PC-CRASH

A Simulation Program for Vehicle Accidents

Operating Manual

Version 8.1

November 2007

Linz, Austria

~~Distributed and Supported in North America by:~~

MACINNIS ENGINEERING *Associates*

Consulting Engineers

11-11151 Horseshoe Way
Richmond, BC Canada V7A 4S5
Tel: 604 277 3040
Fax: 604 277 3020
pccrash@maceng.com
www.maceng.com

Contents

LICENSE AGREEMENT - PC-CRASH

SINGLE USER PRODUCT - This is a legal agreement between you, the end user, and the software company, Dr. Steffan Datentechnik Ges.m.b.H.

BY OPENING THE SEALED DISK PACKAGE, YOU ARE AGREEING TO BE BOUND BE THE TERMS OF THIS AGREEMENT. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, PROMPTLY RETURN THE UNOPENED DISK PACKAGE AND THE ACCOMPANYING ITEMS (including written materials and binders or other containers) TO THE PLACE YOU OBTAINED THEM FOR A FULL REFUND.

1. GRANT OF LICENSE - DSD grants you the right to use one copy of the enclosed PC-Crash software program (the "SOFTWARE") on a single terminal connected to a single computer. Installation on a network server for the sole purpose of distribution to one or more other computers shall not constitute "use" for which a separate license is required. You may not network the software or otherwise use it on more than one computer or computer terminal at a time.
2. COPYRIGHT - The SOFTWARE is owned by Dr. Steffan Datentechnik Ges.m.b.H. and is protected by copyright laws, international treaty provisions, and all other applicable national laws. You may not copy the product manual or written materials accompanying the SOFTWARE.
3. OTHER RESTRICTIONS - You may not rent or lease the software. You may not reverse engineer, de-compile or disassemble the software.
4. NO LIABILITY FOR CONSEQUENTIAL DAMAGES - To the maximum extent permitted by applicable law, in no event shall the company Dr. Steffan Datentechnik or its representatives or suppliers be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or other pecuniary loss) arising out of the use or inability to use this product, even if Dr. Steffan Datentechnik has been advised of the possibility of such damages.
5. NOTICE: Affiliated companies (collectively the "Publisher") make no representations or warranties of any kind whatsoever with respect to the contents herein and specifically disclaim any implied warranties or merchantability or fitness for any particular purpose. The Publisher shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this program or its contents.

WARNINGS AND DISCLAIMER - Specs VEHICLE DATABASE

The "Specs" vehicle database has been developed by Transport Canada, and is supplied with their authority free of charge for the convenience of PC-Crash users.

Transport Canada issues the following warnings and conditions on the use of the Canadian Vehicle Specifications System:

1. The data in the databases is not guaranteed to be 100% accurate. The responsibility of the accuracy of all input data remains with the user. It is possible that you may find errors in dimensions or weights. Transport Canada asks that you inform them of any such errors so that they can make the necessary corrections.
2. Transport Canada does not maintain a list of users of the system. Therefore, if you wish to obtain the specifications for the next model year it is necessary to submit a new request each year (new model years are normally available around April). Such a request would be a complete system revision, since programs and/or data may have changed in the interim.
3. The version of the system supplied contains programs that only query the database. Transport Canada asks that you do not attempt to make any changes to the data.
4. Transport Canada provides this data and system free of charge and by so doing absolves itself of any requirement to guarantee the data under scrutiny from any source.

The KBA vehicle database is based on the manufacturer and model files of the German department for motor vehicles (Kraftfahrt-Bundesamt), Fördestraße 16, D-24944 Flensburg.

Chapter 1
Installation

Introduction

PC-Crash is a powerful program for the simulation of motor vehicle accidents, covering many different accident situations. It takes advantage of the latest hardware and software developments, which allows increasingly complex calculations to be performed on a personal computer. PC-Crash was developed as a Microsoft Windows application for ease of use and compatibility with other programs.

PC-Crash contains several different calculation models, including an impulse-momentum crash model, a stiffness based impact model, a kinetics model for realistic trajectory simulations, and a simple kinematics model for time-distance studies. For maximum versatility, PC-Crash simulation results can be viewed and outputted in scale plan and elevation views, 3D perspective view, and in numerous diagrams and tables.

The PC-Crash Software Package

The PC-Crash software package contains:

- A letter with specific installation instructions;
- The program CD, including sample files and animations, DXF vehicle shapes, and several vehicle databases;
- The PC-Crash Operating and Technical manuals;
- A hardware lock for connection to the parallel or USB port.

README.TXT

Please read the file README.TXT, if it exists, on the PC-Crash CD. It contains information which is not included the manual.

Hardware Requirements

PC-Crash is suitable for all Pentium and higher computers, with the user interface Microsoft Windows 98/ME, or Windows NT/2000/XP.

A minimum random-access memory (RAM) of 128 megabytes (MB) is required, and more is recommended when working with large bitmap files. A VGA video card with a minimum resolution of 1024 x 768 pixels and 32 bit color depth is required. A video card that supports DirectX graphics acceleration is desirable.

Use of Mouse and Printers

A mouse or other tracking device is required for PC-Crash.

All printers supported by Microsoft Windows can be used with PC-Crash. For some printers, not all features can be used (e.g. scanned pictures cannot be printed on a plotter).

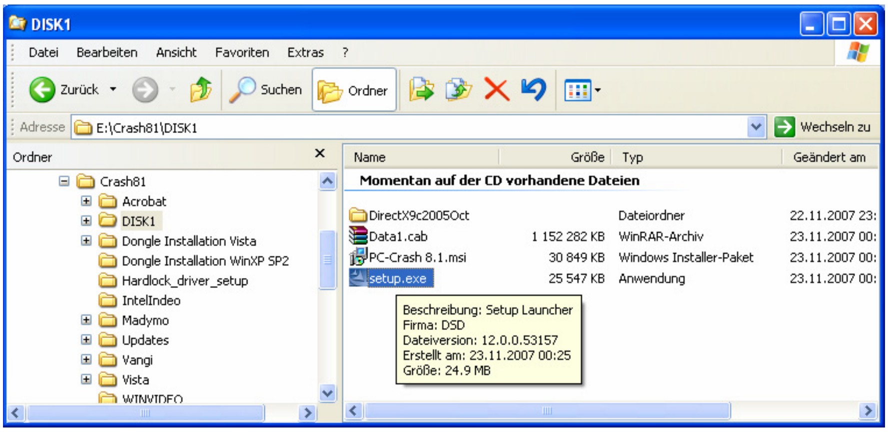
The settings are listed in the individual manuals of the printer drivers.

Installation of PC-Crash

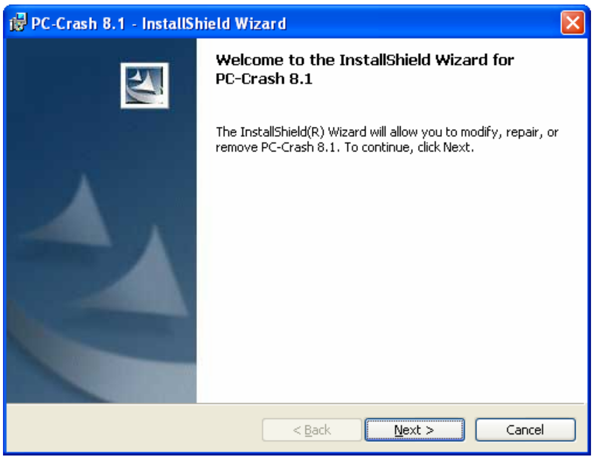
Before installing PC-Crash, it is recommended the user be familiar with the use of Microsoft Windows.

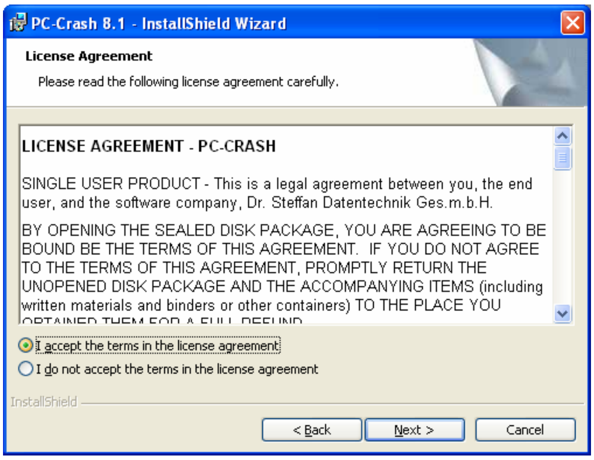
PC-Crash cannot be run from the CD. PC-Crash must be installed on the hard drive, as follows:

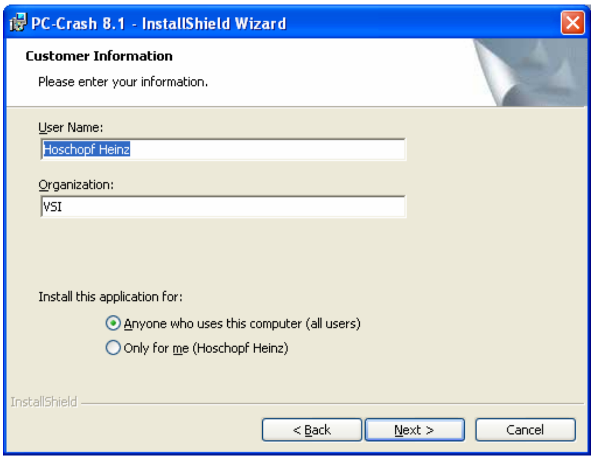
1. Insert the PC-Crash CD in the CD drive.
2. Use the Run command in the Microsoft Windows Start menu (or the Windows Explorer).
3. Select **Setup.exe** from the **PC-Crash\Disk1** directory on the CD.



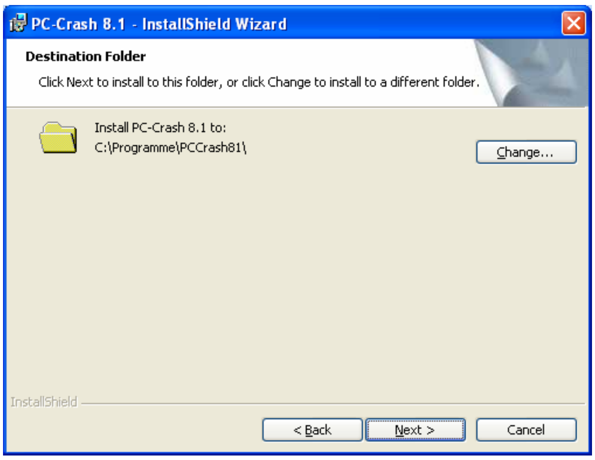
4. Follow the instructions given by the installation wizard program.



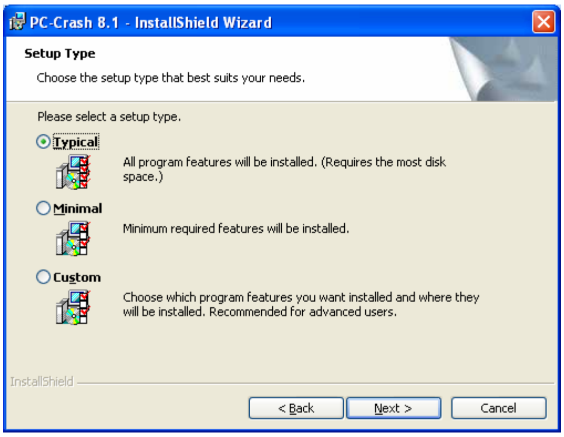


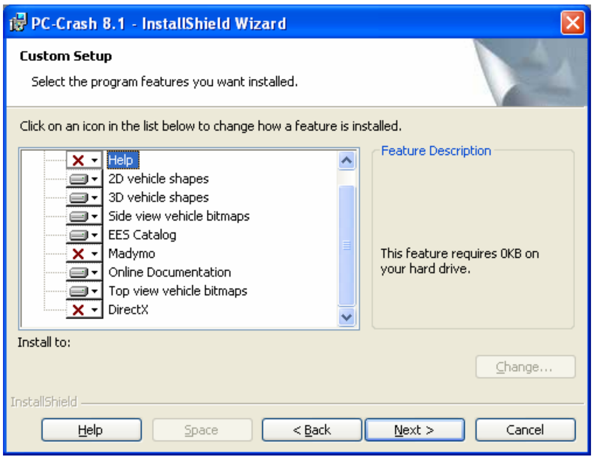


5. All necessary files are copied into the directory **C:\Program files\PCCrash81** by default. The user can change the name of this directory during installation (The installation could last longer).

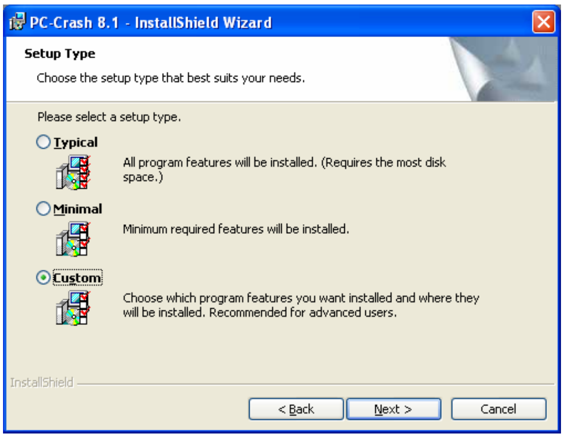


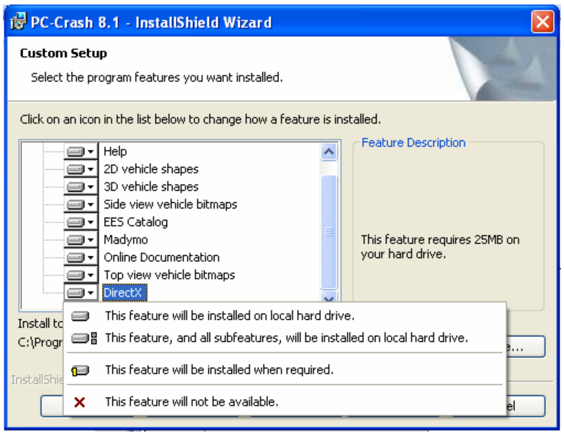
6. You may either install the default components or select the components to install manually by selecting **Custom**.

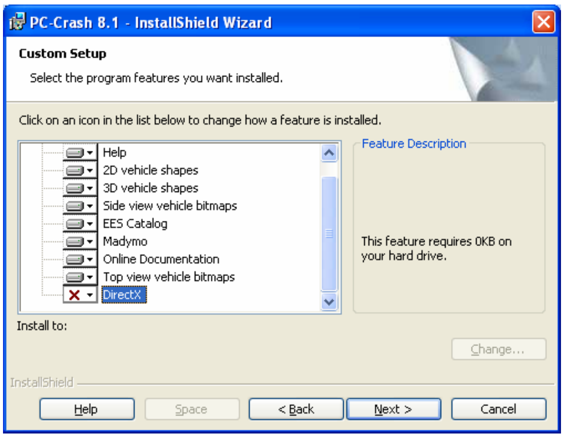




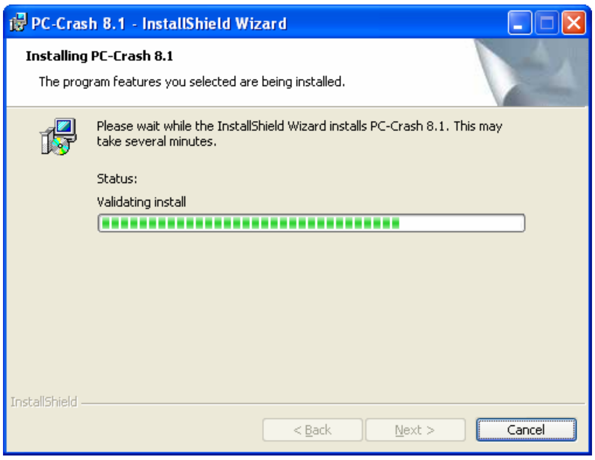
Vista: for installation under Windows Vista select Custom setup and deactivate DirectX







7. After having selected the components the copy process will start.



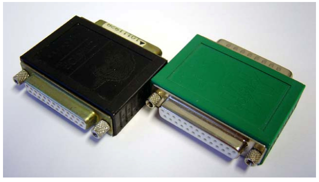
If you make an error during the installation process, you can go back to make changes by pressing the <Back button. The Cancel button can be used to abort the installation at any time. After completing the installation you can repair or modify your installation of PC Crash by starting the setup from the CD again.

Licensing PC-Crash

PC-Crash is protected against copying by a hardware lock (also called a "Dongle") and a license file. There are different types of dongles distributed with PC-Crash. New dongles and USB dongles, which especially are recommended for notebook users, are installed automatically with the setup of PC Crash. No further steps are necessary, just start the program with the dongle connected and your license will be detected.

Small USB dongle

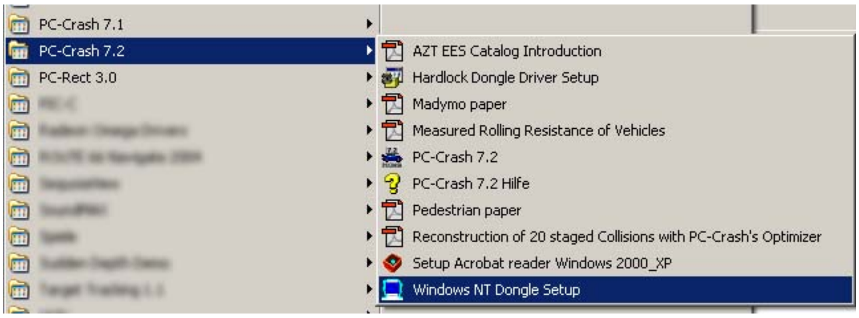
New dongle



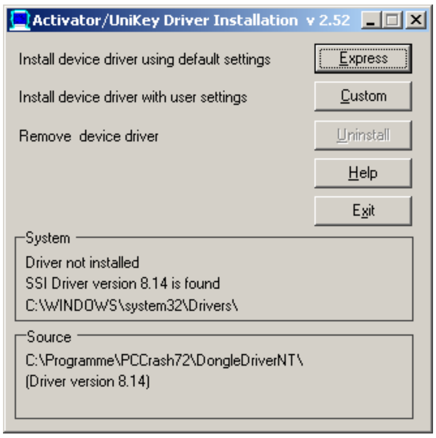
Old dongle

Users that have older dongles (compare the pictures) have to proceed with the following steps to get their license running:

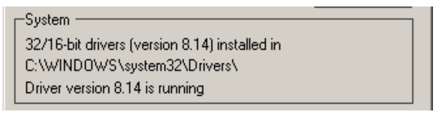
1. Start the Windows NT Dongle Setup located in the Windows Start Menu.



2. If, in the now appearing window, the **Express** button is deactivated the dongle driver has been installed already and you can close the window and start PC-Crash. Otherwise press the **Express** button.

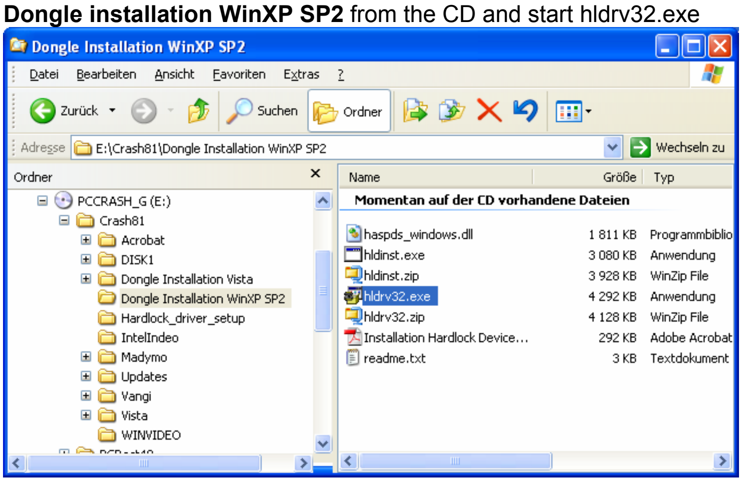


3. The **Express** button should now be deactivated and following message should be visible in the window:

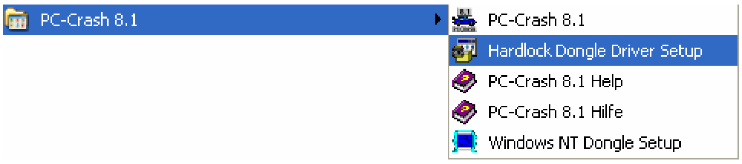


If a new or USB Dongle (compare the pictures) this is not recognized, then the Dongle should be initialized. Proceed with the following steps to get your license running:

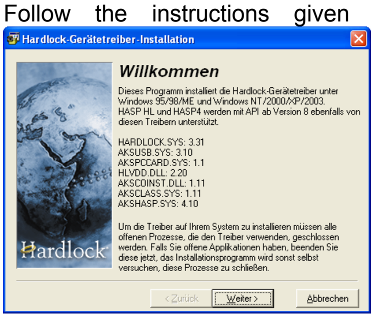
1. Depending upon operating system start either **Dongle installation Vista** or **Dongle installation WinXP SP2** from the CD and start hldrv32.exe



or of the program group of **PC Crash xxxx** start **Hardlock Dongle Driver Setup.**



2. Follow the instructions given by the installation wizard program.



Demo-Version

PC-Crash can be run as a licensed version (with hardware lock and license file installed) or as a "demo-version". The licensed version automatically shifts into the demo-version when the hardware lock is removed from the parallel port.

The demo-version permits loading and playing existing files and setting up new projects, but prevents new simulations from being performed. The title bar indicates which version of PC-Crash is currently running. The demo-version has "Demoversion" in the title bar while the licensed version has the name of the licensee in the title bar.

Since PC-Crash will run only in the demo-version without the hardware lock, it is permissible to make copies of the program for others.

For operation of PC-Crash in the demo-version, startup can be made quicker by deleting the license file Strlib32.dll from the PCCrash73 directory or by pressing the ESC button during startup.

PC-Crash Features

- Simultaneous simulation of up to 2 vehicles (PC-Crash 2D) or 32 vehicles (PC-Crash 3D)
- Interface to Specs (North American), ADAC, Vyskocil, DSD (European and Japanese) and KBA (as of October 2005) vehicle databases.
- 2D or 3D kinetic calculation model
- Front/rear brake force distribution model
- Specification of driver reaction, accelerating, braking, steering and other parameters, in the form of sequences
- Definition of different road elevations, slopes and friction coefficients in specific polygonal areas
- Impact model by Kudlich-Slibar, based on conservation of linear and angular momentum, with "full" and "sliding" impacts possible
- Backwards calculation of impacts is possible
- Specification of impact elasticity with restitution or separation velocity
- 2D or 3D impact model, with unlimited number of impacts
- Automatic calculation of all secondary impacts
- Collision optimizer, for the automatic determination of impact speeds and seven other impact parameters, based on rest and/or up to five intermediate vehicle positions
- Automatic kinematic calculation of accident avoidance.
- Forwards automatic avoidance simulation (velocity decrease, brake increase)
- Various diagrams for wheel forces, etc,
- Kinematic and kinetic (default mode) specification of vehicle paths
- Backtracking tire marks with a kinematic skidding calculation to determine post-impact conditions is possible.
- Measurement tool
- Printout of report of input/output values, including all collision and trajectory parameters and character counting
- Detailed vehicle shapes can be specified using DXF files, with change of shape at impact possible
- Scene DXF and VRML drawings and/or bitmaps can be imported into the simulation
- Integrated drawing program for drawing/modifying scene drawings and vehicle DXF shapes
- Calculation of rollovers and vaults
- Choice of two tire models (Linear or TM-Easy)
- Calculation of acceleration due to engine power and air resistance with up to 16 transmission ratios and the ability to gear down when going up grades
- Calculation of the effects of wind and air resistance, including down force and uplift
- Direct switching between different units systems (e.g. km/h, mph, m/s, f/s)
- Direct switching between different languages
- Auto save feature, with user-definable intervals

- "Undo" up to 50 prior operations
- Interactive help
- Improved vehicle suspension bump-stop model
- Interface to optional Madymo® occupant modeler
- Collision Optimizer Monte Carlo (random) algorithm
- New AZT EES catalog of European vehicle damage photographs
- Individual damaged wheel steering and positioning
- Additional Kinetic Path steering model features
- Up to five axles per vehicle
- North American symbol library
- Additional drawing tool features
- Multiple scene bitmap importing
- Revamped User Manual with more detailed explanations
- Improved templates for simple exchange of data between PC-CRASH and WinWord
- Extended wizard for kinematics simulation
- New simulation model for electronic stability control systems (ESP)
- Mouse Wheel support for all input windows
- Updated Crash 3 database (Stand 02/2007)
- KBA 2007
- Bitmaps are projected also on slopes
- Measurement grid can be extended at arbitrary edge
- Improved representation and expression of bitmaps (interpolation and smoothing)
- Transparency option for bitmaps
- Mirror function for limit method
- Drawing program toolbar
- User defined menus and toolbars
- Bitmap Toolbar for handling of bitmaps
- Adjustable indication sequence for bitmaps (foreground/background)
- Friction polygons and road slope toolbar
- Default settings new

Additional Features of PC-Crash 3D

- Simulation and collision analysis of trailers (steered, non-steered, semi-trailer), with more than one trailer per tow vehicle possible. Offsets at the hitch point can be specified.
- Multiple collisions between different vehicles
- New High Resolution 3D Vehicle models
- 3D perspective view, with display of 3D vehicles and scene 2D or 3D DXF drawings and rectified bitmaps

- VRML and FCE Vehicle models can be imported
- Generation of 3D video animations with fixed or moving camera position, playable with Windows Media Player
- Tool for constructing or importing complicated 3D scenes, including those created from total station survey files or car interior.
- Multibody pedestrian model
- Multibody motorcycle, bicycle and unrestrained occupant models
- Multiple multibody objects in one simulation, and on sloped surfaces
- Simulation of movable load
- Belt modeling
- Trailer steering model (based on articulation angle)
- Crash 3 impact module with interface to NHTSA vehicle database
- Visualization of Crash 3 deformations
- Side View window for analyzing vehicle interaction in rear-end impacts, with European vehicle side view bitmaps
- 2D and 3D vehicle DXF automatic deformation model
- 3D window dynamic viewing
- Direct X 3D graphics, for improved rendering
- New stiffness based crash simulation model
- New stiffness database with real crash test to be used in stiffness based crash simulation
- Improved occupant simulation in PC-CRASH including seatbelts and car interior.
- New mesh based impact model with improved structural stiffness and deformation calculation at vehicle/vehicle and vehicle/slope collisions.
- Key-numbers searching for KBA-database
- Calculation of tracks caused by tire contact
- Bounds method within the Drawing Tool
- Square measurement grid within the Drawing Tool
- Crash backwards calculation with momentum/angular momentum combination
- Adapted impact analysis backwards
- Possibility to save PC-Crash project files for different versions (7.0, 7.1, 7.2, 7.3, Pocket Crash)
- Refresh-display of point of impact (POI) velocities
- Refresh-display of intersection areas of momentum mirror method (backward method), with momentum diagram (scale 0.001:1 m for 1000 Ns)
- Adapted v-s-t window (point of reaction, reaction time, lag time adjustable)
- Camera rotation with roll and pitch
- Vehicle administration (copy, delete, exchange)
- Mesh model with X61/FCE vehicles
- Expansion of FCE vehicles
- EES calculation for Crash 3 model
- 64 bit version of PC-Crash available

- Adapted multibody simulation model (faster calculation, new joint types)
- Sort function within Crash3 data base
- Sort function within EES catalog
- Apply function within measurement grid
- Apply function within limit method
- New 3D vehicle models
- Selection of the pre-impact impulse direction for EES backwards procedures
- Support of DFF files for 3D vehicles (Renderware)
- Rest- and intermediate position can be switched on and off separately
- Optimization of multibody calculations (further optimization in progress)
- Preview for vehicle Dxf dialogue

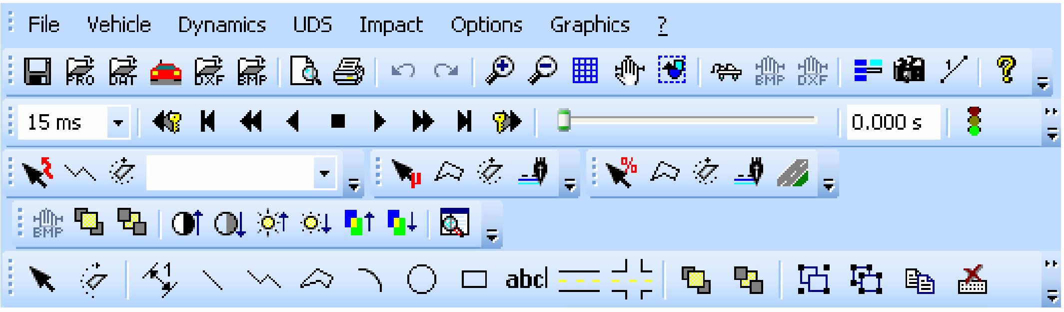
Chapter 2
Working with PC-Crash

This chapter leads the new user through the basic features of PC-Crash. For more details on a particular feature, refer to the subsequent chapters.

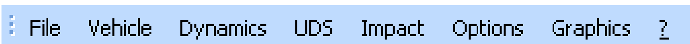
Menu Options

After the program has been started, the menu options are listed across the top of the PC-Crash main screen.

Note, that if you start the program the first time, you can select the language the user interface (all menu options and texts) has. The language can be changed later by selecting **Options – Options…**.



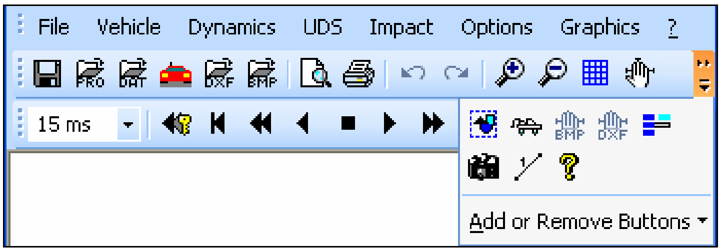
The following menu options can be selected:



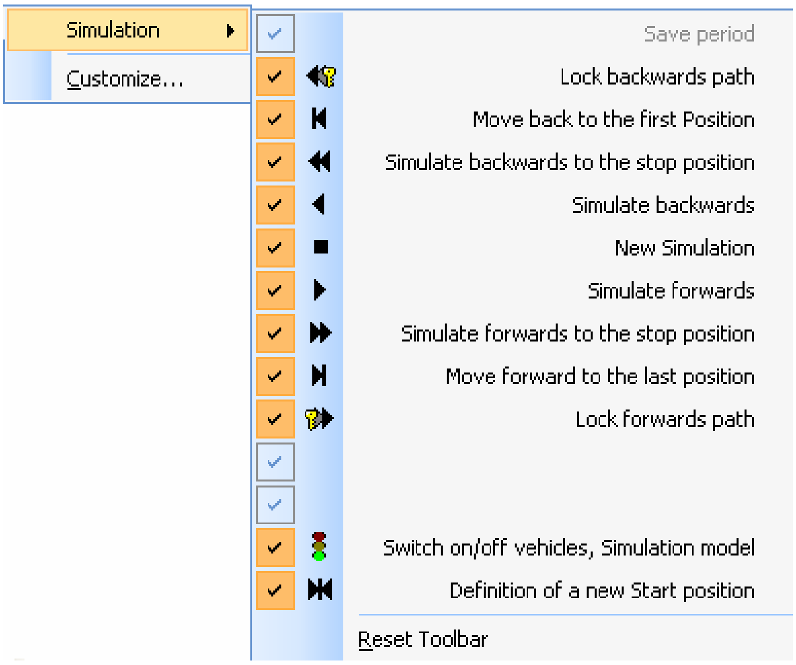
- **File** – Load and save files, printing operations
- **Vehicle** – Load, save and modify vehicles
- **Dynamics** – Specify initial conditions, sequences, scene data
- **UDS** – View data collected from a UDS vehicle data recorder
- **Impact** – Perform crash simulations
- **Options** – Accesses tables, diagrams, program settings, main screen view settings, selection of Side and 3D views
- **Graphics**–Scene bitmap operations, Scene drawing operations
- **?** – Access the Help file, which contains the Operating Manual and index.

The toolbars contain buttons with pictures or text (the same pictures or text from the appropriate menu options). All important menu options can be selected over the toolbars.

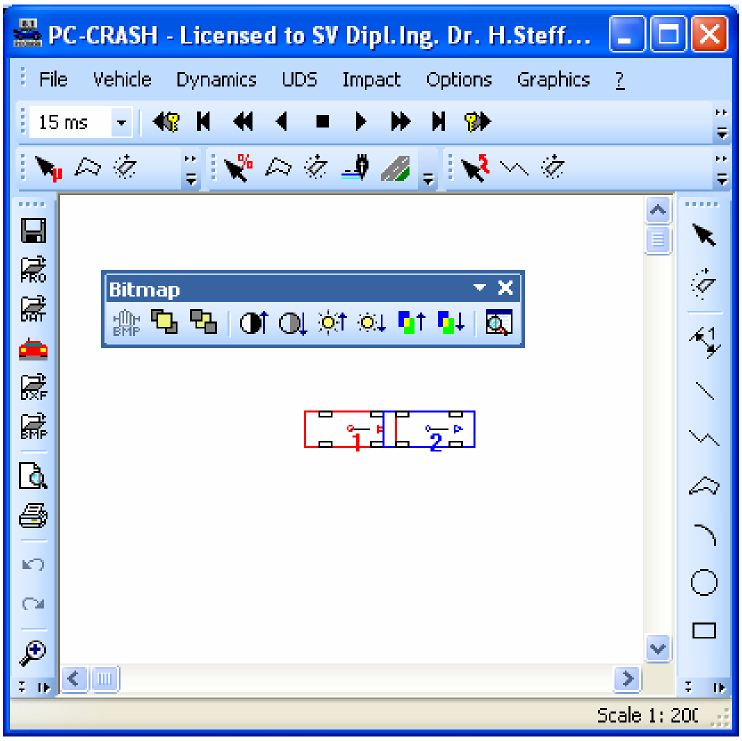
The toolbars can be extended by clicking with the left mouse button, if still further instructions are present, or be adapted over the symbol (Add or Remove Buttons).



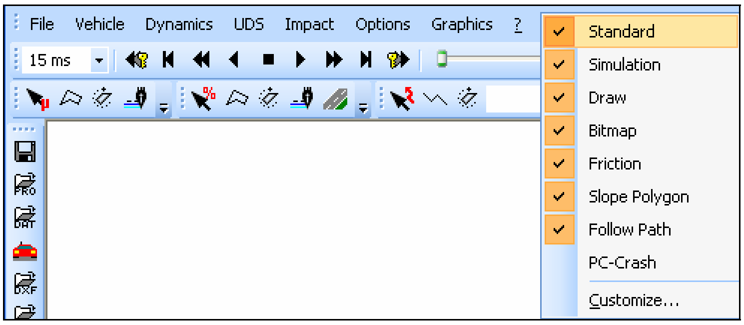
A submenu appears, in which the individual points can be activated or deactivated.



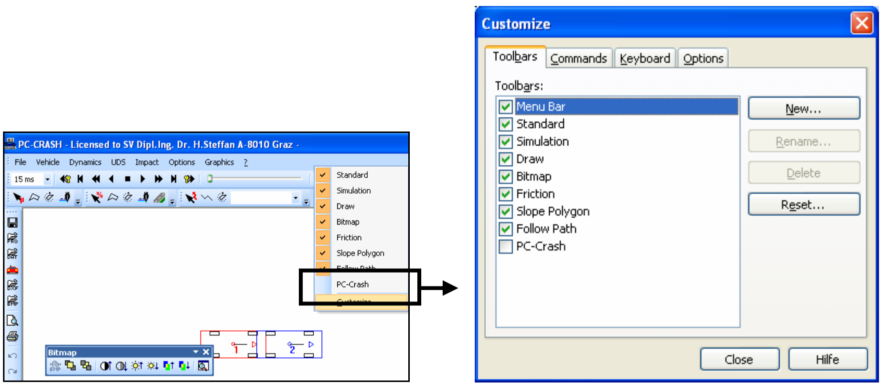
Using the symbol on the left side of the individual toolbars, these can be shifted or arranged. The mouse pointer changes and with held left mouse button the toolbar can be positioned. The toolbars can be arranged into the work area as individual toolbar by pushing into the work area or they can be assigned to the edges in the PC Crash window by shifting to the left or right.



The individual toolbars can be activated/deactivated by clicking with the right mouse button into the toolbar range in the activated submenu.

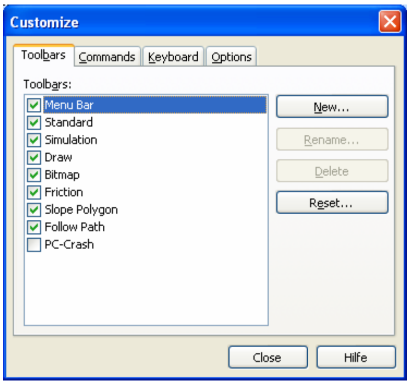


Using the option **Customize...**, the "Customize" window will appear and within this window user-defined toolbars can be generated and/or the existing toolbars can be adapted or changed.



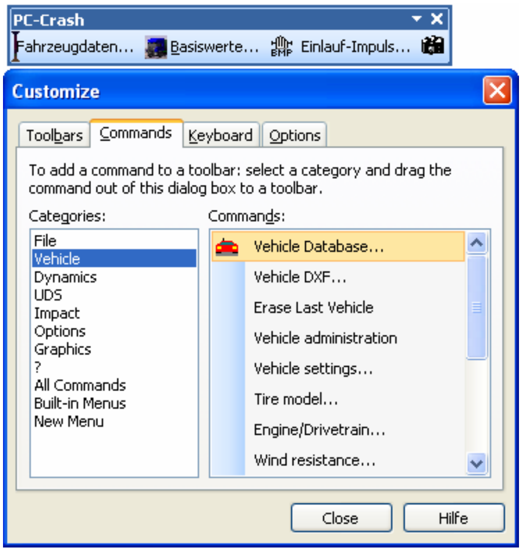
Toolbars

The individual toolbars can be activated and deactivated, using the New... button a new toolbar can be provided and arranged individual.

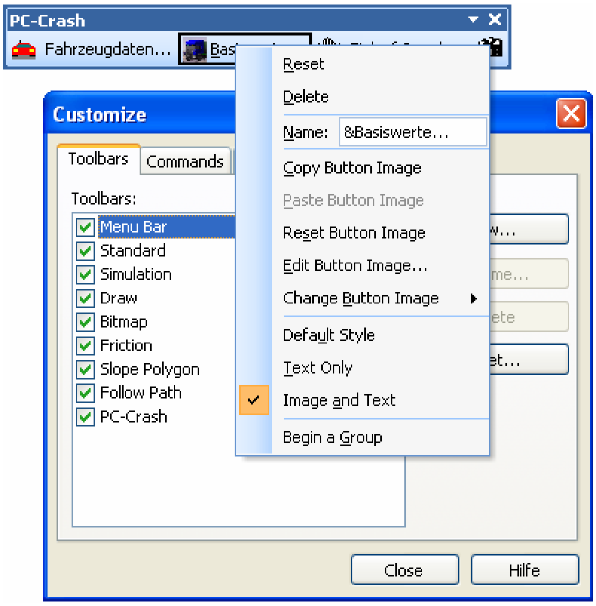


Commands

Commands can be shifted in the individual toolbars or menus. Also instructions can be shifted directly over the menus into the individual toolbars, with Strg-key held the command is copied, otherwise the command is only shifted and disappears thereby from the menu. This option works also in reverse, i.e. individual commands can be assigned to the menu options.

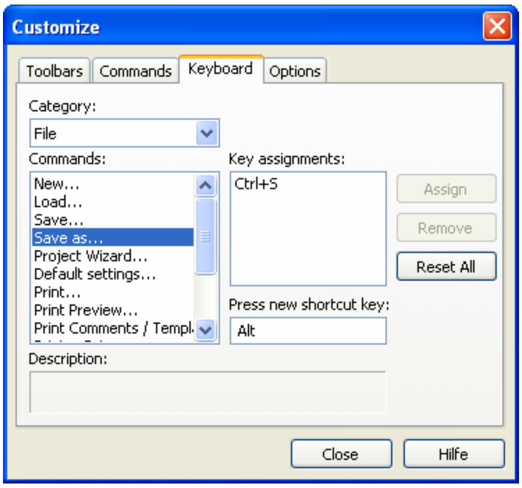


A **New Menu** can be generated and different menu options can be assigned (by shifting from other menu options or using the “Commands" window). The menu option can be designed or formatted additionally by clicking with the right mouse button on the menu option or the command.



Keyboard

Keyboard commands (e.g. Save as … with Ctrl+S in the picture below) can be assigned to individual commands



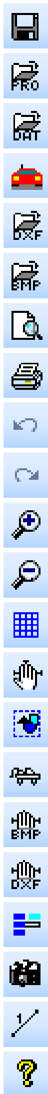
Options

Different design options for the toolbars

The most important menu options can be chosen with buttons on the **Standard toolbar**.

Place the cursor on each button to display a brief description on the screen.

The main toolbar contains following functions:



Save project under previously defined name

Load/save project

Load/save vehicle

Load vehicle from a vehicle database

Load/save DXF drawing

Load/save bitmap

Print preview

Print

Undo last action

Redo last action previously undone

Zoom in

Zoom out

Show grid

Pan drawing area

Zoom window

Move vehicle

Move bitmap

Move DXF drawing

Define display settings

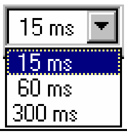
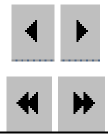
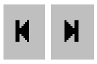
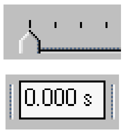
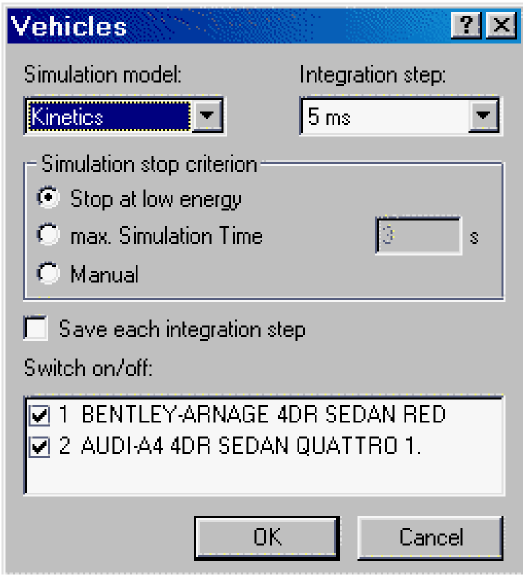
Set camera position

Measuring function

Help functions

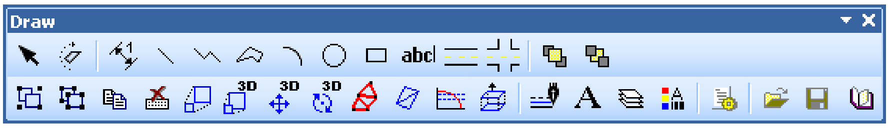
Additionally, most commonly used menu items can be accessed by right-clicking on the main screen.

The simulation options can be chosen with the buttons on the simulation toolbar. This toolbar can be opened or closed by clicking with the right mouse button into the toolbar range in the activated submenu.

	The screen display time interval is viewed and changed here. This does not affect the calculation integration time step, which is 5ms by default.
	The New Simulation button moves the vehicles to the start positions and erases the former simulation.
	The Forward or Backward Simulation buttons will start the time forward or backward simulation. These buttons should be used only when a new simulation is being performed. Single step forward/backward. Continuous simulation forward/backward. Pressing the right mouse button or the ESC key stops the simulation.
	Moves the vehicles to their start or end positions.
	Locks the simulation before time=0.
	Locks the simulation after time=0.
	The scroll bar is used to move the vehicles along their previously calculated paths of motion. A time can also be entered in the text box. Vehicles will be positioned according to the time specified.
	The Simulation Model button is for the selection of the simulation model (kinetic or kinematic), the integration time step (default = 5ms), the simulation stop criteria and the vehicles that are to run in the simulation. 

	Define a new start position. Current positions of active vehicles are defined as the new start positions if the No button is selected in the window that appears. 

Draw Toolbar



For a detailed description of the individual menu options see Draw Toolbar (page 222)

 Select, move

 Rotate selected

 Measure line

 Line

 Polyline

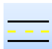
 Polygon

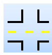
 Arc

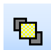
 Circle

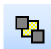
 Rectangle

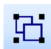
 Text

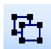
 Generate road element

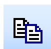
 Generate intersection

 Bring to front

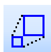
 Move to back

 Group selected

 Ungroup selected

 Copy selected

 Delete selected

 Scale selected

 Scale selected 3D

Move selected 3D

Rotate selected 3D

Triangulate selected

Measurement grid

Limit method

Extrude selected

Change linestyles

Change font, text color

edit Layers

Change selected

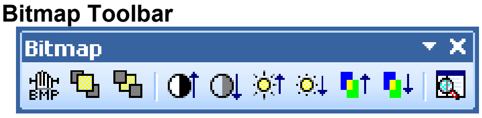
Snap

Load objects

Save selected Objects

Symbol library

Bitmap Toolbar



Move bitmap

Bring to front

Move to back

Rise contrast

Reduce contrast

Rise brightness

Reduce brightness

Rise transparency

Reduce transparency

Scale & Grid Spacing

Friction Toolbar

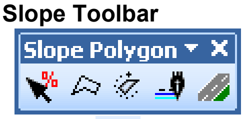
Select, move friction polygon

Define friction polygon

Rotate friction polygon

Linestyle

Slope Toolbar



Select, move slope polygon

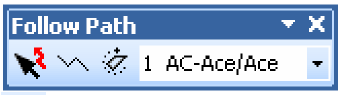
Define polygon polygon

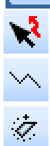
Rotate polygon polygon

Linestyle

Design 3D road objects

Follow Path Toolbar





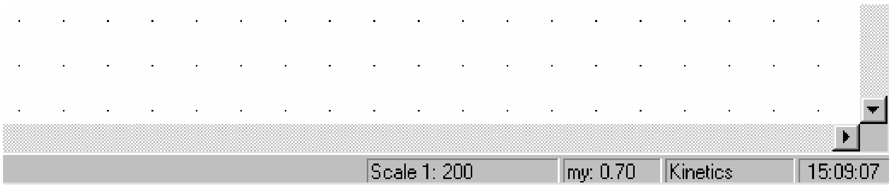
Select, move Follow Path

Define Follow Path

Rotate Follow Path

Select vehicle

The **status bar** at the bottom of the screen displays useful information for the user, and can be opened or closed by activating the menu option **Options - Status Bar**. Double-clicking on a particular area of the status bar allows the user to change the displayed value.



Hot Keys

Keyboard keys provide shortcuts to menu items, as follows:

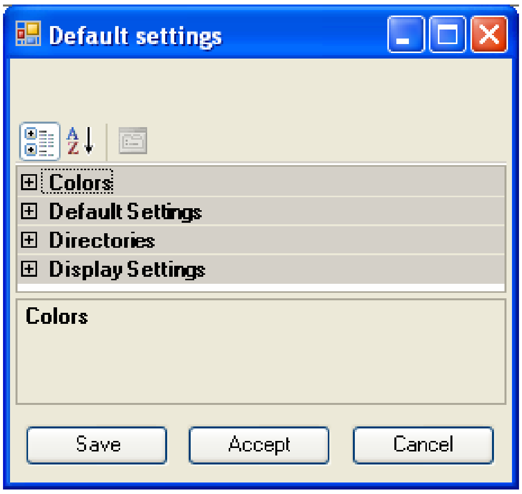
CTRL P Print

F1	Help
F2	Diagrams
F3	Zoom Previous
F4	Values
F5	Screen Refresh
F6	Sequences
F7	Position & Velocity
F8	Crash Simulation
F9	3D Window
F10	Kinematic Calculations
SHIFT F6	Display UDS
SHIFT F12	Copy Screen

Default Settings

There are two options to define the default settings.

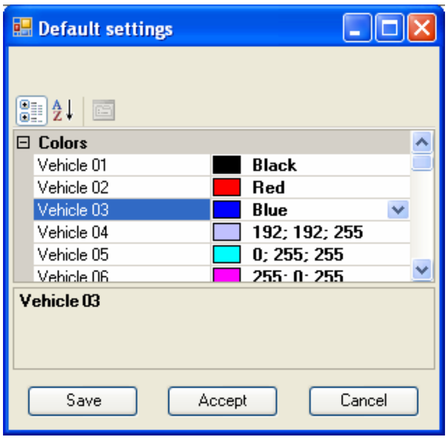
The first one is the menu option **File – Default options**. A dialog box appears with consists of 4 categories (Colors, Default Settings, Directories, Display Settings)



Using the Button sorting according to the categories and using Button alphabetically sort can be done. During the alphabetical sorting all basic adjustment options are indicated, during the sorting of categories main categories with the appropriate submenus are displayed. These can be extended and/or reduced via the symbols .

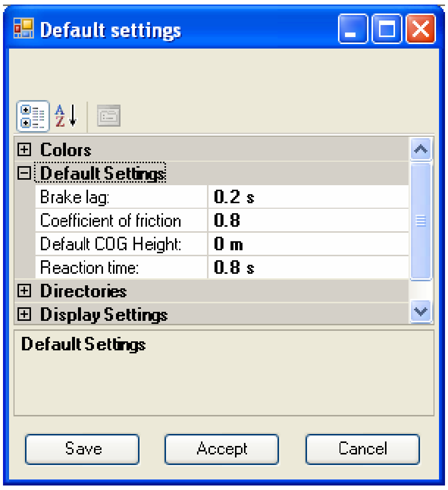
Colors

For the color adjustment of the different vehicles.

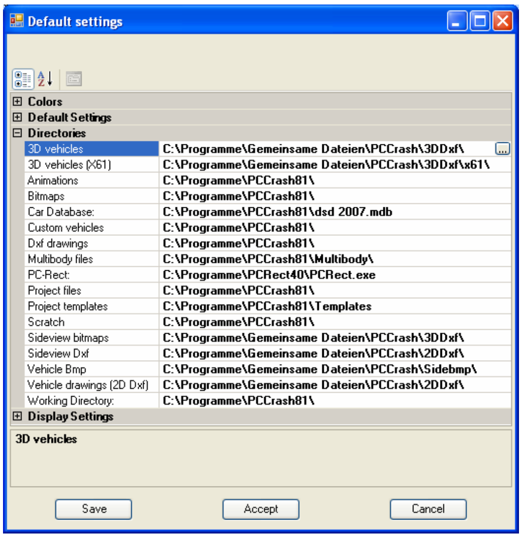


Default settings

Global defaults for PC Crash. Brake lag, global coefficient of friction, default center of gravity height and reaction time are defined. These values are defined global but can be adapted and changed however at any time via the appropriate menu options.



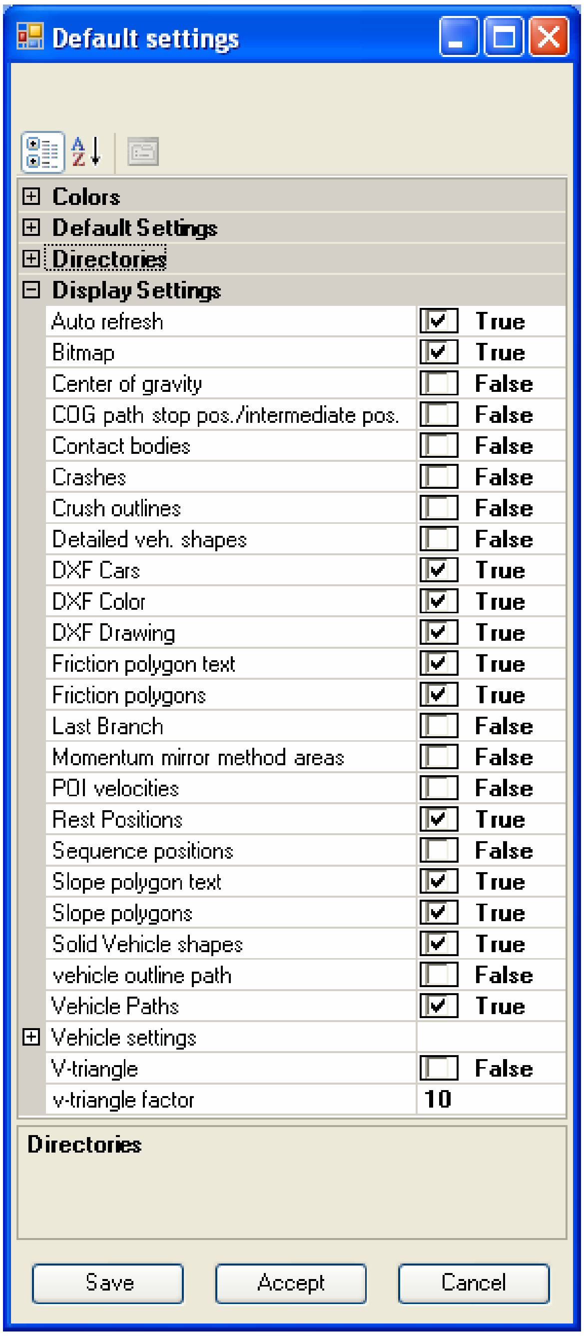
Directories



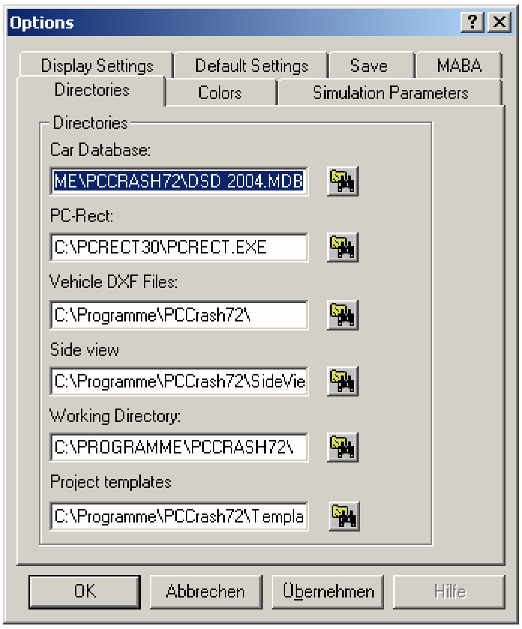
3D vehicles	<Vehicle> <Vehicle Dxf…> <File…> <3D Mapped> <Load 3D vehicle…>
3D Fahrzeuge (X61)	
Animationen	<Options> <3D Window> <Animation>
Arbeitsverzeichnis	
Bitmaps	<File> <Import> <Bitmap> <File> <Export> <Bitmap>
Dxf Zeichnungen	<File> <Import> <Dxf Drawing> <File> <Export> <Dxf Drawing >
Fahrzeug Bmp	<Vehicle> <Vehicle Dxf …> < File …> <Plan View> <Load Bmp …>
Fahrzeugdaten	<File> <Import> <Custom vehicle> <Datei> <Export> < Custom vehicle>
KFZ-Datenbank	<Vehicle> < Vehicle Database>
KFZ-Zeichnungen (2D Dxf)	<Vehicle> < Vehicle -Dxf…> <File…> <Plan View> <Load Dxf …>
Mehrkörpersysteme	
Photoentzerrung (PC-Rect)	<Options> <PC-RECT>
Projektdateien	<File> <Save as>
Projektvorlagen	<File> <Project Wizard…>
Scratch	
Seitenansicht Bmp	<Vehicle> < Vehicle Dxf…> <File…> <Side view> < load Bmp …>
Seitenansicht Dxf	<Vehicle> < Vehicle Dxf…> <File…> <Side view> < load Dxf …>

Display Settings

Parameters for the representation and the appearance of PC Crash. For detailed description see Display Settings beginning from page 213.

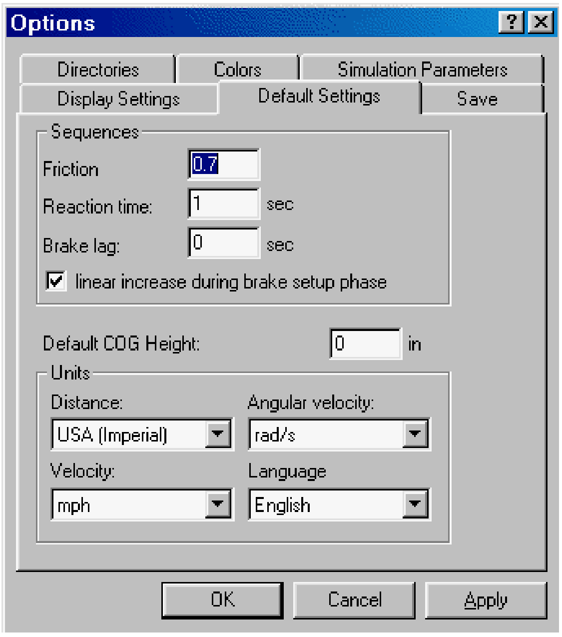


The second option for the definition of all PC-Crash's default directories, colors, simulation parameters, and settings is by selecting the menu option **Options** - **Options**. The first folder of the dialog box that appears is the Directories folder, where paths to various types of files can be specified.



Units

Both metric and U.S. units can be used in PC-Crash. The default units are metric (distance in meters, speed in km/h). To change to U.S. units, select the menu option **Options - Options - Default Settings**. Use the Distance drop down box to choose between metric distances (vehicle and scene dimensions in meters) and U.S. distances (vehicle dimensions in inches, scene dimensions in feet). Use the Velocity drop down box to choose between km/h, mph, m/s (meters/second) or ft/s (feet/second).



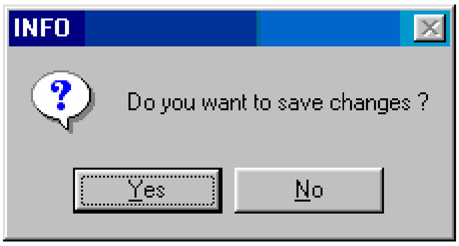
Press the OK button to accept the changes and close the dialog box (the Apply button incorporates the changes without closing the dialog box).

Starting a New Project

After PC-Crash is opened, a new project can be started. The following list contains most of the basic operations required to complete a project. They are shown in the recommended order that they are to be performed, although this can be left somewhat to the individual user's preference. The first six operations listed can be performed even with PC-Crash in the demo mode. Details on how to perform the operations can be found later in this chapter and the next chapter.

1. Each vehicle is loaded from a database or as a custom vehicle using the menu options **Vehicle – Vehicle Database** or **File – Import – Custom Vehicle** .
2. Any necessary changes to vehicle geometry or other parameters are made using **Vehicle – Vehicle Settings**.
3. This is a good time to choose a file name for the project and save it using **File – Save** . This button should be used at regular intervals when working on a project, although there is also an automatic file save feature.
4. Scene information can be inputted, by loading or creating a 2D or 3D DXF drawing, or loading a scanned drawing, aerial photograph or rectified photograph, and selecting scene friction properties.
5. The vehicles are positioned at the start points with the mouse using **Dynamics – Move/Rotate Vehicle** or by entering the X and Y coordinates using **Dynamics – Position & Velocity**.
6. Sequences that control the wheel braking, steering and acceleration forces, and other functions that affect vehicle motion such as driver reaction time, are applied to each vehicle using **Dynamics - Sequences**.
7. An impact analysis can be performed, using **Impact – Crash Simulation**, along with a post-impact trajectory analysis, using the Forwards Simulation tools . If the Collision Optimizer is to be used to solve for collision speeds and other parameters, the vehicle rest and/or intermediate positions have to be specified first using **Impact – Rest Positions** or **Impact – Intermediate Positions**.
8. After the impact and post-impact motion has been solved for, the pre-impact motion can be examined using the Backward Simulation tools . Backward simulations have some limitations, which are discussed later in this chapter.
9. A scale diagram of the main screen, a report of the input and output parameters, and tables and diagrams defining the vehicle motion can be viewed, printed or output as text files.
10. Animations of the simulation, from a fixed or moving camera position, can be rendered (PC-Crash 3D only).

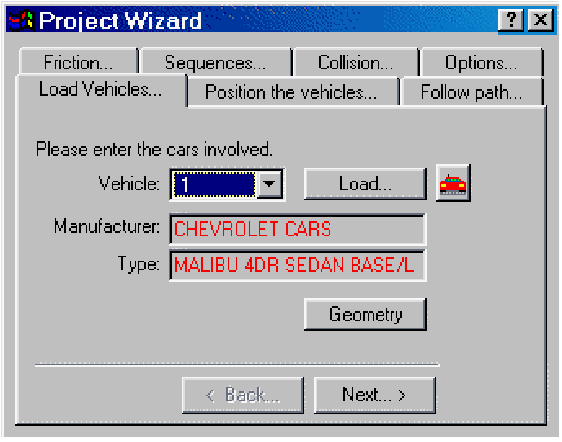
If the user wishes to begin a new project after another project has already been started, all previous input data can be erased by selecting the **File** - **New** menu option. If changes to the existing file were not saved, the program asks if the user wishes to save the changes.



If "Yes" is selected and the existing project has not been saved previously, the user will be prompted to enter a file name.

Project Wizard

The Project Wizard can be used as a guide when starting a new project. This feature leads the user through most of the tasks required to set up a new project. The Project Wizard is accessed by selecting **File – Project Wizard**. After choosing a project template, the following dialog box appears. Using the Next button after each operation is completed will open the dialog boxes in a logical order to complete a project.



Loading or Saving Projects

Existing projects are loaded using the menu option **File – Load** .

While working on a project, it is recommended that it be saved at regular time intervals. This can be done using the menu option **File** – **Save** . If a project name was previously defined, the project will automatically be saved to that file name. If no project name was previously defined, a new file name will be requested in a dialog box. After saving the project, the specified name will be displayed in the title bar. The current status of a saved simulation can be reloaded at any time, using **File** – **Load** .

File – **Save As** can be used to save the current project under a different file name. This is normally done when the user wishes to have several project variations of the same incident.

Autosave

The Autosave feature of PC-Crash can be used to automatically save the current working session every 1 to 10 minutes. Autosave is accessed using the menu option **Options - Options - Save**.

The Autosave feature writes to the file recover.pro, and therefore does not overwrite the working file. In case of a system or software failure, when PC-Crash is re-opened it will ask if you want to recover the last session. If this is done, use **File – Save As** to save the loaded Recover.pro file to the desired file name.

Main Screen Display

Display Settings

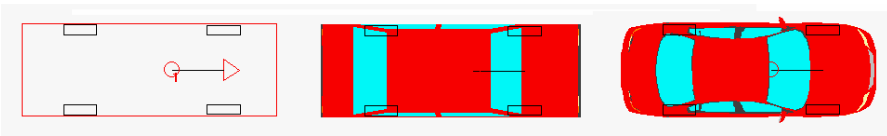
You can change the configuration of the display settings in PC-Crash in the menu option **Options** – **Options** - **Display Settings** or **File – Default settings**

Vehicle outlines (last position and last branch), trajectories of the wheels (full trajectories or just the visible tire marks) and center of gravity, sequence positions, crashes, velocity triangles, etc. can be displayed by clicking on the relevant check box. A single click activates the respective function and a further click deactivates it.

Vehicle Shape

In a simulation it is possible to show the vehicles in either a simple or a more complicated form. The default vehicle shape is Simple, which represents the body of each vehicle with a rectangular outline. A second available shape is Detailed, which adds lines for the windows and bumpers. This, however, increases the simulation refresh time, as there are more lines in the drawing. To select this option, check Detailed veh. shapes in **Options – Options - Display Settings** or **File – Default settings**. This shape is governed by the settings in **Vehicle – Vehicle Settings – Vehicle Geometry (Type)** and (in the case of automobiles) **– Vehicle Shape**.

PC-Crash also allows the user to attach realistic vehicle shapes to the simulation model vehicles, including 3D vehicle shapes, in the form of DXF drawings or bitmaps. These shapes can be loaded using the menu option **Vehicle** - **Vehicle DXF**.



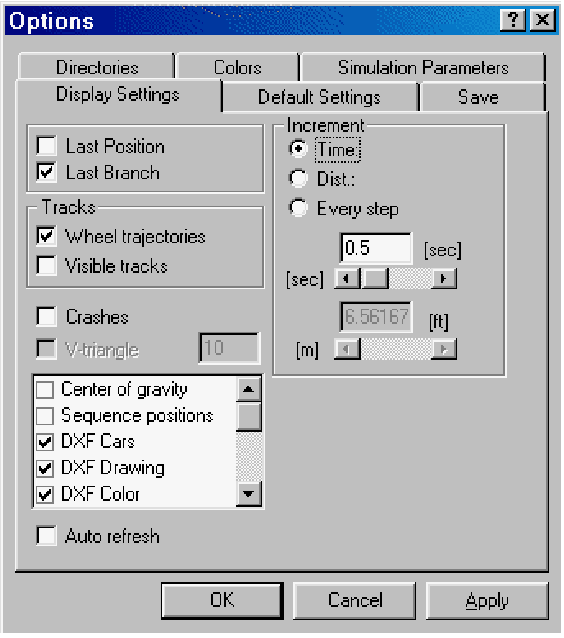
Simple **Detailed** **DXF Shape**

Performing the analysis with DXF vehicle shapes may result in lengthy refresh times. The shapes can, however, be turned off temporarily by de-activating DXF Cars in **Options – Options - Display Settings** or **File – Default settings**..

Last Branch

If Last Branch in **Display Settings** is activated, the vehicles will be displays at even time or distance increments, by selecting the appropriate option button.

Selecting the Every Step option button displays the vehicle outline every 15ms, so do not use this option unless DXF Cars and Detailed veh. shapes are both turned off.



Any changes in the Display Settings dialog box are accepted by closing the window with the OK button.

Refreshing the Screen

A screen refresh can be done at any time during the PC-Crash session by selecting the menu option **View - Refresh** or pressing the function key F5.

Automatic refreshing of the screen (after most operations) can also be activated with Auto Refresh in **Display Settings**.

If Auto Refresh is activated, the 3D Visualization window (**Options – 3D Window**) will also be automatic refreshed.

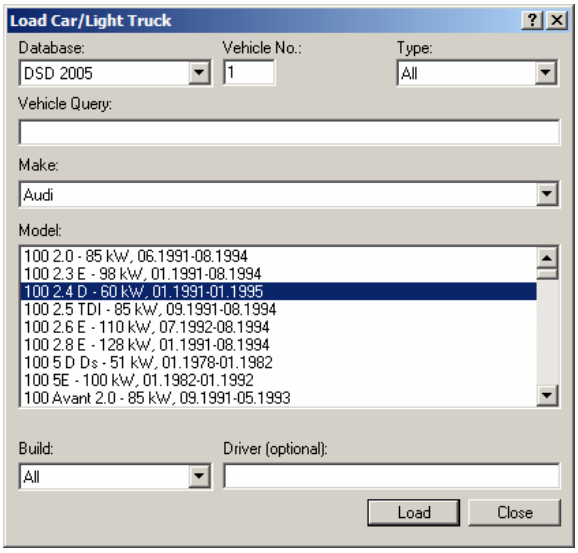
Loading Vehicles

The first step when simulating a new accident is the selection of the vehicles. PC-Crash is supplied with several vehicle databases that include most passenger cars and light trucks. Heavy vehicles, trailers, fixed objects, pedestrians, two-wheeled vehicles and vehicle occupants can be loaded as custom vehicles. The user can also create vehicles not included with PC-Crash by modifying an existing vehicle file.

Vehicle Databases

General

On the first start-up of PC-Crash, the user must select a vehicle database using the Change button beside the Car Database text box in the **Options** - **Options** menu option. After that, selecting **Vehicle** - **Vehicle Database**  will display the selected database in the Vehicle Database dialog box.



In PC-Crash interfaces to several database files have been implemented as follows:

- **DSD xxxx.mdb** - New European standard database of DSD.
- **KBA xxxx.mdb** – Database of the German department for motor vehicles.
- **DSDJapan2000.mdb** - New Japanese standard database of DSD.
- **ADAC95.dbf** - 1995 European database.
- **Spe****.dbf** - Canadian Specs Database. **** is the year of manufacture. Covers most cars and light trucks sold in North America.

Before loading the vehicle, the name of the driver can be entered in the Driver text box, if desired.

The Vehicle Database dialog box contains a text box for the vehicle number. This is the number by which the vehicle is addressed after loading. It is not possible to load vehicles out of sequence (i.e. changing the default number of Vehicle 1 to 2 prior to loading). However, it is possible to enter the number of an existing vehicle when loading a new vehicle. In this case, the new vehicle replaces the existing vehicle.

The Vehicle Database dialog box contains a drop down box for the vehicle database or year and list boxes for the vehicle manufacturer and model. The manufacturers and models available in the database are displayed alphabetically.

After selecting the database, click on the desired manufacturer name in the left list box. Then, click on the correct vehicle model in the right list box. A second dialog box appears, which lists all the database information for the chosen vehicle.

The vehicle is loaded into PC-Crash by pressing the Load button, and is immediately available for use in the simulation. Pressing the Close button closes both dialog boxes. Any vehicles that have been loaded will appear on the screen. Vehicle 1 is loaded with its center of gravity at point 0, 0 in the scene and subsequent vehicles are loaded 4m (13.1 feet) to the right of the previous vehicle loaded.

DSD

The DSD database is also automatically loaded into the same directory as the PC-Crash program files are. This database allows the user to search by specifying additional criteria.

The user can specify the date of manufacture. Once the user enters a manufacture date in the Build drop down list, only those vehicles that have been built on this date (year of manufacture is between ‘built from’ and ‘built to’) will be displayed.

Entering the vehicle make in the Vehicle Query text box will access that make in the left list box, which may be quicker than scrolling down to find the make. It is also possible to search for a certain vehicle. The Vehicle Query text has to start with the year of construction (YY or YYYY, e.g. 99 or 1999) or the manufacturer, whereby the manufacturer must be always entered. The inputs must be separated by blanks, the TAB key starts the search. No differentiation is made between large and small letters.

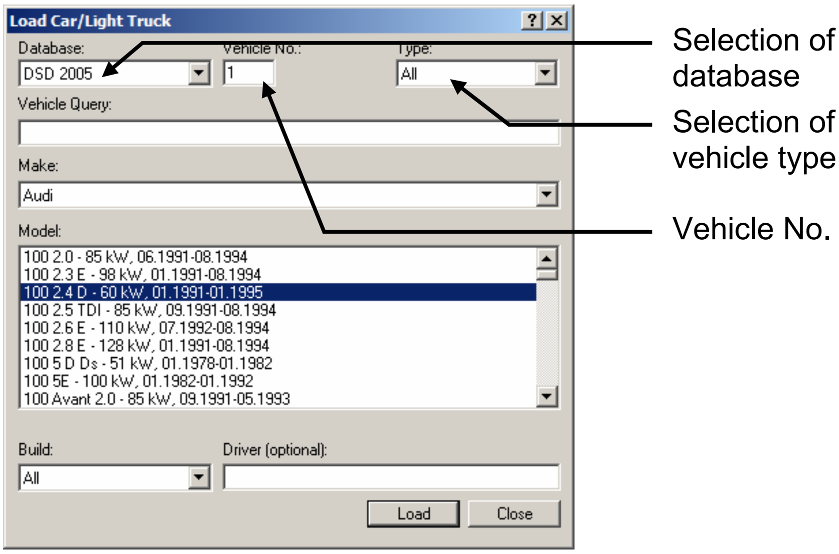
Search words: [(built YY or YYYY)] (manufacturer) [(search criterion)]

e.g. 96 bmw 520; 1991 MERCEDES 190; peugeot 40 (all vehicles with X40 kW and the 40X series are displayed).

It is also possible to display only vehicles of a certain type, as follows:

- Cars
- Trucks
- Trailers
- Busses
- Motorcycles

After selecting a vehicle type the list of manufacturers will be updated to include only those in the category selected.



The DSD database also contains more parameters - a reference to the 2D European vehicle sketches of Ratschbacher's database (purchased separately), engine torque diagrams and gearbox parameters.

These data are loaded automatically. The corresponding vehicle sketches are loaded if the correct path to the main directory of Ratschbacher’s database is specified in **Options – Options – Directories – Vehicle DXF Files**.

DSDJapan2000 and Vyskocil

The Japanese database DSDJapan2000 and Vyskocil databases are automatically loaded into the same directory as the PC-Crash program files are. The Vyskocil database is an older version of the DSD database, with fewer features. The Vyskocil database includes trucks and motorcycles. The DSDJapan2000 is for Japanese vehicles.

