T_D — Internal temperature of the rotor or drum at node 5.
- **Exterior Rotor/Drum Temperature,** T_{DS} — Temperature at the rotor or drum interface with the outside air (node 1). *(Per the node layout in Figure 30-1, the drum exterior surface is the last drum node, node 11.)*

T_L, $T_{L/D}$ and T_D are available in the Vehicle Output Tracks, Wheel Group (Key Results or Variable Output; see Chapter 15, Event Model).

Model Equations

The fundamental equations for the solution of $T_{L/D}$ and T_N were shown earlier in matrix form (see eq. 30-3). The following sections provide details for the calculation of the interface temperature and matrix coefficients.

Interface Temperature

The temperature at the interface between the lining and drum/rotor is:

$$T_{L/D} = \frac{\dot{H} + 21K_D T_{D_1} + 3K_L T_{D_{10}}}{21K_D + 3K_L} \qquad (30\text{-}4)$$

where T_{D_1} is the temperature of the drum/rotor node adjacent to the interface (node 1) and $T_{D_{10}}$ (as printed in the original) refers to the temperature of the lining node adjacent to the interface (node 12 in Figure 30-1).

Capacitance Matrix, M

The thermal capacitance coefficients are calculated as follows:

Rotor/Drum

$$C_D = C_{PD}\,\rho_D\,V_D \qquad (30\text{-}5)$$

where

Symbol	Description
C_D	Capacitance coefficient for rotor or drum
C_{PD}	Specific heat of rotor or drum material
ρ_D	Density of rotor or drum material
V_D	Volume of rotor or drum

$$V_D = \frac{\pi}{4}(D_o^2 - D_i^2)x_D \quad \text{(Rotor)}$$

where D_o = rotor outer diameter, D_i = rotor inner diameter and x_D = rotor thickness;

$$V_D = \pi D_D W_L x_D \quad \text{(Drum)}$$

where D_D = drum diameter, W_L = lining width and x_D = drum thickness.

Lining

$$C_L = C_{PL}\,\rho_L\,V_L \qquad (30\text{-}6)$$

where

Symbol	Description
C_L	Capacitance coefficient for lining
C_{PL}	Specific heat of lining material
ρ_L	Density of lining material
V_L	Volume of lining material

$$V_L = \frac{\gamma_L\,\pi}{1440}(D_o^2 - D_i^2)x_D \quad \text{(Rotor brake pad)}$$

where γ_L = pad arc length (deg), D_o = outer rotor diameter, D_i = inner rotor diameter and x_D = pad thickness; and

$$V_L = (\frac{\alpha_{o,Pri} + \alpha_{o,Sec}}{360})\pi D_D W_L x_D \quad \text{(Drum brake lining)}$$

where $\alpha_{o,Pri}$ = arc length of primary shoe (deg), $\alpha_{o,Sec}$ = arc length of secondary shoe (deg), D_D = drum diameter, W_L = lining width and x_D = lining thickness.

Using these capacitance coefficients, C_D and C_L, the capacitance matrix, M, for the 13-node lumped mass model is the diagonal matrix:

$$[M] = \text{diag}(\tfrac{2}{21}C_D,\ \underbrace{\cdots}_{\text{drum nodes 1…10}},\ \tfrac{2}{21}C_D,\ \tfrac{C_D}{21},\ \tfrac{2}{3}C_L,\ \tfrac{C_L}{3})$$

(the first ten diagonal entries, for the interior drum/rotor nodes, are $\frac{2}{21}C_D$; the eleventh, for the drum/rotor exterior surface node, is $\frac{C_D}{21}$; the twelfth, for the lining node, is $\frac{2}{3}C_L$; and the thirteenth, for the lining exterior surface node, is $\frac{C_L}{3}$).

Internal Temperature Coefficient Matrix

The internal temperature coefficient matrix is composed of both conduction and convection coefficients. The conduction coefficients are calculated as follows:

Rotor/Drum

$$K_D = \frac{k_D A_D}{x_D} \qquad (30\text{-}7)$$

where

Symbol	Description
K_D	Conductivity coefficient of rotor or drum
k_D	Conductivity of rotor or drum material
A_D	Heat transfer area of rotor or drum
x_D	Thickness of rotor or drum

$$A_D = \frac{\pi}{4}(D_o^2 - D_i^2) \quad \text{(rotor)}$$

where D_o = rotor outer diameter and D_i = rotor inner diameter;

$$A_D = \pi D_D W_L \quad \text{(drum)}$$

where D_D = drum diameter and W_L = lining width.

Lining

$$K_L = \frac{k_L A_L}{x_L} \qquad (30\text{-}8)$$

where

Symbol	Description
K_L	Conductivity coefficient for lining
k_L	Conductivity of lining material
A_L	Heat transfer area of lining
x_L	Thickness of lining

$$A_L = \frac{\gamma \pi}{1440}(D_o^2 - D_i^2) \quad \text{(Disc brake pad)}$$

where γ = included pad angle, D_o = outer rotor diameter and D_i = inner rotor diameter; and

$$A_L = (\frac{\alpha_{o,Pri} + \alpha_{o,Sec}}{360})\pi D_D W_L \quad \text{(Drum brake shoes)}$$

where $\alpha_{o,Pri}$ = included angle of primary shoe, $\alpha_{o,Sec}$ = included angle of secondary shoe, D_D = drum diameter and W_L = lining width.

The convection coefficients are calculated as follows:

Rotor/Drum

$$H_D = (H_{0D} + H_{1D} V) A_D \qquad (30\text{-}9)$$

where

Symbol	Description
H_D	Convectivity matrix coefficient for rotor or drum
H_{0D}	Convection coefficient for rotor or drum
H_{1D}	Velocity-dependent convection coefficient
V	Vehicle linear velocity
A_D	Convective surface area for rotor or drum (same as previously defined rotor or drum area)

Lining

$$H_L = (H_{0L} + H_{1L} V)\, A_L \qquad (30\text{-}10)$$

where

Symbol	Description
H_L	Convectivity matrix coefficient for pad or lining
H_{0L}	Convection coefficient for pad or lining
H_{1L}	Velocity-dependent convection coefficient
V	Vehicle linear velocity
A_L	Convective surface area for pad or lining (same as previously defined pad or lining area)

(The original printing of eq. 30-10 shows "H_{1D}" inside the parentheses; the lining velocity-dependent coefficient H_{1L} is intended, as defined in the symbol list.)

The internal temperature coefficient matrix for the 13-node lumped mass model is the tridiagonal matrix:

$$[S] = \begin{bmatrix}
-\frac{63}{2}K_D & \frac{21}{2}K_D & & & & & & & & & & & \\
\frac{21}{2}K_D & -21K_D & \frac{21}{2}K_D & & & & & & & & & & \\
 & \frac{21}{2}K_D & -21K_D & \frac{21}{2}K_D & & & & & & & & & \\
 & & \ddots & \ddots & \ddots & & & & & & & & \\
 & & & \frac{21}{2}K_D & -\frac{63}{2}K_D & 21K_D & & & & & & & \\
 & & & & 21K_D & -21K_D - H_D & & & & & & & \\
 & & & & & -\frac{9}{2}K_L & \frac{3}{2}K_L & & & & & & \\
 & & & & & \frac{3}{2}K_L & -\frac{3}{2}K_L - H_L & & & & & &
\end{bmatrix}$$

Rows 1 through 10 form the drum/rotor conduction chain; row 11 is the drum/rotor exterior surface node, which includes the drum convection loss term; rows 12 and 13 are the lining node and lining exterior surface node, the latter including the lining convection loss term H_L. *(The original printing shows "$-21K_D - H_L$" in row 11; the drum surface convection coefficient H_D is intended there.)*

Internal Temperature Matrix, T

T is a matrix of dependent temperature results for each of the 10 rotor (or drum) nodes, the lining node and the two surface nodes. The T matrix is as follows:

$$[T] = \begin{bmatrix} T_1 \\ T_2 \\ T_3 \\ T_4 \\ T_5 \\ T_6 \\ T_7 \\ T_8 \\ T_9 \\ T_{10} \\ T_{11} \\ T_{12} \\ T_{13} \end{bmatrix}$$

The T matrix contains the brake model outputs:

- **Lining Temperature,** T_{12} — Temperature at node 12, 1/3 of the distance from the interface to the lining exterior.
- **Rotor/Drum Temperature,** T_5 — Temperature at node 5, approximately half way from the interface to the rotor/drum exterior.

Boundary Condition C Matrix

The boundary condition C matrix provides the heat conduction coefficients for the drum and lining. The conduction coefficients were defined earlier; see eq. 30-7 for the rotor/drum conduction coefficients and eq. 30-8 for the lining conduction coefficients.

The resulting matrix for the 13-node lumped mass model is as follows:

$$[C] = \begin{bmatrix} 21K_D \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 3K_L \\ 0 \end{bmatrix}$$

(the interface heat input enters the system at drum node 1 and lining node 12, the two nodes adjacent to the lining/drum interface).

Boundary Condition D Matrix

The boundary condition D matrix provides the heat convection coefficients between the drum and lining and the surrounding air. The convection coefficients were defined earlier; see eq. 30-9 for the rotor/drum convection coefficients and eq. 30-10 for the lining convection coefficients.

The resulting matrix for the 13-node lumped mass model is as follows:

$$[D] = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ H_D \\ 0 \\ H_L \end{bmatrix}$$

(convective losses to ambient air occur at the drum/rotor exterior surface, node 11, and the lining exterior surface, node 13; the original printing shows these entries as "KA_D" and "KA_L", i.e., the convection coefficient times the surface area, which is the definition of H_D and H_L in eqs. 30-9 and 30-10).

Solution Procedure

The M, S, T, C and D matrices are used in the solution for the current temperature matrix, T. Eq. 30-3, defined earlier, is rearranged by inverting the M matrix. Then, three new working matrices are defined:

$$[S'] = [M]^{-1}[S]$$

$$[C'] = [M]^{-1}[C]$$

$$[D'] = [M]^{-1}[D]$$

where $[M]^{-1}$ is the inverted $[M]$ matrix.

Rearranging eq. 30-3 to solve for $[\frac{dT}{dt}]$ yields:

$$[\frac{dT}{dt}] = [S'][T_{Prev}] + [C']T_{L/D} + [D']T_A \qquad (30\text{-}11)$$

where

Symbol	Description
$[\frac{dT}{dt}]$	Time rate of change in temperature
$[T_{Prev}]$	Temperature at each node during the previous timestep

Finally, the current temperature at each node can be calculated:

$$[T] = [\frac{dT}{dt}]\Delta t + [T_{Prev}] \qquad (30\text{-}12)$$

where

Symbol	Description
Δt	Integration timestep

The resulting temperature at nodes 5 and 12 (internal drum and lining temperatures, respectively) and the temperature at the lining/drum (or pad/rotor) interface are available in the output tracks (Wheels Group). In addition, the lining temperature is used (along with the current wheel spin velocity) by the wheel brake torque calculations (see Chapter 29) to assign the current lining friction coefficient.

Chapter 31 — Antilock Braking Systems

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Chapter 31, pages 31-1 through 31-22, revised to describe the current HVE ABS Designer.

Overview

Most vehicles built today are fitted with anti-lock braking systems (ABS). Accurate simulation modeling of these vehicles during braking, as well as combined braking and steering maneuvers, thus requires the effects of the ABS to be included. This is especially true for the simulation of complex crash avoidance maneuvers.

This chapter describes the ABS model implemented in the HVE user environment. It is a general purpose model, available for use by any HVE-compatible vehicle simulation model. The model is applicable to the design of ABS systems, as well as the study of loss-of-control crashes of ABS-equipped vehicles.

This chapter covers the following topics:

- **ABS Methodologies** — A basic overview describing how ABS works
- **ABS State Variables** — A list of the parameters that determine the current state of the ABS system
- **Typical Hardware** — A description of the hardware components common to most ABS systems
- **HVE ABS User Interface** — A detailed description of the parameters included in the HVE ABS dialogs and how the parameters are used to model various ABS systems
- **Current Models** — An overview of the ABS models currently implemented in HVE
- **Procedures** — A step-by-step overview of the procedures for simulating an ABS system on a vehicle

ABS Methodologies

The basic concept behind ABS is quite simple and can be demonstrated by the graph of normalized braking force vs longitudinal tire slip shown in Figure 31-1. This graph is traditionally called a *mu-slip curve*. It defines the relationship between longitudinal tire slip and the resulting longitudinal (braking) force. A key observation is that the maximum braking force occurs at the peak braking coefficient, μ_p, in the vicinity of 10 to 15 percent longitudinal tire slip (this varies somewhat from tire to tire). Also, as the tire slip continues to increase to 100 percent, the available braking force falls off. The region of tire slip between μ_p and μ_s (slide friction coefficient, or 100 percent longitudinal tire slip, associated with locked-wheel braking) is a region of dynamic instability. As slip begins to increase beyond μ_p it quickly increases to 100 percent (i.e., the tire locks) with a commensurate loss in available braking force.

The goal of an ABS system is simply to prevent the tire slip from increasing significantly beyond μ_p — regardless of how much brake pedal force is applied by the driver. By keeping the longitudinal slip in the stable region below the peak (see Figure 31-1), the tire continues to roll and, therefore, maintains directional control capability (i.e., the driver can steer the vehicle). In addition, the available braking force is larger than for a locked tire and, therefore, braking distance can be reduced.

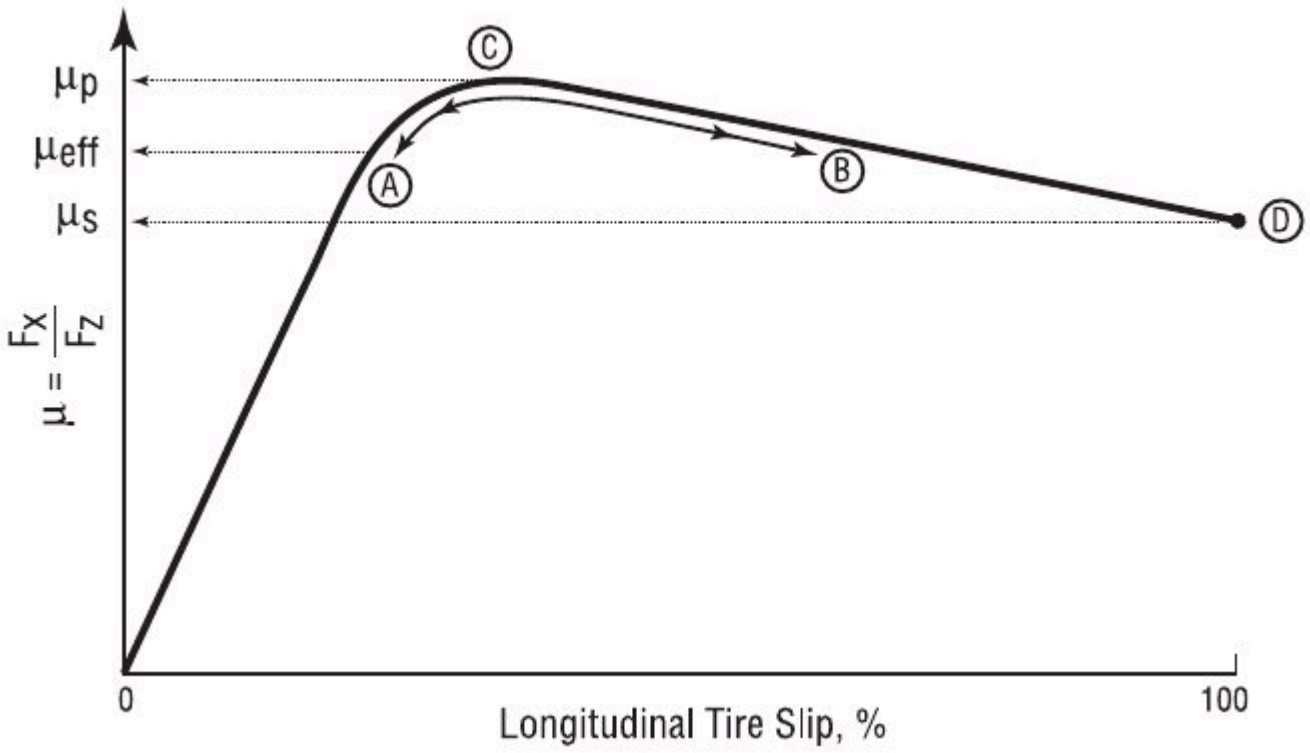


Figure 31-1 — Typical graph of normalized braking force vs longitudinal tire slip, also called a mu-slip curve.

ABS simulation takes advantage of the simulation model's wheel spin degree of freedom, wherein the braking force is calculated from first principles, rather than simply specified as a force at the tire-road interface. To truly simulate ABS, the algorithm must modulate the simulated brake pressure at the wheel, just as it does on an actual vehicle. The procedures for accomplishing this task are described below.

All ABS methodologies work by controlling tire longitudinal slip. This is accomplished through the use of wheel sensors that compare the tire circumferential velocity to the current reference velocity, V_r, normally calculated using the current spin velocities of two or more wheels (see reference 4.41 for a detailed discussion of the calculation of reference velocity). On a vehicle, the longitudinal tire slip cannot be measured directly. Instead, slip is calculated:

$$Slip = \frac{V_r - \dot{\Omega}_w \times R_{tire}}{V_r} \qquad (31\text{-}1)$$

where

Symbol	Description
V_r	Reference velocity
$\dot{\Omega}_w$	Wheel spin velocity
R_{tire}	Current tire rolling radius

State Variables

To accomplish the required control of longitudinal slip, the following state variables are monitored or calculated by the vehicle's ABS control module:

- **Vehicle Velocity** — Linear velocity of the vehicle sprung mass
- **Wheel Spin Velocity** — Angular velocity of each wheel (or axle on some systems)
- **Tire Longitudinal Slip** — Relative velocity between the tire and road, expressed as a fraction of vehicle velocity
- **Wheel Spin Acceleration** — Angular acceleration of each wheel
- **Tire-Road Surface Friction** — Ratio of the maximum braking force to the normal tire force
- **Brake System Pressure** — Pressure produced as a result of the driver's brake pedal application (input variable)
- **Wheel Brake Pressure** — Pressure supplied to the wheel brake assembly (output variable)

Typical Hardware

To monitor or calculate the above state variables, the typical ABS system includes the following hardware components:

- **Electronic Control Unit (ECU)** — This is the vehicle's microcomputer. It is programmed with the algorithm that reads the current state variables, determines the required pressure at each wheel and sends the appropriate signals to the brake pressure modulator (see below).
- **Wheel Speed Sensors** — These components directly measure the wheel spin velocity of each wheel using a wheel-mounted pulse rotor (a notched metal ring) and a fixed, magnetic sensor that measures the rotation of the pulse rotor.
- **Brake Pressure Modulator** — This component (or components, depending on the system) controls the wheel brake pressure according to the control conditions specified by the ECU.
- **Brake Master Cylinder/Air Compressor** — This component provides the fluid pressure source.
- **Wheel Brake Caliper/Cylinder/Chamber** — This component applies the specified braking force at each wheel according to the wheel brake pressure.

The basic hardware requirements are generally the same for all vehicle types, ranging from passenger cars to on-highway trucks. Reference 4.41 provides a detailed description of these required components.

ABS User Interface

The HVE ABS user interface allows the user to select an ABS algorithm and to enter and edit the independent parameters required by the selected ABS algorithm. The interface includes numerous options; thus, various algorithms may be supported. The interface is divided into two sections:

- **System Variables** — Variables that are applicable to the entire vehicle
- **Wheel Variables** — Variables that may be specified independently for each wheel

The interface, dialogs and associated variables are described below.

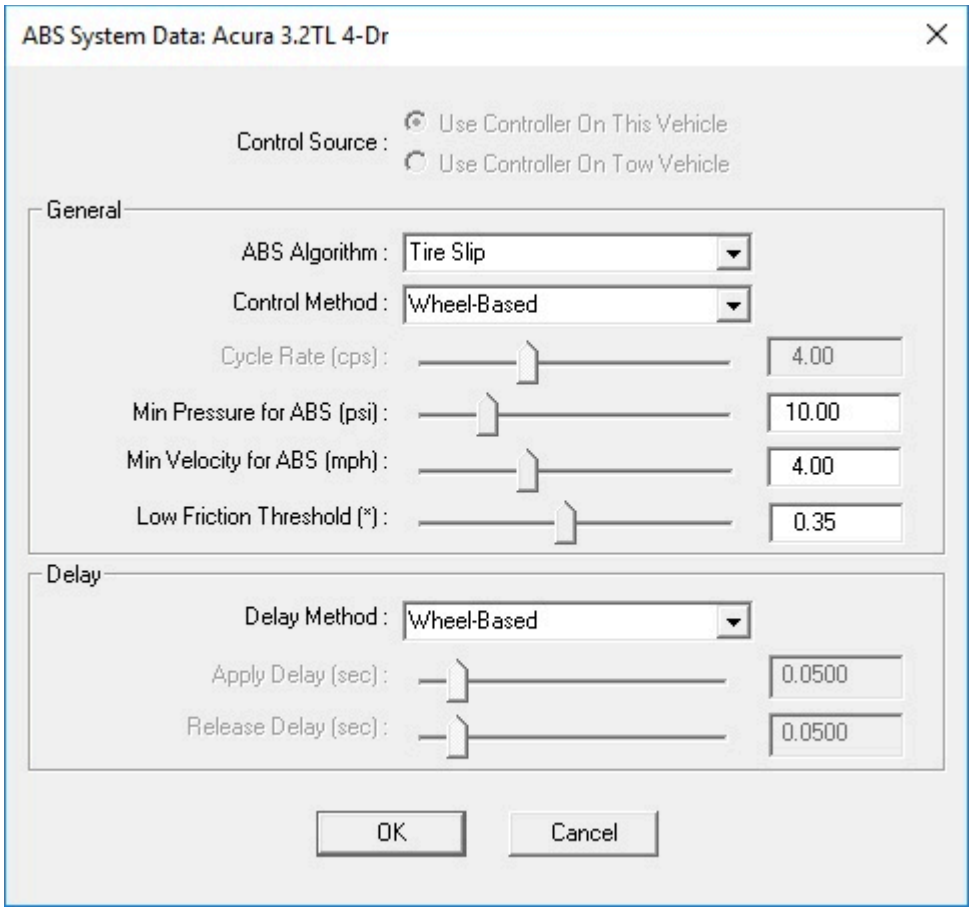


Figure 31-2 — ABS System Data dialog. These parameters affect the entire vehicle (compare with the ABS Wheel Data dialog).

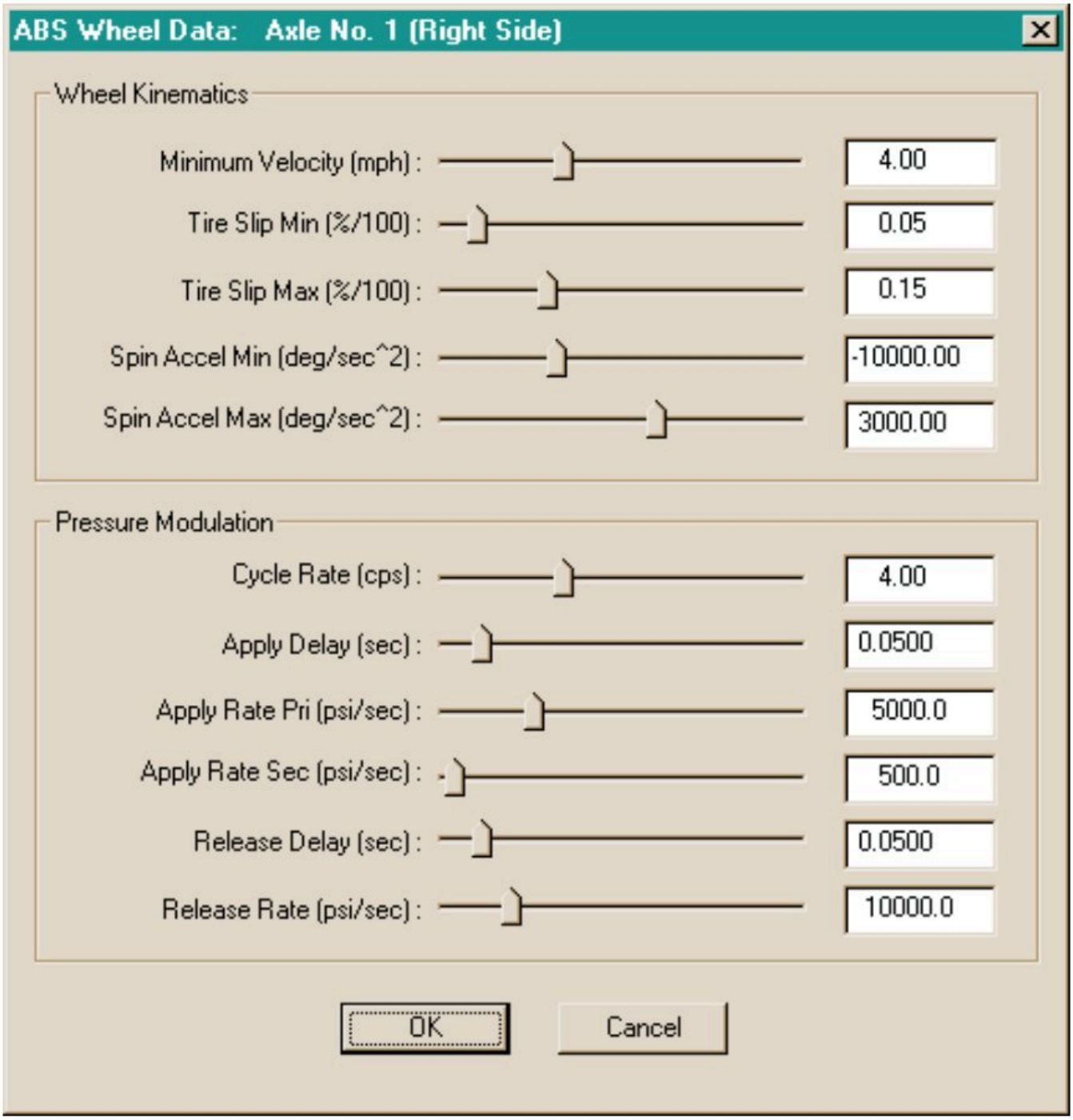


Figure 31-3 — ABS Wheel Data dialog. These parameters affect the selected wheel (compare with the ABS System Data dialog).

Table 31-1 — HVE ABS System variables. These variables apply to the entire vehicle (compare with Wheel Variables in Table 31-2).

Parameter	Units (US / SI)	Description
Control Source	—	Location of ABS controller (This Vehicle or Tow Vehicle)
Algorithm	—	ABS algorithm selected from a list of available algorithms
Control Method	—	ABS control method (Vehicle-Based, Axle-Based or Wheel-Based)
Cycle Rate	cps	Determines the time required for a complete ABS cycle
Threshold Brake System Pressure	psi / kPa	Minimum system pressure for ABS activation
Threshold Vehicle Velocity	mph / km/h	Minimum vehicle velocity for ABS activation
Low Friction Threshold	—	Tire-terrain surface friction threshold
Delay Method	—	ABS delay method (Vehicle-Based, Axle-Based or Wheel-Based)
Apply Delay	sec	Time delay for start of controlled output pressure increase
Release Delay	sec	Time delay for start of controlled output pressure decrease

Table 31-2 — HVE ABS Wheel variables. These variables apply to the selected axle or wheel (compare with System Variables in Table 31-1).

Parameter	Units (US / SI)	Description
Wheel Minimum Velocity	mph / km/h	Minimum wheel forward velocity for ABS control
Tire Slip, Minimum	%/100	Minimum tire longitudinal slip for pressure modulation
Tire Slip, Maximum	%/100	Maximum tire longitudinal slip for pressure modulation
Spin Accel, Minimum	deg/sec²	Minimum wheel angular acceleration for pressure modulation
Spin Accel, Maximum	deg/sec²	Maximum wheel angular acceleration for pressure modulation
Cycle Rate	cps	Determines the time required for one complete ABS cycle
Apply Delay	sec	Time delay for start of controlled wheel output pressure increase
Apply Rate, Primary	psi/sec / kPa/sec	Primary rate of controlled wheel output pressure increase
Apply Rate, Secondary	psi/sec / kPa/sec	Secondary rate of controlled wheel output pressure increase
Release Delay	sec	Time delay for start of controlled output pressure decrease
Release Rate	psi/sec / kPa/sec	Rate of controlled wheel output pressure decrease

System Variables

The ABS System variables included in the HVE ABS model are included in the ABS System Data dialog. The variables and a brief description are shown in Table 31-1. A detailed description of each variable follows.

- **Controller Source** — This selection defines the location of the ABS control unit (ECU; see Typical Hardware, above). For unit vehicles (e.g., passenger cars, vans), it is always This Vehicle. However, the controller for a trailer may be on This Vehicle or the Tow Vehicle. Selecting Tow Vehicle causes the remaining parameters on the ABS System dialog to be disabled; the ABS system parameters are assigned according to those selected for the tow vehicle.

- **ABS Algorithm** — This is the ABS algorithm, selected from a list of the various ABS algorithms available to the user. The algorithms currently available are **Tire Slip** and **Bosch Version 1**. This list is updated as new algorithms become available.
- **Control Method** — This option determines if all wheels are controlled by a single controller or if the individual wheels or axles are controlled separately. Vehicle-based control uses the same control cycle (see below) for all wheels; Axle-based control allows different control cycles for each axle (typically only the rear axle is controlled); Wheel-based control allows different control cycles for each wheel.
- **Cycle Rate** — If the Control Method is Vehicle-based, this parameter determines the maximum time required to perform a complete ABS cycle. It is the same for all wheels.

Specifying the cycle rate is a way an ABS algorithm can enforce a specified periodic cycling of the brake pressure. However, most ABS algorithms do not use the Cycle Rate parameter. Instead, the cycle rate is the natural result of the Apply and Release Delays and the Apply and Release Rates (see below).

> **NOTE:** Cycle Rate is disabled if the Control Method is Axle-based or Wheel-based. Instead, the Cycle Rate is set separately for each wheel (see ABS Wheel Data dialog).

- **Minimum ABS Pressure (Threshold Brake System Pressure)** — This parameter provides a minimum system pressure threshold for ABS actuation. ABS is bypassed when system pressure falls below this value.
- **Minimum Vehicle Velocity (Threshold Vehicle Velocity)** — This parameter provides a minimum vehicle velocity threshold for ABS actuation. ABS is bypassed when vehicle velocity falls below this value.
- **Low Surface Friction Threshold** — This parameter sets a threshold defining low friction surfaces. Algorithms can use this parameter to invoke friction-dependent modulation behaviors. For example, the Apply Delay or Release Delay (see below) may be reduced for low friction surfaces.
- **Delay Method** — This parameter determines if control pressure delays are Vehicle-based, Axle-based or Wheel-based. Vehicle-based delay uses the same delay period for each wheel; Axle-based delay allows different delay periods for each axle; Wheel-based delay allows different delay periods for each wheel.
- **Apply Delay** — If the Delay Method is Vehicle-based, this parameter determines the delay period for all wheels before output pressure is increased. (See Wheel Variables for Axle-based and Wheel-based delay.) This value defines the period of constant pressure before pressure begins to increase (see Figures 31-5 through 31-7).
- **Release Delay** — If the Delay Method is Vehicle-based, this parameter determines the delay period for all wheels before output pressure is reduced. (See Wheel Variables for Axle-based and Wheel-based delay.) This value defines the period of constant pressure before pressure begins to decrease (see Figures 31-5 through 31-7).