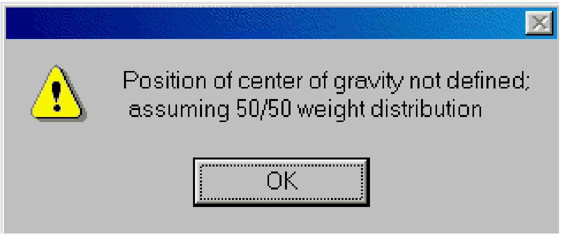
Specs

The North American Specs database is supplied in a separate directory on the PC-Crash CD and must be installed by the user onto the hard drive. The default directory it is installed in is named Specs, but this can be changed to suit the user's preference. The Specs database contains vehicle dimensions and weights of cars and light trucks. No power or transmission information, or motorcycle or heavier truck data is provided, as it is in some of the European databases.

To make the Specs database vehicles accessible, select any of the files in the Specs directory named Spe****.dbf (where **** signifies the model year) using **Options – Options – Directories – Car Database**. The only difference between the selection of these is that the year selected will be the default year that the vehicle database dialog box is initially set at. Any other vehicle year can be selected at any time when **Vehicle - Vehicle database** is selected, however, making any of the Specs vehicle data files available from the vehicle database dialog box.

Note: *If the user transfers the other supplied databases into the directory where the linked database is located (or vice versa), all of the supplied databases can be selected from the Vehicle Database dialog box without having to change the default directory in the* ***Options – Options – Directories – Car Database*** *menu option.*

Example*: Load a 96 BMW Cabrio 2.5 from the DSD database. Choose this vehicle in the Vehicle Database dialog box by selecting the year first (1996) from the drop down box, then the manufacturer (BMW) from the left list box, then the model (Cabrio 2.5) from the right list box, use the Vehicle Query option…. An information message will appear to indicate that the position of the center of gravity is not defined and a 50/50 weight distribution has been assumed. This is currently true for some vehicles in the database.*

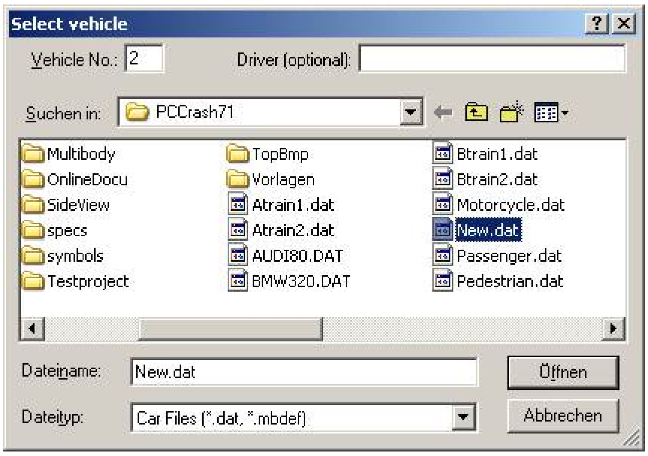


If a different weight distribution is desired, it can be changed in the Vehicle Geometry dialog box, which appears when the menu option ***Vehicle*** *-* ***Vehicle settings – Vehicle Geometry*** *is selected.*

Custom Vehicles

Vehicles not contained in a database and objects such as walls and trees must be loaded as a custom vehicle. A number of custom vehicles (*.dat), as well as multibody pedestrians and motorcycles (*.mbdef), are included in the directory PC-Crash is installed in. To load a custom vehicle, select **File – Import - Custom Vehicle** .

Example*: Replace the 96 BMW Cabrio 2.5 loaded from the DSD database as Vehicle 1 by the vehicle New.dat.*



Vehicle Construction

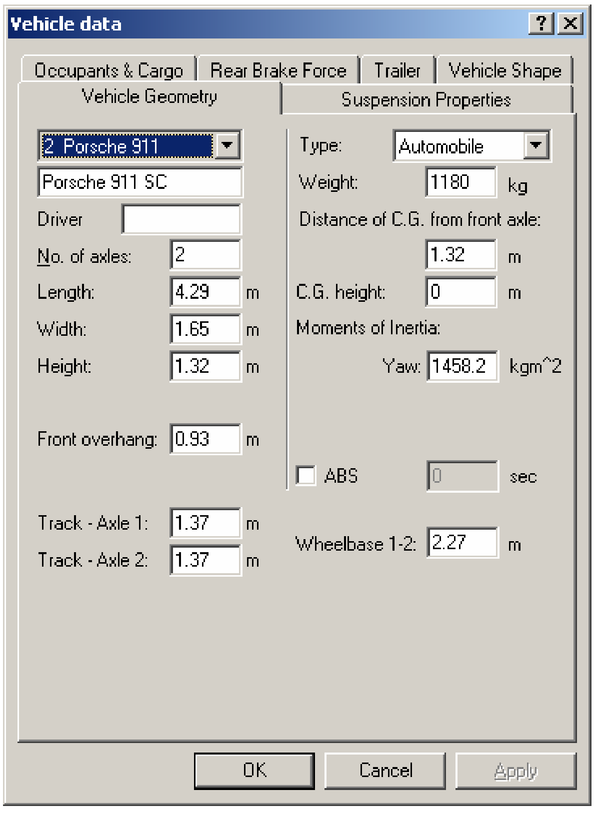
If the desired vehicle is not contained in the supplied PC-Crash databases or custom vehicles, use **File – Import - Custom Vehicle**  and select the vehicle "New.dat" (or any other vehicle) out of the listed vehicle types.

The next step is to input the correct geometry and load parameters for this vehicle. Open the **Vehicle** - **Vehicle settings – Vehicle Geometry** menu option. Ensure that the vehicle number is the correct one and input the proper parameters into the applicable text boxes.

Example: *Load the vehicle New.dat as vehicle number 2 and modify its data according to the data for a Porsche 911 SC, as follows:*

Name :	*Porsche 911 SC*
Length :	*4.29 m*
Width :	*1.65 m*
Height :	*1.32 m*
Wheelbase :	*2.27 m*
Front Overhang :	*0.93 m*
Track Width :	*1.37 m*
Weight :	*1180 kg*
Dist. center of gravity to front axle :	*1.32 m*
Height center of gravity :	*0 m*

Note: *Selecting a 0 C.G. height means that the program calculates the simulation with planar 2D vehicle models.*



The Yaw (z-axis) angular moment of inertia is calculated by the program based on the vehicle size and mass. The user can change this value by overwriting it in the corresponding window (See the **Vehicle Geometry** menu description in the next chapter).

Selecting a vehicle C.G. height of 0 (the default in PC-Crash) means that the simulation is calculated with planar 2D vehicle models that can have yaw, but no roll or pitch motions (on a 2D surface). If a positive C.G. height is selected, values are calculated for the vehicle moments of inertia about the x and z axes as well, and roll and pitch are calculated. The two calculation models are the same, but there are no forces that can cause roll or pitch when the C.G. height is 0.

Finally, you can select whether the vehicle is to have ABS (Anti-lock Braking System). If you select ☑ ABS an additional dialog box will appear in which you can enter the ABS cycling time. A typical cycling time would be approximately 0.07-0.10 seconds.

To close the dialog box and confirm all entered parameters, click on the OK button. If you click on the Cancel button, the previous parameters are retained.

Vehicle Settings

Vehicle geometry and other parameters can be viewed and changed by selecting the **Vehicle** - **Vehicle Settings** menu option. This opens a group of dialog boxes that contain the following:

- **Vehicle Geometry** – Basic dimensions, inertia, anti-lock braking. The basic dimensions and weight are normally the only information loaded from the vehicle database or list of custom vehicles. All other data contained in Vehicle - Vehicle settings is calculated by the program and should be checked by the user to ensure that values are realistic.

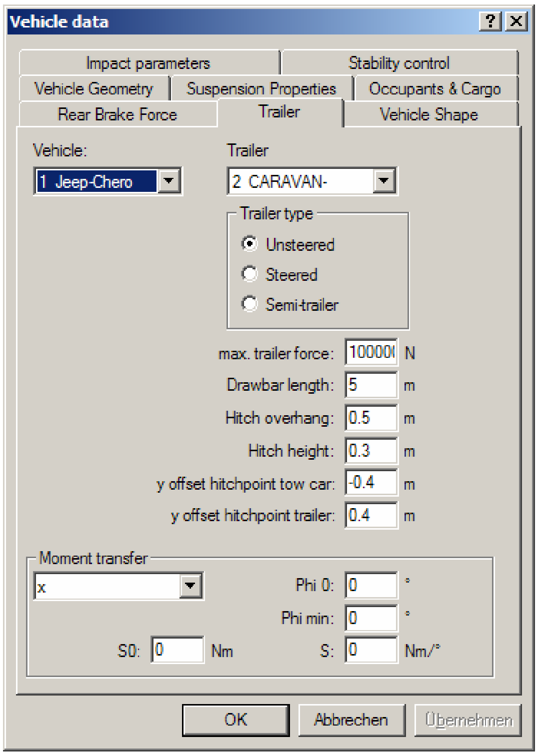
- **Suspension Properties** – Spring and damping characteristics, body-ground contact values for rollovers.

- **Occupants & Cargo** – Passenger, trunk and roof-rack loads.
- **Rear Brake Force** – Rear brake proportioning valve settings for 3D vehicles.
- **Trailers** – Trailer connection criteria.
- **Vehicle Shape** – Basic vehicle body dimensions can be modified here when a 3D shape has not been attached.
- **Impact parameters** – Vehicle specific default parameters for automatic impact simulation.
- **Stability control** – Parameters for simulating stability control devices.

Trailers

The trailer model In PC-Crash is force-based. The vehicle-trailer connection forces necessary to have the same acceleration of the vehicle and the trailer at the hitch point are calculated. Driving stability, road space requirements and impacts to the trailer and tow vehicle can be simulated.

Trailers are loaded as a custom vehicle (select **File - Import - Custom Vehicle** ) or using the DSD database. After loading a trailer, its geometry has to be checked or modified to suit the desired values. Use **Vehicle - Vehicle settings - Trailer** to specify the hitch data.



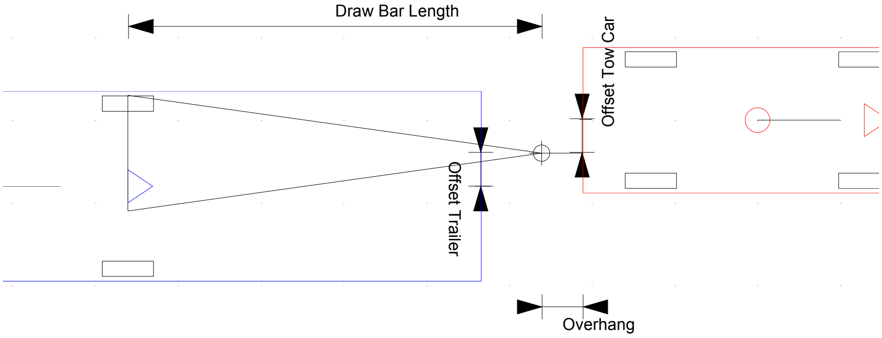
The vehicle that is to be connected with the trailer is selected in the Vehicle drop down box of the Trailer dialog box. The matching trailer is then chosen in the Trailer drop down box. For a second trailer on a tow vehicle, select the first trailer in the vehicle drop down box and the second trailer in the trailer drop down box.

In order to disconnect a vehicle from the trailer the word "None" has to be chosen in the Trailer drop down box.

The drawbar length (distance of the front trailer axle to the hitch point) and the hitch overhang (distance from the rear of the vehicle to the hitch point) have to be specified.

In a three dimensional simulation (C.G. height defined for car and trailer) the height of the trailer hitch above the road also has to be specified. Other parameters can be chosen to specify torsional resistances for the trailer hitch about the x, y and z axes.

If the hitch point is not at the center of the truck or trailer, the offsets may be specified individually as shown in the drawing below:



If you click on the OK button in the Trailer dialogue box, the connection is activated and the trailer is connected with the vehicle.

PC-Crash automatically checks whether the starting situation of the trailer and tow vehicle correspond in position and velocity. If not, all motion parameters of the trailer are recalculated.

***Note:** For trailers, rearward simulations are not possible in PC-Crash. Collisions can be performed between a trailer and a vehicle, between a tow vehicle and another vehicle, between a trailer and its tow vehicle or between two trailers. A kinematic path simulation cannot be performed for a tow vehicle and trailer.*

The start conditions for the trailer are determined automatically by the program, based on those of the tow vehicle.

Due to the calculation method being based on a set numerical accuracy, small location differences between tow vehicle and trailer at the hitch point can develop in long simulations. These differences do not influence the driving stability. If the distance exceeds 10 cm (4 inches), all the definitions should be checked once more and the simulation repeated.

Saving a Vehicle

Vehicles can be saved in individual files for future use. To do this, select **File - Export** - **Custom Vehicle**.

First, change the vehicle number in the Vehicle No. text box to suit the vehicle being saved.

Next enter the name of the driver in the Driver text box (optional) and the name of the vehicle in the Vehicle text box.

Press the Save button. Now the newly stored vehicle will be available as a custom vehicle for future use.

***Example**: Save the Porsche using the file name 911.dat*

Deleting or Replacing a Vehicle

If more than one vehicle is loaded into PC-Crash, the last vehicle loaded can be deleted at any time. This is done with the menu option **Vehicle** - **Erase Last Vehicle**.

***Example**: Delete the Porsche.*

Only the vehicle with the highest number can be deleted. However, vehicles with lower numbers can be replaced using the menu option **File - Import - Custom Vehicle** or **Vehicle - Vehicle Database**. Before loading the new vehicle, change the default vehicle number to that of the vehicle to be replaced.

Attaching 2D and 3D Shapes to Vehicles

Vehicle 2D or 3D drawing shapes and 2D bitmaps can be attached to vehicles, for a realistic appearance on the main screen and, in the case of 3D vehicle shapes, in the 3D window and rendered animations.

To load a vehicle 2D top view or 3D drawing shape, or a top view bitmap, select **Vehicle – Vehicle DXF – File – Plan View - Load DXF** or **- Load BMP**.

Scene Data Input

2D Scenes

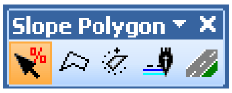
2D Scene data can be entered as follows:

- **Import a Scene Drawing** – A scene DXF drawing can be loaded using **File – Import – DXF Drawing**. Note that PC-Crash assumes the drawing was drawn using meters as units. Other drawings must be scaled accordingly, either before importing or after, using the Scale tool in the Draw Toolbar.
- **Create a Scene Drawing** - Use the Road and Intersection tools in the Draw Toolbar to build a scene in PC-Crash. Crosswalk lines can be drawn on the roads using the line draw and modify tools, and road markings, highway signs and other shapes can be imported with the symbol library tool. Other tools can be used to rotate and scale objects, apply text, etc.
- **Import a Scene Bitmap Image** – A bitmap image of a scene can be loaded using **File – Import – Bitmap**. The image can be a scanned drawing, an aerial photograph or a photograph rectified from PC-Rect. Images not brought in directly from PC-Rect must be scaled, using **Bitmap – Scale**.

Note: *Some DXF scenes drawn by surveyors may have the drawing origin at a large numbered coordinate, such as X, Y = 20 000, 20 000. In these cases, the drawing will not be visible in PC-Crash unless* ***Graphics – Zoom All*** *is selected. In some cases, the coordinates may be so large that the drawing will be outside the maximum viewing area possible in PC-Crash. It is thus generally advisable to place the drawing close to the point 0, 0 in a drawing program prior to importing into PC-Crash.*

3D Scenes

3D Scenes can be used in PC-Crash. In order to have the vehicle tires recognize the individual polygons of the 3D surface, the scene must be imported or created as 3D Road Object with triangular surface polygons using **Dynamics – Define Road Slope** and/ or the **Slope Polygon Toolbar**



.

Importing a Surfaced 3D Scene Drawing

A 3D scene DXF drawing can be imported directly as a 3D Road Object using the Generate 3D Road Object button and then Import... (change the "Files of type" to DXF). The 3D drawing must include all required surfaces, in the form of triangular polygons with normal vectors facing upwards.

Note: *If a 3D scene is imported using **File – Import – DXF Drawing**, the vehicles will not recognize the 3D surfaces. However, it is often useful to import a 3D scene twice – once as a 3D Road Object and once as a 3D DXF drawing. The DXF drawing can be embellished with road lines and other features such as trees that are not suitable for the 3D Road Object conversion. In this case, turn off Friction Polygons in **Display Settings**, which turns off the display (but not the function) of all Friction and Slope polygons. In this case, only the embellished 3D drawing shows on the main screen and in the 3D window, but the vehicles will still follow the 3D Road Object surfaces.*

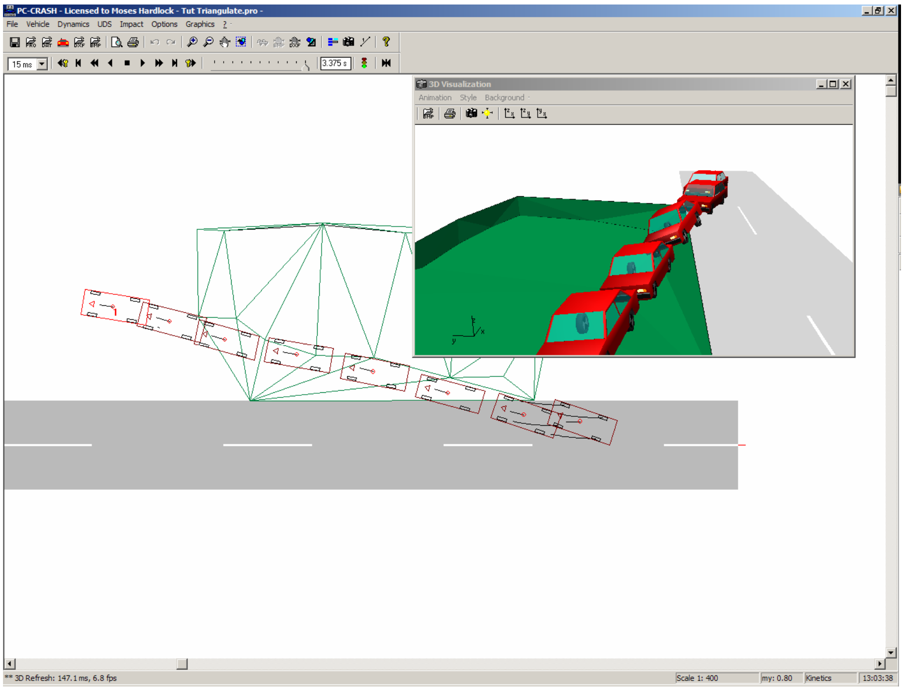
Importing a Survey DXF

A DXF drawing of surveyed points in 3D space can be brought in using **File – Import – DXF Drawing**. Start a New Project before doing this, so no vehicles or other objects will be in the 3D drawing subsequently created and exported.

Open the Draw Toolbar and select the points desired to define the 3D surfaces (see the next chapter for Draw tool operations). Then, by selecting Triangulate, 3D triangular surfaces will be automatically created between all selected points.

A massage box will appear asking if the triangulated area should be used as slope polygon.

If you choose Yes the triangulated area will be used as slope polygon and can be assessed using **Dynamics – Define Road Slope**.



If you choose No the triangulated area is handled as DXF drawing, and once all desired surfaces have been created, save the drawing using **File – Export – DXF Drawing**. Finally, import it into the desired project as a 3D Road Object using  - Import... .

Creating a 3D Scene

3D scenes can be created in PC-Crash as follows:

Slope Polygons

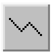
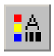
Using the  tool in **Dynamics – Define Road Slope**, one or more sloped planes can be created. Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points of the polygon, click the right mouse button to close the last side of the polygon. A dialog box appears, allowing you to enter the desired elevation and slope(s). The first slope polygon point drawn is defined by a red circle on the main screen and is the one for which the elevation (Zref) is specified. To add additional points to a slope polygon select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

If it is desired that several polygons are to be adjacent to one another with no space between, Snap should be turned on before drawing them. Snap and Snap Spacing can be accessed by the Snap-Button  of the Draw toolbar.

3D Road Objects

Complex multi-faceted 3D road profiles, with sloped banks or ditches, can be created using the Generate 3D Road Object tool . See the description of this menu item in the next chapter.

Contour Lines

3D scenes can be drawn from scratch. This is done by defining contour lines, using the 2D drawing tools  or  in the Draw toolbar. After being drawn, the elevations of each complete contour line can be adjusted up or down using the Shift Selected 3D menu option (right-click in the Draw Toolbar title bar to access this menu). Individual points can be adjusted using the Change Object window  of the Draw toolbar, where the X, Y and Z coordinate of each point can be changed.

When all the lines and points have been adjusted to the correct locations, follow the procedure outlined in the previous section "Importing a Survey DXF" to triangulate and/or export the drawing, and then import it as a 3D Road Object.

Friction Polygons

Areas of different friction in the scene can be defined by drawing polygonal areas, using **Dynamics - Define Friction Polygons** and/or the **Friction Toolbar.**

.

 is for defining a friction polygon. Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points of the polygon, click the right mouse button to close the last side of the polygon. A dialog box appears, allowing you to enter the desired friction coefficient. To add additional points to a friction polygon select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

Note: *Friction and slope areas are handled and calculated due to the order they are created, e.g. if two friction areas have an overlap the values of the first defined friction polygon are used for the calculation in the overlap area.*

Defining Initial Vehicle Conditions

Position and Velocity

The initial position and velocity of each vehicle can be defined using the menu option **Dynamics** - **Position & Velocity.**

The drop down box at the top of the Position & Velocity dialog box is used to choose the vehicle number for which the parameters are indicated and/or changed. It contains a list of all loaded vehicles.

Example: *Change the parameters for Vehicle 1 in the Position & Velocity dialog box. Using the TAB key or mouse, move the cursor to the applicable text boxes to input the coordinates of the center of gravity. Specify -2.5 for the x-coordinate and 0 for the y-coordinate in the given example. Input a start speed of 50 km/h in the Velocity text box. Press the OK button to close the dialog box.*

The Tow Truck Tool

PC-Crash offers an additional method of positioning the individual vehicles. Select **Dynamics** - **Move/Rotate Vehicle** . The mouse cursor will change into the shape of a tow truck. The vehicles can then be moved by clicking on them with the cursor at the center of gravity (max. distance to the center of gravity: 5 feet or 1.5 meters).

While keeping the left mouse button pressed, the vehicle can be moved with the mouse. When clicking on or near the vehicle at a distance between 5 feet and 16.5 feet (between 1.5 and 5 meters) from the center of gravity, the vehicle can be rotated.

Example*: Move and rotate Vehicle 1 on the screen using* .

Positioning Vehicles on 3D Surfaces

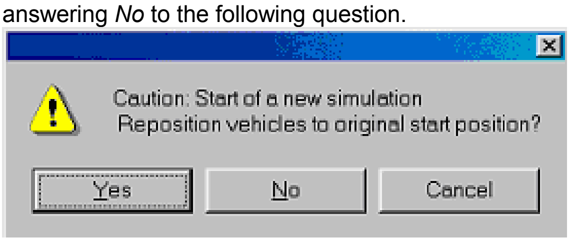
If the vehicle is placed on a 3D road surface or slope polygon with the Tow Truck tool , the vehicle elevation, pitch and roll will be calculated automatically, based on the slope polygon the vehicle's center of gravity is located over. The New Simulation button should be pressed after placing the vehicle to also position the tires on the surface correctly.

For cases where the vehicle's wheels are over different polygons than the center of gravity at the start, the user must "drive" the vehicle on the 3D surface until its suspension and wheels reach a steady-state condition. This is achieved by doing the following:

1. Move the vehicle back from the desired start point a short distance with the Tow Truck tool . About 3m or 10 feet is generally sufficient.

2. Run the simulation forward using the button with the vehicle at the selected start velocity until the vehicle reaches the desired start point. Check that the vehicle traveled a distance great enough to reach tire-ground steady-state conditions. The best evidence of this is a horizontal Tire Normal Force curve for each of its wheels in the Diagrams window.

3. With the button define the current vehicle position as the new star point by answering *No* to the following question.

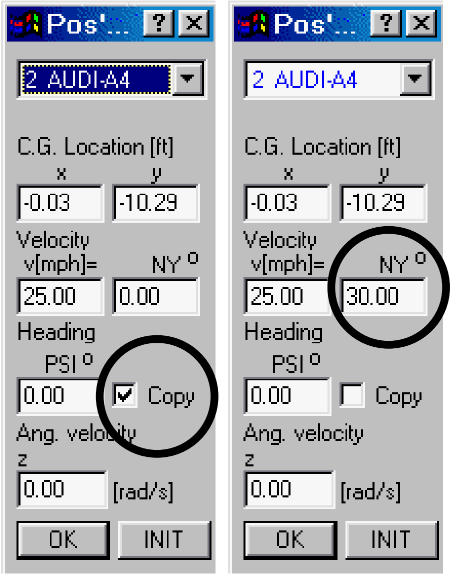


4. In the Position & Velocity dialog box, with time=0 check that the velocity is at the exact desired value and change it, if necessary. Do not change any other values.

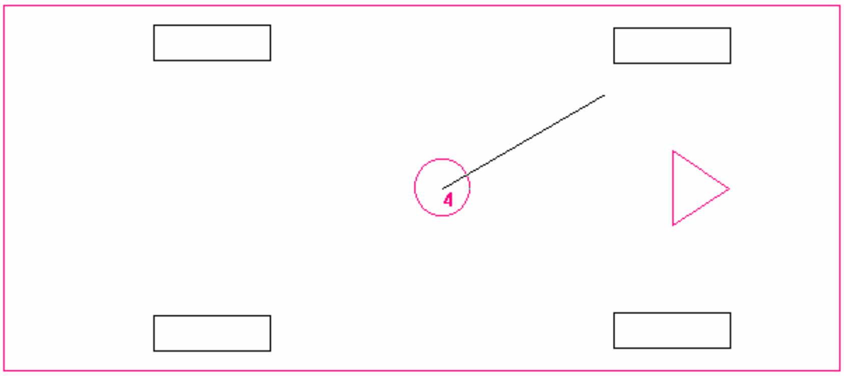
5. The simulation can now be run with the vehicle initial conditions at the correct values.

Initial Yaw

If Copy in the Position & Velocity dialog box is activated, then the velocity direction (NY°) will be automatically correlated with the heading (PSI°) of the vehicle (i.e. no yaw). If this box is turned off, the vehicle heading and the velocity vector can be changed independently, resulting in an initial yaw angle.



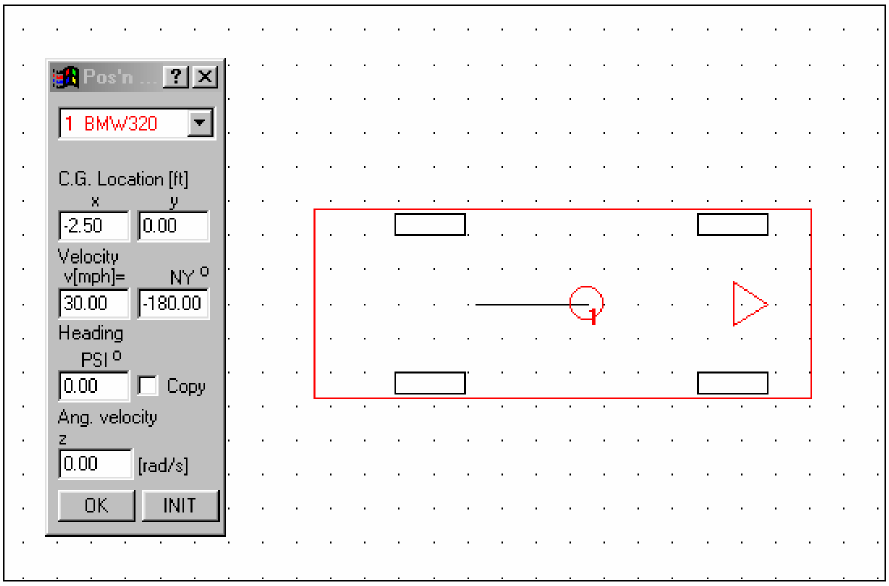
The line extending from the vehicle's center of gravity on the screen indicates the direction of the velocity vector. The direction of this line changes during the simulation to suit the actual velocity direction at any instant.



If the Dynamics - Position & Velocity dialog box is left open, the position and velocity values of the vehicle at any point in the simulation can be viewed.

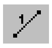
Backing up

Driving backwards can be simulated in PC-Crash easily. In the Position & Velocity dialog box, turn off Copy and change the direction of the velocity vector (NY°) by 180°, so that the velocity vector points to the rear of the vehicle.



Scene Distances and Scaling

Measuring Distances

Select **Options** - **Measure**  to measure distances between points on the screen.

Move the cursor to the point where you want to start the measurement, then press and hold the left mouse button. As the mouse is moved the distance and angle values appear in the Measure window. The connecting line is also shown on the screen. When the left mouse button is released the values are stored in the window until you press the left mouse button again. Close the Measure window by clicking on the close symbol (**X**) in its title bar.

Grid

Distances can be estimated by using the **Options** - **Grid** menu option. This covers the screen with a grid of equal density.

The default grid density is one meter. You can change the density of the grid using **Graphics** - **Scale & Grid Spacing**.

Scale

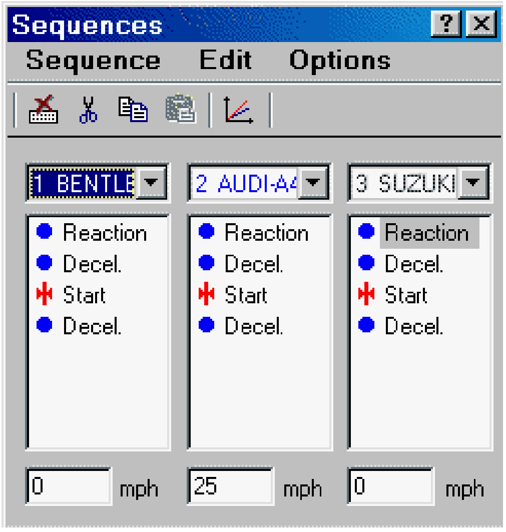
On starting PC-Crash the default scale is 1:200. **Graphics** - **Scale & Grid Spacing** can be used to change the scale of the screen and scene printout to any value desired between 1:1 and 1:1,000,000. This option can be more easily accessed by double-clicking in the scale area of the status bar at the bottom of the screen.

The toolbar buttons "Zoom in"  or "Zoom out"  can also be used to change the scale in even increments.

Motion Sequences (F6)

In PC-Crash different motion sequences for each vehicle, such as braking, acceleration, steering and driver reaction can be defined.

Select **Dynamics - Sequences** (F6) to access the Sequences dialog box, in which different sequences can be combined to reflect the driver's actions for each vehicle.



After loading a vehicle four sequences are specified for it by default: **Reaction**, **Decel**, **Start** and a further **Decel** sequence.

Sequences above the **Start** sequence in the Sequences window are before time=0 (which is normally pre-impact) and sequences below it are after time=0 (usually post-impact). Thus, by default there is a driver reaction, pre-impact braking and post-impact braking sequence for each vehicle, which covers most normal collision reconstructions.

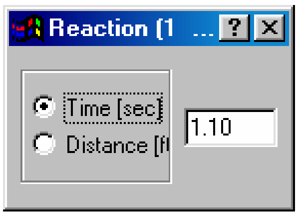
Double-clicking on a selected sequence accesses the sequence data for viewing and changing.

To create a sequence, select the Sequence menu in the Sequences dialog box. The new sequence is automatically placed after the current active (highlighted) sequence. Be careful to ensure that the new sequence is placed in the column of the intended vehicle.

To move sequences select a sequence with the left mouse button and drag the sequence symbol <u>onto</u> the sequence that the moved sequence is to go after.

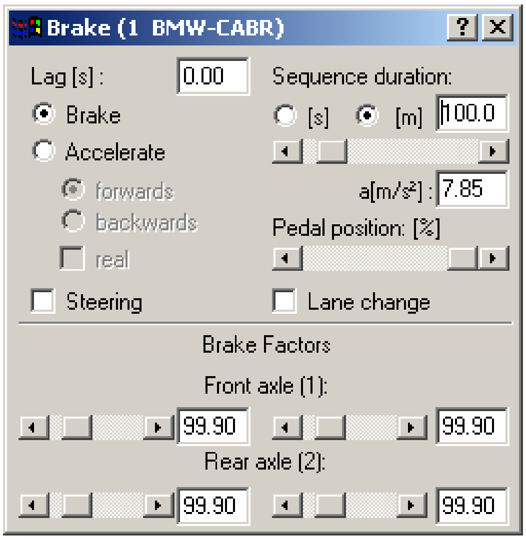
Example: *Create a second reaction sequence. Use the menu option* ***Sequence - Vehicle/Driver - Reaction****.*

In order to enter the correct parameter for the Reaction sequence, move the mouse to the Reaction sequence and double-click on it. A Reaction dialog box appears, which has the Time option button activated by default.



Now modify a Decel. sequence. Double-click on the post-impact Decel. sequence to input the parameters for this sequence. The Brake dialog box appears. The option buttons on the left side select whether the sequence is for braking or acceleration.

For Vehicle 1 input a brake lag time of 0.00 seconds and change the brake pedal position to full braking. Leave the default sequence length at 100 m, which is more than enough to allow braking to a stop from the selected speed of 50 km/h. Specify this speed either in the Sequences dialog box or the Position & Velocity dialog box. By closing the sequences dialog box all of the entered parameters are accepted. Using *the Forward button in the Simulation toolbar, run the simulation from the pre-braking speed of 50 km/h to rest.*



*In the **Demo-version** this process is locked if any changes are made. However, you can continue by loading the project file EXAMPLE1.PRO (this shows a fully-braked vehicle with a tire-to-roadway coefficient of friction of 0.7 decelerating to a stop from 30 mph). Using **File – Load**, select the project EXAMPLE1.PRO from the Examples subdirectory of the PCCrash73 directory.*

For further details on Sequences, refer to the **Programming Sequences** chapter.

Collision Analysis

Preparing for a Crash Simulation

PC-Crash allows crashes to be integrated into the simulation at any time so that the entire process of an accident can be simulated.

To start a new simulation when the user has been working on another simulation, select the menu option **File** - **New**.

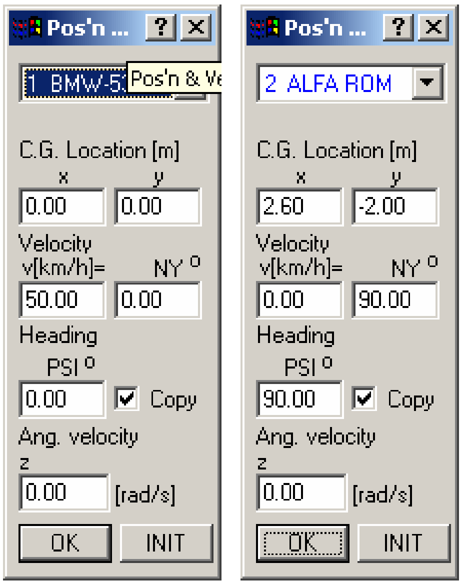
Loading all vehicles that were involved in the accident is the first step in the crash simulation. As stated previously, they can be loaded with **File - Import - Custom Vehicles** or **Vehicle – Vehicle Database**.

After checking the vehicle settings using **Vehicle – Vehicle Settings** and loading or creating a scene diagram, the next step is to position the vehicles in their impact position. For this, use **Dynamics** - **Position & Velocity**, or **Dynamics** - **Move/Rotate Vehicle**.

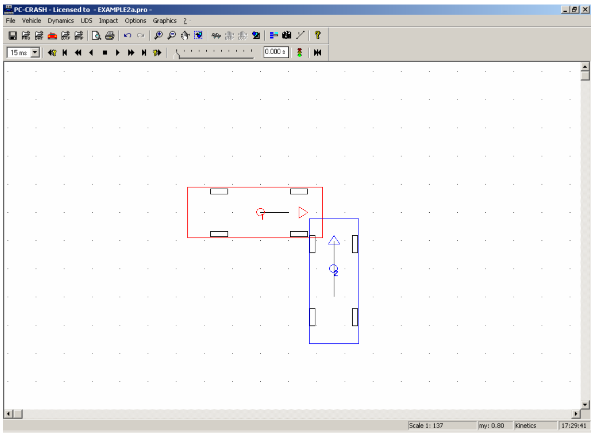
***Example:** A 90° crash is to be simulated between a vehicle traveling at 50 km/h and a stationary vehicle. Select **File** – **New** to start a new project. From the DSD database supplied with PC-Crash, select the 1999 BMW 530 d as Vehicle 1 and the 1998 Alfa Romeo 156 2.4 JTD as Vehicle 2. Keep all the default parameters for the vehicle geometry. Use **Dynamics** - **Position & Velocity** to input* the *following parameters:*

- *The BMW remains in the default position x=0, y=0, NY=PSI=0*
- *BMW start velocity = 50 km/h*
- *The Alfa's coordinates are changed to x = 2.6 m, y = -2.0 m, NY=PSI=90*

- *Alfa start velocity = 0 mph.*



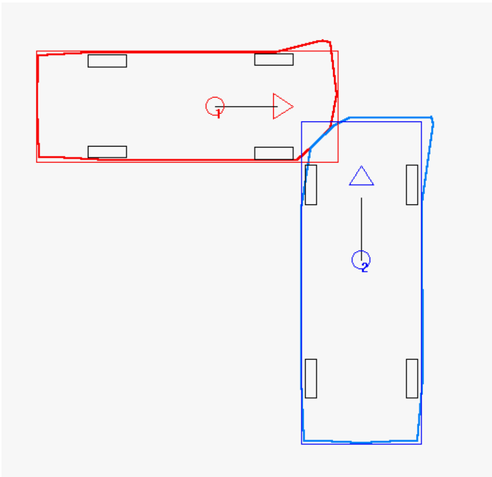
At this point, save the project file, using ***File – Save*** *. Enter the file name EXAMPLE2a in the file name text box of the Select Project Filename dialog box and press the Save button. Now this title will be shown in the title bar of the screen.*



The vehicles will be positioned on your screen as shown above, after the input of the stated parameters (in the figure the scale has been increased in size from the default 1:200).

Impact Position

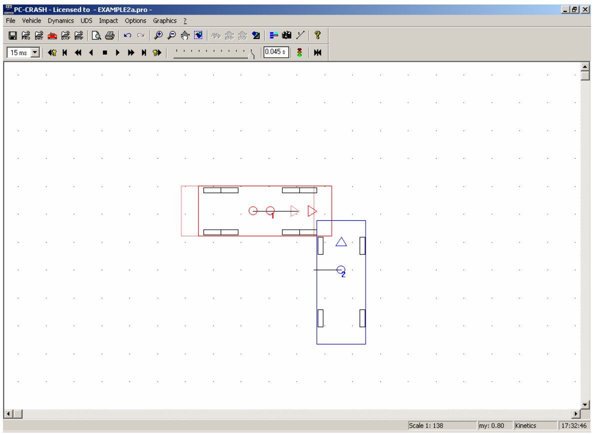
In order to get a realistic simulation of a collision it is important to position the vehicles at impact with the proper amount of overlap, when the exchange of the principal crash forces occurs. This is particularly important for the conservation of angular momentum portion of the collision model. The best way to do this is to position the vehicles at the impact point in an overlapped position corresponding to the amount of vehicle crush. This can easily be accomplished by loading a scene drawing incorporating 2D crushed outlines of the vehicles at impact, or attaching crushed vehicle shape DXF files to each vehicle using **Vehicle – Vehicle DXF**.



Another way of positioning the vehicles at impact with the proper amount of overlap is to drive the vehicles into the impact position. First, put the vehicles into the positions of their first contact. Then move them forwards according to the estimated pre-impact speeds *(50 km/h for the BMW in the example)* for about 30 – 60ms. This is done by defining a time step of 15ms in the Simulation toolbar, and then operating the Forward button two to four times. This moves the vehicles into their approximate positions at maximum engagement.

Although the amount of overlap seems to be relatively large, recall that it is the sum of the crush to both vehicles.

***Example:** Press the Forward button three times with the time step at 15ms. The following figure shows the resulting vehicle overlap.*



Definition of a new Start Position

After the vehicles have been driven into the proper impact position, a new start position should be defined for the simulation. This allows different simulation configurations to be compared with each other as quickly as possible. This is because the vehicles can easily be moved back to the start position at any time with the New Simulation button in the Simulation toolbar, so that a new simulation can be started immediately.

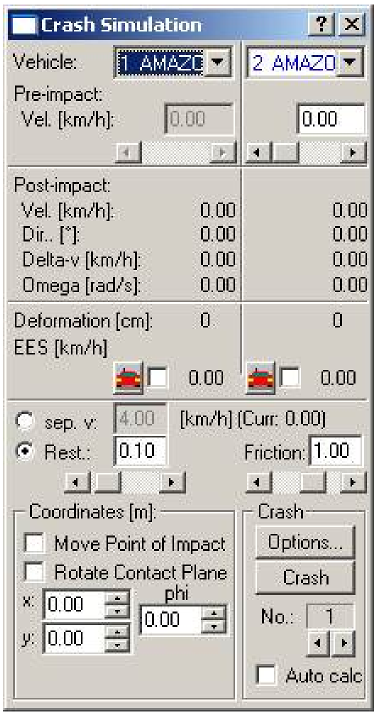
Once the vehicles are in the desired start position, use the button (Definition of a New Start Position). If the vehicles are not in their original start positions, an information box appears. Pressing the No button enables the simulation to be started from the new vehicle position.

Demo version: *If you only have the demo-version, you cannot move the vehicles forward and you cannot determine a new start point. In this case load the file EXAMPLE2.PRO from the Examples subdirectory in the PCCrash73 directory. In this similar file the vehicles are already correctly positioned and the following crash simulation has been performed.*

Crash Simulation

After the two vehicles have been positioned in their impact positions, you can begin with the actual crash simulation. For this, open the Crash Simulation dialog box with **Impact - Crash Simulation** (F8).

PC-Crash's crash simulation is based on the crash model by Kudlich-Slibar. The theory behind this linear and angular momentum-based crash model is described in the PC-Crash **Technical Manual**. The Kudlich-Slibar crash model is characterized by the definition of the point of impact. The point of impact is the point where the total crash force is assumed to be exchanged. The elasticity of the collision is considered, based on a coefficient of restitution. This model also considers the sliding of one vehicle along another vehicle or a fixed object, based on a contact plane angle and friction coefficient. These collision parameters can all be changed by the user.



The vehicle pre-impact speeds are immediately shown in the Crash Simulation dialog box when it is opened.

Definition of the Point of Impact

The first step after opening the Crash Simulation dialog box is to define the point of impact. Do this by activating Move Point of Impact. The point of impact can then be defined in one of the following ways:

- Click the left mouse button with the cursor outside the overlapping area of the vehicles. This will center the point of impact in the overlapping area, and rotate the contact plane to a reasonable value. This is usually the best place to start.
- Use the left mouse button to click on the desired point of impact inside the overlapping area of the vehicles (hold the button down to drag the position).
- Type in the desired impact x and y coordinates in the corresponding text boxes. You also can modify the coordinates using the spin buttons attached to the text boxes. Note that the longer you hold down the mouse button (or the keyboard direction button) the faster the values will change.

Example: *Use the text boxes in the Crash Simulation dialog box to input the point of impact coordinates: x = 2.0 m, y = -0.55 m*

Keep the restitution and contact plane friction coefficient at the default values of 0.10 and 1.0, respectively.

Note: *If both vehicles in an impact are 3D vehicles (i.e. they both have C.G. heights > 0), then the point of impact height must also be specified. This is done by entering an appropriate value (in feet or meters) in the "z" text box at the lower left of the Crash Simulation window.*

Rotating the Contact Plane

This can be done after activating Rotate Contact Plane. This automatically de-activates Move Point of Impact. The contact plane is rotated either by clicking the left mouse button with the cursor near the contact plane on the screen, then rotating it to the desired angle, or by entering the desired angle in the contact plane angle (phi) text box. As described at the point of impact, coordinates you can also use the spin buttons to modify the value in the text box

Example: *Rotate the contact plane to 45 degrees by entering this value in the phi text box. You can now observe that the vehicles change direction as a result of the exchange of collision forces.*

Note: *When making changes to the crash parameters, make sure the vehicles are positioned at the correct position for the crash being analyzed. If the Forward Simulation buttons or have been used and then a crash parameter in the Crash Simulation window is changed, the program assumes a second impact is being defined. This can always be checked by ensuring that the Crash No. at the lower right corner of the Crash Simulation window is at the correct number (i.e. 1 for the first impact).*

Display of Post-Impact Positions

After the point of impact or contact plane has been specified, or the Crash button has been pushed, PC-Crash shows the position of both vehicles 200 milliseconds after the collision. This allows the user to immediately see the effect of different input values (for speed, impact position, contact plane angle, etc.) on the vehicles' post-impact motion.

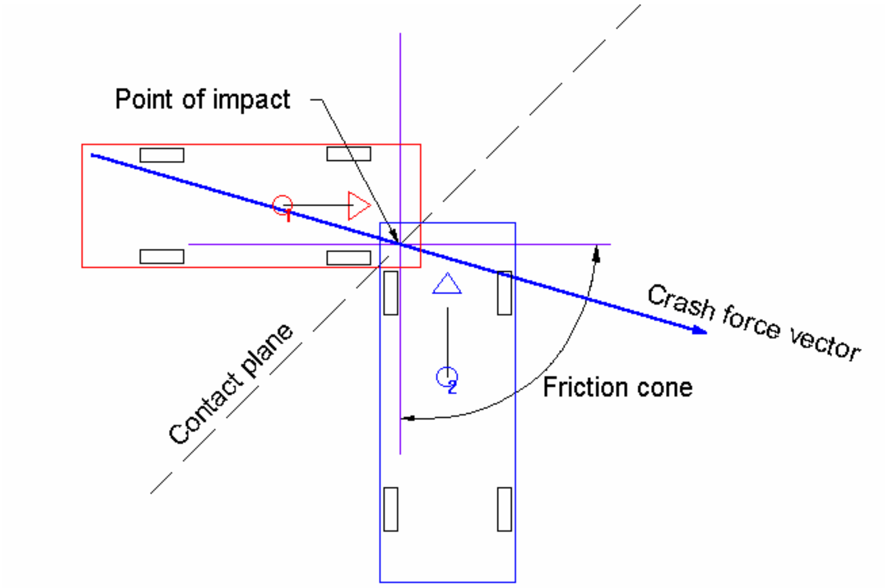
Other Crash Parameters

In addition to the vehicle positions 200 milliseconds after the collision, the following are displayed on the screen:

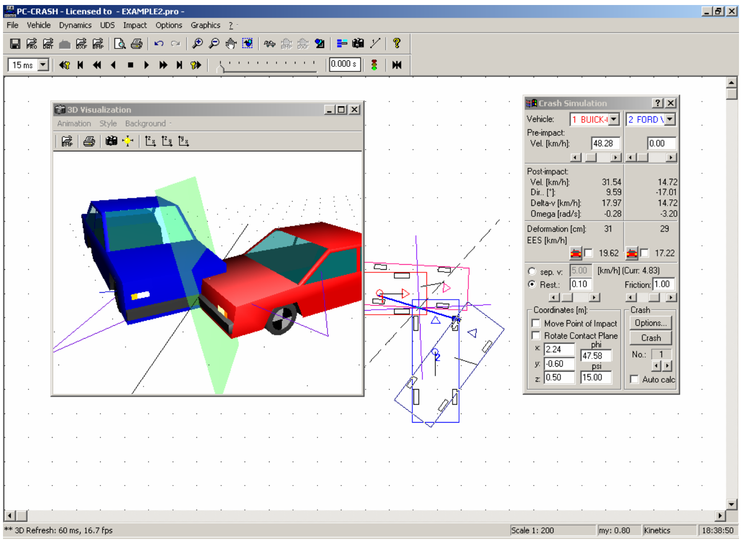
- Point of impact;
- Contact plane;

- Friction cone (the two lines which define the impact force angle limits for which no sliding will take place);
- Crash force vector (PDOF).

The Crash force vector, the friction cone and the contact plane can be visualized within the 3D Visualization window (**Options – 3D Window**) by activating Crashes within the Options – Display settings window (**Options – Options – Display settings**). The drawings of the crash



The Crash force vector, the friction cone and the contact plane can be visualized within the 3D Visualization window (**Options – 3D Window**) by activating Crashes within the Options – Display settings window (**Options – Options – Display settings**).



Note that if the contact plane is rotated to an angle where the crash force vector is on the edge of the friction cone, the impact becomes a sideswipe type with sliding taking place between the vehicle contact surfaces.

Example: *Now you can easily check the effects of changing a single input value. Click ☑ Move Point of Impact once again and shift the point of impact on the screen.*

Immediately you can see the effect of your choice of the point of impact on the post-impact movement of the two vehicles.

You can analyze the influence of the elasticity of the impact by shifting the scroll bar for the coefficient of restitution, or by typing a new value in its text box.

Although it would not be reasonable for the given example, the crash can be changed into a sideswipe collision by rotating the contact plane and/or reducing the vehicle-to-vehicle coefficient of friction.

EES Values

PC-Crash includes a calculation of the deformation energy (EES, or Equivalent Energy Speed) in the crash simulation. The total deformation energy will be distributed between the two vehicles, based upon the relation of the masses of the vehicles as well as of the respective deformation depths. The equations for this calculation are given in the **Technical Manual**.

The EES values for the vehicles can be seen within the Diagrams window (**Options – Diagrams – Diagrams – Vehicles – EES**).

If a different distribution of the total deformation energy is desired, the EES for one of the vehicles can be specified. This is done by clicking on the EES check box for that vehicle, and then entering the desired value in the text box beside the check box. The EES corresponding to the remaining deformation energy is calculated automatically for the second vehicle.

Example: *Activate the EES check box for the first vehicle and enter the desired value. The EES value for the second vehicle is then calculated and is shown in the second vehicle's text box.*

Post-Impact Trajectories

PC-Crash simulates the post-impact trajectory of each vehicle, based on the crash parameters and the post-impact sequences (after time=0, below the Start sequence) in the **Sequences** menu option.

The post-impact trajectories can be calculated one step at a time by pressing the Forward button in the Simulation toolbar. Pressing the Forward button runs the simulation until the vehicles reach their rest positions. In the latter case, the simulation can be interrupted at any time by pressing the button a second time, by pressing the right mouse button, or by using the ESC key.

Once the simulation has been run to the rest positions, do not use the Simulate Forward or Backward buttons to move the vehicles. Instead, use the slider bar to move the vehicles to any position desired. The vehicles can also be moved to any point by typing a specific time in the time display text box 0.000 s. Press the TAB key or click in the main screen after changing the time this way, then press the F5 function key to refresh the screen.

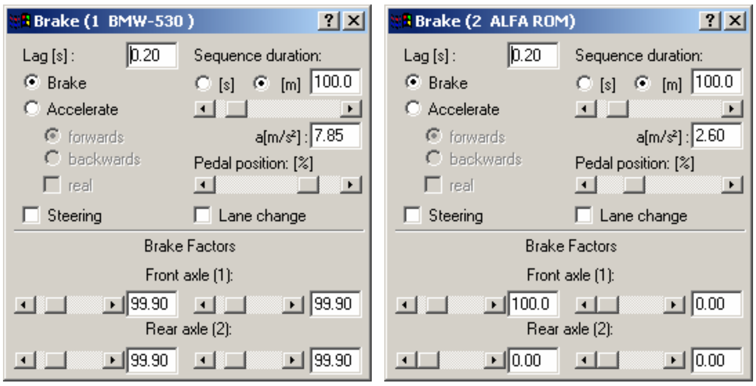
Press the New Simulation button only if you want to perform a new crash simulation with different parameters. This will erase the existing trajectories. Then, make the desired changes and run the simulation again.

The post-impact trajectories of vehicles with different braking conditions can easily be compared. Change the braking by accessing the vehicles' braking sequences in the Sequences window by selecting **Dynamics - Sequences** (F6).

Example: *Use the Sequences window to define a brake sequence for each vehicle as follows:*

- *BMW - Use the Pedal position slider bar to define full braking on all 4 wheels*

- *Alfa - Use the individual wheel slider bars to define 100% braking on LF wheel (to simulate a wheel locked due to damage) and no braking on the other 3 wheels.*



*Note that the coefficient of friction for this example is 0.8, which is the default coefficient of friction in PC-Crash. The default value can be changed to another value by selecting the menu option **Options - Options – Default Settings**, or more easily by double-clicking on the "my" area of the status bar at the bottom of the screen.*

After the braking sequence values have been entered for each vehicle, push the New Simulation button in the Simulation toolbar and then the Crash button in the Crash Simulation window. Run the vehicles to their rest positions (by applying the Simulation toolbar's Forward button). The resulting vehicle post-impact trajectories are shown on the main screen.

Note: *Auto calc in the Crash Simulation window can be used instead of the Simulation toolbar Forward button . This feature will run the vehicles to their rest positions without showing intermediate vehicle movement whenever a change is made in the Crash Simulation window, allowing the effect of a change on the rest positions to be immediately seen.*

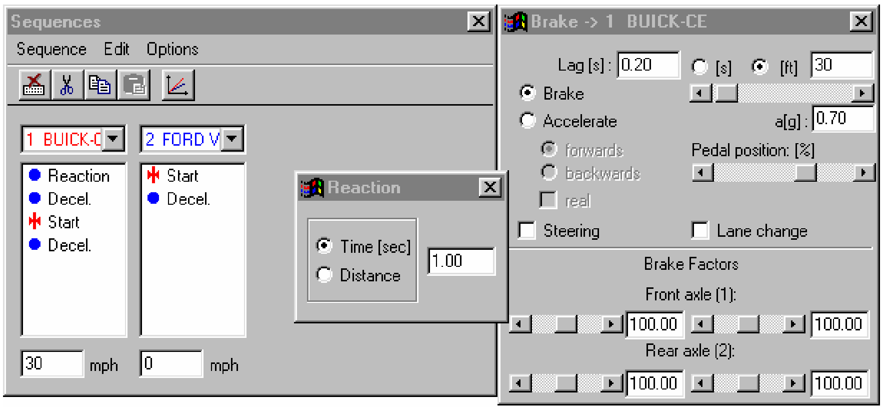
Pre-Impact Trajectories

As for post-impact sequences, the **Dynamics - Sequences** menu option (F6) is used for pre-impact sequences such as driver reaction and pre-impact braking. Pre-impact sequences are those above the Start sequence (time = 0) in the Sequences dialog box.

To avoid deleting the post-impact simulation results by accident when changing the pre-impact parameters, lock the post-impact results by activating the Lock Forwards Path button in the Simulation toolbar. Once the post-impact results are locked the vehicle positions and velocities at time=0 cannot be changed, because this would influence the post-impact movement.

Example: *Load the sample project file EXAMPLE2.PRO. For the Buick, add pre-impact reaction and braking sequences, using **Sequences – Sequence – Vehicle/Driver – Reaction** and **Brake**. After adding these sequences, you will have to drag the Start sequence down to place the added sequences above the Start sequence. Then, double-click on the reaction sequence and change the time to 1.1 seconds. Double-click on the pre-impact Decel. Sequence and apply 100% braking over a distance of 30 feet.*

The pre-start section of the simulation can now be run. As this section is before the start (time = 0), it is simulated by activating the Backward buttons or *in the Simulation toolbar.*



***Note:** Trailers cannot be backed up before time=0.*

Complex Pre-Impact Trajectories

When using pre-start sequences, it is normally only possible to simulate straight line motions, because the inversion of the system of differential equations used by PC-Crash makes backwards simulation of complex yaw or steer motions inaccurate. Also, trailers cannot be backed up before time=0.

There are several ways of overcoming these limitations, however, as follows:

- Don't have the impact at time = 0. Start the simulation at some time before impact, so all the pre-impact and post-impact motion is included. As many pre-impact sequences can be put in as necessary to define the correct pre-impact driving maneuvers. The weakness of this option is that, if speeds are not known, any change in one vehicle's start speed will cause it to arrive at the impact point at the wrong time with respect to the other vehicle. Unless the impact speeds are fairly minor, this option usually takes too much trial and error time.

- Reconstruct the pre-impact portion of the incident as a separate project, starting the vehicles out at positions before the impact. Alter their start conditions individually so that they arrive at the impact point at the correct speed and attitude. The timing with respect to one another can be changed easily using **Options - Diagrams - Options - Origin offset**, as detailed in the Section **Synchronization of Vehicles** later in this chapter.

  If the whole pre-impact simulation needs to be viewed from the start to end, animations must be made from the two projects separately and then joined outside of PC-Crash. This can be done using video editing software of the type normally provided with video output cards.

- The Kinematics path feature can be used for cases not involving trailers where the pre-impact location and timing of the vehicle is important, but having the program check that it is obeying the laws of physics is not. An example is a left-turning vehicle in an intersection accelerating from rest across an oncoming vehicle's path.

  For this option, have the impact at time = 0 and solve for the post-impact motion as usual. Then, lock the post-impact motion with the Lock Forwards Path button  and change the simulation model from Kinetics to Kinematics using the Simulation Model button . Lastly, draw a curved path for the turning vehicle using the menu option **Dynamics – Define Path Points** (see the **Paths** section later in this chapter).

  When the backwards simulation buttons  or  are used, the turning vehicle will follow its defined path exactly. Because the Kinematics model is used for the pre-impact motion, the user must check that the motion along the defined path is physically possible for the speed and acceleration used.

Saving the Simulation as a Drawing

The PC-Crash vehicle positions and other data can be saved as a DXF drawing file, if desired. To do this, proceed as follows:

1. Run the simulation to the vehicles' rest positions.
2. Save the simulation as a DXF file by choosing the **File - Export - DXF** menu option. *In the example enter the file name EXAMPLE2 in the text box and press the Save button (PC-Crash automatically appends the DXF file extension)*. Note that all vehicle positions and tire marks that appear on the screen will be saved in the DXF file. If it is desired that only some items be saved in the DXF file, then the other items must be turned off in **Options – Options - Display Settings** or **File – Default settings**.
3. The DXF drawing containing the vehicle positions and other data can be brought back into PC-Crash using **File – Import – DXF**  and selecting the saved file *(EXAMPLE2.DXF).* The saved vehicle positions will now be shown on the screen as a DXF drawing. The DXF file can also be imported into other programs, such as Microsoft Word.

Automatic Collision Analysis

The Collision Optimizer in PC-Crash is an automatic collision analysis feature that eliminates the potentially lengthy manual trial and error process of determining vehicle speeds and other impact parameters. The Collision Optimizer determines the best solutions for impact speeds and other parameters based on specified impact, intermediate and rest positions.

Rest and Intermediate Position Definition

To specify the rest or intermediate positions of the vehicles choose the menu option **Impact** - **Rest Positions** or **Impact – Intermediate Position 1 - 5**. The cursor changes to a tow truck (different from the tow truck that positions the vehicles at the start) that is used to move and rotate each vehicle from its initial position to its rest or intermediate position. The initial vehicle positions remain unchanged.

Collision Optimization

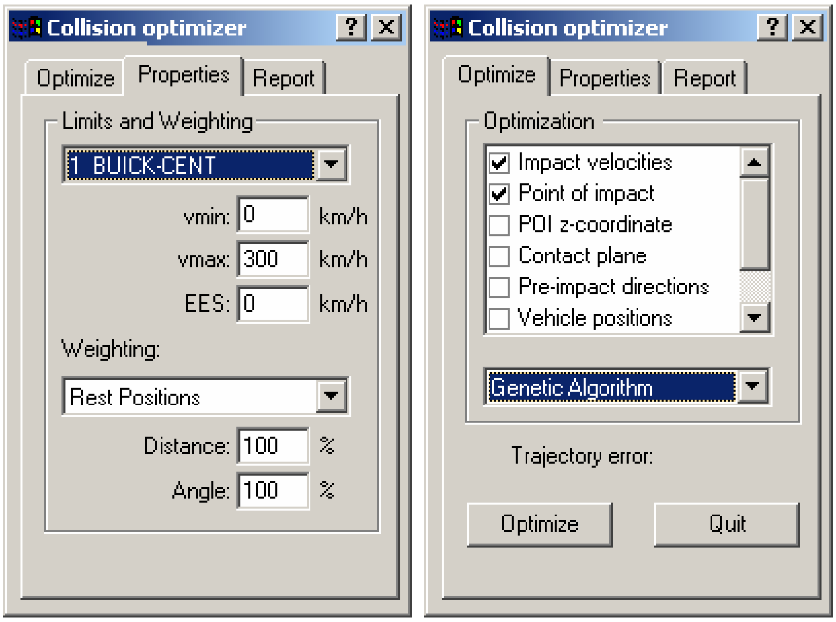
Once the vehicle rest and/or intermediate positions have been placed, the collision optimizer can be used to automatically calculate the impact velocities, the point of impact, the contact plane, and other values. Select the menu item **Impact - Collision optimizer**.

It is generally more efficient to optimize only two or three parameters at once. This allows the user to view the results after a short optimization time and then make a decision on what to optimize on in the next optimization process, if it is needed.

The default Genetic algorithm generally finds a solution quicker than the optional linear algorithm.

A velocity range for each vehicle can be specified. For example, if it is known a vehicle was stopped prior to impact, a minimum and maximum velocity of 0 can be assigned.

The user can control the weighting of linear and angular displacement errors for the specified rest and intermediate positions and EES value. The weighting influences the relative importance of angular and linear displacements for each vehicle in the optimization process.



To start the optimization, push the Optimize button. The optimization process can be stopped at any time by a single click on the right mouse button or by pressing the ESC key.

The better the start conditions are, the faster the optimization process will find an optimized solution.

During the process, the items checked in the Optimization parameter list are optimized one at a time, such that the program doesn't go back to the first item after optimizing the second, and so on. The optimization process should therefore be performed at least twice if two or more items are checked. The optimization process can be repeated several times and the program will always keep the best solution found as a result.

Example: *First prepare for the simulation by saving the rest positions in the EXAMPLE2.pro project file in a DXF drawing file (EXAMPLE2.DXF), as described in the previous section. Import EXAMPLE2.DXF into EXAMPLE2.pro using* ***File – Import – DXF Drawing****. Use* ***Impact - Rest Positions*** *to define the rest positions, based on the rest positions in EXAMPLE2.DXF. Put the vehicles into the start (impact) position with the New Simulation button . Select* ***Impact – Collision Optimizer*** *and change the vehicle speeds to different values in the Crash Simulation dialog box, to simulate an unknown starting speed for each vehicle. You could also move the impact point to a different position to simulate that its exact location is unknown. Select Impact Velocities in the Optimizer dialog box (and Point of Impact, if you moved the impact point) and run the Optimizer by pressing the Optimize button.*

Secondary Impacts

If Automatic Calculation of Secondary Impacts and Crash Detection in the Simulation Parameters dialog box (**Options – Options – Simulation Parameters**) are activated, secondary impacts will be automatically calculated. Crash detection can also be selected or de-selected in the **Impact** menu. The penetration depth of the secondary impact (given in milliseconds after initial contact) can be changed in the Depth of Penetration drop down box of the Simulation Parameters dialog box. A reasonable value is generally in the 30ms to 60ms range.

After running a simulation where there is a secondary impact, the user can move the vehicles to the secondary impact positions to change parameters of that impact only. This is done by using the Crash No. scroll box in the Crash Simulation dialog box to scroll to 2 (or to a higher number if there are more than two impacts). The vehicles will then automatically move to their positions at the time of the secondary impact. When this is done, the crash parameters for the secondary impact can be adjusted manually in a similar manner as they were for the primary impact.

If ☑ Automatic Calculation of Secondary Impacts is not selected in the Simulation Parameters dialog box and ☑ Crash Detection is, the simulation will stop when it detects a secondary impact.

Shifting the Start Point

Sometimes it is advantageous to repeat a simulation starting from a newly defined start position. Sequences which have originally been calculated by a backwards simulation from the start can be done with a forwards simulation. To do this, the start position of the vehicle(s) must be moved to where they would be at the start of the first sequence and the position of the Start sequence has to be changed.

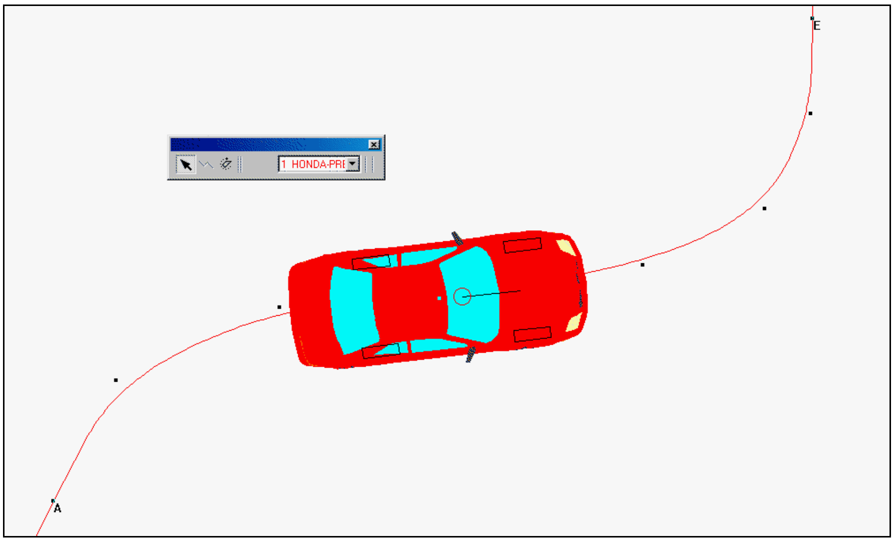
Example: *In the example EXAMPLE2.PRO, shift the start position of the Buick to the beginning of the 1.1-second reaction sequence entered earlier. First, move the Buick into this position by activating the Backward button . Now this position has to be fixed as the new start position. This is done by activating the New Start position button in the Simulation toolbar. Press the No button in the information box question that asks if the vehicles should be repositioned to the previous start position.*

The next step is to rearrange the sequences. Using ***Dynamics - Sequences*** *(F6), shift the Start sequence of the Buick to the top position.*

Now, repeat the simulation of the driving motion of the Buick from the new start position, ensuring that Impact Detection is turned on. If you get a slightly different result for the rest positions, it is because the brake lag on the Buick was not set at 0. In this case the original simulation, with no pre-impact brake sequence, was applying only 50% braking during the lag time of the post-impact brake sequence, because the initial condition, with no pre-impact sequences, was no braking. The new simulation would have 100% braking during the lag time because the initial condition (defined by the pre-impact brake sequence) is 100% braking.

Paths

Paths can be created for the vehicles in PC-Crash. This allows the user to specify complicated vehicle maneuvers without having to use multiple steering sequences.



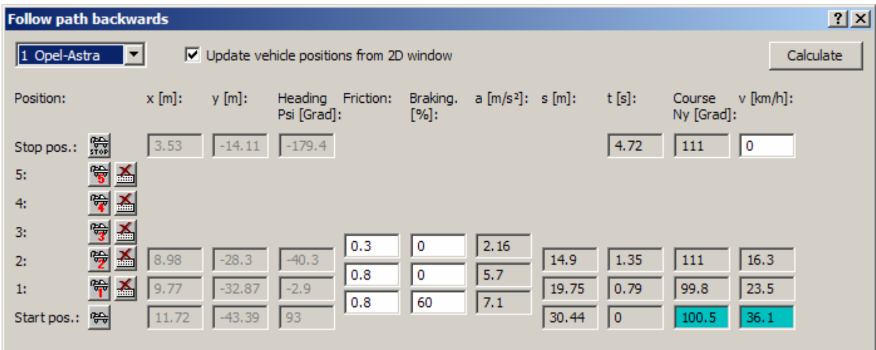
Path definition is done with **Dynamics - Define Path Points** (see the **Menu Description** chapter) and/or the **Follow Path Toolbar.**

Post impact analysis using tire marks

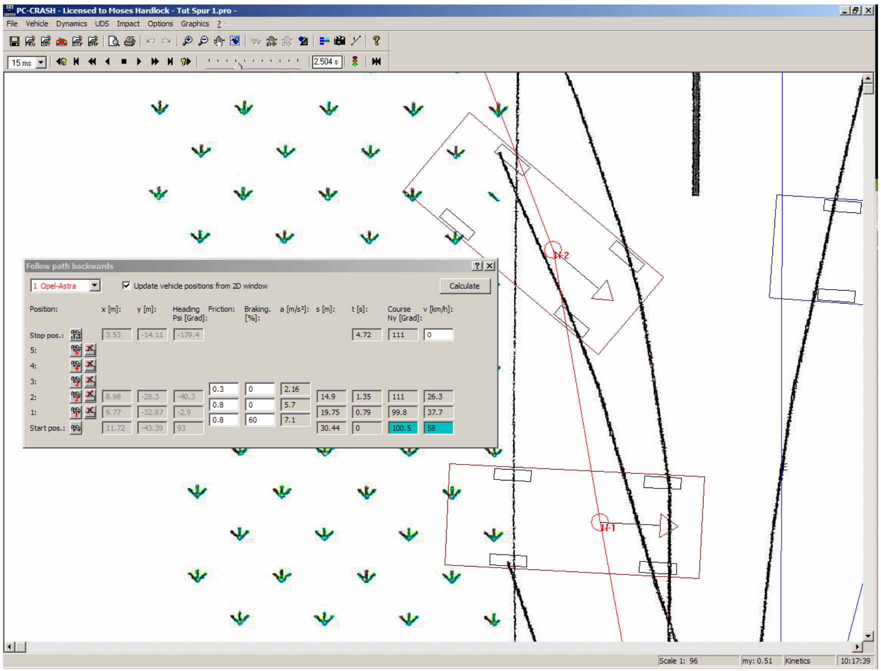
If there are tire marks on the road surface, the post-impact situation can be easily reconstructed by back-tracking the vehicle motion along the wheel trajectories. With this calculation the situation immediately after the collision, for example post-impact velocities, can be determined.

In PC-Crash it is necessary to know at least the collision position and the rest position of a vehicle. Intermediate positions may be defined along the tire marks to improve precision.

In the menu select the command **<Dynamics> <Kinematic follow path backwards…>** to open the input window.



First, the rest position of the vehicle has to be defined. Use the button to drag the rest position from the vehicle position to the desired position. The handling with the mouse is the same as described in chapter *The Tow Truck Tool* on page 43. Up to 5 intermediate positions may be defined using the buttons along the tire marks. To apply the positions for the calculation the option **Update vehicle position from 2D window** has to be turned on.



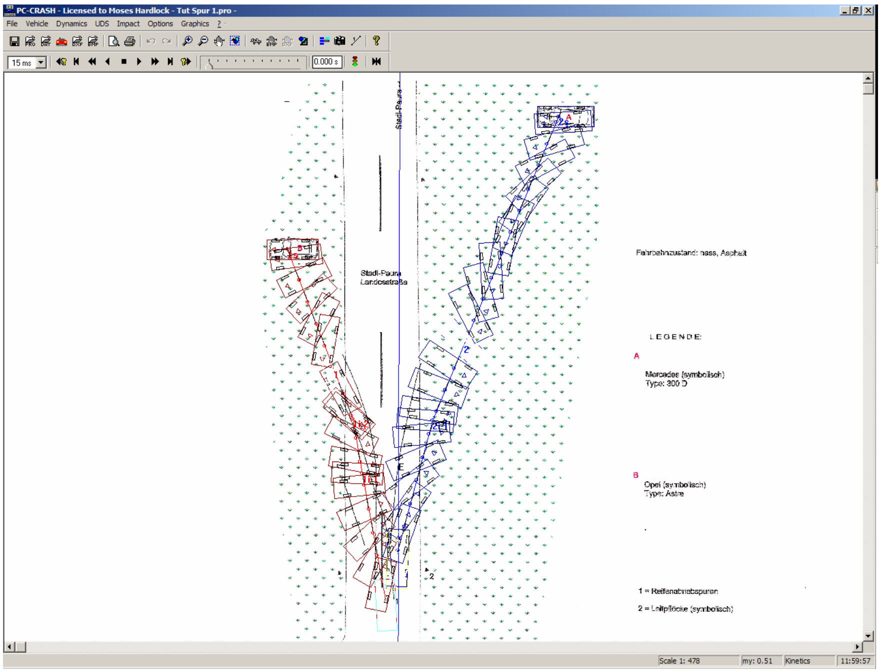
For every movement between two defined positions a friction coefficient and a brake factor can be entered. The brake factor is only applied at the vehicle's longitudinal movement. Full brake force is applied at the lateral movement of the vehicle due to the cornering forces.

In the result fields the velocity, the course angle, the deceleration, this distance and the time are displayed for each position.

By pressing the **Calculate** button the movement for the selected vehicle is calculated and applied.