> **NOTE:** Apply Delay and Release Delay are disabled if the Delay Method is Axle-based or Wheel-based. Instead, the Apply and Release Delays are set separately for each wheel (see ABS Wheel Data).

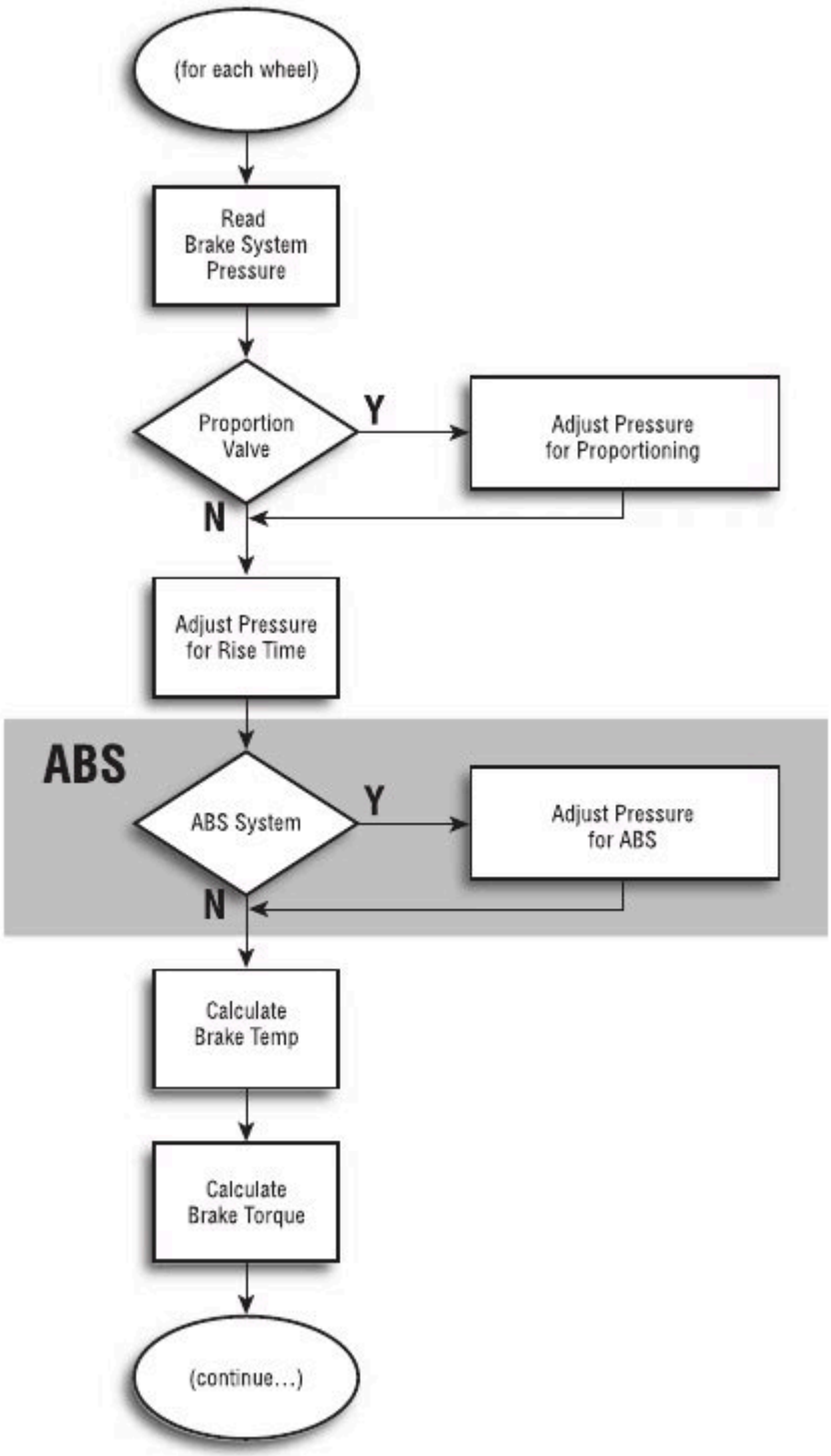


Figure 31-4 — Flow chart showing integration of the HVE ABS model into a vehicle simulation.

Wheel Variables

The ABS Wheel variables are shown in Table 31-2. A brief description of each variable follows.

- **Minimum Wheel Velocity** — This parameter specifies a minimum wheel velocity threshold for ABS actuation. ABS is bypassed when the wheel velocity falls below this value.

- **Minimum Tire Slip** — This parameter may be used in an algorithm to establish a lower threshold for tire longitudinal slip.
- **Maximum Tire Slip** — This parameter may be used in an algorithm to establish an upper threshold for tire longitudinal slip.
- **Minimum Wheel Spin Acceleration** — This parameter may be used in an algorithm to establish a lower threshold for wheel spin acceleration.
- **Maximum Wheel Spin Acceleration** — This parameter may be used in an algorithm to establish an upper threshold for wheel spin acceleration.
- **Cycle Rate** — If the ABS System Control Method is Axle-based or Wheel-based, this parameter determines the maximum time required to perform a complete ABS cycle for the selected axle or wheel.

Specifying the cycle rate is a way an ABS algorithm can enforce a specified periodic cycling of the brake pressure. However, most ABS algorithms do not use the Cycle Rate parameter. Instead, the cycle rate is the natural result of the Apply and Release Delays and the Apply and Release Rates (see below).

> **NOTE:** Cycle Rate is disabled if the Control Method is Vehicle-based. Instead, Cycle Rate is set in the ABS System Data dialog.
>
> **NOTE:** If the Control Method is Axle-based, the specified cycle rate is applied to both wheels on the selected axle.

- **Apply Delay** — If the Delay Method is Axle-based or Wheel-based, this parameter determines the control pressure delay period before output pressure is increased at the selected wheel. This value defines the period of constant pressure before pressure begins to increase (see Figures 31-5 through 31-7).
- **Primary Application Rate** — This parameter determines the initial rate of pressure increase. This value defines the slope as pressure increases (see Figures 31-5 through 31-7).
- **Secondary Application Rate** — This parameter provides an alternative rate of pressure increase, usually substantially lower than the Primary Application Rate. This value defines the slope as pressure increases (see Figures 31-5 through 31-7).
- **Release Delay** — If the Delay Method is Axle-based or Wheel-based, this parameter determines the control pressure delay period before application pressure is reduced at the selected wheel. This value defines the period of constant pressure before pressure begins to decrease (see Figures 31-5 through 31-7).
- **Release Rate** — This parameter determines the rate of pressure decrease (see Figures 31-5 through 31-7).

Selection of the values for the above parameters effectively determines the cycle rate for an ABS system. Using this approach (instead of assigning a specific Cycle Rate; see earlier description of the Cycle Rate variable) has the advantage of varying the cycle time according to the current operating conditions.

> **NOTE:** Apply Delay and Release Delay are disabled if the Delay Method is Vehicle-based. Instead, the Apply and Release Delays are set in the ABS System Data dialog.
>
> **NOTE:** If the Delay Method is Axle-based, the specified Apply Delay and Release Delay are automatically assigned to both wheels on the selected axle.

The ABS System and Wheel Data variables are provided as a palette of parameters available to the designer of an ABS system algorithm. The selection of individual variables and their effect on the simulated characteristics of any specific ABS system are algorithm-dependent.

Implementation

To integrate the ABS model into a vehicle dynamic simulation requires that the simulation have the following features:

- A brake system model, including a pressure source and optional proportioning, lag times and rise times
- A wheel brake model that computes brake torque ratio as a function of current wheel brake pressure
- A spin degree of freedom for each wheel

A flowchart showing the integration of the ABS model into an existing vehicle simulation is shown in Figure 31-4. See also reference 4.42.

Current Algorithms

Two ABS algorithms are currently implemented. These are Tire Slip and Bosch Version 1. These algorithms are described below.

Tire Slip Algorithm

This is a simple and straight-forward ABS algorithm. It is designed based on the fundamental goal of an ABS system, that is, to maintain tire slip in the vicinity of the peak friction coefficient (refer to Figure 31-1).

On brake pressure application, once ABS is invoked (that is, the minimum vehicle velocity and brake pressure thresholds are exceeded), the algorithm incorporates switching logic depending on the current tire slip:

- **Tire Slip ≤ Minimum Tire Slip** — Under this condition, the status of the ABS during the previous sample determines how pressure is modulated for the current cycle. If the ABS modulation status was off, the output pressure is set equal to the input pressure and the ABS system parameters (delays, etc.) are reset. Otherwise, brake pressure will be controlled. One of two possibilities exists: (a) Input pressure is decreasing — in this case, output pressure is set equal to input pressure and the ABS status is turned off; or (b) Input pressure is constant or increasing — in this case, the output pressure is maintained constant for the specified Apply Delay, after which output pressure is increased according to the Primary Apply Rate.
- **Minimum Tire Slip < Tire Slip < Maximum Tire Slip** — Under this condition, pressure will be controlled. One of two possibilities exists: (a) Input pressure is decreasing — in this case, output pressure is set equal to input pressure and the ABS status is turned off; or (b) Input pressure is constant or increasing — in this case, the output pressure is maintained constant for the specified Apply Delay, after which output pressure is increased according to the Secondary Apply Rate.
- **Tire Slip ≥ Maximum Tire Slip** — Under this condition, pressure will be controlled. The output pressure is maintained constant for the specified Release Delay, after which the output pressure is reduced according to the Release Rate.

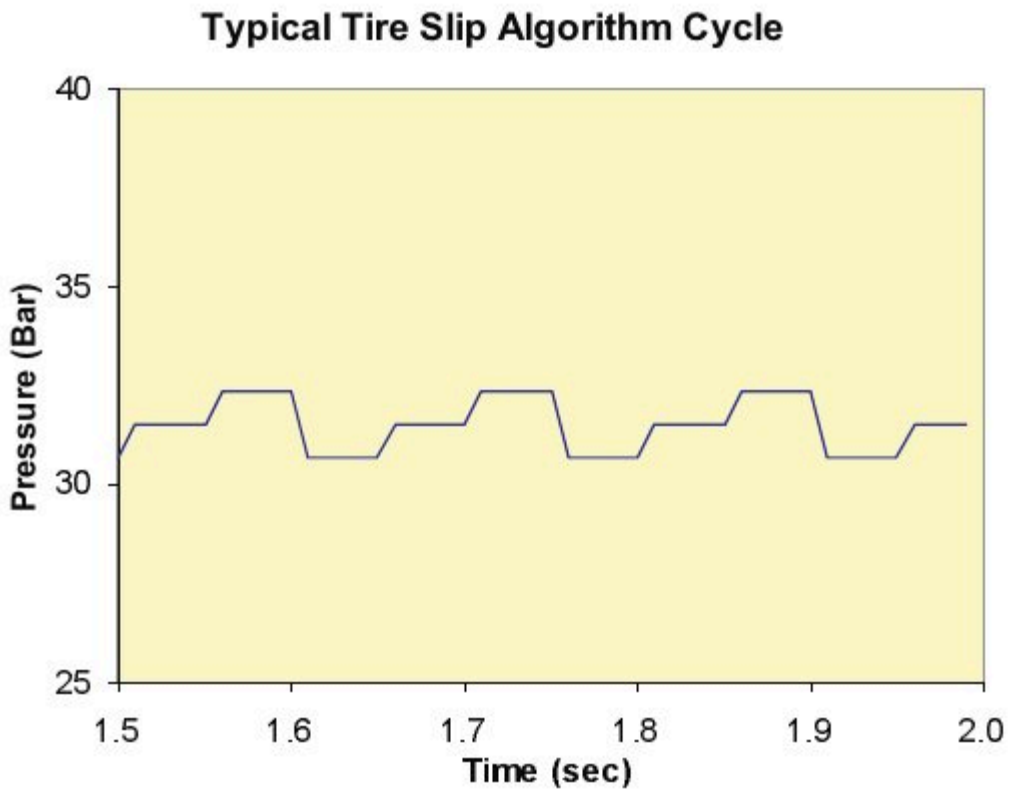


Figure 31-5 — Typical control cycles for the Tire Slip algorithm (wheel pressure vs time during a hard brake application).

Figure 31-5 shows a typical pressure vs time history for a few cycles of a hard brake pedal application (i.e., enough system pressure to lock the wheel). The parameters used by the Tire Slip algorithm are shown in Table 31-3. Additional details for the Tire Slip algorithm, including a flow chart, are provided in reference 4.42.

Table 31-3 — Variables used by the Tire Slip ABS algorithm

Parameter
Control Method
Threshold Brake System Pressure
Threshold Vehicle Velocity
Tire Longitudinal Slip, Minimum
Tire Longitudinal Slip, Maximum
Delay Method
Apply Delay
Apply Rate, Primary
Apply Rate, Secondary
Release Delay
Release Rate

HVE Bosch Version 1 Algorithm

The HVE Bosch Version 1 ABS algorithm is based on the algorithm described in reference 4.41. The Bosch ABS system is used on many passenger cars. The algorithm is based on wheel spin acceleration and a critical tire slip threshold.

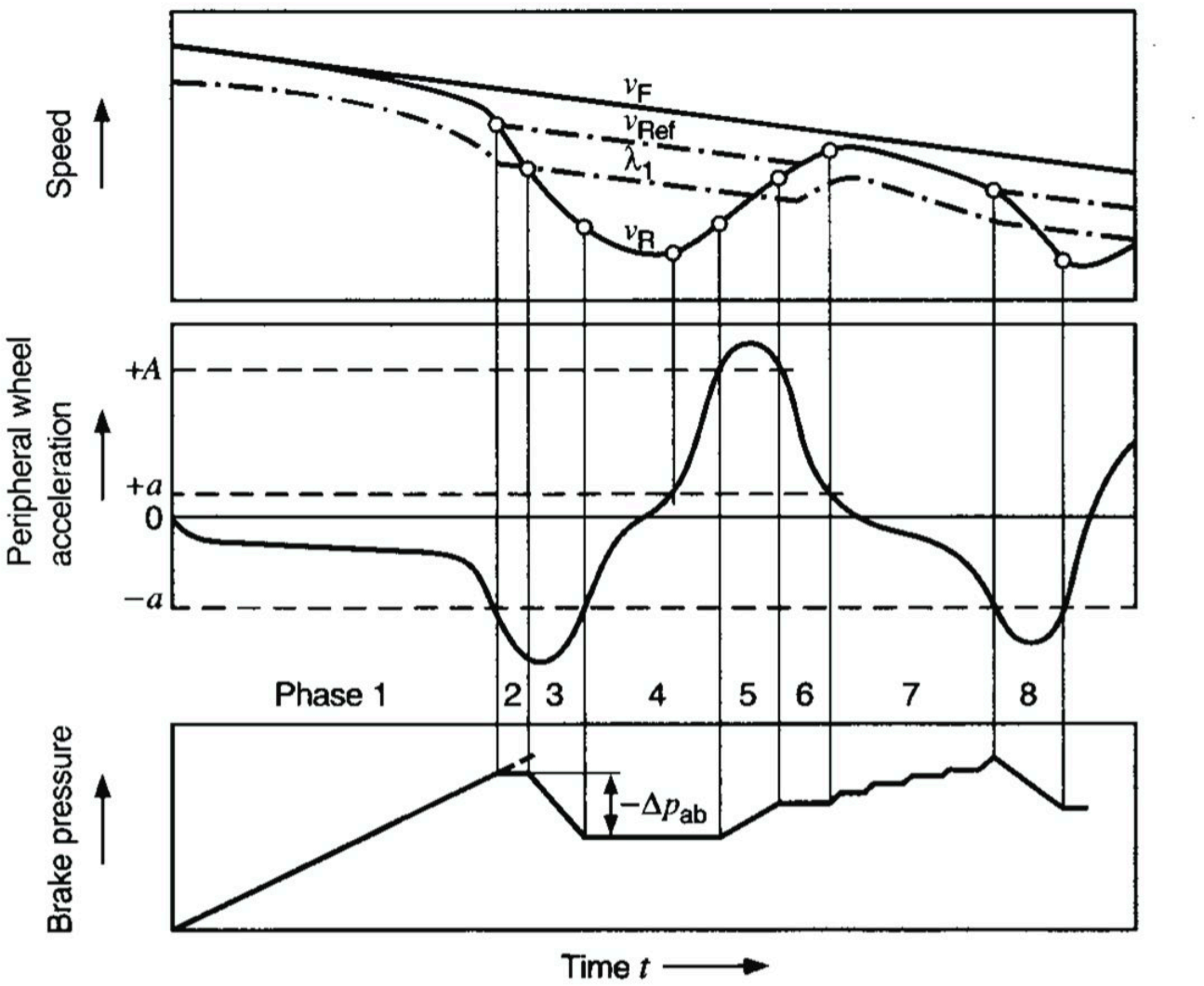


Figure 31-6 — Idealized control sequence for the Bosch algorithm (reprinted from reference 4.41). Compare the lower portion with the pressure cycle displayed in Figure 31-7.

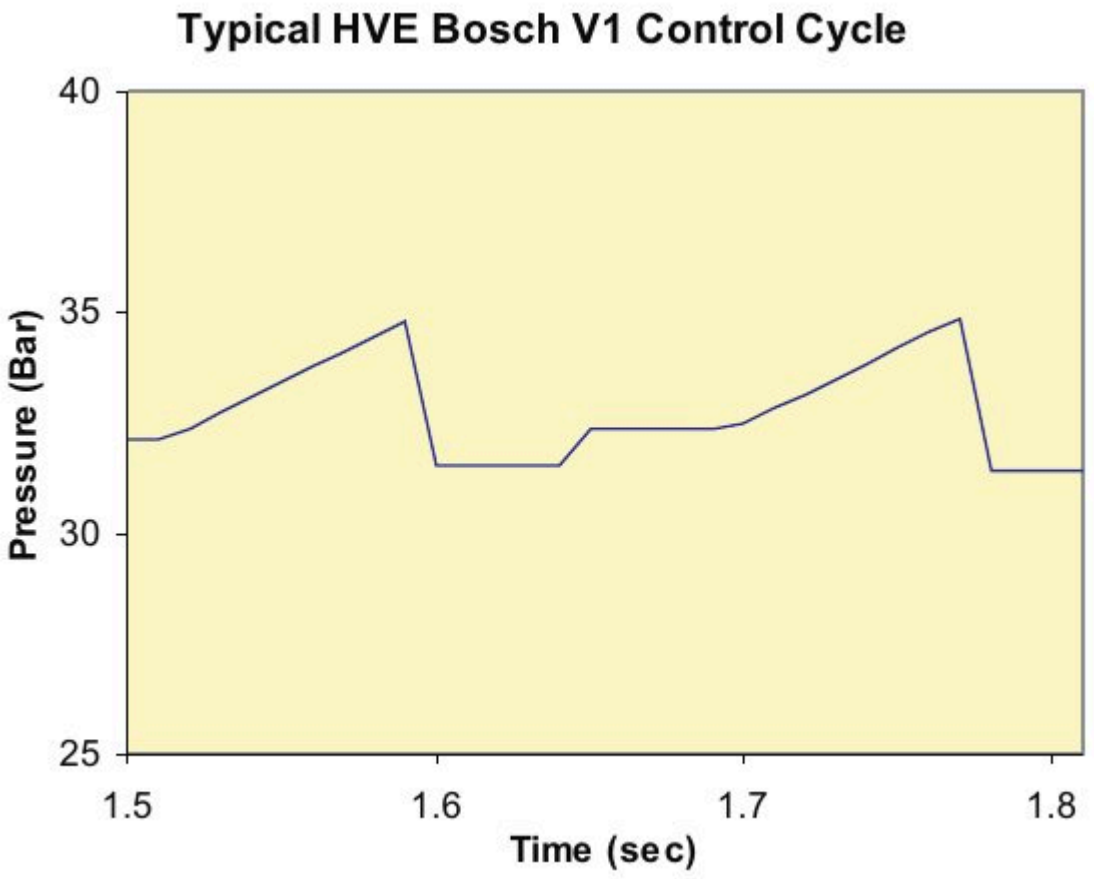


Figure 31-7 — Simulated control cycle for the HVE Bosch Version 1 algorithm. Compare with the lower portion of Figure 31-6.

Upon brake pressure application, once ABS is invoked (i.e., the thresholds are exceeded), the current brake pressure application is divided into eight phases (see Figure 31-6):

- **Phase 1 — Initial application.** Output pressure is set equal to input pressure. This phase continues until the wheel angular acceleration (negative) drops below the Wheel Minimum Wheel Spin Acceleration, $-a$.
- **Phase 2 — Maintain pressure.** Output pressure is set equal to the previous pressure. This phase continues until the tire longitudinal slip exceeds the slip associated with the Slip Threshold. At this time, the current tire slip is stored and used as a criterion in later phases. This slip corresponds to the maximum slip; the tire is beginning to lock.
- **Phase 3 — Reduce pressure.** Output pressure is decreased according to the Release Rate until the wheel spin acceleration again becomes positive (this is a slight modification to the sequence shown in Figure 31-6, in which the pressure is decreased until the wheel spin acceleration exceeds $-a$).
- **Phase 4 — Maintain pressure.** Output pressure is set equal to the previous pressure for the specified Apply Delay, or until the wheel spin acceleration (positive) exceeds $+A$, a multiple (normally 10×) of the Wheel Maximum Spin Acceleration (signifying the wheel spin velocity is increasing at an excessive rate).
- **Phase 5 — Increase pressure.** Output pressure increases according to the Primary Apply Rate. This phase continues until the wheel spin acceleration drops and again becomes negative (this is a slight modification to the sequence shown in Figure 31-6, in which the pressure is increased until the wheel spin acceleration drops below $+A$).
- **Phase 6 — Maintain pressure.** Output pressure is set equal to the previous pressure for the specified Apply Delay, or until wheel angular acceleration again exceeds the Wheel Minimum Spin Acceleration (negative).

- **Phase 7 — Apply pressure.** Output pressure increases according to the Secondary Apply Rate, normally a fraction (1/10) of the Primary Apply Rate. This achieves greater braking performance while minimizing the potential for wheel lock-up at tire longitudinal slip in the vicinity of peak friction. This phase continues until wheel angular acceleration drops below the Wheel Minimum Angular Acceleration (negative), indicating wheel lock-up is imminent.
- **Phase 8 — Release pressure.** At this point an individual cycle is complete, the process returns to Phase 3 and a new control cycle begins.

As stated, some minor differences exist between the HVE implementation and the Bosch description provided in reference 4.41. These differences reflect some inconsistencies between the acceleration and pressure profiles shown in Figure 31-6. For example, unless the throttle is applied, it is physically inconsistent that the wheel spin acceleration would be positive, let alone increase (as shown in Phase 5), in the presence of increased brake pressure (and, therefore, brake torque).

Each of the above phases begins with a comparison between the current tire longitudinal slip and the value stored during Phase 2. If the current slip exceeds this value, the normal logic is bypassed and resumed at Phase 3. This effectively allows the algorithm to "learn" the wheel slip associated with wheel lock-up on the current surface. This is referred to as *adaptive learning*, and is a key to the success of this ABS algorithm. As the tire travels onto surfaces with differing friction characteristics, the ABS model is able to maximize its performance accordingly.

Default parameters used by the HVE Bosch Version 1 algorithm were developed through the evaluation of numerous simulation runs. See reference 4.42 for typical parameters applicable to a P195/75R14 passenger car tire.

Figure 31-7 shows a typical pressure vs time history for a single cycle of a hard brake pedal application (i.e., enough system pressure to lock the wheel). The parameters used by the HVE Bosch Version 1 algorithm are shown in Table 31-4. Additional details for the HVE Bosch Version 1 algorithm, including a flow chart, are provided in reference 4.42.

Table 31-4 — Variables used by the HVE Bosch Version 1 ABS algorithm

Parameter
Control Method
Threshold Brake System Pressure
Threshold Vehicle Velocity
Tire Longitudinal Slip, Minimum
Wheel Spin Acceleration, Minimum
Wheel Spin Acceleration, Maximum
Delay Method
Apply Delay
Apply Rate, Primary
Apply Rate, Secondary
Release Delay
Release Rate

Other ABS Algorithms

The ABS model implemented in HVE is not restrictive in terms of the algorithms it can support, other than its need to provide the parameters required by the algorithm. Endless tweaking of an algorithm is possible, resulting in different ABS system characteristics, each with its advantages and disadvantages. Thus, it is certain that new ABS algorithms will be developed and implemented in HVE over time, both to develop and to model new ABS systems.

Procedures

The steps for using the HVE ABS model are:

1. Display the Brake System Information dialog.
2. Click the ABS Installed checkbox.
3. Optionally, edit the vehicle's ABS System parameters.
4. Optionally, edit the vehicle's ABS Wheel parameters for each wheel.

These steps are described below.

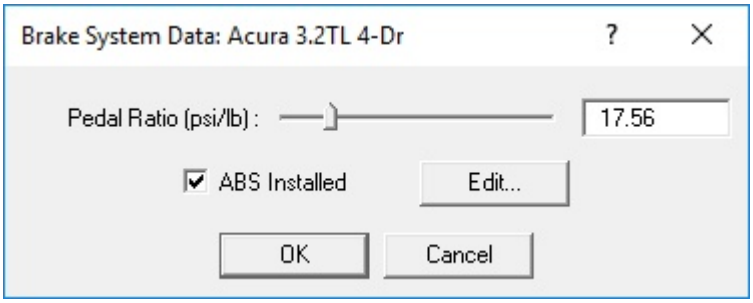


Figure 31-8 — Vehicle Brake System Information dialog. Click on ABS Installed to invoke the ABS model. Click on Edit to display the ABS System Data dialog for the selected vehicle.

Invoking the ABS Model

Vehicles in the EDC Generic Database (Generic.db) and the EDC Custom Vehicle Database (EDC.db) are currently created with the ABS model turned off.

To model an ABS system, perform the following steps:

1. Choose Vehicle Mode. The Vehicle Editor displays the current vehicle.
2. Choose the desired vehicle from the list of active vehicles. The vehicle is displayed in the viewer.
3. Click on the vehicle's brake system icon. The Brake System Information dialog is displayed (see Figure 31-8).
4. Click the ABS Installed checkbox. The Edit button becomes enabled.

The vehicle's ABS parameters may now be edited.

Editing ABS System Parameters

After invoking the ABS model, as described in the previous section, the ABS System parameters may be edited. To edit the ABS System parameters, use the Brake System Information dialog (see Figure 31-8), and perform the following steps:

1. Press Edit. The ABS System Data dialog is displayed (see Figure 31-2).
2. Edit the various ABS system parameters as desired.

Editing ABS Wheel Parameters

After invoking the ABS model, as described earlier, the ABS wheel parameters may be edited. To edit the ABS wheel parameters, perform the following steps:

1. Click on the desired wheel and choose Brakes from the Wheel Data pop-up menu. The Brake Assembly dialog is displayed.
2. Press the ABS Designer button to display the ABS Wheel Data dialog for the selected wheel.
3. Edit the ABS parameters as desired.
4. Press OK. The Brake Assembly dialog is still displayed.
5. Press Copy to Other Side if you wish the left-side and right-side wheels to have the same properties.

> **NOTE:** If the Control Method and/or Delay Method are Axle-based, those parameters are automatically copied to the other side.

1. Press OK to assign the ABS Wheel parameters for the selected wheel(s).

Any event that supports the HVE ABS model (e.g., SIMON) will now include the effects of the ABS system when the event is executed.

Tips and Suggestions

The following tips and suggestions are helpful when using the HVE ABS Model:

- Although the ABS Model will be executed each trajectory integration timestep, the simulation will output results according to the currently specified output time interval (default value 0.1000 sec). Therefore, if you wish to evaluate detailed results from an ABS event, you should reduce the output interval (see Simulation Controls dialog) to 0.0100 seconds (or less). Otherwise, the (rather large) output interval will act as a filter, and the details of the pressure modulation will be lost.
- To monitor results from an ABS simulation, use the Variable Selection dialog and choose the desired output parameters. The most interesting results for an ABS simulation include the following parameters:
- Tire Group — Fx', Fy', Fz'; Long Slip; Skidmark
- Wheel Group — Spin Velocity; Spin Accel; Brake Pressure
- Driver Group — Brake (System) Pressure

Using Variable Output to graph these results is most helpful. - Two conditions are required to cause a tire to leave a skidmark. First, the tire must be skidding (as determined by the value of the Skidmark output variable), and second, the radial load on the tire (Fz') must be sufficient to leave a mark (see Minimum Fz for Skidmark; this value is assigned in the Vehicle Editor, Tire Information, Physical Data dialog). Otherwise, even a locked tire may not leave a black rubber tire mark on the road. - In early versions of HVE, the skidmark was either ON or OFF. HVE now has the capability to vary the skidmark intensity according to the current level of tire longitudinal and lateral slip and vertical tire load. This capability is particularly useful for ABS simulation, wherein the skidmarks are faint, rather than pure black. HVE simulates this effect. - To illustrate the presence of faint skidmarks on a textured road surface, remove the texture (right-click on the road surface and choose No Texture). You can actually see the faint skidmarks on the untextured surface.

Back Matter: Tutorial, Appendices and Index

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), back matter: the HVE Tutorial, Appendices I–X, revised to reflect the current HVE software where applicable.

Contents

Item	Original location
HVE Tutorial (Lessons 1–4: Navigating, Running a Simulation, Combining Multiple Events, Creating a Video)	Tutorial section, pages 32-1 to 32-60
Appendix I — Installation & Setup (system requirements, installation, licensing, HVE file system)	I-1 to I-10
Appendix II — Messages (input validation, event verify, execution and output report messages)	II-1 to II-14
Appendix III — Coordinate Systems (earth-, vehicle-, tire- and human-fixed)	III-1 to III-8
Appendix IV — Units (units files, unit names and program units by group)	IV-1 to IV-14
Appendix V — Collision Deformation Classification (CDC)	V-1 to V-8
Appendix VI — References	VI-1 to VI-10
Appendices VII–IX — Developer's Toolkit Overview, Databases, Quick Reference (reserved in the original)	VII-1 to IX-2
Appendix X — Index (replaced by a topic-to-chapter locator table)	X-1 to X-6

Notes on this edition

- The Tutorial's video lesson (Lesson 4) has been updated to the current file-based AVI recording workflow; tape-deck steps are flagged as legacy. See also Section Nine — Video Output.
- Appendix I's installation/licensing procedures describe the 2006-era CD-ROM and parallel-port EDKEY distribution and are retained as legacy reference; the HVE file system tables have been checked against the file locations used by the current version of HVE.
- Appendix IV's Environment Units table contained several descriptions mis-copied from the Engine table in the original printing; these have been corrected.
- The original page-number index (Appendix X) is replaced by a topic-to-chapter table, since page numbers do not apply to the Markdown edition.

Chapter 32 — HVE Tutorial

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Tutorial section, pages 32-1 through 32-60, revised to reflect the current HVE software.