Important note:

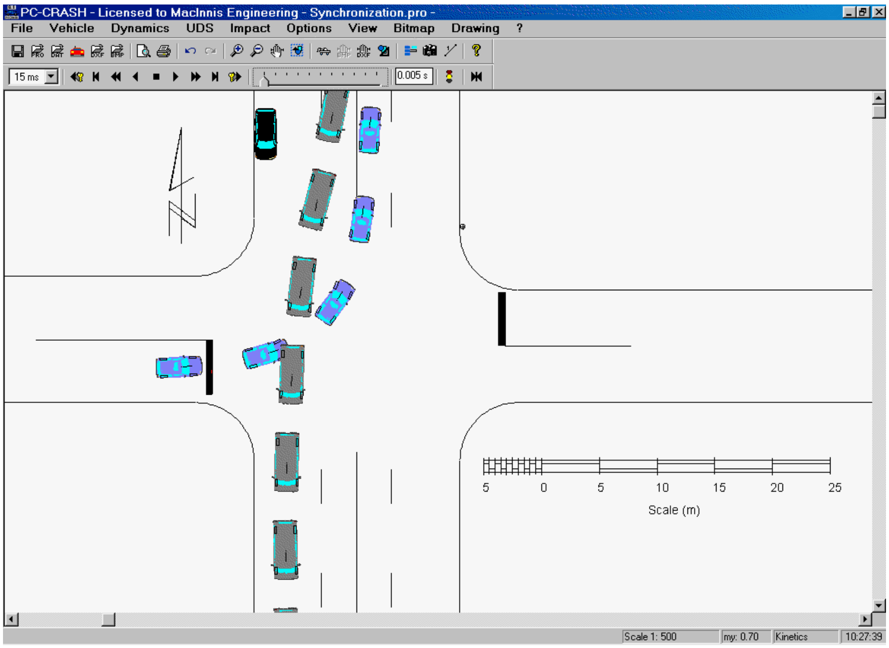
The vehicle movement using this calculation is performed as specified by the positions and values, not by physical constraints. Always double check your results by making a kinetic forward simulation.



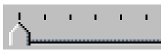
Synchronization of Vehicles

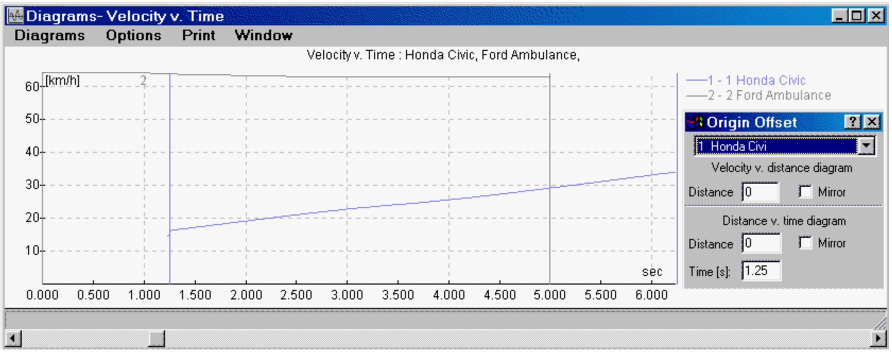
The motion of different vehicles can be synchronized after each vehicle's motion has been determined independently.

The following figure shows a two-vehicle simulation where the individual motion of each vehicle has been defined correctly, but where they do not start at the correct time with respect to one another.

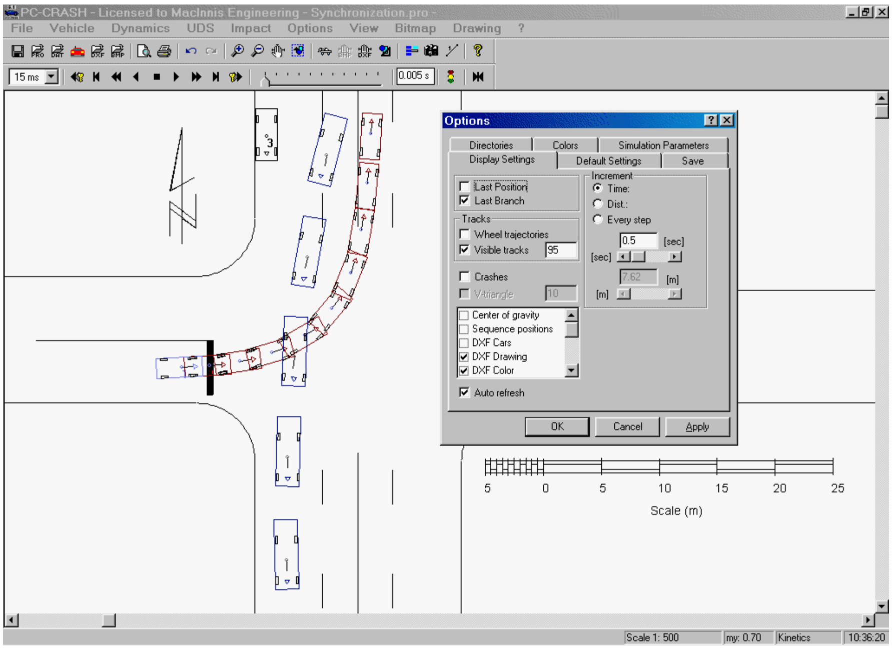


To synchronize the vehicles the correct time offset must be specified. This is done in the Diagrams window by selecting **Options - Diagrams - Options** - **Origin offset** and changing the origin offset time for one of the vehicles.

This origin offset specified in the Diagrams window will be used for the Simulation toolbar slide control  as well as in the rendering of animations. Use the Simulation toolbar slide control to move the vehicles synchronously and determine the necessary origin offset time by trial and error. In the following figure, a time of 1.25s has been put in for the Honda. Note how its Velocity v. Time graph has been shifted 1.25s to the right of the origin.



Use the Last Branch checkbox in **Display Settings** to display vehicle positions at selected distance or time intervals, if desired. In the following figure the last branch has been set at 0.5s.  Dxf Cars has also been turned off, so that the vehicles are displayed as simple rectangles to avoid long refresh times.

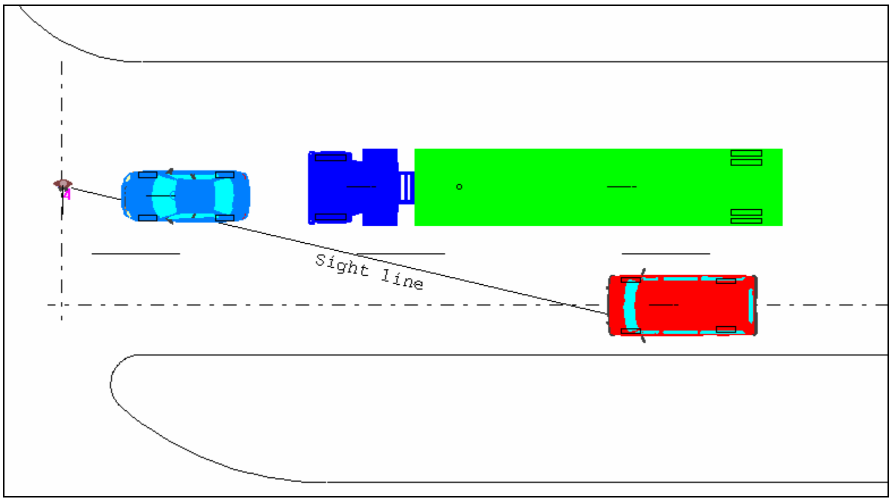


Pedestrians

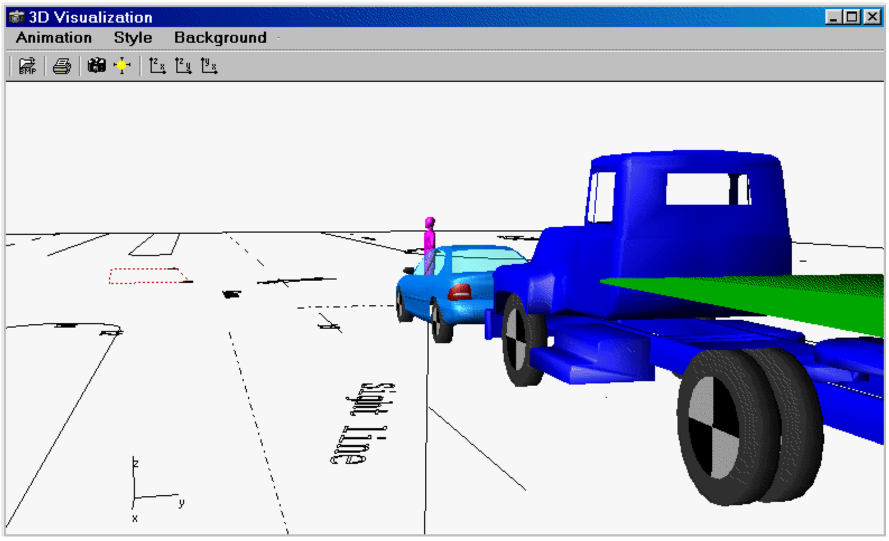
The Simple Pedestrian Model

This models the pedestrian as a simple block-shaped vehicle. It is useful for determining pedestrian visibility around other objects in the scene and for doing time-distance studies.

Using **File - Import - Custom Vehicle** , load Pedestrian.dat. Move the pedestrian to the desired start position using the tow truck tool . The pedestrian's velocity can be entered in the Position and Velocity dialog box.



For a more realistic appearance, a 3D DXF drawing of a pedestrian can be attached to the pedestrian, using **Vehicle – Vehicle DXF**.



The Multibody Model

The multibody model is used for modeling vehicle-pedestrian collisions, vehicle-motorcycle collisions, and unrestrained and restraint occupant motion in a vehicle. This model calculates impacts automatically between the multibody's individual body components and a vehicle's exterior or (in the case of the multibody occupant) interior.

The multibody models react to gravity and ground and vehicle impact forces. The multibody pedestrian cannot be made to walk or run the way a human can. For this reason, it is best to start the simulation immediately before impact. Otherwise, the pedestrian will fall down due to gravity prior to being struck.

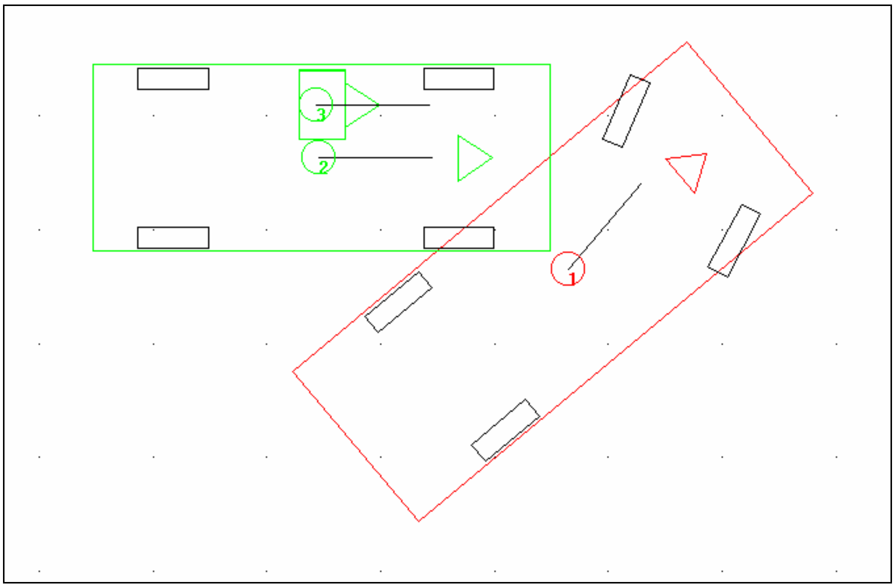
See the **Multibody Model** chapter for complete details on the use of multibody systems.

Occupant Models

The "Passenger" Model

This models the occupant as a simple block. It is useful for determining the direction an occupant moves relative to the vehicle during an impact.

Using **File - Import - Custom Vehicle** , load Passenger.dat. Move the passenger to the correct position in the vehicle using the tow truck tool . The vehicle's start velocity is automatically applied to the passenger. This can be checked in the Position and Velocity dialog box.



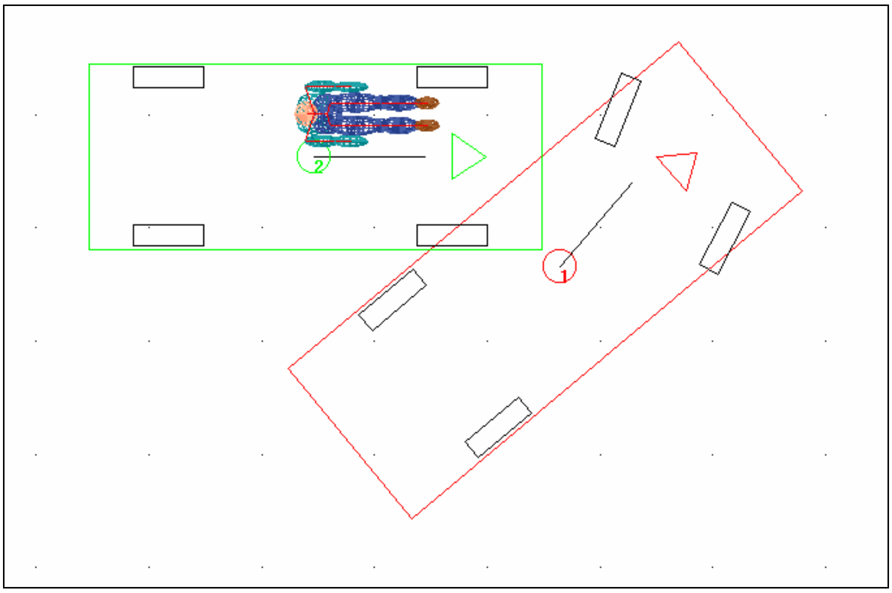
In a simulation the passenger will follow the vehicle's motion during normal driving conditions. However, in the case of a collision or other event in which the vehicle acceleration exceeds 2g, the passenger moves in the proper direction with respect to the vehicle.

The passenger is not restrained by the vehicle outline or a seat belt and can therefore exit the vehicle, unless an impact is defined (using **Impact – Crash Simulation**) between the passenger and the vehicle.

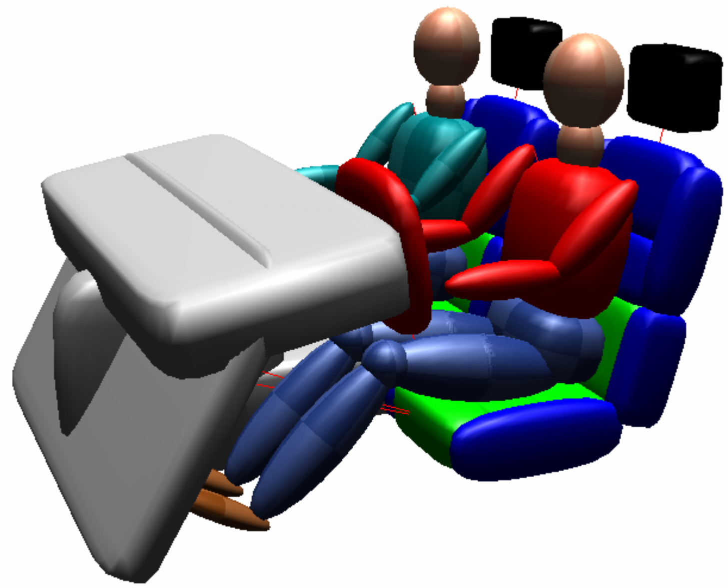
A relative passenger-vehicle deceleration of up to 2g is applied as a result of the program trying to match the passenger's speed with the vehicle. This acceleration can cause the passenger's post-impact path to curve, especially in lower speed impacts. For this reason, this feature should only be used to determine which direction the passenger would move with respect to the vehicle immediately after the impact.

The Multibody Occupant

One or more multi-body occupant models can be placed in a vehicle. This model is similar to the multi-body pedestrian model, except it is in a seated position. This model allows impacts to be automatically calculated between the occupant's multi-body components and the vehicle's interior or other occupant multi-bodies.



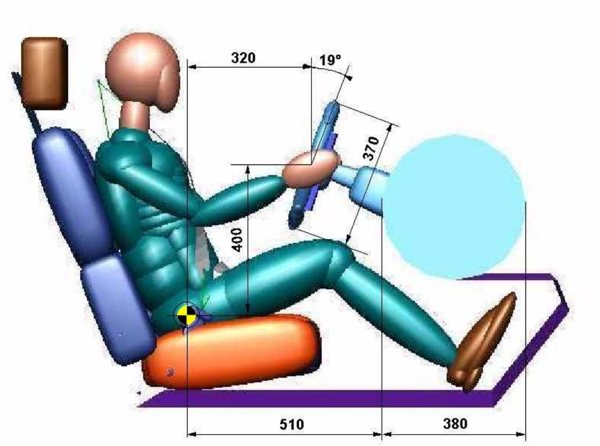
The multi-body passenger model is not connected to the vehicle in any way. Therefore, the multi-body passenger will accelerate at 1g downward due to gravity. In this case, it is best to use this model only when the simulation starts right at impact, or shortly before. Another option is to use the "Seat + Occupant" multi-body, which includes a 3D seat. There are additional multi-body occupant systems delivered with PC-Crash including multiple seats/rear-seats with and without seatbelts as well as occupants with cockpits.



The multibody occupant is described in detail in the **Multibody Model** chapter.

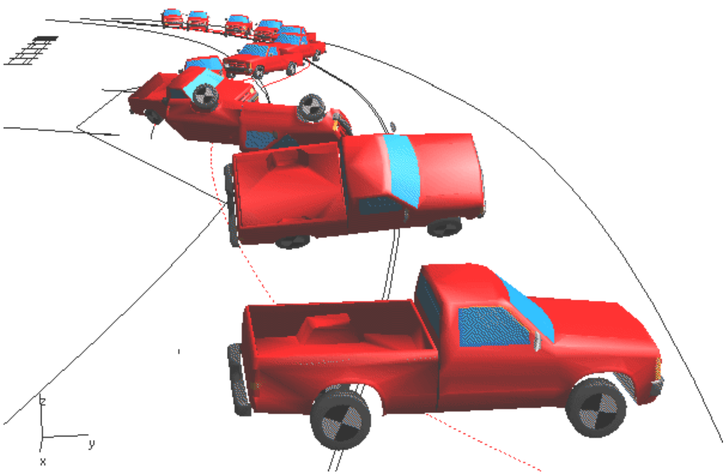
The Madymo Occupant

Madymo (MAthematical DYnamic MOdel) is a more comprehensive multibody occupant. It is a model of a 50th percentile Hybrid III dummy. Madymo includes a seat with easily changeable parameters, a steering wheel, lap and/or torso seat belts, and airbags. Only one Madymo occupant (driver or right front passenger) can be modeled at a time.



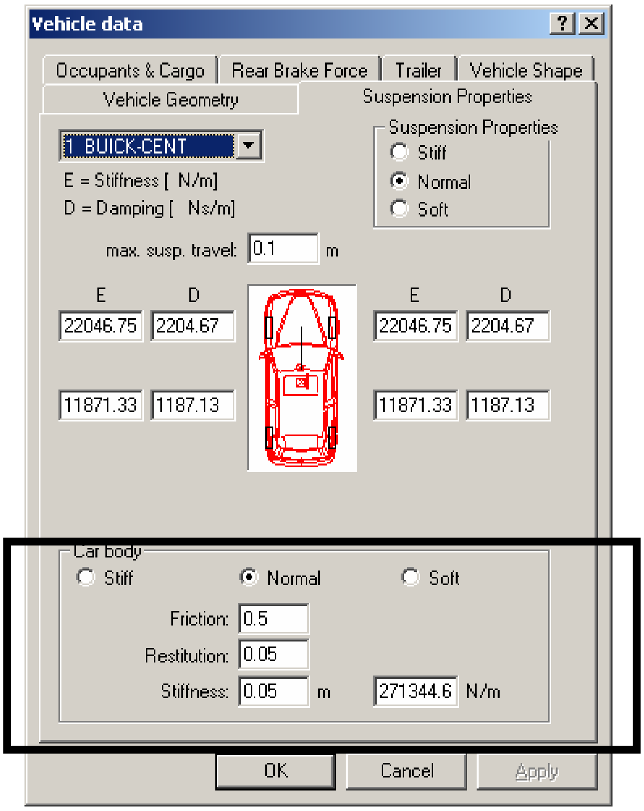
The optional Madymo model is described in detail in the **Madymo Occupant Model** chapter.

Rollovers



The rollover model is based on the calculation of contact forces between the vehicle body and the ground. Several contact ellipsoids are used to represent the vehicle shape. The geometry of these ellipsoids is based on the vehicle type specified in the menu item **Vehicle - Vehicle Settings**. Refer to the **Technical Manual** for more details on this.

The values for rollover ellipsoid body to ground friction, restitution and stiffness are defined in the Car Body field of the **Suspension Properties** tab of **Vehicle Settings**.



Vehicle speed changes and accelerations that occur as a result of vehicle impacts with the ground can be viewed in the Diagrams window, using **Options - Diagrams – Diagrams – Sensor Signals**.

When **Rollover Detection** is checked in the **Dynamics** menu (default condition), vehicle body-ground contact forces are taken into account when a vehicle rolls over. Otherwise, the vehicle falls through the road surface when inverted.

Note: *When working with a 3D road surface made up of multiple slope polygons, Rollover Detection will increase calculation time significantly. In these cases, turn off Rollover Detection, run the simulation until the vehicle is at the point of rollover, stop the simulation manually, and then turn on Rollover Detection and run the simulation to completion. Rollover Detection can be left off if there are no rollovers.*

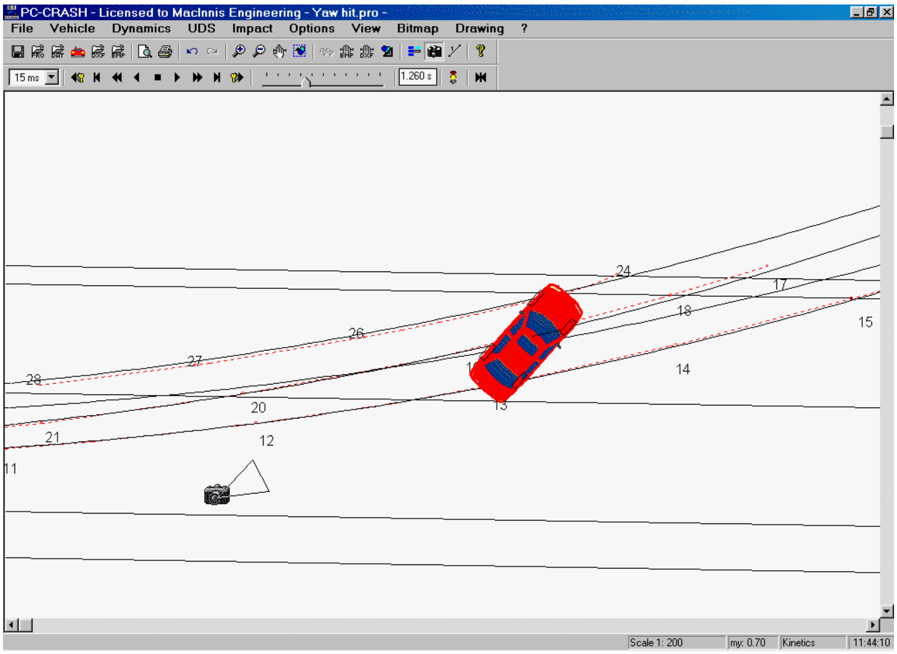
Body-ground impacts are calculated at a point where the tangent of the body ellipsoid surface is parallel to and in contact with the ground. If this is not possible, an impact point will not be located and no contact will be calculated. This can occur if the vehicle ellipsoid's tangent point is beyond the edge of the contacting plane, such as when a vehicle goes off a cliff at a slow enough speed to have underbody contact. In this case, the best solution is to add one or more angled planes to extend the edge of the ground plane at an angle that will be parallel to and in contact with the desired portion of the ellipsoid at the correct time.

3D View and Animation

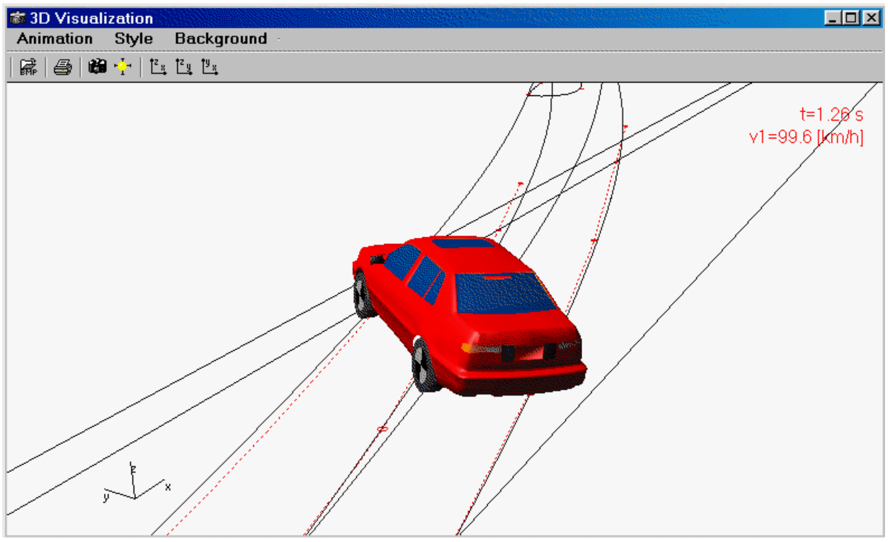
PC-Crash 3D has a 3D perspective view feature. This can be used to view the accident scene from any position desired and then make an animation of the simulation.

3D View

Use **View - 3D Camera Position** to select the viewing position. A camera will appear on the screen. It can be moved and rotated in the plan view by moving the cursor with the left mouse button pressed. If the camera is not visible in the drawing area a single click (left button) at any position will move the camera to that position. To change the viewing direction, click the cursor inside the viewing cone and rotate it.



Select **Options - 3D Window** (function key F9) to display the 3D Visualization window. This window can also be opened by double-clicking on the camera symbol.



For changing the camera's vertical position and other parameters, use the **Animation - Set Camera Position** menu option in the 3D Visualization window, to open the Camera Settings dialog box.

The view can also be changed in the 3D window dynamically, as follows:

- **Moving the Camera** - Hold the left mouse button down and move the cursor in the 3D window.
- **Panning (Rotating) the Camera** - Hold the SHIFT key and the left mouse button down and move the cursor in the 3D window.
- **Zooming the Camera** - Hold the CTRL key and the left mouse button down and move the cursor up or down in the 3D window to move the camera away or towards the scene.

Animation Construction

Once a view has been selected, animations can be created using the **Animation** menu option in the 3D window.

Animations can be made from a fixed camera position or a camera moving with one of the vehicles. The correct placement of a moving camera must be done when the simulation is at time = 0. The choice of a constant camera position or one attached to a vehicle is made after selecting **Animation – Render** in the 3D window, then selecting **Options – Camera** after a file name has been chosen for the animation.

Input and Output Data

Printouts

Printing a Scene Diagram

A scale diagram of the scene, including the PC-Crash vehicles and tire marks on the DXF drawing and/or bitmap image, can be printed to any desired scale.

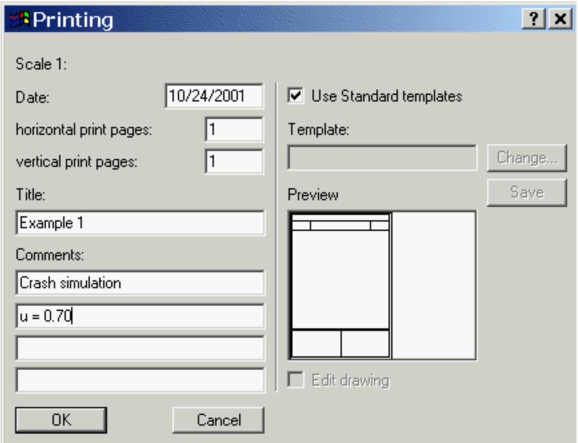
Use **File – Print**  to print the scene diagram. The scale shown in the status bar at the bottom of the screen is used for the printout.

Printing a Report

At the end of any simulation a report can be printed. The report printout contains all parameters that are necessary to duplicate the simulation. Choose the menu option **File - Print Report.** The report can also be viewed and printed in the Values window. Print Preview is available in either case.

Printout Comments

Choose the menu option **File** - **Print Comments** to access a window in which you can add descriptive text for the scene or report printout. The printout template can also be changed or modified in this window.

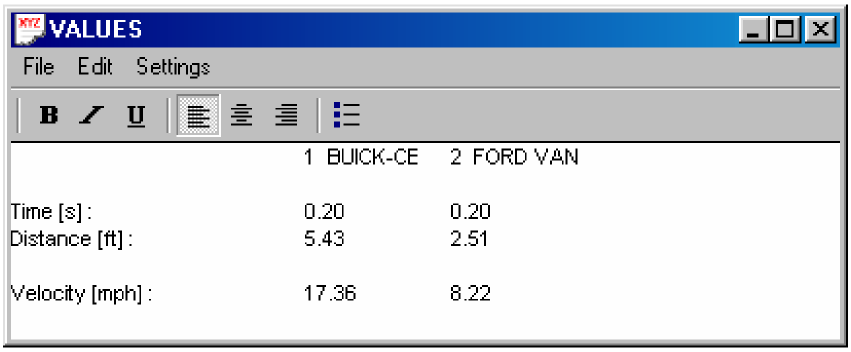


Values

The Values window is used to view the calculated values of the simulation, or a summary of the calculated sequences of the simulation.

Open the Values window with **Options** - **Values** (F4).

The default configuration in this window is the elapsed time, the distance traveled and the velocity of each vehicle in the simulation at any selected point in the simulation.



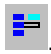
Other values can be chosen in the **Settings** menu option of the Values window. Upon selection of this option, a cascade menu appears which shows the following choices:

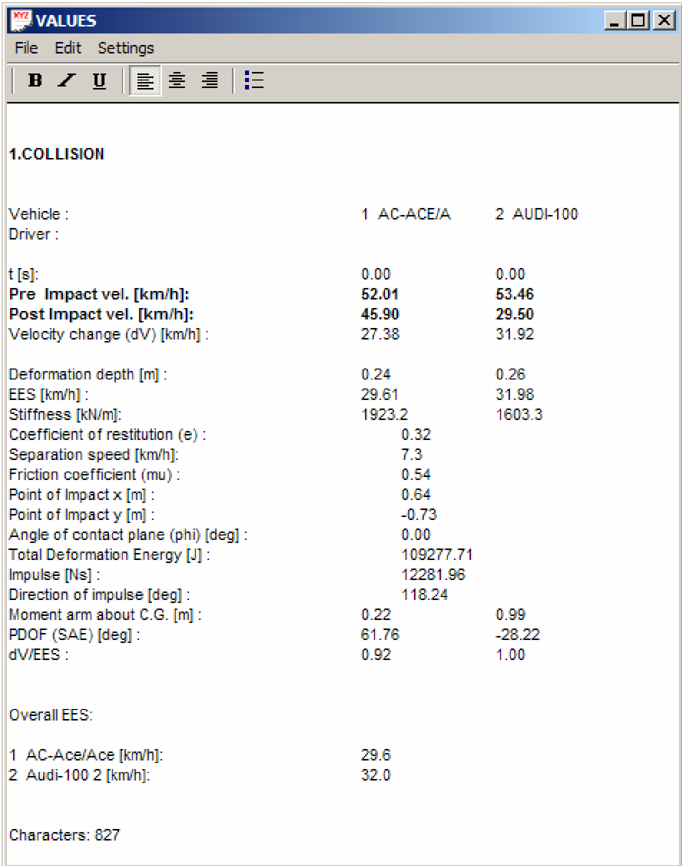
- Distance/Time
- Tire normal forces
- Tire lateral forces
- Brake forces
- Energy (kinetic)
- Coefficient of friction
- Velocity
- Acceleration.

In addition, there are other menu options in the Settings cascade menu of the Values window as follows:

- Sequences
- Sections
- Crash Parameters

- Vehicle Dynamics
- Report
- Use Template
- Report Settings.

For example, selecting Vehicle Dynamics displays the vehicle position, speed and rotation at intervals defined in the Last Branch section of **Display Settings** . Selecting Crash Parameters displays all the crash values, including Delta V.


VALUES

File Edit Settings

1.COLLISION

Vehicle :	1 AC-ACE/A	2 AUDI-100
Driver :		
t [s]:	0.00	0.00
Pre Impact vel. [km/h]:	**52.01**	**53.46**
Post Impact vel. [km/h]:	**45.90**	**29.50**
Velocity change (dV) [km/h] :	27.38	31.92
Deformation depth [m] :	0.24	0.26
EES [km/h] :	29.61	31.98
Stiffness [kN/m]:	1923.2	1603.3
Coefficient of restitution (e) :	0.32	
Separation speed [km/h]:	7.3	
Friction coefficient (mu) :	0.54	
Point of Impact x [m] :	0.64	
Point of Impact y [m] :	-0.73	
Angle of contact plane (phi) [deg] :	0.00	
Total Deformation Energy [J] :	109277.71	
Impulse [Ns] :	12281.96	
Direction of impulse [deg] :	118.24	
Moment arm about C.G. [m] :	0.22	0.99
PDOF (SAE) [deg] :	61.76	-28.22
dV/EES :	0.92	1.00

Overall EES:

1 AC-Ace/Ace [km/h]:	29.6
2 Audi-100 2 [km/h]:	32.0

Characters: 827

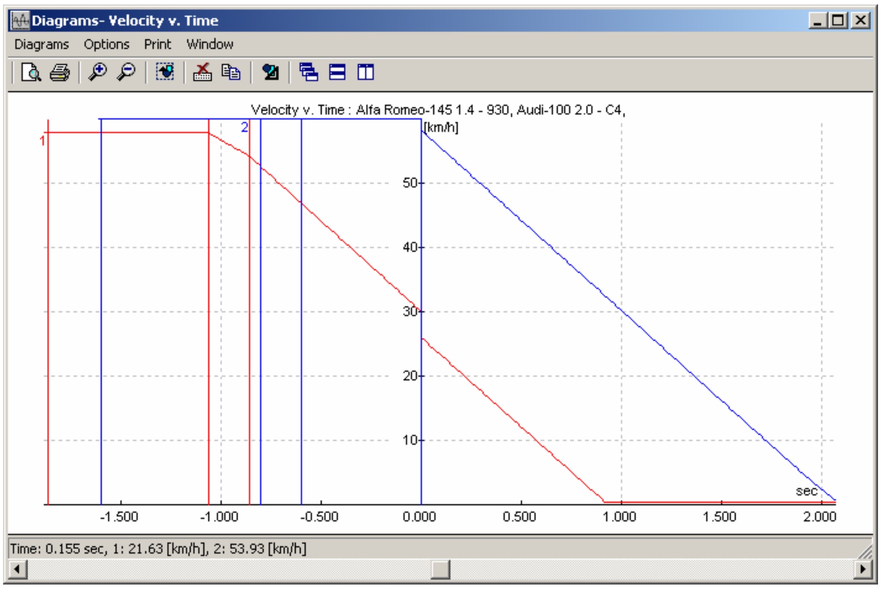
To view any of the vehicle parameters listed below the line in the Values – Settings menu option list, any option above the line that has been selected must be turned off by selecting it again.

The Values window also displays the total character count of the current report displayed including space characters.

Diagrams

Diagrams of variables can be viewed with respect to distance or time. Open the Diagrams window using **Options** - **Diagrams** (F2).

The default diagram in PC-Crash is the distance-velocity diagram. Many other values can be graphed, by selecting them from the **Diagrams** menu option in the Diagrams window.



Multiple diagrams can be viewed at one time in different windows using the **Window – Open new diagram window** menu option in the Diagrams window. Any of the diagrams can be printed with the **Print** menu in the Diagrams window. If multiple diagrams are displayed, they will be printed on a single page.

External Program Interface

Exporting PC-Crash information to word processing programs or spreadsheet programs can be done easily.

Exporting to the Windows Clipboard

The main screen can be copied by selecting **File - Copy Screen**. The copied window can then be inserted in your document using the Edit - Paste or Paste Special functions.

Any active PC-Crash window can be copied by selecting the keys ALT-PRINT SCREEN and then pasted into another document.

Values window text or other output window text can be copied using the Values window **Edit – Copy** menu item or by right-clicking on the selected text and selecting **Copy** from the menu that appears.

Exporting using DXF Files

The simulation results on the main screen can be saved as a DXF file. Select **File – Export - DXF** after finishing the simulation. This stores the screen contents (not including bitmaps) based on the settings in **Options – Options - Display Settings** , in a DXF file. This file can then be imported into a word processing program or a CAD program.

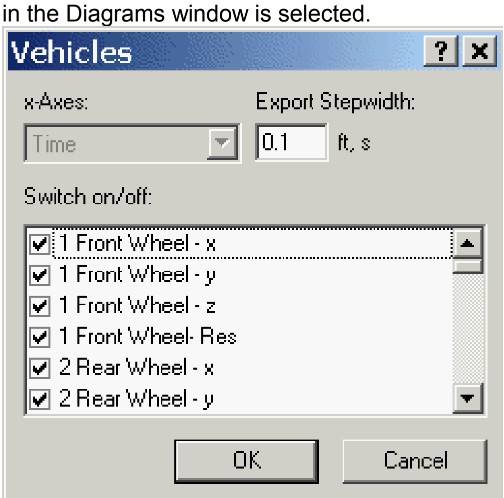
Because vector graphics and not bitmaps are exported in DXF files, there are two advantages with this procedure: the file requires less memory, and the printing quality of lines in a word processing program is improved.

Exporting Diagram Data

The diagram data can be exported in two ways:

- Select the **Diagrams** window menu option **Diagrams – Export Diagram**. This will save the diagram information as a text file.

- Select **Options – Copy** (CTRL +C) in the Diagrams window. This will allow the diagram to be pasted either as an image or a text file, depending on the option chosen under Paste Special in the destination document. The text file step width is selected in the Vehicles window that appears when **Options – Diagrams/Axes** in the Diagrams window is selected.



Example Truckload

At a velocity of 95 km/h (according to tachograph) the LKW driver detects a radar box and initiates a braking in an elongated left bend. By the braking the locked charge - 9 Wire Rod Coils in a series - slips forward against the front wall, whereby the brake hoses are damaged, so that the wheels of the plank bed semi trailer block. 2 of the Coils make are lost. The truck stops on a waters bridge, after an exit.

1. (a) Load vehicles or (b) define vehicles

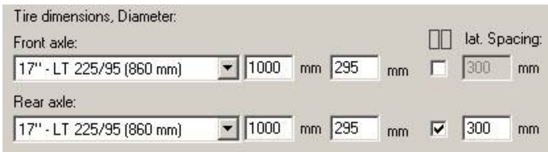
(a) <File> <Import> <Custom Vehicle...> or  and <Open> the files.

(b) <File> <Import> <Custom Vehicle...> or  and New.dat <Open>

<Vehicle> <Vehicle settings...>, “Vehicle data“ – window <Vehicle Geometry>

Name	truck	semi trailer
Driver		
No. of axles	2	3
Length	6 m	13 m
Width	2.5 m	2.5 m
Height	1.1 m	1.2 m
Front overhang	1.2 m	8.5 m
Track – Axle 1	1.9 m	2 m
Track – Axle 2	1.9 m	2 m
Type	Truck	Semi-trailer
Weight	10800 kg	6000 kg
Distance of C.G. from axle	1.9 m	-1.35 m
C.G. height	1.1 m	1.1 m
Moments of Inertia	*calculated autom.*	*calculated autom.*
ABS		
Wheelbase 1-2	3.8 m	1.3 m
Wheelbase 2-3		1.3 m

<Vehicle> <Tire model...>, “Tire model“ – window <General> for the truck Front axle Diameter 1000 mm and for the Rear axle 1000 mm and dual wheels with lat. Spacing 300 mm. For all wheels set a width of 295 mm.



2. *Couple vehicles*

<Vehicle> <Vehicle settings...>, “Vehicle data“ – window <Trailer>

Vehicle	truck
Trailer	semi trailer
Trailer type	*Semi trailer*
Drawbar length	7 m
Hitch overhang	-2 m
Hitch height	1.1 m
Moment transfer x-axis	
Phi 0	0 °
Phi min	0 °
S	5000 Nm/°
S0	0 Nm
Moment transfer y-axis	
Phi 0	0 °
Phi min	15 °
S	5000 Nm/°
S0	0 Nm
Moment transfer z-axis	
Phi 0	0 °
Phi min	0 °
S	0 Nm/°
S0	0 Nm

3. *Suspension properties*

<Vehicle> <Vehicle settings...>, “Vehicle data“ – window <Suspension properties> for the “truck” select the Suspension Properties Stiff

<Vehicle> <Vehicle settings...>, “Vehicle data“ – window <Vehicle Geometry>

For the “semi trailer” define a Weight of 30 000 kg (considers the additional load) and then select the Suspension Properties Stiff within the “Vehicle data“ – window <Suspension properties> for the “semi trailer”. Afterwards reset the Weight to 6 000 kg.

4. *Vehicle Dxf*

<Vehicle> <Vehicle-DXF...> within the “Vehicle-DXF“ – window for the “semi trailer“ select <New…> and activate <Edit drawing>.

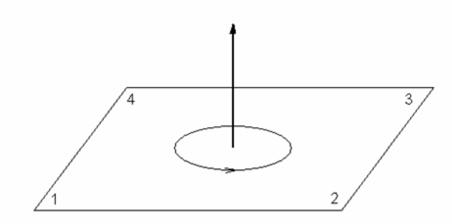
Delete the drawing and select (rectangle). Draw a rectangle around the plan view and select a color with Color and filled within the Drawing-window (change line styles) for the semi trailer. Select the rectangle and move it using the system menu of the Drawing-window (click on the Windows symbol), select <Shift selected (3D) ...> and define 1.2 m for the z-axis within the “Shift selected“ - window.

Perform the same procedure for the „truck“ and define 1.0 m for the z-axis.

The front wall of the semi trailer can be defined using the Drawing-window, for example draw any rectangle and select (Change object) where the coordinates can be defined.

	1	2	3	4	
x:	7.15	7.15	7.15	7.15	m
y:	-1.25	-1.25	1.25	1.25	m
z:	1.2	3.2	3.2	1.2	m

Important: the points for the contact areas must be defined in such a way that the normal vector shows toward the contact.



5. *Multibody System definition*

To be able to operate with the Multibody-system a Multibody system must be loaded in a first step.. <File> <Import> <Custom Vehicle...> or  and file type: Multibody system (*.mbdef)

(PCCrash73/Multibody/Block.mbdef)

With <Vehicle> <Multibody system> in the “Multibody system“ - window this Multibody-system can be modified. Within the “Multibody system“ - window <Bodies> this element can be adapted.

Name	Block1
System ID	30000
Geometry	
a	0.65 m
b	0.65 m
c	0.56 m
n	4
Mass	2800 kg
Moments of inertia	*calculated autom.*
Stiffness	400000 N/m
Restitution	0.1
Friction-Cars	0.4
Friction Ground	0.5
Color 1	128/128/128
Color 2 (changes to red during contact)	192/192/192

In the “Multibody system“ - window <Bodies> additional bodies can be inserted with <Insert>. Definitions are equal as described above. This process must be executed for all 9 bodies. For each body different System ID should be used to facilitate the following positioning and inserting of joints.

Important: After die definition of the bodies the “Multibody system“ – window should be closed once with <OK> for initializing all body data.

6. *Multibody System settings*

The individual bodies can be arranged in the “Multibody system“ – window <Settings>. In the first combo box a selection can be made whether the inputs refer to an individual body or to all systems. For the individual bodies the following specifications are made (select a single system and act. system). For all systems select <Syst. Properties> and activate 3D Dxf car contact. If for all bodies a System ID 3xxxx was defined:

System	*Pedestrian 1*	*Pedestrian 2*	*Pedestrian 3*	*Pedestrian 4*	*Pedestrian 5*
vxy	95 km/h	95 km/h	95 km/h	95 km/h	95 km/h
PhiVel	0 °	0 °	0 °	0 °	0 °
vz	0 km/h	0 km/h	0 km/h	0 km/h	0 km/h
xmin	-12.3 m	-11.1 m	-9.35 m	-8.17 m	-6.45 m
ymin	-0.65 m	-0.919 m	-0.65 m	-0.919 m	-0.65 m
zmin	1.2 m	1.2 m	1.2 m	1.2 m	1.2 m

Phi	0 °	45 °	0 °	45 °	0 °
System	*Pedestrian 6*	*Pedestrian 7*	*Pedestrian 8*	*Pedestrian 9*	
vxy	95 km/h	95 km/h	95 km/h	95 km/h	
PhiVel	0 °	0 °	0 °	0 °	
vz	0 km/h	0 km/h	0 km/h	0 km/h	
xmin	-5.3 m	-3.61 m	-2.47 m	-0.75 m	
ymin	-0.919 m	-0.65 m	-0.919 m	-0.65 m	
zmin	1.2 m	1.2 m	1.2 m	1.2 m	
Phi	45 °	0 °	45 °	0 °	

7. Multibody System Spring/Damper

The bodies can be joined with the vehicle using “Multibody system“ – window <Spring/Damper>. Select the single systems in the first combo box and insert a new spring/damper joint using <Insert>. The second reference body is vehicle 2.

Joint location 1	*x*	*y*	*z*	*Joint location 2*	*x*	*y*	*z*
Block 1	*0*	*-0.65*	*0.5*	*Vehicle 2*	*-5.1*	*-1.2*	*0.2*
Block 1	*0*	*0.65*	*0.5*	*Vehicle 2*	*-5.1*	*1.2*	*0.2*
Block 2	*-0.45*	*-0.45*	*0.45*	*Vehicle 2*	*-3.62*	*-1.2*	*0.2*
Block 2	*-0.45*	*0.45*	*0.45*	*Vehicle 2*	*-3.62*	*1.2*	*0.2*
Block 3	*0*	*-0.5*	*0.5*	*Vehicle 2*	*-2.13*	*-1.2*	*0.2*
Block 3	*0*	*0.5*	*0.5*	*Vehicle 2*	*-2.13*	*1.2*	*0.2*
Block 4	*0.65*	*-0.65*	*0*	*Vehicle 2*	*-0.73*	*-1.2*	*0.2*
Block 4	*0.65*	*-0.65*	*0*	*Vehicle 2*	*-0.73*	*1.2*	*0.2*
Block 5	*0*	*0.5*	*0.5*	*Vehicle 2*	*0.74*	*1.2*	*0.2*
Block 5	*0*	*-0.5*	*0.5*	*Vehicle 2*	*0.74*	*-1.2*	*0.2*
Block 7	*0*	*0.5*	*0.5*	*Vehicle 2*	*3.62*	*1.2*	*0.2*
Block 7	*0*	*-0.5*	*0.5*	*Vehicle 2*	*3.62*	*-1.2*	*0.2*
Block 8	*-0.45*	*-0.45*	*0.45*	*Vehicle 2*	*5*	*-1.2*	*0.2*
Block 8	*0.45*	*0.45*	*0.45*	*Vehicle 2*	*5*	*1.2*	*0.2*
Block 9	*0*	*-0.5*	*0.5*	*Vehicle 2*	*6.42*	*-1.2*	*0.2*
Block 9	*0*	*0.5*	*0.5*	*Vehicle 2*	*6.42*	*1.2*	*0.2*

For all Spring/Damper systems:

trans. stiffness	400000 N/m
Damping	1000 Ns/m
Fmax	11500 N
Tension	6000 N
Tension only	x

8. Multibody System contacts

The contacts between the systems can be activated/deactivated within “Multibody system“ – window <Contacts>. Activate the contacts for the single systems.

9. Sequences

The sequences for the vehicles can be defined with <Dynamics><Sequences>.

Truck		**Semi trailer**	
Sequence1	***Dec./St.***	***Sequence1***	***Dec./St.***
Brake	x	*Brake*	x
duration	1.5 s	*duration*	1.5 s
a	4.00 m/s²	*a*	4.00 m/s²
Steering	x		
Turning circle	500 m		
Time	0.5 s		
Name	***Dec./St***	***Name***	***Dec./St***
Brake	x	*Brake*	x
Geometry	1.0 s	*Geometry*	1.0 s
a	4.00 m/s²	*a*	4.00 m/s²
Steering	x		

Turning circle	0 m		
Time	0.5 s		
Name	***Decel***	***Name***	***Decel***
Brake	x	*Brake*	x
duration	10 s	*duration*	10 s
a	4.00 m/s²	*a*	7.85 m/s²
		Brake factors	500

10. Visualization

For the visualisation and the video rendering a Dxf-drawing can be attached to the “truck”.

<Vehicle> <Vehicle-DXF...> within the “Vehicle-DXF“ – window for the “truck“ select <File…> <Plan view> <Load DXF...> and select a proper file.

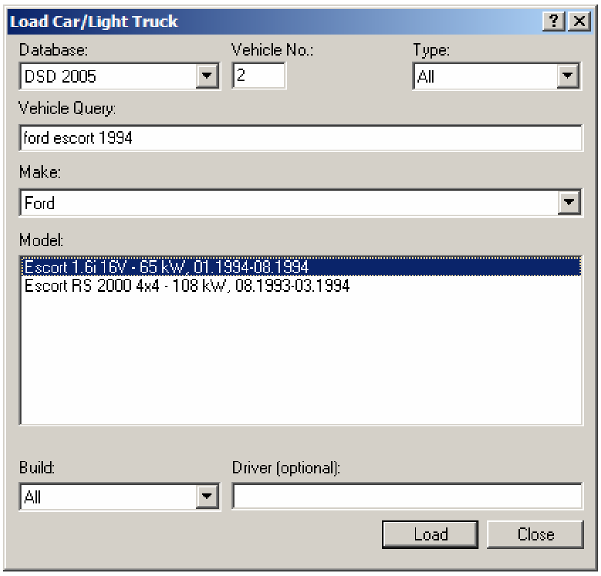
Creating vehicle interior for occupant simulation

Most 3D vehicle models in PC-Crash have no interior assigned to the model. Thus, occupant movements are only meaningful a short period of time after the impact because contacts with the interior is missing.

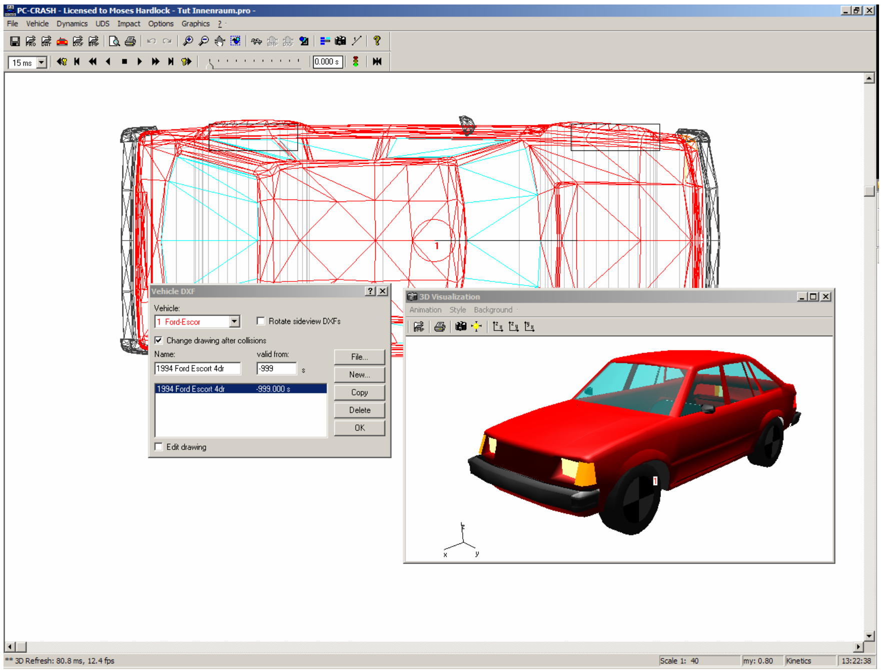
To make better statements about occupant movements and injury mechanics the existence of a basic car interior is an essential requirement.

This example will show how to create a basic car interior using the Extrude function of PC-Crash.

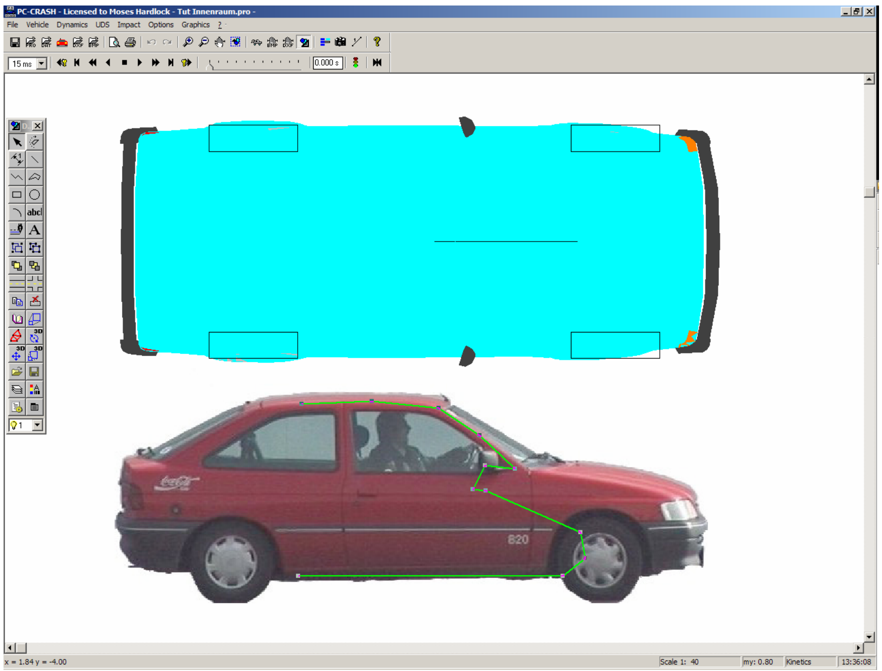
For the beginning load a car - the *Ford Escort 1.6i – 65KW 1994* - from the DSD database.



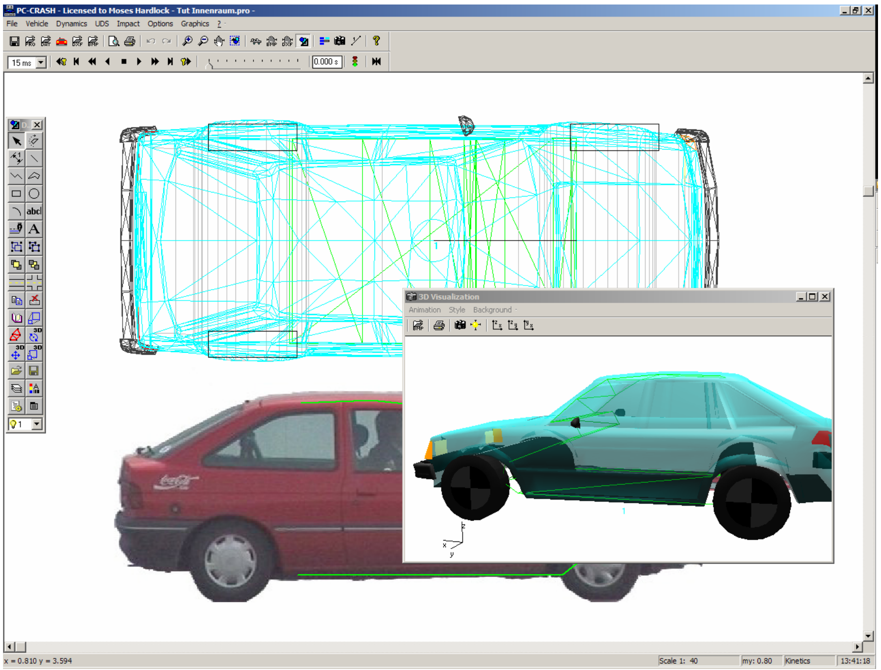
After loading the vehicle a 3D DXF model has to be assigned for the car exterior by selecting the command **<Vehicle> <Vehicle DXF…> <File> <Plan View> <Load DXF>** from the menu. Select the file `3DDxf\cars\1994 Ford Escort 4dr.idf` from the PC-Crash directory.



To create the car interior a bitmap or DXF side view drawing can be used. In this example use the side view bitmap of the Ford Escort and scale it using the command **<Graphics> <Bitmap> <Scale>** to a total length of 13.58 ft. Then draw a ploy-line on the bitmap outlining the approximate car interior as shown in the picture below.

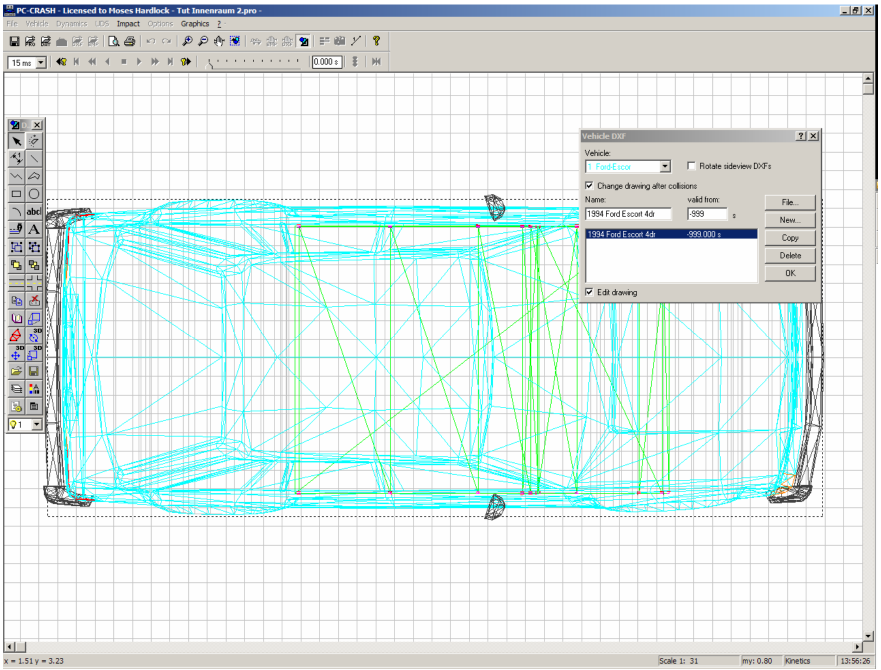


Extrusion can be used to make a 3D Model of a 2D shape. To create a 3D interior of the just created poly-line select it and use the **Extrude** button to extrude the line in z direction about the vehicle width of 4.69 ft. Now rotate the extruded object 90° about the x-Axis to match the orientation with the Ford Escort. Move the object into the vehicle. Use the 3D window to adjust the z position of the interior.



If the interior fits in the car use the **Save Object** button to save the interior as *.idf file. The *.idf format saves the surface normals, which are necessary for computing the contacts with the occupant later. *.dxf files do not store surface normals.

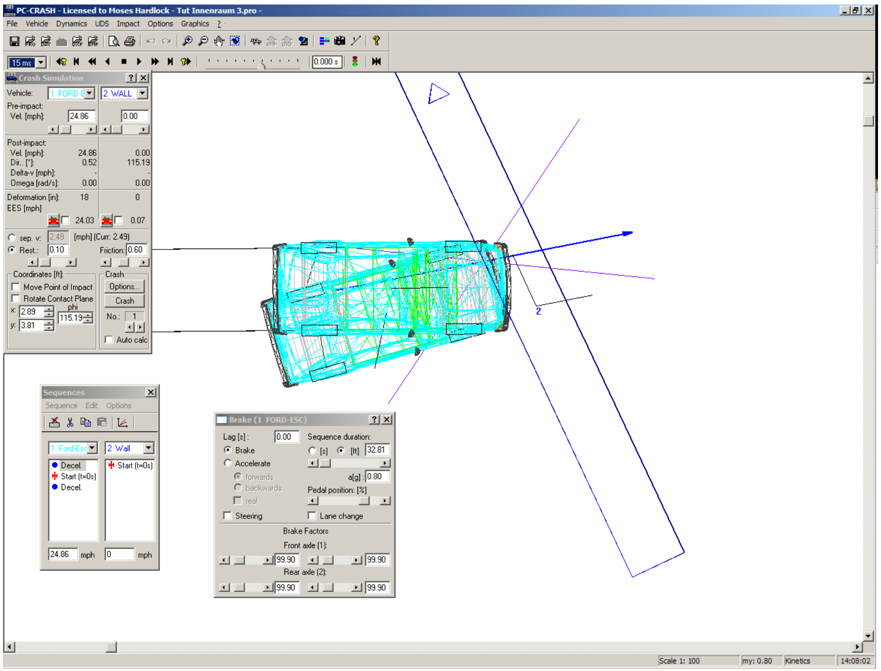
To assign the interior to the vehicle it has to be loaded as part of the vehicle DXF. Use the command **<Vehicle> <Vehicle DXF…>** and then the option **edit drawing** to change the vehicle DXF. Use the **Insert object** button from the drawing program to add the just saved interior object to the vehicle. Use the 3D window to verify the position of the interior again.



It is now time to start with the simulation. The first step of an occupant simulation is the crash of the car. After the impact results are satisfying, the occupant can be added.

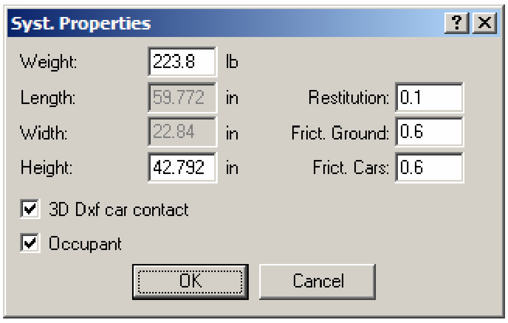
In this example a crash against a rigid wall is simulated with an impact velocity of 24.8 mph and a braking distance of 32.8 ft. Use the sequences window to create the appropriate sequences.

Then the wall `WALL.DAT` has to be loaded by using the command **<File> <Import> <Custom Vehicle…>**. Place the two objects in the impact positions and calculate the impact using the command **<Impact> <Crash Simulation…>** and the pre- and post impact movements.

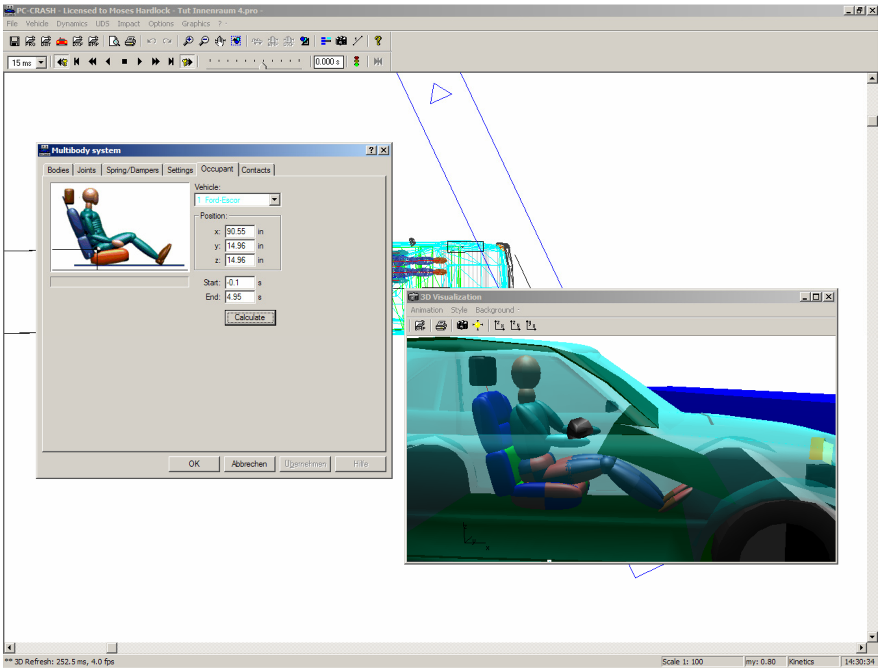


If you are satisfied with the impact motion the occupant can be inserted. Select the command **<File> <Import> <Custom Vehicle…>** from the menu to load the multi-body occupant `Seat + Occupant 010910.mbdef`. The occupant is comprised of a person and a seat without belts.

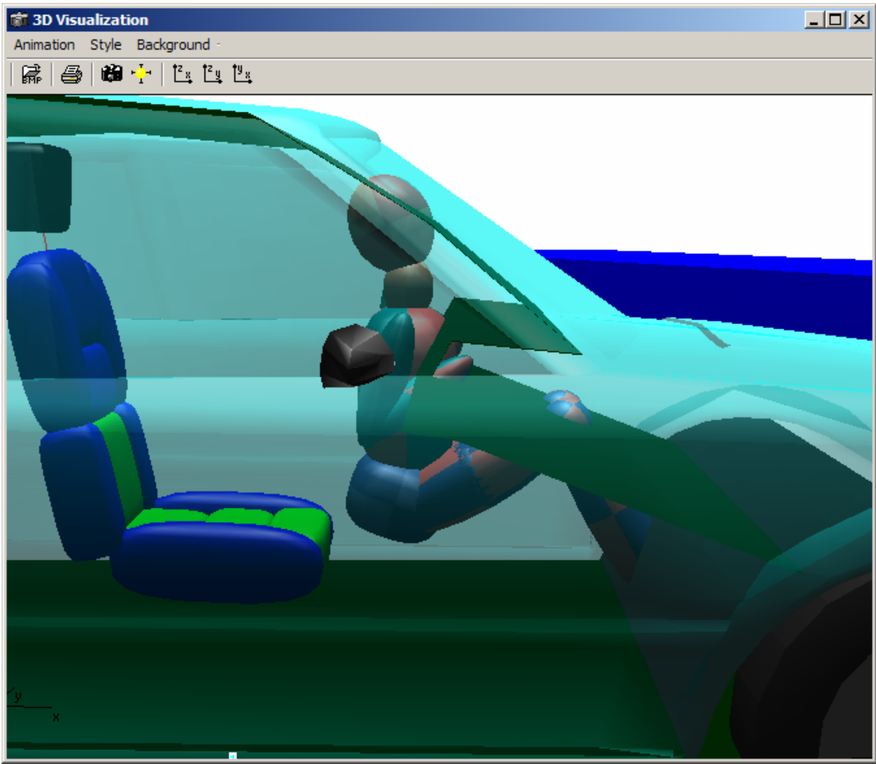
Select the command **<Vehicle> <Multibody System>** to open the multi-body dialog. On the page **Settings** open press the button **Syst. Properties** to select the option **3D Dxf car contact**. This option has to be active to make the multi-body system recognize the DXF car body with the interior.



On the page **Occupant** the seat position can be specified. By selecting the value 0 for both, the start and end times and pressing the **Calculate** button, the position of the multi-body system can be verified in the 3D window without calculation. No part of the system must penetrate any part of the vehicle.



Then specify the start and end times for the occupant simulation and press the **Calculate** button. You can now use the 3D window to watch the occupant hit the car interior.



Chapter 3
Menu Description

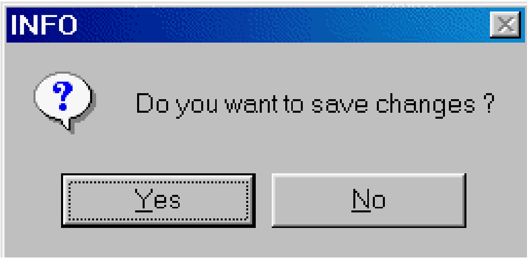
The menu descriptions are presented in the order that they appear, from left to right across the top of the PC-Crash main screen.

File

This menu is for loading, saving and printing operations.

New

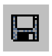
This option is for defining a new project. When selected, the following Info dialog box appears. If the No button is pressed, all existing data is deleted and the program returns to the same configuration as when it is initially opened. Pressing the Yes button returns the program to the state it was in prior to selecting this menu option, so that the user can save the project.



Load

Open a previously saved project. A dialog box appears in which the file can be loaded from any directory selected in it. After a project is loaded, the name of the project file will be shown in the Title Bar next to the license name.

Save

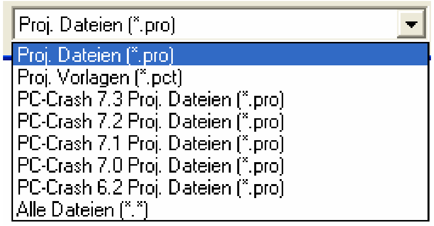


This tool saves the current project using a previously defined name. If no name was defined, the Project Save dialog box is opened and the user must type a name in the File name text box. All the vehicle positions and settings are automatically saved.

Save As

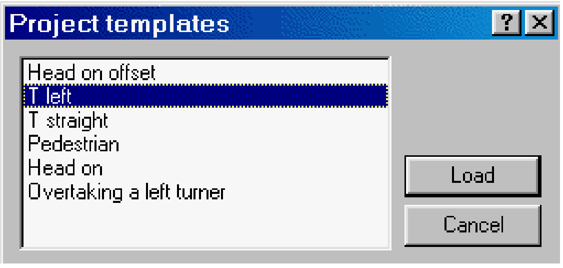
This allows the user to change the name or directory location of the project to be saved. During operation of the Save button all active data are stored in a file with the specified name (file name) and the selected path.

Additionally the PC-Crash program version can be defined under type of file; this determines the format of the project file and the program version respectively.



Project Wizard

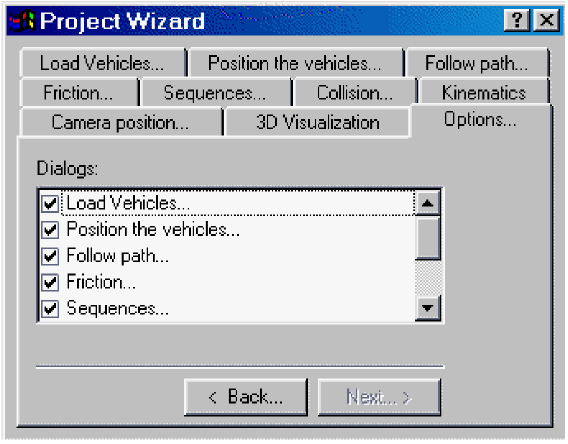
Project Wizard allows quick setup of a new project by guiding the user through most of the necessary steps to set up a project. In a new project, the following dialog box appears when Project Wizard is selected. One of the provided templates can be selected from this list.



Additionally, users can create their own custom templates. This is done by setting up a project as desired, and then saving it as a project template file (*.pct) in the PC-Crash template directory in, using **File – Save As**. You can change the PC-Crash template directory on the dialog page **Directories** accessed by selecting the menu option **Options – Options…**.

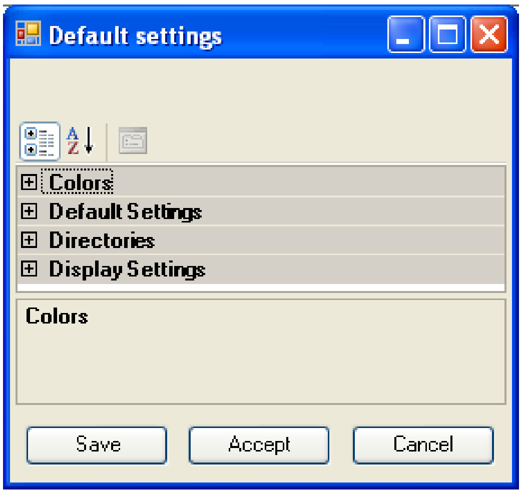
After a project template is selected, or if Project Wizard is selected after a project has been started, the following group of dialog boxes appears. Using the Next button after each operation is completed will open the dialog boxes in a logical order to complete a project.

The Options dialog box in the Project Wizard group allows the user to define the operations that will be included.



Default Settings

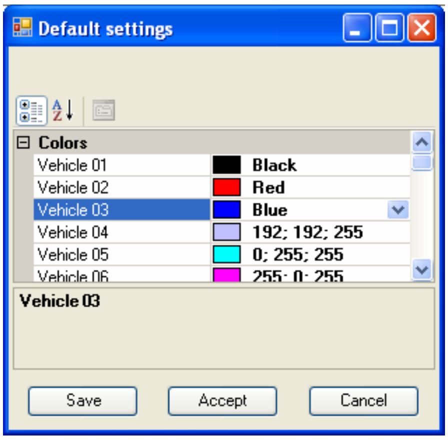
The menu option **File – Default settings**. A dialog box appears with consists of 4 categories (Colors, Default Settings, Directories, Display Settings)



Using the Button  sorting according to the categories and using Button  alphabetically sort can be done. During the alphabetical sorting all basic adjustment options are indicated, during the sorting of categories main categories with the appropriate submenus are displayed. These can be extended and/or reduced via the symbols  .

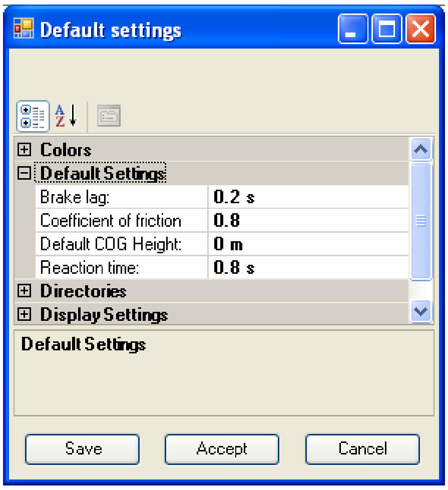
Colors

For the color adjustment of the different vehicles.