This section of the HVE User's Manual contains a tutorial illustrating HVE's features. For any new HVE user, this section is very important. The tutorial describes the use of HVE's basic components, but even more important, the tutorial provides a basic framework for using HVE. Users will soon realize they perform the same basic operations each time they use HVE, so understanding these basic operations is fundamental to its use.

The purpose of the HVE Tutorial is to provide a solid grasp of the fundamentals of using HVE, and to accelerate the user's learning curve.

Overview

This chapter presents a series of basic tutorials that demonstrate various HVE techniques and capabilities. This is accomplished through a series of high-level, task-oriented procedures using a step-by-step approach. While this tutorial explores many of HVE's important features, it does not provide detailed explanations. For these details, the tutorial refers to specific chapters, and the user should refer to those chapters when necessary.

> **NOTE:** In addition to this tutorial, every EDC physics model has a Program Manual that includes a comprehensive tutorial specific to that physics model. Manuals from 3rd-party vendors typically include a tutorial as well.

The Tutorial is divided into four individual lessons:

- **Lesson 1 — Navigating in HVE** — Techniques that promote efficiency; how to move quickly within HVE to perform fundamental tasks.
- **Lesson 2 — Running a Simulation Model** — The best way to set up and execute HVE-compatible physics models (reconstructions and simulations involving human and vehicle dynamics).
- **Lesson 3 — Combining Multiple Events** — How to merge two or more individual simulations into a coherent and seamless sequence involving multiple humans and vehicles.
- **Lesson 4 — Creating a Video** — A quick and easy lesson describing how to create your first video.

Lesson 1 — Navigating in HVE

Lesson 1 is a lesson in fundamentals, with an emphasis on efficiency. In this lesson you will learn to perform the following fundamental tasks: Starting HVE; Learning About the Menu Bar; Managing Dialogs and Viewers; Choosing an Editor; Using Dialogs; Using Viewers; Shutting Down HVE.

Do not skip this lesson! By spending time performing these tasks, you will be able to navigate within HVE more quickly, more efficiently and (perhaps most important) more intuitively.

Starting HVE

To start HVE, use the Start menu (or desktop shortcut) to select the HVE program icon, just as you would start any other program on your computer. After starting HVE, the HVE Menu Bar and current HVE Editor dialog are displayed.

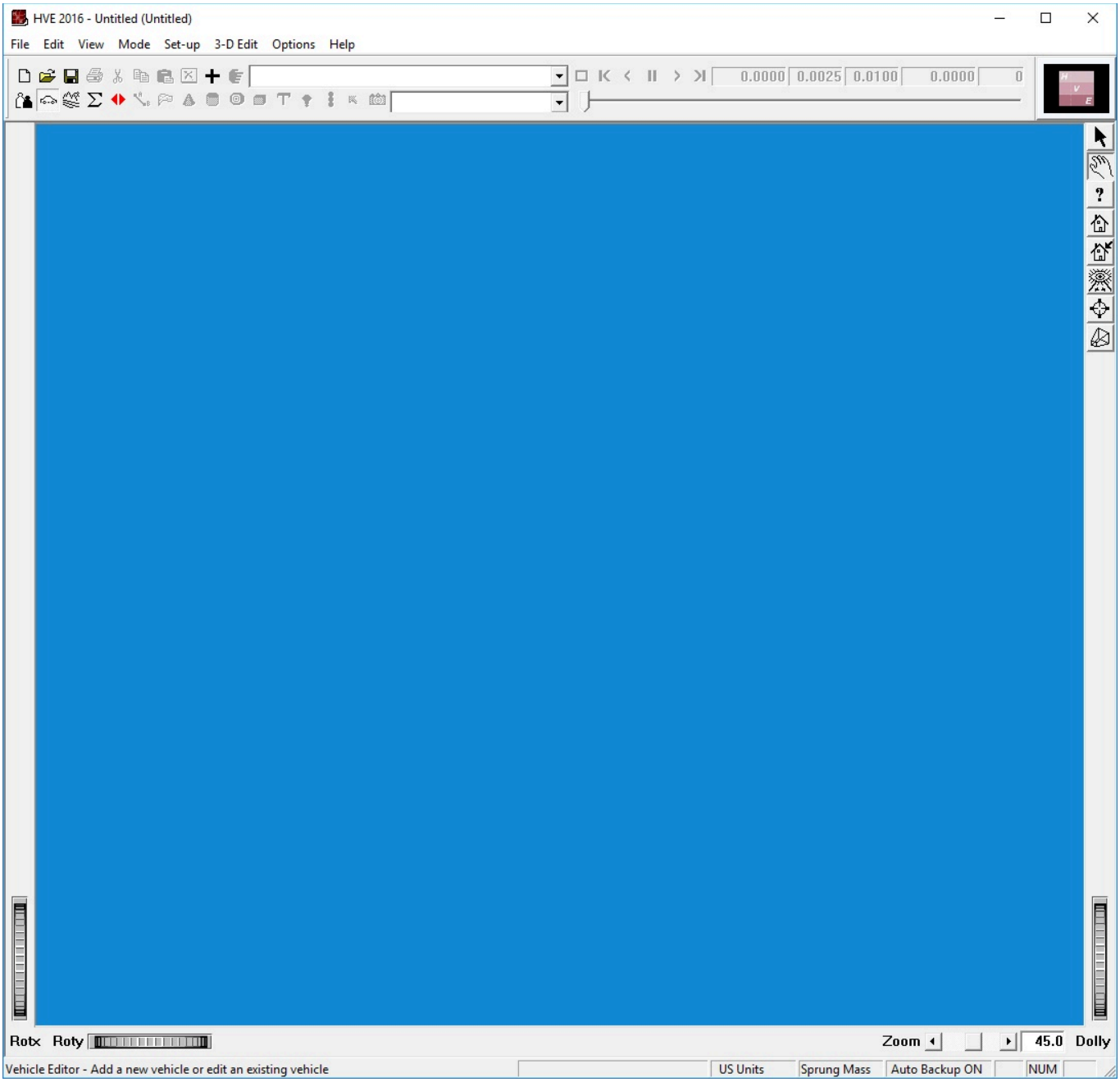


Figure 32-1 — The Desktop after starting HVE: main program window with Menu Bar, Tool Bar, integrated Event and Playback Controller, Viewer Manipulators and Zoom Slider.

> **NOTE:** HVE saves the current editor when you exit; thus, the editor now displayed is the one you were using when you last exited HVE.

Learning About the Menu Bar

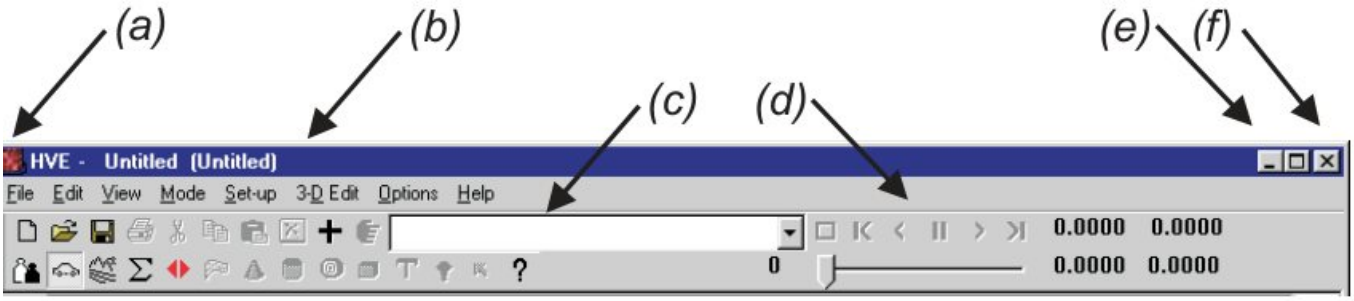


Figure 32-2 — Menu Bar. HVE's menu bar is very traditional, with File, Edit, View, Mode, Set-up, 3D-Edit, Options and Help menus. It also includes the Dialog Control button, the Title Bar, the Active Objects List, the Event Controller, the Minimize/Maximize buttons, the Close button, and several toolbar buttons for quick access to functions.

The Menu Bar contains the program's fundamental options: File, Edit, View, Mode, Set-up, 3D-Edit, Options and Help.

- **File** — Basic system operations, such as starting new cases, opening and saving files, working with video, printing and exiting HVE.
- **Edit** — Operations that allow you to edit selected objects. (No Edit menu options are available until an object is selected. Other menus also have disabled options until the appropriate situation is present.)
- **View** — Operations that determine how you look at simulations.
- **Mode** — Selects which Editor you are currently using, and adds new or previous objects into the Editor (e.g., Select Mode, Add New, Add Previous).
- **Set-up** — Inputs for setting up your simulation or reconstruction events.
- **3D-Edit** — Launches and closes the 3-D Editor while in Environment mode, plus edits material attributes, colors and textures or changes manipulators while in the 3-D Editor.
- **Options** — Several user-definable options and preferences.
- **Help** — Access to the various HVE Help System options.

The **Dialog Control button** is located in the upper left corner of the Menu Bar. Its options allow you to move the dialog, change its size, minimize it, and maximize it. All dialogs and viewers have a Dialog Control button.

The **Title Bar** displays the current case title, followed by the current filename in parentheses. Clicking on the Title Bar and dragging the mouse moves the dialog — a fundamental navigation task that allows you to move dialogs and viewers anywhere you wish.

The **Active Objects List** is integrated into the Menu Bar. It displays a drop-down list of active objects while in the Human, Vehicle, Environment and Event modes. The current object is displayed in the viewer. To add a new object to the list, click on the Add New Object button (a + sign) on the toolbar. To display the Object Information dialog for the current object, click on the Object Info button (a pointing hand) immediately to the left of the list.

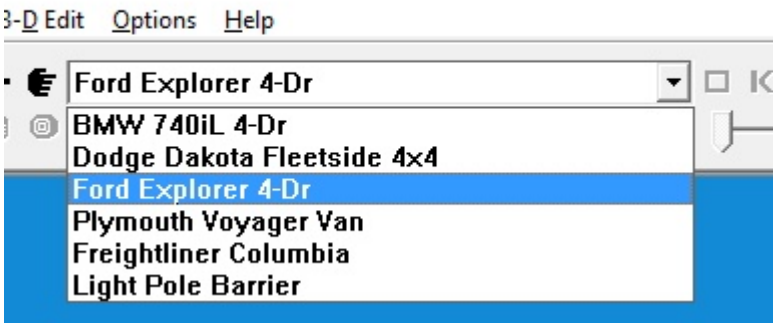


Figure 32-3 — Active Objects List drop-down shown in Vehicle Mode.

The **Event (and Playback) Controller** is integrated directly into the Menu Bar. Its information displays are:

- **Current Simulation Frame** — The current frame of simulation output (a frame is equivalent to one timestep of output).
- **Max Simulation Time** — The selected Simulation Controls Maximum Simulation Time termination condition.
- **Simulation Timestep** — The integration timestep used for typical vehicle trajectory calculations, as set in Simulation Controls.
- **Current Simulation Time** — The current timestep of the complete event.
- **Output Interval** — The timestep interval used for displayed results, as set in Simulation Controls.

The Event Controller Slider can be used to move forward or backward through each simulation frame — an easy means to locate a specific point in a simulation without having to Play and Stop.

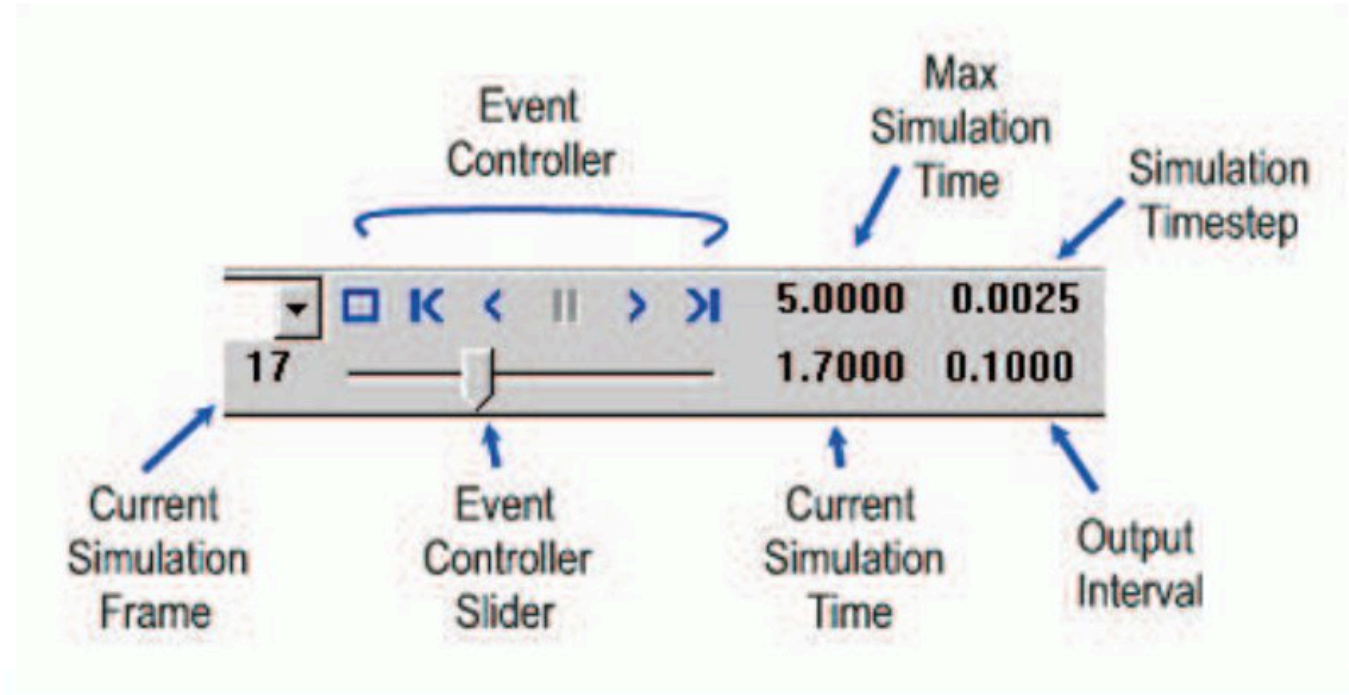


Figure 32-4 — Event Controller.

The Menu Bar also includes **Minimize** and **Maximize** buttons and a resizable dialog frame. Clicking the frame and dragging resizes the dialog (some dialogs cannot be resized); clicking on the frame also raises the dialog to the top of the display.

> **NOTE:** Maximizing the program window allows you to use the whole screen for your work. A high display resolution provides an even larger work area.

Viewers

Viewers are used to display 3-D objects.

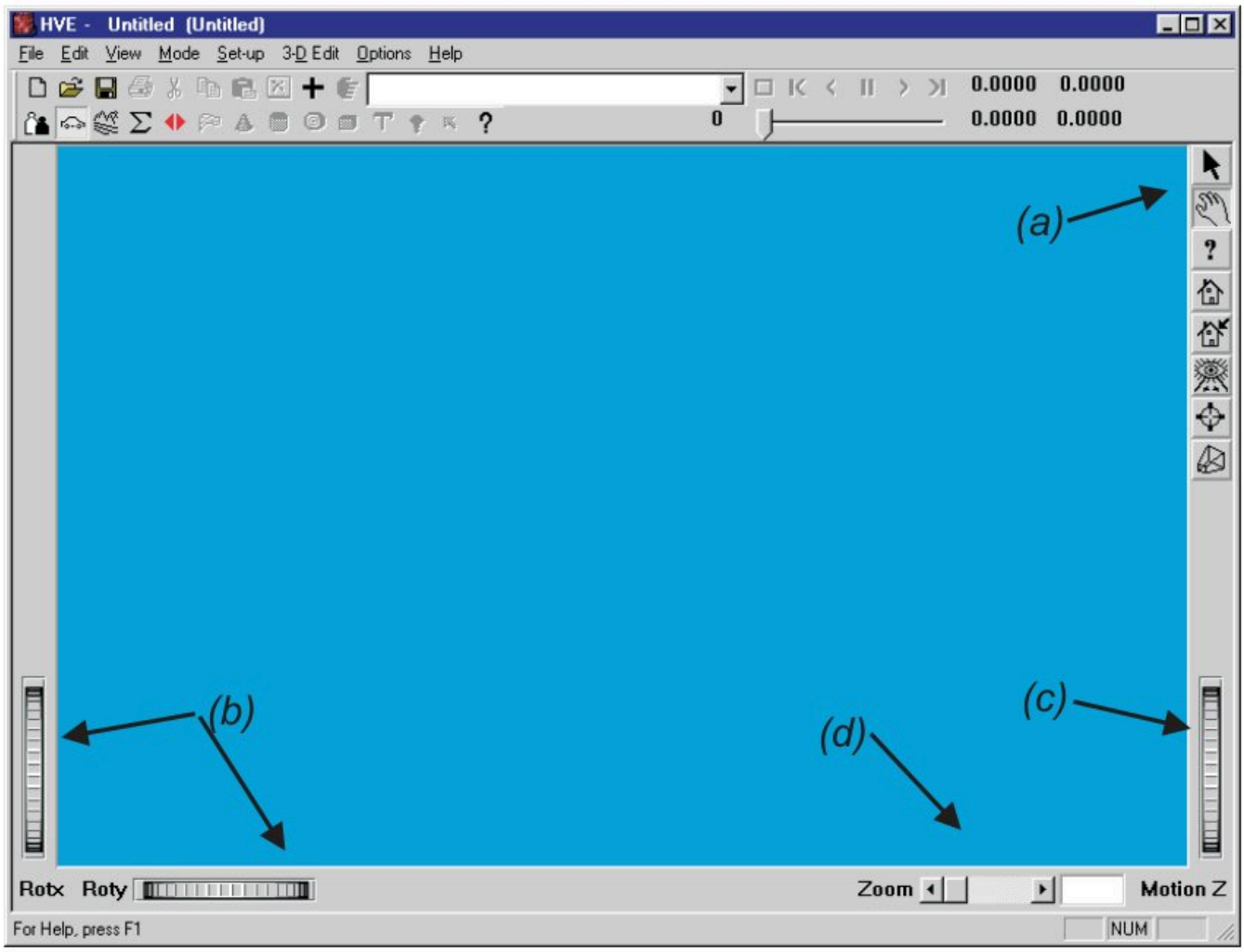


Figure 32-5 — Typical Viewer. Pick Mode or Manipulate Mode is selected by choosing the arrow or hand icon near the upper right edge of the viewer. RotX and RotY thumbwheels rotate the object about the viewer's X and Y axes. The Dolly thumbwheel moves the camera toward or away from the object. The Zoom slider changes the viewer's included angle.

Viewers have the same window manager components as the Menu Bar, plus:

- **Viewer Mode Selector** — Switches between Pick mode (used for choosing objects in the viewer and performing editing operations) and Manipulate mode (used for changing the 3-D view using direct manipulation).
- **Viewer Controls** — Viewer X and Y rotation thumbwheels, Dolly thumbwheel, and Zoom slider.

HVE Mode Selector

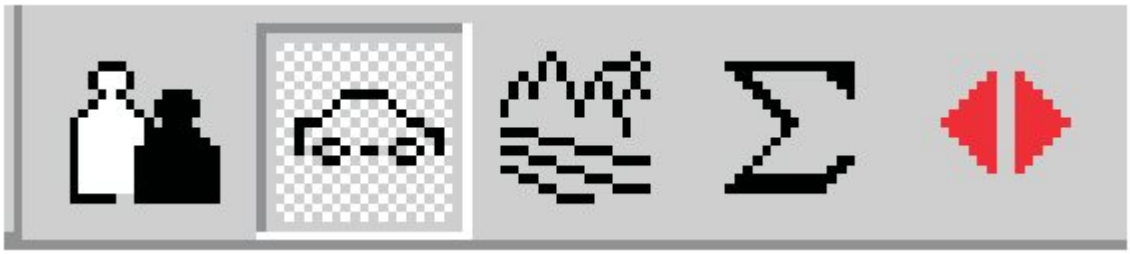


Figure 32-6 — HVE Mode Selector, located at the left side of the toolbar.

The HVE Mode Selector has five buttons (left to right): Human Editor, Vehicle Editor, Environment Editor, Event Editor and Playback Editor. Click on each button in the Mode Selector to display the various HVE editors.

Managing Dialogs and Viewers

Becoming an efficient HVE user requires a few simple, but important, procedures:

- Arranging your desktop to allow you to be as efficient as possible
- Raising viewers and dialogs to the top of the desktop
- Minimizing windows to help improve system performance (especially for 3-D viewers that must be rendered)

Minimizing Windows. While using HVE, your desktop may become cluttered with numerous data windows and viewers, especially in the Playback Editor when a case has multiple events with several output reports, or multiple Trajectory Simulation windows. In these instances, minimize windows so they take less screen space.

> **NOTE:** A minimized window maintains all the data associated with it; minimizing simply replaces the visual representation of the window with an icon.
>
> **NOTE:** Minimized 3-D viewer windows are not rendered. Because it takes time and computer resources to render a window, minimizing 3-D viewer windows can significantly speed up your work!

Practice (using the supplied tutorial case): open the case file `EdhisTutorialEDC.hve` (File menu, Open), choose Playback mode, click on a Traj Sim window to bring it to the top, then use its Dialog Control button and choose Minimize.

Figures 32-7 and 32-8 — Playback Editor with multiple reports, before and after one report window is minimized.

Choosing an Editor

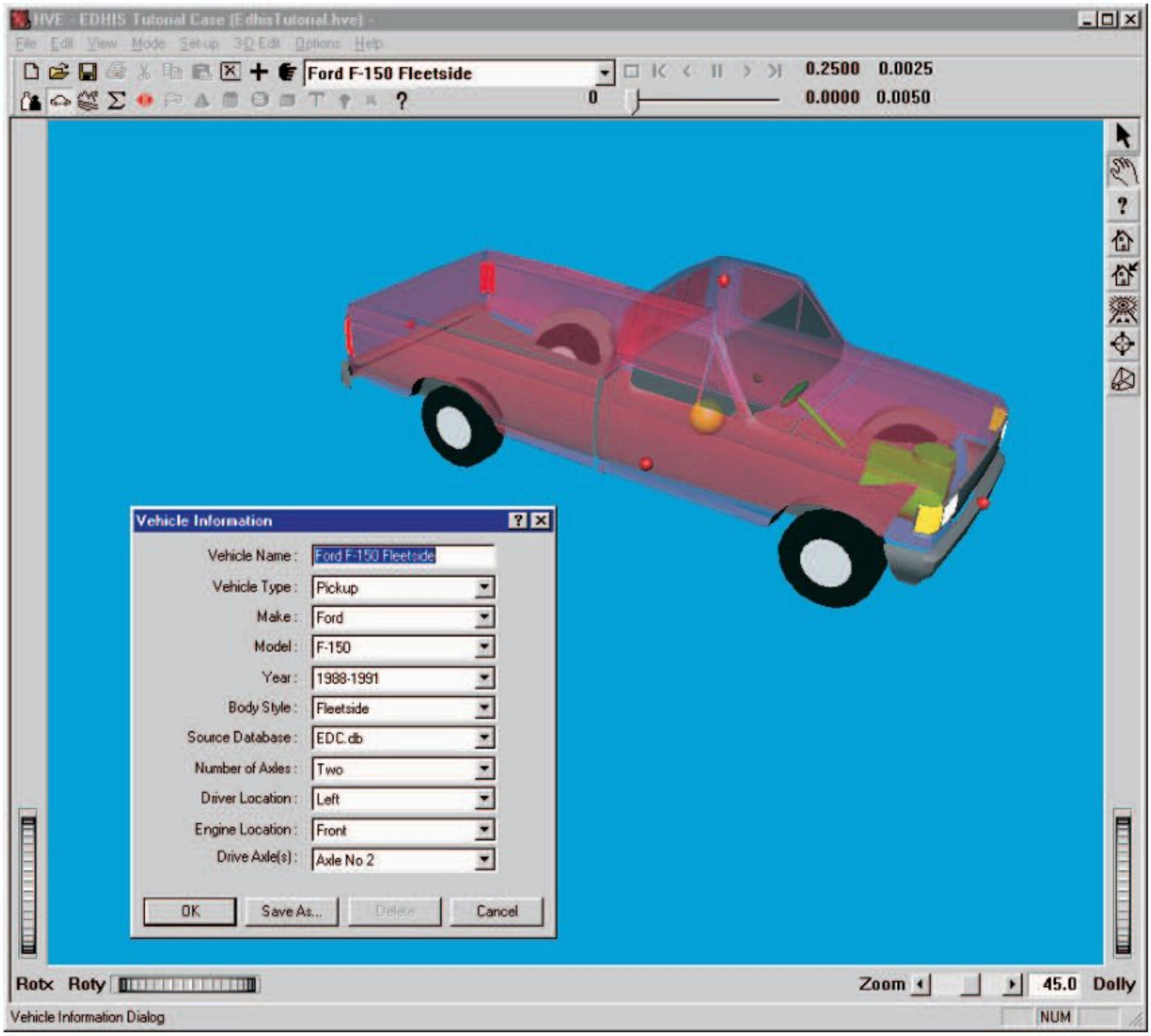
HVE is designed around five fundamental editors:

- **Human Editor** — Selecting and editing 3-D humans
- **Vehicle Editor** — Selecting and editing 3-D vehicles
- **Environment Editor** — Selecting and editing 3-D environments
- **Event Editor** — Setting up and executing reconstruction and simulation models
- **Playback Editor** — Creating visual and numeric output reports and videos

Switching editors is performed using the Mode Selector. Using the tutorial case, press each of the Mode Selector's pushbuttons in turn and observe: the Human Editor with the current human displayed in the Human 3-D viewer (Figure 32-9); the Vehicle Editor with the current vehicle displayed, including contact surfaces if Show Contacts is selected (Figure 32-10); the Environment Editor with the current environment (Figure 32-11 — note there is no Active Environments list, because there is only one environment); the Event Editor with the current event (Figure 32-12); and the Playback Editor with its report windows (Figure 32-13).

Figures 32-9 through 32-13 — The five HVE editors.

To inspect an object's basic attributes, select it in the Active Objects list and click the Object Info button on the toolbar. For a vehicle, the Vehicle Information dialog shows its basic attributes (Driver Location, Engine Location, Number of Axles, Drive Axles; Figure 32-14). For the environment, the Environment Information dialog (Figure 32-15) is used for assigning physical event information (sun position; wind direction and velocity for aerodynamics calculations; air temperature and pressure for use in air density calculations; and gravity constant), for assigning a 3-D geometry file to drive on, and for sky color and fog for visibility studies.



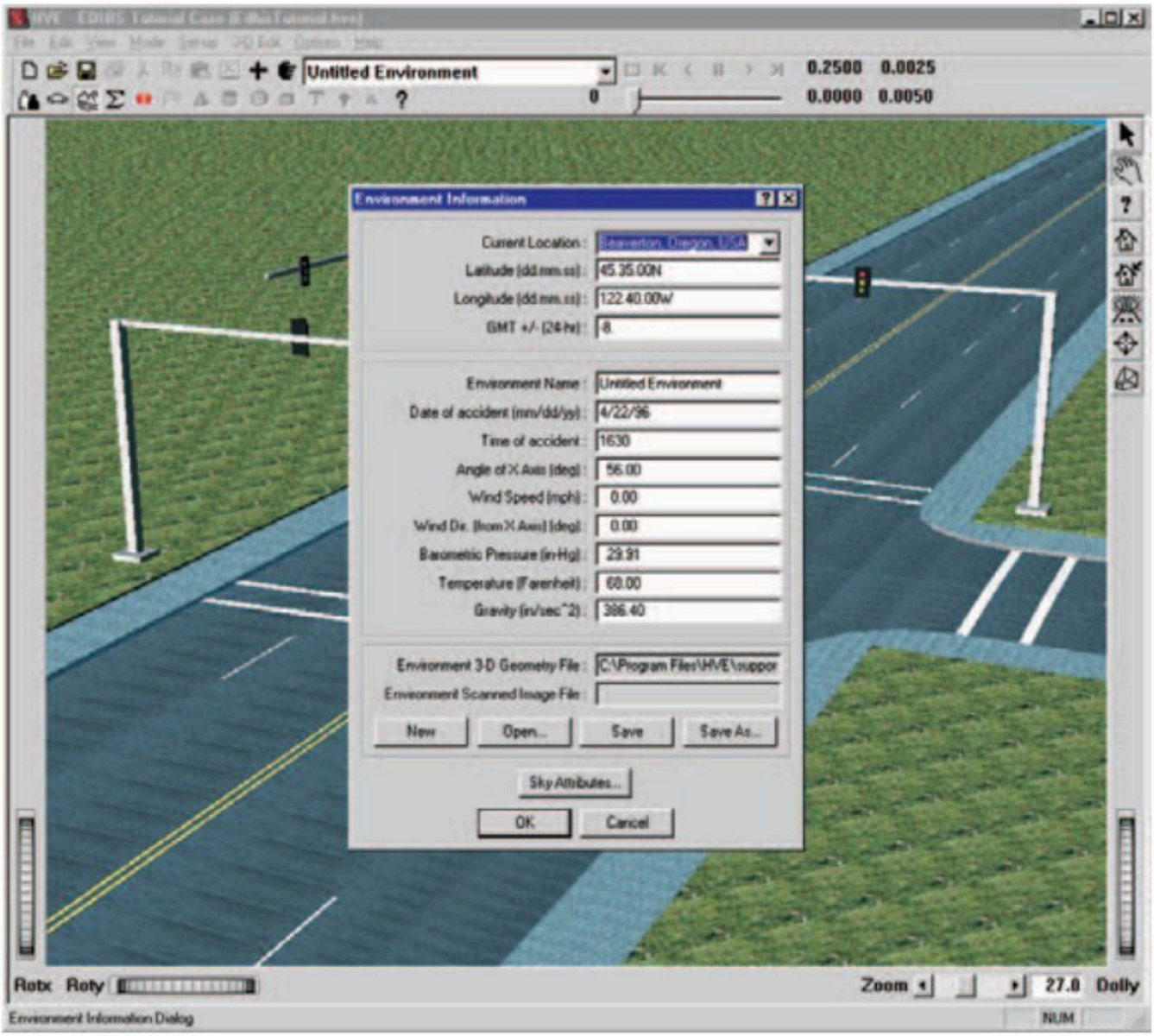


Figure 32-14 — Vehicle Information dialog. Figure 32-15 — Environment Information dialog.

Using Dialogs

HVE has more than 200 dialogs for entering and editing information. All HVE dialogs have default information already assigned; therefore, when using a dialog, you are always editing existing default data.

Modal vs Modeless Dialogs. HVE (like most applications) uses two types of dialogs:

- A **modal** dialog requires the user to complete an action by pressing OK or Cancel before another action can be performed. (You cannot make any other program selection until you remove the modal dialog — HVE beeps if you try. The File Selection dialog is a typical modal dialog.)
- A **modeless** dialog allows you to enter and edit data in the dialog, as well as make selections from the menu bar and manipulate objects in viewers, while the dialog is still displayed. The most important example is the

Position/Velocity dialog used by the Event Editor (Figure 32-16). You can tell it is modeless because it does not have OK/Cancel/Help buttons.

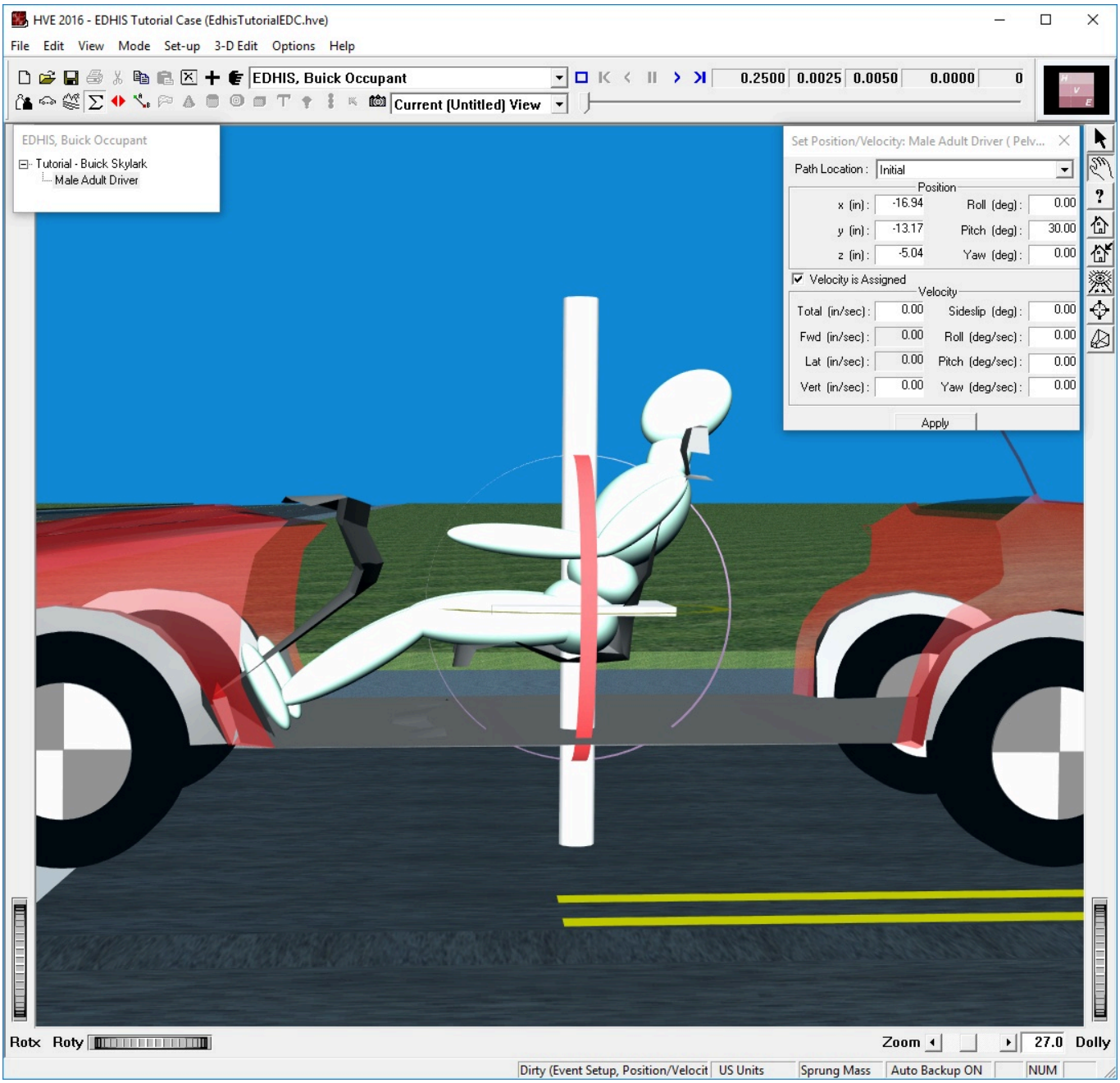


Figure 32-16 — HVE Event Editor's Position/Velocity dialog.

> **NOTE:** Nothing happens in a modeless dialog until you press the `<Enter>` key (or the Apply button). Pressing OK on a modal dialog is like pressing `<Enter>`; since a modeless dialog does not have an OK button, you must explicitly press `<Enter>` for the value(s) in the dialog to be updated.
>
> **NOTE:** You can enter several values into the dialog, then press `<Enter>` once to have all the values updated at the same time. This can save time, since HVE re-renders the viewer whenever the viewer's contents change.

Using 3-D Viewers

HVE uses 3-D viewers to display humans, vehicles and the environment, and to visualize motion during Event and Playback modes. The user can perform the following viewer operations:

Resizing Viewers. Click on any edge of the viewer frame (the cursor changes shape) and drag to grow the viewer in width and/or height; drag a corner to resize in both directions. Clicking the edge of a partially covered window also brings it to the top of the desktop. While 3-D viewers may be resized, dialogs normally cannot. In general, make your viewers nice and BIG!

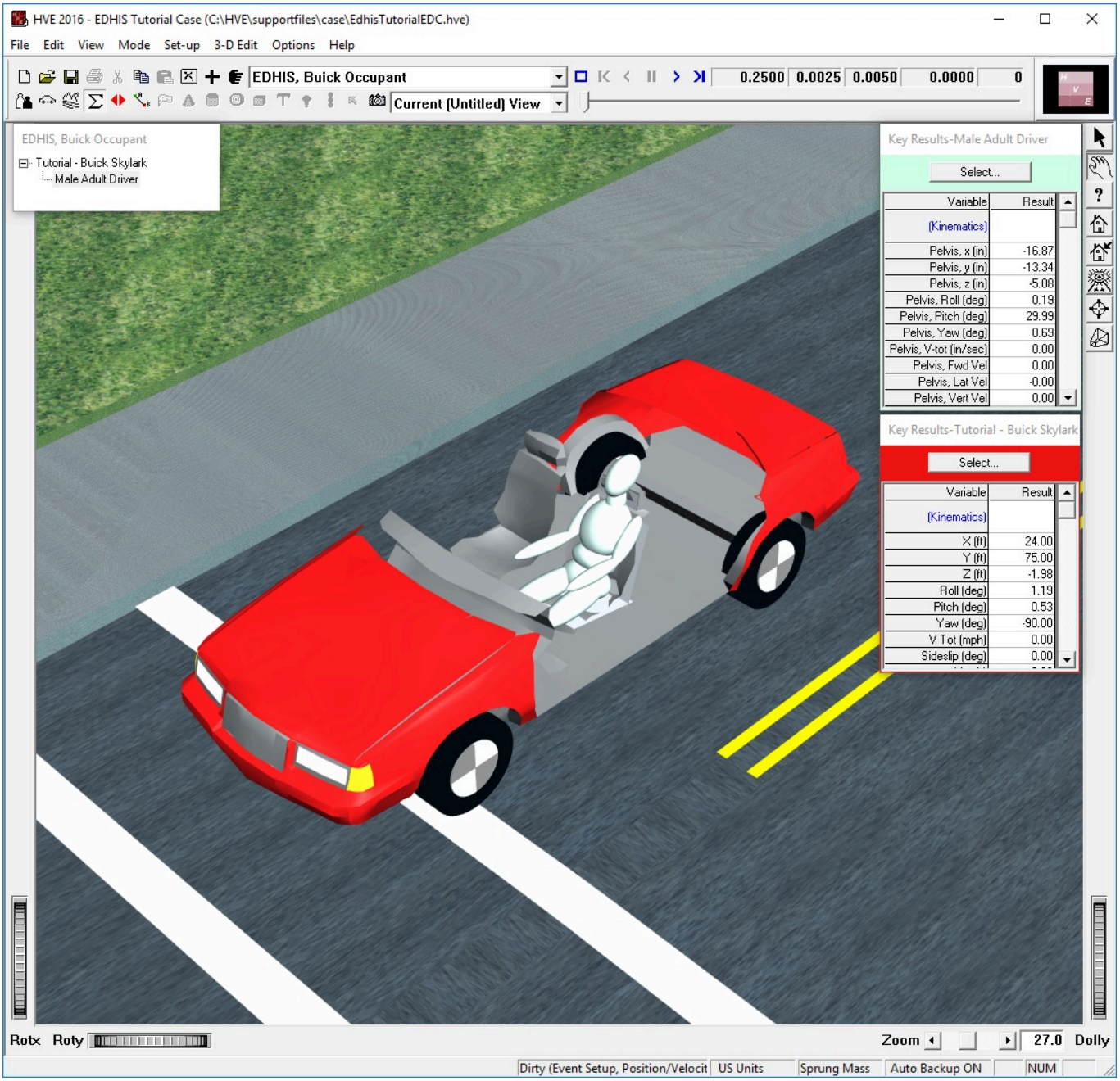


Figure 32-17 — Using HVE 3-D viewers.

Panning. Panning moves the scene horizontally and vertically within the viewer. Confirm the viewer is in Manipulate mode (click the hand icon), place the mouse cursor in the middle of the viewer, then press the middle mouse button and

drag (or press the left mouse button while holding <Shift>).

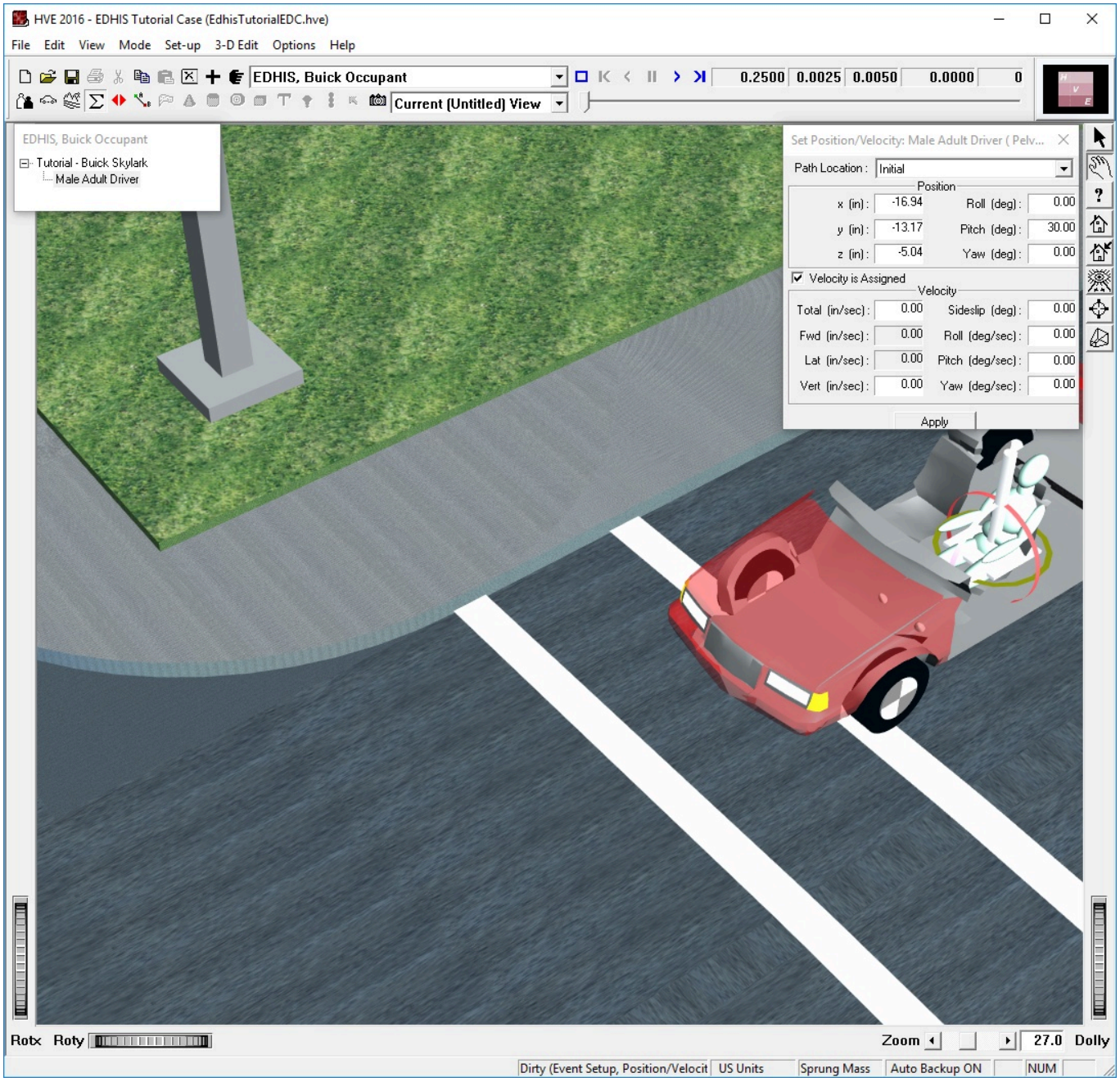


Figure 32-18 — Panning the camera in a 3-D viewer.

Zooming In and Out. Zooming changes the lens of the virtual camera. The zoom slider is located along the lower right edge of the viewer. Dragging the slider zooms in (analogous to changing from a 50 mm lens to a 150 mm telephoto lens) or out.

Dollying In and Out. Dollying changes the camera position, moving it closer to or farther from the object (just like filming a movie scene). Press both the left and middle mouse buttons and drag down/up (or press the left mouse button with <Shift>+<Ctrl>), or use the Dolly thumbwheel. Pressing <Escape> with the cursor in the viewer toggles between Pick and Manipulate modes — equivalent to clicking the viewer mode icons.

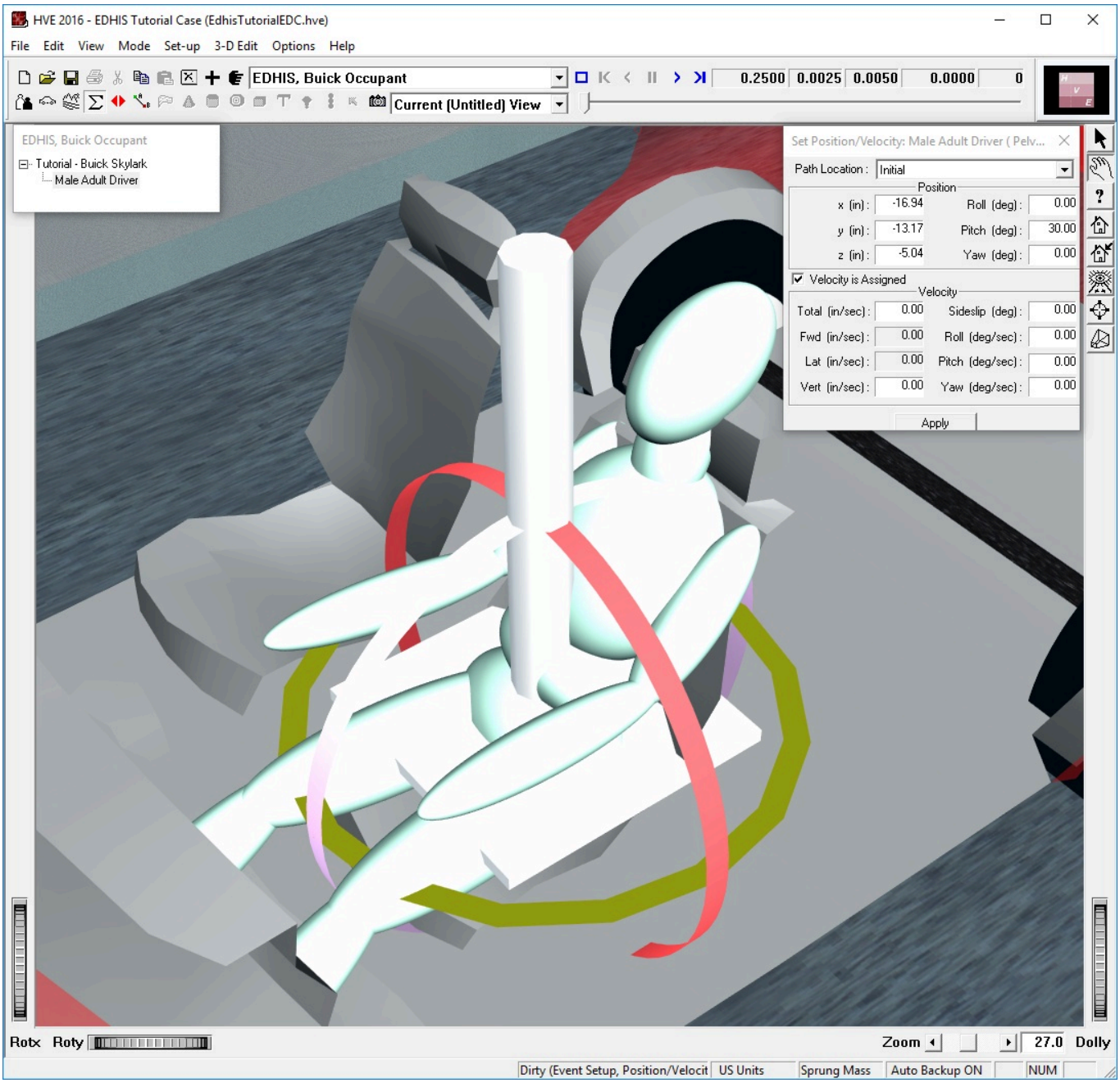


Figure 32-19 — Dollying the camera in a 3-D viewer.

> **NOTE:** The thumbwheel's range of motion is not limited by its visible size — click it and keep dragging.
>
> **NOTE:** Both dollying and zooming appear to have the same effect (making objects appear larger). However, if you zoom in too close, the objects become distorted as if you were using a fisheye lens. For this reason, we recommend dollying instead of zooming if your goal is a close-up view.

Changing Camera Position. In Manipulate mode, click the left mouse button and drag horizontally: the object spins about the viewer's vertical axis; drag vertically and it spins about the viewer's horizontal axis.

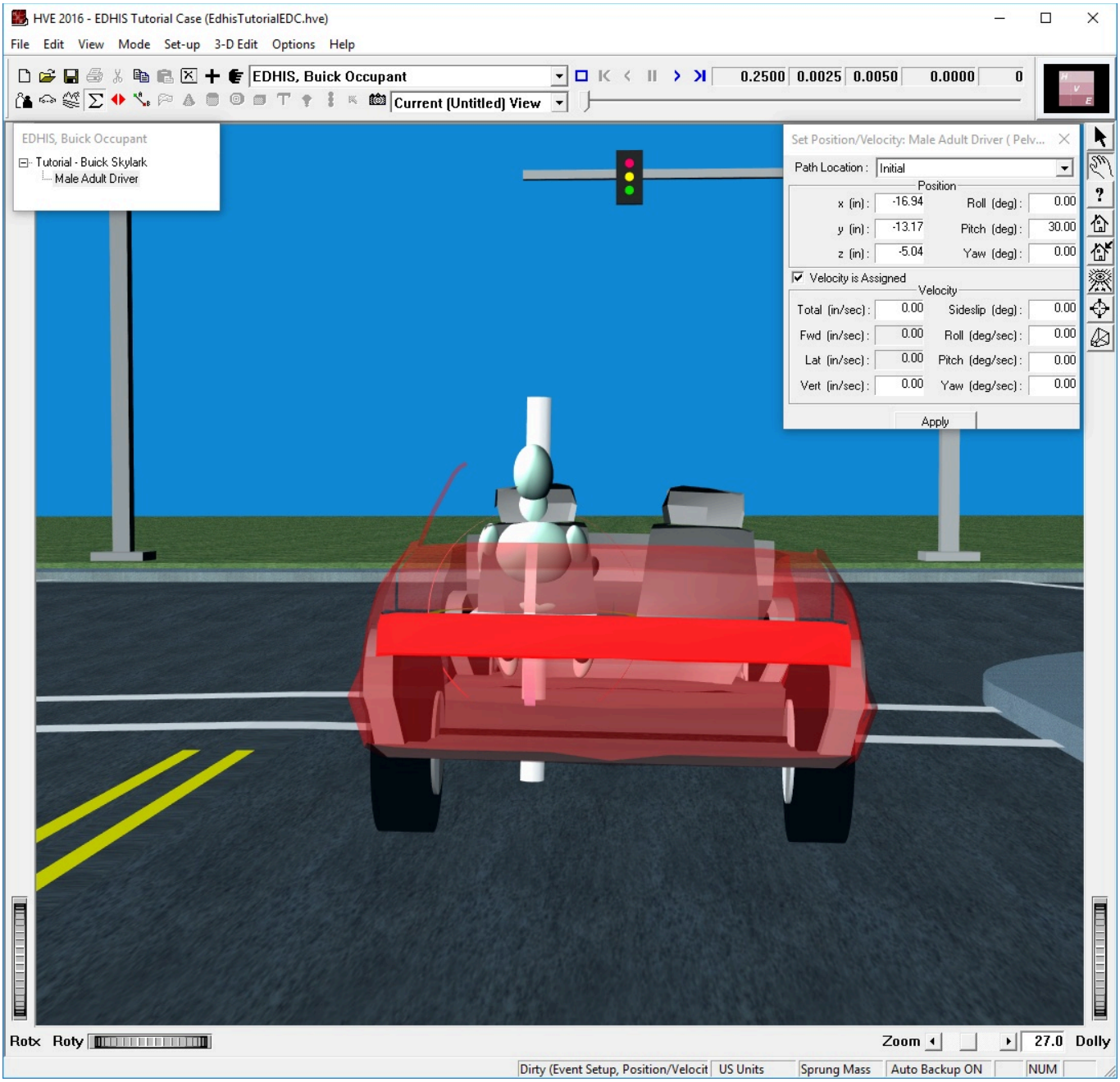


Figure 32-20 — Moving the camera in a 3-D viewer.

> **NOTE:** This is an important concept: while using direct manipulation, the scene always spins about the center of the viewer.

Spend time practicing direct manipulation combined with panning, dollying and rotating. By mastering viewer manipulation you will benefit every time you use HVE.

Shutting Down HVE

When you are finished working with HVE, end your session by saving your work and exiting HVE. It is important to exit HVE gracefully because HVE asks you to save your work if it has changed since your last save, and because HVE

updates your configuration file so your next HVE session uses the same preferences and options selected in the current session. HVE also saves the current editor when you exit.

To shut down HVE, click on the File menu and choose Exit, then answer the save-changes prompt.

Lesson 2 — Running a Simulation Model

Lesson 2 is a high-level overview describing how to use HVE to execute a single event (reconstruction or simulation program). In this tutorial we execute an EDVSM simulation. You will learn: Selecting Humans; Selecting Vehicles; Selecting Environments; Creating Events; Setting Up Events; Executing Events. Mastering these tasks is important because they are the same tasks you'll use every time you use HVE — regardless of the reconstruction or simulation model you're executing.

Selecting Humans

1. Switch to Human mode. The Human Editor is displayed.
2. Click Add New Object. The Human Information dialog is displayed. It includes option buttons allowing the user to select a seat position within the vehicle (alternatively, Pedestrian could be selected), and assign the human's attributes according to Sex, Age, Weight Percentile and Height Percentile.
3. Choose: Location = Front, Left; Sex = Male; Age = Adult; Weight Percentile = 50; Height Percentile = 50.
4. Replace the default name with a more meaningful one — enter `Fred` — and press OK.

Fred is displayed in the Human Editor's 3-D viewer (Figure 32-21), in a seated position because we assigned him as an occupant. Spend a few moments visualizing Fred using the techniques from Lesson 1.

Selecting Vehicles

1. Choose Vehicle Mode. The Vehicle Editor is displayed.
2. Click Add New Object. The Vehicle Information dialog is displayed, allowing selection of the basic vehicle attributes according to Type, Make, Model, Year and Body Style.

> **NOTE:** The Vehicle Information dialog also allows you to edit the Driver Location, Engine Location, Number of Axles and Drive Axle(s). These options affect the basic vehicle configuration and do not need to be changed for this tutorial.

1. Choose: Type = Passenger Car; Make = Dodge; Model = Charger; Year = 1968–1970; Body Style = 2-Door; Source Database = Tutorial.db.
2. Click OK to add the Dodge Charger to the Active Vehicles list.

The Charger is displayed in the Vehicle Editor's 3-D viewer (Figure 32-22).

Selecting Environments

1. Switch to Environment mode and click Add New Object. The Environment Information dialog is displayed.
2. Choose Open. The 3-D Geometry File Selection dialog is displayed (Figure 32-23). If necessary, set Files of Type to HVE files.
3. Choose `4x2_Intersection.h3d` and press OK.

4. Replace the default environment name — enter `Tutorial Lesson 2 Intersection` — and press OK.

The selected environment is displayed in the Environment Editor's 3-D viewer (Figure 32-24).

Event Editor

Using the Event Editor involves three steps:

- **Creating the Event** — Choosing the humans, vehicles and the calculation method (reconstruction or simulation model) for the event.
- **Setting Up the Event** — Assigning the position and velocity for each human and vehicle in the event; may also involve driver controls, restraint systems, damage profiles, other event set-up options, simulation controls and calculation-specific options.
- **Executing the Event** — Pressing the Execute pushbutton and observing the results; normally also making adjustments to inputs and performing range-checking runs to identify sensitivities.

To create the event:

1. Switch to Event Mode and click Add New Object. The Event Information dialog is displayed (Figure 32-25), containing the Active Humans list, Active Vehicles list and Calculation Method option list.
2. Select the Dodge Charger from the Active Vehicles list; it is added to the Event Humans & Vehicles list.
3. Choose EDVSM from the Calculation Method option list.
4. Change the default event name to `Sample Event` and press OK.

To set up the event (initial position/velocity and a steer table):

1. Select the Dodge Charger in the Event Humans & Vehicles list, then choose Set-up from the menu bar. The Event Set-up options for EDVSM are displayed (Figure 32-26).

> **NOTE:** If no object is selected, no options are enabled in the Set-up menu. Only the set-up options applicable to the chosen physics program are displayed — each physics program tells HVE which set-up options to make available.

1. Select Position/Velocity. The Initial Position/Velocity dialog is displayed and the Charger appears at the earth-fixed origin (Figure 32-27).
2. Enter the initial position: X = −70.0 ft, Y = 37.5 ft; and initial Total Velocity = 25.0 mph, followed by `<Enter>`.

> **NOTE:** By waiting until all the data are entered before pressing `<Enter>`, the 3-D viewer is only re-rendered once. Clicking Apply is the same as pressing `<Enter>`.

1. Choose Set-up, Driver Controls, and select the Steer tab. Enter the steer table (Figure 32-28):

Table 32-1 — Steer Table entries for the Dodge Charger's left turn

Time (sec)	Steering Wheel Angle (deg)
0.00	0.0
0.50	0.0
1.00	−150.0
3.50	−150.0
4.00	0.0

1. Press OK to accept the steer table.

Use the Event Controller (Figure 32-29) to execute the event. The Event Controller's buttons function like a VCR controller: **Reset** (reinitialize the calculation model for re-execution), **Rewind to Start**, **Reverse**, **Pause**, **Execute/Play**, and **Advance to End**.

> **NOTE:** If you make changes to any of the event set-up options, those changes have no effect unless you press Reset before pressing Play.

Press Execute/Play. The Dodge Charger begins moving as its motion is simulated according to the vehicle model, the current environment terrain, the initial position and velocity, and the specified driver steering inputs (Figure 32-30). The simulation continues until the default maximum simulation time, 5.0 seconds, is reached.

Replay the event using Rewind — this redisplays the current output tracks without re-executing the physics calculations.

Key Results. Numeric results are displayed in Key Results windows for each timestep during execution. If a Key Results window is not displayed, choose Options, Show Key Results. By default, the vehicle's Sprung Mass Kinematics results are displayed. To add tire forces:

1. Click Select Variables in the Key Results window. The Variable Selection dialog is displayed (Figure 32-31).
2. Click Tires, Axle 1, Right, Outer; select Fx, Fy and Fz; press OK.
3. Press Execute/Play to redisplay the event with the selected Key Results at each timestep (Figure 32-32).

> **NOTE:** Like all events, this EDVSM event produces several output reports (Accident History, Messages, Vehicle Data and others). The Playback Editor is used to view (and print) these reports. Refer to the program manual for the specific simulation program for further information.

Lesson 3 — Combining Multiple Events

Most crashes involve a sequence of events (e.g., loss of control, collision, occupant ejection). HVE's open architecture allows users to choose the tool best suited for each specific event from a variety of reconstruction and simulation tools. The HVE Playback Editor combines these individual events into a single window, and edits the sequence so each event is timed properly with respect to the other events. The result is a seamless, real-time view of the entire multiple-event sequence.

In this lesson you will learn: Displaying Multiple Trajectory Simulation Events; Combining the Multiple Events Into a Single Playback Window; Editing the Time Sequence for the Events. These tasks are also a prerequisite for creating video output, the subject of Lesson 4.

Open the prepared case: File, Open; select `HveTutorialLesson3EDC` and press OK. Choose Event mode. The tutorial case includes five events (Figure 32-33):

- **Event 1** — EDSMAC4, 3-Vehicle Crash (Ford Taurus, Honda Accord, Chevrolet Suburban)
- **Event 2** — EDVSM, Ford Taurus Post-impact
- **Event 3** — EDVSM, Honda Accord Post-impact
- **Event 4** — EDVSM, Chevrolet Suburban Post-impact
- **Event 5** — EDVDS, Tractor-trailer Avoidance

Review each event: choose it in the Active Events list and press Play in the Event Controller, watching it in the Event 3-D viewer from different perspectives.

> **NOTE:** The post-impact events begin at separation, not at the start of the entire sequence.

Displaying Multiple Trajectory Simulations

1. Choose Playback mode. The Playback Editor is displayed (Figure 32-34); initially no reports are displayed.
2. Click Add New Object. The Report Window Information dialog is displayed (Figure 32-35), showing each event in the Active Events list.
3. Select the first event, EDSMAC4, 3-Vehicle Crash; choose Traj Sim from the Report Type list; press OK. The Trajectory Simulation window for the first event is displayed (Figure 32-36).
4. Repeat for the remaining four events. After adding the fifth window and arranging them, the desktop appears as in Figure 32-38.

Editing the Time Sequence

Three of the five events require editing the start time so the motion for each event begins at the proper time. Because these events begin at separation following the 3-vehicle collision, the times are available in the Accident History report of the first event. Add it via Add New Object, select EDSMAC4, 3-Vehicle Crash, choose Accident History, press OK (Figure 32-39). From the report:

- **Event 2** — EDVSM, Ford Taurus Post-impact: separation begins at 8.99 seconds; however, the vehicle still has residual velocity at the end of the simulation time (10.00 seconds), so the final position is used for the start of this event. *(This vehicle is involved in two impacts; the end of the second impact, with the Chevrolet Suburban, is chosen for the separation conditions.)*
- **Event 3** — EDVSM, Honda Accord Post-impact: separation occurs at 8.44 seconds.
- **Event 4** — EDVSM, Chevrolet Suburban Post-impact: the event's start must be delayed until separation at 8.99 seconds.

Combining the Multiple Events

1. Select Options, Add Playback Window from the main menu. The Playback Window is displayed (Figure 32-41). The Active Trajectory Simulations list displays all five Trajectory Simulation windows in creation order (Figure 32-42).
2. Make sure all five Trajectory Simulations are selected for display (check boxes next to the event names).
3. Edit the Window Name field from `Untitled Playback Window` to `Combining Multiple Events`.