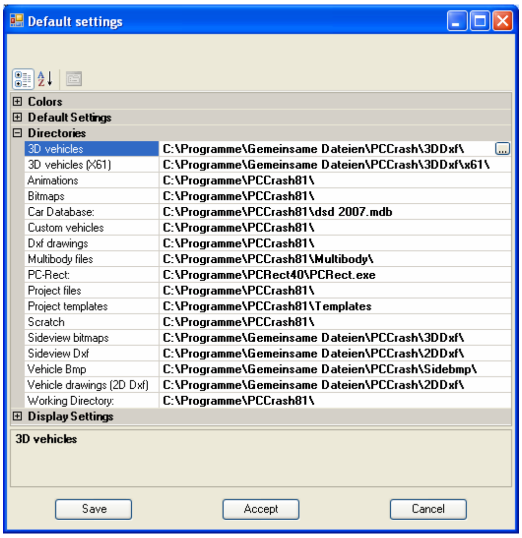


Default settings

Global defaults for PC Crash. Brake lag, global coefficient of friction, default center of gravity height and reaction time are defined. These values are defined global but can be adapted and changed however at any time via the appropriate menu options.



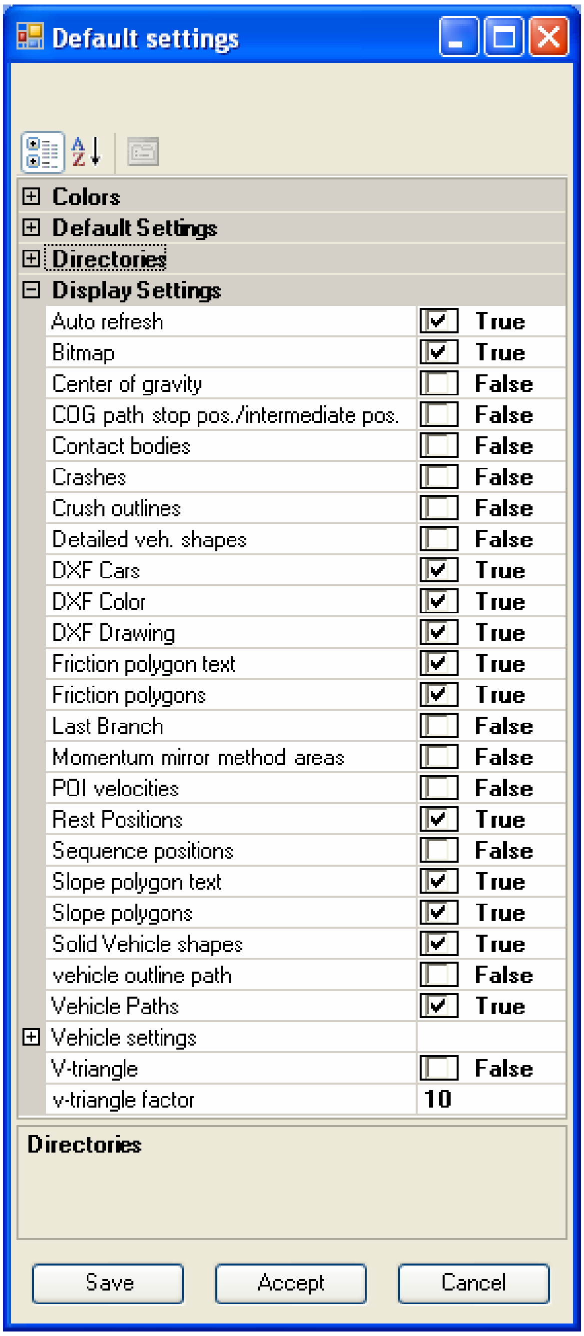
Directories



3D vehicles	<Vehicle> <Vehicle Dxf…> <File…> <3D Mapped> <Load 3D vehicle…>
3D Fahrzeuge (X61)	
Animationen	<Options> <3D Window> <Animation>
Arbeitsverzeichnis	
Bitmaps	<File> <Import> <Bitmap> <File> <Export> <Bitmap>
Dxf Zeichnungen	<File> <Import> <Dxf Drawing> <File> <Export> <Dxf Drawing >
Fahrzeug Bmp	<Vehicle> <Vehicle Dxf …> < File …> <Plan View> <Load Bmp …>
Fahrzeugdaten	<File> <Import> <Custom vehicle> <Datei> <Export> < Custom vehicle>
KFZ-Datenbank	<Vehicle> < Vehicle Database>
KFZ-Zeichnungen (2D Dxf)	<Vehicle> < Vehicle -Dxf…> <File…> <Plan View> <Load Dxf …>
Mehrkörpersysteme	
Photoentzerrung (PC-Rect)	<Options> <PC-RECT>
Projektdateien	<File> <Save as>
Projektvorlagen	<File> <Project Wizard…>
Scratch	
Seitenansicht Bmp	<Vehicle> < Vehicle Dxf…> <File…> <Side view> < load Bmp …>
Seitenansicht Dxf	<Vehicle> < Vehicle Dxf…> <File…> <Side view> < load Dxf …>

Display Settings

Parameters for the representation and the appearance of PC Crash. For detailed description see Display Settings beginning from page 213.



Import

Scan

If a scanner is present on the user's operating system, this option enables pictures to be scanned directly into PC-Crash.

Scanner Selection

If one or more scanners are present on the user's operating system, this option can be used to select the desired scanner.

Bitmap 

This tool imports bitmaps of plan view scene drawings or photographs (aerial view or those rectified into a plan view with PC-Rect), for use as an underlay to the simulation. A bitmap resolution of 100-300 dpi is adequate in most cases.

PC-Crash supports bitmap graphic files in the following formats:

- Bitmap (*.BMP)
- Encapsulated Postscript (*.EPS)
- Graphics Interchange Format (*.GIF)
- JPEG (*.JPG)
- PCX (*.PCX)
- TIFF (*.TIF).

Tools for scaling, rotating and moving the bitmap after importing are included in the **Bitmap** menu. There are several options for Bitmaps by selecting the menu command **<Options> <Scale & Grid Spacing…>**. These options include selecting, rotating, scaling, moving and hiding bitmaps.

DXF Drawing 

This tool imports scene DXF and VRML drawing files, such as those created with external drawing programs (AutoCAD, AutoSketch, etc.), for use as an underlay to the simulation.

The following DXF objects can be interpreted by PC-Crash:

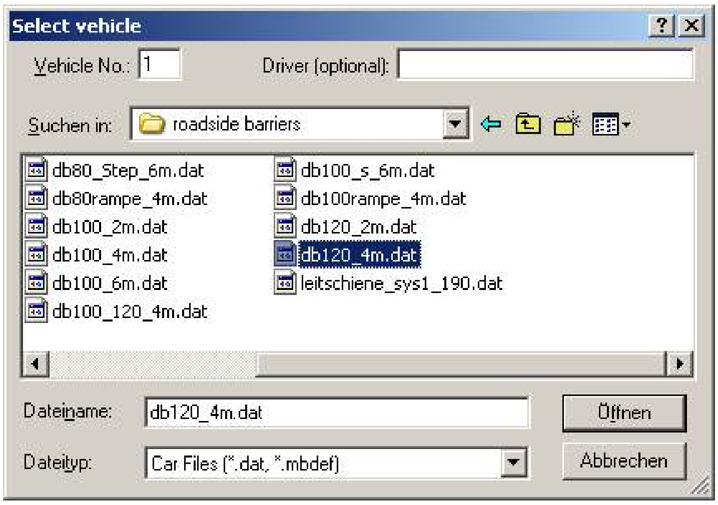
- LINE
- CIRCLE
- ARC
- POINT
- SOLID
- TRACE
- TEXT

Plines are changed into normal lines. Non-basic colors are not recognized, and will be changed to black. Thus, some objects many have to be redrawn or re-colored after importing. Grouping objects by layers is normally a better choice than using blocks, because PC-Crash groups objects on the same layer together as a block.

PC-Crash assumes imported drawings are scaled in meters. If drawn in other units, they must be re-scaled. Tools for scaling, rotating, moving and modifying the drawing after importing are included in the **Drawing** menu.

Custom Vehicle 

This loads custom vehicles (those not in a database). A number of custom vehicles are provided in the directory PC-Crash is installed in. Multibody pedestrians and two-wheeled vehicles are included in the Multibody subdirectory.

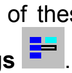


Export

Bitmap

This saves bitmap files that have been modified in PC-Crash.

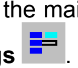
DXF Drawing

This saves DXF files created or modified in PC-Crash. Note that in addition to the drawing components, all vehicle positions that appear on the screen, along with the tire marks, will be saved in the DXF file. If it is desired that some of these not be saved in the DXF file, then they must be turned off in **Display Settings** .

Custom Vehicle

Vehicles modified in PC-Crash can be saved. The vehicle data files are simple ASCII data files. They can be created and modified with any text processor. However, it is easier to load an existing vehicle, modify it in the Vehicle Geometry dialog box, and then save it under the desired name in the Select Vehicle dialog box.

Print (CTRL + P)

This prints a scale scene drawing from the main screen. The printout depends on the parameters selected in **Display Settings** .

When activating this menu option a window appears, which allows print parameters to be changed before printing.

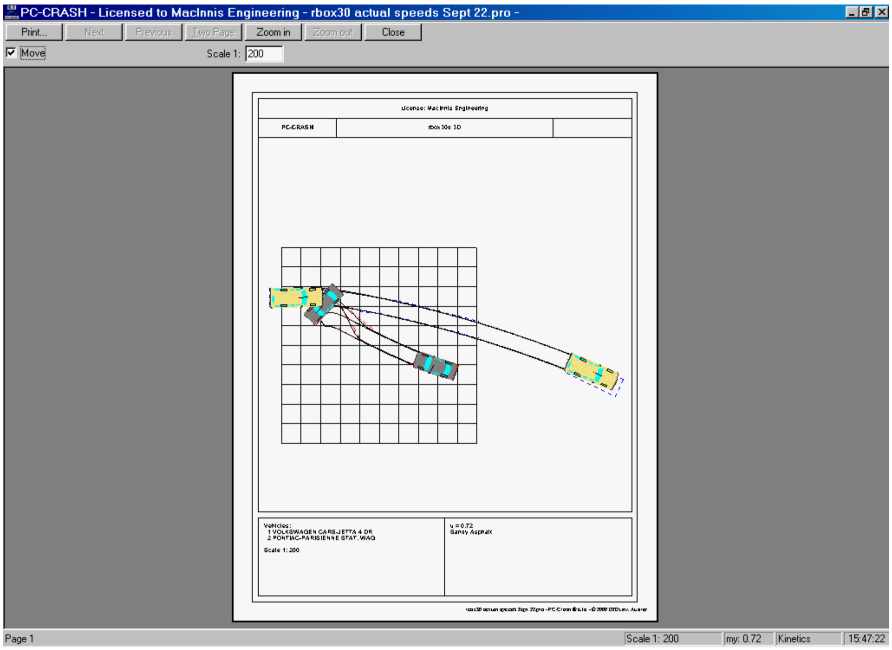
The default scale of the printout will be that of the main screen, which is indicated in the status bar. The scale can be changed with the Zoom In and Zoom Out tools , or with the menu option **Graphics – Scale & Grid Spacing**. If **Print Preview** is selected instead of **Print**, the scale and position of the printout can more easily be adjusted to suit the page (see next item).

Print comments can be added to the printout using the menu option **File – Print Comments / Template**.

Print Preview

(F10)

After selecting this option, a print preview is displayed.



Move and Scale at the top of the Print Preview window allow the printout to be maximized in the page space available, without affecting the main screen image.

When selecting **Print** from this window the file is sent to the printer.

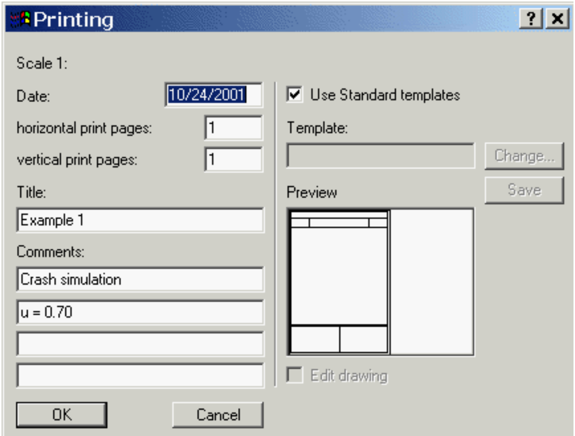
Print Comments / Template

This option allows comments to be printed with each printout. These comments are saved and can be modified any time.

The date can be modified or deleted. The text of the first line is printed as a title at the top of the page. The scale of the printout is displayed.

There is also the option of creating custom page layouts.

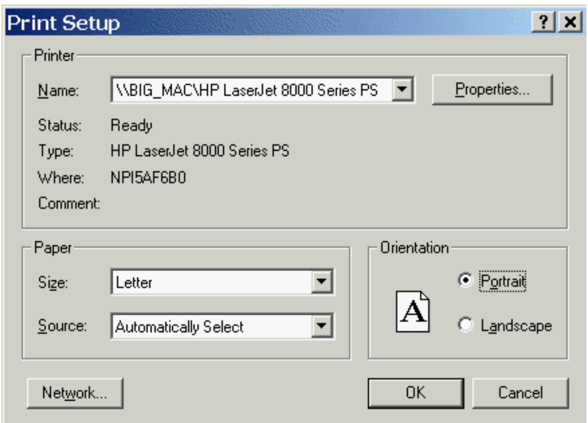
By selecting Edit Drawing, the template can be modified using the drawing tools. The modified template can be stored as file in a desired directory using the Save button. The modified template can then be opened by deselecting Use Standard Templates, pressing the Change button and selecting the appropriate path.



Using Horizontal Print Pages and Vertical Print pages, the printout of the main screen can be done over several pages, which can then be glued together to make a larger scale drawing.

Printer Setup

This allows printer properties, paper size and orientation to be changed. The following Windows dialog box appears:



See also the previous Print Comments/Template section for printing the main screen over several pages.

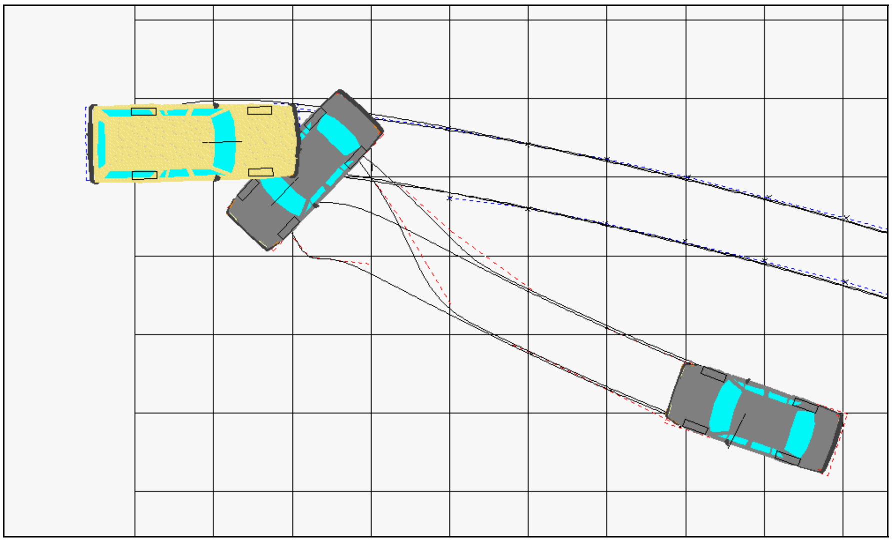
Hardcopy (F12)

This gives a 1:1 copy of the screen on the printer. The printing is displayed on the screen in print preview form.

Note: *This menu option can only be activated if the screen content has been copied into the Windows Clipboard first, using the* PRINT SCREEN *key or the* ***Copy Screen*** *menu item. The* PRINT SCREEN *key copies the whole screen including title bar while the* ***Copy Screen*** *menu option copies only the main window (see next menu item).*

Copy Screen (CTRL C)

This copies the main window contents to the Windows Clipboard, for pasting into other Windows programs or for printing (see previous menu item).



The above was copied to the clipboard using **Copy Screen** and then pasted into this manual using Edit - Paste. The frame was added afterwards.

Print Report

A text report of all the input and output values of the current PC-Crash project is printed, after showing a print preview. This can also be done from the Values window (select **Options - Values - Settings - Report**, where a print preview is also available). The contents of the report can be changed using **Options – Values – Settings – Report Settings**.

Recently Loaded Projects

The last 4 projects loaded are listed at the bottom of the **File** menu and can be selected from this area directly.

Exit

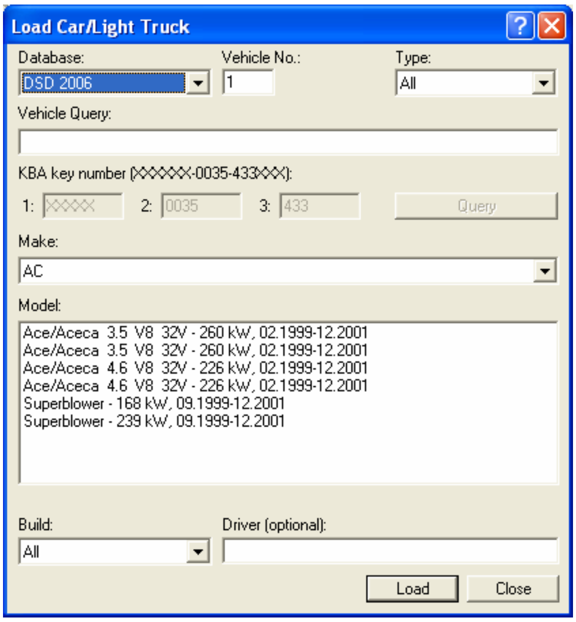
Closes PC-Crash.

Vehicle

This menu is for loading, saving and modifying vehicles.

Vehicle Database

This tool is for loading vehicles contained in a database. It brings up a vehicle database dialog box, which lists vehicles included in the database.



The Vehicle database Load Car/Light Truck dialog box includes the following:

Database

This allows the selection of database to be used. If all databases are in the same directory, and one of them has been selected in the **Options - Options – Directories - Car Database** box, then any of these databases can be selected from this drop down list.

Currently the following databases are supported:

- Specs (North American)
- ADAC (European)
- Vyskocil (European)
- DSD (European and Japanese)
- KBA (German department for motor vehicles)

Refer to the chapter **"Getting Started"** for more details on setting up the vehicle databases.

Vehicle No:

This will be the identification number of the vehicle being loaded. These are normally consecutively higher numbers as vehicles are added to the simulation project.

However, vehicles that have been previously loaded can be overwritten by changing the Vehicle No. in this text box to that of the vehicle which is to be replaced.

Type (only available for DSD Databases)

Only vehicles of the selected type are displayed, based on the following choices:

- Cars
- Trucks
- Trailers
- Busses
- Motorcycles

Selecting **All** displays all vehicles in the database regardless of their type. After changing the vehicle type the list of manufacturers will be updated.

Vehicle Query

Entering the vehicle make in the Vehicle Query text box will access that make in the left list box, which may be quicker than scrolling down to find the make. It is also possible to search for a certain vehicle. The Vehicle Query text has to start with the year of construction (YY or YYYY, e.g. 99 or 1999) or the manufacturer, whereby the manufacturer must be always entered. The inputs must be separated by blanks, the TAB key starts the search. No differentiation is made between large and small letters.

Search words: [(built YY or YYYY)] (manufacturer) [(search criterion)]

e.g. 96 bmw 520; 1991 MERCEDES 190; peugeot 40 (all vehicles with X40 kW and the 40X series are displayed).

KBA key number (XXXXXX-0000-000XXX)

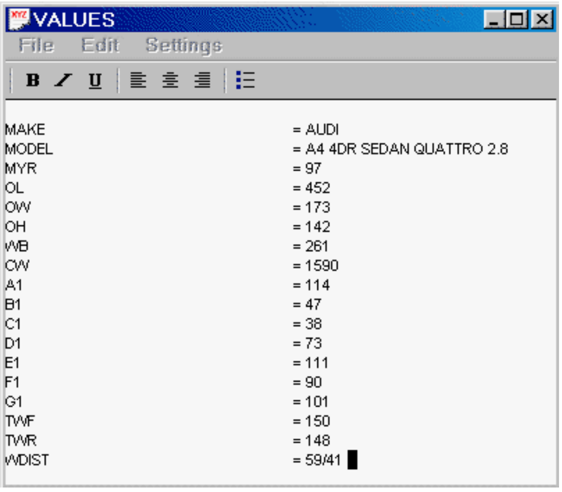
With the KBA field HERTYP (Make: first 4 numbers, Type: digit 5 to 7) a vehicle query can be made. The query fields are activated, if Database KBA xxxx is selected.

Make

The makes of available vehicles in the database are shown in the first (left) list box. By clicking once on the desired make, all the available vehicle models are shown in the second (right) list box.

Model

By clicking once on the desired vehicle model in the right list box, the model details are shown in an additional window. Regardless of whether U.S. or metric units have been selected in the Default Settings dialog box, the dimensions are shown in metric (m and kg) in this window, as shown in the following figure. After loading the vehicle, however, the units are automatically converted to U.S. units when this option has been chosen in **Options – Options – Default Settings**.



Built (only available for DSD Databases)

The user can specify the year of manufacture. Once the user enters a manufacture date in the Build drop down list, only those vehicles that have been built on this date (year of manufacture is between 'built from' and 'built to') will be displayed.

Driver

The name of the driver is optional. If entered, this name will be indicated with the vehicle make and model in the report printout.

Load

Activating the Load button (or double-clicking on the selected vehicle model name) loads the selected vehicle into PC-Crash.

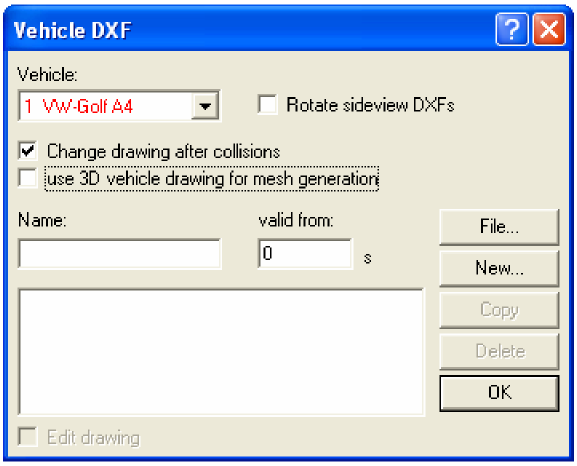
Close

Closes the dialog box after the desired vehicles have been loaded.

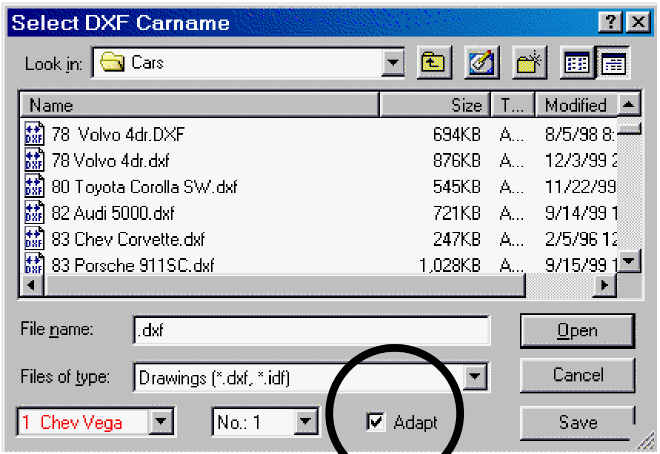
Vehicle DXF

This feature allows 2D or 3D drawing files, or 2D bitmap files to be attached to simulation vehicles.

You can use the provided vehicle shapes, as well your own. Both 2D and 3D vehicle shapes show on the main screen of PC-Crash, but only 3D shapes will show in the 3D window.



For scaling an imported vehicle drawing to the correct size, PC-Crash assumes the drawing was drawn in metric. However, Adapt can be used to scale the drawing to the length of the vehicle, after **File - Load DXF** has been selected in the Vehicle DXF dialog box.



When vehicle bitmap files are imported, they are automatically scaled to the vehicle length.

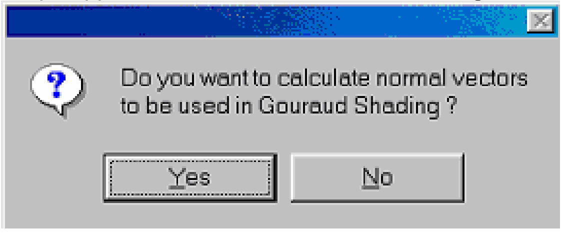
Vehicle shape drawing files must be in DXF, IDF or VRML (*.WRL) format, with the front of the vehicle in the direction of the positive X-axis. DXF (Data eXchange Format) is a standard format used by most CAD programs for exchanging vector drawings. IDF files are similar to DXF files, but are unique to PC-Crash. IDF files have two advantages over DXF files:

- IDF files are smaller than DXF files;
- IDF files can contain normal vector information that smoothes the shape of vehicles when Gouroud shading is selected in the 3D window. An IDF file containing normal vector information can be imported much more quickly than a DXF file, because the normal vectors don't have to be calculated.

The VRML format was initially designed as a textual object-oriented description language for real-time virtual worlds viewed in web browsers. Thus the format contains information about 3D models, lighting and animation. VRML also has proved useful for exchanging 3D models between programs.

The 3D vehicle shapes provided with PC-Crash are in IDF format, with the normal vectors calculated. If the user wishes to change a third party vehicle DXF shape into IDF or VRML format, this can be done as follows:

1. Using **Vehicle – Vehicle DXF - File – Plan View – Load DXF - Open**, import the file and attach it to a vehicle in PC-Crash. If "Yes" is selected when the following information window appears, normal vectors will be calculated so the vehicle shape appears smooth when Gouroud shading in selected in the 3D window.



2. Using **Vehicle – Vehicle DXF - File – Plan View – Load DXF - Save**, export the file as an IDF or VRML file. When this IDF or VRML file is imported, there will be no time delay while normal vectors are calculated.

Conversely, if the user wishes to change an IDF file back into DXF format (i.e. so that it can be imported into a CAD program), import the IDF file into PC-Crash and then export it as a DXF file by selecting the DXF file type when exporting.

Modifying Third Party 3D Vehicle Shapes for PC-Crash

Vehicle 3D files purchased from third party companies must usually be modified in a CAD program for use in PC-Crash. The following details how vehicle shapes need to be modified for PC-Crash, using AutoCAD:

1. **Import** file into AutoCAD.
2. Use **Explode** command and select **All** to explode all meshes & blocks.
3. Use **Scale** command and select **All**, then choose **0, 0, 0** and enter scale-factor (e.g. of **0.3048** to change from feet to metres).
4. Establish height **h** of point **0, 0, 0** above bottom of tires. Use **Move** command and select **All** to move **0, +h, 0.** (e.g. Viewpoint Engineering uses **0, 0, 0** for centre of vehicle, which needs to be at ground level for use in PC-Crash).
5. Use **Rotate3D** command, select **All**, choose **X-axis** and **0, 0, 0** then enter **+90º**.
6. Use **Rotate3D** command, select **All**, choose **Z-axis** and **0, 0, 0** then enter **+90º**.
7. Use **Properties** and select **All** to change colour of entire vehicle from 'By-Layer' to a specific colour by number, such as **#1** for **Red**. Then color individual parts as follows, using **Properties**:

Glass	color **151**	(Transparent Blue for Windows)
Bumpers	color **253**	(Grey for Bumpers)
Headlights	color **50**	(Light Yellow for Headlights)
Brake lights	color **20**	(Red for brake lights, which changes to bright red when braking)
Turn signals	color **40**	(Orange for Turn Signals)
Grille	color **131**	(Steel Blue for Grille/Chrome).
Tires	color **7**	(Black for Tires)
Rims	color **9**	(Grey for Rims)

   Tires and rims should normally be deleted, as PC-Crash vehicles already have tires and wheels.

8. Go to the **File - Drawing Utilities** menu and **Purge - All** to delete blocks and unused layers.
9. **Save** and **Export** DXF file to use in PC-Crash.

10. If any surface polygons on the vehicle have their faces pointing inward instead of outward, triangular holes will appear in the vehicle body when viewed in the 3D window of PC-Crash. To correct this, these polygons must be redrawn. Import the DXF into AutoCAD again, erase these polygons, and redraw them in a counter-clockwise direction (when viewed from the outside).

Modeling Crush Using Different Vehicle Shapes

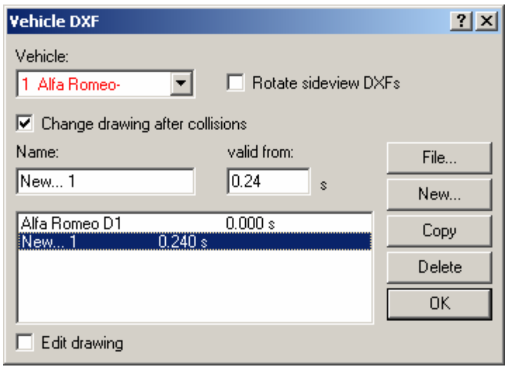
Having vehicle crush occur to a 2D or 3D vehicle drawing shape at the correct time during a simulation can be done in two ways: by importing damaged vehicle shapes, or by using the Deform Drawing feature in PC-Crash.

Importing Damaged Vehicle Shapes

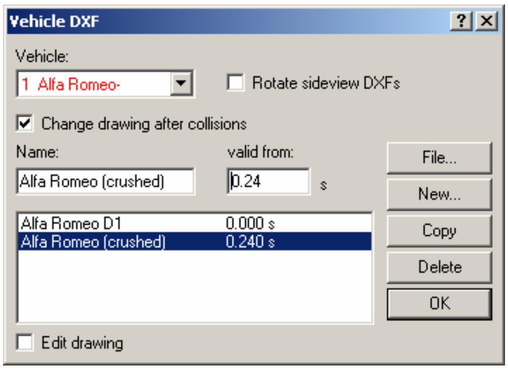
Different drawing shapes made in an external CAD program can be applied to each vehicle at different times so that crush damage from each crash can be shown properly in the simulation. This is done as follows:

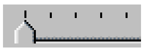
1. Bring in the undamaged vehicle shape using **Vehicle – Vehicle DXF – File – Plan View – Load DXF**. By default, it is valid at the start of the simulation.

2. Select the New button in the Vehicle DXF window. This will set a place for a second shape file following the undamaged one. By default, the New file will be valid from the time of the first crash.



3. Click on the New file to select it, and then use the File button to load the vehicle shape that has been modified with the desired crush in an external CAD program. Make sure ☑ Change Drawing After Collisions is activated.

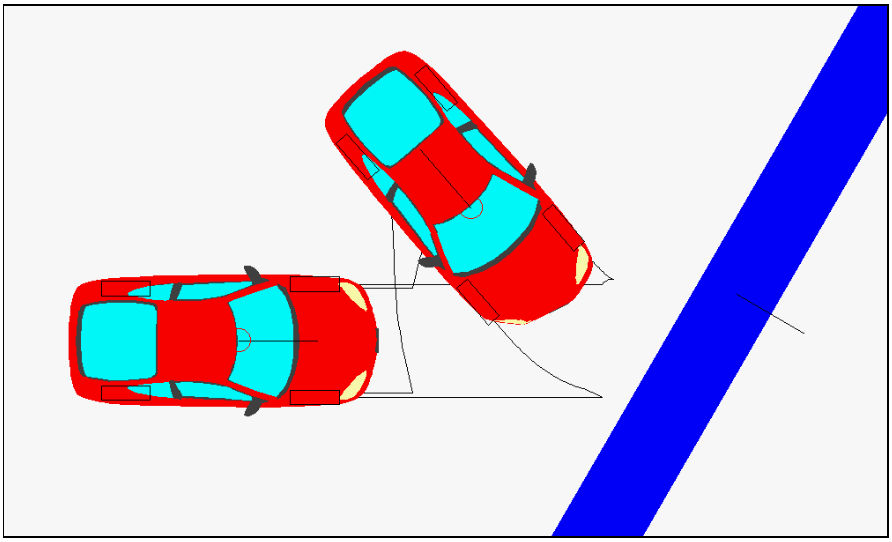


4. Now, use the simulation toolbar  to run the vehicles from the start to after the crash. With the Refresh key (F5) you will see the vehicle shape change from the initial (undamaged) shape to the second (damaged) shape. If 3D vehicle shapes were used, you will also see the shapes change in the 3D window and in rendered animations.

5. Repeat the above steps 2 to 4 to load subsequent shapes, if there is more than one crash which causes crush.

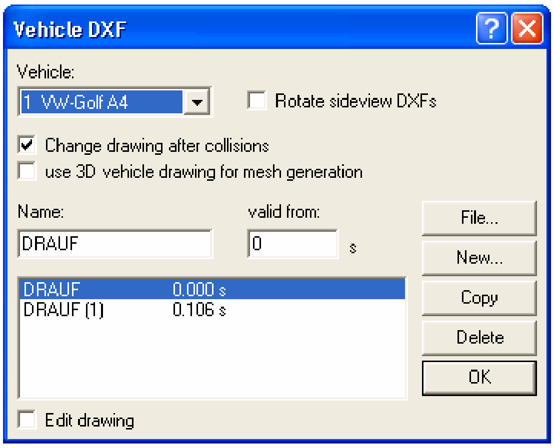
Using the Deform Drawing Feature

PC-Crash can automatically apply crush to each vehicle during each collision. After completing a simulation, simply select **Vehicle - Vehicle DXF - File – Deform Drawing**. Make sure  Change Drawing After Collisions is activated. Now, use the simulation toolbar  to run the vehicles from the start to after the crash. With the Refresh key (F5) you will see the vehicle shape change from the initial (undamaged) shape to the subsequent (damaged) shape(s).



If 3D vehicle shapes were used, you will also see the shapes change in the 3D window and in rendered animations.

Regardless of which method is used to provide for crush deformation, the Edit Drawing feature (described later in this section) can be used to make changes to the vehicle drawing. This is best used for 2D changes, however, as Edit Drawing is not a comprehensive 3D drawing tool.



Vehicle DXF includes the following:

Vehicle

This drop down list is for selecting which vehicle the drawing or bitmap file is to be applied to.

Rotate sideview DXFs

Activation means that the vehicle drawing, which was selected via <File…> <Side view> <Load DXF…> for the active vehicle is rotated 180° around the Z-axis before it is loaded.

Change drawing after collisions

Select whether the vehicle drawing will change when collisions occur. If only one drawing is applied to the vehicle, no changes will take place.

use 3D vehicle drawing for mesh generation

3D vehicle drawings will be used for the mesh generation which can be used for the Mesh contact model (activation: <Impact> <Use mesh based impact model>). If this option is not activated, the standard vehicle shapes or those surfaces defined under 'Vehicle Shape' are used for the Mesh model. For this case the vehicles are meshed with an edge length of approx. 0.02 m.

Name

This is the name of the DXF or IDF file which has been loaded. If more than one file has been loaded for the current vehicle, the name of the drawing selected in the list box below is shown here.

Valid from

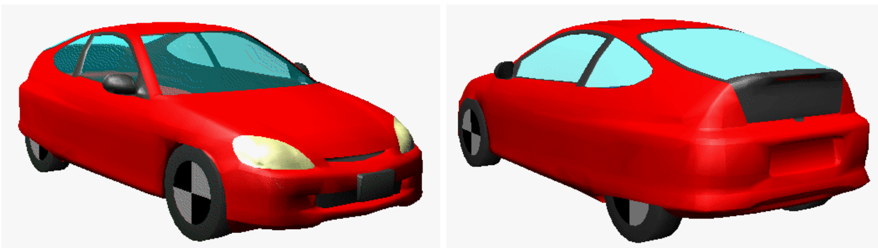
This indicates the time in the simulation when the highlighted drawing will be applied to the vehicle. For a crushed vehicle shape, the chosen time for an impact at t = 0 should be about -0.01 seconds, if the user wishes to have the crushed shape appear at t=0 (the start of the simulation).

File

The File button brings up a menu that includes the following:

Plan View

For loading 3D vehicle shapes and 2D plan view shapes (DXF, IDF, VRML and bitmap files). Many 3D vehicle IDF shapes are provided in the 3DDxf subdirectory in the PCCrash directory. Most of the vehicle shapes were made from digitizing full-scale vehicles, and are thus very accurate.

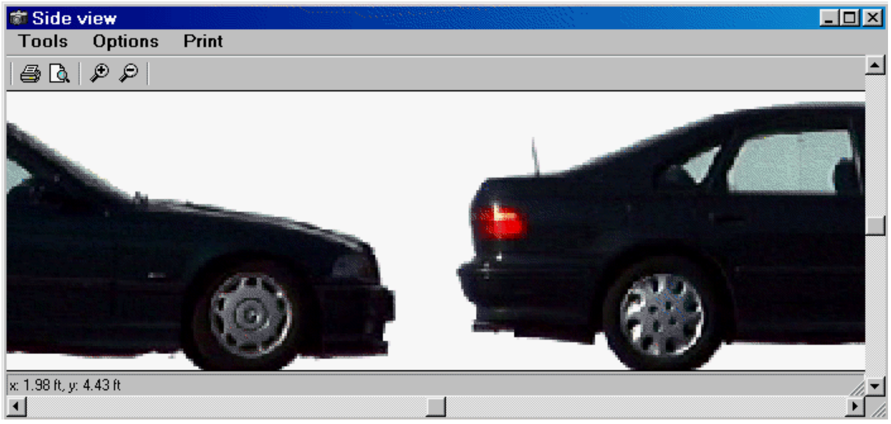


The following sub-menu items are included in the Plan View menu:

- Load DXF
- Load BMP
- Delete BMP

Side View

For loading 2D side view vehicle drawings and bitmaps. When loaded, these can be viewed by selecting **Options - Side View Window**. This feature allows bumper heights in rear-ender impacts to be compared easily. These mainly European vehicle bitmaps are scaled according to the length of the vehicle they are being attached to. They are included in the Side View subdirectory.



The following sub-menu items are included in the Side View menu:

- Load DXF
- Delete DXF
- Load BMP
- Delete BMP

When loading a 2D DXF for the side view, it may be that the orientation of the drawing is mirrored to the vehicle orientation. This is due to an inappropriate coordinate system of the DXF drawing that is used. To correct this issue, select the checkbox **Rotate side view DXFs** before loading a side view DXF file. This will automatically invert the DXF drawing along its x-axis.

3D Mapped

For loading mapped vehicles. These vehicles, which have the file extension *.x61, are visible only in the 3D window. They are comprised of a vehicle shape with bitmaps of components, such as grilles and lights, mapped onto the surface. These vehicle shapes are not as accurate as the IDF shapes provided, but give a realistic appearance because of the bitmapped features. These mainly European vehicles are included in the 3DDxf subdirectory.

PC-Crash additionally can import car models provided in the *.fce file format and the *.dff format. As the *.x61 format the *.fce files and *.dff files are visible only in the 3D window and are comprised of the shape and texture bitmaps. Many *.dff and *.fce files provided with PC-Crash have very high resolution and are very detailed.

If *.dff files are used, additionally different components can be selected/deselected using the **Vehicle – Vehicle DXF…** and the **File – 3D Mapped – Features** option in the Vehicle DXF window.


The following sub-menu items are included in the 3D Mapped menu:

- Load 3D Vehicle
- Delete 3D Vehicle

New

The New button applies a “new” vehicle shape to the selected vehicle. The “new” vehicle shape is a simple rectangle sized to fit the overall vehicle length and width. This is a good starting point if the vehicle 2D shape is to be drawn in PC-Crash (using Edit Drawing).

The New vehicle button should also be selected before loading a second vehicle shape for a vehicle; otherwise the original vehicle shape will be overwritten.

Copy

Copies the selected vehicle shape to the same vehicle, so that changes can be made (using Edit Drawing) to show damage that occurs during the impact.

Delete

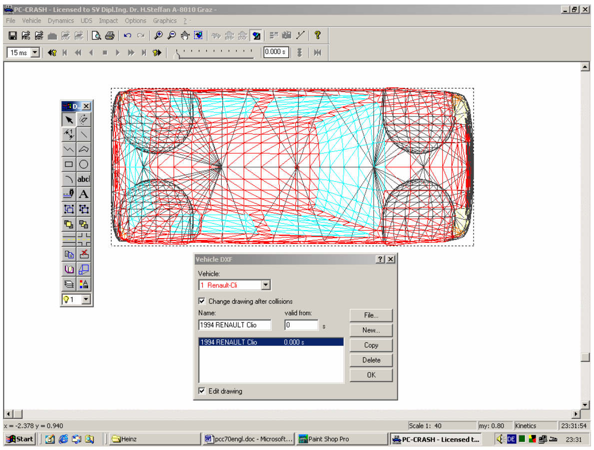
Deletes the selected vehicle shape.

OK

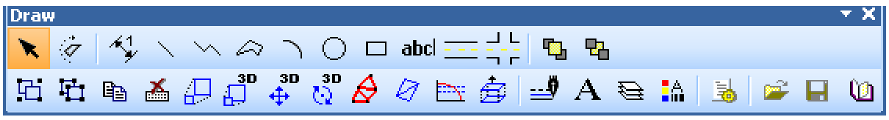
Accepts the changes made and closes the dialog box.

Edit drawing

This allows the modification of the selected vehicle drawing. The vehicle drawing appears on the main screen. Move the Vehicle shapes dialog box aside (do not close it) to access the whole drawing. Use the tools in the Draw group box to make changes to the drawing. For a complete description of these tools, refer to the menu item **Drawing – Draw Toolbar**.



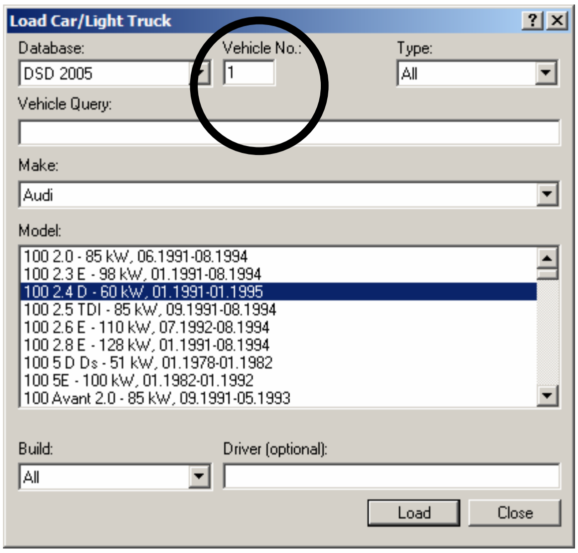
All the tools in the Draw Toolbar can be used to adapt the vehicle shape.



Erase Last Vehicle

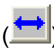
This deletes the last vehicle loaded (the one with the highest vehicle number).

Vehicle No. 1 cannot be deleted. To delete this vehicle the program must be initialized using **File** - **New**. Alternatively, Vehicle No. 1 can be replaced with another by changing the Vehicle No. to 1 when loading a new vehicle using **Vehicle – Vehicle Database** or **File – Import – Load Custom Vehicle** .



Vehicle administration

Menu option for the organization of the vehicles; vehicles can be exchanged, deleted and copied.

Exchange vehicles: over the selection boxes Vehicle 1 and Vehicle 2 those vehicles are selected, which should be exchanged. By activating the vehicles exchange button ()the vehicle are exchanged and thus the sequence of the vehicles is changed.

 Copy vehicle: with the vehicle copy button the vehicle selected under Vehicle 1 can be copied; the copied vehicle is arranged as last vehicle.

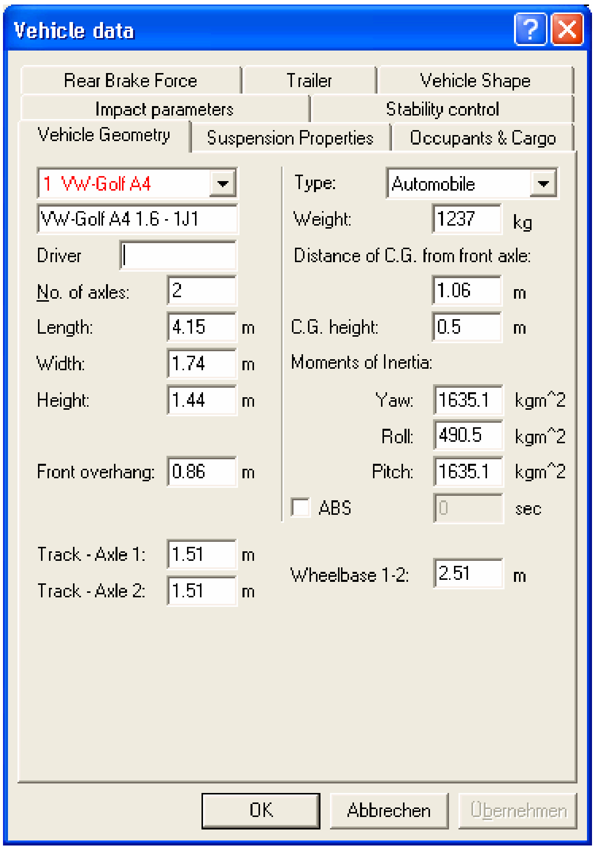
 Delete vehicle: with the vehicle delete button the vehicle selected under Vehicle 1 can be deleted and the vehicle listing is re-calculated.

Vehicle Settings

Selection of this menu item opens the Vehicle Data dialog box group, which includes the following tabs:

- Vehicle Geometry
- Suspension Properties
- Occupants & Cargo
- Rear Brake Force
- Trailer
- Vehicle Shape.
- Impact parameters
- Stability control

Vehicle Geometry



This dialog box is for viewing and changing vehicle geometry, mass or weight, moment(s) of inertia, and vehicle type. ABS braking can also be defined here.

Vehicle Geometry includes the following:

Vehicle

This drop down list identifies the name and number of the vehicle which the data is for. All the specific vehicle geometry values shown are related to this particular vehicle.

Vehicle Name

This text box shows the vehicle name corresponding to the vehicle number selected. The vehicle name can be edited here.

No. of axles

The number of axles a vehicle has is specified here. Valid values are 1 (trailers only), 2, 3, 4 or 5. In the normal case of a vehicle with two axles the text boxes for the third and higher axles' wheelbase and track width will not be shown.

Length, Width, Height

These text boxes show the overall dimensions of the vehicle.

Wheelbase

This is the distance between the first and second axle and, if a third axle exists, the distance between the second and third axle.

Front overhang

This distance is from the front of the vehicle to the front axle. It is normally measured from the center front, where the vehicle is longest.

Track

The track is the distance between the center of the left wheel and the center of the right wheel, for each axle.

Type

This drop down box is for selecting vehicle type. Possible selections are:

- Automobile
- Truck
- Wall
- Tree
- Motorcycle
- Unsteered trailer
- Steered trailer
- Semi-trailer
- Occupant
- Tramway

The default vehicle shape is changed in the 3D window to suit the specified vehicle type.

Weight

The empty mass (metric system - kg) or weight (U.S. system - lb) of the vehicle is shown here.

If the user desires, the loaded mass or weight of the vehicle can be specified in this text box. In this case, ensure all occupant and cargo loads are set to zero in the Vehicle - Occupants & Cargo dialog box.

Distance of C.G. from front axle

This is the distance of the center of gravity behind the front axle. Negative numbers can be used here for trailers, when applicable.

C.G. height

This is the vertical distance of the center of gravity from the roadway, under static conditions.

If the height of the center of gravity is 0, the simulation is performed in the 2D mode, and the pitch and roll moments of inertia are not shown.

Moments of Inertia

These are the moments of inertia of the vehicle about its three main axes:

- **Yaw**, about the vertical axis
- **Roll**, about the longitudinal axis
- **Pitch**, about the transverse axis.

The program automatically calculates the moment of inertia using the following formulas (these values can be overwritten by the user at any time):

- For most vehicles, the Burg formula is used:

  $Iz = 0.1269\ (m)\ (WB)\ (L)$,
  $Iy = Iz$,
  $Ix = 0.3\ (Iz)$, where

  Iz = Yaw moment of inertia
  m = Mass
  WB = Wheelbase
  L = Overall length
  Iy = Pitch moment of inertia
  Ix = Roll moment of inertia

- For a truck or a trailer the moment of inertia is calculated by the following formula:

  $Iz = m\ (L^2 + B^2)/12$,
  $Iy = Iz$,
  $Ix = 2\ (m)\ (B^2)/12$, where

  L = Overall length
  B = Overall width

Note: *All the moments of inertia in this dialog box represent the unloaded vehicle. Use the* ***Vehicle data*** - ***Occupants & Cargo*** *menu option to add additional loading for occupants and cargo. PC-Crash automatically modifies the moment(s) of inertia as follows:*

$$I_{loaded} = I_{empty}\ (m_{loaded}/m_{empty})$$

☑ ABS

This activates the ABS brake model. When ABS is activated, a text box appears in which the ABS cycle time in seconds can be entered. The ABS model functions differently for the Linear and TM-Easy tire models, as described in the **Technical Manual**.

OK

Accepts the data entered and closes the dialog box.

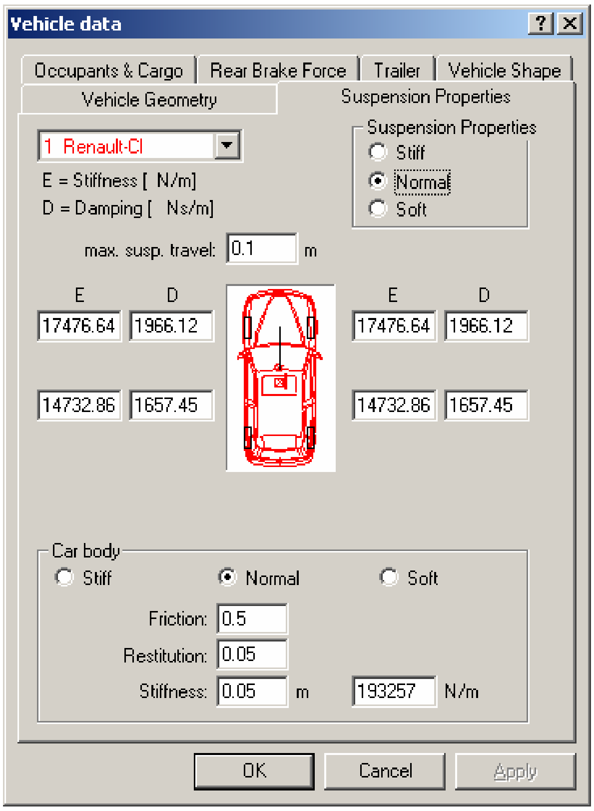
Cancel

Closes the dialog box without accepting the data.

Note: *When the geometry of a vehicle is changed and a 3D simulation is being performed, the suspension properties should be redefined so that they are based on the new wheel loads. This is also true after a* 5^{th} *wheel trailer is connected to a vehicle.*

Suspension Properties

Suspension stiffness and damping parameters for each wheel can be specified here. Additionally, vehicle body parameters can be specified for the rollover model and the stiffness based impact model.



Suspension Properties includes:

Vehicle

This drop down box is for selecting which vehicle the suspension parameters are being modified for.

Max. Susp. Travel

The maximum suspension compression travel (measured at the wheel) is defined here. The spring rate (E) doubles beyond this value. For extension, the wheel travel stops when the spring force is zero, which depends on the selected spring rate (a softer spring rate will allow more downward wheel travel).

Stiffness (E)

This text box shows the spring stiffness coefficient for each wheel.

Damping (D)

This text box shows the damping coefficient for each wheel.

Suspension Properties

Default values for the stiffness and damping coefficients can be chosen by activating the Stiff, Normal or Soft suspension option buttons. These correspond to the following:

- Stiff: 10cm (3.9") static spring compression
- Normal: 15cm (5.9") static spring compression
- Soft: 20cm (7.8") static spring compression

Note: *Whenever you load a new vehicle, the correct values for the suspension should be checked. This can be done by entering the exact values desired or by using the buttons Stiff, Normal or Soft.*

When performing a 2D simulation (height of C.G. equals zero) the suspension characteristics are not important. However, for numerical reasons they must not be unreasonable numbers.

Suspension values are checked automatically. In the case of unrealistic parameters, a warning appears in the status bar at the bottom of the screen:

Unusual suspension characteristics for vehicle ...

Check and correct the parameters if this message appears.

Car Body

Vehicle body to ground impacts are calculated in the case of a vehicle rollover, if **Rollover Detection** is turned on in the **Dynamics** menu. Refer to the **Technical Manual** for more details on the Rollover Model.

A stiffness based impact model is used if **Impact – Use stiffness based impact model** is activated. The crash is calculated using the values specified within the Car body section.

Properties for stiffness based impacts can be specified in the Car Body field, as follows:

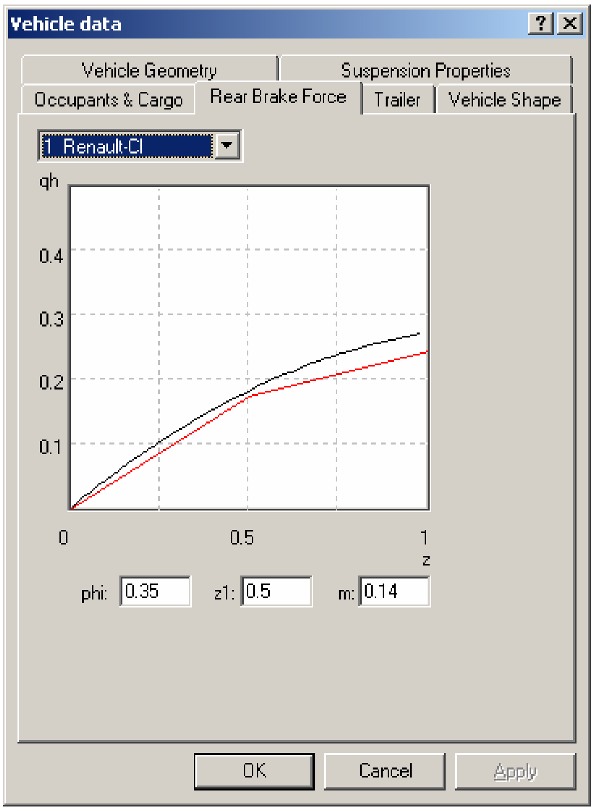
- **Friction** – This is the coefficient of friction for the vehicle body. The tire to ground friction is not specified here – it is the lesser of 0.8 or as specified elsewhere: for the general scene, in Friction Polygons, or in the Sequence window.
- **Restitution** – This is the coefficient of restitution for vehicle impacts. If the stiffness based impact model is used only one restitution for all concerned vehicles should be used. If different values are specified the lowest value is used for the calculation.
- **Stiffness** – The "stiffness" is expressed as a deformation distance. The stiffness is defined by the deformation of the car body due to its static weight - the user specifies the deformation and the stiffness that is calculated. The value of stiffness used for the tire ellipsoids is half of the value specified for the vehicle body. For automobiles, the specified stiffness is for the lower part of the vehicle body only. For the roof, one quarter of the specified stiffness is used. For other vehicles, the specified stiffness is for all body ellipsoids. In addition the value for the stiffness can be specified (linear relationship between load and deformation), the corresponding deformation distance is calculated automatically.

Vehicle speed changes and accelerations that occur as a result of stiffness based impacts can be viewed in graphical form in the Diagrams window, using the menu option **Options - Diagrams – Diagrams – Vehicles** or **Options - Diagrams – Diagrams – Sensor Signals**. Refer to the description of this menu option later in this chapter.

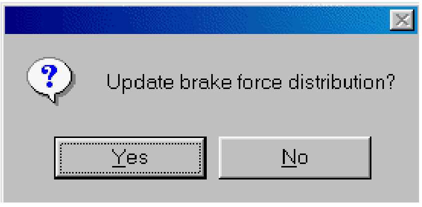
Rear Brake Force

The brake force distribution between the front and rear wheels can be viewed and changed here.

A graph shows the proportion of rear brake force (qh) to the front brake force (z). The theoretically ideal curve as well as the assumed or defined distribution is shown. Immediately after entering a C.G. height in the Vehicle Geometry dialog box, a suggested distribution curve is automatically calculated. The user can modify the curve if desired. Based on the defined distribution the correct brake forces will be calculated when braking is applied in **Dynamics - Sequences**.



Many modern vehicles are equipped with automatic re-adjustment of brake distribution as a function of vehicle cargo. This is taken into account if the user answers yes when the following message appears after adding occupant or cargo mass (see next section).



Rear Brake Force includes the following:

Vehicle Number

This drop down box is for selecting which vehicle the brake force distribution is for.

phi, z', m

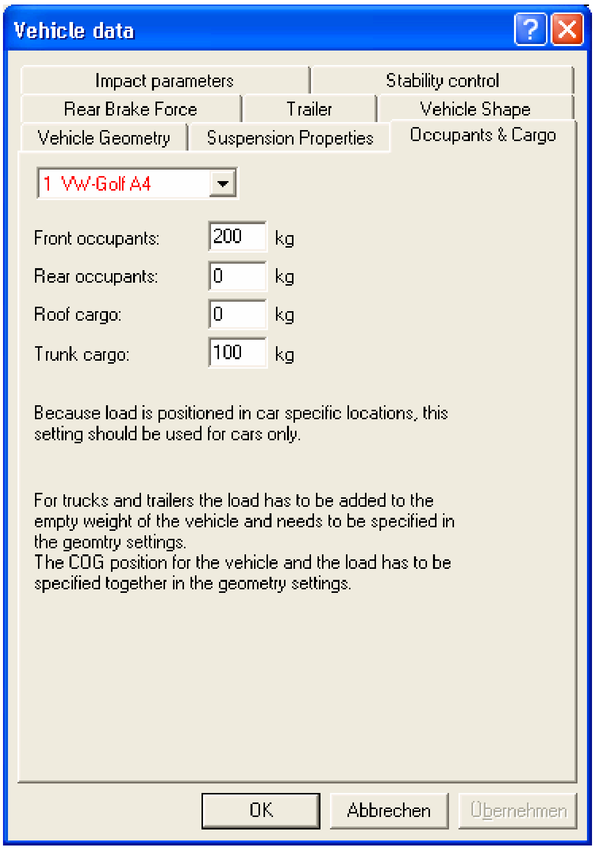
These parameters identify the defined brake force distribution, as follows:

- **Phi** Slope of the first line
- **z1** Point where slope changes
- **m** Slope of the second line.

The values can either be entered numerically in the text boxes, or the distribution may be changed interactively in the diagram by shifting points on the curve with the mouse.

Occupants & Cargo

Vehicle passenger and cargo loads are specified here.



Occupants & Cargo includes the following:

Vehicle

This drop down box is for selecting which vehicle the occupant and cargo loads are being specified for.

Front occupants

The weight or mass of occupants in the front seat is applied in the middle of the vehicle at the same height as the center of gravity and 15% of the wheelbase (front to rearmost axle) in front of the vehicle mid-point.

Rear occupants

The weight or mass of occupants in the remaining seats is applied in the middle of the vehicle at the same height as the center of gravity and 20% of the wheelbase behind the vehicle mid-point.

Roof cargo

The weight or mass of roof cargo is centered directly above the center of gravity, 0.3m (1 foot) above the overall vehicle height. However, for 2D simulations, the roof load height is set to zero, along with all other heights.

Trunk cargo

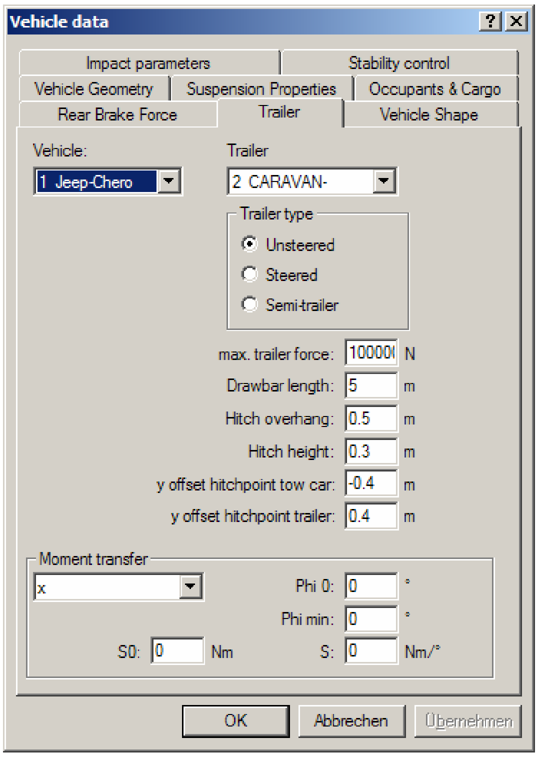
The weight or mass of trunk cargo is applied in the middle of the vehicle at the height of the center of gravity and 10% of the wheelbase behind the rear axle.

Note: *The program automatically recalculates values for the vehicle's center of gravity, mass or weight, and all the moments of inertia as a result of any changes in the vehicle load due to occupants or cargo. The moments of inertia are corrected for cargo with the formula:*

$$I_{loaded} = I_{empty}\ (m_{loaded}/m_{empty}).$$

Trailer

Trailer parameters for steered, unsteered and semi-trailers can be defined here. In addition to calculating tow vehicle and trailer dynamics, the trailer model is useful for determining off-tracking of truck and trailer combinations. Refer to the Technical Manual for details on the theory behind the trailer model.



Trailer includes the following:

Vehicle

This drop down box is for selecting which vehicle is to be connected to the trailer.

A vehicle is coupled to a trailer by selecting the desired vehicle in the drop down box. Then the correct trailer must be selected in the Trailer drop down box. As soon as a trailer has been selected, both are coupled. Note that the trailer must have been previously loaded into PC-Crash, using **File - Import** - **Custom Vehicle** or **Vehicle – Vehicle Database**.

If the vehicle has already been connected to a trailer, the corresponding trailer will be shown in the Trailer drop down box. If no trailer is connected, the Trailer drop down box contains the word "None".

Trailer

This drop down box is for selecting which trailer is to be connected to the tow vehicle. If "None" is chosen, any trailer that was previously connected to a tow vehicle is disconnected.

Trailer Type

These option buttons are for specifying trailer type, as follows:

- Unsteered
- Steered (at least 2 axles required)
- Semi-trailer.

max. trailer force

This value specifies the maximum force the hitch withstands. If the force at the hitch is higher than the specified amount, the trailer decouples from the towing car.

The maximum trailer force can be used to simulate uncoupling of truck trailer combinations due to external force (i.e. crashes).

Drawbar Length

The drawbar length of the trailer is the distance between the hitch and the front trailer axle.

Hitch Overhang

This is the overhang of the trailer hitch outside the rear bumper of the tow vehicle. Negative overhangs are used for 5th wheel hitches.

Hitch Height

The height of the hitch point above the ground can only be defined for 3D simulations (if the height of the center of gravity for both the tow vehicle and the trailer are > 0).

Y offset hitch point tow car

If the hitch point is not at the center of the tow car, an eccentric hitch point can be specified by entering an offset value. The offset specifies the distance from the vehicles longitudinal axis.

Y offset hitch point trailer

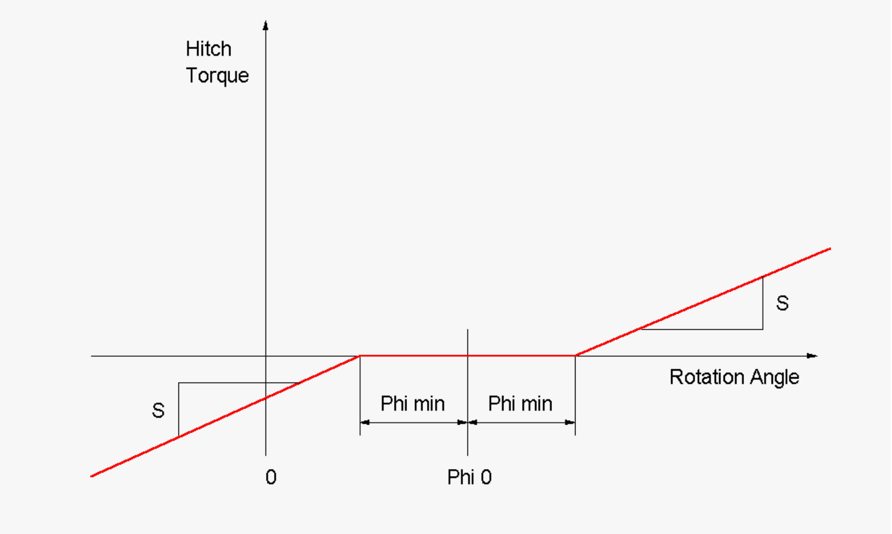
If the hitch point is not at the center of the trailer, an eccentric hitch point can be specified by entering an offset value. The offset specifies the distance from the trailers longitudinal axis.

Moment Transfer

This area of the Trailer dialog box is used to define moment transfer values about the x, y and z axes of the hitch, as follows:

- **S0** - Constant frictional torque about the specified axis.
- **Phi 0** - This is an offset value from 0°, useful mainly for pitching about the y axis of the hitch. If the rotation between tow vehicle and trailer is more than Phi 0 + Phi min or less than Phi 0 – Phi min, the linearly increasing torque S is applied. If Phi 0 = 0°, the torque S is symmetrical for positive and negative rotations.
- **Phi min** - Angular rotation of the trailer about the hitch with respect to Phi 0 at which the rotational torque S starts.

- **S** - Linearly increasing resistive torque about the specified axis, starting at Phi 0 + Phi min.



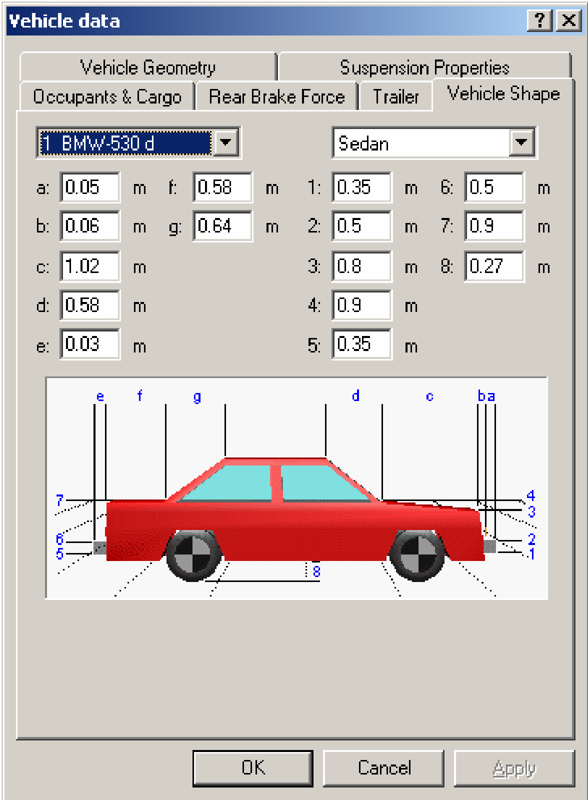
Note: *For trailers, rearward simulations are not possible in PC-Crash. Collisions can be performed between a trailer and a vehicle, between a tow vehicle and another vehicle, between a trailer and its tow vehicle or between two trailers.*

The start conditions for the trailer are determined automatically by the program, based on those of the tow vehicle.

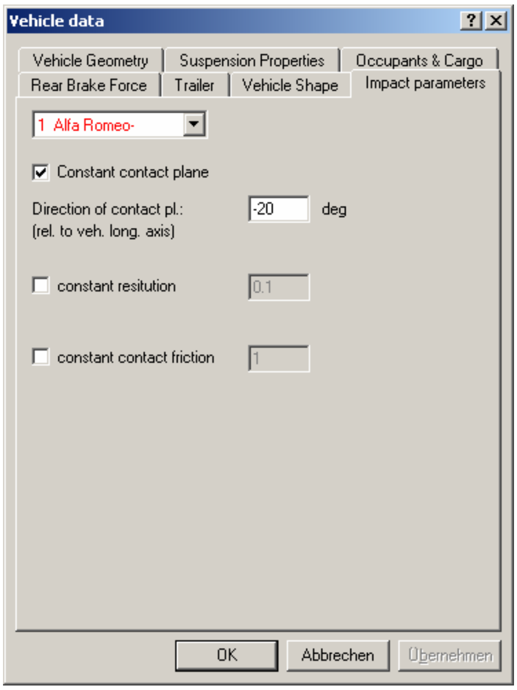
Due to the calculation method being based on a set numerical accuracy, small location differences between tow vehicle and trailer at the hitch point can develop in long simulations. These differences do not influence the driving stability. If the distance exceeds 10 cm (4 inches), all the definitions should be checked once more and the simulation repeated.

Vehicle Shape

This dialog box is for the definition of the vehicle body shape that will appear in the 3D window when a vehicle DXF/IDF shape is not used. Default settings to be selected are Sedan, Hatchback or Van. Additionally, users can input custom dimensions. The vehicle shape specified here is used to determine the vehicle ellipsoid sizes for the stiffness based impact/rollover model. The shape is also used for collisions with multibody objects, unless a DXF/IDF vehicle shape is specified for these contacts.



Impact parameters



In this window the impact parameters can be made constant for a vehicle. The means, instead of determining the impact parameters by calculations the parameters specified on this page are used.

Constant contact plane

The direction of the contact plane can be set to a constant value relative to the vehicles longitudinal axis. If this option is turned off, the contact plane will be determined by the impulse vectors of the colliding vehicles.

Constant restitution

For collisions with this vehicle the factor of restitution can be set to a constant value.

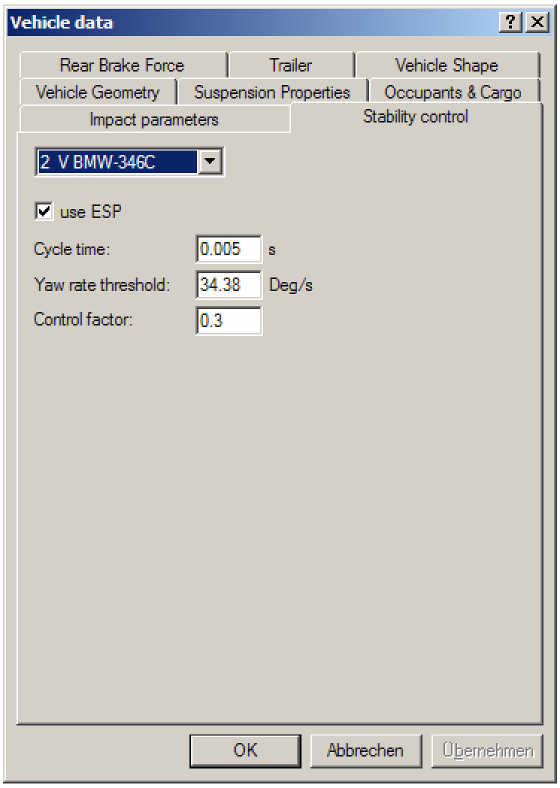
Constant contact friction

For collisions with this vehicle the friction can be set to a constant value.

Stability Control

Many modern vehicles have built-in devices for increasing driving stability for avoiding breakaways of the vehicle. The stability is gained by applying break forces to single wheels, thus decreasing improper yaw angular velocity.

PC-Crash has a device manufacturer independent model for simulating such electronic devices. On the page **Stability Control** the parameters for this model can be accessed.



use ESP

This option determines if the stability control simulation should be activated for the selected vehicle.

Cycle time

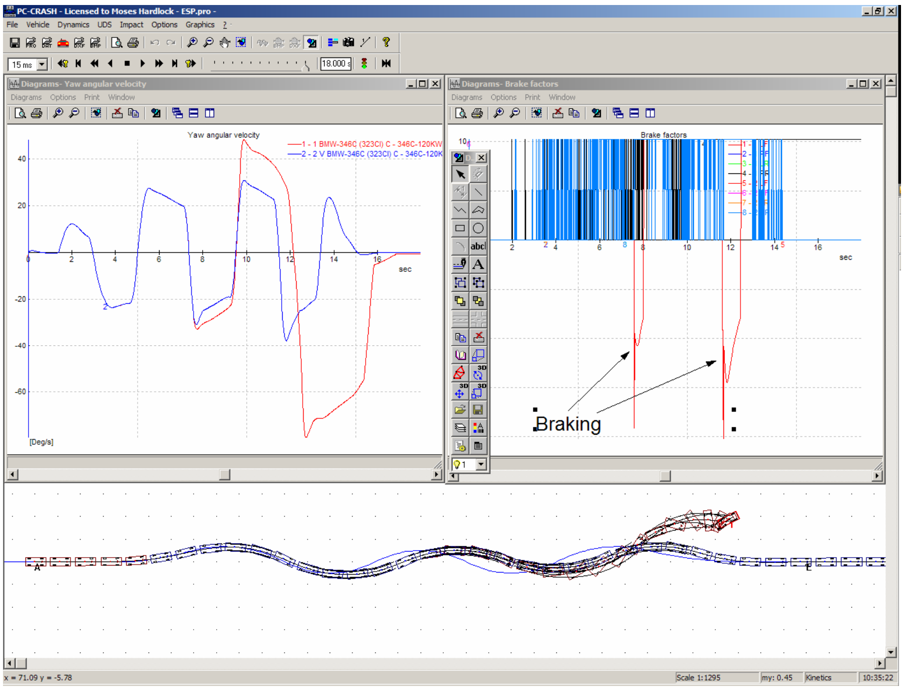
The cycle time determines the update frequency of the stability control device. After the specified time the simulation will determine if the vehicle tends to become instable and will react properly.