4. Set the timing: enter Tdelay = 10.0000 sec for EDVSM, Ford Taurus Post-impact; Tdelay = 8.4400 sec for EDVSM, Honda Accord Post-impact; and Tdelay = 8.9900 sec for EDVSM, Chevrolet Suburban Post-impact — pressing `<Enter>` after each (this is a modeless dialog).
5. Press Play. The playback window shows the entire sequence properly timed. Although you cannot tell by inspection, the motion for all three colliding vehicles is first handled by EDSMAC4, then transferred to EDVSM following their impact.

Audit Trail. Press Audit Trail to display a table report of each event in the Playback window (Figure 32-43) — it tells the user which events are controlling the motion of each object. Press OK to remove it.

Finally, save the case file (File, Save): enter case title `Combining Multiple Events` and filename `HveTutorialLesson4`, then press OK. The file is saved in the `HVE\supportFiles\case` subdirectory. (The Case Title appears as a heading on all printed output reports.)

Lesson 4 — Creating a Video

The final task while using HVE often includes creating a video of the crash sequence. Because it is visual, a video presentation is often the best means of describing and explaining how a crash occurred.

HVE's Playback Editor includes an integrated video output interface that allows you to produce a simulation movie without additional software. (If you want to create a full-length movie containing multiple views of the crash sequence, use a software program dedicated to editing movie clips together.)

In this lesson you will learn to: route a multi-event crash sequence to the video recorder; replay the sequence in real time; and save the sequence to a Video for Windows (AVI) file.

First confirm the video options. Open the Video Setup dialog and check the compressor selection (Figure 32-44).
(updated: the original tutorial recommended selecting the Cinepak Codec from choices such as Full Frames (Uncompressed) and Microsoft Video 1; those codecs are obsolete — choose any modern codec installed on your system, or Full Frames (Uncompressed) for a lossless master. In current HVE the Video Options dialog is opened with the Video Setup button on the Playback window, and also sets the movie size and frame rate; see Chapter 27.)

If you just finished Lesson 3 and are still in the Playback Editor, proceed directly. Otherwise open the prepared case `HveTutorialLesson4EDC` and choose Playback mode.

> **NOTE:** The Traj Sim reports might be minimized in a single stack in the bottom left corner of the desktop. You may wish to leave them minimized to reduce the load on your graphics card, which would otherwise refresh all viewers at every frame of output.

Route to Video Recorder

The HVE video interface has three simple components:

- **Playback Controller** — a VCR-like panel that allows you to display and record the results of your simulation
- **Source Option List** — all the possible sources of the simulation displayed in the Playback Window
- **Destination Option List** — all the possible destinations that can receive the simulation displayed in the Playback Window

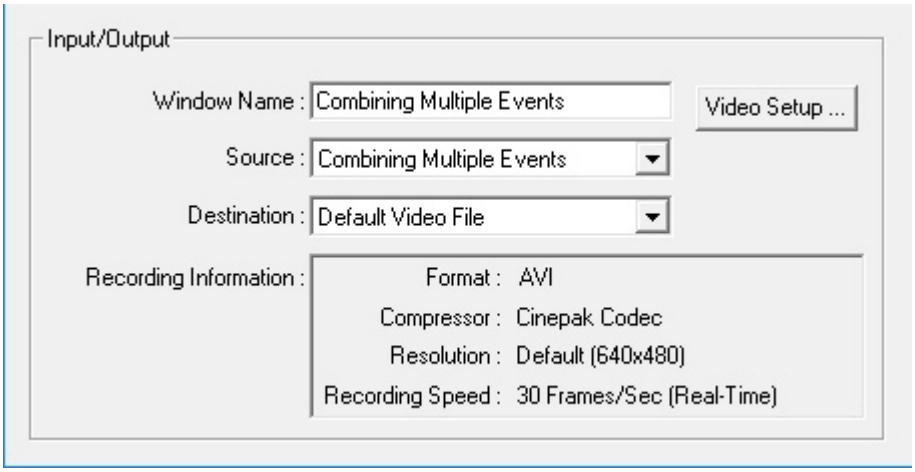


Figure 32-45 — Close-up of part of the Playback Window dialog, used for playing and recording simulations.

Think about the Source and Destination options for a moment. It is really a simple concept: get the simulation from the Source and put it in the Destination. To record our simulation:

1. Click on the Source option list and choose `Combining Multiple Events` (the Playback Window). *(updated: on legacy systems the list showed the playback window, S-Video/Composite Video and AVI; current HVE lists the playback window and saved movie files.)*
2. Click on the Destination option list and choose the video file (AVI).
3. Confirm the current simulation time is 0.0000 seconds (otherwise recording would start after the beginning of the sequence).
4. Click on the Options menu and choose Playback. Confirm the Playback Interval is set to 0.0333 seconds (30 frames/sec) or 0.0250 seconds (25 frames/sec). Press OK.

> **NOTE:** If you halve the Playback Interval, the simulation will be recorded in slow motion (twice as slow as normal).

1. Confirm you have set the view in the Playback Window to your liking (Figure 32-46).
2. Confirm no other windows are covering the Playback Window — otherwise they will show up in your simulation movie.
3. Press Play. The simulation is recorded to disk, and is also displayed in the Playback Window so you can monitor its progress.

> **NOTE:** Recording is not real-time — each frame is rendered and compressed individually. For a complex sequence, you may wish to go get a cup of coffee (or tea) while recording.

Replay in Real Time

After the simulation has been recorded to disk, you can replay the entire sequence in real time. All you need to do is switch the Source and Destination:

1. Click on the Source option list and choose the movie file (AVI).
2. Click on the Destination option list and choose `Combining Multiple Events` (the Playback Window).
3. Press Play.

The simulation data is retrieved from the disk and displayed in real time in the Playback Window.

> **NOTE:** The viewer controls are not available while replaying a movie. That's because the view was established when the simulation was recorded and cannot be modified afterward.

Route to Videotape *(legacy)*

On legacy systems with video-out hardware, an optional step was to create a videotape: choose S-Video/Composite Video as the Destination, activate the computer's video-out capability, press Play to activate the special full-frame display, turn on the VCR, insert a good-quality blank tape, advance the tape about 20 seconds of leader, press the VCR's Record button, wait 5 seconds (recording an initial still image, which prepares the viewer for what they are about to see), then press Play on the special display. After the sequence finished, wait another 5 seconds before pressing the VCR's Stop button.

(updated: current HVE has no video-device destination; to produce physical media, record the AVI file and use standard authoring tools.)

Saving the AVI File

The final step is to save the AVI file for convenient future viewing:

1. Open the Video Setup dialog.
2. Choose Save As. The File Selection dialog is displayed.
3. Enter a filename: `HveTutorialLesson4`. Press OK.

The AVI video is saved as `HveTutorialLesson4.avi` (HVE appends the `.avi` extension) in the `hve\supportFiles\images\movies` subdirectory.

Changing the View

Figure 32-46 shows a view of the crash scene from an interesting perspective — all the collision action is displayed. However, it does not show what each driver saw as the crash sequence evolved. These views are important: by viewing the sequence as seen by each driver, it becomes obvious how the crash occurred and which driver made the first mistake.

To view the sequence from each driver's perspective, attach a virtual camera to each vehicle and replay the simulation:

1. Click on the View menu and choose Set Camera. The Set Camera dialog is displayed (Figure 32-47).
2. Click the Object Name list in the View From group and choose Honda Accord.
3. Replace the View From coordinates with a location above and behind the driver's head: x = −13.0 ft, y = −1.5 ft, z = −5.0 ft.
4. Click the Object Name list in the Look At group and choose Chevrolet Suburban, with Look At coordinates x = 0.0, y = 0.0, z = 0.0 ft (the Suburban's CG).
5. Press OK. The scene is viewed from the Honda driver's perspective (Figure 32-48).
6. Press Play. The Playback viewer displays the entire sequence as viewed by the driver of the Honda.

Repeat, using the Set Camera dialog to watch the sequence from each driver's perspective — simply select the desired vehicles in the Object Name lists of the View From and Look At groups; the coordinates don't require adjusting.

> **NOTE:** Even though you used the Set Camera dialog to set the view, you can still use the viewer controls (thumbwheels and direct manipulation) to fine-tune the view.

Ending the Tutorial

This concludes Lesson 4 — Creating a Video. In Lesson 4, we learned how to create an AVI video file, play it back on the computer in real time, and save the simulation movie file for future viewing using HVE (or any other AVI viewer).

Appendix I — Installation & Setup

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix I, pages I-1 through I-10. The installation and licensing procedures below describe the 2006-era distribution (CD-ROM, parallel-port EDKEY, Windows 95–XP) and are retained as legacy reference; consult EDC (www.edccorp.com) for current system requirements, installation media and licensing. The HVE file system section describes the file locations used by the current version of HVE.

This section of the manual describes the basic computer hardware system requirements for using HVE and also how to install HVE software.

System Requirements *(legacy)*

HVE runs on computers running a Windows operating system. The recommended requirements printed in the original edition were:

- Intel Pentium IV processor (or similar)
- 128 MB RAM
- 9 GB hard disk
- Windows 95/98/ME or NT/2000/XP
- CD-ROM drive
- OpenGL-compatible graphics card (32 MB or larger)
- Video output device
- 3-button mouse
- 17-inch monitor

(updated: these figures are far below any modern PC; current HVE is a 64-bit Windows application. For the latest information about HVE and recommended hardware configurations, contact EDC Technical Support or visit www.edccorp.com.)

HVE Software Installation *(legacy)*

For the first installation and for software updates, HVE was installed directly from CD-ROM.

> **NOTE:** Please review the next section, Software Security and Protection, before installing your HVE software.

Installation From CD-ROM

- **Windows 95/98/Me:** Insert the HVE CD-ROM; the installation program starts automatically.
- **Windows NT/2000/XP:** You must log in as Administrator to install HVE. Insert the CD; the installation program starts automatically.

> **NOTE:** If the installation program does not start automatically, start it by clicking Start, Run..., and typing `F:\Setup` (where F is the drive letter for your CD-ROM drive).

The installation program guides you through the installation. You will be asked to type in your name, your company name and your User ID. This information should correspond to the information printed on the license media included

with your HVE program CD. If you are installing an update, this information is already filled in; confirm that it is correct.

You will be asked to choose one of two license options:

- *I will use the license provided by EDC* — a file browser is displayed; browse to the location of your license file (HVE license files use the `.LIC` extension), then select and open it.
- *No license. I will run in Demo Mode* — the license file selection is bypassed. HVE will run in Demo Mode, with certain features disabled:
- Vehicles cannot be selected from the EDC Custom Vehicle database (vehicles are selectable from the Generic and Tutorial databases).
- New events cannot be created; however, you can replay, modify and re-execute existing events from previously saved HVE or HVE-2D cases.
- Case files cannot be saved.

Before installing files, the installation program displays all your selections; choose Back to correct anything. After installation is complete, you will be asked to restart your computer.

> **NOTE:** You may choose not to restart your computer; however, HVE cannot be used until your computer has been restarted!
>
> **NOTE:** If you installed a software update, the installation overwrites existing files of the same name, but does not remove any other files, so user-saved data files will not be damaged.
>
> **NOTE:** The installation program makes a copy of your `Language.rsc` file before copying the new one. If you have made edits to your previous `Language.rsc` file that you wish to transfer to your new HVE installation, you can copy and paste the modified portions of your previous file into the new one. EDC STRONGLY RECOMMENDS that you make a back-up copy of the new `Language.rsc` file before making changes. HVE will not run with a defective `Language.rsc` file!

Software Security and Protection *(legacy)*

Your HVE software and its components are protected from unauthorized use by one of two methods, chosen at time of purchase:

Lock to Your Computer's Hard Disk

Prior to the time of purchase, you may have received a program from EDC called `ECHOID.EXE`:

1. Execute the ECHOID program. A dialog with two lines of information appears. The first line, of the form `4 - XXXXX`, is the information you will need to provide EDC Customer Service to receive your license file. Record the complete six-digit code.
2. Phone, fax or email this six-digit code, along with your User ID number and contact information, to EDC Customer Service.
3. The appropriate license file will be included with your software package.

If you did not receive a license file with your HVE software package: install HVE in Demo Mode, then select Start, Programs, HVE, Show Computer ID and follow the same code-reporting procedure. EDC will send the license file via

email or mail; while waiting (normally a few hours) you can use HVE in Demo Mode.

Once you have received the license file, copy it to your HVE program directory: either reinstall HVE and choose the license file during installation, or copy the license file (supplied with a filename of the form `hveXXX-X.lic`) directly to your `\Hve` program folder and rename it `lservrc`.

> **NOTE:** Be very careful when you rename the file. It should be in lower case, and should include no period/dot or extension.

Lock to Your EDKEY

If you chose this method, a license file was supplied with your HVE software and must be selected during installation. In addition, you must install the EDKEY (a parallel-port hardware key) before your HVE software can be used:

1. Locate your computer's parallel port (25-pin female connector).
2. If a printer cable is connected, disconnect it.
3. Plug the EDKEY into the parallel port.
4. If a printer cable was connected, reconnect it to the other end of the EDKEY (printer data passes through the EDKEY).

> **NOTE:** The User ID on the EDKEY must match the User ID on the license file. Otherwise, HVE will run only in Demo Mode.
>
> **NOTE:** Your computer may have additional hardware dongles attached to the parallel port. These may normally be stacked back-to-back without affecting the operation of other programs.

HVE File System

It is important that users of HVE know and understand its file system in order to perform system maintenance as well as to copy files between computer systems.

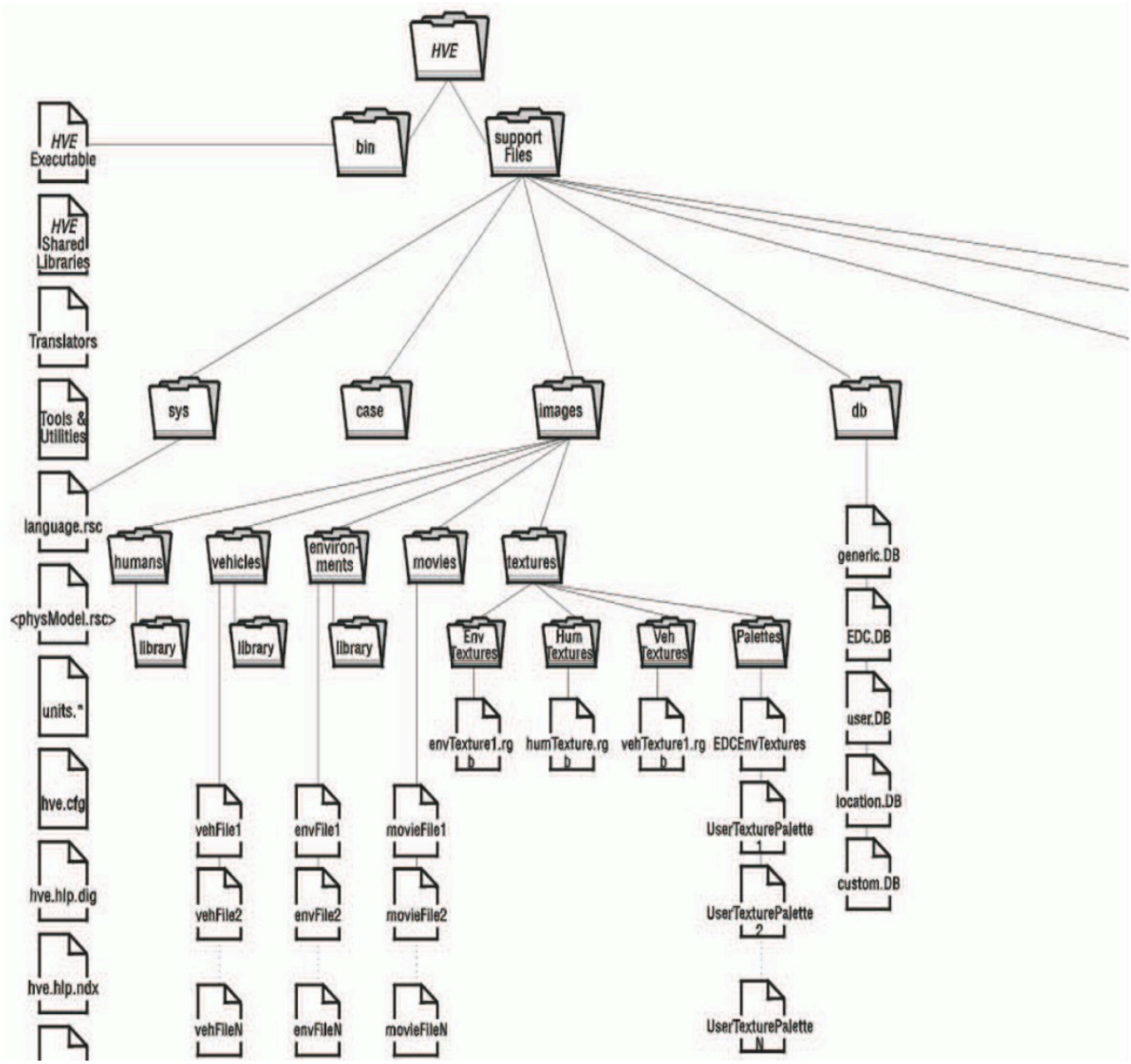


Figure I-1 — HVE File System diagram.

Table I-1 — Directory Names and Descriptions

Directory Name	Description
`hve`	Main installation directory for HVE
`hve\supportFiles`	Root subdirectory for the various support file systems required by HVE
`hve\supportFiles\case`	Subdirectory containing all HVE case files
`hve\supportFiles\images`	Root subdirectory for all image files (2-D bitmaps and 3-D geometry)
`hve\supportFiles\images\humans`	Subdirectory containing 2-D bitmap and 3-D geometry files for humans
`hve\supportFiles\images\vehicles`	Subdirectory containing 2-D bitmap and 3-D geometry files for vehicles
`hve\supportFiles\images\environments`	Subdirectory containing 2-D bitmap and 3-D geometry files for environments
`hve\supportFiles\images\movies`	Subdirectory containing movie files (e.g., AVI) created using the Playback Editor, and (in a `frames` subdirectory) single-frame image output *(updated: the original table printed this as `supportFiles\movies`; HVE places movies under `supportFiles\images\movies` — in current installations under the public user directory, e.g. `C:\Users\Public\HVE\supportFiles\images\movies` — and the location may be overridden in the Video Options dialog)*
`hve\supportFiles\db`	Subdirectory containing human, vehicle and tire databases; also contains location and vehicle contact surface materials databases
`hve\supportFiles\colPulse`	Subdirectory containing user-saved collision pulse files
`hve\supportFiles\sys`	Subdirectory containing various HVE system and resource files

Table I-2 — Files in `\hve` directory

File Name	Description
`hve` (executable)	HVE executable
`hve*.lic`	HVE license file

Table I-3 — Files in `\hve\supportFiles\sys` subdirectory

File Name	Description
`language.rsc`	Resource file for HVE; contains text strings, unit name format and error/warning ranges for each parameter in the user interface. Also contains all units conversion factors (used for converting user units to program units)
`<phys_model>.rsc`	Resource file for each installed reconstruction or simulation model (contains text strings for all output report strings)
`units.si`	Resource file for SI (metric) units; contains user name string for each program unit
`units.us`	Resource file for US units; contains user name string for each program unit
`hve.cfg`	Configuration file; saves current user preferences and other parameters on HVE shutdown

Appendix II — Messages

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix II, pages II-1 through II-14.

HVE displays a number of messages alerting the user to various conditions that have occurred. There are two main types of messages:

- **Information** — These messages provide information to the user and require no response.
- **Warning or Error** — These messages tell the user something has gone wrong and require the user to respond by taking corrective action.

This appendix describes the messages produced by HVE.

Input Parameter Validation

While entering data, each parameter is validated to determine that it is within an allowable range. HVE performs two levels of validation: Warning and Error. The *warning range* represents a range of inputs that is outside the normal range for a parameter, but is still allowed. The user is alerted to the suspicious data and may choose to view and change the data or ignore the warning. The *error range* represents a range of values that is not allowed, either because the calculation model would fail (e.g., a division by zero), or because the entered value is simply preposterous (e.g., an engine speed of 3,000,000 RPM). In this case, the user must correct the errant value before HVE will allow the user to continue. The allowable range is displayed in the dialog.

> **NOTE:** These ranges are assigned in the `language.rsc` file and are user-editable.

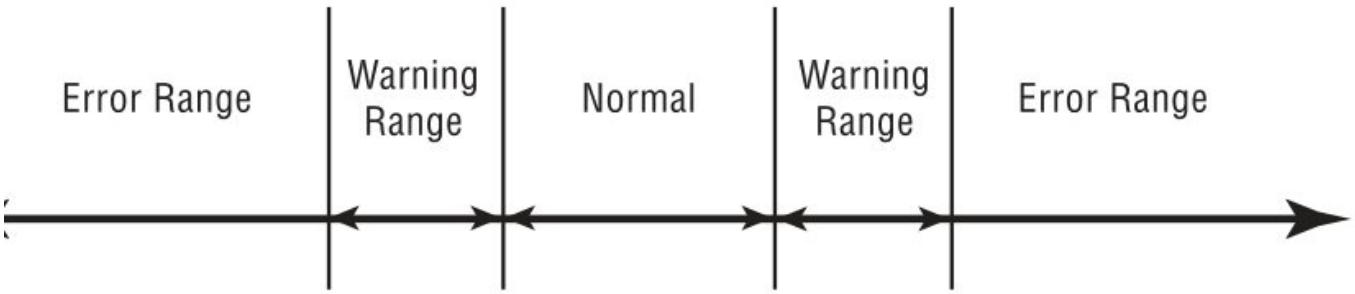




Figure II-1 — Warning/Error range checking for all inputs. Figure II-2 — Warning Range dialog. Figure II-3 — Error Range dialog.

Event-related Messages

While executing events, a number of conditions may occur that require a response by the user. These conditions are described below.

> **NOTE:** These messages may be produced by HVE or by the current calculation model. Messages produced by the calculation model may also appear in greater detail in the event's Messages output report during Playback.

Inter-process Communication

- **'Event' Died Unexpectedly!** — The current calculation model stopped running before it returned. The abnormal termination may be caused by a variety of conditions, but usually is the result of an untrapped calculation error (e.g., division by zero) or operating system error. After clearing the message dialog, check all running processes to confirm no unwanted processes are running.
- **Writing to message queue failed!** — A failure occurred in the communication (the passing of data) between HVE and the current calculation model (this communication occurs in a virtual device called the message queue). After clearing the message dialog, check the inter-process communication status to confirm the message queue is clear.

Event Verify

To execute an event, the user first must select the human(s) and vehicle(s) and a calculation model using the Event Information dialog. When OK is pressed, HVE verifies the compatibility between the selected objects and calculation model.

> **NOTE:** Actually, it is up to the calculation model to perform these diagnostics and return any errors to HVE.

If an incompatibility is discovered, a message dialog is displayed and the user must correct the problem before continuing. The Event Verify error messages are listed below.

General Incompatibility

- **Incompatible Object(s): Bad Data** — The human and/or vehicle data are incompatible with the calculation model (the least descriptive of all messages; normally one of the more detailed messages below is displayed instead).
- **Incompatible Object(s): Wrong Object Type** — The calculation model was expecting a different type of object (e.g., the user selected a human object and a simulator that expects vehicles).
- **Incompatible Object(s): Wrong Number Of Objects** — The user selected either too many or too few objects for the selected calculation model.
- **Incompatible Object(s): Number Of Axles Not Supported** — The vehicle has a different number of axles than allowed by the selected simulator.
- **Incompatible Object(s): Tandem Axles Not Supported** — The vehicle has tandem axles and the simulator does not allow tandem axles.
- **Incompatible Object(s): Dual Tires Not Supported** — The vehicle has dual tires and the simulator does not allow dual tires.

- **Incompatible Object(s): Bad Combination Vehicle Configuration** — The set of vehicles is not compatible with the selected calculation model (for example, two trailers without a tow vehicle). Remember that the selection order of the vehicles (as displayed in the Selected Objects list) implies the order assigned by the simulator: the first object will be the tow vehicle, next the first trailer, and so forth.
- **Incompatible Object(s): Suspension Type Not Supported** — The suspension type is not allowed by the calculation model (e.g., a commercial vehicle simulator may not allow independent suspensions).

Incompatible Humans

- **Bad or Incompatible Human Data** — The human data are incompatible with the calculation model (least descriptive message).
- **... Check Inertias** — The inertial properties of the selected human(s) are not compatible with the selected calculation model.
- **... Check Ellipsoids** — The contact ellipsoid properties are not compatible with the selected calculation model.
- **... Check Joints** — The joint properties are not compatible with the selected calculation model.
- **... Check Tolerances** — The injury tolerances are not compatible with the selected calculation model.

Incompatible Vehicles

- **Bad or Incompatible Vehicle Data** — The vehicle data are incompatible with the calculation model (least descriptive message).
- **... Check Dimensions** — Dimensional properties not compatible.
- **... Check Inertias** — Inertial properties not compatible.
- **... Check Stiffness Coefficients** — Stiffness coefficients not compatible.
- **... Check Contact Surfaces** — Contact surface properties not compatible.
- **... Check Belt Restraints** — Belt restraint system properties not compatible.
- **... Check Airbag Restraints** — Airbag restraint system properties not compatible.
- **... Check Inter-vehicle Connections** — The inter-vehicle connection properties (hitch, kingpin, etc.) for the selected vehicle(s) are not compatible with each other (for example, the tow vehicle has a ball and the trailer has a kingpin). It is also possible that one or more vehicles has no connection at all.
- **... Check Drag Coefficients** — Drag coefficients not compatible.
- **... Check Drivetrain** — Drivetrain properties not compatible.
- **... Check Engine** — Engine data not compatible.
- **... Check Transmission** — Transmission data not compatible.
- **... Check Differential** — Differential data not compatible.
- **... Check Tire Properties** — Tire data not compatible.
- **... Check Suspension Properties** — Suspension data not compatible.
- **... Check Brake Assembly Properties** — Wheel brake data not compatible.
- **... Check Vehicle Brake System** — Brake system (master cylinder) data not compatible.
- **... Check Vehicle Steer System** — Steer system data not compatible.

Incompatible Environment

- **Bad or Incompatible Environment Data** — The environment data are incompatible with the calculation model (least descriptive message).
- **... Check Wind Speed/Direction** — Wind speed or direction data not compatible.
- **... Check Gravity** — The gravitational constant is not compatible.
- **... Check Air Temperature/Pressure** — Ambient air temperature or pressure data not compatible.
- **... Check Road Surface Polygons** — The 3-D polygon database in the current environment is not compatible with the selected calculation model.

Incompatible Event

- **Bad or Incompatible Event Data** — The event set-up data are incompatible with the calculation model (least descriptive message).
- **... Check Position(s)** — The position(s) supplied for one or more objects are not compatible with the selected calculation model.
- **... Check Position — POC Lies On A Straight Line** — The point-on-curve position supplied for one or more vehicles lies on a straight line between separation and end-of-rotation or rest. This condition is not allowed because it will cause a singularity during the path definition calculations.
- **... Check [Initial, ..., Final/Rest] Position** — The indicated position(s) are not compatible with the selected calculation model.
- **... Check Position Angle** — The orientation supplied for one or more objects is not compatible.
- **... Check Position Limits** — The position supplied for one or more objects is beyond the allowable range.
- **... Check Velocity** — The velocity supplied for one or more objects is beyond the allowable range.
- **... Check Driver [Throttle, ..., Gear] Table** — The driver input table (throttle, brakes, gear selection or steering) supplied for one or more vehicles is not compatible with the selected calculation model.
- **... Check Damage Profile Data** — The damage profile supplied for one or more vehicles is not compatible.
- **... Check Damage — Invalid CDC** — The Collision Deformation Classification (CDC) supplied for one or more vehicles is not compatible.
- **... Check Damage — Invalid Crush Depths** — The crush depths supplied are not compatible.
- **... Check Damage — Invalid Damage Width** — The damage width supplied is not compatible.
- **... Check Damage — Invalid Damage Offset** — The damage offset supplied is not compatible.
- **... Check Damage Energy** — A failure occurred while calculating the damage energy for one or more vehicles (check damage-based results for damage energy).
- **... Check PDOFs** — The user-entered PDOFs are not opposing each other.
- **... Check Damage — Force Imbalance (Newton's 3rd Law)** — The Damage Analysis (based on user-entered damage profiles, PDOFs and stiffness coefficients) resulted in a significant difference in the calculated forces on each vehicle (according to Newton's 3rd Law, they should be equal).
- **... Check Collision Pulse** — The collision pulse supplied for one or more objects is not compatible.
- **... Check Contact Surfaces** — The contact surface data supplied for one or more vehicles are not compatible.
- **... Check Payload** — The Payload Data supplied are not compatible.
- **... Check Restraints — Invalid Belt Data** — The Belt Restraint Data supplied are not compatible.
- **... Check Restraints — Invalid Airbag Data** — The Airbag Data supplied are not compatible.

- **... Check Simulation Controls** — The Simulation Control Data supplied are not compatible.
- **... Check Program Options** — The Program Option Data supplied are not compatible.
- **... Check Human Ellipsoids — No [Head, Knee, Chest] Specified** — A contact ellipsoid required by the current calculation model was not supplied.
- **... Check Airbag — No Backside Contact Specified** — No backside contact surface was supplied for the current vehicle (airbag calculations require a vehicle contact surface behind the airbag).
- **... Check Vehicle — No Seat Contact Specified** — No contact surface named 'Seat' was supplied for the current vehicle (some occupant models require a 'Seat' contact surface).
- **... Belt not installed at selected seat position** — The vehicle does not have a belt restraint system installed at the selected occupant position. Belt restraints cannot be assigned.
- **... Airbag not installed at selected seat position** — The vehicle does not have an airbag restraint system installed at the selected occupant position. Airbag restraints cannot be assigned.

Incompatible Tires

- **Bad or Incompatible Tire Data** — The tire data are incompatible with the calculation model (least descriptive message).
- **... Invalid Tire Size Specification** — The tire size specification (e.g., P225/75R14) supplied for one or more vehicles is not compatible.
- **... Invalid Tire Dimensions** — The tire dimensions supplied are not compatible.
- **... Tire Type Not Supported** — The tire type supplied is not compatible with the selected calculation model.

Event Execution Messages

The following normal-completion messages do not indicate an error:

- **Done** — The current calculation model ran to completion without error.
- **Simulation interrupted by user** — The user paused the simulation before it reached a termination condition.
- **Termination Velocity Reached** — All the humans and/or vehicles reached the termination velocity (i.e., they stopped moving).
- **Maximum simulation time reached** — The current simulation time reached the maximum simulation time.

Event termination and warning messages:

- **Event Termination: Wheel Lift-off** — During execution, the vertical force at one or more wheels became zero or negative (some simulations do not allow this condition).
- **Event Termination: Excessive Roll or Pitch** — The vehicle roll or pitch angle exceeded the allowable value.
- **Event Termination: Excessive Surface Slope** — A wheel traveled on a surface with an excessive slope.
- **Event Termination: Integration Error** — The integration routine failed; the integration timestep was halved more than the allowable number of times. This usually indicates an instability in the numerical integration — either a very dynamic system or errant derivative calculations.
- **Event Termination: Integration Timesteps Too Large For 3-D Simulation** — The current integration timestep is too large for a 3-D simulator. Select the Simulation Controls dialog and reduce the integration timestep(s).
- **Event Termination: Excessive Longitudinal Tire Slip** — The current level of longitudinal tire slip is too large for a linear tire model.

- **Event Termination: Excessive Lateral Tire Slip** — The current level of lateral tire slip is too large for a linear tire model.
- **Event Termination: Radial Tire Model Error** — The radial tire model failed.
- **Event Termination: Tire Model Integration Error** — The tire model failed due to an integration failure.
- **Event Termination: Tire is on a bad surface** — The surface beneath the tire contact patch has missing or defective information.
- **Event Termination: Entered initial position is illegal** — The initial position is not allowed, probably because the initial orientation angle is excessive.
- **Event Termination: Engine RPM exceeded maximum table value (threw a rod!)** — The current engine RPM is higher than the RPM range for the vehicle engine model. This normally occurs because of a combination of high initial velocity and low gear entered in the transmission table.
- **Warning: Inter-vehicle forces Violate Newton's 3rd Law** — The calculated collision force on each vehicle is significantly different (they should be equal, ignoring tire and aerodynamic forces).
- **Event Termination: Zero Length Collision Vector** — During the collision phase, the calculation model calculated a zero (or negative) length collision vector. This suggests excessive crush depth; the collision severity may be beyond the scope of the collision model.
- **Event Termination: Invalid Collision Vector Angle** — The angle between collision vectors was exactly 90 degrees; this condition is not allowed.
- **Event Termination: Too Many Collision Vectors** — The number of collision vectors required to span the damage range exceeded the allowable number.
- **Event Termination: Too Many Collision Vector Length Adjustments** — The calculation model adjusted the collision vectors too many times, suggesting the collision phase calculations are unstable. Try reducing the overlap at impact or the collision phase integration timestep.
- **Warning: Too Many Damage Ranges (Remaining Are Ignored)** — Too many damage peaks were recorded during the collision phase (sometimes occurs during sustained impact). The simulation results are valid, but tabulations based on accumulated damage arrays may not include the latter peaks.
- **Event Termination: Collision Vector Length Negative After Restitution** — The restitution calculations resulted in a negative vector length. This is a fatal error; it might be eliminated by reducing the integration timestep or adjustment length per timestep.
- **Warning: Unmatched Damage Range** — The collision phase simulation was unable to match the PDOF with a particular damage profile (for vehicles with multiple damage zones). Informative only; results are not affected.
- **Event Termination: Common Velocity Error** — The separation velocity results violated the common velocity assumption (inherent to damage-based delta-V calculations). Usually considered a fatal error. See also Common Velocity Warning.
- **Warning: Common Velocity Adjustment Performed** — The separation velocities were adjusted to be consistent with the common velocity assumption. See also Common Velocity Error.
- **Event Termination: Spinout Error** — The calculated separation velocities were inconsistent with the laws of physics. Displayed by reconstructions that use the modified Marquard method for calculating separation velocities; usually indicates excessive wheel lock-ups or too much spin over a short path length.
- **Event Termination: Excessive Joint Articulation Angle** — The current joint articulation angle exceeds the maximum angle; normally indicates errant initial joint angles.

- **Event Termination: Too Many Human Ellipsoids** — The number of ellipsoids exceeds the number allowed by the simulation model; usually solved by removing one or more contact ellipsoids from the selected human.
- **Event Termination: Too Many Vehicle Contact Surfaces** — The number of contact surfaces exceeds the number allowed; usually solved by removing one or more contact surfaces from the selected vehicle.
- **Event Termination: Bad Human vs Vehicle Contact Force** — The simulation model calculated an excessive contact force; normally indicates a deeper problem in the simulation model calculations.
- **Event Termination: Excessive Inter-vehicle Articulation Angle** — The articulation angle between a tow vehicle and trailer exceeded the allowable value specified for the tow vehicle. This usually indicates a jackknife condition.
- **Event Termination: Error in Derivative Calculations** — An error occurred while calculating the derivatives (accelerations) for the current timestep; normally indicates a deeper problem in the simulation model calculations.
- **Event Termination: Singular Matrix (Bad News!)** — This message should never occur for a well-behaved dynamic system; it indicates a fundamental problem in the simulation model.
- **Event Termination: Collinear Collision — Can't Use Momentum** — A collinear collision: attempting to use linear momentum as a final value problem (separation velocities known, impact velocities to be calculated) results in a mathematical singularity (division by zero).
- **Event Termination: No [Human, Vehicle] Position(s) Assigned!** — One or more required positions were not assigned; enter the required positions.
- **Event Termination: No [Human, Vehicle] Velocity(ies) Assigned** — One or more required velocities were not assigned; enter the required velocities.

Output Report Errors

All of the following indicate a missing or defective physics resource file (the resource file contains the language-specific text for output reports produced by the current calculation model):

- **Output Report Error: Trouble parsing physics' resource reference filename**
- **Output Report Error: Bad keyword in physics' resource reference file**
- **Output Report Error: Cannot open physics resource reference file**
- **Output Report Error: Empty physics resource reference file**
- **Output Report Error: HVEFILES not available to physics process** — The resource file could not be found because the `HVEFILES` environment variable is missing or defective.
- **Output Report Error: Bad resource reference definition string** — The resource file contains a defective string.
- **Output Report Error: Unexpected EOF in physics resource reference file** — The resource file is defective.

Appendix III — Coordinate Systems

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix III, pages III-1 through III-8.

Coordinate systems are used to define the locations of objects, such as humans and vehicles. HVE uses the following coordinate systems:

- **Earth-fixed** — Used to define the location of objects relative to the earth
- **Vehicle-fixed** — Used to define the location of objects relative to a vehicle
- **Tire-fixed** — Used to define the position and orientation of a tire
- **Human-fixed** — Used to define the location of objects relative to a human

These coordinate systems are described in detail below.

Earth-fixed

The earth-fixed coordinate system may be thought of as a large coordinate system attached to the surface of the earth. It is never moved. It locates objects (vehicles and human pedestrians) on or above the earth's surface.

The origin of the earth-fixed coordinate system may be placed anywhere, and the Z axis points down, towards the center of the earth (in the direction of the gravity vector). The X-Y plane is normal to the Z axis; the X axis may point in any direction in this plane and the Y axis points to the right (i.e., clockwise 90 degrees) from X.

All angles are defined relative to the X, Y, Z axes. The heading angle (Ψ) is defined about the Z axis; roll (Φ) and pitch (Θ) are about the X and Y axes, respectively.

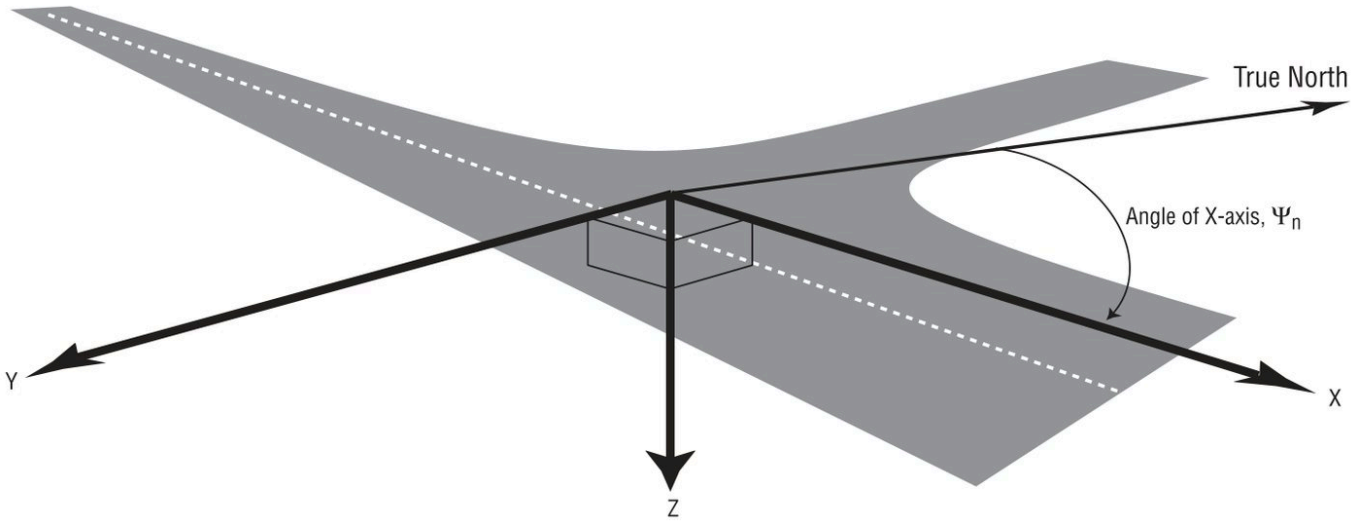


Figure III-1 — Earth-fixed coordinate system. Z is in the direction of gravity (down); X and Y are orthogonal to each other, as well as to Z.

Vehicle and pedestrian path coordinates and heading angles are defined relative to the earth-fixed coordinate system.

Vehicle-fixed

The vehicle-fixed coordinate system may be thought of as a smaller coordinate system attached firmly to the vehicle; it never moves with respect to the vehicle, but moves along with the vehicle with respect to the earth-fixed coordinate system.

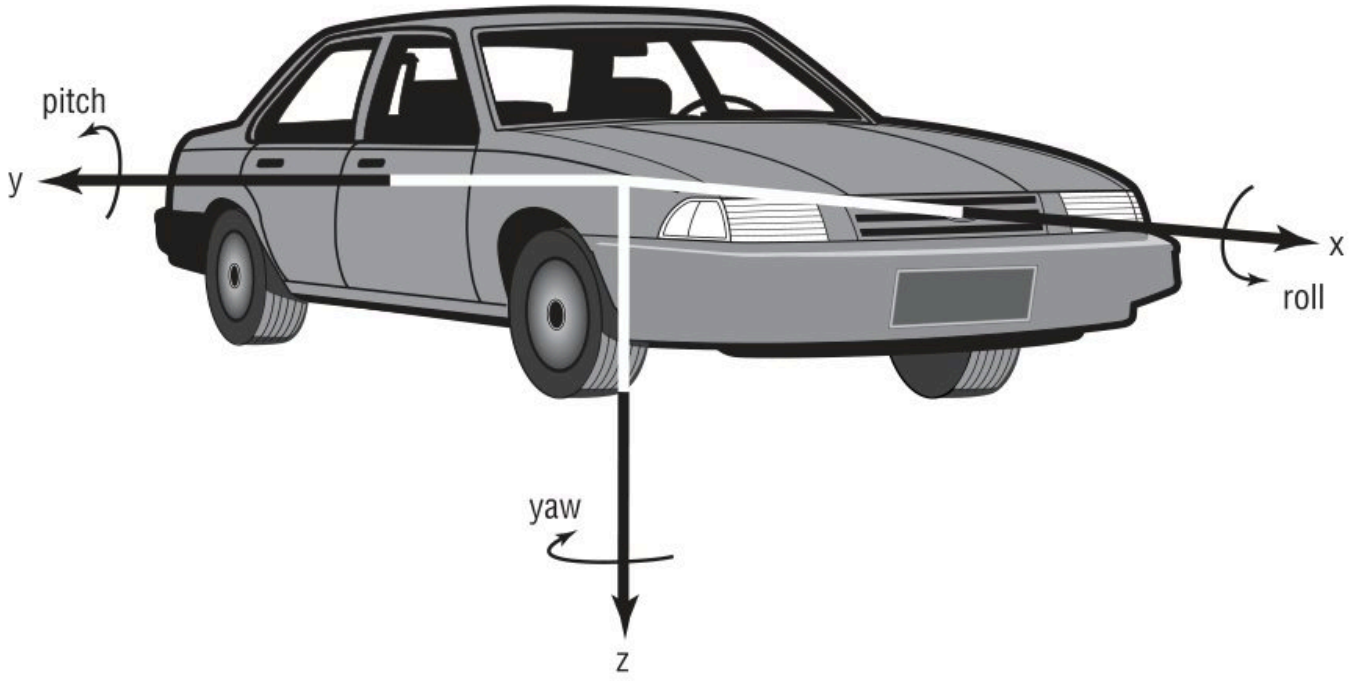


Figure III-2 — Vehicle-fixed coordinate system.

The orientation of the vehicle-fixed coordinate system is not arbitrary. Its origin is at the vehicle's center of gravity (CG). The x axis points towards the front of the vehicle, and defines the vehicle's heading angle with respect to the earth-fixed coordinate system. The y axis points to the right, and the z axis points down.

(The original printing said "the y axis points towards the front"; per SAE J670e and Figure III-2, the longitudinal forward axis is x.)

> **NOTE:** Consistent with SAE J670e, the earth-fixed axes are denoted by upper-case characters (X, Y, Z), while the vehicle-fixed axes are denoted by lower-case characters (x, y, z).

All vehicle-related data, such as wheel locations, steer angles and sideslip angles, are defined relative to the vehicle-fixed coordinate system.

Articulated Vehicles

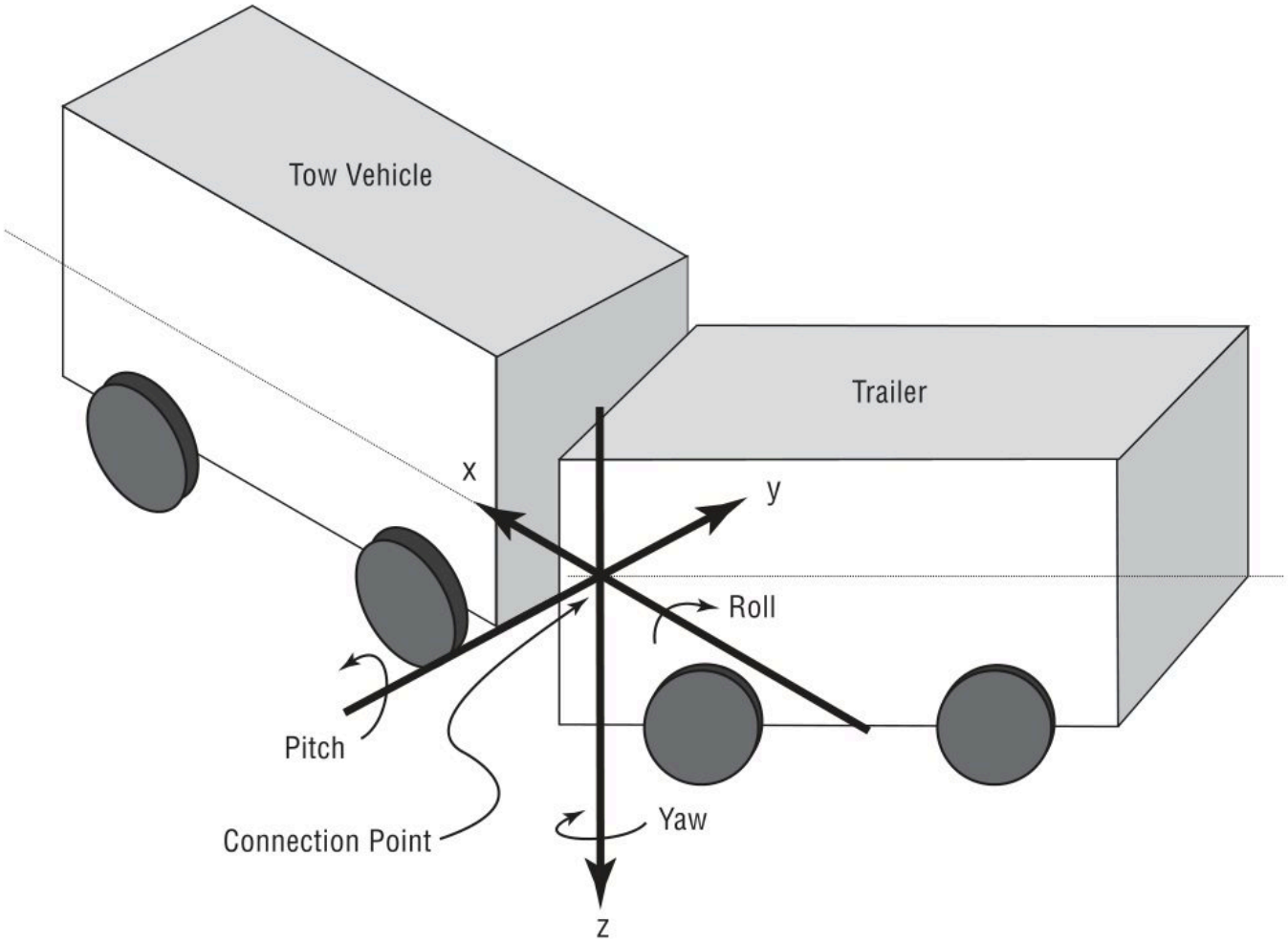


Figure III-3 — Articulated vehicles. Roll, pitch and yaw articulation angles are defined about the vehicle-fixed x, y and z axes, respectively.

Articulated vehicles (i.e., vehicle trailers and dollies) are positioned using the vehicle-fixed coordinate system, using their roll, pitch and yaw articulation angles with respect to the towing vehicle.

Tire-fixed

The tire axis system is attached to the vehicle and defines the orientation of a tire. The origin of the tire axis system is the center of the tire contact patch.

The X' axis is the intersection of the wheel plane and the road surface plane, with the positive direction pointing forward. The Z' axis is normal to the X' axis with the positive direction downward, normal to the road surface plane. The Y' axis is in the road surface plane, its direction chosen to be orthogonal to X' and Z'.

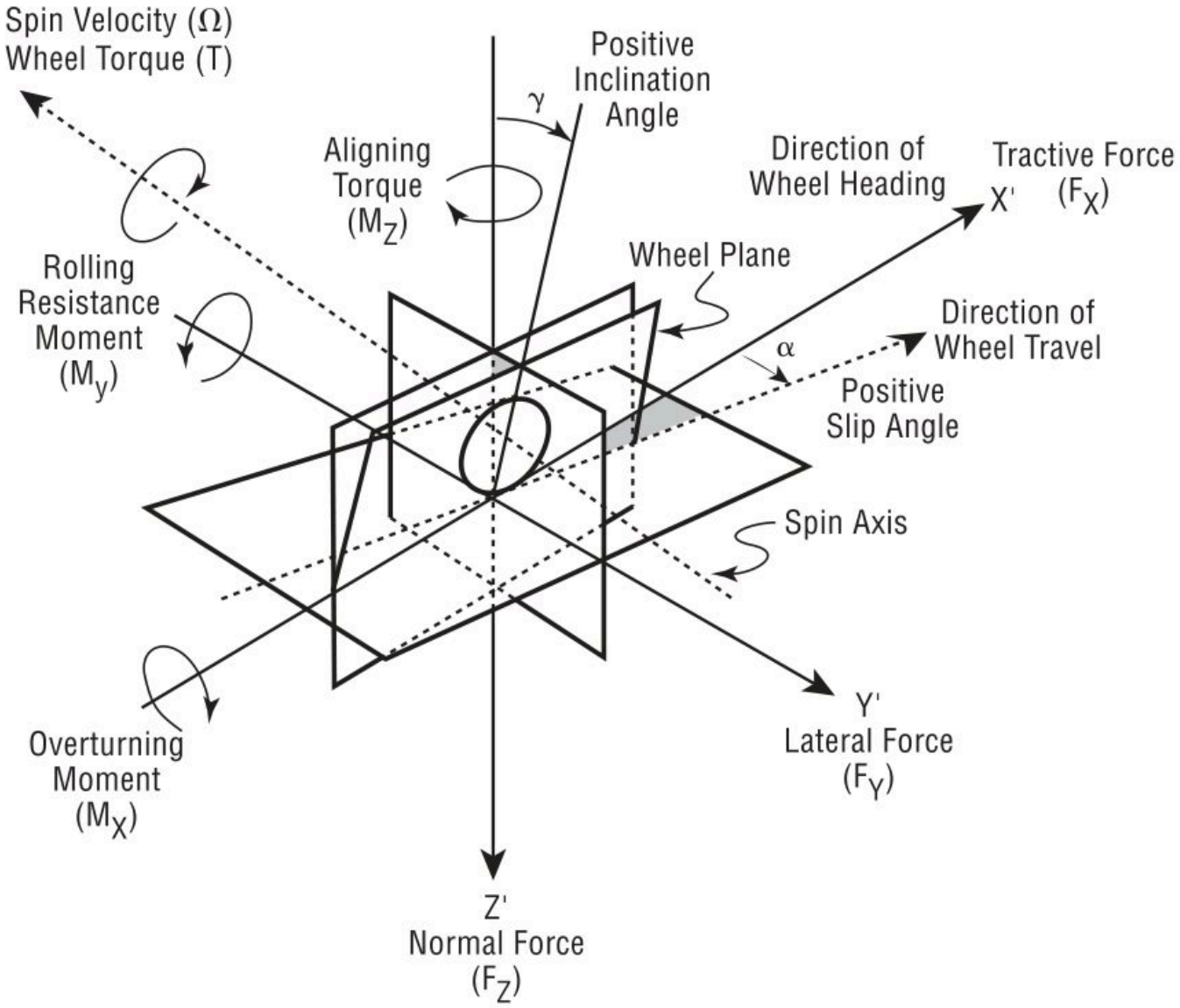


Figure III-4 — Tire Coordinate Axis System [6.2].

Human-fixed

The human-fixed coordinate system may be thought of as a smaller coordinate system attached firmly to the human; it never moves with respect to the human, but moves along with the human with respect to the earth-fixed coordinate system.

The human model is composed of 15 segments and 14 joints. Each segment has an i, j, k coordinate axis system with its origin at the segment CG. Joints and contact ellipsoids are positioned relative to the i, j, k coordinate axis system. Roll, pitch and yaw rotations are defined about the i, j and k segment axes, respectively.

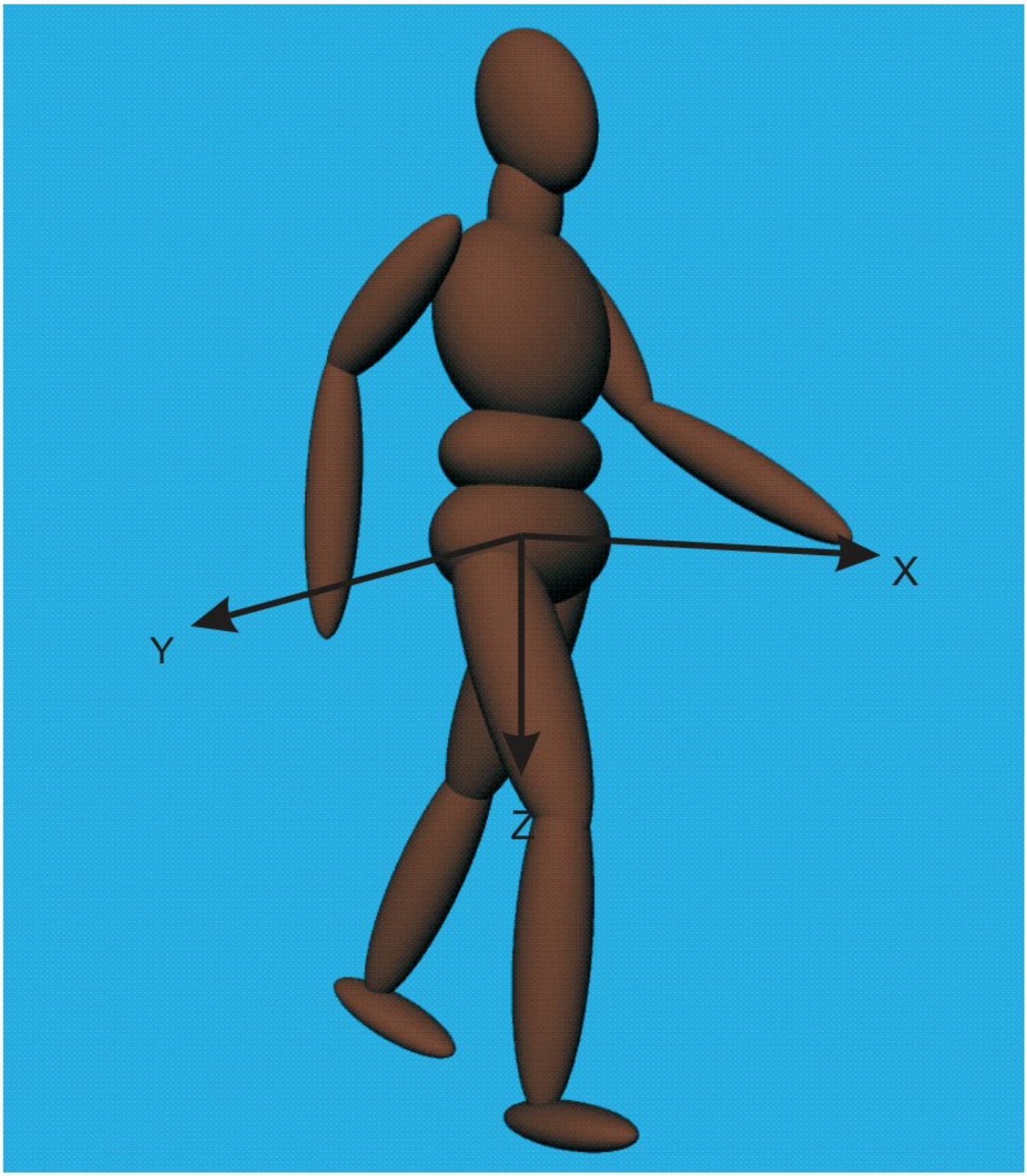


Figure III-5 — Human-fixed coordinate system.

Human positions are defined relative to either the earth-fixed (for human pedestrians) or vehicle-fixed (for human occupants) coordinate systems. The origin of the human-fixed coordinate system is the center of gravity (CG) of the pelvis segment. Its forward (x), lateral (y) and vertical (z) directions are coincident with the i, j, k axes for the pelvis segment (see Figure III-5). Thus, when you position a human relative to either the earth or vehicle, you are really

positioning the pelvis. The position and orientation of the other 14 segments are defined by their yaw, pitch and roll articulation angles with respect to adjacent upstream segments (i.e., the articulation angles for the upper leg are relative to the pelvis, the articulation angles for the lower leg are relative to the upper leg, and so on).

Appendix IV — Units

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix IV, pages IV-1 through IV-14. Several obviously mis-copied descriptions in the original Environment Units table have been corrected; corrections are noted.

HVE allows the user to define the current system of units by selecting Units from the Options menu (see Menu Reference). Every input and output variable in HVE has a unit name. For example, `UtVehForce` is the unit name for force applied to the vehicle and `UtHumForce` is the unit name for force applied to humans. Inside HVE, the program units for force will always be lb. However, if desired, they can be assigned different user units.

> **NOTE:** *Program Units* are the units stored and used inside HVE. Program units are always pounds, seconds and inches (and their derivatives). However, the user sees all input and output values in *User Units*. User units are completely programmable by the user.

In general, HVE allows the user to assign units to the following object groups:

- **General** — General-purpose units
- **Brake System** — Units related to the brake system
- **Engine** — Units related to the engine
- **Environment** — Units related to the environment
- **Human** — Units related to humans
- **Steering System** — Units related to the steering system
- **Suspension** — Units related to the suspension
- **Tire** — Units related to the vehicle's tires
- **Vehicle** — Units related to vehicles

This appendix describes how to assign units to any parameter.

Units Files

HVE has two units files, `units.us` and `units.si`. These files contain the unit names (e.g., `UtHumDispAngle`) and current value (e.g., `deg`). A file named `language.rsc` contains the associated program unit (e.g., `rad`) and conversion factor (e.g., 57.29577951).

To assign a unit to any parameter, perform the following steps:

1. Identify the unit name. The parameter tables elsewhere in this manual give each parameter's units; the unit names used in the units files are listed by group in Tables IV-1 through IV-11 of this appendix.
2. Identify the Program Units for the selected parameter. The Program Units for each parameter group are listed in this appendix (see Tables IV-1 through IV-11).
3. Modify the `units.us` or `units.si` file by editing the desired unit. For example, to change vehicle velocity from mph to ft/sec, edit `units.us`: find `UtVehVelLinear` and simply change the corresponding string from `mph` to `ft/sec`.
4. Confirm the conversion factor exists in the `language.rsc` file. In this case, the following lines may be found in the `language.rsc` file:

```
Convert    in/sec    in/sec    1.0
Convert    in/sec    ft/sec    0.083333333
Convert    in/sec    mph       0.056818182
Convert    in/sec    km/h      0.091440000
Convert    in/sec    cm/sec    2.540000000
Convert    in/sec    m/sec     0.025400000
```

> **NOTE:** If `ft/sec` were not already in the `language.rsc` file, it would need to be added before HVE could fulfill the user's request.

The remainder of this appendix lists the individual unit names and program units for each group.