Yaw rate threshold

The optimal yaw angular velocity is determined by the vehicle's velocity and the steering angle. If the actual yaw angular velocity is off by the given threshold the stability control device will react properly by braking a wheel.

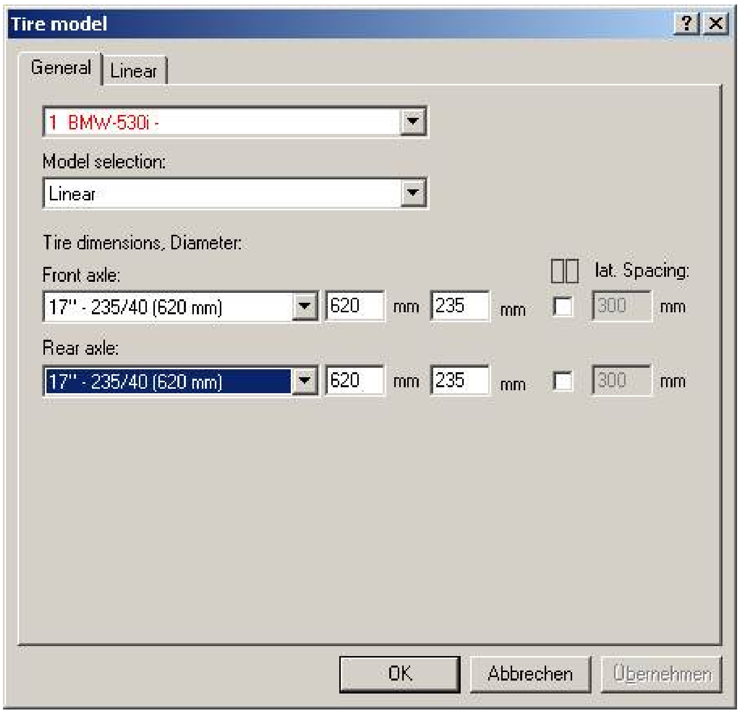
Control factor

The control factor determines how fast and intense the reaction of the stability control device will be. The brake force is calculated by integrating the angular velocity that is above the threshold, multiplied by the control factor. A high control factor will almost immediately fully brake a wheel even at the smallest possibility of getting unstable. A low control factor will increase the brake factor slower and less intensive having the risk of a too late intervention and not being able to stabilize the vehicle. The default values provided are appropriate for most situations.



Tire Model

The menu item **Vehicle - Tire model** is for viewing and defining tire parameters. The user can select tire models (Linear or TM-Easy), change tire model parameters, change tire size, and select duel tires when desired.



Tire Model (General) includes the following:

Vehicle

This drop down list identifies the vehicle for which the tire characteristics are being defined.

Model selection

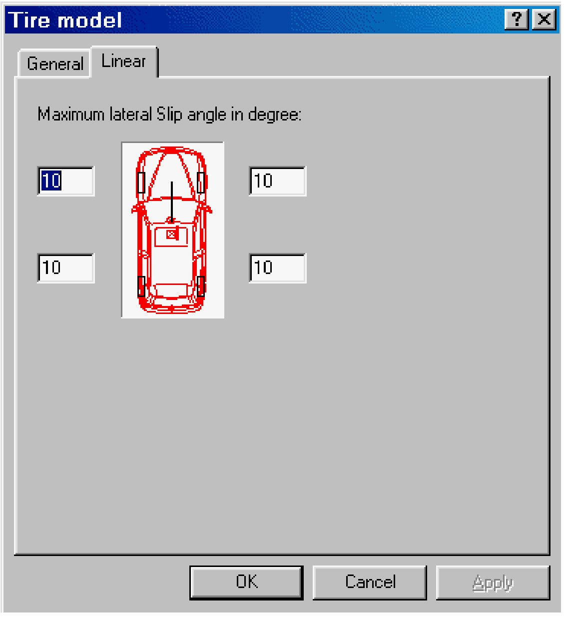
In this drop down list the tire model for each vehicle can be selected. There are two models currently available:

- Linear
- TM-Easy

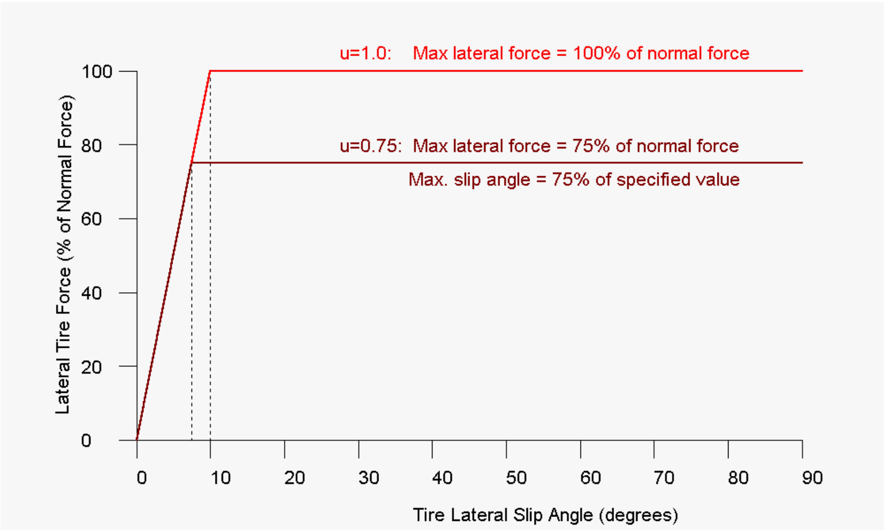
Linear Tire Model

The linear model dialog box displays the maximum lateral slip angle for each tire.

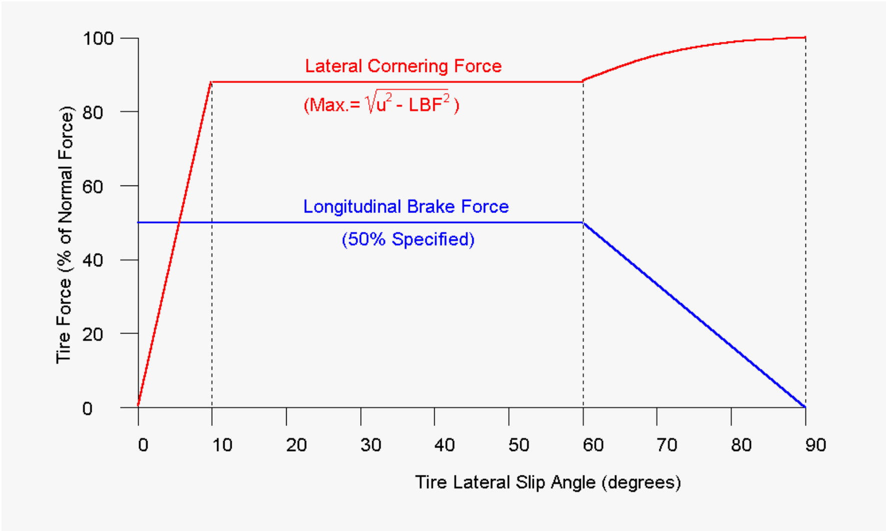
The maximum lateral slip angle defines the stiffness of the tire on each wheel. The model simulates the tire forces by assuming a linear increase of lateral tire force with increasing lateral slip angle up to the specified value. The lateral tire force increases no further once the lateral slip angle exceeds this value (typically about 10° for a 70 series tire on a high friction surface – lower profile tires will have a lower maximum lateral slip angle).



The specified maximum lateral slip angle (default 10°) is for a unity coefficient of friction *u* = 1.0. If a different coefficient of friction is specified, the tire slip angle at which lateral tire force saturates will vary accordingly. Different surface friction coefficients have no effect on the stiffness of a tire, and thus a lower coefficient of friction, which cannot produce as much side force on a tire, will result in a lower maximum slip angle. For example, with a specified maximum tire slip angle of 10° on a surface with a coefficient of friction of 0.75, the maximum tire slip angle possible is 7.5°. This is the angle at which the maximum lateral tire force will be reached, which will be 75% of normal force in this case.

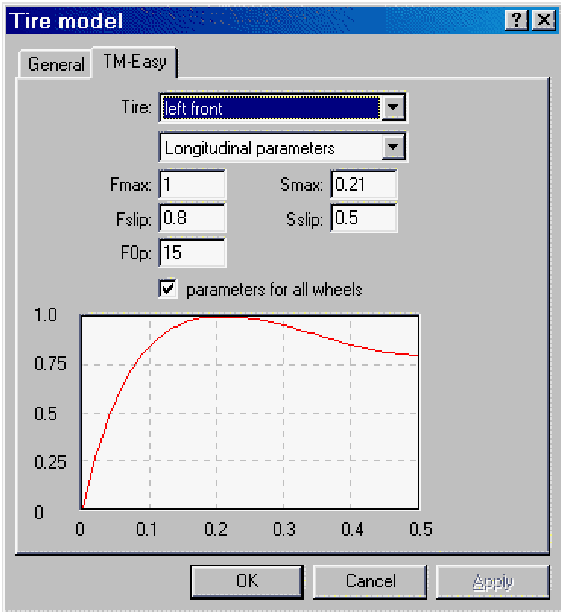


If braking is applied, the maximum lateral tire cornering force will decrease according to the friction circle rule. For example, with a specified brake level of 50% (of the static normal tire load), the maximum lateral force will be as shown in the following diagram (*u* = 1.0).



TM-Easy Tire Model

The dialog box for the TM-Easy tire model, which allows non-linear tire effects to be modeled, can be accessed when this option is chosen in the Model Selection drop down list.



Tire Model (TM-Easy) includes the following:

Tire

The top drop down list is for specifying which tire the parameters refer to. If the ☑ Parameters for all Wheels is activated, the parameters will be the same for all wheels.

The second drop down list allows the longitudinal and lateral tire parameters to be specified independently.

Fmax

The peak frictional force value. The maximum friction is this value (default = 1) multiplied by the specified scene friction coefficient.

Smax

The slip value at which Fmax occurs. For lateral tire properties, the x-axis of the tire model graph is tan (α), where α is the lateral slip angle. For longitudinal properties, the x-axis is a fractional value, with 0.5 meaning 50% slip.

Fslip

The sliding frictional force value. The maximum friction is this value (default = 0.8) multiplied by the specified friction coefficient.

***Note:** When switching from the Linear tire model to the TM-Easy tire model, the change in vehicle motion will be due almost entirely to the difference between the specified Fslip value and 1. This is because the default coefficient of friction is the <u>sliding</u> value when using the Linear tire model, but is the <u>static</u> value when using the TM-Easy tire model and Fmax = 1.*

Sslip

The slip value at which F slip occurs.

FOp

The slope of the tire model curve at the origin.

Tire dimensions, Diameter, Width

Most tire sizes can be selected from the Front axle and Rear axle drop down lists to obtain the correct tire diameter. For other tire sizes, the diameter must be entered by the user (the information in the drop down lists is ignored). The tire diameter is used in the Engine & Drive train gearing calculations and also in the 3D Window.

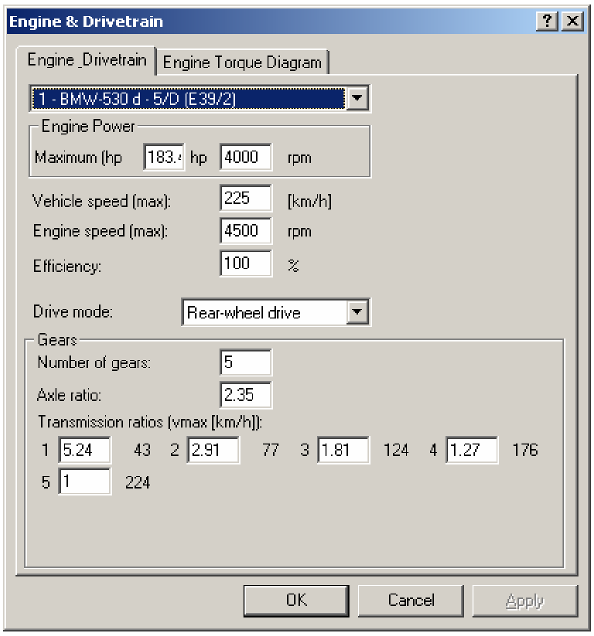
The tire width can be entered independently from the tire diameter. The width has no effect in the simulations but will provide a more realistic look in the 2D and 3D views.

The Lateral Spacing check boxes  are for specifying dual tires and the text boxes to the right are for specifying the dual spacing (center to center).

Engine/Drivetrain

This option allows the definition of engine and gear ratio parameters. The engine power curve characteristics, transmission and differential ratios can be entered in this dialog box. Realistic acceleration motions can then be applied to the vehicle by using the Real Acceleration option in that vehicle's acceleration sequence (see the **Programming Sequences** chapter). Ensure that the correct tire diameter has been entered in the Tire Model dialog box.

Some of the engine/gear parameters are included in the European vehicle databases, but not in the present Specs North American database.



Engine & Drivetrain includes the following:

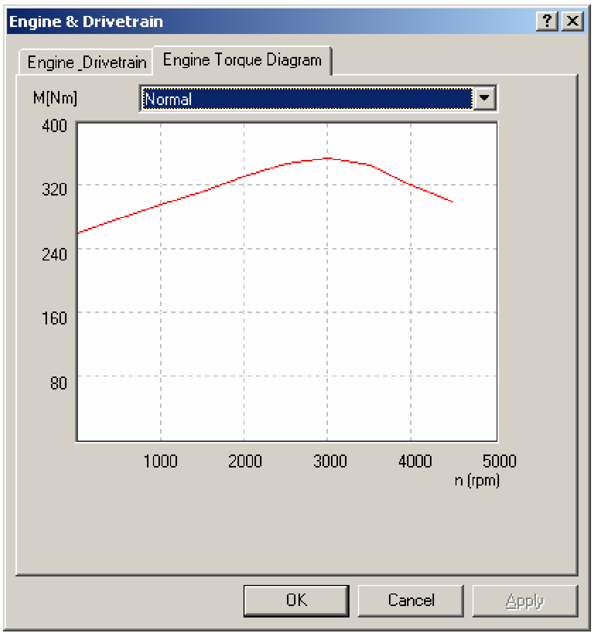
Vehicle

Drop down list for identifying which vehicle the values are to be applied to.

Engine Power

Maximum engine power (HP) and the corresponding engine speed (rpm) at which it is developed. Note that this is not the maximum engine speed.

After selecting the desired power and rpm, use the Engine Torque Diagram tab to view the torque curve. Different torque curves can be selected and - if desired - modified.



The current torque curve is displayed. To ensure that changes made to engine power or rpm have been accepted, select one of the standard torque curves from the drop down list above, as follows:

- Normal
- Sport
- Diesel
- Turbo Diesel.

After selection of one of these curves, it can be modified it by moving points on the curve with the mouse.

Vehicle speed (max)

Maximum vehicle speed. PC-Crash compares this specified value with the available horsepower to determine a value for the air resistance at this speed. This calculation of the air resistance assumes the gearing is such that the engine's speed will be at the maximum horsepower level at maximum vehicle speed.

The air resistance is applied at all other speeds based on the assumption it is proportional to the square of the vehicle's speed. The air resistance forces are added to the vehicle's wheel brake forces, evenly divided between all wheels.

Note: *If wind resistance has been applied to a vehicle by activating ☑ Calculate Air Resistance in the **Vehicle - Wind Resistance** menu item, that air resistance is applied to the vehicle <u>in place of</u> the air resistance determined from the specified top speed in the **Engine/Drivetrain** menu option.*

Engine speed (max)

Maximum revolutions per minute (rpm) of the engine.

Efficiency

The efficiency of the drivetrain is specified here. A value of 70 to 90% is normal.

Drive mode

Different types of drives can be selected in this drop down box, as follows:

- Front wheel drive
- Rear wheel drive
- Four wheel drive (50% front / 50% rear)
- Four wheel drive (30% front / 70% rear).

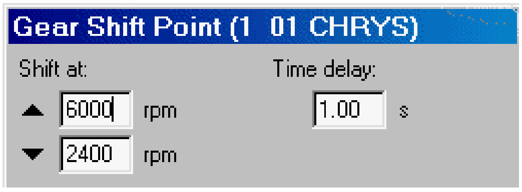
Gears

Text boxes for the number of gears (maximum = 16), their ratios, and the drive axle ratio. The maximum vehicle speed for each gear (based on the specified maximum engine RPM and ignoring air resistance) is shown to the right of the gear ratios.

Note: *After defining a new engine power or nominal engine speed the following information box appears to adjust the Moment Diagram when the Engine/Drivetrain dialog box is exited.*

Selecting the Yes option will adjust the engine torque diagram using the selected characteristics.

Note: *The parameters specified in Engine/Drivetrain are only used when Real Acceleration is specified in the vehicle's acceleration sequence, accessed using* ***Dynamics – Sequences*** *(F6). Once an acceleration sequence is specified for the vehicle and opened by double-clicking on it, Real Acceleration can be activated. A window appears where the user can specify the engine speed for shifting gears as well as the time necessary to perform the shift. Make sure the acceleration sequence pedal slider bar is moved to the desired throttle position, or the vehicle won't accelerate at all.*



Wind Resistance

Vehicle air resistance, down force and wind parameters can be specified in the Air resistance/Wind dialog boxes. The effect of air resistance and wind is calculated only when Calculate Air Resistance in the Wind dialog box is checked.

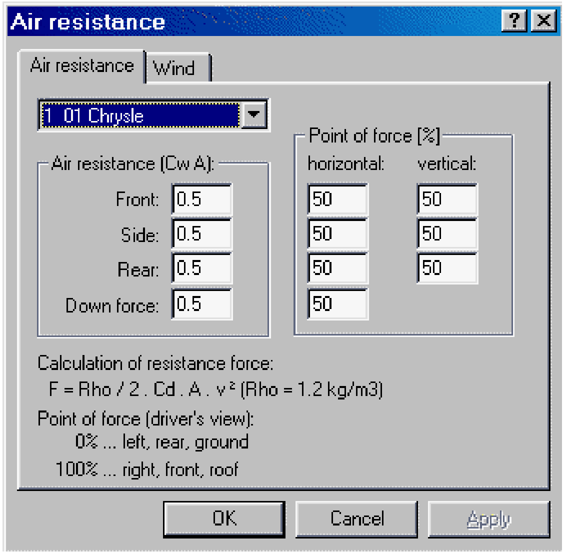
Air Resistance

PC-Crash enables the user to consider the influence of wind resistance depending on the vehicle speed and predefined external wind parameters.

The resistance force is calculated using the following formula separately for front, side and rear:

$$F = \frac{\rho}{2} \cdot c_w \cdot A \cdot v_{rel}^2$$

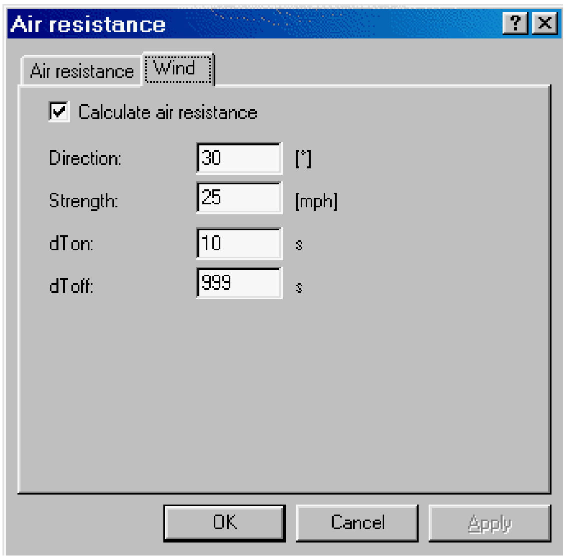
The user has to define the air resistance value $c_w \cdot A$ for front, side and rear. The density of air is assumed to be 1.2 [kg/m^3]. The relative velocity v_{rel} depends on the vehicle speed and on the wind speed (see the next item).



The point of force in the horizontal and vertical direction is also defined. This input is in percentage of the vehicle length, width and height, where 0 % is a point of force on the rear, ground or left side and 100% is on the front, top or right side. Vehicle down force or lift (using a negative down force coefficient) can also be modeled.

Wind

This dialog box enables the direction and strength of the wind to be defined.



The user can consider the influence of air resistance during a simulation by activating Calculate Air Resistance. By default, this is not activated.

Direction

The wind direction is specified, counterclockwise from the global x-axis. A wind direction of 0 degrees means the wind is blowing towards the east, if north is towards the top of the screen.

Strength

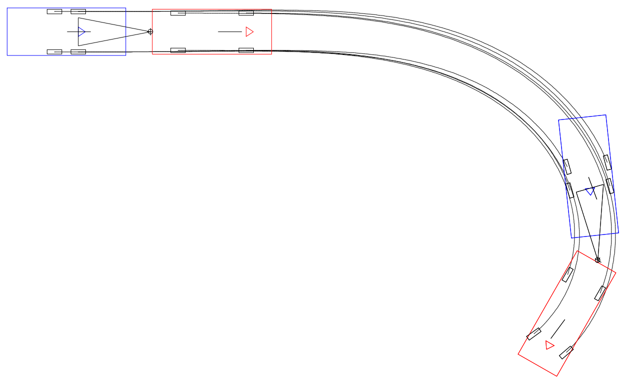
The wind speed is specified, in the units chosen in the **Options - Options** menu option.

dT On and dT Off

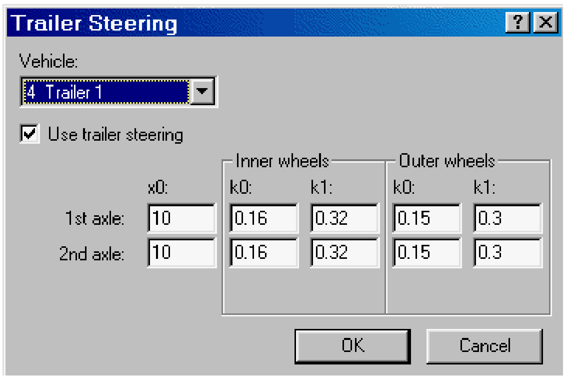
Wind gust conditions can be specified. For example, if a dT On time of 2 seconds is chosen with a dT Off time of 1 second, there will be 2 seconds of wind followed by 1 second of calm, with this combination repeated to the end of the simulation. If a steady wind is desired, choose a dT On time longer than the simulation.

***Note:** If Real Acceleration and Air Resistance are used at the same time, the air resistance which is calculated from the defined maximum vehicle speed is ignored and the air resistance based on values specified in the Air Resistance dialog box is used.*

Trailer Steering



Trailer steering, as a function of tow vehicle – trailer articulation angle, can be specified in this dialog box, which is accessible only when a trailer is coupled to a tow vehicle.



The steering for each trailer wheel can be specified in two steps, as follows:

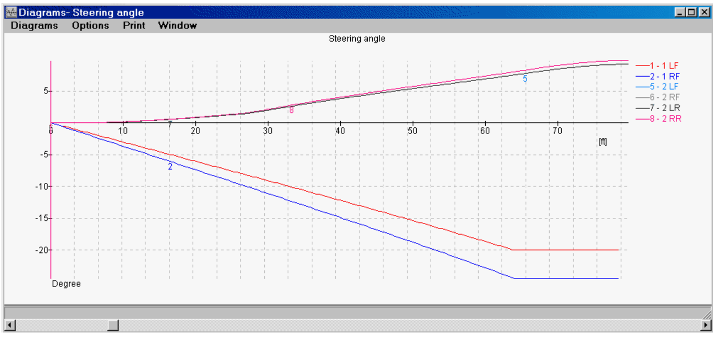
For $\varphi_S \le x0$, $\varphi_S = k0\phi_A$, and

for $\varphi_S > x0$, $\varphi_S = k0x0 + k1(\phi_A - x0)$

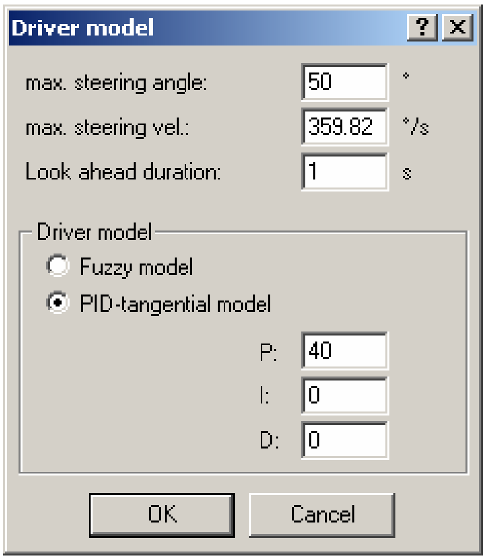
where φ_S = trailer wheel steer angle

φ_A = tow vehicle – trailer articulation angle

$x0$ = angle (degrees) at step change
$k0, k1$ = multipliers.



Driver Model



When a Path is defined for a vehicle, various parameters can be set in this dialog box, as follows:

Max. Steering Angle

This is a limit on how far the front wheels will steer. The selected value, in degrees from straight-ahead, is for the outside front wheel. The inside front wheel angle is slightly higher, based on Ackermann effects.

Max. Steering Vel.

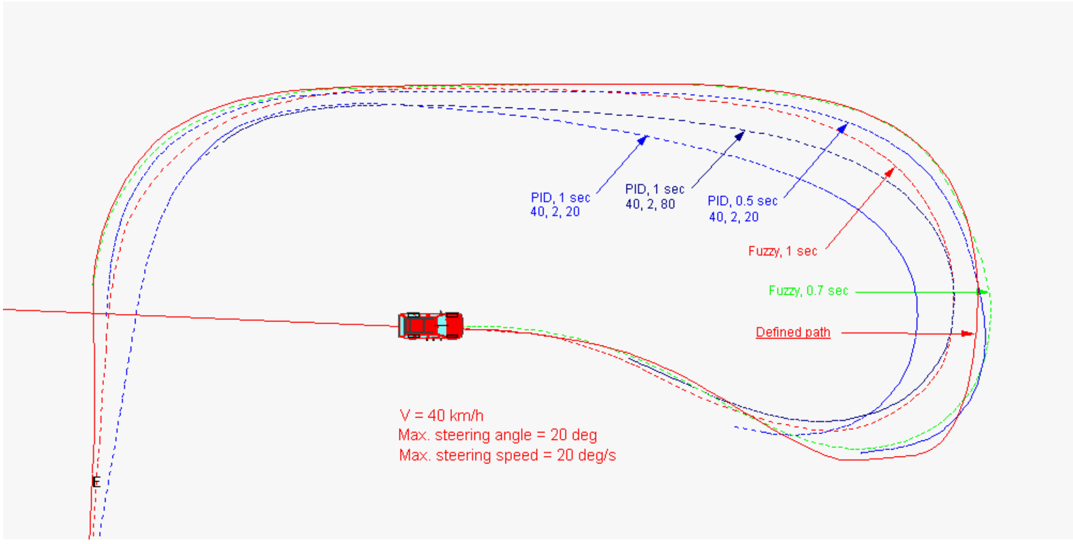
This is a limit on how fast the front wheels can be steered. The selected value, an angular velocity in degr/second, is for the outside front wheel. The inside front wheel steering velocity is slightly higher, based on Ackermann effects.

Look Ahead Duration

This time, combined with the current vehicle speed and heading, is used to calculate a look-ahead distance vector. The Path model compares the location of the end of this vector with the defined path to determine the current steering angle.

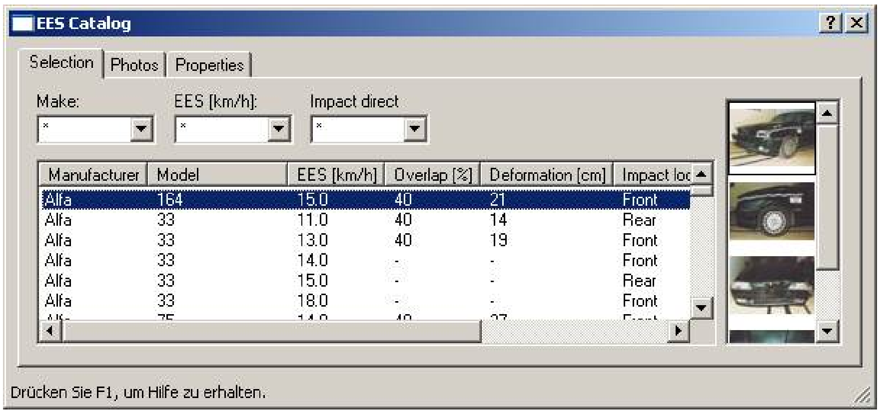
Driver Model

The Fuzzy or PID-Tangential model can be selected here. Refer to the **Technical Manual** for more details. The following figure shows the effects of different models and settings for them. In this case, the Fuzzy Model with a look-ahead duration of 0.7 seconds enables the defined path to be followed quite closely, except near the lower right, where the defined path radius is too sharp for the speed.



EES Catalog

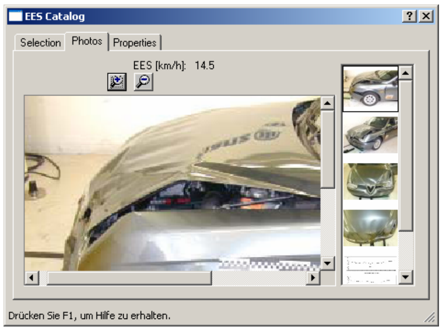
The EES catalog contains photos of damaged vehicles categorized into vehicle model and collision severity groups. This enables the user to quickly see if the EES of the calculated impact is reasonable, based on a visual comparison of the damage.



The three drop down lists at the top of the Selection dialog box enable the selection of vehicle make, EES range and impact location on the vehicle. The list box below will then show the vehicles in the EES catalog that fit the specified criteria.

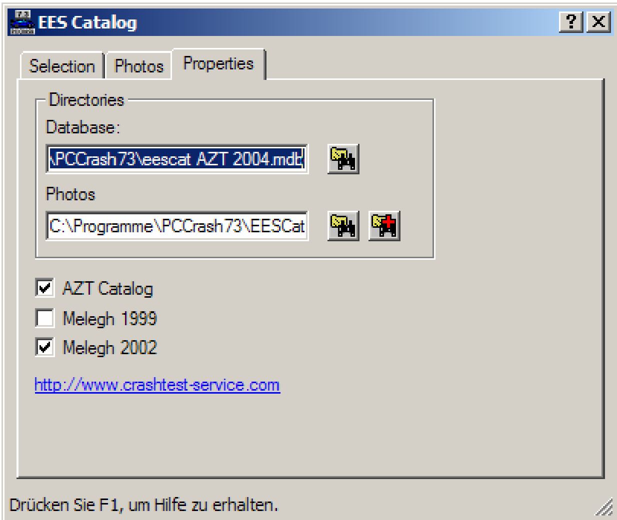
Double clicking on a photo in the preview window will open the photo in an edit window that can be expanded to full screen, if desired. Also the photos can be seen in detail in the Photos tab. Use the Buttons and to magnify or shrink the displayed image.

The photos of the PC-Crash EES Catalog are limited regarding the resolution. The photos with the original resolution are included in the EES CDs, which are available separately.



In the Properties tab the location of the database can be chosen. Normally there is no need to change this value, since all three available EES Catalogues can be chosen by selecting the check boxes:

- AZT Catalog
- Melegh 1999
- Melegh 2002



As there can be multiple EES catalogues selected simultaneously there can be specified multiple search paths to the EES images available on the EES CDs (which are available separately to PC-Crash). Press the button to add a directory to the search paths.

The internet link http://www.crashtest-service.com opens your internet browser with the given address. Crash test-Service provides – against payment of a fee – a large online database for determining EES values.

Note that you can resize the EES Catalog at the bottom right corner. This will give you a better look on some data in the list and the picture.

Crash 3 – EBS Calculation

Vehicle crush energy can be calculated according to the CRASH3 (Calspan Reconstruction of Accident Speeds on the Highway) damage algorithm, based on published NHTSA (National Highway Traffic Safety Administration) test results. This allows a comparison of the CRASH 3 crush energy with the crush energy determined using the PC-Crash impulse-restitution impact model.

EBS can be compared directly with EES as long as the EBS is above about 10 mph (16 km/h), so that the effects of restitution energy (included only in the EES calculation) are negligible. For smaller values, manual EES calculation of crush energy should be done instead of CRASH 3 EBS calculations. This is especially true because most of the tests in the provided NHTSA database typically have an EBS of 30 or 35 mph, such that the accuracy of the CRASH 3 EBS calculations will decrease for EBS values not in this speed range.

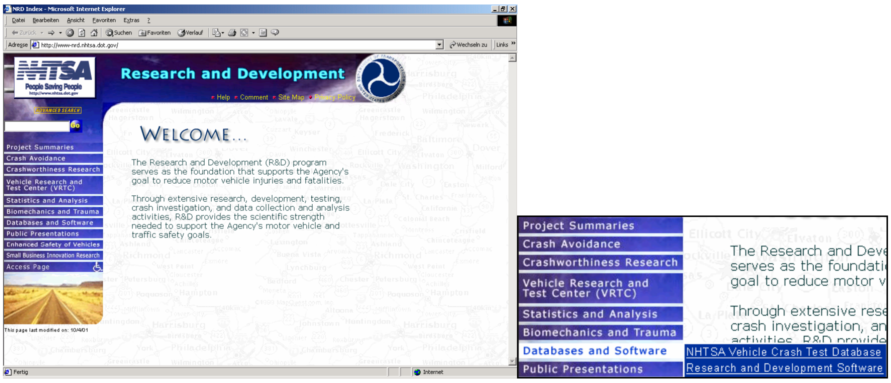
The EBS values calculated using this option can also be used as an EES input to the Collision Optimizer. Remember to increase the EES weighting from the default 0% in this case.

Crash 3 – EBS Calculation opens a group of dialog boxes, as follows:

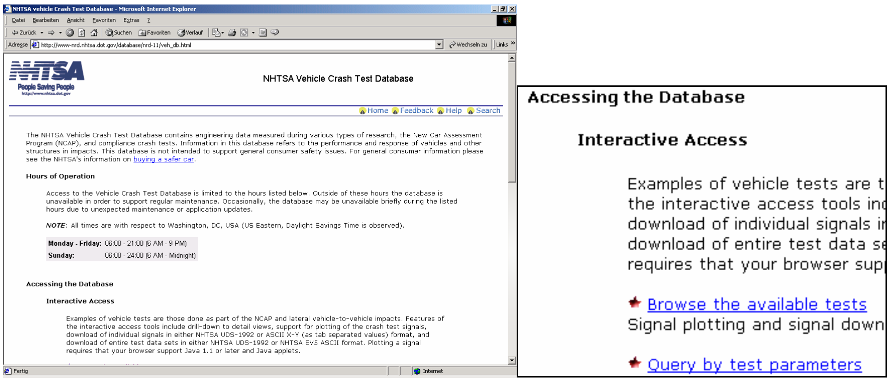
NHTSA Database

Database with tested vehicles as basis for the EBS calculation.

For detailed analysis of the tests the homepage of the National Highway Traffic Safety Administration (NHTSA) is linked (http://www-nrd.nhtsa.dot.gov/).

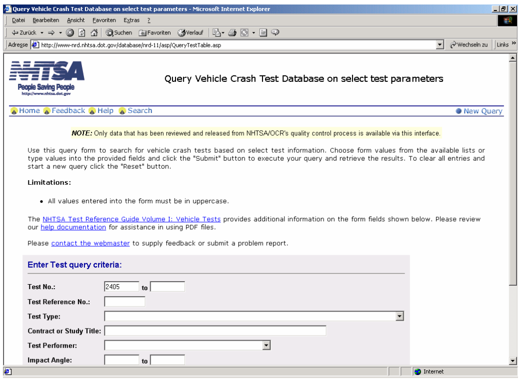


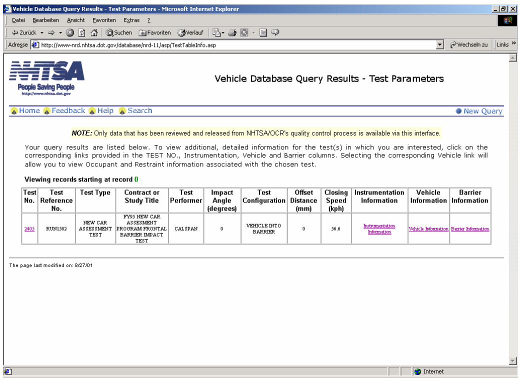
Follow the link under ‘Databases and Software’ to the ‘NHTSA Vehicle Crash Test Database’.



In the lower part of this side (‘Accessing the Database, Interactive Access’) you will find a link ‘Query by test parameters’ which will open the side ‘Query Vehicle Crash Test Database on select test parameters’ (Address: http://www-nrd.nhtsa.dot.gov/database/nrd-11/asp/QueryTestTable.asp).

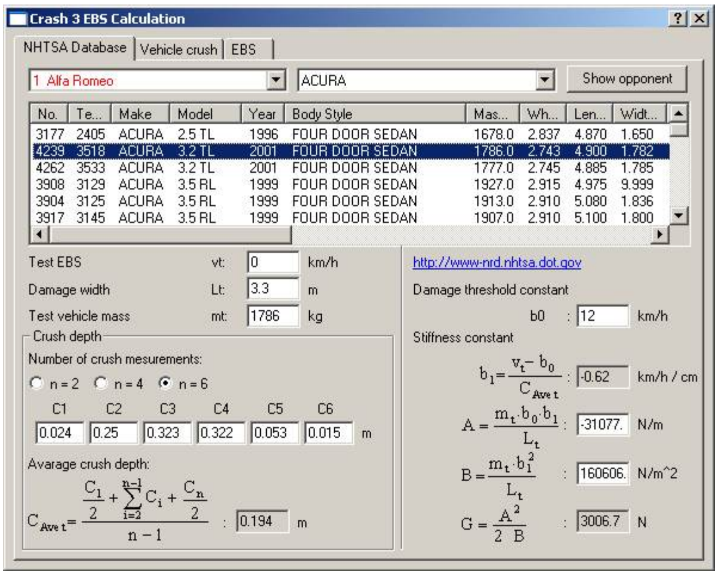
Use for example the **Test No.** from the second column within the database as query criteria.





Selecting Test No. a description of the test will be displayed and it is possible to download the complete testreport.

Under Instrumentation Information the diagrams of the individual sensors and dynamometers are linked, further a specification of the tested vehicle (Vehicle information) and a description of the Crash barrier (Barrier information).

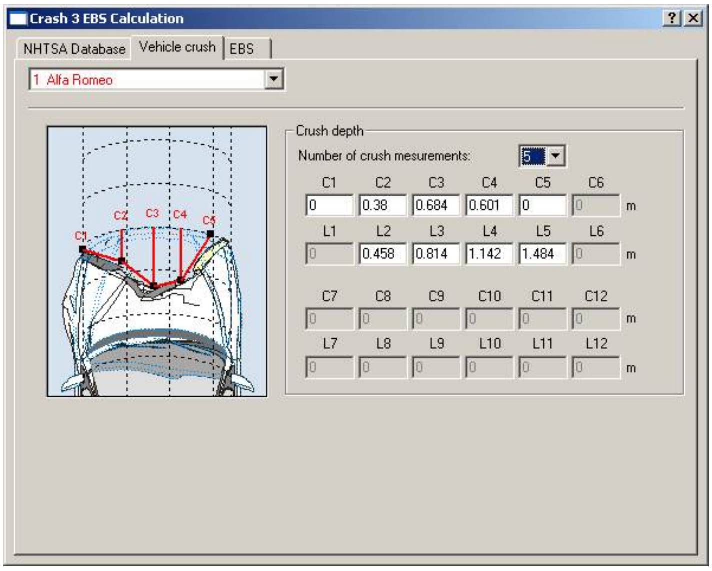


This dialog box contains a list box of NHTSA barrier impact tests. By selecting a particular test, the values needed to calculate the A and B crush coefficients will appear in the text boxes below. The A and B coefficients will be calculated automatically, based on the test EBS (Equivalent Barrier Speed), crush and weight, and the assumption of a linear crush relationship between the user-defined Damage Threshold EBS and the test EBS. The calculated average crush $C_{ave\ t}$ is a slight approximation, as discussed by Prasad ("CRASH3 Damage Algorithm Reformulation for Front and Rear Collisions", SAE 900098).

Note: *The test EBS is only correct for the calculation if the test vehicle struck a fixed non-deformable barrier. The EBS must be calculated and entered manually by the user for NHTSA tests involving moving or deformable barriers. If the barrier is another vehicle in the database the button **"Show opponent"** selects the barrier.*

Test data not contained in the provided database can be entered manually in the lower left portion of the dialog box to calculate A and B crush coefficients. The A and B coefficients can also be typed in the lower right portion of the dialog box if they are known from an external source.

Vehicle Crush

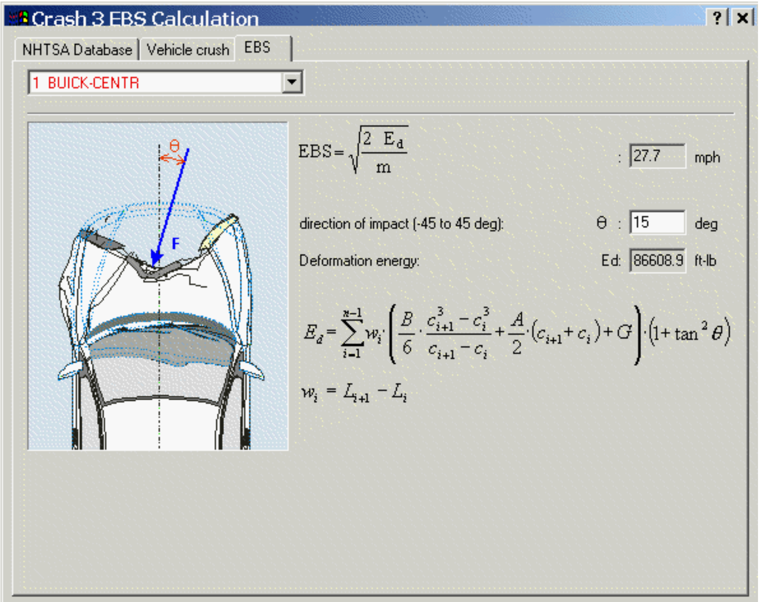


The incident vehicle crush is entered here. Two to twelve crush measurements can be entered. The crush measurements do not need to be spaced at even intervals.

The crush measurements can be entered by using the appropriate text boxes or by placing them with the mouse in the picture. The image correlates with the values in the text boxes to visualize the deformation. The grid shown in the picture always measures 0.5 m.

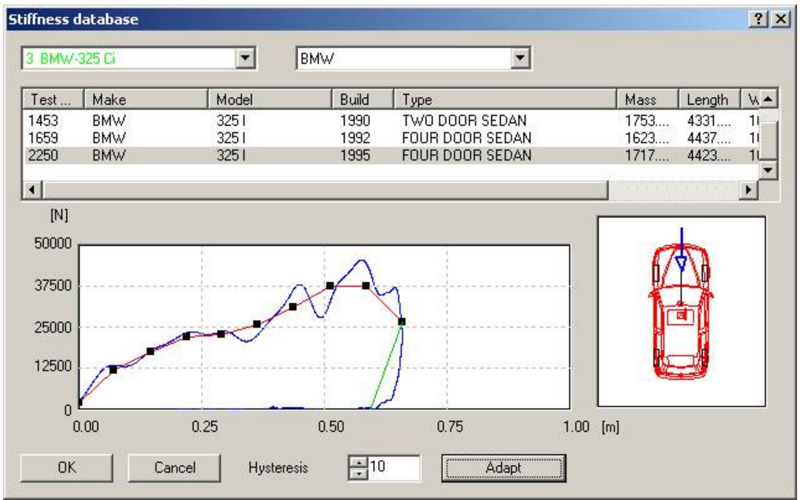
There is no need to locate the crush with respect to the COG, since only the EBS (and not the resulting vehicle rotation change) will be calculated with this option.

EBS



The Equivalent Barrier Speed and deformation energy of the incident vehicle are shown here, based on the defined impact force direction. This angle is limited to a maximum of +/-45°.

Stiffness Database



The stiffness database hold stiffness parameters of numerous vehicles collected from real crash tests. These parameters are provided as force/distance characteristics.

These characteristics can be used in the stiffness based impact model instead of the linear stiffness function to make the penetration forces more realistic. For more details on the stiffness based impact model see chapter **Stiffness based impact model** on page 176.

Vehicle

This is the selection window for the vehicle you want to apply the stiffness characteristics to.

Test vehicle

First select the manufacturer of the test vehicle and then select one of the available test records in the database. When a record is selected its force/distance characteristics will be displayed as blue line. The principle direction of force of the test case is shown as a blue arrow in the schematic picture on the right side of the dialog.

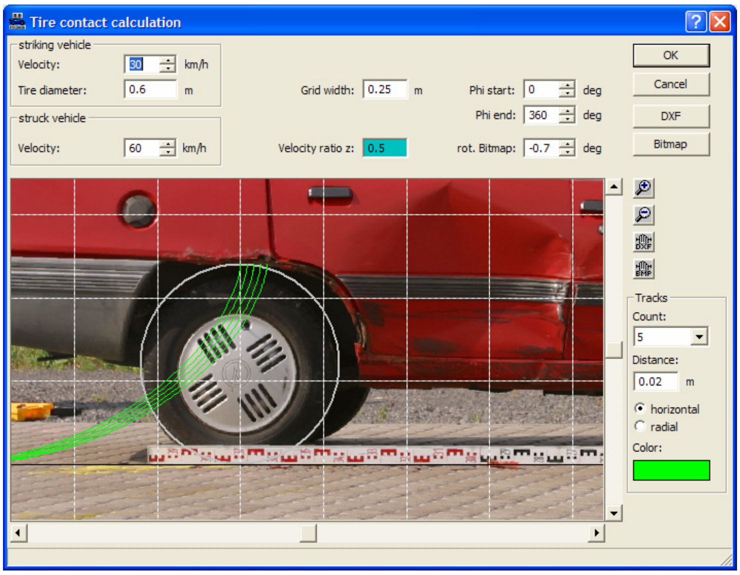
Adapt

Press this button to adapt the red line with its ten grid points to the blue line. You can manually modify the red line by dragging the grid points with the mouse to customize your stiffness function.

Hysteresis

The hysteresis (green line) is described as percent-value of the total deformation distance. It shows the elasticity of the vehicle. Usually a value of 10 percent works well.

Tire contact calculation



The 'Tire contact calculation' window is activated. This is a program tool for the calculation and analysis of tire contact tracks.

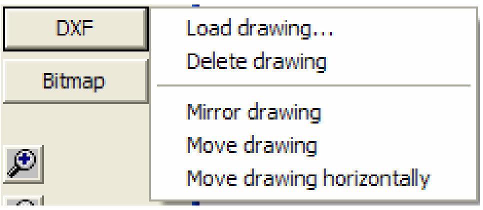
OK

Accepts the changes made and closes the dialog box..

Cancel

Closes the dialog box without accepting the data.

DXF



A new menue option is displayed:

Load drawing…: activates the „Select DXF-filename" window

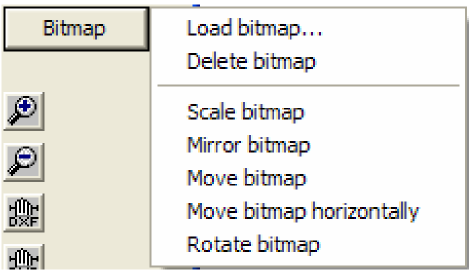
Delete drawing: deletes the active drawing

Mirror drawing: the drawing can also be mirrored

Move drawing: the drawing can be moved, the same function as the (Move DXF-drawing) Button.

Move drawing horizontally: the drawing can be moved horizontally, the same function as the (Move DXF-drawing) Button in combination with the pressed <SHIFT> key.

Bitmap

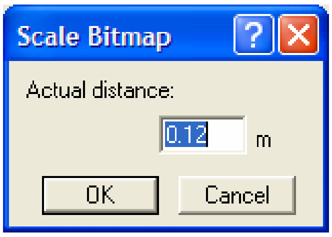


Activates an option menu:

Load Bitmap ...: This loads a bitmap file into the “Tire contact calculation” view.

Delete bitmap: This deletes a bitmap that has been loaded into the “Tire contact calculation” view.

Scale bitmap: The size of the bitmap can be changed. This is done by entering the “Actual distance” in the Scale Bitmap dialog box.



Mirror bitmap: active bitmap is mirrored

Move bitmap: This moves the bitmap (with the left mouse button held down) to any position desired.

Move bitmap horizontally: This moves the bitmap only horizontally (with the left mouse button held down) to any position desired.

Rotate bitmap: This enables the selected bitmap to be rotated by a defined angle. Press and hold the left mouse button while moving the cursor along any desired reference line in the bitmap.

Move the viewpoint closer to the vehicles

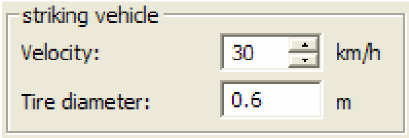
Move the viewpoint further from the vehicles

This enables the drawing to be moved by putting the cursor on the drawing and holding the left mouse button, and drawing a line to the new location.

This enables the bitmap to be moved by putting the cursor on the bitmap and holding the left mouse button, and drawing a line to the new location.

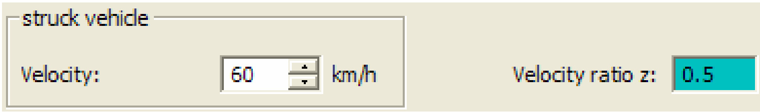
Striking vehicle

For the striking vehicle the velocity and the Tire diameter have to be defined. The shape of the tire diameter is displayed in grey color.



Struck vehicle

For the struck vehicle the velocity has to be defined. Basis fort he calculation is the velocity ratio of the two vehicles. The appropriate value is displayed as Velocity ratio, which is defined as ratio of velocity of striking vehicle and velocity of struck vehicle



Grid width

Defines the spacing between the grid lines.

Phi Start

Defines the starting point of the drawing of the tire contact traces in relation to the rotating angle of the striking tire.

Phi End

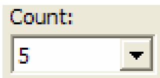
Defines the end-point of the drawing of the tire contact traces in relation to the rotating angle of the striking tire.

Rot. Bitmap

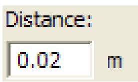
The bitmap can be rotated using this input box by defining the rotation angle. The rotating angle of the active bitmap is displayed in this box.

Tracks

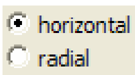
Input area for the parameters for the track drawing.



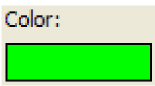
Selection box for the definition of the **count** of tracks; possible are 1 – 20 tracks, the distance between the drawings is defined under *Distance*



the **distance** between the drawings is defined, the count of the tracks is defined under *Count*



the tracks are displayed **horizontal** or **radial**



Selection box for the display color of the tracks.

Multibody System

The multibody system parameters can be viewed and modified using this menu item. Multibody systems can be pedestrians, passengers, two-wheeled vehicles or other custom objects. Each multibody system consists of one body or a number of bodies interconnected by joints and/or spring/damper systems.

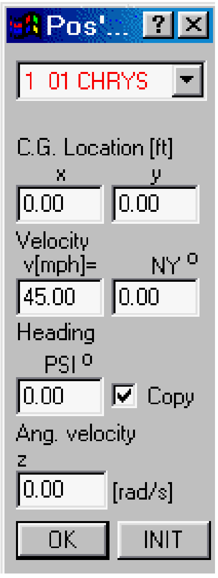
See the **Multibody Model** chapter for complete details on the use of multibody systems.

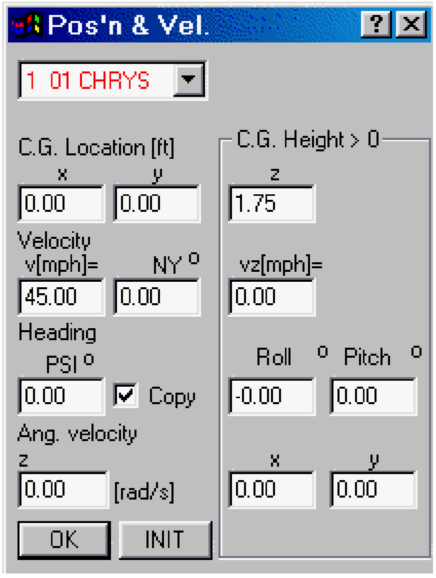
Dynamics

This menu is for positioning and applying initial conditions to vehicles, for performing kinematic calculations, for defining vehicle paths, and for defining scene characteristics.

Position & Velocity (F7)

The Position & Velocity dialog box allows the initial location, velocity, heading and rotational velocities to be specified for each vehicle. In addition, for 3D simulations (center of gravity height > 0), the vertical position and velocity, and the roll and pitch angles and rotational velocities can be specified. The intermediate values of all of these parameters, at any point along a vehicle's path, can also be viewed in this dialog box.





Description - Position & Velocity dialog box items

Vehicle

This drop down box is for selecting the vehicle for which the parameters are to be defined.

C.G. Location

The position of the center of gravity of each vehicle in the scene can be defined in the text boxes for the X, Y and (for 3D simulations) Z coordinates. The static height of the center of gravity defined in the Vehicle Geometry dialog box is the default value for the Z coordinate.

Velocity

Three values are used to define the velocity:

- **V** Velocity in X-Y plane
- **NY** Velocity direction (Degrees CCW from global X axis)

- **Vz** Velocity in Z direction (for 3D simulations only, positive upward)

Angles

The yaw and (for 3D simulations) roll and pitch describe the rotation of the vehicle in reference to its three main axes:

- **PSI** - Vehicle heading angle (Degrees CCW from global X axis). The yaw angle of a vehicle is equal to NY minus PSI.
- **Roll** - Rotation about the longitudinal axis (Degrees CCW from level, if viewed from the front).
- **Pitch** - Rotation about the lateral axis (Degrees CCW from level, if viewed from the left).

Copy

The velocity direction NY will be copied to the vehicle heading angle PSI, if this is selected. In this case, there is no yaw at the start of the simulation.

If this check box is deactivated, different values can be chosen for NY and PSI, in which case an initial yaw condition is present. Additionally, a reversing vehicle can be defined by making NY opposite to PSI.

Angular velocity

The three values of the angular velocity around the respective vehicle axes are:

- **z** axis - Yaw velocity (positive CCW)
- **x** axis - Roll velocity (positive CCW, viewed from front)
- **y** axis - Pitch velocity (positive CCW, viewed from left).

All values are in radians/second (1 radian = 57.296^{o}) or deg/second, as defined in **Options – Options – Default Settings**.

OK

Accepts the data entered and closes the dialog box.

INIT

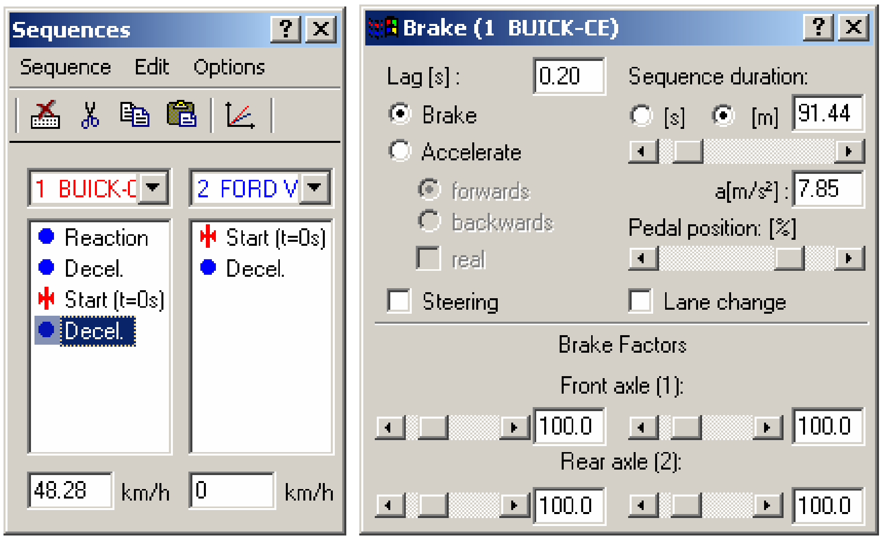
After pressing the Initialize button the following values are set to zero:

- Roll - Rotation about the longitudinal axis
- Pitch - Rotation about the lateral axis
- Rotational velocity about the X axis
- Rotational velocity about the Y axis
- Rotational velocity about the Z axis.

Note: *In the case of a vehicle with a trailer, the position and velocity values are adjusted automatically for the trailer when changes are made to the tow vehicle's values.*

Sequences (F6)

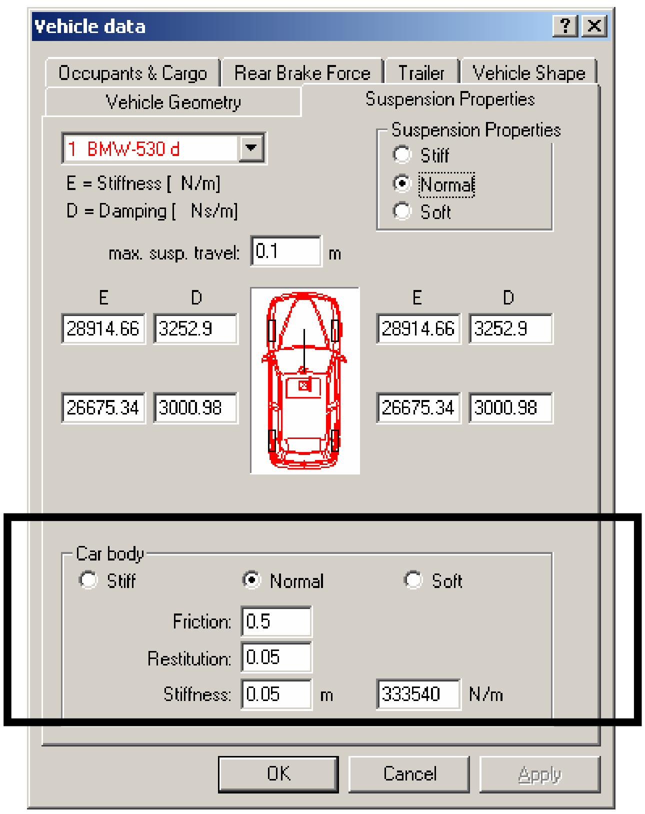
In the Sequences dialog box, various sequences can be specified for each vehicle, including driver reaction, vehicle braking, acceleration and steering.



Refer to the **Programming Sequences** chapter for a detailed description.

Rollover Detection

When Rollover Detection is selected, vehicle body to ground contact forces will be taken into account, based on the Car Body values specified in **Vehicle – Vehicle Settings – Suspension Properties**. Otherwise, the vehicle will fall through the road surface when inverted.



Refer to the description earlier in this chapter under **Vehicle – Vehicle Settings – Suspension Properties**, and also the **Technical Manual**, for more details on the rollover model.

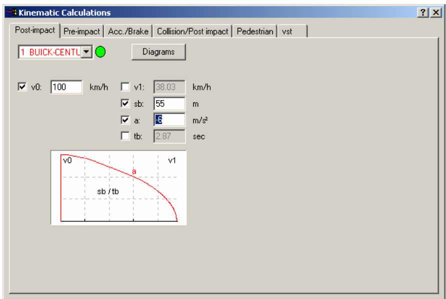
Kinematic Calculations

Window for performing kinematic calculations, whereby the calculated sequences can be assigned to the vehicles.

Selection of this menu item opens the Kinematic Calculations dialog box group, which includes the following tabs:

- Post-impact
- Pre-impact
- Acc./Brake
- Collision/Post impact
- Pedestrian
- Pedestrian throwing distance
- vst

Post-impact

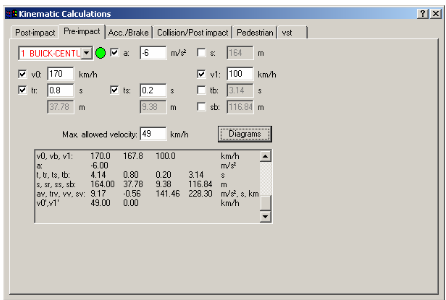


Enables the calculation of post-impact situations as pure braking / acceleration procedures. Select 3 of the 5 parameters, pushing the TAB key after the last text box input performs the calculations:

v0	Starting velocity
v1	Final velocity
sb	Acceleration / braking distance
a	Acceleration / deceleration
tb	Acceleration / braking time

The Diagrams button Diagrams opens the Diagrams window for viewing graphs of the calculated values. The correct sequences are added in the Sequences window simultaneously with this button.

Pre-impact



Pre impact driver reaction and braking before the Start (t = 0) can be examined with this option. Select 5 of the 8 parameters, pushing the TAB key after the last text box input performs the calculations:

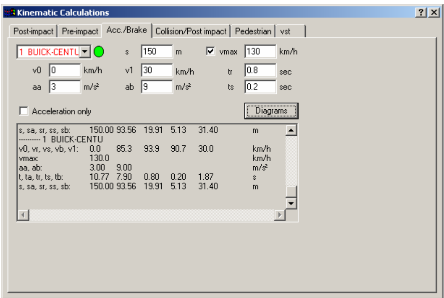
v0	Starting velocity
tr	Reaction time
ab	Braking deceleration
ts	Brake lag time
s	Stopping distance (sr + ss + sb)
v1	Final (impact) velocity
tb	Braking time
sb	Braking distance
vlimit	Maximum allowed velocity (speed limit)

In addition to these values, avoidance values and the impact velocity for the specified vlimit are calculated.

v0	Starting velocity
vb	Velocity at start of braking
v1	Impact velocity
a	Braking deceleration
t	Total time
tr	Reaction time
ts	Brake lag time
tb	Braking time
s	Total distance
sr	Reaction distance
ss	Brake lag distance
sb	Braking distance
av	Avoidance deceleration
trv	Avoidance reaction time
vv	Avoidance velocity
sv	Avoidance distance
v0'	Maximum allowed velocity (vlimit)
v1'	Impact velocity using maximum allowed velocity **v0'** instead of **v0**

The Diagrams button Diagrams opens the Diagrams window for viewing graphs of the calculated values. The correct sequences are added in the Sequences window simultaneously with this button.

Acc./Brake



Pre impact acceleration followed by braking before the Start (t = 0) can be examined with this option. The vehicle's speed can be limited by selecting vmax. When the specified acceleration values are sufficient to exceed this speed, the vehicle will maintain this constant speed once it has been reached.

v0	Starting velocity
aa	Acceleration during driver's reaction time
s	Total acceleration + braking distance
v1	Impact velocity
ab	Braking deceleration
vmax	Maximum allowed velocity

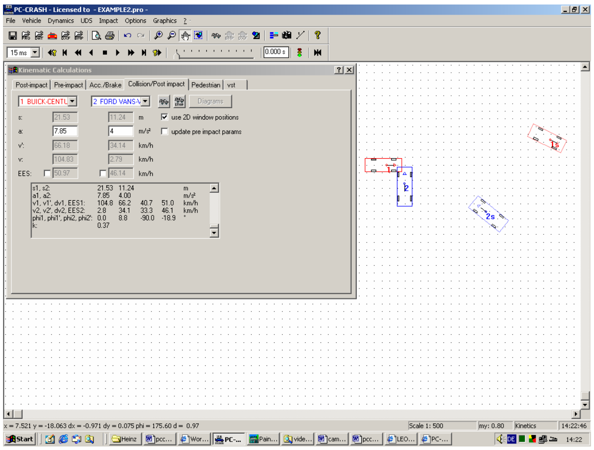
tr Reaction time
ts Brake lag time

Following values are calculated.

v0 Starting velocity
vr Velocity at start of reaction
vs Velocity at start of brake lag
vb Velocity at start of braking
v1 Impact velocity
vmax Vehicle's speed limit
aa Acceleration
ab Braking deceleration
t Total time
ta Acceleration time
tr Reaction time
ts Brake lag time
tb Braking time
s Total distance
sa Acceleration distance
sr Reaction distance
ss Brake lag distance
sb Braking distance

The Diagrams button Diagrams opens the Diagrams window for viewing graphs of the calculated values. The correct sequences are added in the Sequences window simultaneously with this button.

Collision/Post impact



The following values can be examined in this field

v Pre impact velocity
v' Post impact velocity
EES Equivalent energy speed
s Post impact distance
a Average post-impact deceleration

There are two methods of determining the pre-impact velocity, based on a specified average deceleration after the impact:

- Deactivate use 2D window position and specify a deceleration distance and rate in the provided text boxes.
- Activate use 2D window position and define the start and rest position by moving the vehicles with the mouse. To position the vehicle at the start click the 'Place Vehicle in Start Position' button and for the rest position, click

the ‘Place Vehicle in Rest Position’ button . Specify the average deceleration rate as above (the effect of vehicle orientation is not considered, as this is a simple kinematic calculation).

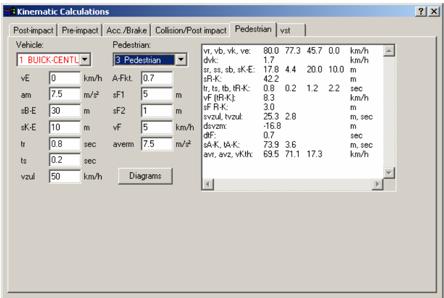
The values in the textboxes are updated immediately. If update pre-impact parameters is activated the collision velocity is updated for the pre-impact calculations.

The EES value for a vehicle can also be defined. The value for the second vehicle is then updated accordingly.

Following values are calculated.

s1	post impact distance vehicle1
s2	post impact distance vehicle2
a1	mean post impact deceleration vehicle1
a2	mean post impact deceleration vehicle2
v1	impact velocity vehicle1
v1’	post impact velocity vehicle1
dv1	Delta v vehicle1
EES1	EES for vehicle1
v2	impact velocity vehicle2
v2’	post impact velocity vehicle2
dv2	Delta v vehicle2
EES2	EES for vehicle2
phi1	pre-impact direction of velocity vector (course-angle) veh1
phi1’	post-impact direction of velocity vector (course-angle) veh1
phi2	pre-impact direction of velocity vector (course-angle) veh2
phi2’	post-impact direction of velocity vector (course-angle) veh2
k	coefficient of restitution

Pedestrian



The following values can be examined in this field

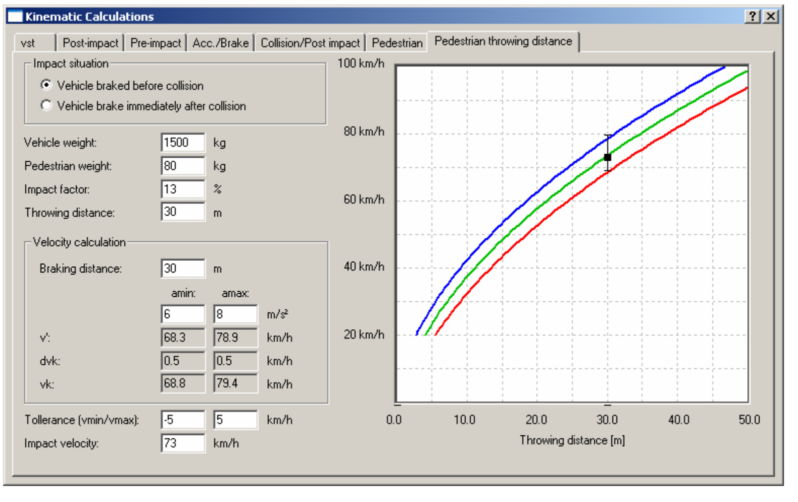
vE	Velocity at the end of the investigated distance
am	mean brake deceleration
sB-E	Distance start of braking - rest position
sK-E	Distance collision - rest position
tr	Reaction time
ts	Brake lag time
vzul	allowed velocity
A-Fkt	Collision factor
sF1	Distance up to collision
sF2	Needed distance for the pedestrian starting from the collision, in order to leave the danger area
vF	Pedestrian velocity<
averm	mean acceleration for avoidance investigation

Following values are calculated.

vr	Velocity at start of reaction
vb	Velocity at start of braking
vk	Collision velocity

ve Velocity at end of investigated distance
dvk Change of velocity due to collision with pedestrian
sr Distance during reaction
ss Brake lag distance
sb Braking distance
sK-E Distance collision-rest position
sR-K Distance reaction-collision
tr Reaction time
ts Brake lag time
tb Braking time
tR-K Time reaction-collision
vF (tR-K) Calculated velocity of the pedestrian, in order to cover the pre-impact distance during reaction and collision
sF R-K Pre-impact distance of the pedestrian, calculated with the vF (tR-K)
svzul Stopping distance out of vzul
tvzul Stopping time out of vzul
dsvzm Distance rest position out of vzul - collision
dtF Time the pedestrian would have needed to leave the danger area, starting from point of impact
sA-K Distance reaction-collision
tA-K Time reaction-collision
vr Velocity for distance dependent avoidance
vz Velocity for time dependent avoidance
vKth Theoretical velocity of the pedestrian passing the collision scene

Pedestrian throwing distance



On this dialog page the impact velocity of pedestrian accidents can be calculated by determining the throwing distance of the pedestrian. This calculation is based on statistical evaluation of real pedestrian accidents where two different situations have been discovered to influence the throwing distance:

- Vehicle braked before collision
- Vehicle braked immediately after collision

Vehicle weight

Specifies the total weight of the vehicle

Pedestrian weight

Specifies the weight of the pedestrian

Impact factor

The impact factor is determined by several components like impact height, vehicle type and other parameters.

Throwing distance

The throwing distance specifies the measured throwing distance of the pedestrian. The Measurement begins at the point of impact.

Braking distance

This value specifies the total braking distance of the vehicle.

amin and amax

These values specify the minimum and maximum deceleration of the vehicle while braking.

v'

These values specify the post-impact velocities at amin resp. at amax.

dkv

These values specify the loss of velocity at the point of impact at amin resp. at amax.

vk

These values specify the pre-impact velocity (the impact velocity) at amin resp. at amax. These values are displayed in the diagram as a vertical black line.

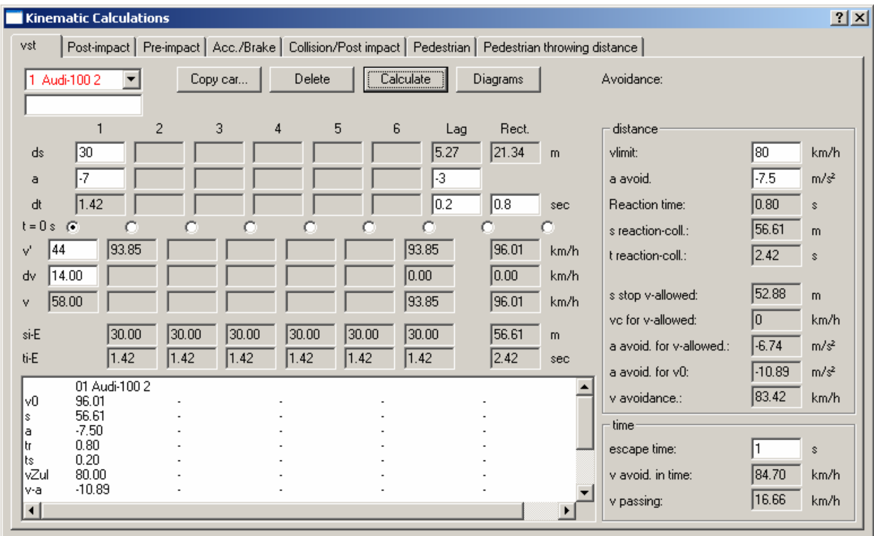
Tolerance (vmin / vmax)

The tolerance velocities specify the tolerance lines in the Distance-Velocity diagram, i.e. the blue and the red graph.

Impact velocity

In this field the suspected impact velocity can be entered. The value is displayed in the diagram as black marker. If all values have been entered correctly, the marker should now reside between the tolerances graphs in the diagram.

vst



Kinematic calculation, if different sequences are known.

The input takes place from left to the right. Start with a pre-impact velocity **v'** and a change of velocity **dv** (if there is no impact use dv = 0). Then the input of a sequence can be done, define 2 values out of the 3 parameters **ds**, **a**, **dt**. By clicking the **Calculate** Button the calculation is started.

Within the right range of the input window a brake lag (**Lag**) - and a reaction (**Rect**.) sequence can be defined.

Within the textboxes **si-E** and **ti-E** the summed up distances and times of the sequences are displayed.

Pressing the **Diagrams** button the sequences are transferred to the selected vehicle and represented in the diagram window. At the same time the appropriate sections (Accelerate. Brake, Crash) are inserted in the Sequences window. The position of the sequences respective the Start (**t=0s**) sequence can be defined using the option field between the different sections.

For avoidance investigations an avoidance acceleration (**a avoid**) and an allowed velocity (**vlimit**) can be defined.

Spatial avoidance

The avoidance values (**a avoid** and **vlimit**) are used together with the **reaction time**, the distance from the point of reaction to the point of impact (**s reaction-coll.**) and the time from the point of reaction to the point of impact (**t reaction-coll.**) to calculate some situations that could have avoided the impact.

If the vehicle had been driving with the velocity **vlimit**, then it would have needed a distance of **s stop v-allowed** to brake, a deceleration value of **a avoid. for v-allowed** to stop before the point of impact or it would have had an impact velocity of **vc for v-allowed**.

Additionally the necessary deceleration value **a avoid. for v0** to stop the vehicle with the current velocity and the maximum velocity **v avoidance** that would just have avoided an impact, are displayed.

Temporary avoidance

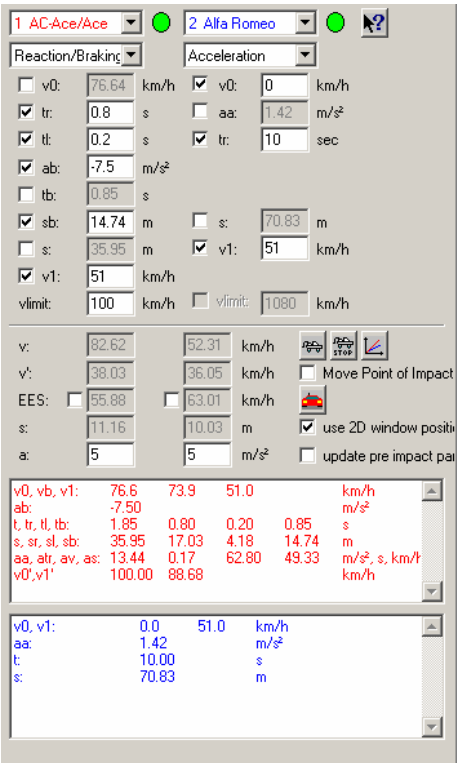
If the **escape time** needed for the collision partner to leave the area of impact safely can be specified, the maximum velocity of the vehicle **v avoid. in time** to just pass the collision partner and the velocity **v passing** that the vehicle has when passing the partner, can be calculated.

If the escape time is so high that the avoidance velocity must be so small that the vehicle would have stopped before the area of impact (i.e. **v passing** less or equal 0), no values are calculated.

Kinematics Toolbar

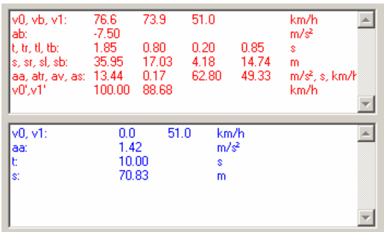
This opens the Kinematics toolbar, which is for examining velocity, distance, time and acceleration. One of the following three pre-impact sequence options can be selected for each vehicle, using the drop down lists at the top of the Kinematics toolbar:

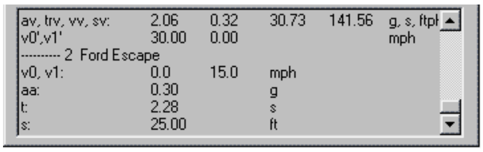
- Acceleration
- Acc./Braking
- Reaction/Braking



For the Acceleration and Reaction/Braking options, parameters can be changed only after activating the corresponding check box. A certain number of parameters must be selected for these two options before the calculation can be done. This is indicated when the control light beside the vehicle selection box changes from red to green. The Acc./Braking option needs a value for every parameter. Pushing the TAB key after the last text box input performs the calculations.

The calculated values are shown in the text areas at the bottom of the Kinematics toolbar. If the screen resolution is large enough, for each vehicle in the toolbar there is a separate text area. Previously calculated values can be viewed by scrolling up through this text area with the scroll bar on the right.

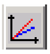


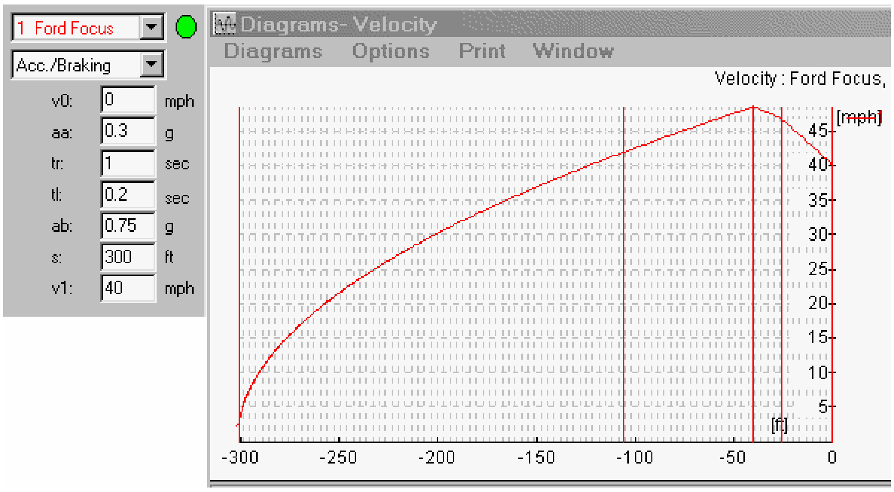


The following variables can appear in this text area, depending on the option chosen:

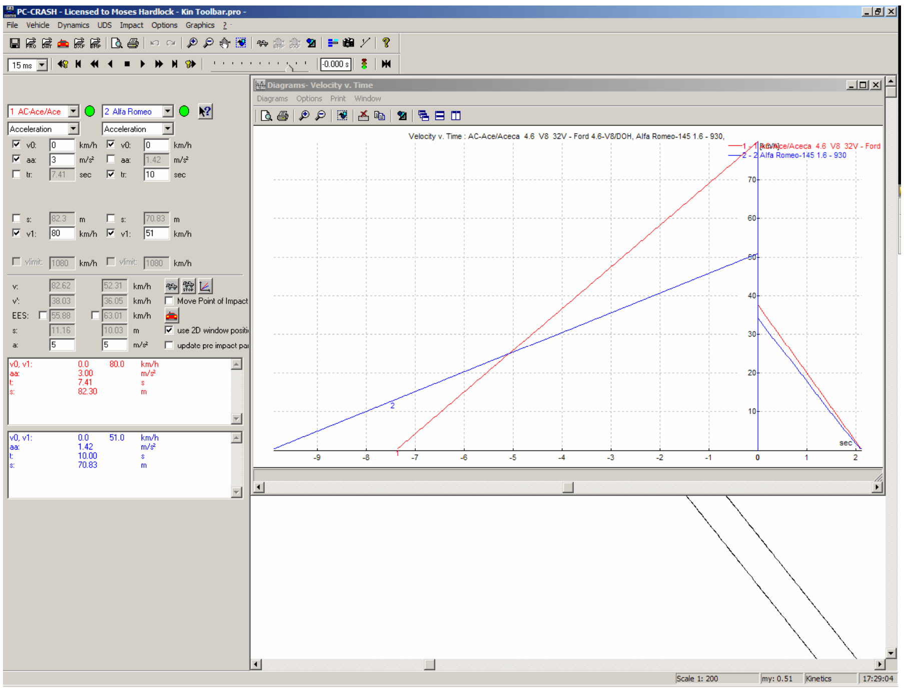
v0	Starting velocity
vr	Velocity at start of reaction
vl	Velocity at start of braking
vb	Velocity at end of brake lag time
v1	Impact velocity
ab	Braking deceleration
aa	Acceleration
t	Total time
tr	Reaction time
tl	Brake lag time
tb	Braking time
s	Total distance
sr	Reaction distance

sl Brake lag distance
sb Braking distance
av Avoidance deceleration
atr Avoidance reaction time
av Avoidance velocity
as Avoidance distance
v0' Maximum allowed velocity (vlimit)
v1' Impact velocity using maximum allowed velocity **v0'** instead of **v0**

The Diagrams button  opens the Diagrams window for viewing graphs of the calculated values. The correct pre-impact sequences are added in the Sequences window simultaneously with this button.



Acceleration

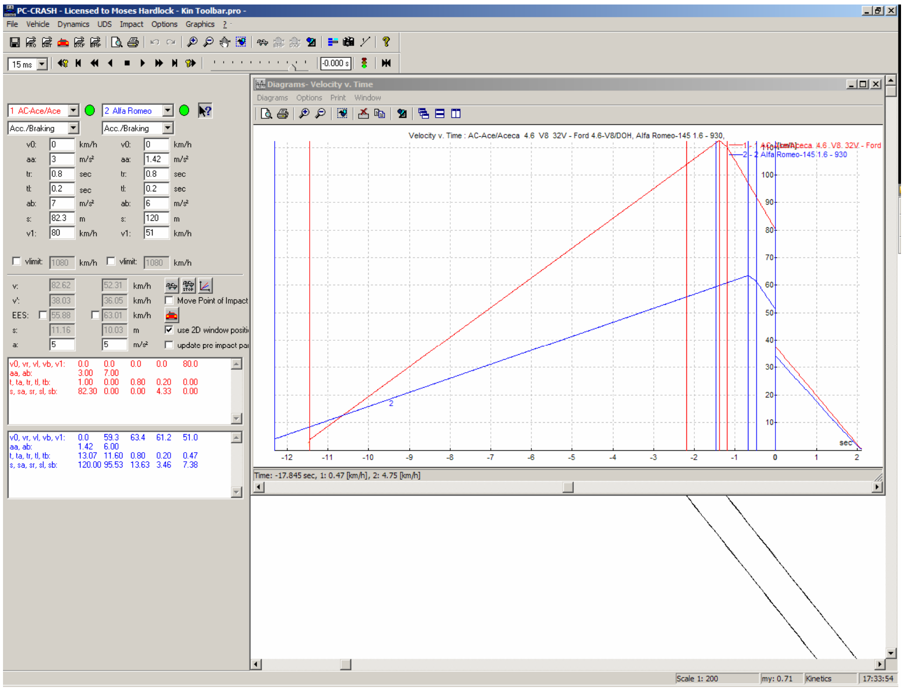


Pre impact acceleration before the Start (t = 0) can be examined when this option is selected in the drop down list. Select 3 of the top 5 parameters:

v0 Starting velocity

aa Acceleration
tr Acceleration time
s Acceleration distance
v1 Impact velocity

Acc./Braking

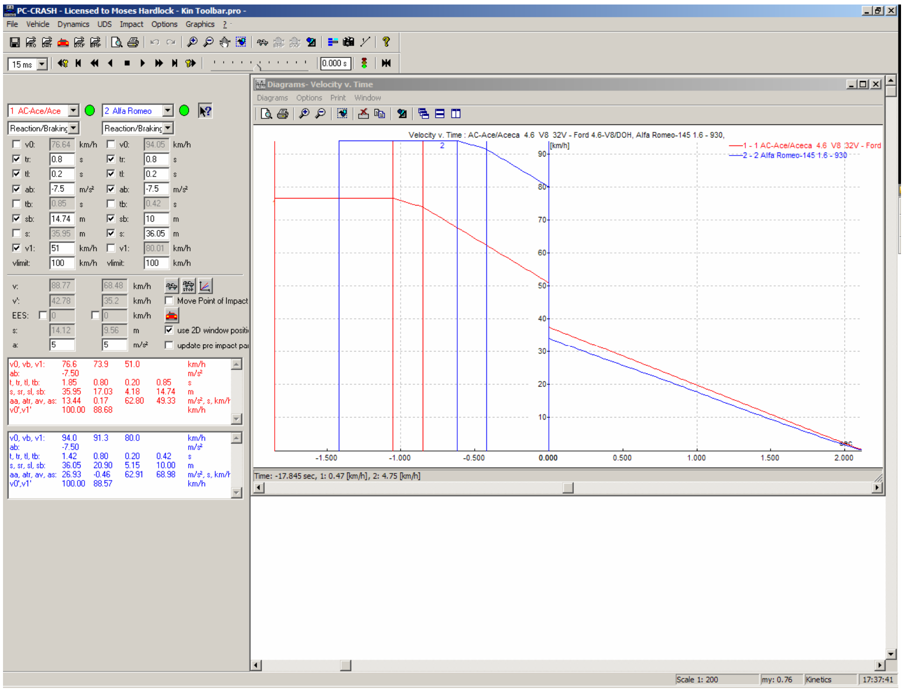


Pre impact acceleration followed by braking before the Start (t = 0) can be examined with this option. The parameters are:

v0 Starting velocity
aa Acceleration during driver's reaction time
tr Reaction time
tl Brake lag time
ab Braking deceleration
s Total acceleration + braking distance
v1 Impact velocity
vlimit Maximum allowed velocity

The vehicle's speed can be limited by selecting **vlimit**. When the specified acceleration values are sufficient to exceed this speed, the vehicle will maintain this constant speed once it has been reached.

Reaction/Braking



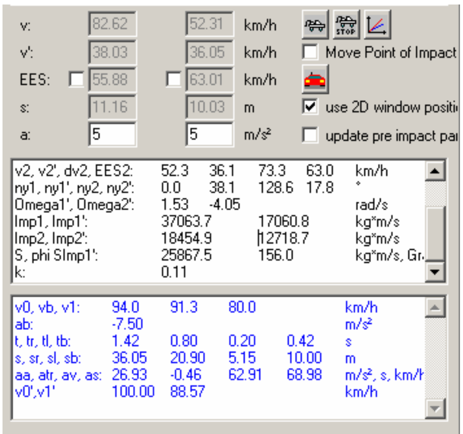
Pre impact driver reaction and braking before the Start (t = 0) can be examined with this option.

Select 5 of the top 8 parameters:

v0	Starting velocity
tr	Reaction time
tl	Brake lag time
ab	Braking deceleration
tb	Braking time
sb	Braking distance
s	Stopping distance (sr + ss + sb)
v1	Final velocity
vlimit	Maximum allowed velocity (speed limit)

In addition to these values, avoidance values and the impact velocity for the specified **vlimit** are calculated.

Post-Impact



The following values can be examined in this field

v	Pre impact velocity
v'	Post impact velocity
EES	Equivalent energy speed
s	Post impact distance
a	Average post-impact deceleration

There are two methods of determining the pre-impact velocity, based on a specified average deceleration after the impact:

- Deactivate **Use 2D Window Position** and specify a deceleration distance and rate in the provided text boxes.
- Activate **Use 2D Window Position** and define the start and rest position by moving the vehicles with the mouse. To position the vehicle at the start click the Place Vehicle in Start Position button and for the rest position, click the Place Vehicle in Rest Position button. Specify the average deceleration rate as above (the effect of vehicle orientation is not considered, as this is a simple kinematic calculation).

The values in the Kinematics toolbar are updated immediately. If **Update Pre-impact Parameters** is activated the collision velocity is updated for the pre-impact calculations.

The EES value for a vehicle can also be defined. The value for the second vehicle is then updated accordingly. Use the EES button to access the EES catalog for a comparison of damage.

The Diagrams button opens the Diagrams window for viewing graphs of the calculated values. The correct pre-impact sequences are added in the Sequences window simultaneously.

Avoidance in time

This option allows the calculation of temporal avoidance situations based on the computed simulation beginning from the point of reaction. This means, that the velocity of the chosen vehicle is gradually reduced until the vehicles are passing and no collisions occur. The reduced velocity is then considered as avoidance velocity of the chosen vehicle.

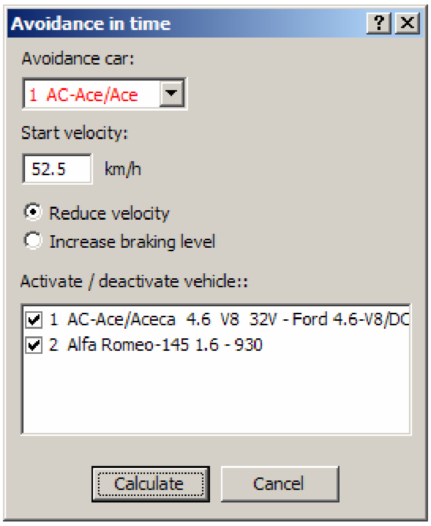
To make this function work properly the simulation must start in the reaction point of the chosen vehicle. Due to the fact that most simulations have their beginning in the first point of impact (because this is the situation where probably many parameters are known best) several steps have to be performed to shift the starting point to the point of reaction of the chosen vehicle.

First, simulate the pre-impact situation by simulating backwards. Make sure that all vehicles are simulated in the point of impact of the chosen vehicle. If e.g. sequences are not specified long enough, some vehicles may stop before (i.e. “after” in terms of time) the chosen car reaches the point of reaction by simulating backwards. In such a situation modify the sequences of these vehicles.

With the time slider set the simulation time to the point of reaction of the chosen car. Then press the button to set the starting point to the chosen time.

Eventually it is necessary to modify the sequences of the vehicles. All sequences that have been after the point of reaction of the chosen car must now be moved after the starting point (i.e. the starting point is now the point of reaction of the chosen car). By making a forward simulation you can verify if your sequences are now correct.

Now you can calculate the avoidance in time by choosing the appropriate menu option.



Avoidance car

This is the car you have chosen. For that car the velocity will be gradually reduced.

Reduce velocity

The velocity will be decreased in steps of 1 km/h to avoid a collision.

Increase braking level

The braking force will be increased in steps of 0.1 m/s² to avoid a collision. The avoidance by increasing the brake factor heaviyly depends on the available braking distance and the friction of the ground.

Start velocity

The start velocity of the avoidance car.

Switch on/off

Here you can specify which vehicles will be simulated during the calculation. Please note, that even when a vehicle is not active it can cause a collision by resting in the path of another vehicle. If any collision is detected the situation will not be considered an avoidance situation.

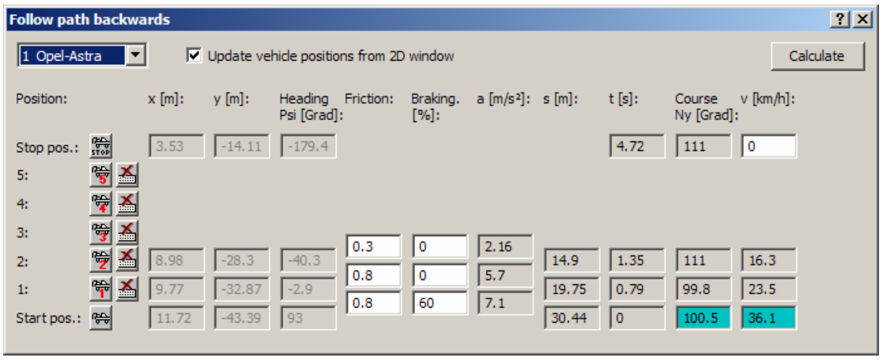
Calculate

The calculation will be performed until a velocity has been found where no collisions occur.

Kinematic follow path backwards…

By using this function of PC-Crash skid marks of vehicles can be tracked to determine the post-impact situation using a kinematic calculation model. The calculated post-impact velocities and other parameters can be used for further investigating the pre-impact situation.

When activating this menu option the following window appears:



In addition to the start position (collision position) the rest position and up to 5 intermediate positions can be specified. The rest position must always be specified, the intermediate positions are optional. Intermediate positions can be removed by selecting the button.

By deselecting the option **Update vehicle positions from 2D window**, previously defined positions can be entered in the appropriate input fields by specifying their coordinates and heading instead of the visually defined positions.

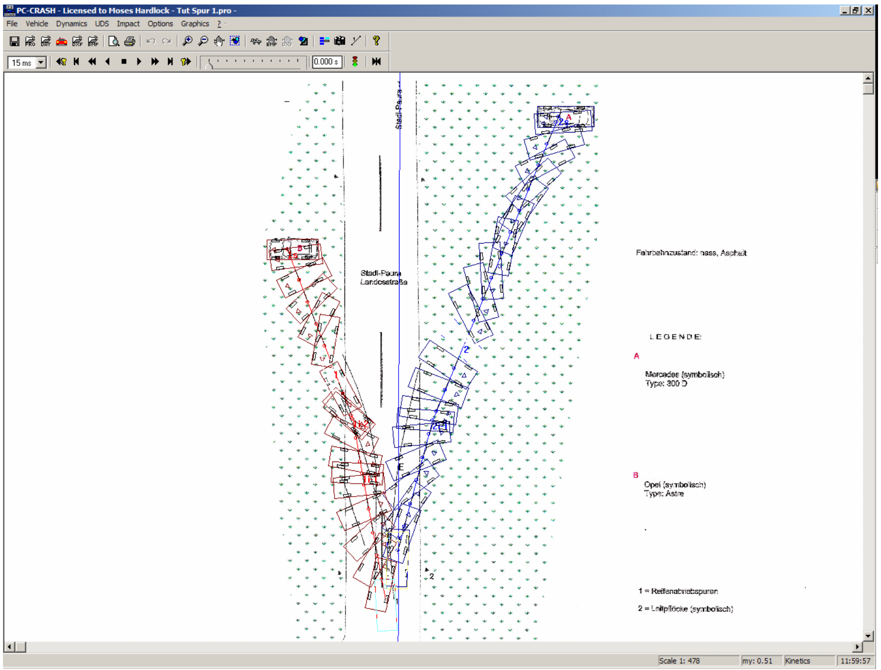
For each sequence between two positions a friction value between tires and ground and the amount of braking in percent can be specified. From these values and the given positions the results are calculated.

a	The deceleration value calculated for the sequence between two points. The value is determined by the amount of braking and the friction. For the lateral part of the vehicle movement a brake value of 100% is assumed due to he lateral tire forces.
s	The total distance of the position, beginning at the rest position.
t	The total time the vehicle reaches the position, beginning from the start position.
Course Ny	The general moving direction of the vehicle (applied to the center of gravity). The post- impact course direction is displayed in the blue input field.
v	The calculated velocity of the vehicle at the position. The post-impact velocity is displayed in the blue input field. The velocity at the rest position can be set to a different value that 0 to indicate that the rest position is not the end of the movement.

By pressing the **Calculate** button, the calculation is performed and the results are applied to the specified vehicle. The vehicle will now move as specified by the positions.

Important note:

The vehicle movement using this calculation is performed as specified by the positions and values, not by physical constraints. Always double check your results by making a kinetic forward simulation.



Define Path Points

Paths for each vehicle can be specified. This allows the user to specify complicated vehicle motions without having to use multiple steering sequences. The specified paths are based on curves (modified B-spline 2nd order) defined by points selected by the user.

When activating this menu option the following toolbar appears or the Follow Path Toolbar is activated:

The functions of these tools are similar to those in the drawing program.

 moves and adjusts a previously-created path. A path can be deleted by clicking on it with this tool and then pressing the DELETE key.

 draws a new path. Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points, click the right mouse button. Note that the created path will always continue on to infinity before the start point and after the end point.

To add additional points to a path select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

 is used to rotate a path, after it has been selected with . The rotation center  is moved to the desired position on the screen, and then the path is rotated, both using the mouse with the left mouse button held down.

is to select the vehicle for which the path is to be drawn or modified.

A vehicle follows a specified path differently depending on whether the normal Kinetics simulation model or the Kinematics simulation model is used. The model is changed using the Simulation Model tool .

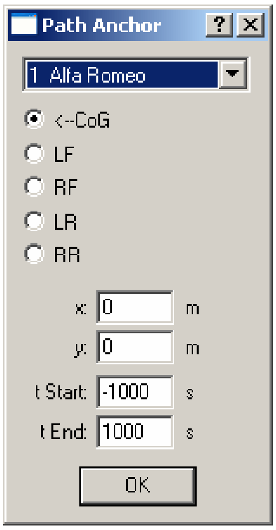
When the **Kinetics** simulation model is used, the vehicle motion will be governed by the laws of physics, the specified vehicle properties, and the **Driver Model** parameters. In this case the vehicle will not follow the path exactly, especially if the speed is too high. Refer to the **Technical Manual** for details on the Path physical model.

When the **Kinematics** simulation model is used, the vehicle will follow the specified path at the specified speed, regardless of whether it is obeying the laws of physics. It is up to the user to ensure that the vehicle motion is possible, as the kinematics model does not check this.

The **Define Path Points** menu option is active until it is turned off or another option requiring the mouse is performed.

Vehicle Anchor Point

This option is for selecting the point of the vehicle (C.G. or one of the wheels) that is to be anchored to the path specified in **Dynamics - Define Path Points**. The following dialog box appears.



The **x** and **y** text boxes allow the anchor point to be offset with respect to the Center of Gravity by any distance desired from the selected item (x is positive towards the vehicle's front, y is positive to the vehicle's left).

There is also the possibility to define the time span during the driver model will steer the vehicle along the defined path points. The **t Start** and **t End** text boxes specify the starting and the ending time of the driver model. These settings allow to have the car follow the path e.g. only in the pre-impact simulation.

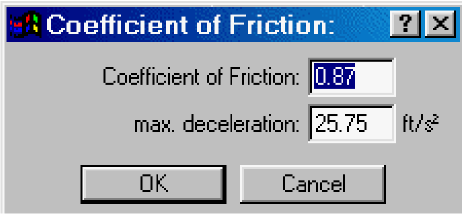
Define Friction Polygons

Areas of different friction can be defined directly on the screen. The following toolbar appears when selecting this menu option or the Friction Toolbar is activated::

The functions of these tools are similar to those in the drawing program.

moves and adjusts a previously-created polygon. A polygon can be deleted by clicking on it with this tool and then pressing the DELETE key.

is for defining a friction polygon. Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points of the polygon, click the right mouse button to close the last side of the polygon. A dialog box appears, allowing you to enter the desired friction coefficient.



Enter a value of friction for the polygon area. You can change the values at any time by double clicking in the polygon with the **Define Friction Polygons** menu option active.

To add additional points to a friction polygon select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

is used to rotate a polygon, after it has been selected with . The rotation center is moved to the desired position on the screen, and then the polygon is rotated, both using the mouse with the left mouse button held down.

is used to change the appearance of a polygon.

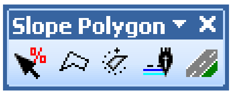
The **Define Friction Polygons** menu option is active until it is turned off or another option requiring the mouse is performed.

Note: *In case of overlapping polygons the values of the overlapping areas are determined by the values of the polygon which has been defined first. To avoid overlapping, use the Snap feature when drawing friction polygons. The Snap feature can be accessed by activating the Snap button of the Draw toolbar.*

When a vehicle goes from one polygon to another, the position of each wheel is used independently to determine which friction applies to it.

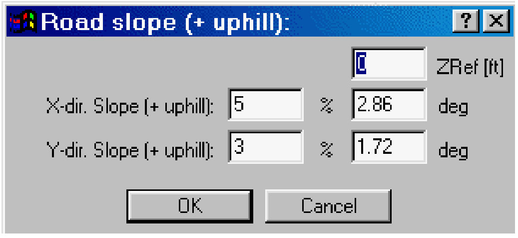
*Friction sequences defined using the **Sequences** menu option have a higher priority than friction polygons. In this case when a vehicle passes through a polygon the polygon friction coefficient is ignored.*

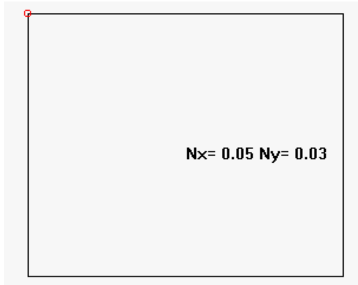
Define Road Slope



Road slopes can be entered in the Define Road Slope dialogue box. After creating the road slope, if the vehicle is to start out on it, the initial conditions must by set properly. This is done by placing the vehicle on the road slope with the Tow Truck tool and then pressing the New Simulation button .

is for defining a slope polygon. Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points of the polygon, click the right mouse button to close the last side of the polygon. A dialog box appears, allowing you to enter the desired elevation and slope(s). The first slope polygon point drawn is defined by a red circle on the main screen and is the one for which the elevation (Zref) is specified.





After closing, this box can be re-opened by double-clicking on the slope polygon with highlighted.

To add additional points to a slope polygon select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

is used to rotate a polygon, after it has been selected with . The rotation center is moved to the desired position on the screen, and then the polygon is rotated, both using the mouse with the left mouse button held down.

is used to change the appearance of a polygon.

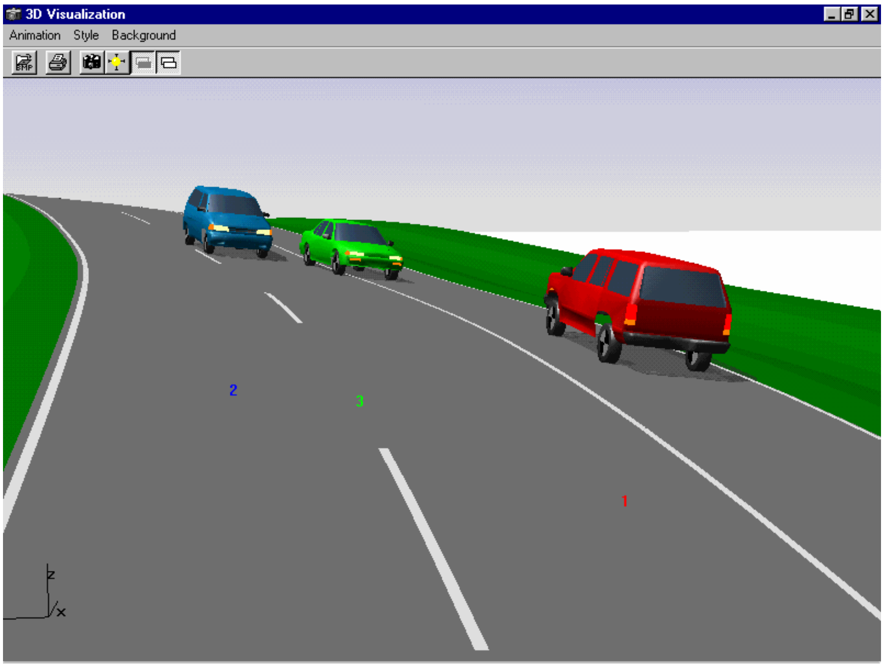
The **Define Slope Polygons** menu option is active until it is turned off or another option requiring the mouse is performed.

Note: *In case of overlapping polygons the values of the overlapping areas are determined by the values of the polygon which has been defined first. To avoid overlapping, use the* ***Snap*** *feature when drawing slope polygons. The Snap feature can be accessed by activating the Snap button of the Draw toolbar.*

When a vehicle goes from one polygon to another, the position of each wheel is used independently to determine which slope and elevation apply to it.

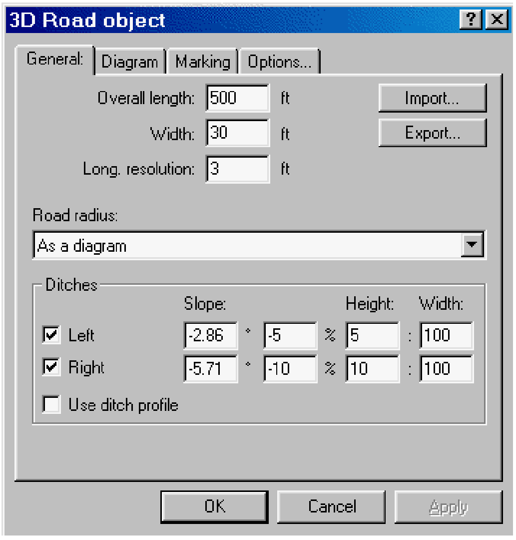
Generate 3D Road Object

Complex 3D road surfaces can be created or imported using this feature. The vehicles in the simulation will recognize the 3D road object surfaces during the simulation. After creating the 3D road object, if the vehicle is to start out on it, the initial conditions must by set properly. This is done by placing the vehicle on the 3D road object with the Tow Truck tool and then pressing the New Simulation button .



There are four tabs in the 3D Road Object dialog box - General, Diagram, Marking and Options.

General



Overall length

Input the overall length of the 3D road object. The default length is 100 meters or 328 feet.

Width

Input the width of the 3D road object roadway, excluding the banks or ditches on each side. Normally, this includes the paved area outside of the fog lines. The default width is 4 meters or 13 feet.

Long. Resolution

The longitudinal resolution of the 3D road object is entered here. A larger resolution than the default value will result in a slightly lower redraw time, but will increase the elevation error at points where the slope changes. The default resolution is 1 meter or 3 feet.

Road Radius

This drop down list allows the road radius to be specified as a diagram (see the Diagram tab in the 3D Road Object dialogue box), as a straight road (default value), or by using "follow points". When follow points are selected, the road centerline will follow a path drawn using **Dynamics – Define Path Points**.

Import

This button allows 3D road objects, in the form of a road definition file (*.txt) or a 3D drawing (*.dxf), to be imported.

The easiest way to construct a 3D road definition file is to first export a 3D road created in PC-Crash into a spreadsheet program such as Excel™. Then, modify it by changing the values to the desired ones and save the file. It can then be imported back into PC-Crash to have a 3D road section with the new values.

3D DXF scene drawings must have surfaces applied before being imported. This can be done in one of two ways:

1. Use a 3D drawing program such as AutoCAD® to apply the surfaces in the form of triangular polygons between 3D points.
2. Starting with a New Project in PC-Crash, import 3D scene points from a total station DXF file using File – Import – DXF Drawing or use the Drawing Editor to define 3D points. Then, select the Drawing Editor (Drawing – Draw Toolbar or). Use the tool to select the drawing components that are to be made into a 3D drawing. Right-click in the Drawing toolbox title bar and, in the menu that appears, select Triangulate selected. Triangular polygons will be applied between the 3D points to make a complete 3D surface. A massage box will appear asking if the triangulated area should be used as slope polygon.

If you choose Yes the triangulated area will be used as slope polygon and can be assessed using **Dynamics – Define Road Slope**.

Export

This button allows 3D road slopes, in the form of a road definition file (*.txt) or a 3D drawing (*.dxf), to be exported.

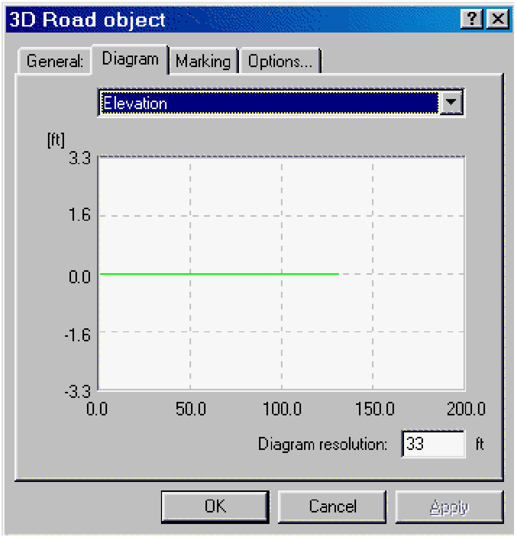
Ditches

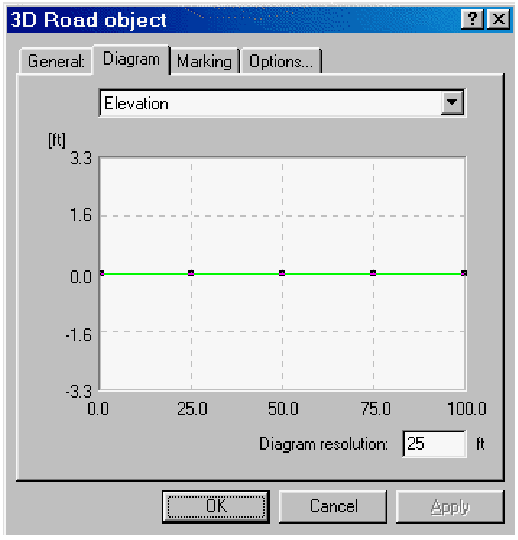
Banks on each side of the road that have constant slopes or a ditch profile can be selected with ☑ Left and ☑ Right. When ☑ Use Ditch Profile is also selected, elevations at 1 meter (3.3 foot) increments across the bank from the road can be specified in the Diagrams tab.

When Use Ditch Profile is not selected, a constant slope of the banks on each side of the road can be specified in degrees or percent grade. The height and width shown in the text boxes to the right of the slope text boxes are for information only, to give the grade. The horizontal width across the bank is always the lesser of 20 meters (65.5 feet) or that which is necessary to reach elevation zero along a negative slope (downward from the road), and 10 meters for a positive slope (upward from the road).

Diagram

The Diagram tab allows the user to specify various parameters at even intervals along the length of the 3D road object. These intervals are specified in the Diagram Resolution text box. The default diagram parameter is **Elevation** and the default resolution is 10 meters or 33 feet. After changing the resolution to the desired one, use the TAB key to exit this text box and the new value will be accepted on the diagram and the diagram will be scaled to the road length. Click on the diagram line and the intervals will show on it, as seen in the following figure.

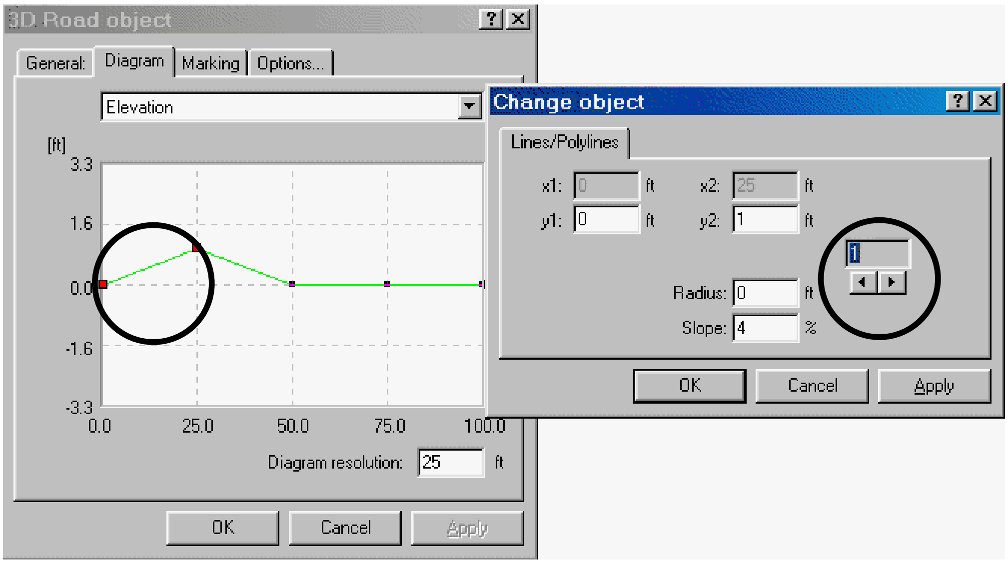




The next step is to input the elevations at each interval. This can be done in two ways:

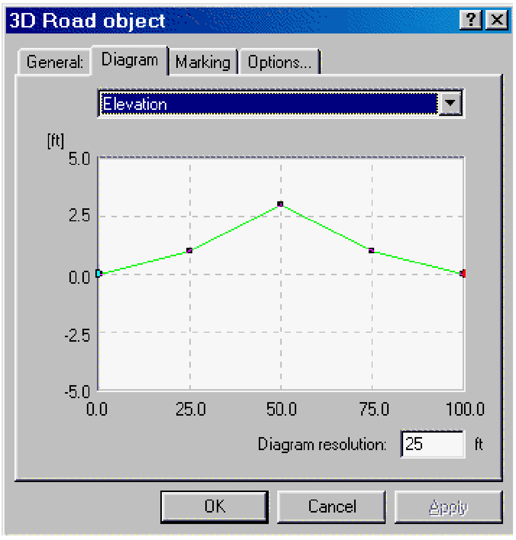
1. Click on the line and drag it up or down to the desired elevation at each point. The vertical axis of the diagram will change to a larger scale, if necessary.

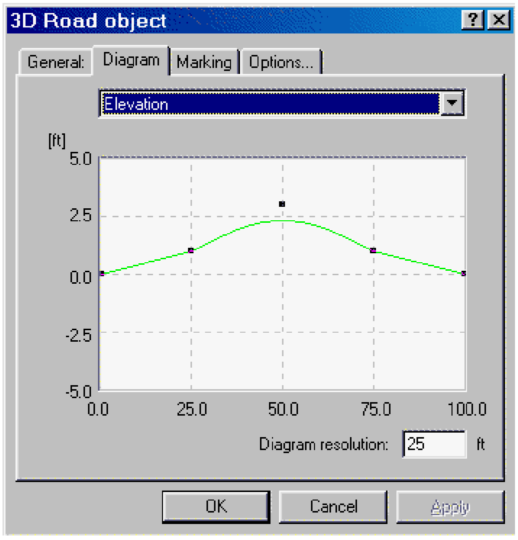
2. Double-click on the line and enter the values in the Change Object dialog box that appears. If this box is moved away from over the diagram, the section of the road that is being modified can be seen with red endpoints. The section number is also shown at the right of the Change Object dialog box.



The y1 and y2 text boxes are for specifying the y-axis value (in this case the elevation) at each end of the highlighted section. After changing these values for one section, use the arrow buttons below the section number box to move to the next section.

After entering the y values for all the sections, they can be seen on the diagram (following figure, left side). Depending on the elevations chosen, there may be an unrealistic slope change between each section. This can be smoothed using the Radius text box in the Change object dialog box, to apply a radiused transition from one section to another. For example, the right figure below shows a road profile that has been changed from the one in the left figure by specifying a radius of 200 feet for the second section (the specified radius is for the right end of the selected section). Note that the radius causes the road elevation to be different than the specified elevation, so you may have to readjust the specified elevation to account for this. The specified radius must not be so large that it would extend past the far ends of the two adjoining sections.

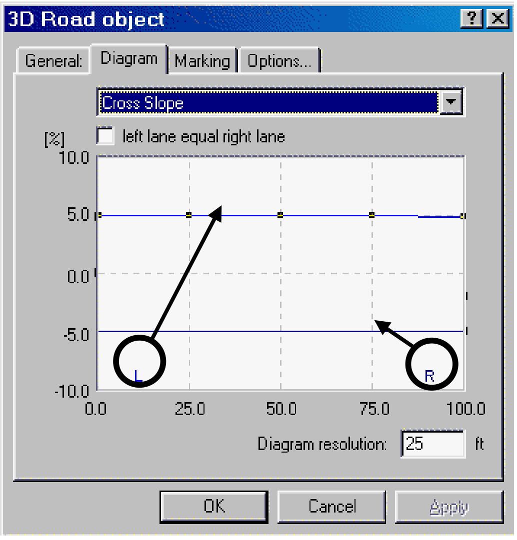


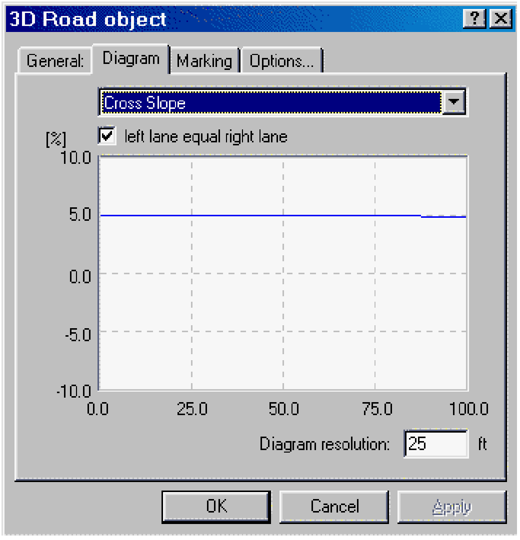


In addition to Elevation, parameters that can be varied at each interval along the road are:

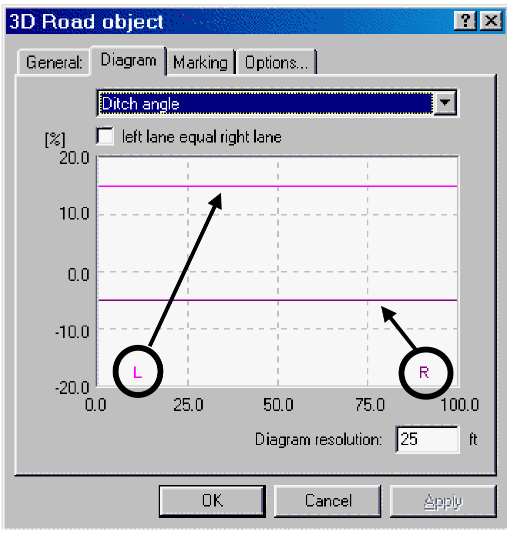
- **Cross slope** – The cross slope or super-elevation of each side of the road surface can be specified at intervals along the road axis. A positive grade on either side means the road slopes up from the center. Thus, if the cross slope is the same on both sides of the road, it will be a crowned or a dished road. If the cross slope is negative on one side of the road and positive on the other (as in the following figure, left side), it will be a banked road. Note that the two lines on this figure (one for the left side of the road and one for the right side) are overlapping at the value 0 when this window is first opened.

If the road is banked with the same cross slope on both sides, an easier way of specifying this is to select ☑ Left Lane Equals Right Lane. This applies the specified left lane cross slope across the whole road (shown in the following figure, right side).

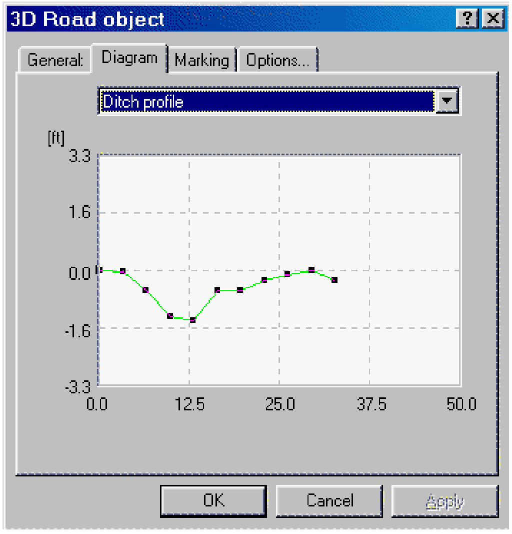




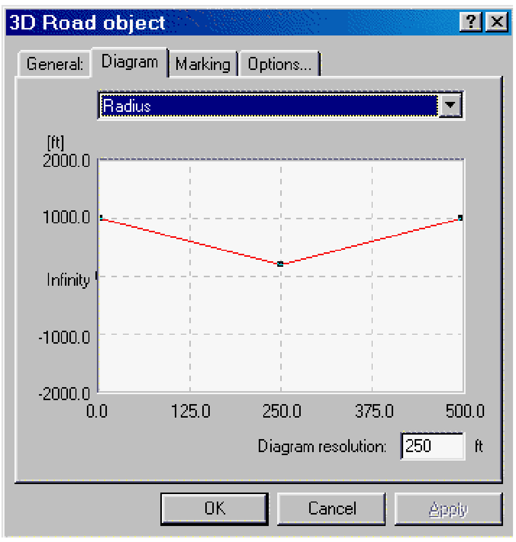
- **Ditch Angle** - If ☑ Left and/or ☑ Right Ditch in the General tab have been activated, bank slopes can be varied along the axis of the road. The horizontal width across the bank is always the lesser of 20 meters (65.5 feet) or that which is necessary to reach elevation zero along a negative slope (downward from the road), and 10 meters for a positive slope (upward from the road). As in Cross Slope, ☑ Left Lane Equals Right Lane can be selected for adjusting the slopes of both side banks to the same value. The following shows the Ditch Angle diagram with the values left as the constant ones specified in the General tab.

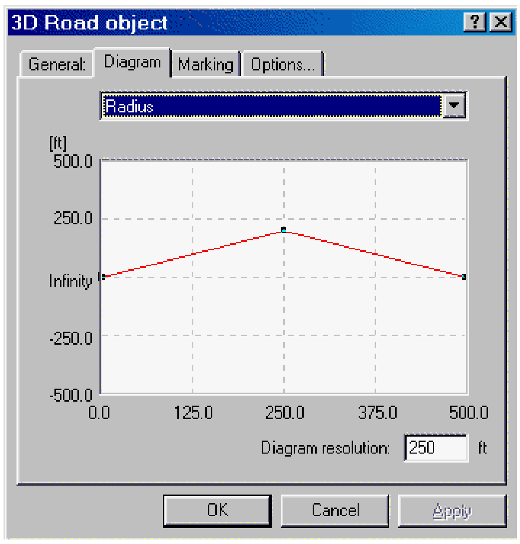


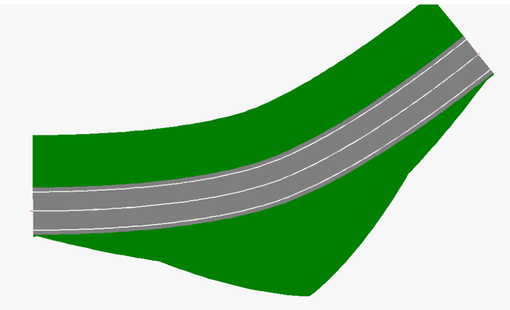
- **Ditch Profile** – If ☑ Use Ditch Profile in the General tab has been selected, road bank elevations at 1 meter (3.3 foot) increments across the bank from the road edge can be specified. The two sides of the road cannot be varied independently, and the specified ditch profile is continuous (similar to an extrusion) along the whole length of the 3D road object.

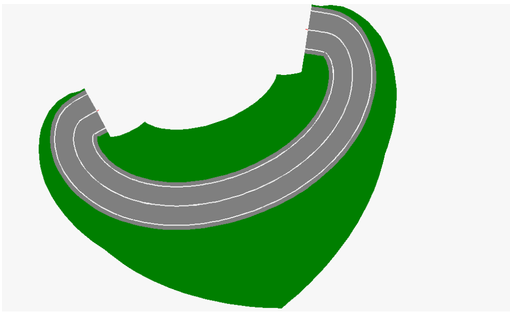


- **Radius** - The radius of the road (at the centerline) can be specified. A specified radius of -10 to +10 meters (-33 to +33 feet) will be taken as "Infinity", which means the road is straight. When the user wishes to have a smooth transition from a straight road to a curve, it is better to use a large radius for the "straight" portion. This avoids having a sharp radius at the point when the road goes from a "0" radius (straight section) through a 10 meter radius (a very sharp turn) and then to the specified radius. In the figures below, the left ones represent the recommended method of having transitions into and out of a 200 foot radius curve, while the right one was done leaving the ends of the road at radius "0".

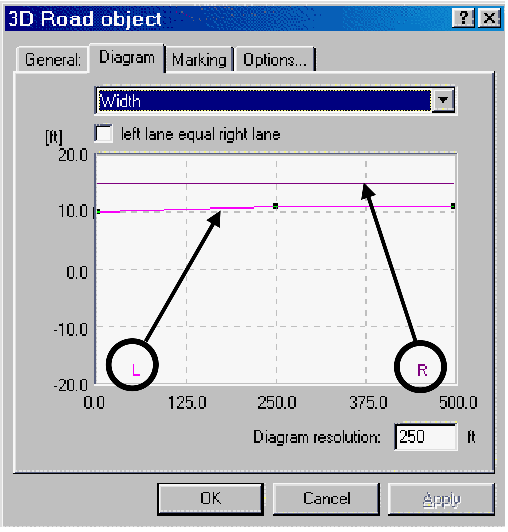






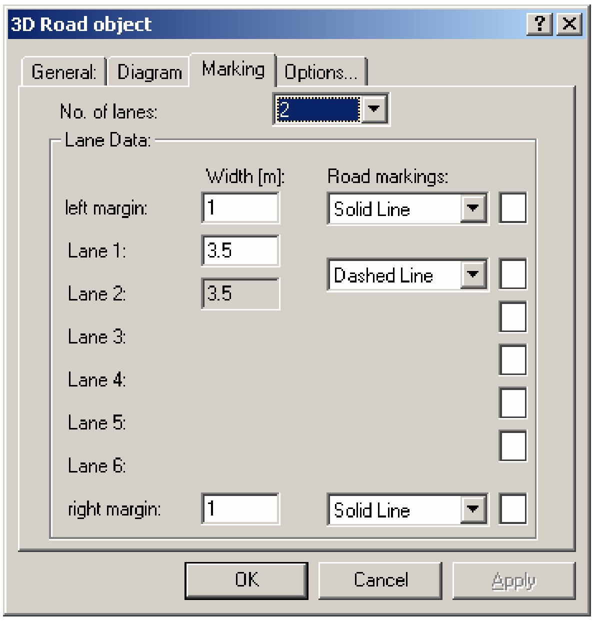


- **Width** - The width of the road can be varied from point to point. There are two lines on this diagram, so that the width of each side of the road can be varied independently. As in Cross Slope, the lines overlap when the window is first opened, and Left Lane Equals Right Lane can be selected for adjusting the widths of both sides to the same value.



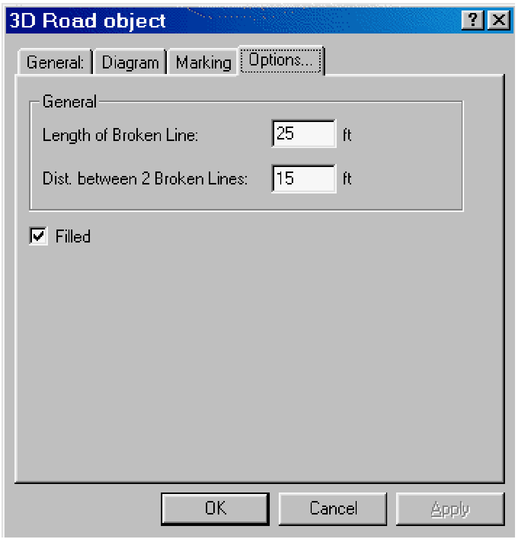
Marking

The Marking tab allows the user to specify the number of lanes (1 to 6), the lane and margin (paved shoulder) widths, and the line type between the lanes. The width of the last lane is calculated automatically, based on the width of the road, the other lanes and the margins. The color of the road markings can be defined using the color buttons on the right side.



Options

In the Options tab the dashed line spacing can be chosen.  Filled defines how the 3D road object will appear (filled or just outlined) on the main screen.



Note: *To modify an existing 3D Road Slope after closing the 3D Road Object dialog box, do not click on the*  *button again. This will create a second 3D road object. Instead, double-click on the road with the*  *button activated, which will open the 3D Road Object dialog box for the current 3D road object.*

If a second 3D road object is created, it is possible to join it to the first one. Simply click on the second one and drag its end to that of the first one. The angle and elevation of the second 3D road object will change to match the end of the first one.

Move/Rotate Vehicle 

This tool is for moving and rotating vehicles, to place them at the desired start position.

Position the tow truck cursor's hook at the vehicle's center of gravity and move the vehicle to the desired position with the left mouse button pressed. If the hook is positioned 1.5 to 5 meters (5 to 16.5 feet) from the center of gravity, the vehicle can be rotated.

Trailers: If a trailer is connected to the vehicle, it will move and rotate with the tow vehicle when this tool is used on the tow vehicle. Relative rotation of the trailer with respect to the tow vehicle is accomplished by using this tool on the trailer.

Note: *A vehicle can only be moved if it is in its start position (time =0).*

UDS

This Menu is for interaction with UDS vehicle data collection device. UDS is presently not widely available in North America.

Impact

This menu is for performing crash simulations.

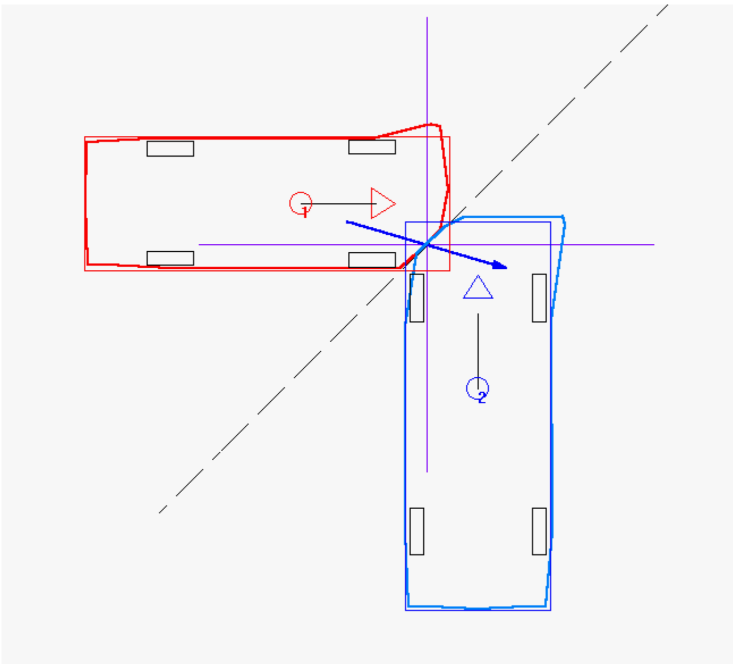
Crash Simulation (F8)

The Crash Simulation dialog box is used to define and change crash parameters.

PC-Crash's crash simulation is based on the impulse-restitution crash model by Kudlich-Slibar (extended for 3D cases). The theory behind this crash model is described in the PC-Crash **Technical Manual**. The Kudlich-Slibar crash model is characterized by the definition of the point of impact. The point of impact is the point where the crash force is assumed to be exchanged. The elasticity of the collision is considered, based on a coefficient of restitution. This model also considers the sliding of one vehicle along another vehicle or a fixed object, based on a contact plane angle and friction coefficient. The user can change all of these collision parameters.

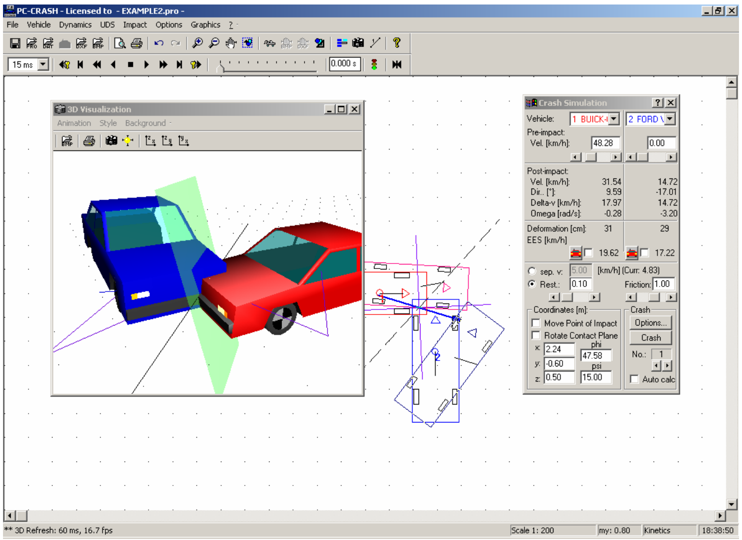
The driving conditions for both vehicles are specified before the crash. Then, from conservation of linear and angular momentum and the use of the Newtonian Crash Hypothesis, the post-impact conditions are determined.

Once a crash is specified, the vehicle positions at the crash and 200msec after the crash are shown on the main screen of PC-Crash. This allows the user to immediately see the influence of different pre-impact speeds and other parameters.



Also shown on the main screen are the impact impulse vector (heavy blue line), the contact plane (dotted line) and the friction cone (purple lines).

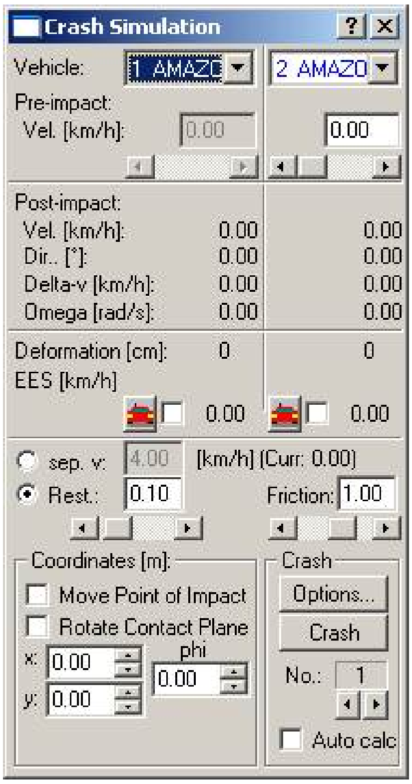
The Crash force vector, the friction cone and the contact plane can be visualized within the 3D Visualization window (**Options – 3D Window**) by activating Crashes within the Options – Display settings window (**Options – Options – Display settings**).



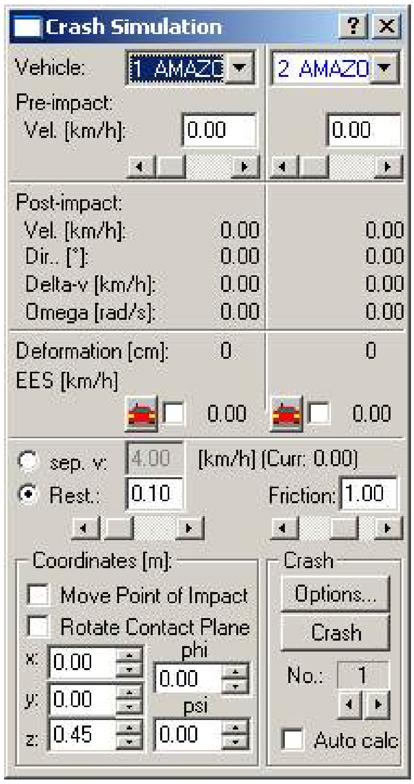
In addition to the pre-impact conditions for each vehicle, the following impact parameters can be altered:

- Position of the point of impact
- Impact elasticity, defined by either coefficient of restitution or separation velocity
- Orientation of the contact plane
- The inter-vehicle contact friction coefficient along the contact plane.

PC-Crash uses a 2D crash model when one or both of the vehicles in the crash have a center of gravity height of 0 (as defined in **Vehicle – Vehicle settings – Vehicle Geometry**). Only when a center of gravity > 0 is defined for both vehicles is the 3D model used. The 3D model Crash Simulation dialog box has two additional parameters: the impact height and the contact plane angle in the vertical plane.



2D Model dialog box



3D Model dialog box

Crash Simulation includes the following:

Vehicle:

These two drop down lists are for defining which vehicles are involved in each crash.

Pre-Impact:
Vel.:

The velocities of the two vehicles specified in the Position & Velocity dialog box are adopted as values for the pre-impact phase. In order to simplify making variations in pre-impact speed, however, they can also be modified in this dialog box by typing them in or adjusting them with the help of a scroll bar. The post-impact parameters are automatically updated.

Note: *If the vehicles are not in their starting position the values for the pre-impact velocity in the Crash Simulation dialog box cannot be changed.*

Post-Impact:

Below this heading the output values for the post-impact phase are shown, as follows:

- **Vel.** is the speed of the two vehicles.
- **Dir.** is the velocity vector direction of the two vehicles.
- **Delta-v** is the impact velocity change
- **Omega** is the rotational yaw velocity of the two vehicles.

Deformation

As a control, the deformation depth of each vehicle, based on the defined positions of the vehicles and point of impact, is indicated. The deformation depth of each vehicle, calculated in the direction of the crash force vector, is the distance from the point of impact to the outside of the undeformed rectangular vehicle outline.

EES

As a control, the Equivalent Energy Speed (EES) of the impact is indicated for each vehicle. EES is similar to EBS, or Equivalent Barrier Speed, which is a term commonly used in North America. The difference between the two is that EES also considers the rebound velocity of the impact. Thus, the EES can be quite different from the EBS in low speed impacts with higher coefficients of restitution. For impacts with an EBS above about 10 mph, however, the difference between EES and EBS is generally insignificant.

The distribution of the deformation energy between two vehicles in collision depends on the vehicle masses and the deformation depths:

$$\frac{EES_1}{EES_2} = \sqrt{\frac{m_2}{m_1}\frac{s_{Def1}}{s_{Def2}}}$$

$$EES_2 = \sqrt{\frac{2E_D}{m_2\left(\frac{s_{Def1}}{s_{Def2}} + 1\right)}}$$ where

m_1, m_2 = Mass of each vehicle

s_{Def1}, s_{Def2} = Crush depth of each vehicle, outer surface to impact point, in line with impact force.

E_D = Energy lost by both vehicles in collision due to damage.

Additionally, one of the two EES check boxes can be activated. In this case, the EES value can be defined for the corresponding vehicle. The EES value for the second vehicle will be calculated.

The calculation of the second value follows the following formula:

$$EES_2 = \sqrt{\frac{E_D - m_1 EES_1^2}{m_2}}$$

If both EES check boxes are de-activated, both EES values are calculated.

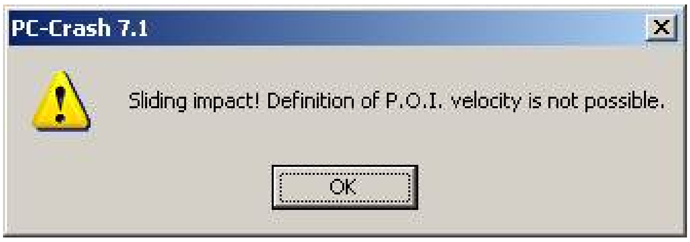
The button beside the EES text box accesses the EES catalog. See the description under the menu item Vehicle – EES Catalog.

Sep. v

As an alternative to defining the impact coefficient of restitution (see next item), the separation velocity of the contacting areas of the two vehicles can be defined. When this option button is selected and a separation speed is entered, the resulting impact coefficient of restitution is automatically calculated.

The post-impact separation velocity is usually quite constant, regardless of most crash parameters. This is due to the fact that elastic forces are quite constant and there is only a small dependence on the extent of deformation. Investigations indicate the post-impact separation velocity averages about 3 – 4 mph (5 - 7 km/h).

Defining the impact elasticity with a post-impact separation velocity can only be done when a sliding impact can be ruled out. When calculating a sliding impact a warning appears, informing you that you cannot define a post-impact separation velocity. The coefficient of restitution shown will be used for the simulation, instead.



If a post-impact separation velocity is defined that calls for a coefficient of restitution > 1, the coefficient of restitution is set to 1 and the corresponding separation velocity is calculated.

Rest.

The coefficient of restitution is a value normally between 0 and 1 (PC-Crash default = 0.10) which defines the elasticity of the impact following the Newtonian Crash Hypothesis. It can be varied with the scroll bar or by typing the new value in the text box. The separation velocity is calculated automatically.

A negative coefficient of restitution can also be specified, to cover impacts where one vehicle tears through a portion of another vehicle and no common velocity is reached. This is useful for offset frontal impacts where the vehicles continue past one another after tearing off each others' left front wheels or fenders, and in lateral curb impacts where the tire and wheel bend under, allowing the vehicle to continue past the curb. Close attention must be paid to the speed change and deformation energy when specifying negative coefficients of restitution to ensure realistic values are used.

Friction

This is the inter-vehicle friction coefficient along the contact plane. As long as the impact impulse vector lies within the friction cone shown on the main screen of PC-Crash, there will be no sliding of one vehicle along the other. On the other hand, if the impulse vector is in line with the outside of the friction cone, the specified inter-vehicle friction has likely limited the impulse angle. If so, inter-vehicle sliding takes place, and the contact friction coefficient describes the relation between the normal and tangential components of the impulse vector.

Coordinates

This area contains two check boxes and three to five text boxes, as follows:

- ☑ **Move Point of Impact** - When this is activated, the point of impact (where the average exchange of forces between the two vehicles takes place) can be placed. This is done by moving the mouse cursor on the main screen within the area of vehicle overlap with the left button depressed. Alternatively, clicking the left mouse button with the cursor outside of the area of vehicle overlap will place the point of impact in the center of the area of overlap <u>and</u> rotate the contact plane to an angle consistent with the overlapped position of the vehicles. The point of impact can also be placed by entering the desired coordinates in the applicable text boxes below.
- ☑ **Rotate contact plane -** After positioning the point of impact the contact plane and friction cone are depicted. On activation of ☑ Rotate Contact Plane, a new contact plane angle (phi) can be defined by holding the left mouse button and moving the cursor on the main screen. The contact plane angle can also be entered in the phi text box.

x, y, z - Text boxes for the input and output of the impact coordinates. The **z** coordinate, which is the height above the current road plane normal to its surface in global coordinates (i.e. feet when in U.S. units), is only visible for 3D simulations. The values can be changed also with the spin buttons located inside the edit boxes. The longer a button is pressed down the faster the value will change. Spin buttons can be controlled with the keyboard up/down direction keys too.

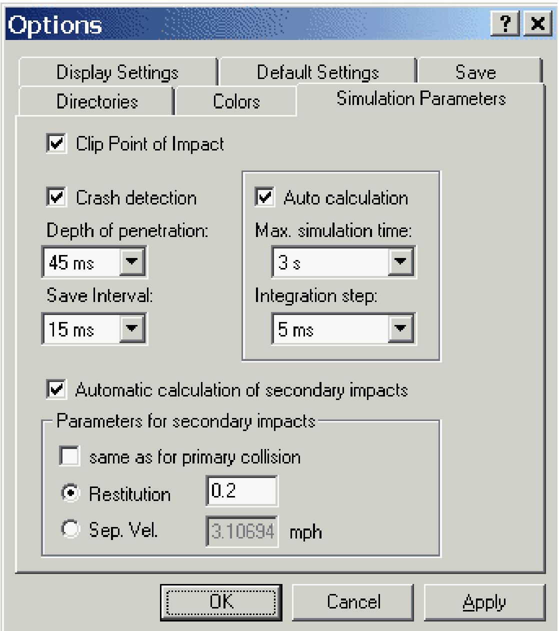
- **phi** - Angle of the contact plane between the vehicles (degrees CCW from the global X-axis).
- **psi** - Angle which defines the rotation of the contact plane with respect to the vertical plane (3D simulations only). An angle of 0° means the contact plane is vertical. This angle can be viewed in the 3D window with ☑ Crashes activated in **Display Settings** .

Crash

This area contains two buttons, a scroll bar and a check box, as follows:

Options

This button accesses the Simulation Parameters dialog box, which can also be accessed in the **Options** menu. Please refer to the menu item **Options – Options – Simulation Parameters** for a description of the items in this dialog box.



Crash

This button performs the impact calculation. It is used to specify a crash in a new simulation when no changes have been made to any values in the Crash Simulation dialog box (impacts are calculated automatically without use of the Crash button whenever a change is made to any value in the Crash Simulation dialog box).

Crash No.

This shows which crash the parameters are currently being set for. By changing this value, parameters of all crashes that have occurred can be accessed. An unlimited number of crashes between up to 32 vehicles can be performed. These crashes are saved until a New Simulation is specified.

Using the scroll arrows automatically moves the vehicles between the different impacts, changing the crash numbers accordingly. When shifting to the last number the vehicles will be shifted to the last simulation position and a further impact calculation including post-impact movement can be appended, if desired.

Auto calc.

When activated, the simulation will run to the vehicle rest positions, or for a pre-defined time period, whenever a change is made in the Crash Simulation window. The pre-defined time period can be changed using Max. Simulation Time in **Options – Options - Simulation parameters**.

The vehicles' positions at rest or at the pre-defined time interval are shown on the main screen instead of the positions 200 msec after impact, based on all defined sequences and road conditions.

Crash Detection

This menu option is used to activate an automatic crash detection feature. This feature continuously checks for contact between the 2D vehicle rectangular outlines. If two vehicles touch each other, the simulation will perform an impact calculation automatically after the selected Depth of Penetration time is reached, if Automatic Calculation of Secondary Impacts is activated. If not, the simulation will stop at this time, allowing the user to specify the crash parameters manually. Both of these items are located in **Options – Options** - **Simulation Parameters.** Crash Detection can also be activated in **Simulation Parameters** using Crash Detection.

If two vehicles are less than a pre-defined minimum distance apart, the program automatically runs at the smallest time step to allow a more accurate timing of the automatically detected crash.

For secondary impacts between two vehicles after a primary impact (one at time = 0), impact parameters can be different than those of the primary impact, as specified in **Simulation Parameters**.

For secondary impacts when vehicles are already overlapping, **Crash Detection** checks if the area of overlap is still increasing after the Depth of Penetration time from the previous impact elapses. If it has, another impact will be calculated.

When not using **Crash Detection**, secondary impact values can be defined manually in the Crash Simulation dialog box. In this case, drive the vehicles to the secondary impact position using the Simulate Forwards button , rather than the Simulate Forwards to Stop Position button .