General Purpose Units

Table IV-1 — General-purpose Units

Unit Name	Program Units	Description
UtNone	N/A	Used for dimensionless parameters
UtTime	sec	Simulation time units (see also UtHumTime, UtVehTime, UtSteTime, UtEnvTime, UtBraTime, UtEngTime)
UtFrequency	sec^{-1}	Inverse of the period of oscillation for harmonic motion
UtVelAngular	rad/sec	Angular velocity units (see also UtHumVelAngular, UtVehVelAngular, UtEnvVelAngular, UtEngSpeed, UtSusVelAngular, UtTirVelAngular)
UtVelLinear	in/sec	Linear velocity units (see also UtHumVelLinear, UtVehVelLinear, UtEnvVelLinear, UtSusVelLinear, UtTirVelLinear)

Brake System Units

Table IV-2 — Brake System Units

Unit Name	Program Units	Description
UtBraForce	lb	Units of force used for brake system parameters
UtBraExpansionCoef	in/degF	Coefficient for increasing stroke in S-cam brake assemblies due to thermal expansion
UtBraPedalRatio	lb/in²-lb	Ratio of brake system pressure to brake pedal force
UtBraPercent	%/100	Percent units used for brake system parameters
UtBraPress	lb/in²	Units of pressure
UtBraTime	sec	Brake system time
UtBraTorque	in-lb	Brake system torque
UtBraTorqueRatio	in-lb-in²/lb	Brake system torque ratio
UtBraPressRate	lb/in²-sec	Rate of change in wheel brake pressure

Engine Units

Table IV-3 — Engine Units

Unit Name	Program Units	Description
UtEngForce	lb	Units of force used for engine parameters
UtEngPercent	%/100	Percent units used for engine parameters
UtEngPower	in-lb/sec	Units of power (rate of work) for engine parameters
UtEngSpeed	rad/sec	Units of rotational velocity for engine parameters
UtEngTime	sec	Engine time
UtEngTorque	in-lb	Engine torque

Environment Units

Table IV-4 — Environment Units *(several descriptions in the original table were mis-copied from the Engine table; they are corrected here)*

Unit Name	Program Units	Description
UtEnvTime	sec	Units of time used for environment parameters
UtEnvCameraFocalLength	in	Camera focal length units
UtEnvDensity	lb/in³	Weight per unit volume
UtEnvDispLength	in	Linear displacement units for the environment
UtEnvDispAngle	rad	Angular displacement units for the environment
UtEnvEnergy	in-lb	Energy units for the environment
UtEnvForce	lb	Force units for the environment
UtEnvGravity	in/sec²	Gravitational acceleration units
UtEnvKc	lb/in^(N+2)	Cohesive soil constant for sinkage
UtEnvKphi	lb/in^(N+1)	Frictional soil constant for sinkage
UtEnvPercent	%/100	Percent units for the environment
UtEnvPress	lb/in²	Units of pressure for the environment
UtEnvShear	lb/in²	Units of shear stress for the environment
UtEnvTemp	Fahrenheit	Units of temperature for the environment
UtEnvTorque	in-lb	Units of torque for the environment
UtEnvVelAngular	rad/sec	Units of angular velocity for objects in the environment
UtEnvVelLinear	in/sec	Units of linear velocity for objects in the environment
UtEnvVolume	in³	Units of environment volume

Human Units

Table IV-5 — Human Units

Unit Name	Program Units	Description
UtHumAccelAngular	rad/sec²	Angular acceleration for humans
UtHumAccelLinear	in/sec²	Linear acceleration for humans
UtHumChestSI	in/sec²	Chest Severity Index
UtHumDamp	in-lb-sec/rad	Damping for humans
UtHumDispAngle	rad	Angular displacements for humans
UtHumDispLength	in	Linear distances for humans
UtHumElastLinear	in-lb/rad	Linear elastic torque
UtHumElastQuad	in-lb/rad²	Second-order elastic torque
UtHumElastCubic	in-lb/rad³	Third-order elastic torque
UtHumEnergy	in-lb	Energy units for humans
UtHumForce	lb	Force units for humans
UtHumInertia	lb-sec²-in	Rotational inertia units for humans
UtHumMass	lb-sec²/in	Mass units for humans
UtHumPercent	%/100	Units of percentage for humans
UtHumTime	sec	Time units for humans
UtHumTorque	in-lb	Torque units for humans
UtHumVelAngular	rad/sec	Angular velocity units for humans
UtHumVelLinear	in/sec	Linear velocity units for humans

Steering System Units

Table IV-6 — Steering System Units

Unit Name	Program Units	Description
UtSteDispAngle	rad	Steering system angular displacement units
UtSteTime	sec	Time units for steering
UtSteVel	rad/sec	Steer velocity units
UtSteVelRate	rad/in	Steer correction rate units
UtSteVelDamp	rad-sec/in	Steer correction damping units
UtSteTorque	in-lb	Steer torque units
UtSteTorqueRate	in-lb/in	Steer torque correction rate units
UtSteTorqueDamp	in-lb-sec/in	Steer torque correction damping units

Suspension System Units

Table IV-7 — Suspension System Units

Unit Name	Program Units	Description
UtSusAccelAngular	rad/sec²	Angular acceleration units for suspension
UtSusAccelLinear	in/sec²	Linear acceleration units for suspension
UtSusAntiPitch	lb/in-lb	Anti-pitch units for suspension
UtSusDispAngle	rad	Angular displacement units for suspension
UtSusDispLength	in	Linear displacement units for suspension
UtSusRateLinear	lb/in	Units for suspension linear rate
UtSusRateQuadratic	lb/in²	Units for suspension second-order rate
UtSusRateCubic	lb/in³	Units for suspension third-order rate
UtSusDamp	lb-sec/in	Suspension damping units
UtSusPercent	%/100	Units of percentage for suspension
UtSusRollConst	rad	Roll constant units for suspension
UtSusRollLinear	rad/in	Linear roll rate units for suspension
UtSusRollQuad	rad/in²	Second-order roll rate units for suspension
UtSusRollCubic	rad/in³	Third-order roll rate units for suspension
UtSusRollStiff	in-lb/rad	Roll stiffness units for suspension
UtSusVelAngular	rad/sec	Suspension angular velocity units
UtSusVelLinear	in/sec	Suspension linear velocity units

Tire Units

Table IV-8 — Tire Units

Unit Name	Program Units	Description
UtTirAccelAngular	rad/sec²	Tire angular acceleration units
UtTirAccelLinear	in/sec²	Tire linear acceleration units
UtTirAlignTorque	in-lb/rad	Tire aligning torque units
UtTirCalfa	lb/rad	Tire cornering stiffness units
UtTirCgamma	lb/rad	Tire camber stiffness units
UtTirDispAngle	rad	Tire angular displacement units
UtTirDispLength	in	Tire linear distance units
UtTirForce	lb	Tire force units
UtTirInertia	lb-sec²-in	Tire rotational inertia units
UtTirLongSlip	%/100	Tire longitudinal slip units
UtTirLongStiff	1/lb	Tire longitudinal stiffness units
UtTirPercent	%/100	Tire percentage units
UtTirRateLinear	lb/in	Tire linear rate units
UtTirRolloff	%/100	Tire rolloff units
UtTirTorque	in-lb	Tire torque units
UtTirVelAngular	rad/sec	Tire angular velocity units
UtTirVelDependence	sec/in	Tire velocity dependence units for friction
UtTirVelLinear	in/sec	Tire linear velocity units

Vehicle Units

Table IV-9 — Vehicle Units

Unit Name	Program Units	Description
UtVehAccelAngular	rad/sec²	Vehicle angular acceleration units
UtVehAccelLinear	in/sec²	Vehicle linear acceleration units
UtVehArea	in²	Vehicle area units
UtVehConduction	in-lb/in-sec-degF	Vehicle material conduction units
UtVehConvection	in-lb/in²-sec-degF	Vehicle static convection units
UtVehConvectionVel	in-lb-sec/in²-sec-degF-in	Vehicle velocity-dependent convection units
UtVehDensity	lb/in³	Vehicle density units
UtVehDispAngle	rad	Vehicle angle units
UtVehDispLength	in	Vehicle length units
UtVehDragConst	lb	Vehicle drag constant units
UtVehDragLinear	1/sec	Vehicle velocity-dependent drag units
UtVehDragQuad	in²	Vehicle second-order velocity-dependent drag units
UtVehEnergy	in-lb	Vehicle energy units
UtVehForce	lb	Vehicle force units
UtVehInertia	lb-sec²-in	Vehicle rotational inertia units
UtVehInvLength	1/in	Vehicle inverse length units
UtVehInvLength2	1/in²	Vehicle inverse length-squared units
UtVehLightBright	cd	Vehicle light power units
UtVehMass	lb-sec²/in	Vehicle mass units
UtVehMassRate	lb/sec	Vehicle mass rate units
UtVehMatStiffLinear	lb/in	Vehicle material linear stiffness units
UtVehMatStiffQuad	lb/in²	Vehicle material second-order stiffness units
UtVehMatStiffCubic	lb/in³	Vehicle material third-order stiffness units
UtVehMatDamp	lb-sec/in	Vehicle material damping units
UtVehPercent	%/100	Vehicle percentage units
UtVehPress	lb/in²	Vehicle pressure units
UtVehRelativeVelLinear	in/sec	Vehicle relative velocity units
UtVehSpecHeat	in-lb/lbm-°R	Vehicle specific heat units
UtVehSteeringRatio	rad/rad	Vehicle steering ratio units
UtVehSteeringStiffness	in-lb/rad	Vehicle steering torsional stiffness units
UtVehStiffnessAstf	lb/in	Vehicle A stiffness units
UtVehStiffnessBstf	lb/in²	Vehicle B stiffness units

Unit Name	Program Units	Description
UtVehStiffnessKstf	lb/in²	Vehicle Kv stiffness units
UtVehTime	sec	Vehicle time units
UtVehTorque	in-lb	Vehicle torque units
UtVehVelAngular	rad/sec	Vehicle angular velocity units
UtVehVelLinear	in/sec	Vehicle linear velocity units
UtVehVolume	in³	Vehicle volume units

Combined Material Units

Table IV-10 — Combined Material Units

Unit Name	Program Units	Description
UtComForce	lb	Combined material force units
UtComMatLinear	lb/in	Combined material linear stiffness units
UtComMatQuad	lb/in²	Combined material second-order stiffness units
UtComMatCubic	lb/in³	Combined material third-order stiffness units
UtComMatDamp	lb-sec/in	Combined material damping units
UtComDispLength	in	Combined material length units

Miscellaneous Units

Table IV-11 — Miscellaneous Units

Unit Name	Program Units	Description
UtHardwareLength	in	Length units for hardware (e.g., printer paper dimensions)

Appendix V — Collision Deformation Classification (CDC)

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix V, pages V-1 through V-8.

The Collision Deformation Classification (CDC) is a seven-character code which describes the vehicle damage. The characters in the CDC identify the Principal Direction of Force (PDOF) in clock direction, the impacted surface (front, right, back, or left), the shape of the damage profile, and the extent of maximum penetration. If a more accurate direction than the clock direction is known, the PDOF may be entered in degrees after the CDC.

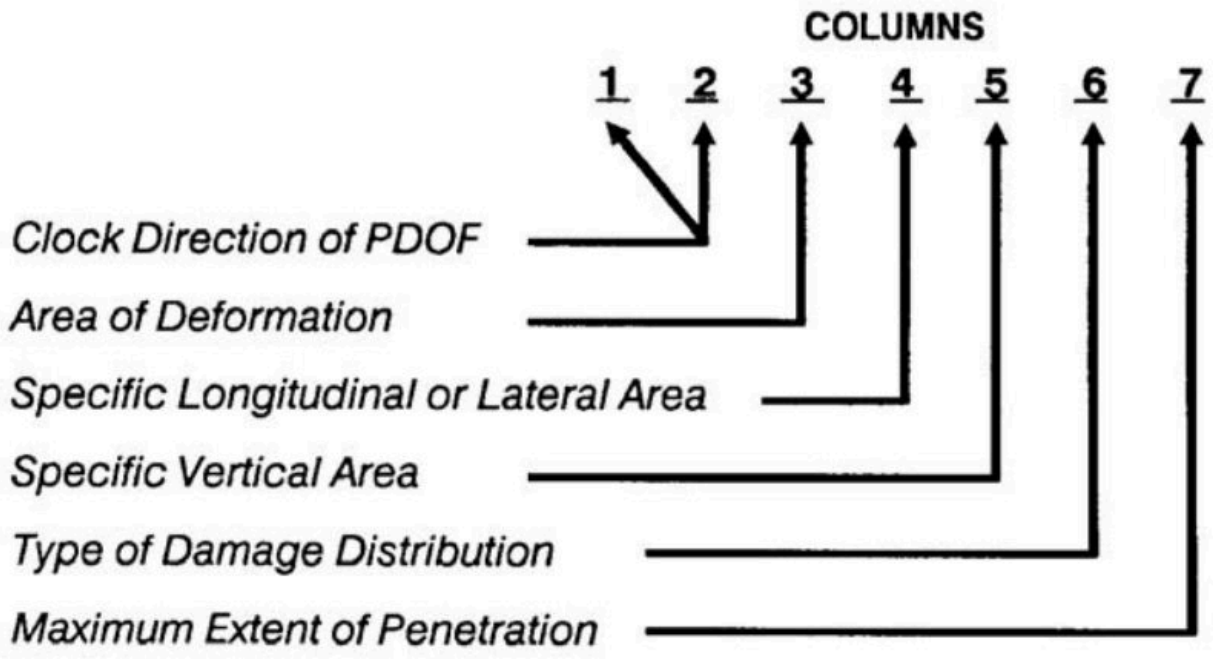


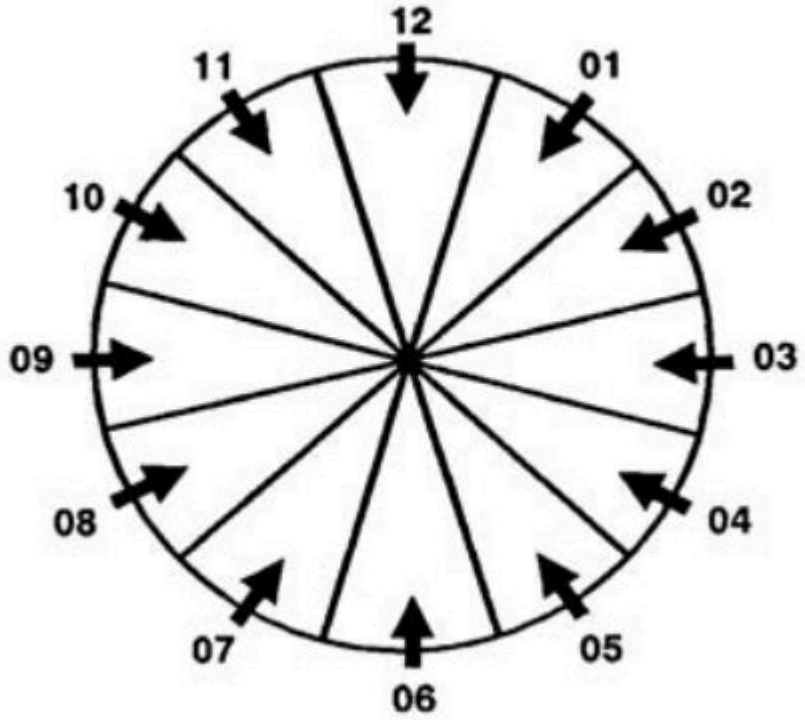
Figure V-1 — Description of each column within the seven-character CDC code.

Columns 1 and 2 — Principal Direction of Force

As defined in the first two columns of the CDC, the Principal Direction of Force (PDOF) during impact is the direction of the force that caused the crush and sheet metal displacement on the damaged vehicle. The PDOF is determined by the vector result of forces normal and tangential to the surface of the vehicle in the area of deformation. In columns 1 and 2 of the CDC, the PDOF is designated by hour sectors on a conventional clock face and referred to as the

clock direction

. 12-o'clock indicates a head-on impact, 06-o'clock refers to a rear-end impact, 03- and 09-o'clock refer to perpendicular impacts to the right and left sides, respectively. Angles in between are noted accordingly.


12
01
02
03
04
05
06
07
08
09
10
11

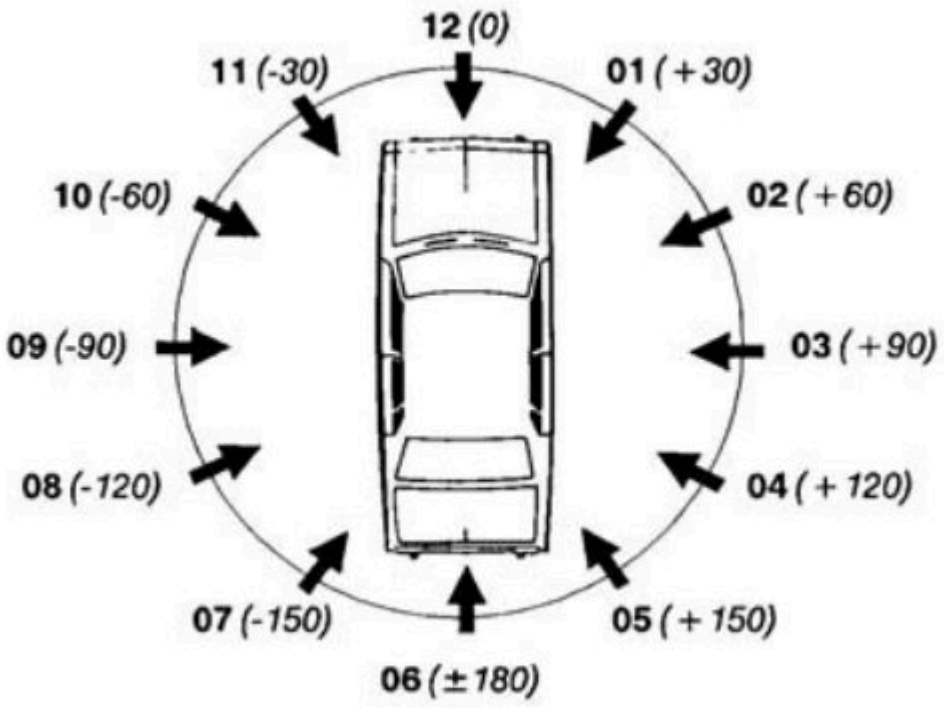
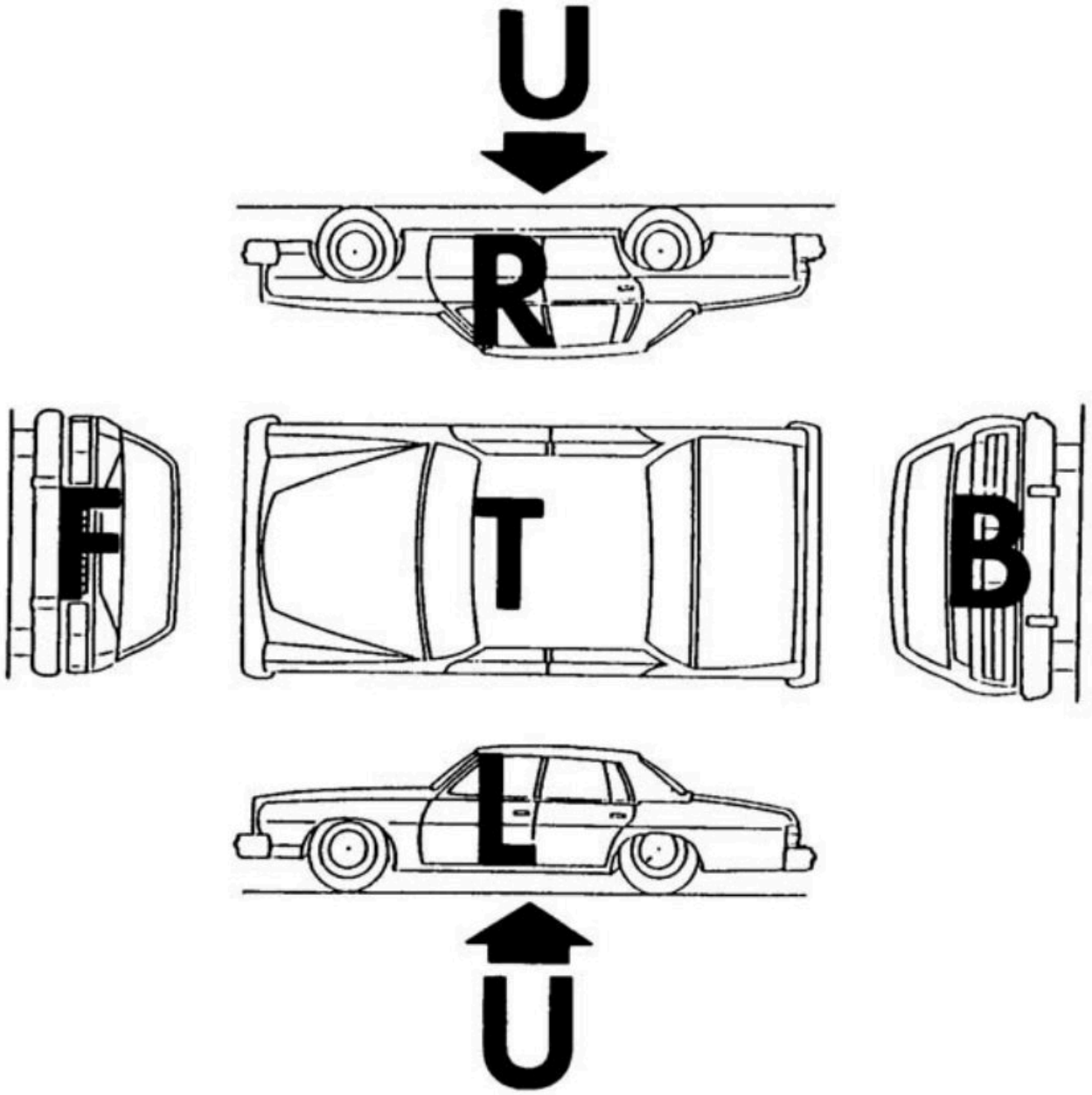


Figure V-2 — Columns 1 and 2, Clock Direction of PDOF. Figure V-3 — Relationship between hour angles and PDOF (degrees).

Note the clock direction is a two-digit code; that is, six-o'clock must be entered as 06, not 6. Note also that, by virtue of the hour increments, the clock direction in columns 1 and 2 is rounded to the nearest 30 degrees.

Column 3 — Deformation Location Code

The third column of the CDC, the Deformation Location Code, specifies the general area of the vehicle which sustained damage: the Front (F), Right (R), Back (B), or Left (L) side. The Top (T) and Undercarriage (U) of the vehicle may also be damaged; however, due to the 2-dimensional restrictions imposed by analysis in 3 degrees of freedom, T and U are not valid CDC entries for HVE.

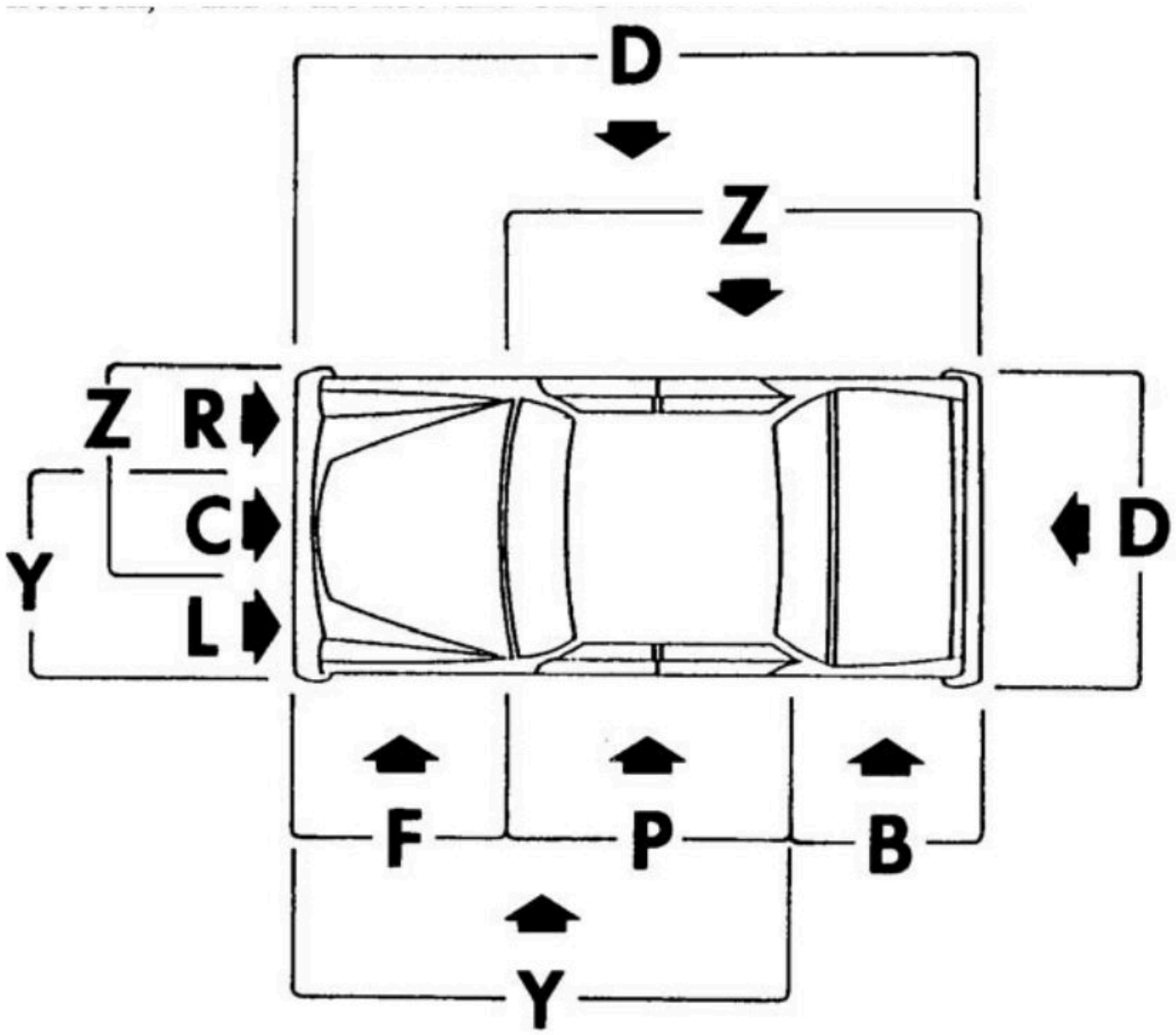


Location	Classification
Front	F
Right Side	R
Back (Rear)	B
Left Side	L
Top	T
Undercarriage	U
Unclassifiable	X

Figure V-4 — Column 3, Deformation Location Code.

Column 4 — Specific Longitudinal or Lateral Location of Deformation

The specific area of damage (Distributed, Left, Center, Right, etc.) can be considered a suffix to column three, in that the Specific Longitudinal or Lateral Location of Deformation describes, in greater detail, the general area of damage (front, right, back or left) entered in column 3. Letters shown on the front and rear may be applied to either the front or rear; letters shown on the sides may be used for either side.

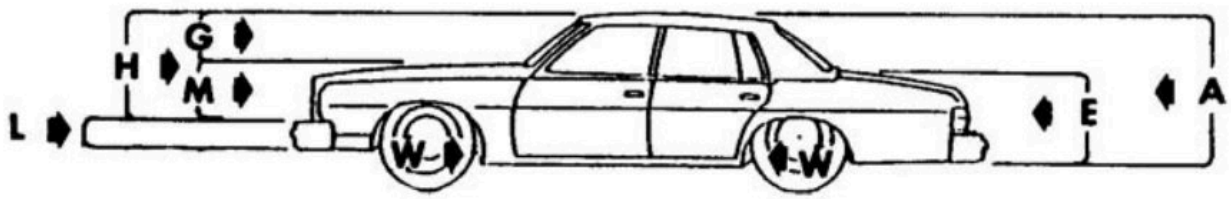


Location	Classification
Distributed—side or end	D
Left—front or rear	L
Center—front or rear	C
Right—front or rear	R
Side front—left or right	F
Side center section—left or right	P
Side rear—left or right	B
Side or end—F + P or L + C	Y
Side or end—B + P or R + C	Z

Figure V-5 — Column 4, Longitudinal or Lateral Location Code.

Column 5 — Specific Vertical Location of Deformation

This entry is also a suffix to column three and refers to vertical damage characteristics. An entry (A, H, E, G, M, L, or W) is necessary for a valid CDC. However, again due to the 2-dimensional analysis, column 5 is not used for any calculations.



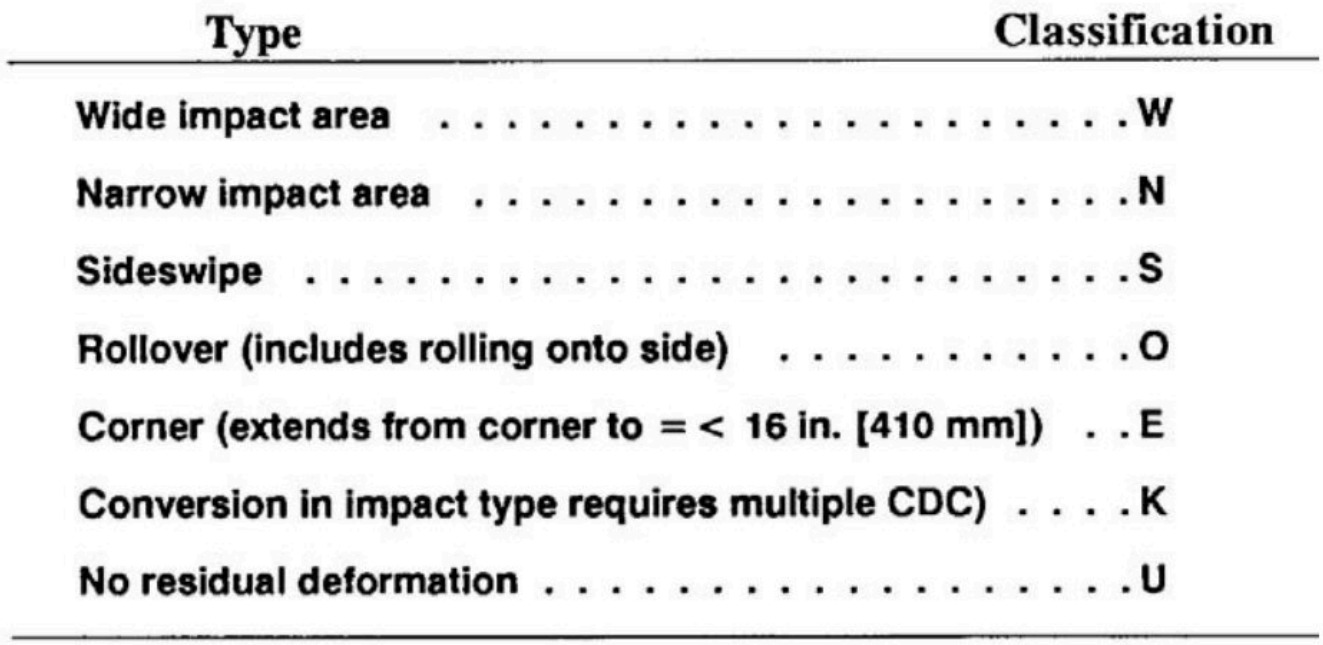
Vertical Location—Front, Rear, or Side Impacts	Classification
All	A
Top of frame to top of vehicle	H
Everything below belt line	E
Belt line and above	G
Middle—top of frame to belt line or hood	M
Frame—top of frame, frame, bottom of frame (incl. undercarriage)	L
Below undercarriage level (wheels and tires only)	W

Figure V-6 — Column 5, Vertical Location Code.

Column 6 — Type of Damage Distribution

The general type of impact or accident is defined in column 6. Although several entries are available, EDCRASH only makes a distinction between Narrow and Wide damage distributions. Further, some entries violate the assumptions of analysis, notably Sideswipe, Rollover, and multiple impact (K).

If Narrow impact is specified, the width of the damage profile is set to 16 inches; otherwise, Wide impact is assumed, and the width of damage is set based on tabular data in conjunction with the Specific Longitudinal or Lateral Location of Deformation entered in column 4 of the CDC.



Type	Classification
Wide impact area	W
Narrow impact area	N
Sideswipe	S
Rollover (includes rolling onto side)	O
Corner (extends from corner to =< 16 in. [410 mm])	E
Conversion in impact type requires multiple CDC)	K
No residual deformation	U

Figure V-7 — Column 6, Type of Damage Distribution.

Column 7 — Maximum Extent of Penetration

The extent of maximum residual deformation to the vehicle is based on a somewhat qualitative system wherein the vehicle is divided into penetration zones numbered from 1 to 9. Along the longitudinal axis of the vehicle, the B-pillar is generally taken as the maximum (i.e., the beginning of zone 9). A frontal crush penetrating rearward to the B-pillar will have a penetration extent of 9; conversely, a rear impact penetrating forward to the B-pillar will also have a penetration extent of 9. Laterally, a penetration extent of 9 is associated with penetrating the entire width of the vehicle. A penetration extent of 0 would imply no permanent deformation, and therefore no CDC.

For damage extents between 0 and 9, the extents are divided into zones normally having equal width. The exception to this rule occurs in the areas which include the windshield and windows.

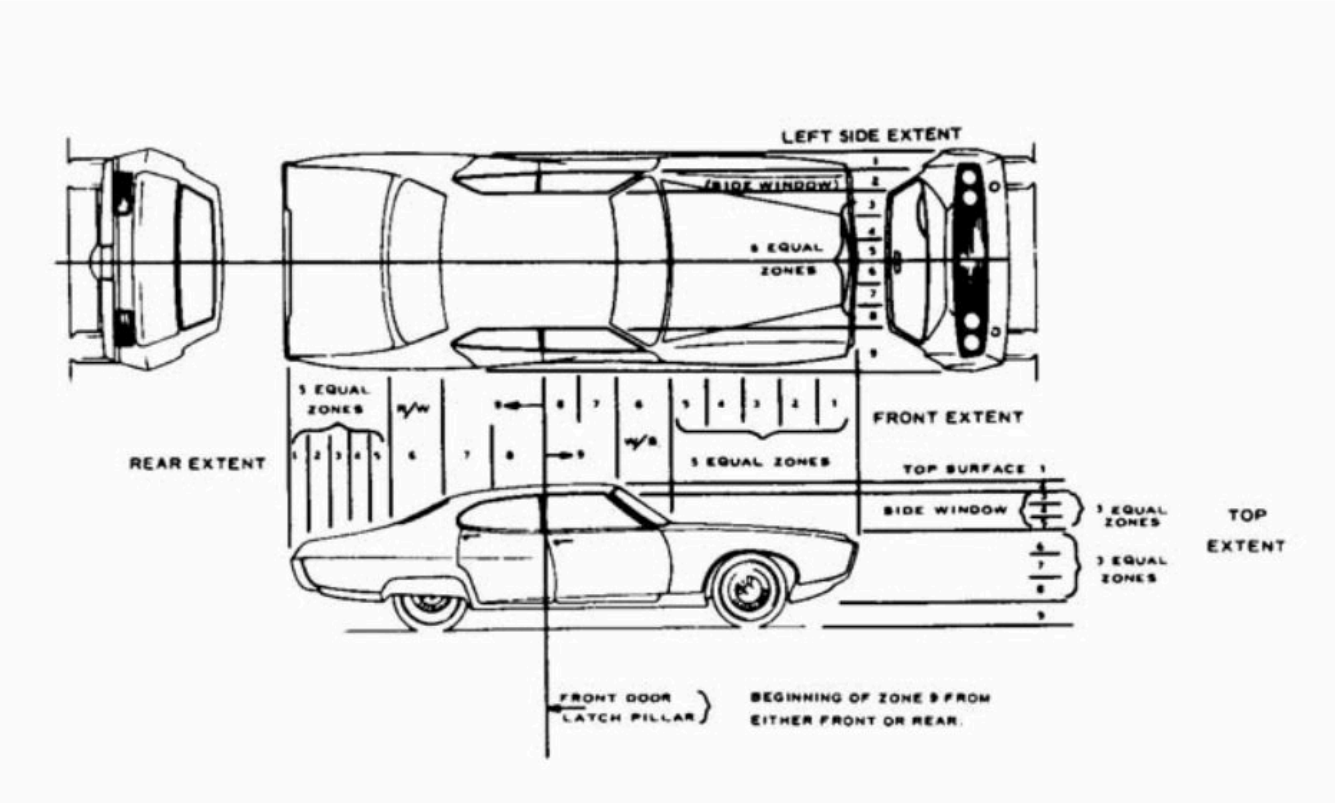


Figure V-8 — Column 7, Maximum Extent of Penetration.

> **NOTE:** See "Collision Deformation Classification", SAE J224 MAR80 [6.3] for further details.

Examples

The following are some examples of acceptable entries:

- **11FDEW1** — Full frontal impact having a PDOF of +30 degrees and a maximum penetration extending approximately 1/9th of the distance from the front of the vehicle to the B-pillar.
- **02RDEW4** — Right side impact having a PDOF of +60 degrees and having a maximum penetration of approximately 4/9th of the vehicle's overall width.

Appendix VI — References

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix VI, pages VI-1 through VI-10. Reference numbers are cited throughout the manual (e.g., [4.31] in the Brake Designer chapter).

The subject of vehicle dynamics as it relates to the study of motor vehicle accidents is very interesting and complex. Fortunately, there are many excellent references available; many are available directly from EDC. The user of HVE is urged to review these references. They provide a basis for the understanding and interpretation of HVE and HVE-compatible reconstruction and simulation models, and an excellent overview of the general field of motor vehicle safety. The references are divided into the following sections:

- **HVE** — Research related to the design and development of HVE
- **Motor Vehicle Safety** — Research related to the field of accident reconstruction
- **Humans** — Research related to modeling the human body
- **Vehicles** — Research related to modeling of vehicles and vehicle components
- **Environments** — Research pertaining to modeling of roadways and related structures
- **Standards and Recommended Practices** — Research related to rule-making and commonly used standards and recommended practices
- **General** — Other pertinent research or reference material

HVE

- 1.1 Day, T.D., "A Computer Graphics Interface Specification for Studying Humans, Vehicles and Their Environment," SAE Paper No. 930103, Society of Automotive Engineers, Warrendale, PA, 1993.
- 1.2 HVE Developer's Toolkit, Engineering Dynamics Corporation, Beaverton, OR, 1996.
- 1.3 Day, T.D., "An Overview of the HVE Developer's Toolkit," SAE Paper No. 940923, Society of Automotive Engineers, Warrendale, PA, 1994.
- 1.4 Day, T.D., "An Overview of the HVE Human Model," SAE Paper No. 950659, Society of Automotive Engineers, Warrendale, PA (presented at 1995 SAE International Congress, February, 1995).
- 1.5 Day, T.D., "An Overview of the HVE Vehicle Model," SAE Paper No. 950308, Society of Automotive Engineers, Warrendale, PA (presented at 1995 SAE International Congress, February, 1995).
- 1.6 HVE Vehicle Database, Engineering Dynamics Corporation, Beaverton, OR, 1996.
- 1.7 Day, T.D., "The Scientific Visualization of Motor Vehicle Accidents," SAE Paper No. 940922, Society of Automotive Engineers, Warrendale, PA, 1994.
- 1.8 Day, T.D., "Validation of Several Reconstruction and Simulation Models in the HVE Scientific Visualization Environment," SAE Paper No. 960891, Society of Automotive Engineers, Warrendale, PA, 1996.
- 1.9 Day, T.D., "Reconstruction and Simulation in the HVE 3-Dimensional Environment," SAE Paper No. 960890, Society of Automotive Engineers, Warrendale, PA, 1996.

Motor Vehicle Safety

- 2.1 Lee, S.N., Fell, J.C., "An Historical Review of the National Highway Safety Administration's Field Accident Investigation Studies," NHTSA, Washington, DC, 1988.

- 2.2 McHenry, R.R., "Development of a Computer Program to Aid the Investigation of Highway Accidents," Calspan Report No. VJ-2979-V-1, DOT HS 800 821, December, 1971.
- 2.3 Vehicle Analysis Package — EDCRASH Program Manual, Engineering Dynamics Corporation, Beaverton, Oregon, 1996.
- 2.4 Vehicle Analysis Package — EDSMAC Program Manual, Engineering Dynamics Corporation, Beaverton, Oregon, 1996.
- 2.5 EDC Simulations Training Manual, Version 4, Engineering Dynamics Corporation, Beaverton, Oregon, 1989.
- 2.6 Kenefect, M., "Crash Safety in Racing Cars," SAE Paper No. 930999, Society of Automotive Engineers, Warrendale, PA, 1993.
- 2.7 Crush Stiffness Coefficients for Vehicle Model Years 1960–1994, Neptune Engineering, Clovis, CA, 1994.
- 2.8 Prasad, A.K., "CRASH3 Damage Algorithm Reformulation for Front and Rear Collisions," SAE Paper No. 900098, Society of Automotive Engineers, Warrendale, PA, 1990.
- 2.9 Strother, C.E., Woolley, R.L., James, M.B., "A Comparison Between NHTSA Crash Test Data and CRASH3 Frontal Stiffness Coefficients," SAE Paper No. 900101, Society of Automotive Engineers, Warrendale, PA, 1990.
- 2.10 Woolley, R.L., Strother, C.E., James, M.B., "Rear Stiffness Coefficients Derived From Barrier Crash Data," SAE Paper No. 910120, Society of Automotive Engineers, Warrendale, PA, 1991.
- 2.11 EDCRASH Version 4 Source Code Listing, EDC Library Ref. No. 2003, Engineering Dynamics Corporation, February, 1989.
- 2.12 EDSMAC Version 2 Source Code Listing, EDC Library Ref. No. 2006, Engineering Dynamics Corporation, May, 1989.
- 2.13 Vehicle Crush Stiffness Coefficients With Damage Profile Supplement for Model Years 1970–1984, EDC Library Ref. No. 1042, Engineering Dynamics Corporation, August, 1987.
- 2.14 Day, T. and Hargens, R., "Differences Between EDCRASH and CRASH3," SAE Paper No. 850253, February, 1985.
- 2.15 Day, T. and Hargens, R., "An Overview of the Way EDCRASH Computes Delta-V," SAE Paper No. 870045, February, 1987.
- 2.16 Day, T. and Hargens, R., "A Personal Computer Program for Drawing Accident Sites," SAE Paper No. 880068, February, 1988.
- 2.17 Day, T. and Hargens, R., "An Overview of the Way EDSMAC Computes Delta-V," SAE Paper No. 880069, February, 1988.
- 2.18 Hargens, R. and Day, T., "Vehicle Data Sources for Accident Reconstruction," SAE Paper No. 880070, February, 1988.
- 2.19 Day, T. and Hargens, R., "Application and Misapplication of Computer Programs for Accident Reconstruction," SAE Paper No. 890738, February, 1989.
- 2.20 Day, T. and Hargens, R., "Further Validation of EDCRASH Using the RICSAC Staged Collisions," SAE Paper No. 890740, February, 1989.
- 2.21 McHenry, R.R., Segal, D.J., Lynch, J.P., Henderson, P.M., "Mathematical Reconstruction of Highway Accidents," NTIS PB-220 150, January, 1973.
- 2.22 Robbins, D.H., Melvin, J.W., Huelke, D.F., Sherman, H.W., "Biological Accident Investigation Methodology," NTIS PB83-184 366, February, 1983.

- 2.23 Campbell, K.L., "Energy Basis of Collision Severity," SAE Paper No. 740565, Society of Automotive Engineers, Warrendale, PA, July, 1974.
- 2.24 Emori, R.I., "Mechanics of Automobile Collisions," First International Conference on Vehicle Mechanics, Wayne State Univ., July, 1968.
- 2.25 Zeidler, F., Schreier, H.H., Stadelmann, R., "Accident Research and Accident Reconstruction by the EES-Accident Reconstruction Method," SAE Paper No. 850256, Society of Automotive Engineers, Warrendale, PA, February, 1985.
- 2.26 Tumbas, N.S., Smith, R.A., "Measurement Protocol for Quantifying Vehicle Damage from an Energy Basis Point of View," SAE Paper No. 880072, Society of Automotive Engineers, Warrendale, PA, February, 1988.

Humans

- 3.1 Robbins, D.H., Bennett, R.O., Roberts, V.L., "HSRI Three-Dimensional Crash Victim Simulator: Analysis, Verification and User's Manual, and Pictorial Section," Report No. BIO M-70-9, HSRI, University of Michigan, June, 1971.
- 3.2 Bennett, R.O., Bowman, D.H., Robbins, D.H., "Supplement to the HSRI Three-Dimensional Crash Victim Simulation Report," HSRI Report No. UM-HSRI-77-20, University of Michigan, Ann Arbor, MI, 1977.
- 3.3 Bowman, B.M., Bennett, R.O., Robbins, D.H., "MVMA Two-Dimensional Crash Victim Simulation, Version 4, Vol. I," Report No. UM-HSRI-79-5-1, University of Michigan, June, 1979.
- 3.4 Bowman, B.M., Robbins, D.H., Bennett, R.O., "MVMA Two-Dimensional Crash Victim Simulation, Version 4, Vol. III," UM-HSRI-79-5-3, University of Michigan, Ann Arbor, 1979.
- 3.5 Fleck, J.T., Butler, F.E., and Vogel, S.L., "An Improved Three Dimensional Computer Simulation of Crash Victims," NHTSA Report Nos. DOT-HS-801-507 through 510, Vols. 1–4, April, 1975.
- 3.6 Fleck, J.T., Butler, F.E., "Validation of the Crash Victim Simulator, Volume I, Engineering Manual — Part 1: Analytical Formulation," DOT-HS-6-01300, Calspan Corp., Buffalo, 1981.
- 3.7 Sanders, W.T., et al., "Performance Comparison of Air Damped Padding with Other Padding Materials," SAE Paper No. 920351, Society of Automotive Engineers, Warrendale, PA, 1992.
- 3.8 Seat Belt Webbing Tests in Accordance with FMVSS 209, Test Report No. DTB04R91-0393, Dayton T. Brown, Engineering and Test Division, LI, NY, 1991.
- 3.9 Day, T.D., "An Overview of the HVE Human Model," SAE Paper No. 950659, Society of Automotive Engineers, Warrendale, PA, 1995.
- 3.10 Gross, M.E., "The GEBOD III Program User's Guide and Description," AL-TR-1991-0102, Beecher Research Co., Dayton, OH, 1991.
- 3.11 Robbins, D.H., Snyder, R.G., McElhaney, J.H., Roberts, V.L., "A Comparison Between Human Kinematics and the Predictions of Mathematical Crash Victim Simulators," SAE Paper No. 710849, November, 1971.
- 3.12 Robbins, D.H., Becker, J.M., Bennett, R.O., Bowman, B.M., "Accident Data Simulation — Pedestrian and Side Impact — 3D," Report No. UM-HSRI-80-75, HSRI, University of Michigan, June, 1980.
- 3.13 Gray's Anatomy, Bounty Books, New York, NY, 1981.
- 3.14 Human Function and Structure, Luciano, D.S., Vander, A.J., Sherman, J.H., McGraw-Hill Book Company, New York, NY, 1978.
- 3.15 Obergefell, L.A., Gardner, T.R., Kaleps, I., Fleck, J.T., "Articulated Total Body Model Enhancements, Volume 2: User's Guide," AAMRL Report No. AAMRL-TR-88-043, January, 1988, NTIS No. A203 566.

- 3.16 MADYMO 3-D User's Manual, Volumes I and II, Version 5.1, TNO Road-Vehicles Research Institute, Delft, The Netherlands, May, 1994.
- 3.17 Webb, P.W., "Bioastronautics Data Book," NASA Report SP-3006, 1964.

Vehicles

- 4.1 Calspan Corporation, "Highway-Vehicle-Object Simulation Model," Calspan Report No. FHWA-RD-76-162, February, 1976.
- 4.2 MacAdam, C.C., Fancher, P.S., Hu, G.T., Gillespie, T.D., "A Computerized Model for Simulating the Braking and Steering Dynamics of Trucks, Tractor-semitrailers, Doubles, and Triples," Report No. UM-HSRI-80-58, HSRI, University of Michigan, June, 1980.
- 4.3 Moncarz, H.T., Bernard, J.E., Fancher, P.S., "A Simplified, Interactive Simulation for Predicting the Braking and Steering Response of Commercial Vehicles," Report No. UM-HSRI-PF-75-8, HSRI, University of Michigan, August, 1975.
- 4.4 Allen, R.W., Szostak, H.T., et al., "Analytical Modeling of Driver Response in Crash Avoidance Maneuvering: Vol. I — Technical Background," DOT-HS-807 270, NHTSA, Washington, DC, 1988.
- 4.5 Nalecz, A.G., "Development and Validation of Light Vehicle Dynamics Simulation (LVDS)," SAE Paper No. 920056, Society of Automotive Engineers, Warrendale, PA, 1992.
- 4.6 Vehicle Analysis Package — EDSVS Program Manual, Engineering Dynamics Corporation, Beaverton, Oregon, 1993.
- 4.7 Vehicle Analysis Package — EDVTS Program Manual, Engineering Dynamics Corporation, Beaverton, Oregon, 1993.
- 4.8 1993 Yearbook, The Tire and Rim Association, Inc., Copley, OH, 1993.
- 4.9 Tapia, G.A., "Extended Tire Testing," Calspan Report No. 6871-V-1, Calspan Corporation, Buffalo, NY, 1983.
- 4.10 Fancher, P.S., Bernard, J.E., "Vehicle-In-Use Limit Performance and Tire Testing," HSRI Report No. UM-HSRI-PF-75-1-1, University of Michigan, Ann Arbor, MI, 1975.
- 4.11 HSRI Flatbed Tire Test Results, University of Michigan, 1981.
- 4.12 Segal, D.J., Kamholz, L.R., Griffith, D.G., "Vehicle Component Characterization, Vol. II: Data Appendices," DOT-HS-807-055, MGA Research Corp., Akron, NY, 1987.
- 4.13 Day, T.D., "An Overview of the HVE Vehicle Model," SAE Paper No. 950308, Society of Automotive Engineers, Warrendale, PA, 1995.
- 4.14 HVE Vehicle Database, Engineering Dynamics Corporation, Beaverton, OR, 1996.
- 4.15 EDSVS Version 4 Source Code Listing, EDC Library Ref. No. 2001, Engineering Dynamics Corporation, May, 1989.
- 4.16 EDVTS Version 4 Source Code Listing, EDC Library Ref. No. 2002, Engineering Dynamics Corporation, May, 1989.
- 4.17 Fancher, P.S., Bernard, J.E., "Vehicle-In-Use Limit Performance and Tire Factors," UM-HSRI-PF-75-1-1, University of Michigan, January, 1975.
- 4.18 Sternberg, E.R., "Heavy-Duty Truck Suspensions," SAE Paper No. 760369, February, 1976.
- 4.19 Iacovoni, D.H., "Fundamentals of Automobile Handling Analysis," March 13, 1969.
- 4.20 Segel, L., "Theoretical Prediction and Experimental Substantiation of the Response of the Automobile to Steering Control," Institution of Mechanical Engineers, 1956.

- 4.21 Taborek, J.J., "Mechanics of Vehicles," Machine Design, The Penton Publishing Co., Cleveland, 1957.
- 4.22 Pacejka, H.B., "Simplified Analysis of Steady Turning Behavior of Motor Vehicles — Parts 2 & 3," Vehicle Systems Dynamics, Vol. 2, No. 4, December, 1973.
- 4.23 Fiala, E., "Seitenkräfte am rollenden Luftreifen" (Lateral Forces on Rolling Pneumatic Tires), Zeitschrift V.D.I., 96, No. 29, October, 1954.
- 4.24 Clark, S.D., Mechanics of Pneumatic Tires, National Bureau of Standards Monograph 122, November, 1971 (out of print).
- 4.25 Segel, L., ME 690 unpublished lecture notes, University of Michigan, 1981.
- 4.26 Siddall, D.E., Day, T.D., "Updating the Vehicle Class Categories," SAE Paper No. 960897, Society of Automotive Engineers, Warrendale, PA, February, 1996.
- 4.27 Neptune, J.A., "Overview of an HVE Vehicle Database," SAE Paper No. 960896, Society of Automotive Engineers, Warrendale, PA, February, 1996.
- 4.28 Neptune, J.A., Blair, G.Y., Flynn, J.E., "A Method for Quantifying Vehicle Crush Stiffness Coefficients," SAE Paper No. 920607, Society of Automotive Engineers, Warrendale, PA, February, 1992.
- 4.29 Neptune, J.A., Flynn, J.E., "A Method for Determining Accident Specific Crush Stiffness Coefficients," SAE Paper No. 940913, Society of Automotive Engineers, Warrendale, PA, February, 1994.
- 4.30 Limpert, R., Brake Design and Safety, Society of Automotive Engineers, Inc., Warrendale, PA, 1992.
- 4.31 Flick, M.A., "The Effect of Brake Adjustment on Braking Performance," National Highway Traffic Safety Administration, U.S. Dept. of Transportation, Report No. DOT HS 807-287, Washington, DC, 1988.
- 4.32 Stroh, G.B., Lawrence, M.H., Deibel, W.T., "Effects of Shoe Force Geometry on Heavy Duty Internal Shoe Brake Performance," SAE Paper No. 680432, Society of Automotive Engineers, Warrendale, PA, February, 1968.
- 4.33 Gurney, J.W., Bernard, J.E., "The Utilization of a Computer Simulation as an Aid to Predict Compliance with MVSS 121," SAE Paper No. 740137, Society of Automotive Engineers, Warrendale, PA, February, 1974.
- 4.34 Bowman, B.L., "Grade Severity Rating System (GSRS) — Users Manual," Goodell-Grivas, Inc., FHWA Report No. IP-88-015, Washington, DC, 1989.
- 4.35 Radlinski, R.W., Williams, S.F., "NHTSA Heavy Duty Vehicle Brake Research Program, Report No. 1 — Stopping Capability of Air Braked Vehicles, Volume 1 — Technical Report," National Highway Traffic Safety Administration, U.S. Department of Transportation, Report No. DOT HS 806-738, Washington, DC, 1985.
- 4.36 Post, T.M., Fancher, P.S., Bernard, J.E., "Torque Characteristics of Commercial Vehicle Brakes," SAE Paper No. 750210, Society of Automotive Engineers, Warrendale, PA, February, 1975.
- 4.37 Johnson, L.K., Fancher, P.S., Gillespie, T.D., "An Empirical Model for the Prediction of the Torque Output of Commercial Vehicle Air Brakes," Highway Safety Research Institute, Report No. UM-HSRI-78-53, University of Michigan, Ann Arbor, 1978.
- 4.38 Day, T.D., "An Overview of the EDSMAC4 Collision Simulation Model," SAE Paper No. 1999-01-0102, Society of Automotive Engineers, Warrendale, PA, March, 1999.
- 4.39 Day, T.D., "Differences Between EDVDS and Phase 4," SAE Paper No. 1999-01-0103, Society of Automotive Engineers, Warrendale, PA, March, 1999.
- 4.40 York, A.R., Day, T.D., "The DyMESH Multi-Vehicle Collision Simulation Model," SAE Paper No. 1999-01-0104, Society of Automotive Engineers, Warrendale, PA, March, 1999.
- 4.41 Bosch Driving-Safety Systems, 2nd Edition, Robert Bosch GmbH, ISBN 0-7680-0511-6, Society of Automotive Engineers, Warrendale, PA, 1999.

- 4.42 Day, T.D., Roberts, S.G., "A Simulation Model for Vehicle Braking Systems Fitted with ABS," SAE Paper No. 2002-01-0559, Society of Automotive Engineers, Warrendale, PA, 2002.
- 4.43 Day, T.D., "Validation of the SIMON Model for Vehicle Handling and Collision Simulation — Comparison of Results with Experiments and Other Models," SAE Paper No. 2004-01-1207, Society of Automotive Engineers, Warrendale, PA, 2004.

Environments

- 5.1 Rudaux, L., DeVaucouleurs, G., LaRousse Encyclopedia of Astronomy, Prometheus Press, New York, 1959.
- 5.2 1993 ASHRAE Handbook, Fundamentals, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc., Atlanta, 1993.
- 5.3 Bekker, M.G., Introduction to Terrain-Vehicle Systems, University of Michigan Press, Ann Arbor, 1969 (out of print; contact EDC for information).

Recommended Practices

- 6.1 Human Tolerance to Impact Conditions as Related to Motor Vehicle Design, SAE Technical Report No. J885 JUL86, Society of Automotive Engineers, Warrendale, PA, 1986.
- 6.2 Vehicle Dynamics Terminology, SAE Technical Report No. J670e, Society of Automotive Engineers, Warrendale, PA, 1976.
- 6.3 Collision Deformation Classification, SAE Technical Report J224 MAR80, March, 1980.
- 6.4 Accident Reconstruction Terminology, SAE Technical Report No. J1675, Society of Automotive Engineers, Warrendale, PA (under development at time of original publication).
- 6.5 Friction Coefficient Identification System for Brake Linings, SAE Technical Report J866 MAR85, Society of Automotive Engineers, Warrendale, PA. *(cited in Chapter 29; not printed in the original Appendix VI list)*

General

- 7.1 AUTOSIM Reference Manual, Version 2, Transportation Research Institute, University of Michigan, Ann Arbor, MI, 1993.
- 7.2 Viewpoint Animation Engineering, Orem, UT, 1998.
- 7.3 Wernecke, J., The Inventor Mentor, Addison-Wesley Publishing Co., Menlo Park, CA, December, 1993.
- 7.4 Neider, J., Davis, T., Woo, M., OpenGL Programming Guide, Addison-Wesley Publishing Co., Menlo Park, CA, June, 1993.
- 7.5 Strang, G., Linear Algebra and its Applications, Second Edition, Academic Press, New York, 1980.
- 7.6 Waite, M., Martin, D., Prata, S., Unix System V Primer, Second Edition, Prentice Hall Publishing, Carmel, IN, 1992.
- 7.7 Quercia, V., O'Reilly, T., X Window System User's Guide, OSF/Motif Edition, O'Reilly and Associates, Sebastopol, CA, 1993.
- 7.8 Welty, J.R., Wicks, C.E., Wilson, R.E., Fundamentals of Momentum, Heat and Mass Transfer, John Wiley & Sons, New York, 1969.

Appendices VII, VIII and IX — Reserved

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendices VII–IX.

The following appendices were reserved (placeholder) sections in the original edition and contained no substantive content:

Appendix VII — Developer's Toolkit Overview

(Reserved.)

> **NOTE:** Contact EDC for information about receiving a copy of the HVE Developer's Toolkit.

The HVE Developer's Toolkit is documented separately; see also references 1.2 and 1.3 in Appendix VI.

Appendix VIII — Databases

(Reserved.) HVE's human, vehicle and tire databases are discussed in Chapter 2 (Using HVE) and in the Human and Vehicle Editor chapters. The database files reside in the `supportFiles\db` directory (see Appendix I, HVE File System).

Appendix IX — Quick Reference

(Reserved.)

Appendix X — Index (Topic Locator)

Updated Markdown edition of the HVE User's Manual (HVE Version 5, Seventh Edition, January 2006), Appendix X. The original page-number index has been replaced by a topic-to-chapter locator table, since page numbers do not apply to the Markdown edition. Chapter numbers refer to the original manual's chapter organization; Roman numerals refer to appendices.

Topic	Where to look
3-D Editor	Chapter 2 (overview), Chapter 19
3-D Geometry (environments)	Chapters 12–13
ABS (Anti-lock Brake System)	Chapter 11 (Brake System dialogs), Chapter 31
ABS algorithms (Tire Slip, Bosch V1)	Chapter 31
Accident History report	Chapters 2, 17; Tutorial Lesson 3
Aerodynamic drag	Chapters 11, 13
Airbag restraints	Chapters 4, 11
Anti-aliasing	Options menu / Render Options (Chapters 2, 6); Chapter 26 Tips
Atmospheric model	Chapter 13
Audit Trail	Chapters 2, 17–18; Tutorial Lesson 3
AutoPosition	Chapters 4, 6
Brake Designer	Chapter 11 (dialogs), Chapters 29–31
Brake system (pedal ratio, proportioning, pressure)	Chapters 11, 16
Brake table (driver controls)	Chapters 2, 4
Brake temperature model	Chapter 30
Calculation models (reconstruction, simulation)	Chapter 2
Calculation options	Chapters 2, 6; physics program manuals
Camera (Set Camera dialog, object-based cameras)	Chapter 5; Tutorial Lesson 4
Case files (open, save, database)	Chapters 2, 3
CDC (Collision Deformation Classification)	Chapters 4, 16; Appendix V
Collision pulse	Chapters 2, 4
Configuration file	Chapter 2; Appendix I
Contacts (contact surfaces, ellipsoids)	Chapters 2, 4
Coordinate systems	Appendix III
Damage profiles, damage energy	Chapters 2, 4, 16
Databases (case, custom, generic, human, tire, vehicle)	Chapter 2
Delta-V	Chapters 2, 16
Developer's Toolkit	Chapter 2; Appendix VII
Differential	Chapters 4, 11
Driver controls (throttle, brake, steering, gear)	Chapters 2, 4
Drivetrain (engine, transmission, differential)	Chapter 11

Topic	Where to look
EES / EBS (Equivalent Energy Speed)	Chapters 2, 4, 16
Environment Editor / Environment model	Chapters 12–13
Event Editor / Event model	Chapters 15–16
Event Controller	Chapters 1, 2, 15; Tutorial
Event positions (initial, impact, POC, separation, rest)	Chapters 2, 4, 15–16
File menu (New, Open, Save, Print, Video, Exit)	Chapter 3
Gear table	Chapters 2, 4
GEBOD	Chapters 8–9
Grid options	Chapters 2, 6
Help menu	Chapters 2, 7
Human Editor / Human model (inertias, ellipsoids, joints, injury tolerance)	Chapters 8–9
Key Results	Chapters 2, 6, 15; Tutorial Lesson 2
Manipulators (3-D Editor)	Chapter 21
Menu Bar	Chapters 1–2; Tutorial Lesson 1
Messages (warnings, errors)	Appendix II
Modes (Human, Vehicle, Environment, Event, Playback)	Chapters 1–2
Multiple events (combining, timing, object precedence)	Chapter 18; Tutorial Lesson 3
Newton's 3rd Law checks	Chapters 4, 16; Appendix II
NTSC / PAL / SECAM	Chapter 26 *(legacy)*
Output reports (graphic, numeric, Traj Sim, Variable Output)	Chapters 2, 17
Payload	Chapters 2, 4
PDOF (Principal Direction of Force)	Chapters 4, 16; Appendix V
Playback Editor / Playback windows	Chapter 17; Tutorial Lessons 3–4
Playback Controller	Chapters 2, 17; Chapter 27
Position/Velocity dialog	Chapters 2, 4
Printing	Chapters 2, 3, 17
Reconstruction vs simulation	Chapter 2
Render options (shading, quality, complexity)	Chapters 2, 6
Restraints (belts, airbags, usage)	Chapters 2, 4, 11
Road surfaces (friction zones, contact rules)	Chapter 13
Scanned images (overlays)	Chapters 2, 5, 12

Topic	Where to look
Set-up menu	Chapters 2, 4
Simulation controls (timesteps, termination conditions)	Chapters 2, 6
Skidmarks	Chapters 2, 6, 11, 16; Chapter 31 Tips
Steer table / steering system	Chapters 2, 4, 11, 16
Stiffness coefficients (A, B, Kv)	Chapters 2, 4, 11
Suspension properties	Chapter 11
Tire properties (friction, cornering stiffness, camber stiffness, slip)	Chapters 11, 16
Trajectory simulation windows	Chapters 2, 17; Tutorial
Transmission	Chapters 4, 11
Units (selecting, units files)	Chapters 2, 6; Appendix IV
User preferences	Chapters 2, 6
Variable output / variable selection	Chapters 6, 17–18
Vehicle Editor / Vehicle model (parameters)	Chapters 10–11
Video basics (recording methods, compression, formats, tips)	Chapter 26
Video interface (setup, source/destination, recording, saving)	Chapter 27; Tutorial Lesson 4
View menu (Set Camera, overlays)	Chapter 5
Viewer controls (pick/manipulate, thumbwheels, dolly, zoom)	Chapter 2; Tutorial Lesson 1
Water, driving on	Chapter 19
Wheel data (locations)	Chapters 2, 4