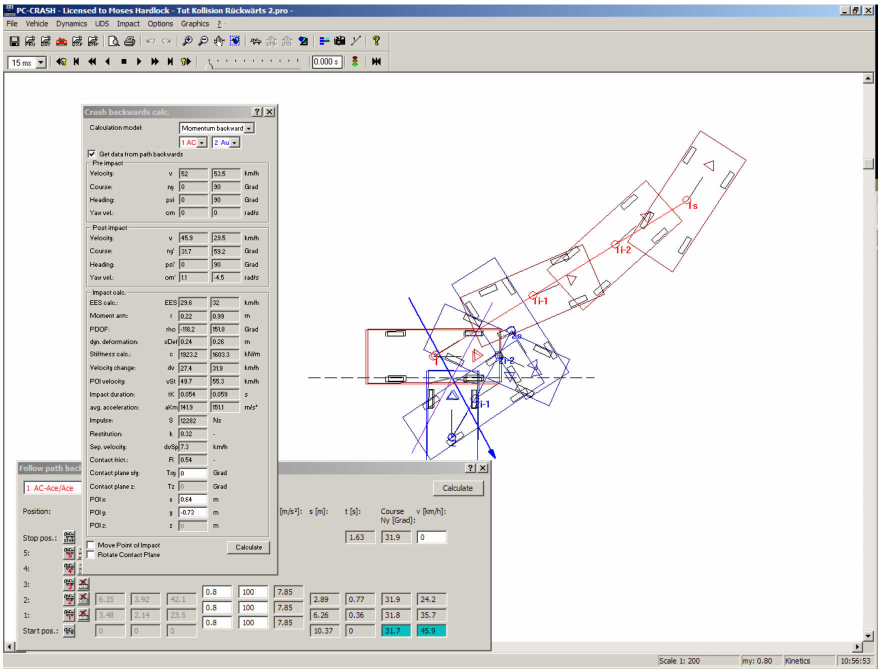
Crash Detection within Truck/Trailer combinations

This is similar to **Crash Detection**, but is for selecting whether or not impacts between a tow vehicle and its trailer(s) are detected, due to jack-knifing.

Crash backwards simulation…

The backwards simulation of an impact can be used to determine the pre-impact situation if the post-impact situation is known (e.g. by making a post-impact analysis by using tire marks described in chapter *Kinematic follow path backwards…* on page 154).

By selecting the menu command **<Impact> <Crash backwards simulation>** a window appears, listing all impact relevant parameters.



The combo box **Calculation model** allows switching between two calculation models:

- **Momentum backwards** uses the momentum and impulse of the post-impact situation to determine the pre-impact situation.
- **EES backwards** uses the deformation energy of the collision to determine the pre-impact situation.

If the post-impact situation has been determined using the kinematic follow path backwards menu command, the option **Get data from path backwards** inserts all post-impact data automatically in the appropriate input fields.

Some of the impact parameters listed here are described in more detail in chapter *Crash Simulation (f8)* on page 168.

Pre impact

v The pre-impact velocity.

ny The course angle of the vehicle before the impact. If the calculation model is set to **EES backwards**, the course angle of the first vehicle has to be set.

psi The heading of the vehicle before the impact.

om The angular velocity of the vehicle before the impact. This value is assumed to be 0.

Post impact

v The post-impact velocity can be automatically defined by the option **Get data from path backwards** or can be defined manually.

ny' The post-impact course angle can be automatically defined by the option **Get data from path backwards** or can be defined manually.

psi' The post-impact vehicle heading can be automatically defined by the option **Get data from path backwards** or can be defined manually.

om' The post-impact angular velocity can be automatically defined by the option **Get data from path backwards** or can be defined manually.

Impact calc.

EES The calculated EES value of the deformation. If the calculation model is set to **EES backwards**, this value can be specified.

r The length of the momentum arm is determined by the position of the point of impact.

rho The angular direction of the momentum arm.

sDef The total dynamic deformation distance during intrusion is determined by the collision positions of the collision partners and the position of the point of impact.

c The calculated structural stiffness.

dv The change of velocity caused by the impact.

vSt The velocity of the vehicle at the point of impact.

tK The total duration of the impact

aKm The average acceleration of the vehicle caused by the impact (during **tK**).

S The total impulse of the impact

r The coefficient of restitution determines the elasticity of the impact. If the calculation model is set to EES backwards, the coefficient of restitution can be specified.

dvSp The separation velocity

R The contact friction between the vehicles that has been calculated.

Txy The angular direction of the contact plane. This value can also be specified by selecting the option **Rotate Contact Plane** and using the mouse to rotate the plane.

Tz The slope angle of the contact plane. This value can only be specified if both impact partners have specified the height of the center of gravity.

x, y, z The coordinates of the point of impact. Instead of entering the x and y coordinates, the point of impact can also be specified by selecting the option **Move Point of Impact** and using the mouse to move the point. The z coordinate can only be specified if both impact partners have specified the height of the center of gravity.

By pressing the button **Calculate**, all values in the window are updated.

Stiffness based impact model

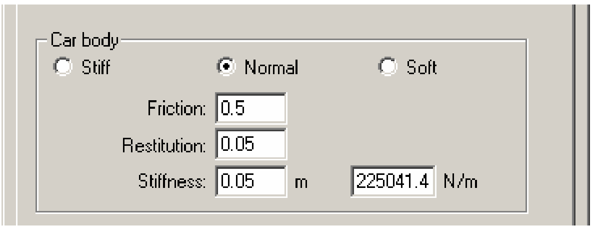
For the analysis of impacts a simple and reliable impact model was developed, which consider vehicle stiffness and resolve the crash in time. Some general assumptions regarding vehicle stiffness were made, in this way a good agreement of the model for several collision situations could be found, on the other hand CPU time could be limited to a minimum.

Motor vehicles are generally regarded as rigid bodies for the simulation of traffic accident collisions with kinetic 3D simulation programs. This simplification is admissible for the most simulations. However, for calculation of contact forces between the vehicle body and the ground or other objects several contact ellipsoids are used to represent the vehicle shape. The geometry of these ellipsoids is based on the vehicle type and values specified. The contact model used to calculate the contact force due to a certain amount of penetration between two bodies or a body and another object are based on a linear stiffness function. A coefficient of restitution can be specified to define the amount of elasticity during the contact. Once the contact normal force is calculated friction forces can be calculated using the specified contact friction between the two bodies. Therefore sliding impacts, partially sliding and full impact situations can be calculated. The calculations of the amount of penetration, the location of the point of impact and the orientation of the contact plane are performed automatically. There is no need of user interaction for the calculation of these impacts once the general body parameters are specified.

Note that the linear stiffness function can be replaced by specifying force/distance characteristics for a vehicle from the stiffness database. For further information on that topic see chapter **Stiffness Database** on page 135.

The vehicle shape specified under **Vehicle – Vehicle settings – Vehicle shape** is used to determine the vehicle ellipsoid sizes for the stiffness based impact/rollover model. These specifications can only be done for Automobiles. Refer to the Technical Manual (Rollover Model) for more details on specifying the contact ellipsoids.

The crash is calculated using the values specified within the Car body section (**Vehicle – Vehicle settings – Suspension Properties**).



Properties for stiffness based impacts can be specified in the Car Body field, as follows:

- **Friction** – This is the coefficient of friction for the vehicle body. The tire to ground friction is not specified here – it is the lesser of 0.8 or as specified elsewhere: for the general scene, in Friction Polygons, or in the Sequence window.
- **Restitution** – This is the coefficient of restitution for vehicle impacts. If the stiffness based impact model is used, only one restitution for all concerned vehicles should be used. If different values are specified the lowest value is used for the calculation.
- **Stiffness** – The "stiffness" is expressed as a deformation distance, the corresponding stiffness is calculated automatically. The stiffness is defined by the deformation of the car body due to its static weight - the user specifies the deformation and the stiffness is calculated. The value of stiffness used for the tire ellipsoids is half of the value specified for the vehicle body. For automobiles, the specified stiffness is for the lower part of the vehicle body only. For the roof, one quarter of the specified stiffness is used. For other vehicles, the specified stiffness is for all body ellipsoids. In addition the value for the stiffness can be specified (linear relationship between load and deformation), the corresponding deformation distance is calculated automatically.

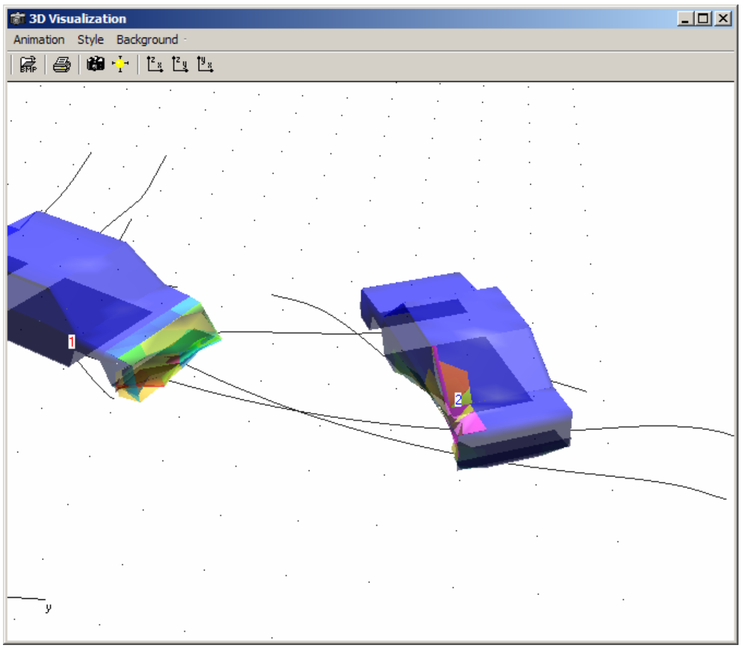
Vehicle speed changes and accelerations that occur as a result of stiffness based impacts can be viewed in graphical form in the Diagrams window, using the menu option **Options - Diagrams – Diagrams – Vehicles** or **Options - Diagrams – Diagrams – Sensor Signals**. Refer to the description of this menu option later in this chapter.

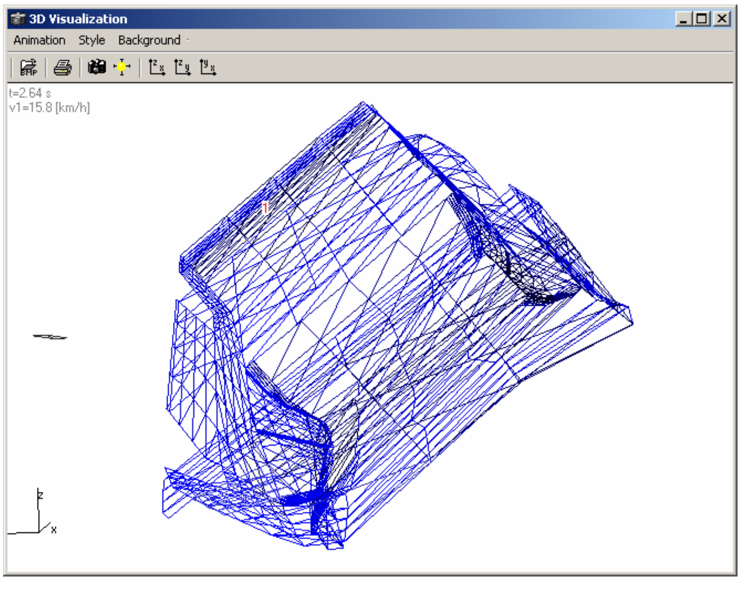
Use mesh based impact model

The mesh based impact model tessellates the vehicle body into small polygons. This mesh has the initial form of the car body. Upon contact with the ground surface, slope polygons or other vehicles the mesh is deformed, reducing the total energy of the movement thus simulating kinetic energy conversion into deformation energy.

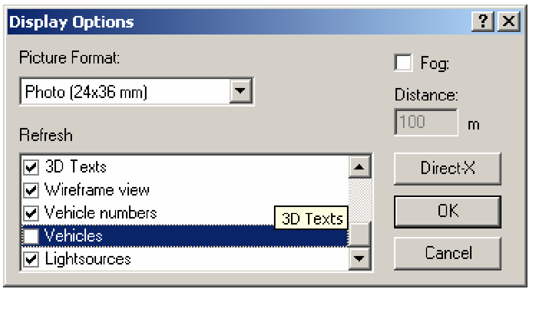
The main difference to the stiffness based impact model (see chapter *Stiffness based impact model* on page 176) is that the vehicle's structural deformation (including restitution) is considered by determining the stiffness function. Already deformed vehicle parts are stiffer than not deformed parts. This is especially important when multiple impacts occur because deformation work is not applied twice to the vehicle.

The deformation results can be examined in the 3D window. The color of the mesh displays the amount of deformation.





Best results are achieved if turning on wire frame view and deactivating the display of vehicles by selecting the menu option **<Style> <Display Options…>**.



Rest Positions

This enables vehicle rest positions to be defined for use with the Collision Optimizer. Initially the rest and start positions of each vehicle are identical, so the user has to move each vehicle from the start position to the rest position, after selecting this option the first time. The cursor changes to a tow truck (of slightly different appearance than the tow truck cursor used for placing the vehicles at time=0), with which the vehicle rest positions can be defined. The initial vehicle positions remain unchanged.

Note: *If the rest positions are not shown or cannot be selected, check that* *Optimizer Positions in* ***Display Settings*** is checked**.**

Intermediate Positions

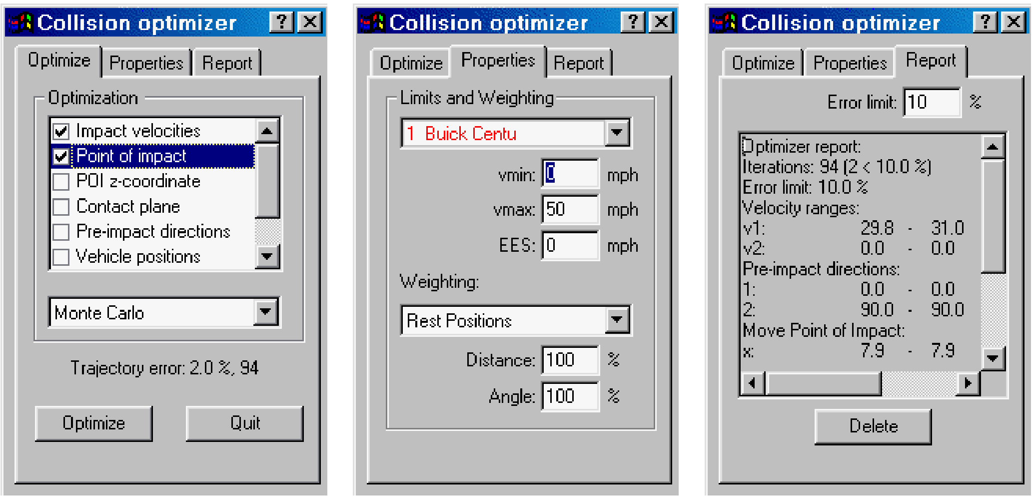
This is similar to **Rest Positions**, but is for selecting a known intermediate position along the vehicle's post-impact path.

Five intermediate positions (Intermediate Positions 1 to 5) can be defined.

Collision Optimizer

The Collision Optimizer controls the automatic optimization of pre-impact velocities and other parameters. Refer to the **Technical Manual** for the theory this model is based on. The aim of the Collision Optimizer is to determine the pre-impact speeds and other parameters, based on the defined rest and/or intermediate positions.

After activating this menu item the following dialog box appears, with Optimize, Properties and Report folders:



Optimize

This dialog box is for defining the strategy of the optimizing process. The following three algorithms are currently available:

- **Genetic** (default)
- **Linear** (applicable only to impact velocity and point of impact)
- **Monte Carlo**

The parameters that can be varied during the optimization process are specified in the Optimize list box. These parameters are optimized in the order that they appear in the list box, except that the point of impact (POI in the x-y plane) is optimized interactively with the POI z-coordinate when optimizing with both of these items checked.

For the Linear and Genetic algorithms, each parameter is optimized in two phases, starting with the higher step width, before the next parameter is optimized, in the following order:

☑ **Impact velocity**

5 km/h, 1 km/h (3 mph, 0.5 mph in US units)

☑ **Point of impact** – POI in the x-y plane

5 cm, 1 cm (1", 0.4")

☑ **POI z-coordinate** – vertical height of the point of impact (for 3D simulations only)

5 cm, 1 cm (1", 0.4")

☑ **Contact plane** - angle phi (see Crash dialog box), degrees CCW from the global x-axis

2.5 deg, 0.25 deg

☑ **Pre-impact directions** – vehicle pre-impact heading angles are varied about the point of impact

2 deg, 0.5 deg

☑ **Vehicle positions**

5 cm, 1cm (1", 0.4")

☑ **Restitution**

0.05, 0.01

☑ **Contact friction**

0.05, 0.01

The Genetic algorithm varies both of the parameters being optimized (such as Velocity 1 and Velocity 2) in an interactive manner. For example, if starting with a speed of 10 km/h for each vehicle, it will initially vary the speeds as follows:

Velocity 1	Velocity 2
10	10
15	10
15	15
10	15
5	15
5	10
5	5
10	5
15	5

If 15, 15 produces the lowest error, it will continue at the larger (5 km/h or 3 mph) step to higher values. Once it starts to achieve lower errors at values less than the highest value, the optimization step will decrease to 1 km/h or 0.5 mph.

The Linear algorithm varies the velocity and the POI differently. For the velocity, it changes Velocity 1 up and down in large steps (5 km/h or 3 mph) until the error decreases no more, then does the same for Velocity 2. It then repeats this at the smaller step. For the POI, the Linear algorithm scans an area of +/-25cm (step width 5cm) and then an area of +/-5cm (step width 1cm). It is basically a raster scan of every point at 5cm intervals in a square 50cm by 50cm, then every point at 1cm intervals in a square 10cm by 10cm, originating at the 5cm interval point with the lowest error in the square 50cm by 50cm. The Linear algorithm does not vary the POI z-coordinate so should not be used for 3D simulations where this parameter is to be optimized on. Also, since the Linear algorithm is applicable only to impact velocity and point of impact, the Genetic algorithm will be used for the other parameters regardless of whether Linear or Genetic is selected.

For the Monte Carlo algorithm, 100 iterations (calculations) are performed, using random values for the selected optimization parameters. This avoids situations where the other algorithms could stop at a local minimum. It also gives a good indication of the expected ranges of velocities and other parameters. It is generally best to assign Vmin and Vmax values in the Properties dialog box when optimizing on Velocity, otherwise random values anywhere between 0 and 300 km/h (186 mph) will be used.

The ranges of the other Monte Carlo random values are:

- **POI in x-y direction** –+/- 0.5m (1.6 feet)
- **POI in z direction** - +/-0.25m (0.8 feet).
- **Contact plane angle** - +/-15°
- **Pre-impact directions** - +/-15°
- **Vehicle positions** - +/-0.1m (0.3 feet)
- **Coefficient of restitution** - +/-0.25
- **Contact plane friction** - +/-0.25.

The Optimizer does not optimize on parameters such as braking, steering or pre-impact rotation. The user must specify these values, and, if necessary, vary them manually.

To start the optimization, press the Optimize button. The optimization process can be stopped at any time by a single click on the right mouse button or by pressing the ESC key.

The better the start conditions are, the faster the optimization process will find an optimized solution. The optimization process can be repeated several times and the program will always keep the best solution found as a result.

The current and lowest weighted total error is displayed just above the Optimize button during optimization. The number of optimization trials that have been completed is also displayed in this area.

Closes the Optimize dialog box.

Properties

This dialog box enables the following values to be defined for each vehicle for the optimization process:

- Velocity range limits (default = 0 to 300 km/h or 186 mph);
- EES;
- Weighting of distance and angular errors for the rest and intermediate positions (default = 100%);
- Weighting of EES (default = 0%).

When the braking level of a vehicle is not known it is advisable to assign little or no weighting to that vehicle's rest position initially. Define a known intermediate position fairly close to the impact point where tire forces have not played a large part in altering the vehicle's trajectory. If only rest positions are known, define the rest position as an intermediate position. In either case, assign braking levels on a trail and error basis between optimizer runs until it looks like reasonable values are being used, and then increase the weighting of the rest positions.

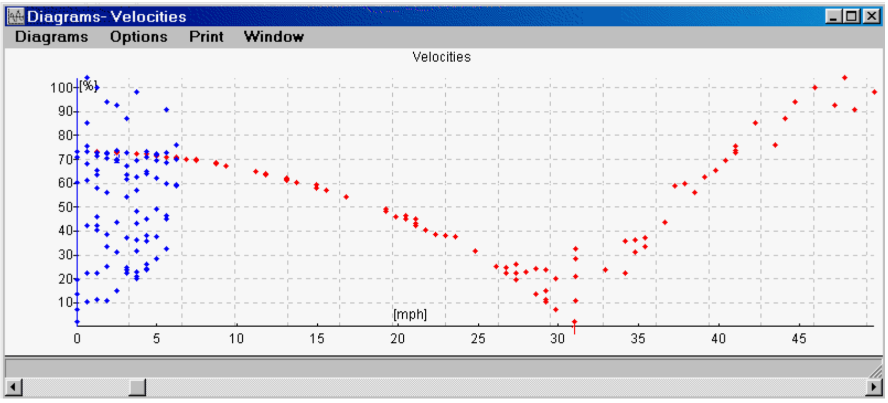
Report

This dialog box shows a report of the Optimizer results. The error ranges of the results are based on the user-selectable Optimizer trajectory error limit.

Deletes all the previous results. If this is not pressed between Optimizer runs, the previous iteration results will be combined with the new ones.

The results can also be viewed in the form of diagrams. Select **Options – Diagrams – Diagrams – Collision Optimizer** to view diagrams for:

- **Velocities**
- **Point of impact**
- **Restitution, contact friction**
- **Pre-impact directions**
- **Pre-impact positions**



All the iteration points from Optimizer runs made since the last time

was pressed will appear on the diagrams.

Madymo® Occupant Simulation

The optional Madymo occupant model is operated using this menu item. See the **Madymo Occupant Model** chapter for complete details on this feature.

Options

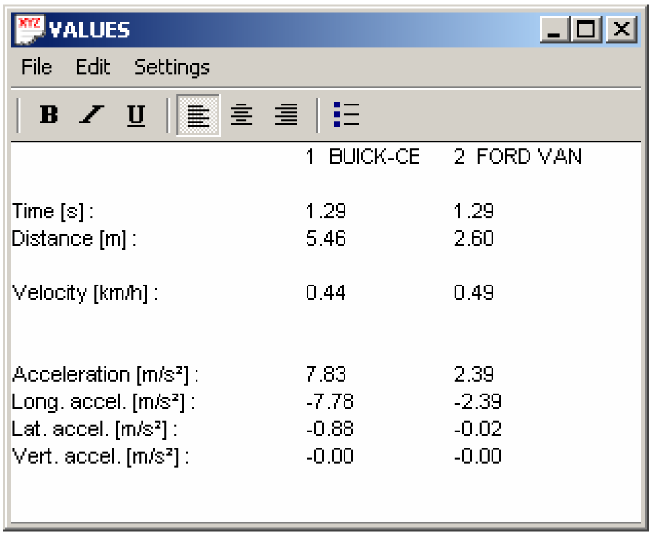
This menu is used to view tables and diagrams, and to change program settings, main screen view settings, selection of Side and 3D views.

Values (F4)

Selecting Values opens a window in which various simulation and/or sequence values are displayed and continuously updated throughout the simulation. This option allows the user to print or output values that are important for the assessment of a simulation run.

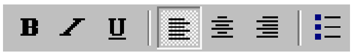
The Values window can also be used as a text editor, for defining vehicle files (*.DAT), for example. This window uses RTF (rich text format) which means it is able to assume text formats. The user can choose proportional scripts and the format can be imported directly into a word processing program.

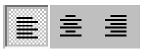
The Values window also displays the character count including space characters of any report created.



Toolbar

The Values window toolbar allows the user to select different text formats and scripts.



	Bold: Activates and deactivates **bold type** of the selected text
	Italic: Activates and deactivates *italic type* of the selected text
	Underline: Activates and deactivates underline type of the selected text
	Left, center or right alignment
	Adds or removes bullets before the selected text.

There are three menus at the top of the Values window, as follows:

File

The File menu is used for loading and saving files, for printing of the Values window contents, and for selecting font styles in this and other windows in PC-Crash.

All indicated parameters in the Values window can be saved or printed under the menu option **File** - **Print**.

Additional notes for a simulation can be added at any place in the window before printing or saving. Move the cursor to the desired position, click the left mouse button, and then type in the desired text.

Note: *There is a limitation regarding the size of supported files. Maximum viewable file size is 64kB, which is sufficient in most cases. However, when trying to view Vehicle Dynamics, where ten columns of values appear for every specified time step in the simulation, this viewable size can be too small. This can be overcome by saving the Values file and then viewing it with any text editor.*

Edit

This menu is used to edit text in the Values window.

Settings

The Settings menu is used to choose which parameters are to be displayed in the Values window.

The first six menu options (those above the line) can only be viewed one at a time. When selected, all other displays are suppressed. Click on the selected menu item above the line again to deselect it and view all the selected menu items below the lower line.

Sequences

Displays all sequence values defined in the Sequences dialog box.

Sections

Displays the actual time, distance and velocity for all defined sequences. The values can only be shown if the whole simulation was previously calculated.

Crash Parameters

Displays all calculated crash parameters for each crash in the simulation.

Vehicle Dynamics

Displays time, position, rotation and velocity of all vehicles in pre-defined time intervals. The time interval is defined in **Display Settings** using Last Branch. After defining the time interval, deactivate Last Branch or the vehicles will be drawn at these intervals on the main screen as well.

When this information is saved as a text file (using Save from the Values File menu), it can be used in other programs (e.g. for video animation or simulation of occupant motion).

Report

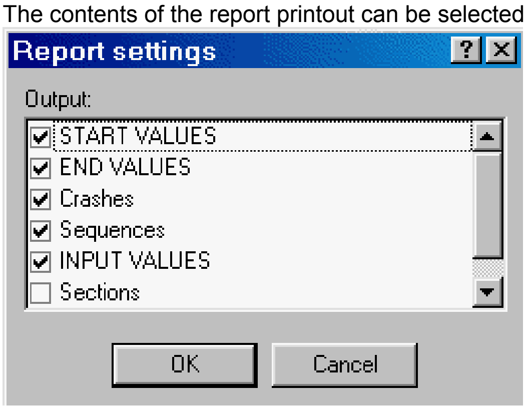
Displays a complete report of the simulation, identical to the printout that occurs with the **File - Print Report** menu option.

Use Template

Allows a report template to be loaded and automatically accept the data from the simulation. Report templates (*.rtf files), mainly useful for reporting values from the Kinematics toolbar, are located in the directory PC-Crash is installed in. The file Keywords.rtf shows all the keywords that can be used in documents.

Report Settings

The contents of the report printout can be selected here.



Distance/Time

Displays the time and distance traveled for the vehicle along its path. These are set to zero at the beginning of every new simulation.

Tire Normal Forces

Displays the tire forces normal to the road surface at each wheel.

Tire Lateral Forces

Displays the forces lateral to the tire's longitudinal axis at each wheel.

Brake forces

Displays the braking/acceleration forces in line with the tire's longitudinal axis at each wheel.

Energy

Displays the kinetic energy of the vehicle.

Coefficient of Friction

Displays the present friction coefficient at each wheel.

Velocity

Displays the center of gravity velocity of the vehicle along its path.

Acceleration

Displays the center of gravity acceleration of the vehicle.

Diagrams (F2)

The Diagrams window is for viewing graphs of various values with respect to distance or time for each vehicle.

Diagram Scaling

The diagrams can be scaled with the Zoom in and Zoom out buttons in the diagram windows toolbar. Using the button resizes the diagram so that it fits in the diagrams window. The displayed area can additionally be changed by selecting the menu option **<Options> <Diagrams / Axes>**. In the dialog the appropriate view area of the diagram may be entered either manually in the input fields (**xmin** – **ymax**) or the view area can be maximized to the diagram extents automatically by selecting the appropriate check boxes.

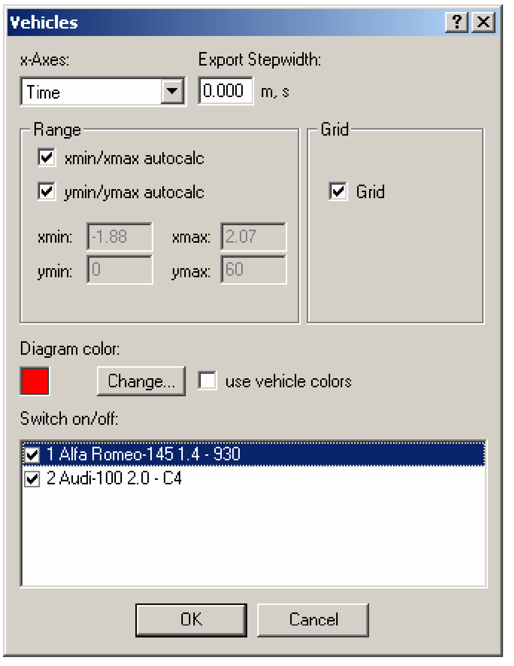


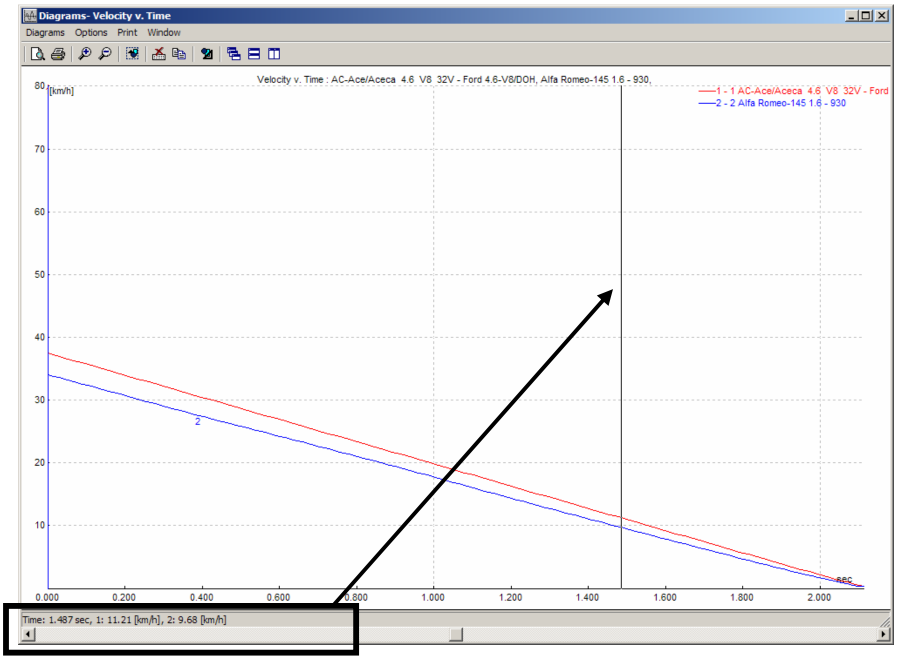
Diagram Line Colors

The colors of the diagram lines correspond to the "Basic" vehicle colors chosen in **Options – Options – Colors**. For example, diagram line 4 will be the same color as Vehicle 4 in the simulation. Because of this, white should be avoided as a vehicle color, as the diagram line corresponding to that vehicle number will not show on the white diagram background.

The color of a line may be changed also by selecting the menu option **<Options> <Diagram / Axes>**. In the dialog the color of each diagram line can be specified individually. By selecting the check box **use vehicle colors** the diagram lines are reset to the vehicle colors.

Diagram Scanning

A scan function allows values anywhere along the graphs to be shown in the Status Bar. Pressing the left mouse button with the cursor at the desired position on the diagram accomplishes this.



The diagrams can be configured with the local cascade menus. These menus are at the top of the Diagrams window, as follows:

Diagrams

This menu is for choosing which graphs are to be viewed. Each menu option can be activated or de-activated by clicking it once. The X-axis of the graphs is distance or time, depending on the setting in **Diagrams - Options – Diagrams / Axes**.

Vehicles

The following vehicle graphs can be selected using this menu item:

- **Velocity** - a vehicle's speed along its path.
- **Distance v. Time** - the elapsed time (seconds) with respect to distance traveled along a vehicle's path.
- **Distance-Time-Velocity** - the elapsed time (seconds) with respect to distance traveled along a vehicle's path and vehicle's speed along its path
- **Heading** - vehicle heading direction, degrees CCW from global X-axis.
- **Yaw angular velocity** - yaw angular velocity, in radians/sec or degrees/sec about the vertical axis, + in CCW direction.
- **Steering angle** - the steering angle of a vehicle's wheels in degrees, + in CCW direction.
- **Brake factors** - the brake/acceleration forces applied to each wheel, in % of normal static force.
- **Coefficient of friction** - the friction coefficient at each wheel.
- **Tire normal forces** - the vertical forces (normal to the road plane) on each tire.
- **Tire lateral force** - the lateral forces (parallel to the road plane) on each tire.
- **Tire brake forces** - the longitudinal forces (parallel to the road plane) on each tire.

- **Acceleration** - the vehicle's center of gravity longitudinal, lateral and vertical accelerations.
- **Roll Angle** - the roll angle of a vehicle about its longitudinal axis, degrees CW from level, if viewed from the rear (only applicable for 3D simulations).
- **Roll angular velocity** - the roll velocity of a vehicle about its longitudinal axis, + in CW direction when viewed from the rear (only applicable for 3D simulations).
- **Pitch Angle** - the pitch angle of a vehicle about its lateral axis, in degrees CW from level, if viewed from the right (only applicable for 3D simulations).
- **Pitch angular velocity** - the pitch velocity of a vehicle, + in CW direction, if viewed from the right (only applicable for 3D simulations).
- **Tire overall slip** - the tire slip due to brake or acceleration forces.
- **Tire rpms** - the wheel rotational speed, in rpm.
- **Trailer Hitch Force** - the lateral, longitudinal and vertical components of the hitch force between a tow vehicle and its trailer.
- **EES** – the calculated EES values.

Multibody Systems

This menu allows graphs of the following to be viewed for each of the 16 components of the multibody pedestrian. For all items except **Energy**, four values are given for each multibody component (the X, Y, and Z global directional values, as well as the resultant).

- **Distance**
- **Velocity**
- **Acceleration**
- **Rotation angle**
- **Angular velocity**
- **Angular acceleration**
- **Energy (kinetic)**
- **Contact Forces**
- **Spring Forces**

Collision Optimizer

This menu allows graphs of the following Collision Optimizer items to be viewed:

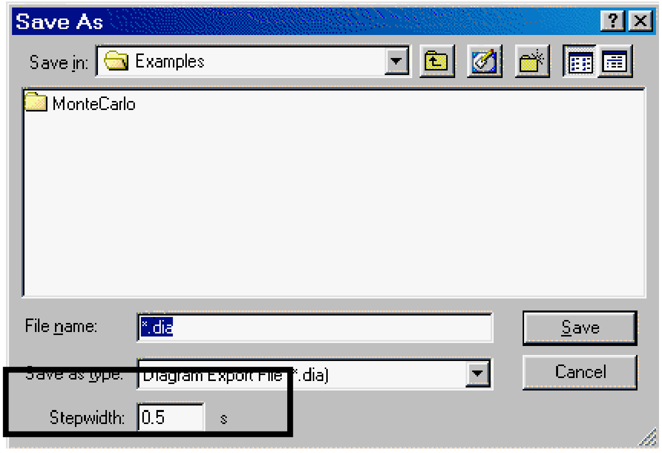
- **Velocities**
- **Point of impact**
- **Restitution, contact friction**
- **Pre-impact directions**
- **Pre-impact positions**

Madymo® Diagrams

See the **Madymo Occupant Model** chapter.

Export Diagram

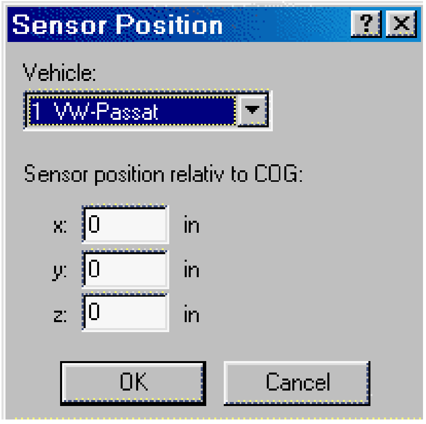
Exports a *.dia or *.dxf text file of the chosen diagram(s). The following dialog box allows the data to be saved to a text file with the selected step width (in distance or time). When working with a time graph, the user will generally want to reduce the 0.5 seconds default value to a smaller one.

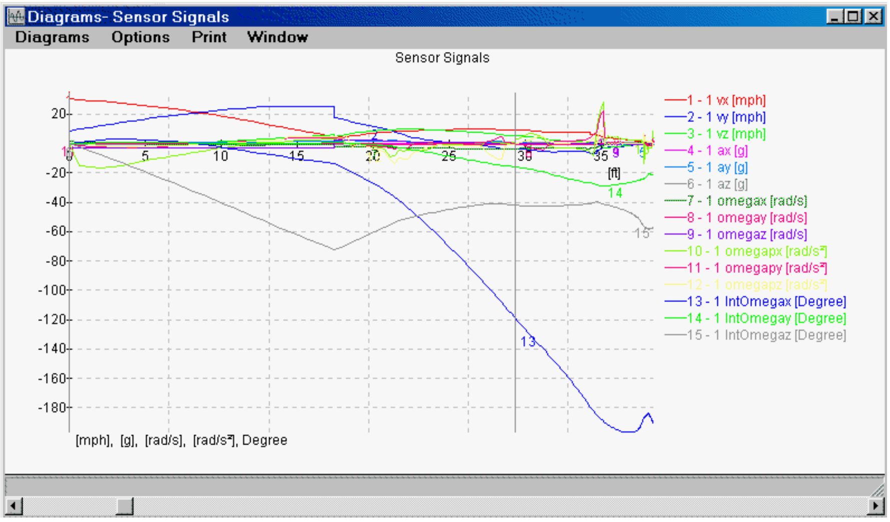


Sensor Signals

This feature allows the user to place a sensor anywhere in the vehicle so that velocities, accelerations and rotational values at a selected point can be viewed on a diagram. The default location is at the C.G. (x = y = z = 0).

Since this feature is intended mainly to see what a particular occupant is subjected to, the acceleration of gravity is included in the acceleration plots. Thus, a vertical acceleration of –1g will show for a stationary vehicle.





Any or all of the following can be shown on the diagram, as selected using the **Options – Diagrams / Axes** menu item in the Diagrams window:

- **Velocity** in the x, y and z directions - Curves 1, 2, 3
- **Acceleration** in the x, y and z directions (-1g in the global Z direction is included for the acceleration of gravity) - Curves 4, 5, 6
- **Rotational velocity** about the x, y and z axes - Curves 7, 8, 9
- **Rotational acceleration** about the x, y and z axes - Curves 10, 11, 12
- **Rotation** about the x, y and z axes - Curves 13, 14, 15

The above Curves 1 to 15 are all for Vehicle 1. When there are more vehicles in the simulation, Curves 16 to 30 are for Vehicle 2, Curves 31 to 45 are for Vehicle 3, etc.

Options

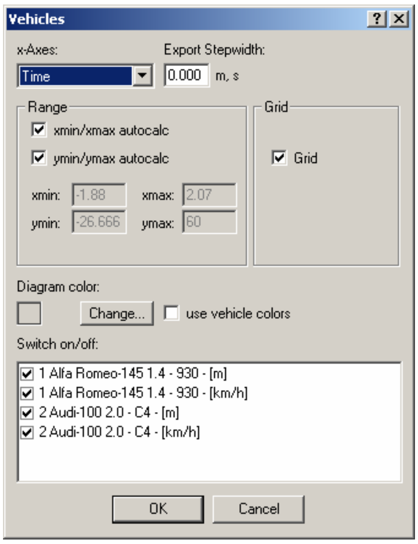
The Options menu is for selecting display options in the Diagrams window.

Copy (CTRL C)

This, or the button , copies the contents of the Diagram window to the Windows clipboard for insertion in another program. Depending on the option chosen under Paste Special in the destination document, the diagram can be pasted either as an image or a text file. The diagram step width is selected with the Diagram/axes option (see below).

Diagrams / Axes

Use this option for selecting distance or time for the x-axis, for selecting which diagrams will be displayed (for multibody diagrams, only time is available), and for selecting the diagram export step width when the Copy option (see above) is selected..

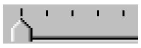


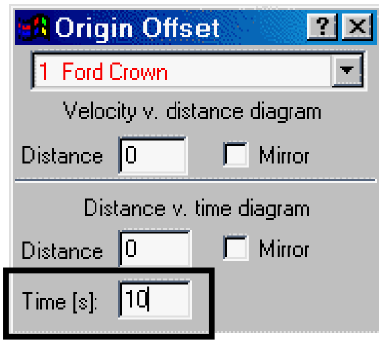
Within the Vehicles window each item can be switched on or off separately with the applicable check boxes. Additionally, all items can be switched on or off by clicking with the right mouse anywhere in the Vehicle dialog box.

Additionally the color of each diagram line and the visible diagram area can bet set.

Origin Offset

This shifts a graph to the left or right by entering the desired distance or time value in the appropriate text box of the Origin Offset dialog box. The Distance v. Time graph can also be shifted vertically.

When a shift in time is selected for the Distance v. Time diagram, the vehicle will also be shifted in time in the simulation when using the Simulation toolbar slider bar  and in rendered animations, in addition to the shift that occurs in the Diagrams window. This is useful for synchronizing the motion of a vehicle with respect to another. Refer to the **Synchronization of Vehicles** section in the previous chapter.



The graphs can also be mirrored with the appropriate  Mirror selection. This is useful when showing diagrams including collisions.

Grid

This menu option toggles whether or not a grid is superimposed on the diagram. The density of this grid is identical to the grid density on the main screen and can be modified with the **Graphics**- **Scale & Grid Spacing** menu option.

Move

Move allows the diagram lines to be shifted back and forth. After activation, a diagram can be shifted with the cursor on it and the left mouse button pressed.

Draw Toolbar

This activates the drawing toolbar, so that text or drawing elements can be added to the diagrams for descriptive purposes. Refer to the **Drawing – Draw Toolbar** menu item description.

Print

The Print menu or the button resp. the button are used for printing the diagram on the printer. A print preview feature allows the printout to be viewed first. If multiple diagram windows are open, all diagrams will be printed on a single page.

The text entered in **File - Print Comments** will be printed with the diagrams.

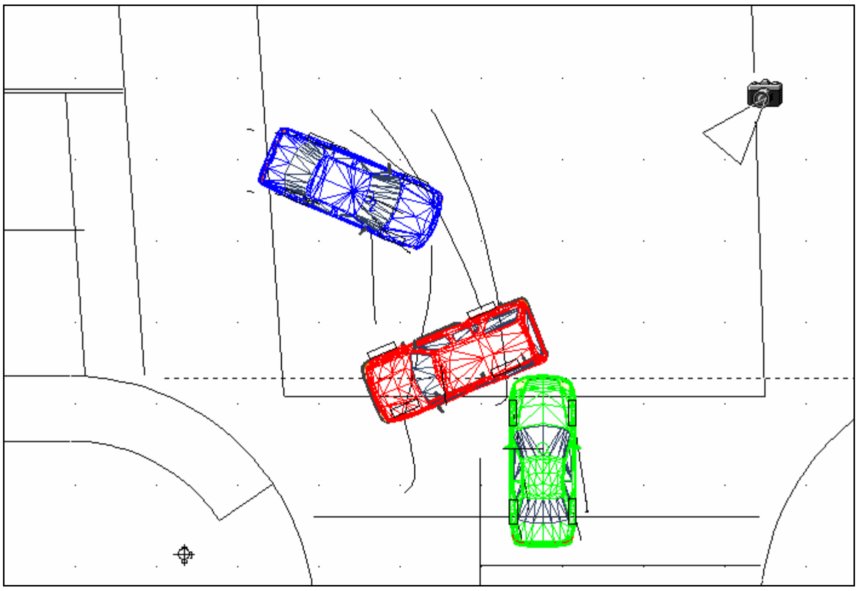
Window

The Window menu is used to open, close and organize the diagram windows for multiple diagram viewing. The windows can also be organized by using the buttons , , and .

3D Camera Position

This tool is used to position a camera on the main screen to define the point from which the 3D vehicles and scene will by viewed (PC-Crash 3D only).

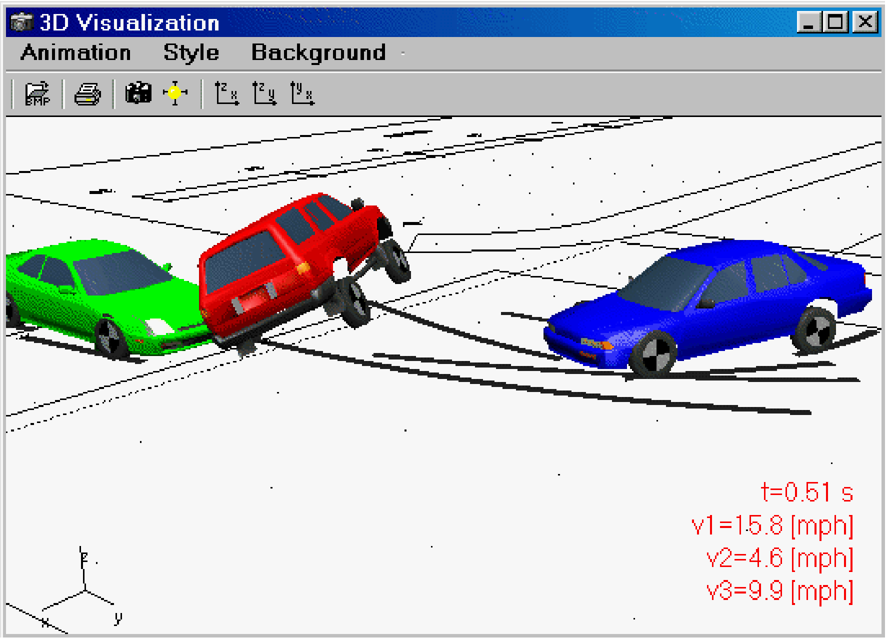
When activated, the cursor changes to a camera symbol on the main screen. The camera can be moved and rotated by moving it with the left mouse button pressed. If the camera is not visible in the screen area a single click of the left mouse button with the cursor at any position will move the camera to that position. To change the viewing direction, hold down the left mouse button with the cursor in the viewing cone and move it to rotate the cone.



3D Window (F9)

This opens the 3D View and Animation window (PC-Crash 3D only). This window can also be opened by double-clicking on the camera symbol.

This window displays a 3D view of the scene with the vehicles positioned to match the current frame in the simulation, along with any other positions defined in **Display Settings** .



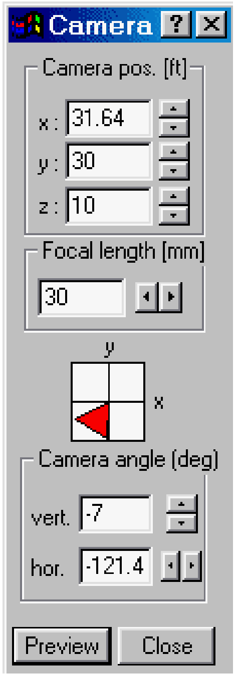
The 3D Visualization window contains three menus at the top (Animation, Style and Background), as follows:

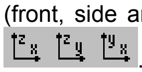
Animation

This menu is for positioning the camera, rendering and playing animations, and printing 3D views.

Set Camera Position

This opens the Camera Settings dialog box, in which camera position, rotation and focal length can be changed to attain the desired view. For "bird's eye" views, a maximum vertical angle of 89° can be selected.



In addition to the camera position specified here, there are three pre-defined views (front, side and top) that can be selected from the 3D Visualization window toolbar .

Dynamic Camera Positioning

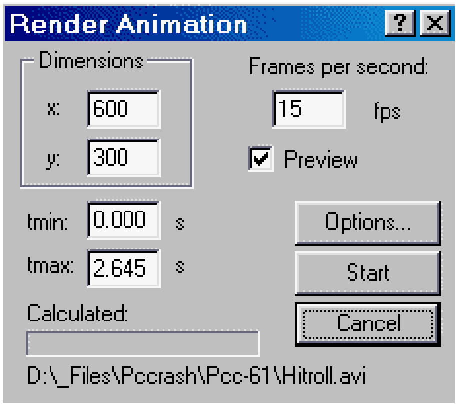
Dynamic camera moving, panning and zooming in the 3D window is possible, as follows:

- **Moving the Camera** - Hold the left mouse button down and move the cursor in the 3D window.
- **Panning (Rotating) the Camera** - Hold the SHIFT key and the left mouse button down and move the cursor in the 3D window.
- **Zooming the Camera** - Hold the CTRL key and the left mouse button down and move the cursor up or down in the 3D window to move the camera away or towards the scene.

Render

This is for rendering animations of simulations. The animations are saved as *.AVI files, which can be played using Windows Media Player.

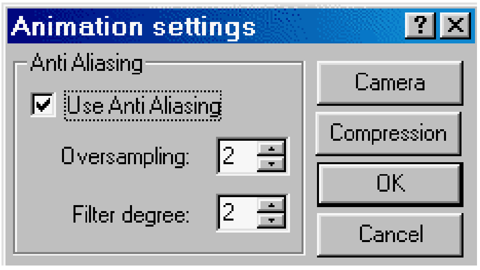
When Render is selected, the desired file name for the animation must be entered in the Save As dialog box. After the new file name has been accepted by pressing the OK button, the Render Animation dialog box appears.



Here, the animation dimensions (in pixels), the number of frames per second, the start and stop times of the animation, and various options can be selected.

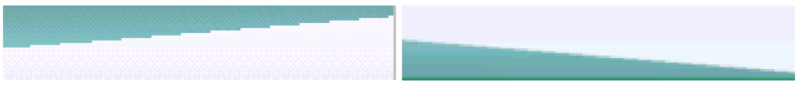
Options

This button opens the Animation settings window, in which anti-aliasing, fixed or moving camera and various compression options can be chosen.



Use Anti-Aliasing

Anti-aliasing smoothes jagged edges by applying intermediate shades of adjoining colors at even intervals along the interface.

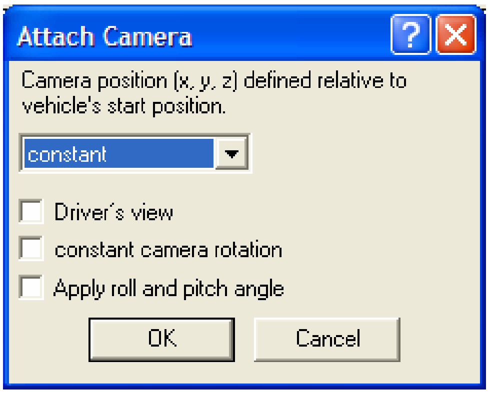


Without Anti Aliasing With Anti Aliasing (2, 2)

Anti-aliasing should only be used for presentation quality animations, as it increases 3D window refresh and rendering time markedly.

Camera

The Camera button allows the user to choose a camera that moves with one of the vehicles, instead of the default fixed camera position. This is done in the drop down list at the top of the Attach camera dialog box that appears.



Driver's View - This attaches the camera to the vehicle, places it in the approximate position of a passenger car driver at an elevation of 3.94 feet (1.2 meters), and assigns it a focal length of 35mm. These values can be changed afterwards, if desired.

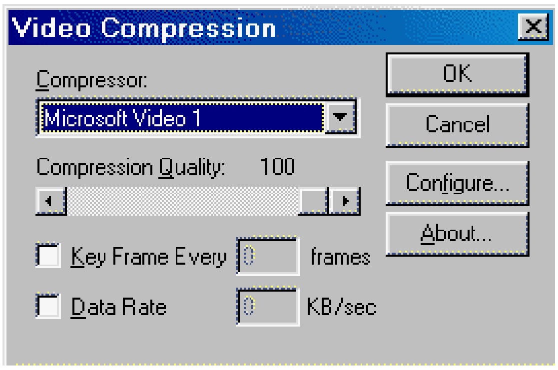
Constant Camera Rotation – This prevents the camera attached to a vehicle from rotating in yaw with the vehicle (the camera never rotates with vehicle roll or pitch).

Apply roll and pitch angle – The roll and pitch angle is used for the calculation of the camera attached to a vehicle

Compression

This button opens the Video Compression dialog box, in which several video compression parameters can be changed. In complex animations, changing these parameters can make a practical balance between animation quality and playback speed. These options are used when the video sequence is saved to disk.

Video sequences are compressed and decompressed by routines called codecs (short for compressor/decompressor). A codec is responsible for compressing raw video data into a format suitable for distribution. When the compressed video sequence is played, the codec performs a reverse role, converting the compressed data into images that can be displayed.



Video for Windows includes several codecs, each of which has advantages and disadvantages. When evaluating a codec, you should consider the following factors:

- Quality: how well does the codec preserve the original colors, motion, and image detail?
- Data rate: does the codec reduce the data-transfer rate to an acceptable level?
- Storage size: how much storage space is saved as a result of compressing?
- Time: how much time is required for compression and playback?

You should experiment with several compression methods to see which works best with your video content and delivery platforms.

The following compression methods are available in the Compressor drop down list in the Video Compression Dialog Box:

Method	Description
Cinepak Codec by SuperMatch	Excellent compression method for video sequences delivered on CD-ROM. Cinepak provides good image and motion quality at CD-ROM data-transfer rates (15 frames per second with a 320-by-240 frame size produces a 150K per second data rate). The video sequence is stored using the 24-bit color format, which preserves much of the color information from the original video. When playing on 8-bit displays, Cinepak uses dithering to transfer the 24-bit color values to an 8-bit palette format. Cinepak uses an asymmetric compression technique, one that takes much longer during compression than during playback. On a fast capture system, allow approximately one hour of compression time per minute of video at 320-by-240 frame size, 15 frames per second.
Intel Indeo(TM) Video R3.2	Provides high-quality video (320-by-240 frame size and 15 frames per second), using an asymmetric compression technique. Indeo 3.1 uses a 24-bit color format. When played on 8-bit video displays, Indeo dithers the 24-bit colors to 8-bit format. Indeo takes advantage of compression hardware on the Intel Smart Video Recorder capture board, allowing real-time hardware compression during capture. Intel recommends a key-frame interval of 4.
Microsoft RLE	Fast compression method useful for computer-generated animation, screen captures, and other sequences with areas of uniform color. Quality and size reduction drop rapidly if there are many changes from frame to frame (for example, as would be the case in a sequence captured from videotape). The RLE method is limited to 8-bit video sequences.
Microsoft Video 1	Combines good playback quality with relatively quick compression times. If you're using the Cinepak codec as your final

	compression method, you might consider using Video 1 for draft versions of the video sequence. Just be sure to retain your original, uncompressed sequence as a master copy. The video sequence is stored in either 8-bit or 16-bit format. Using the 8-bit format, you can specify which palettes to use in the video sequence, so if your sequence is playing strictly on 8-bit machines, Video 1 provides better control over the palette colors than compression methods that use an automatic dithering technique.
Intel Indeo(TM) Video Raw	Uncompressed YUV format produced by the Intel Smart Video Recorder capture board. Capture in this format if you want to use a different compressor than Indeo 3.1.
Full Frames	Not a compression method; instead, Full Frames saves the animation as an uncompressed AVI file (without any loss in quality).

The following options are available in the Video Compression dialog box:

Options	***Description***
Compression Quality	Lets you balance image quality with the specified data rate. Higher quality (moving the slider to the right) increases the importance of clear, accurate images at the expense of smooth motion. Lower quality (moving the slider to the left) decreases the image quality but provides smoother motion. The Configure button also contains a command button that provides compressor-specific settings or information.
☑ Key Frame Every n Frames	Specifies how often a full frame should be stored in the video sequence. Key frames provide a complete frame image and act as a reference point for successive frames that store only differences from the previous frame. Key frames provide convenient jump destinations for the video sequence; users can jump to the key frame, and the full-frame image will be displayed.
☑ Data Rate	Specifies the expected data-transfer rate at playback. The video compressor uses this

information to determine how much information to include on each frame; with higher data rates, more information can be stored, and quality is better. If the video sequence is played from a device that has a slower transfer rate than specified for compression, the playback quality might suffer.

Note: *Don't apply multiple compression methods to your video sequence. Most methods result in some loss of quality, and the loss is compounded if multiple techniques are applied. For example, if you have a video output card with its own compression software, there is no need to use any compression when creating the AVI file in PC-Crash.*

When experimenting with compression settings, save the original uncompressed version of your video sequence as a baseline.

Start

The recording of the video animation is initiated with the Start button. Depending on its complexity, the video animation can take several minutes to create.

Note*: The computer display should be set at 16 bit color or higher. A setting of only 256 colors will cause a 10x increase in rendering time. Check your display setting by clicking on the Display icon in the MS-Windows Control Panel window.*

The created video animations can be played at any time using the **Windows Media Player** program.

The features of this program are documented in the Windows manual.

Playback

This menu item allows the user to choose from, and play, previously rendered animations from within PC-Crash, instead of opening Windows Media Player.

Print Preview

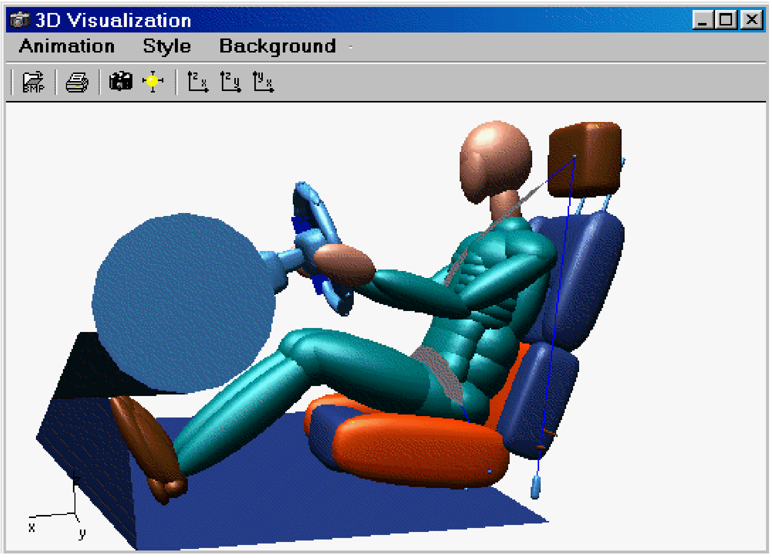
This shows a preview of the 3D window printout prior to printing. The text entered in **File - Print Comments** will be printed with the 3D view.

Print 

This prints the current 3D view on the printer.

Load Madymo Kin3-file

This is for importing a Madymo Kin3-file. The 3D simulation of the multiple body movement can be visualized. Refer to the **Madymo Occupant Model** chapter for more details.

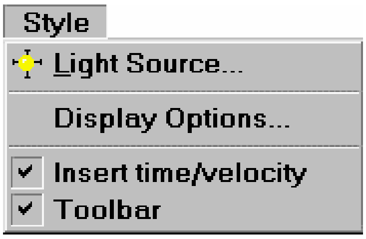


About 3D

Gives a brief information about the PC-Crash animation feature.

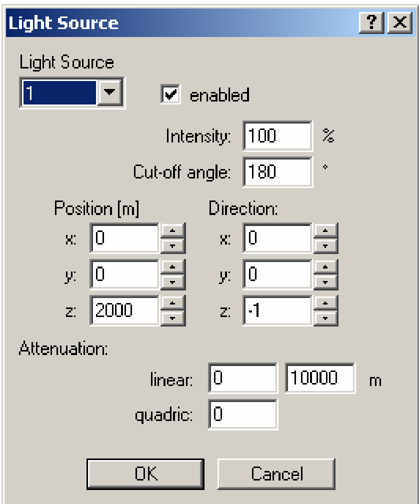
Style

This menu is for altering the appearance of the vehicles and objects in the 3D view.



Light Source

This is used to define the position of the light source(s). The light source position affects the shading of the vehicles and the location of the shadows.



By default, only one light source is active. Use the Light source drop down list and Enabled to activate additional light sources.

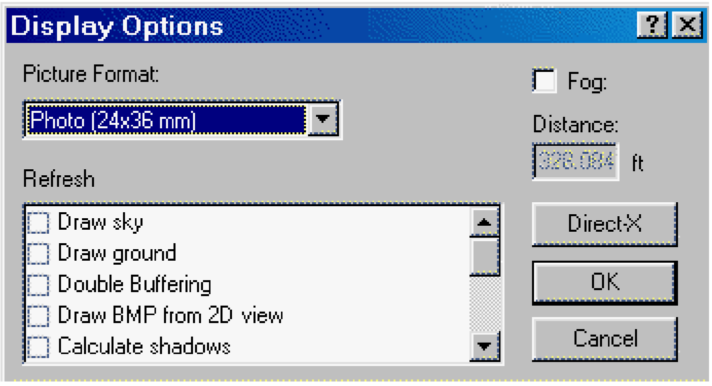
Use the Linear and Quadratic text boxes to change the effects of each light source. The light attenuation *A* is calculated as a function of the distance *D* between light source and object:

$$A = \frac{1}{A_0 + A_{Lin} D + A_{Quad} D^2}$$

Typically the distance between light source and objects is quite far, therefore even small values of A_{Lin} and A_{Quad} will reduce light intensity significantly. These constants are useful for modeling streetlights.

Instead of entering the linear attenuation, you can enter the distance. The linear attenuation will be computed automatically.

Display Options



This dialog box contains various options which can be toggled on or off. Picture Format is only important for animations that will be transferred to video tape.

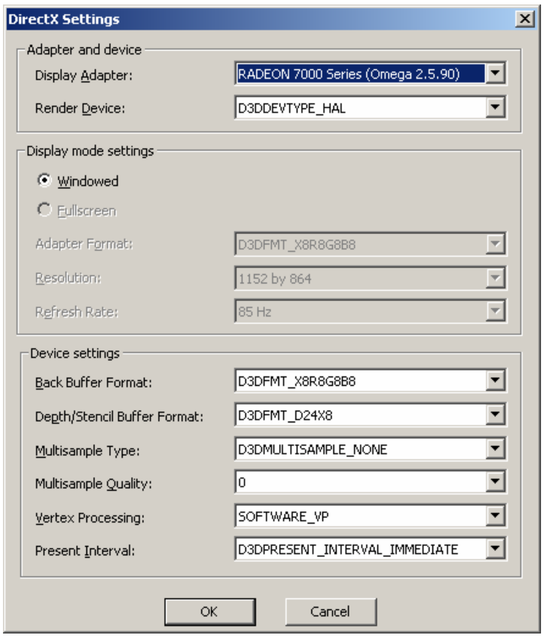
Other options available are:

Fog	This activates fog, with a user-specified visibility distance.
Draw Sky	Shows a graduated blue sky (darker at higher elevations).
Draw Ground	Colors the ground green, except where there is a DXF drawing shape or bitmap.
Double Buffering	Redraws the 3D window as a whole, rather than vehicle by vehicle.
Draw Bitmap from 2D view	Converts the plan view bitmap from the main screen into the correct orientation for the chosen view. This takes some calculation time when the 3D window is first opened, but not subsequently, even when the camera is moved.
Calculate shadows	The shadows of the vehicles are shown.
Gouraud shading	Applies Gouraud shading to the vehicles, which significantly smoothes their shapes. In order for this option to function, the normal vectors of 3D DXF vehicle shapes have to be calculated when the shapes are brought into PC-Crash.
Detailed wheel shapes	The wheels are shown with target wheel covers to

	indicate rotation.
Wide tire marks	Wider tire marks are shown.
3D Texts	Displays DXF drawing text in the 3D view.
Wire frame view	Displays the objects as wire frame models
Vehicle numbers	Displays the vehicle numbers in the 3D window (not applicable for animations)
Vehicles	Displays the vehicles in the 3D window. Turning the vehicles off is a feature used mainly for viewing the Madymo occupant model.
Light sources	The light sources will be displayed as light emitting white balls at their position.

Direct X

This button opens the Select Direct3D® Device dialog box. Depending on what Direct3D® mode(s) the computer's graphics card supports, rendering speed can be increased by selecting the appropriate mode. The default mode is the HAL, the hardware abstraction layer. In this mode the hardware acceleration of the graphics adapter is used. As an alternative REF, the reference rasterizer, may be used. It is significantly slower and not supported by all device drivers, but supports every rendering mode since it is implemented in software.



Additionally you can choose the display adapter, if there are more than one on your computer.

The Back Buffer format specifies the display memory that is used for 3D visualization. It is not required to change this value if the 3D window displays the scene. Otherwise, try different formats to reduce memory consumption.

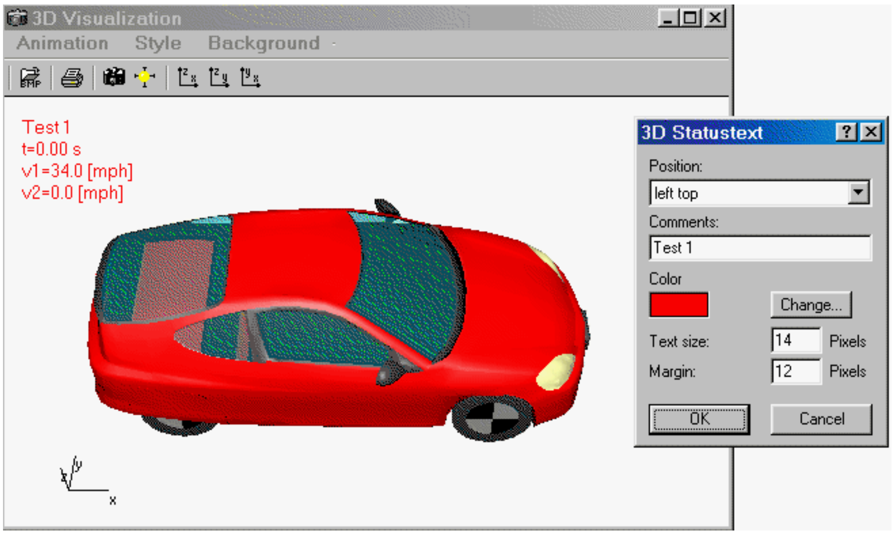
The depth buffer is used to determine which pixel is nearer to the camera and should be displayed. As with the back buffer there are different formats the device may use. If display does not work correctly try different formats to reduce memory consumption.

Additionally you can improve the quality by selecting a multisampling type. Multiple samples are rendered to anti-alias the picture. The better the quality the more time will be needed by the device for drawing the scene.

Vertex processing means the transformation and lighting calculations of the 3D-objects in the scene. You can choose if those calculations are performed by the software or by the graphics hardware by setting the corresponding value. In general, hardware processing is faster but not all operations are supported in every device.

Insert time/velocity

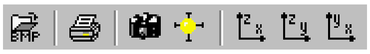
This enables the time and each vehicle's velocity (if it is activated in Simulation Model ) to be placed on each animation frame. Any of the four corners of the animation frames can be selected, with a margin (in pixels) from the vertical and horizontal edges of the frames. The text size and color can be changed, and comments can be added.



To deactivate this option, select "do not display" in the Position drop down list.

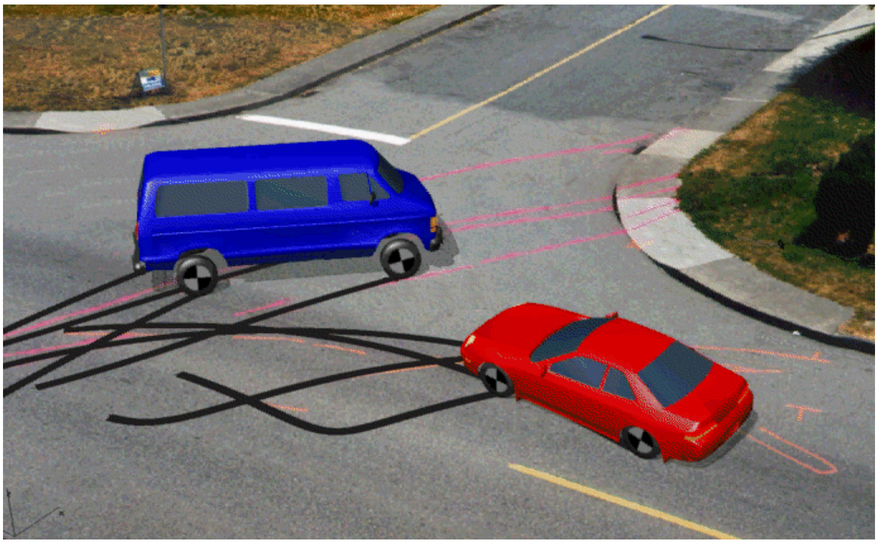
Toolbar

This activates and deactivates the 3D Visualization window toolbar.



Background

The Background menu is for loading and altering bitmap files for use as a background in the 3D view. A perspective photograph of the actual incident scene can be an effective background.



Load Bitmap 

This loads a bitmap file into the 3D view. For scene photographs, PC-Crash's camera position must be adjusted to the position the photograph was taken from.

Erase Bitmap

This deletes a bitmap that has been loaded into the 3D Visualization window.

Position Bitmap

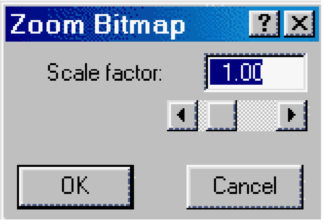
This alters the size and aspect ratio of a loaded bitmap in one step. Drawing a rectangular box (with the left mouse button held down) does this. The bitmap will be distorted to conform to this box, so this option should not be used with photographs.

Move Bitmap

This moves the bitmap (with the left mouse button held down) to any position desired in the 3D view.

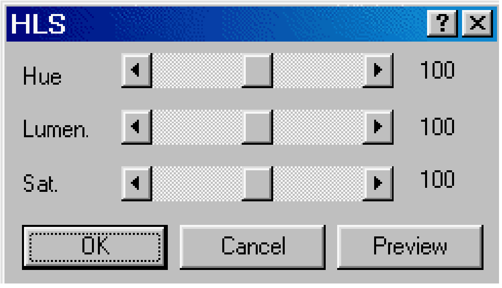
Scale Bitmap

The size of the bitmap can be changed. This is done by entering the desired scale factor in the Zoom Bitmap dialog box, which appears on selection of this menu option.



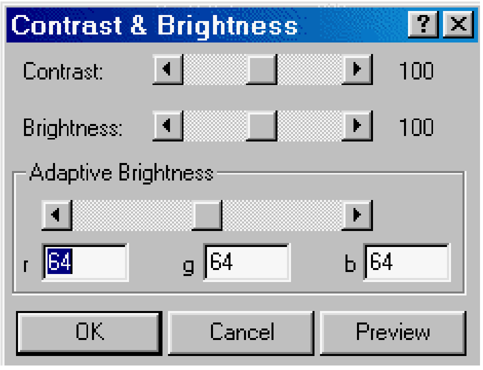
Hue/Lum/Sat

The hue, luminance and saturation of the loaded bitmap background can be changed with the slider bars in the HLS dialog box.



Contrast/Brightness

The contrast, brightness, adaptive brightness and color of the loaded bitmap background can be changed with the slider bars and text boxes in the Contrast & Brightness dialog box.



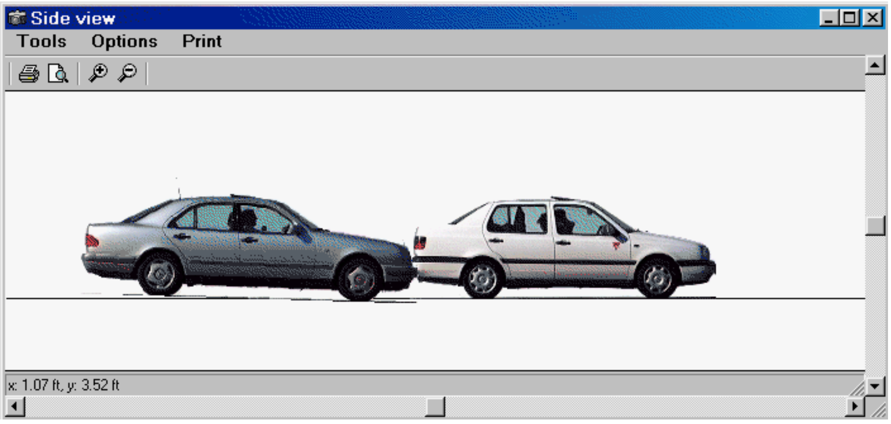
Using the scroll bar for Adaptive Brightness, only brightness and contrast of areas darker than the predefined values are modified.

The Preview button is used to display the changes without modifying the actual picture. This can be done several times. If OK is pressed the actual picture is modified according to the preview.

Side View Window

The Side View window allows vehicles and objects to be viewed in true scale in the vertical plane. A measurement feature, operated by clicking and holding the left mouse button and dragging the cursor from one point to another, is included. The resulting dimension is shown in the Side View window status bar.

The Side View window is especially useful for examining vehicle front-rear bumper interaction in rear-end collisions, as it includes a feature for applying vehicle pitch due to brake dive.



Before opening the Side View window, select a camera position that views the vehicles from the desired side (the distance of the camera from the vehicles does not matter, as the view can be zoomed in or out after the Side View window is open).

The following vehicle shapes can be loaded for use in the Side View window:

- 3D DXF shapes (load with **Vehicle – Vehicle DXF – File - Plan View – Load DXF**)
- 2D DXF side view shapes (load with **Vehicle – Vehicle DXF – File – Side View – Load DXF**)
- Bitmap side views (load with **Vehicle – Vehicle DXF – File - Side View – Bitmap**). A number of European vehicle side view bitmaps are included with PC-Crash, in the Side View subdirectory. Some U.S. and Japanese vehicles, as well as bicycles and motorcycles, are also included. Users can add their own subdirectories of photographs, as desired. The photographs should be taken as far away from the vehicle as possible, with a zoom lens, to minimize perspective distortion.

The following menus are included in the Side View window:

Tools

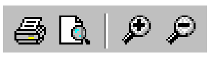
Zoom In 

Move the viewpoint closer to the vehicles.

Zoom Out 

Move the viewpoint further from the vehicles.

Toolbar

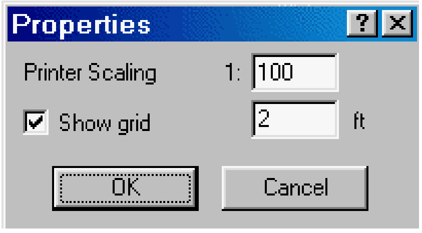
This activates/deactivates the toolbar  with the options Print, Print Preview, Zoom in, and Zoom out.

Status bar

This activates/deactivates the status bar, which shows the mouse position within the window.

Properties

Printer scaling,  Show Grid and grid resolution can be selected here.



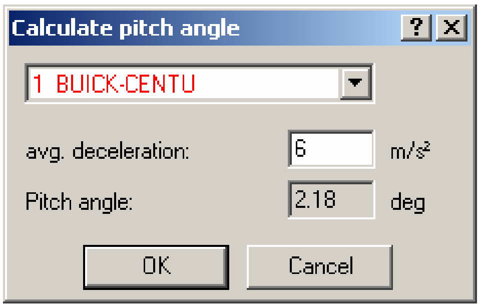
Options

Copy Window

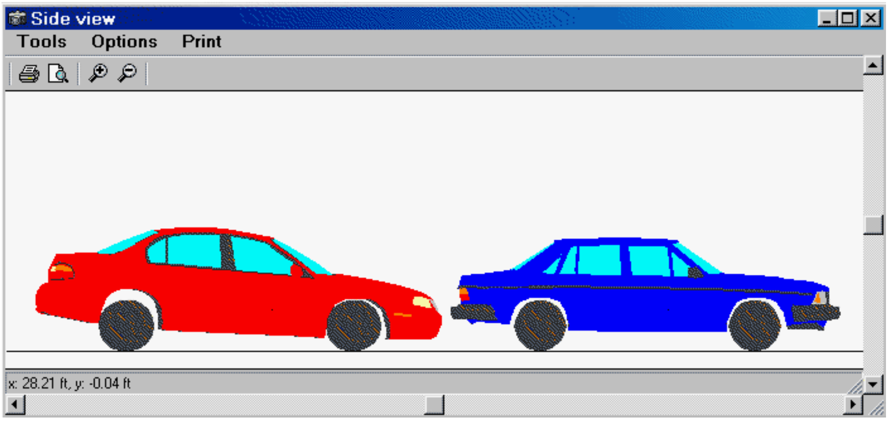
This copies the visible view to the clipboard as a Windows Metafile (WMF), for pasting into MS Word® or other programs that can accept this type of file.

Calculate Pitch Angle

This enables the calculation of vehicle pitch angle, based on the selected braking deceleration in **Dynamics - Sequences** and the vehicle properties in **Vehicle - Vehicle Settings**.



The calculated pitch angle for each vehicle will be shown in the Side View window, to enable a comparison of bumper heights.



Depth Sorting

This allows one vehicle to be shown in front of or behind the other.

Print

Print Preview

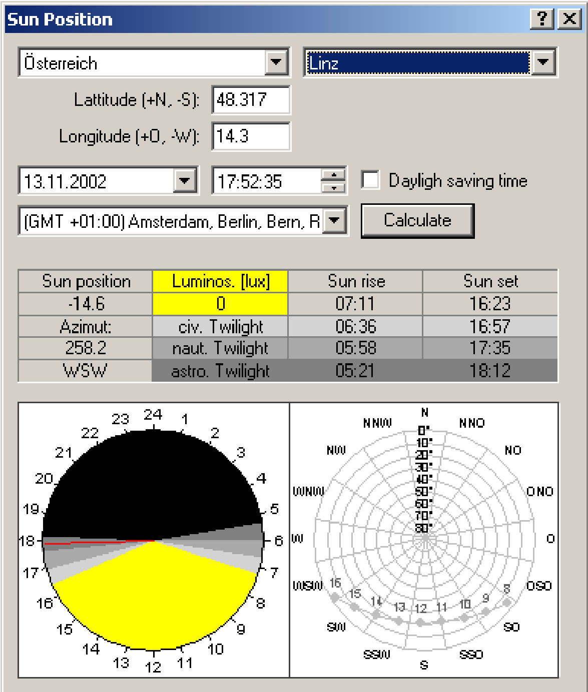
This shows a print preview, based on the scale selected in **Tools – Properties**.

Print

This prints the side view, based on the scale selected in **Tools – Properties**.

Sun position

This menu option is for the calculation of the position of the sun to the desired time to a given date at selected places.



Country

Selection of the country.

Place

Selection of the place. Predefined places can be selected.

Latitude (+N, -S), Longitude (+O, -W)

The coordinates of the selected places are displayed. Values for latitude and longitude can also be defined by hand (north positive, south negative and east positive, west negative)

Date, Time

Selection of the desired date and time.

Daylight saving time

Definition if daylight saving time active.

Time Zone

If coordinates are used for defining a place the corresponding time zone has to be selected.

Values

Sun position Altitude is the angular distance above the horizon (0 < h < 90°),

Azimuth the angular distance, measured along the horizon

Point of the compass the orientation is measured in degree (0 or 360° is equivalent to north, 90° = east, 180° = south and 270° = west). Additionally the direction (e.g. 109.4° = OSO) is indicated as text. In the right diagram the sun is represented in the current position (red pointer) and in the course of the day (grey).

Luminous intensity (lux). The density of light in lux (lx) as measure for the quantity of light on a surface. For the calculation a white surface with cloudless sky is accepted.

Sun rise Sunrise is the time at which the leading limb of the Sun first rises above the horizon. The effect of refraction in the Earth's atmosphere lifts the image of the Sun about half a degree at the horizon, making sunrise about two minutes earlier than would be expected from the actual position of the Sun in space.

Sun set Sunset is the time at which the trailing limb of the Sun first sets below the horizon. The effect of refraction in the Earth's atmosphere lifts the image of the Sun about half a degree at the horizon, making sunset about two minutes later than would be expected from the actual position of the Sun in space (note: in hilly area it comes by the surrounding relief to a so-called horizon shading, i.e. the horizon is not 0 degrees, so that e.g. in valley situations the sun comes up later and in former times goes down)

Civil. twilight. The period of twilight beginning (or ending) when the center of the (refracted) Sun is more than 6.5° below the horizon (0° to 6.5°).

Nautical twilight. The period of twilight beginning (or ending) when the center of the (refracted) Sun is more than 12° below the horizon

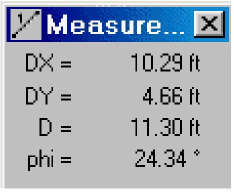
Astronomical twilightthe period of twilight beginning (or ending) when the center of the (refracted) Sun is more than 18° below of horizon

Grid

This activates or deactivates a main screen grid with settings as defined under **Graphics – Scale & Grid Spacing** (see the menu item description). The grid also appears at elevation 0 in the 3D window.

Measure 

The Measure tool enables measurements to be made on the main screen. Measurements are in feet (U.S. units) or meters (metric units).



The Measure dialog box includes the following:

- **DX** - Distance in X direction (horizontal on screen)
- **DY** - Distance in Y direction (vertical on screen)
- **D** - Distance along measurement line
- **phi** - Angle of measurement line (CCW from X axis).

Measuring is done by placing the cursor on the first point of the distance to be measured, and then pushing and holding the left mouse button while moving the cursor to the second point. The measured values remain visible in the Measure dialog box until the left mouse button is pressed again. Deactivation of the measure feature is done by either selecting another menu option or by clicking on this menu option again.

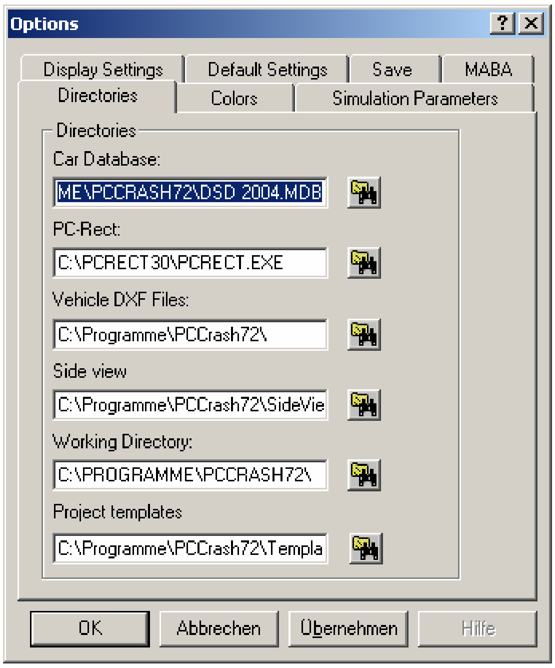
Status Bar

The status bar at the bottom of the screen can be turned on or off. It contains error messages and additional information (scale, friction, time, and simulation model).

Options

PC-Crash's default settings can be defined using **Options**. The settings are automatically saved in the **crash81.ini** file in the Windows directory.

When **Options** is selected the following dialog box appears:

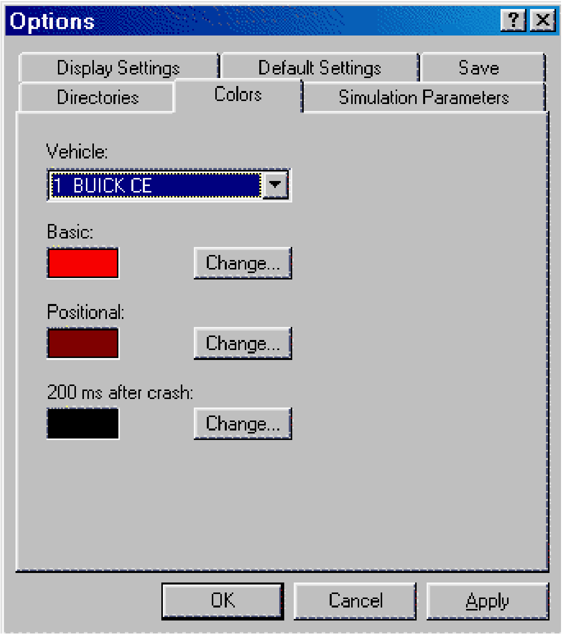


Directories

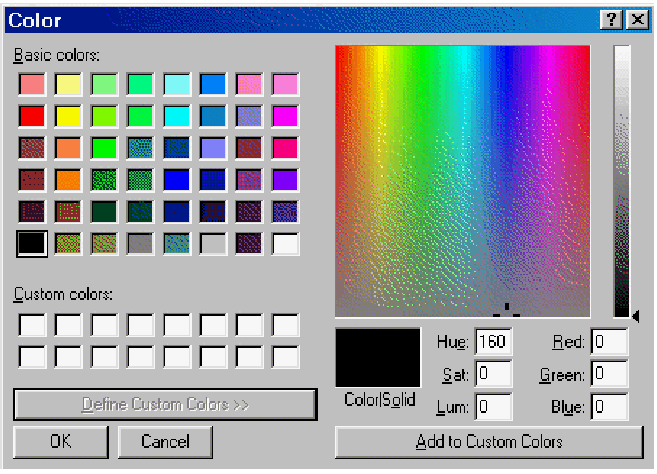
In this area, the directories where PC-Crash looks for the vehicle databases, PC-Rect, vehicle drawing and bitmap files, project files and project template files can be changed, using the respective Change button .

Colors

The base colors of all vehicles can be defined according to personal preferences. The defined colors will be used in all views.



When pressing Change for each vehicle color box (Basic, Positional and 200ms after the crash), the color can be selected from the following Color window. If desired custom colors can be created.



Basic

This color is for the current vehicle position on the main screen and is the color of the vehicle in the 3D window and animations. Additionally, the diagram lines (**Options – Diagrams**) will be the same color.

Positional

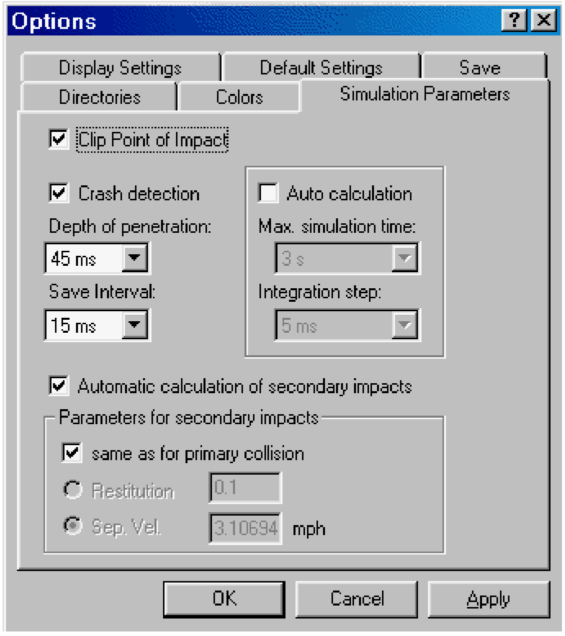
This color is for positions other than the current vehicle position on the main screen, such as start and rest positions.

200 ms After Crash

This color is for the vehicle 200ms after the crash, which shows when a crash is performed, using **Impact – Crash Simulation**.

Simulation Parameters

This menu item allows the user to change simulation parameters. This can also be accessed with the Options button in the Crash Simulation window.



Clip Point of Impact

When this is activated the point of impact can be located only in the overlapping area of the two rectangular vehicle outlines in the collision.

Crash Detection

This is used to activate an automatic crash detection feature, and is identical to the menu option **Impact – Crash Detection**.

Depth of Penetration

This drop down box allows the user to select the time from the first contact between the rectangular vehicle outlines to when the impact is calculated. Normally, a time of 30 to 60msec is realistic - this can be checked against the measured vehicle crush. This time is also the time between secondary impacts, if the vehicles are still engaged and approaching each other.

Auto Calculation

When this is activated the simulation will run to the vehicle rest positions, or for a pre-defined time interval (see next item), whenever a change is made in the Crash Simulation dialog box.

This check box is identical to Auto Calc in the Crash Simulation dialog box (see **Impact – Crash Simulation**).

Max. Simulation Time

When running in Auto Calculation mode, the simulation will be stop at the time specified here, if the vehicles have not come to rest prior to this time.

Integration Step

The Auto Calculation integration time step can be varied. Normally PC-Crash uses a time step of 5ms. However, for the Auto Calculation mode this time step can be increased in duration to speed up the simulation. This larger time step will not be used when the vehicles are near a crash.

Automatic Calculation of Secondary Impacts

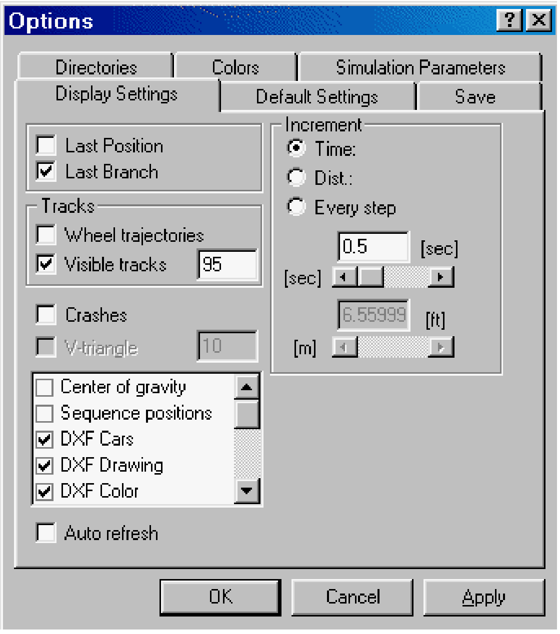
This enables the automatic calculation of secondary impacts when the vehicles overlap by the amount specified in the Depth of penetration drop down list.

Parameters for Secondary Impacts

This area allows secondary impact restitution or separation velocity to be specified differently from the first impact.

Display Settings

This tool is used to change the display settings of the main screen and (for PC-Crash 3D) the 3D Visualization window. See also the description in the previous chapter under **Main Screen Display**.



Last Position

The first, last and present vehicle positions are shown.

Last Branch

The vehicle positions at each specified increment are shown. Selection of this check box displays an additional area in the Display Settings dialog box where the user can specify the increment value with the Time, Distance or Every Step option buttons.

The specified increment influences only what is shown on the screen. It does not affect the calculation frequency.

The specified increment will also be the one used in the Vehicle Dynamics section of the Values window. Last Branch does not have to be activated for the current specified increment value to show in the Values window.

☑ Wheel Trajectories

The paths of all wheels are shown, whether or not they are skidding.

☑ Visible Tracks

Only tire marks that would be visible on the road are indicated. For this, PC-Crash assumes that the marks become visible when the combined longitudinal and lateral tire forces are more than 95% of the available frictional force. The user can alter this default value at any time.

☑ Crashes

The position of all the crashes, with the contact plane, friction cone, and impact impulse are shown.

☑ V-triangle

The velocity vector diagrams, comprising a triangle formed by the pre-impact speed, post-impact speed, and speed change vectors for each vehicle in a collision, are shown on the screen.

In the text box next to ☑ V-triangle the size of the velocity vector diagrams can be scaled. With the default scaling factor of 10, a one meter (3.3 foot) distance on the screen corresponds to a velocity of 10 mph.

Other options available in the Display Settings drop down list are:

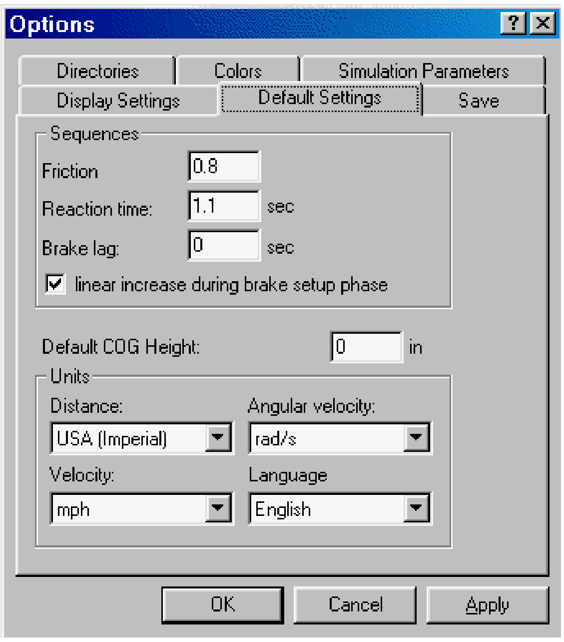
☑ **Center of Gravity**	The path of the center of gravity is shown
☑ **Sequence Positions**	The vehicle positions at the start of each sequence are shown
☑ **DXF Cars**	Allows the vehicles to be represented by 2D or 3D DXF files, if loaded, instead of the simple rectangular shape
☑ **DXF Drawing**	The loaded DXF drawing is shown as a background to the simulation
☑ **DXF Color**	The loaded DXF-drawing is shown in color, rather than black and white
☑ **Bitmap**	The loaded bitmap is shown in the ground plane on the main screen
☑ **Friction polygons**	The outlines of defined friction polygons are shown
☑ **Friction polygon text**	The text referring to defined friction polygons is shown
☑ **Slope polygons**	The outlines of defined slope polygons are shown
☑ **Friction polygon text**	The text referring to defined slope polygons is shown
☑ **Vehicle paths**	Defined vehicle paths are shown
☑ **Crush outlines**	Approximate 2D crush outlines, as defined by the location of the impact point and the overlapping areas of the vehicle outlines, are shown

Optimizer positions	The optimizer intermediate and rest positions are shown
Solid vehicle shapes	3D vehicle DXF shapes are shown in solid, rather than wire-frame, on the main screen
Detailed veh. shapes	Allows the selection of a more detailed vehicle (showing windows and bumpers), on the main screen.
Vehicle outline path	Shows a trace from each corner of the vehicles to define their swept path.
Contact bodies	The ellipsoid bodies for stiffness based contact model are shown
COG path stop pos./intermediate pos	COG position of the intermediate and stop position are linked with lines
POI velocities	Shows the velocities at the point of impact for the crash backward calc..
Momentum mirror method	Display of the momentum mirror method. (scale 0.001:1 m for 1000 Ns)

Auto Refresh

Auto Refresh automatically refreshes the screen when dialog boxes or windows are moved or closed. It also automatically refreshes the 3D window. It is suggested this check box be deactivated only when a slow computer is used.

Default Settings



Default values can be specified for the following items:

Friction

The tire-to-road friction coefficient can be specified here or changed at any time by double-clicking on the "my" area of the status bar at the bottom of the main screen.

Reaction Time

This is the driver perception/reaction time for Reaction sequences.

Brake Lag

The brake lag time for Deceleration sequences is the time between brake pedal application and when the specified braking level occurs at the wheel. A typical panic stop brake lag time is about 0.2s, with longer times for normal stops or air brake systems.

☑ Linear Increase during Brake Setup Phase

This specifies a linear brake force increase from zero to the specified level over the brake lag time. If this is not checked an average of ½ the specified brake force level is assumed during the brake lag time.

Default COG Height

This is the center of gravity height that vehicles loaded into PC-Crash will have. If a height of 0 is kept, the simulation will be a 2D simulation unless the center of gravity height is subsequently changed in **Vehicle - Vehicle Settings - Vehicle Geometry**.

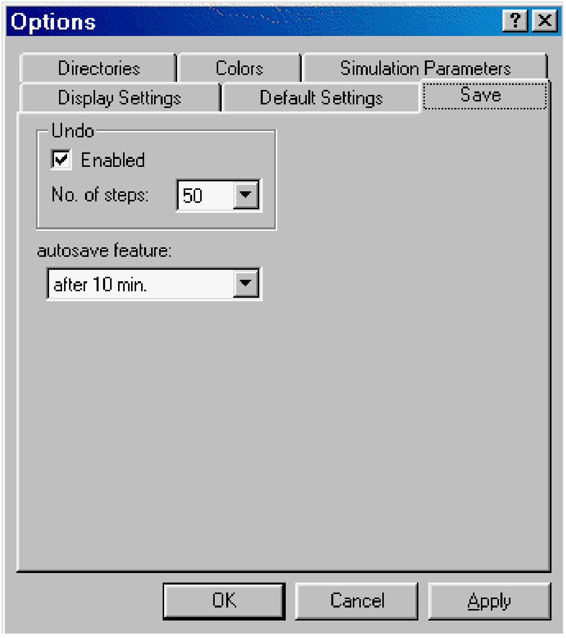
Units

This enables switching between different unit systems. Choose your preferences for distances (metric, U.S.) and velocities (km/h, mph, fps and m/s). Units can be changed at any time when working on a project.

Language

Choose from a number of languages here.

Save



The Save dialog box contains settings for the Undo operation and the Auto save Feature.

Undo

The Undo feature in PC-Crash functions similar to the Undo feature in other programs. Operations can be undone or redone with the "Undo last action" and "Redo last action" buttons . Up to 50 Undo steps can be retained in memory, as selected in the Undo "No. of steps" drop down list.

Autosave Feature

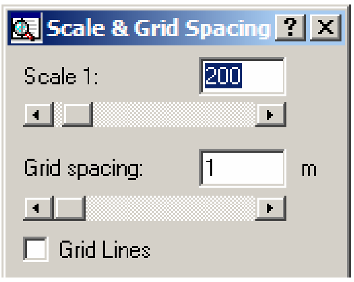
Auto save automatically saves the current project under the name Recover.pro in the Windows\Temp directory (or the directory specified in Window's Control Panel - System) at predefined time intervals (1 to 10 minutes) when PC-Crash is open. In case of software failure, when PC-Crash is re-opened it will ask if you want to load the last session, which will be contained in Recover.pro. After loading Recover.pro, remember to rename it to the desired file name.

Graphics

The Graphics menu includes Scene bitmap operations and scene drawing operations.

Scale & Grid Spacing

This accesses the following dialog box.



Scale

The scale of the screen and printout can be specified here. The scale can also be varied with the Zoom In and Zoom Out buttons  in the main toolbar, or by double-clicking in the "Scale" area of the status bar.

When starting PC-Crash, the default scale is 1:200.

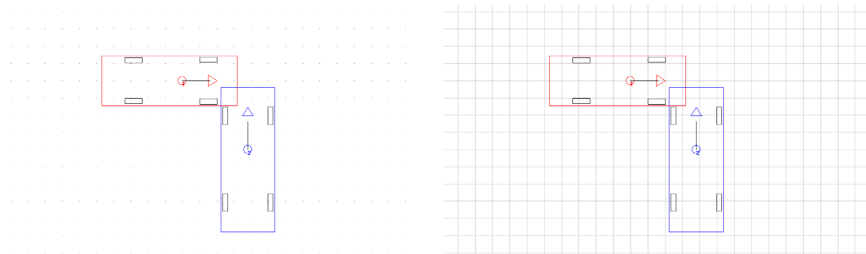
Grid spacing

The grid spacing is specified here, both for the main screen and the Diagrams window. The main screen grid is only visible if the **Grid** menu option (description following) is activated. The Diagrams window grid is only visible if Grid is activated in the Options menu of the Diagrams window.

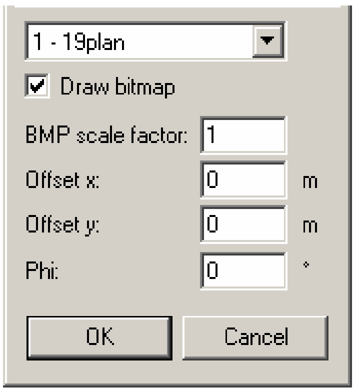
When starting PC-Crash, the default grid spacing is one meter (3.28 feet).

☑ Grid Lines

The default grid shows points only. Selection of this shows full lines instead.



Bitmap



General

PC Crash offers the possibility of working on several bitmaps at the same time. While loading a new bitmap the inquiry takes place whether the current bitmap should be overwritten. The selection of the bitmap can be made either within this window or in the PC Crash work area by clicking with the left mouse button (but however all Buttons must be deactivated in the Toolbar) on the appropriate bitmap.

Draw bitmap

For every bitmap loaded, this option can be turned off to hide the bitmap.

BMP scale factor

Definition of the enlargement of the bitmap. The correct factor can be determined using the menu option **Graphics – Bitmap - Scale**.

Offset x

The active bitmap is shifted in x-direction appropriate the offset

Offset y

The active bitmap is shifted in y-direction appropriate the offset

Phi

The active bitmap is rotated appropriate the angle Phi (positive rotation counter clockwise).

Refresh (F5)

This refreshes the main screen, based on the selected settings in Display Settings .

Zoom Previous (F3)

This changes the main screen area to the previous size.

Zoom Window

Zoom Window allows the user to zoom in on a specific area of the main screen.

This is done by moving the cursor to one corner of the desired area, pressing and holding the left mouse button, moving the cursor to the opposite corner of the constructed rectangle, and then releasing the mouse button.

Zoom All

Zoom All changes the main screen viewing area to include the complete simulation scene, including the bitmap and DXF drawing.

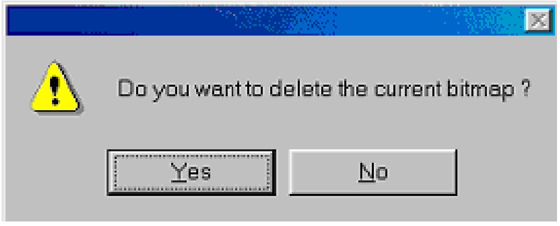
Pan

The Pan tool is used to move the main screen view without changing the scale.

This is done by clicking and moving the cursor on the main screen while holding the left mouse button down, and then releasing it.

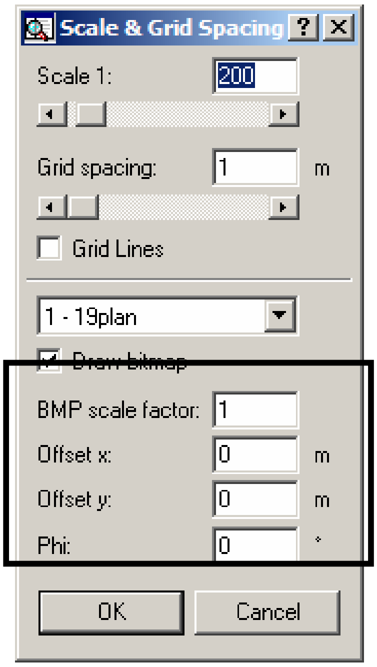
Graphics - Bitmap

This menu is user to perform operations on scene bitmaps, which are loaded using **File – Import - Bitmap** . Multiple scene bitmaps can be loaded. When subsequent bitmaps are loaded, the following message box appears. Select the No button unless you want to replace the existing bitmap.



When more than one bitmap has been loaded, they can be scaled and moved independently. Clicking on the desired bitmap will highlight it, as indicated by the black squares at each corner. The bitmap can then be scaled, moved or rotated independently of the others.

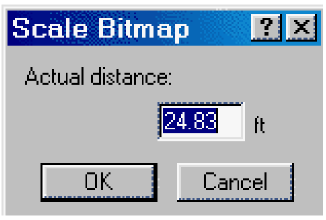
There are two ways of scaling, moving or rotating bitmaps. One is to select **Graphics – Scale & Grid Spacing**. In the lower half of the dialog box that appears, select the desired bitmap from the drop down list and change the values below for it by typing them in the text boxes.



The second method of scaling, moving or rotating bitmaps is to use the following functions in the Bitmap menu.

Scale

Scales the selected bitmap. A known reference distance must be marked in the drawing area. This is done by placing the cursor on one end of the reference distance, then moving the cursor to the other end with the left mouse button pressed. After releasing the mouse button, the following window is displayed, which shows the present length and allows the user to define the correct length.



The procedure may be repeated several times, if an accurate scaling is not achieved the first time.

Move 

This enables the selected bitmap to be moved with respect to the vehicles and drawing in the simulation. Follow the same steps as with the **Graphics** - **Pan** menu option.

Rotate

This enables the selected bitmap to be rotated by a defined angle.

Procedure:

- Put the cursor at the desired center of rotation for the bitmap.
- Press and hold the left mouse button while moving the cursor along any desired reference line in the bitmap.
- After releasing the left mouse button, a reference line is defined. This line can be rotated either way by moving the cursor. The angle of rotation (in degrees) is displayed in the status line.
- When the desired rotation angle is reached, press the left mouse button again. The bitmap will now be rotated by this angle.

***Note:** For every rotation of the bitmap with this menu option, the bitmap's quality will be reduced and its size will be increased. The +90 or -90 degree bitmap rotations in the following two menu options do not result in any loss of bitmap quality.*

Rotate +90 deg

Rotates the selected bitmap 90 degrees counterclockwise.

Rotate -90 deg

Rotates the selected bitmap 90 degrees clockwise.

Grayscale

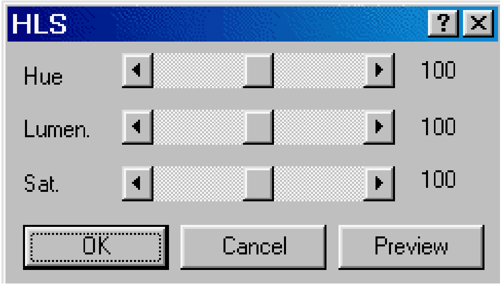
Creates a grayscale picture from a color picture.

Invert

Inverts the colors of the selected bitmap.

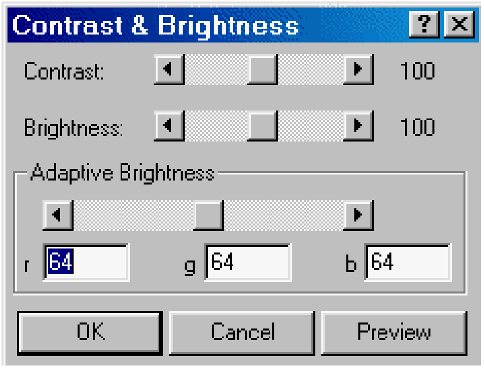
Hue/Lum/Sat

The hue, luminance and saturation of the selected bitmap can be changed with the slider bars in the HLS dialog box.



Contrast/Brightness

The contrast, brightness, adaptive brightness and color of the selected bitmap can be changed with the slider bars and text boxes in the Contrast & Brightness dialog box.



Using the scroll bar for Adaptive Brightness, only brightness and contrast of areas darker than the predefined values are modified.

The Preview button is used to display the changes without modifying the actual picture. This can be done several times. If OK is pressed the actual picture is modified according to the preview.

Delete Bitmap

Deletes the selected bitmap.

Graphics - DXF

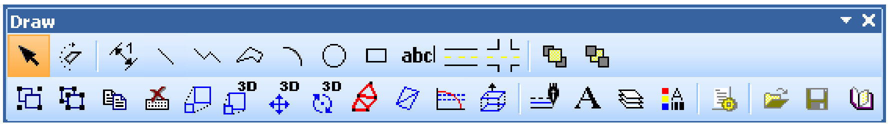
This menu is user to perform scene drawing operations. Scene drawings can also be imported using **File – Import – DXF Drawing** .

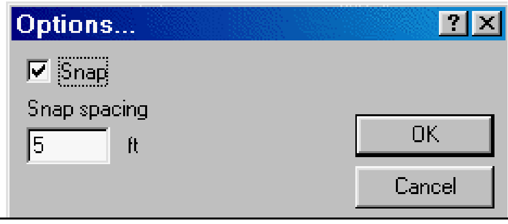
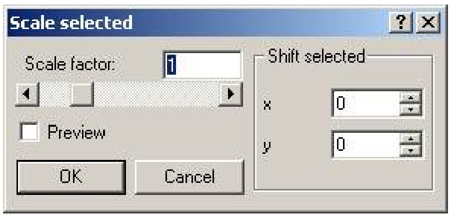
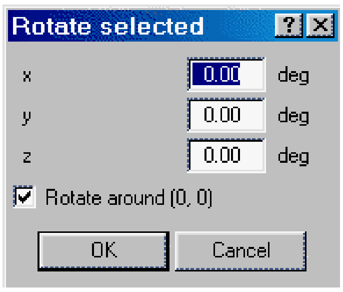
Move 

This enables the drawing to be moved with respect to the vehicles and bitmap in the simulation. Follow the same procedure as with the **Graphics** - **Pan** menu option.

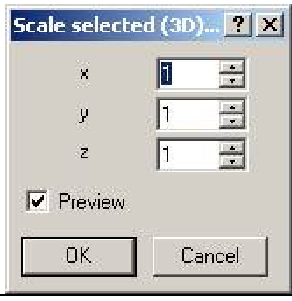
Draw Toolbar

This opens the Draw toolbar. The Draw toolbar contains a number of tools for creating and editing drawings.

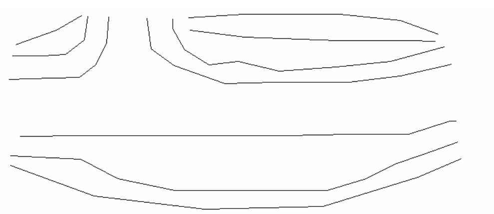


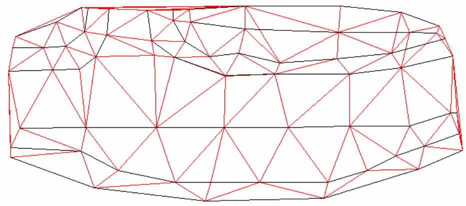
<table>
<tr><td>Snap</td><td>Snap and Snap Spacing are selected here. When drawing with any drawing tool, and also when drawing Friction Polygons, Slope Polygons and Vehicle Paths, the drawn point will snap to the selected spacing.

</td></tr>
<tr><td>Scale</td><td>The selected drawing components can be scaled and moved here using x and y coordinates. Selecting Preview draws the changes immediately after changing. Instead of entering the values in the text boxes you also can change them using the spin buttons.

</td></tr>
<tr><td>Select all</td><td>Selects all drawing components.</td></tr>
<tr><td>Rotate selected (3D)</td><td>The selected drawing components can be rotated about the X, Y or Z axes here.

</td></tr>
</table>

<table>
<tr><td>Shift selected (3D)</td><td>The selected drawing components can be shifted along the X, Y or Z axes. Selecting ☑ Preview draws the changes immediately after changing. Instead of entering the values in the text boxes you also can change them using the spin buttons.

</td></tr>
<tr><td>Scale selected (3D)</td><td>The selected drawing components can be scaled along the X, Y or Z axes separately. Selecting ☑ Preview draws the changes immediately after changing. Instead of entering the values in the text boxes you also can change them using the spin buttons.

</td></tr>
<tr><td>Insert object

Save selected</td><td>DXF drawings (*.dxf, *.wrl, *.idf) can be loaded as "objects". A DXF drawing object can be moved as a unit, independently of the main DXF drawing

The selected drawing components can be saved as a DXF drawing (*.dxf, *.idf, *.wrl)</td></tr>
<tr><td>Triangulate selected</td><td>3D surfaces can be created from a drawing containing 3D points. This feature is useful for creating 3D scenes from total station survey DXF files or contour lines drawn in PC-Crash.
</td></tr>
</table>

	 After selection of the desired points and activation of this feature, 3D triangular surfaces are created between the selected points. A massage box will appear asking if the triangulated area should be used as slope polygon. If you choose Yes the triangulated area will be used as slope polygon and can be assessed using **Dynamics – Define Road Slope**. If you choose No the triangulated area is handled as DXF drawing, once the surfaces are created, use **File – Export – DXF Drawing** to save the drawing. It can then be imported as a 3D Road Object using **Dynamics – Define Road Slope – Generate 3D Road Object** – **Import**. The vehicles in the simulation will now recognize the drawing as a 3D road object. To set the initial conditions properly, place the vehicle on the road object with the Tow Truck tool and press the New Simulation button .

The following tool buttons are contained in the Draw toolbar:

Select, Move

This button is initially active when the Draw toolbar is opened. This enables the selection of existing drawing objects by clicking the cursor on the object with the left mouse button. After selection, objects can be moved, modified or scaled. Multiple elements can be selected by framing all objects within a window drawn with the mouse, or by holding down the SHIFT key while selecting additional objects.

Modifying Objects - If only one single object is active, it can be modified in shape by moving the corner points (marked with small squares) with the left mouse button pressed.

Moving Objects - If one or several objects are active, they can be shifted to a different location by putting the cursor on the object, clicking and holding the left mouse button, and moving the object to the new location.

Selecting all Objects - If the whole drawing is to be moved, rotated or scaled, all objects can be selected using the menu option **Drawing - Select All Objects**.

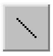
Rotate Selected

Operation: When this button is pressed, the symbol , which identifies the center of rotation, appears. Click and hold the left mouse button with the cursor on this symbol to move the center of rotation. Rotate the selected object(s) by clicking on any point outside this symbol in the drawing area and moving the cursor with the left mouse button pressed. The new position of the objects is displayed immediately.

Measure Line

Creates a dimension line of any length and direction.

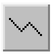
Operation: Mark the starting point of the line with the left mouse button pressed and held. Move the cursor to the opposite end of the line and release the left mouse button. If you simultaneously press the SHIFT key, the angle of the line will always be 0, 45 or 90 degrees.



Line

Creates a line of any length and direction.

Operation: Click and hold the left mouse button to draw the first point and release it at the second point. If you simultaneously press the SHIFT key, the angle of the line will always be 0, 45 or 90 degrees.



Polyline

Creates an open polyline, consisting of several line parts.

Operation: Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. If you simultaneously press the SHIFT key, the angle of the line will always be 0, 45 or 90 degrees. A single click of the right mouse button completes the polyline. To add additional points to a poly line, select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

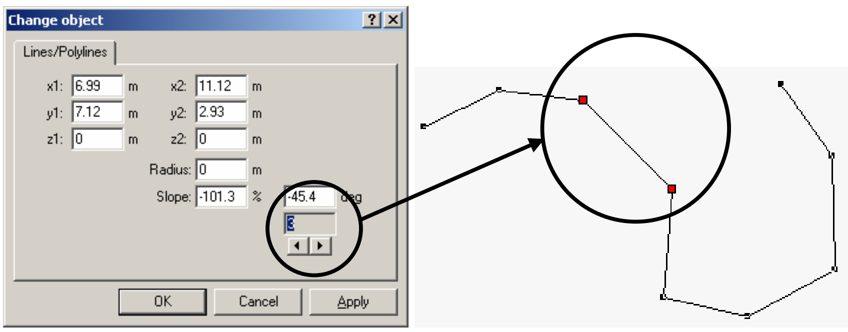
Polygon

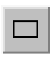
Creates a polygon, consisting of several line parts.

Operation: Click and hold the left mouse button to draw the first point and release it at the second point. Click at the desired locations to define subsequent points. After entering all points of the polygon, click the right mouse button to close the last side of the polygon. If you simultaneously press the SHIFT key, the angle of the line will always be 0, 45 or 90 degrees. A single click of the right mouse button closes the polygon. To add additional points to a polygon select it and click with the right mouse button on the desired location. A popup menu appears that allows you to insert or delete a point at the chosen location.

Note: *Individual points of Lines, Polylines and Polygons can be adjusted using the*  *button (they must be highlighted first) to access the Change Object window.*

The X, Y and Z coordinate of the ends of each line segment can be changed, and a radius can be applied between each line segment. Alternatively a slope between the points can be defined, the second and the following points are adjusted. The current line segment, denoted by a number in the right portion of this dialog box, is highlighted with red endpoints on the main screen.





Rectangle

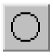
Creates a square or rectangle.

Operation: Mark the starting corner of the square or rectangle with the left mouse button pressed and held. Move the cursor to the opposite corner and release the left mouse button.

Arc

Creates an arc of a circle.

Operation: First, the full circle must be created. Mark the starting corner of the circle with the left mouse button pressed and held. Move the cursor to the opposite corner and release the left mouse button at the desired circle size. Then, define the beginning and end angles of the arc segment with single clicks of the left mouse button.



Circle

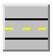
Creates a circle or ellipse.

Operation: Two selected points always define the surrounding rectangle of the circle or ellipse. Mark the starting corner of the circle or ellipse with the left mouse button pressed and held. Move the cursor to the opposite corner and release the left mouse button. If you simultaneously press the SHIFT key, a circle is automatically created.

Text

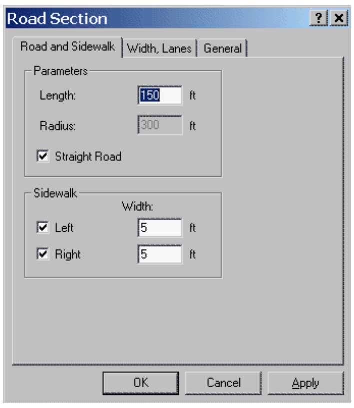
Enters text.

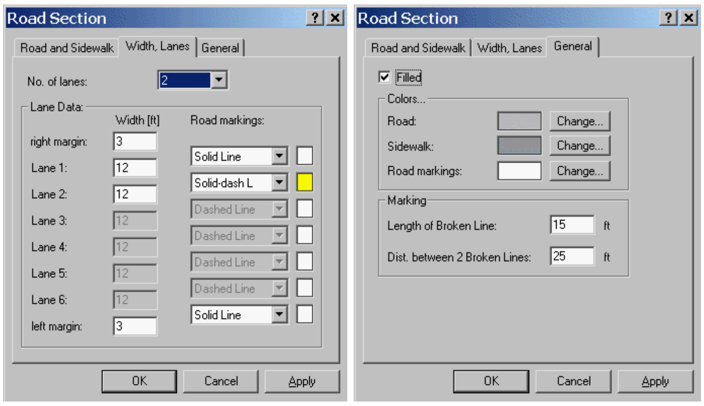
Operation: Mark the position where the text is to be entered in the drawing area by clicking the left mouse button. Then, type in the desired text.



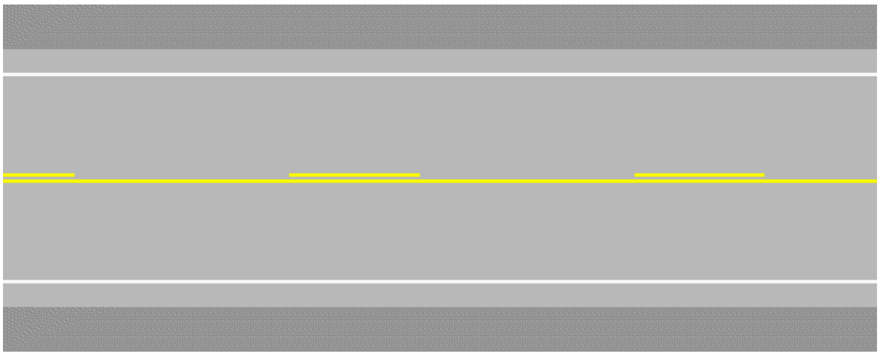
Generate Road Element

Draws a straight or curved road section. Road parameters are defined in the three folders of the Road Section dialog box.





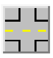
Up to 6 lanes can be chosen. Different colors can be chosen for the different lines and road components. The dashed line spacing can be changed. The Filled check box allows the road to be shown in full color (below), or in outline only.



Road sections can be easily joined to one another or to intersections (see next item) by moving the section so that the red line at the end is near the red line on the end of the other section or the desired street of an intersection. The road section will snap into place at the correct angle.

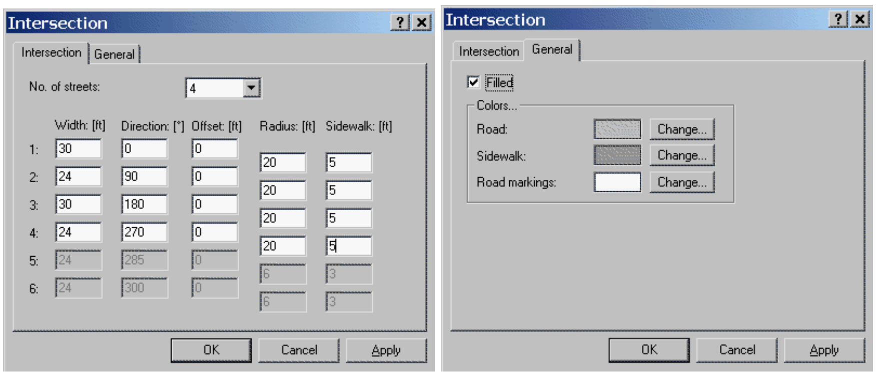
Properties can be redefined at any time by double- clicking on a line or edge of the desired road section or intersection.

Note: *Sidewalks are raised above the road level in the 3D view for a realistic appearance only. They do not define a true elevation change that will affect vehicle dynamics. That must be done using the menu option* ***Dynamics - Define Road Slope****.*

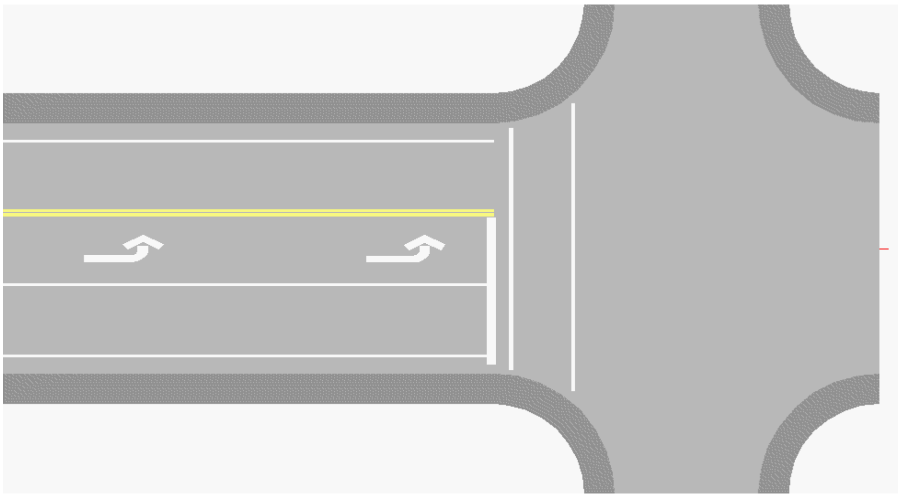


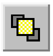
Generate Intersection

Draws an intersection, using the Intersection dialog box. An intersection with up to six streets can be created.



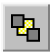
After the roads and intersections are created, lines and symbols can be added using the other Drawing Toolbar tools. In the following figure, crosswalk lines have been drawn and left turn symbols from the symbol library have been added.





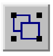
Bring to Front

This and the following tool work between lines only. This tool brings the selected line to the foreground.



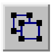
Move to Back

Puts the selected line in the background behind other lines.



Group

Groups selected drawing components into objects, which can then be moved, rotated or scaled as one.



Ungroup

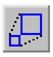
Ungroups selected drawing objects.

Copy Selected

A copy is made of the selected objects. The copy will be attached to the cursor, and can be inserted by moving it to the desired location and clicking the left mouse button.

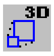
Delete Selected

All selected objects can be deleted with this tool or by pushing the DELETE key.



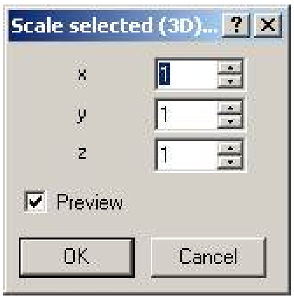
Scale Selected

The selected drawing components are scaled by the amount in the Scale Factor text box. Additionally the selected drawing components can be moved by the amount in the text boxes. The Preview Check box applies all changes immediately to the selected drawing components when you change a value.



Scale Selected 3D

The selected drawing components can be scaled along the X, Y or Z axes separately. Selecting ☑ Preview draws the changes immediately after changing. Instead of entering the values in the text boxes you also can change them using the spin buttons.

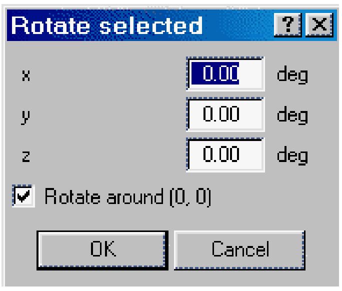


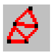
Shift Selected 3D

The selected drawing components can be shifted along the X, Y or Z axes. Selecting ☑ Preview draws the changes immediately after changing. Instead of entering the values in the text boxes you also can change them using the spin buttons.

Rotate Selected 3D

The selected drawing components can be rotated about the X, Y or axes here.



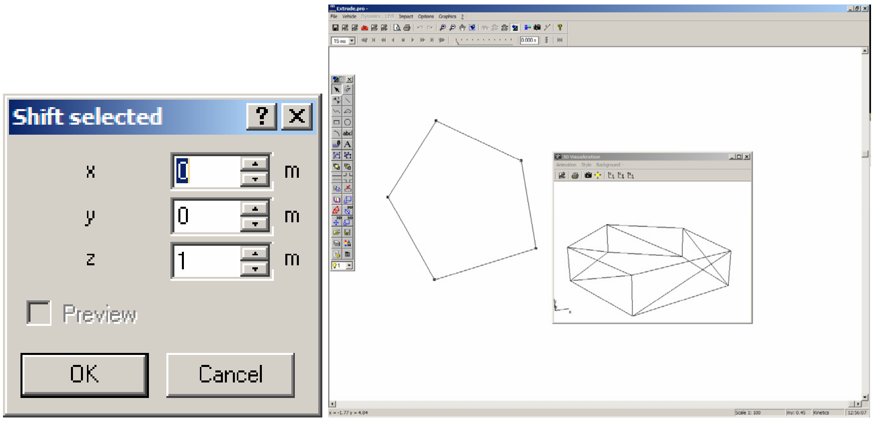


Triangulate selected

3D surfaces can be created from a drawing containing 3D points. This feature is useful for creating 3D scenes from total station survey DXF files or contour lines drawn in PC-Crash.

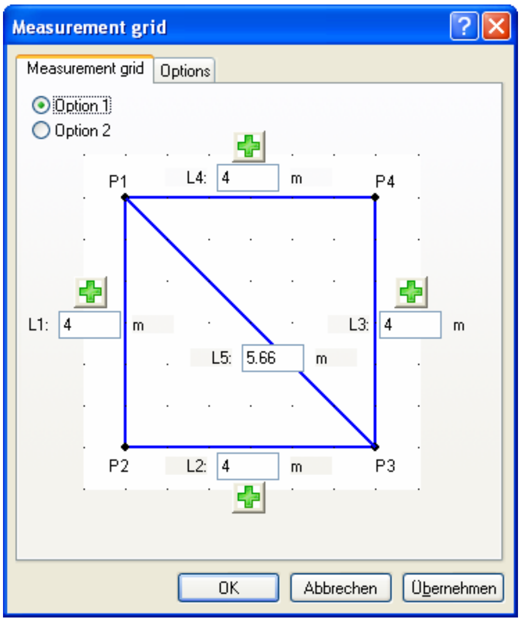
Extrude selected

This operation extrudes a poly-line or polygon along a given vector. Extruding means that a copy of the line or polygon is moved by the vector and the copy and the origin are connected by triangulated surfaces.



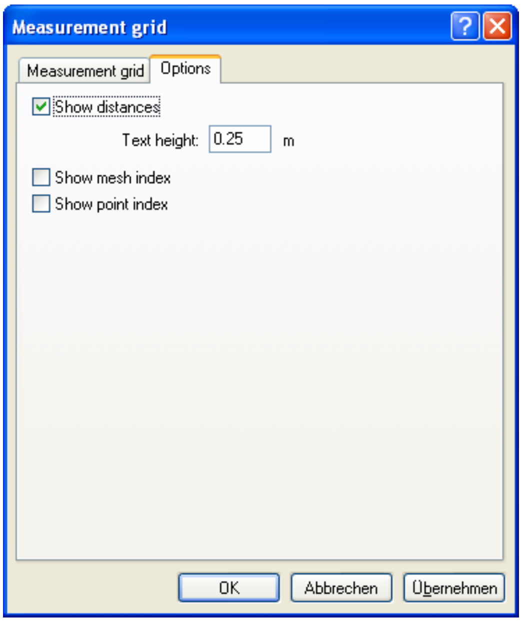
Measurement grid

This operation enables the definition of a four point measurement grid. Activating of this menu option loads a standard four point grid. Double clicking on the grid opens the “Measurement grid”-window.



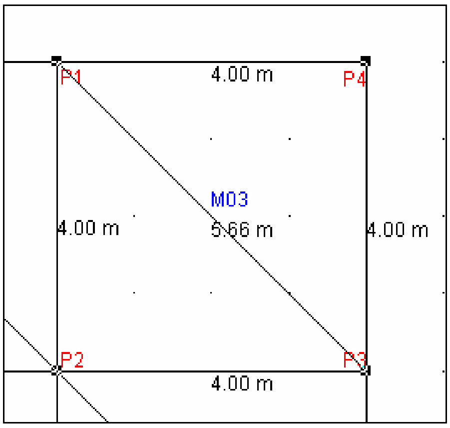
Within this window the distances for the grid can be defined. With **Option 1** or **Option 2** the diagonals are defined. The selection box is for selection of the number of the measurement grid.

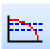
With  Insert a new grid is defined, this new grid is inserted at the appropriate position. Therefore this distance is predefined.



The **Options** window is for activating / deactivating the display of the distances and for defining the text height.

Additionally the **mash index** and the **point index** can be displayed.

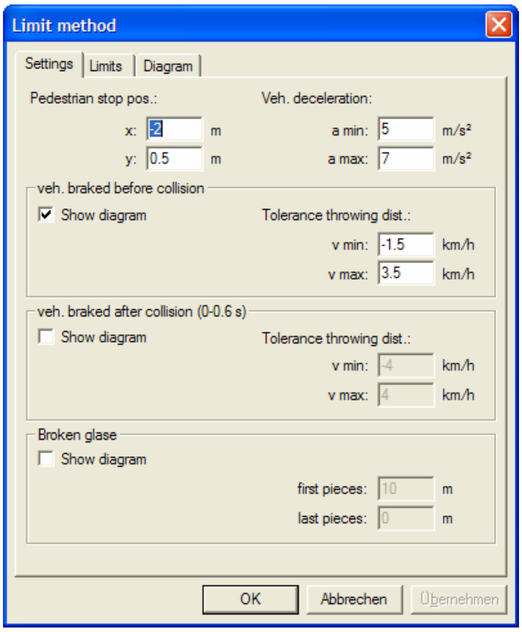




Limit method

The limit method is a graphical method to determine the impact velocity and the impact location for vehicle / pedestrian accidents based on tire marks, the stopping distance of the vehicle and the pedestrian throwing distance. In this method distance limits or corridors as well as velocity limits or corridors can be specified to determine where the impact actually occurred. The calculation is based on the stop position of the vehicle and the rest position of the pedestrian.

After generating a diagram within the drawing program using the menu option “Limit method” in the menu indicated below, the menus has to be moved using the arrow in such a way, that the origin of the diagram corresponds to the rest position of the vehicle. It’s suggested, that the front of the vehicle is used as corresponding location.



In the “Limit method-Settings” window (activated via double click on base line of diagram) the user has to specify the “Pedestrian stop pos.” relative to the stop position of the vehicle in the window “Settings”. Positive values in the x direction are stop positions before the stop position of the vehicle, negative values for the x direction of the pedestrian stop position are stop positions after the stop position of the vehicle. The y position of the pedestrian stop position does not influence the calculations this setting is only used to show where the pedestrian stop position was in the real case.

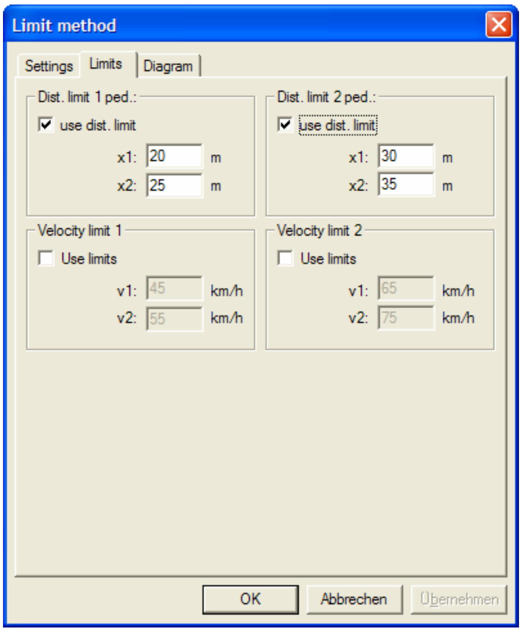
For the vehicle the minimum and maximum deceleration has to be specified and the velocity vs distance diagram is shown based on these decelerations in red color. This diagram starts at the stop position of the vehicle.

For the pedestrian either the throwing distance diagram for collisions where the vehicle was braked at impact already or the diagram for collisions where the vehicle was braked immediately after the collision (0-0.6 s) can be shown or both diagrams can be displayed if the user is not sure whether the vehicle was braked before the collision or not. For each of these two diagrams a tolerance can be specified as velocity tolerances for the throwing distance diagram. The diagrams for vehicles braked before the collision are shown in green color, the light green color is the throwing distance diagram for vehicles braked at collision and the darker green diagrams are the diagrams based on the tolerances specified.

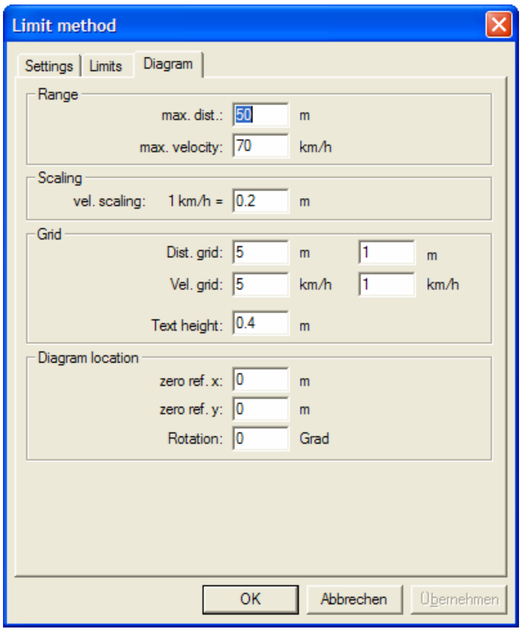
For vehicles braked after the collision the diagrams are shown in blue color, the light blue diagram is the throwing distance diagram without tolerances and the darker blue diagrams are the diagrams with the specified tolerances.

The pedestrian throwing distance diagrams start at the stop position of the pedestrian and these diagrams are shown together with vehicle velocity diagrams.

In addition a diagram for the throwing distance of glass particles can be shown. The position of the first and last glass particles found at the accident scene can be specified relative to the stop position of the vehicle.



In the limits section 2 distance limits and 2 velocity limits can be specified. These limits can be used to define where possible areas for the pedestrian are to cross the road or to limit the velocity of the vehicle. For these limits the intersection of the distance or velocity ranges is calculated together with the vehicle velocity diagram and the throwing distance diagrams and the intersecting area is shown in the diagram as a hatched area in blue color. The velocity ranges are displayed on the velocity axis of the diagram if a distance limit is specified, if a velocity limit is specified the distance ranges are displayed on the distance axis. In this way the impact velocity ranges can be calculated based on a range of possible impact locations or the impact location can be determined based on a velocity range specified.

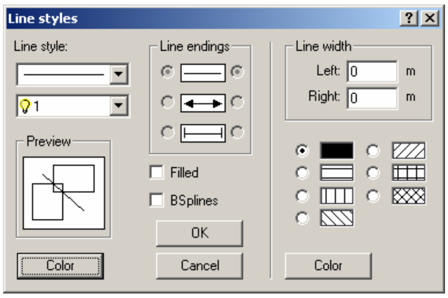


In the diagram dialog the range for the diagram can be specified as a maximum distance and velocity, the velocity scaling can be specified as meter per kph, the grid spacing and text height can be set for solid grid lines and for the dashed grid lines to grid spacing can be entered. The diagram position and rotation can be set in global coordinates as well.

Change Line Style

Changes line styles, layers, colors and other object features.

Operation: After pressing this button, the following dialog box appears:



All changes done in this dialog box refer to objects that have been selected. If this menu option is chosen when no object is selected, the changes are for all future drawing operations.

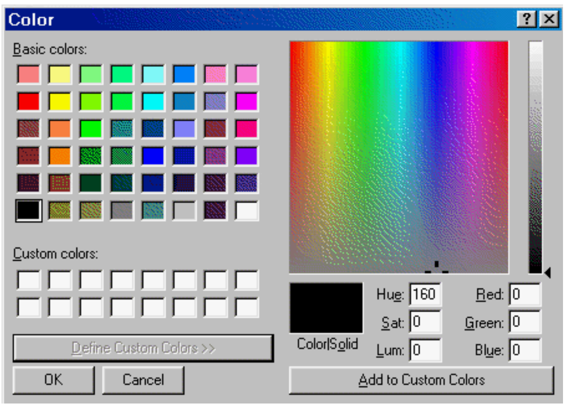
Different line styles can be selected, the layer can be selected, and Line ends can be changed to different types. Line widths can be changed. The total width of the line is the sum of its left and right halves. There are several possibilities for wide fill patterns. These patterns can be applied to lines, poly-lines (including arcs) and polygons, even when the BSplines checkbox is activated.

 **Filled** fills rectangles, ellipses or polygons with a solid color.

 **BSplines** changes straight segments of polylines and polygons to curved segments.

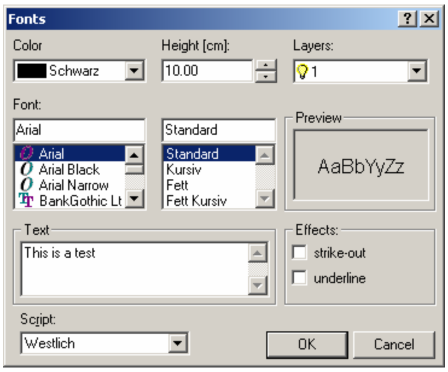
Color Buttons - the colors of all chosen objects can be changed in the Color dialog box that these buttons access. The left Color button is for all objects and lines. The right Color button is for the line fill, if a Line Width > 0 has been selected. To select a new color click on the desired box. By selecting the Define Custom Colors button, additional colors can be defined.

Note: *The 5th color in the 2nd row is a transparent aqua color, useful for vehicle windows.*



Change Font, Text Color

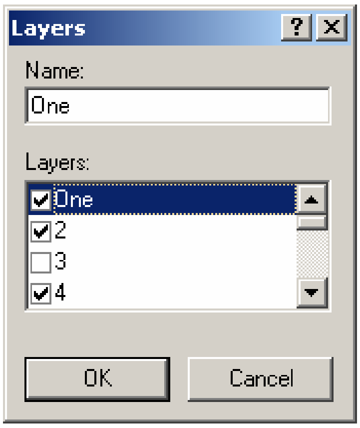
Modifies text. Text can also be modified by double-clicking on it. The following dialog box is displayed:



This dialog box allows the user to change the text color, size, the layer, font and content. Text several lines long can be entered in this window.

Layers

The Name of the Layers can be defined (select the Layer in the Layers box and change the text in the Name textbox). The name will be updated in all other windows. Within the Layers box different layers can be activated/deactivated.

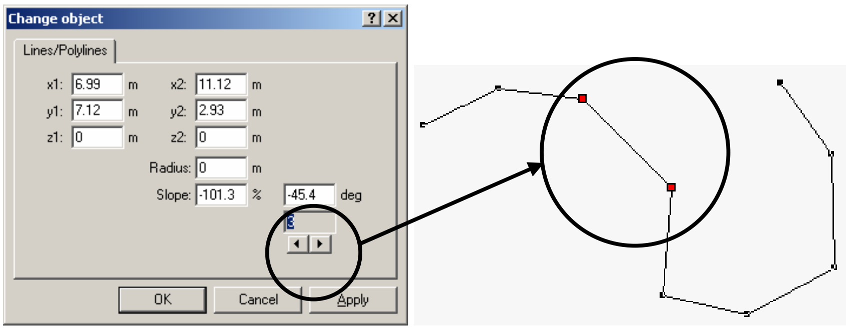


The created drawings and text are relevant for the layer selected in the Layer selection box  at the bottom of the Draw Toolbar. If the layer should be changed select the relevant drawings and use the Line styles window ( change line styles) to change the layer. For text use the Fonts window ( change font, text color) to change the layer.

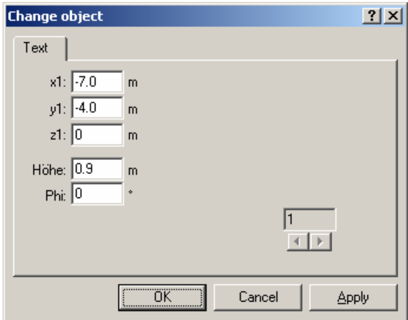
Change objects

Individual points of Lines, Polylines and Polygons can be adjusted using the Change Object window (they must be highlighted first).

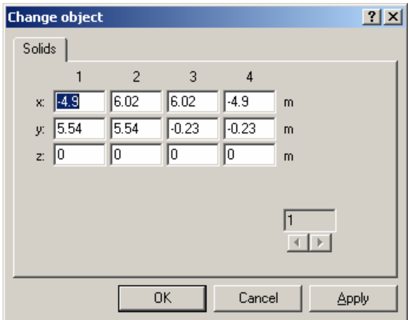
The X, Y and Z coordinate of the ends of each line segment can be changed, and a radius can be applied between each line segment. Alternatively a slope between the points can be defined, the second and the following points are adjusted. The current line segment, denoted by a number in the right portion of this dialog box, is highlighted with red endpoints on the main screen.



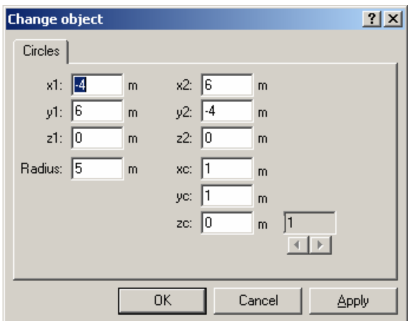
Text can be positioned, scaled and rotated.



Solids:

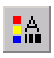


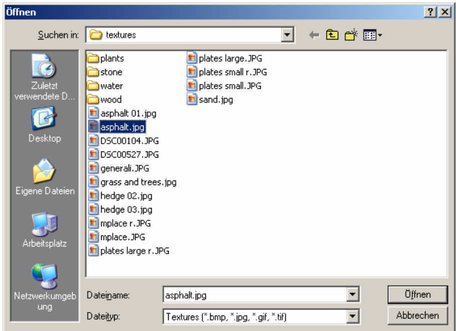
Circles can be defined using the enveloping rectangular or by selecting radius and center.



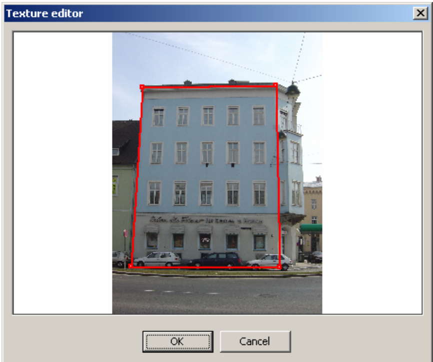
Using textured rectangles in 3D window

Rectangles can have assigned a bitmap texture to use realistic appearing objects, like houses, walls, asphalt or grass areas, in the 3D window.

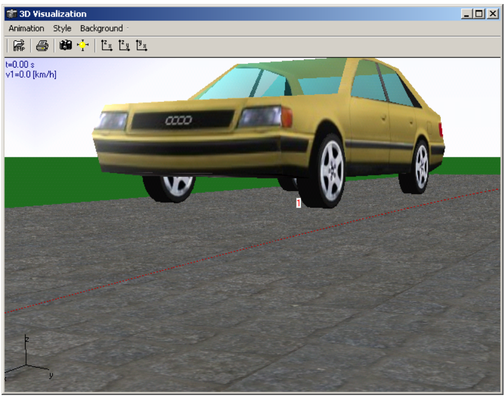
To assign a bitmap texture to a rectangle select the rectangle and press the button  **Change objects**.



In the input field **Texture file** the bitmap file can be specified. If only a part of the file is to be used for texturing the rectangle the button **Edit** can be used.



With the check box **Tile** a texture can be tiled on a rectangle. A reasonable application of tiling a texture is e.g. large asphalt or grass areas repeating always the same pattern. Instead of creating a similar large texture the width and height of a single tile can be entered in the input field **cx** and **cy**.



Snap

☑ Snap and Snap Spacing are selected here. When drawing with any drawing tool, and also when drawing Friction Polygons, Slope Polygons and Vehicle Paths, the drawn point will snap to the selected spacing.

 *Insert object*

DXF drawings (*.dxf, *.wrl, *.idf) can be loaded as "objects". A DXF drawing object can be moved as a unit, independently of the main DXF drawing

 *Save selected*

The selected drawing components can be saved as a DXF drawing (*.dxf, *.idf, *.wrl)

 *Symbol Library*

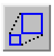
Drawing symbols from a library can be loaded. Refer to the description under the **Drawing – Symbol Library** menu option.

Load Object

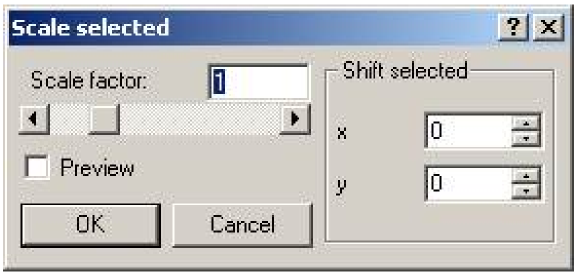
DXF drawings can be loaded as "objects". A DXF drawing object can be moved as a unit, independently of the main DXF drawing.

Select All Objects

All objects on the screen, including the main DXF drawing, can be activated for moving, rotating or performing other modifications as a whole.

Scale Objects 

The selected drawing components are scaled and moved by the amount in the appropriate text boxes.

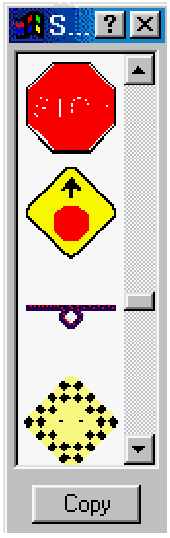


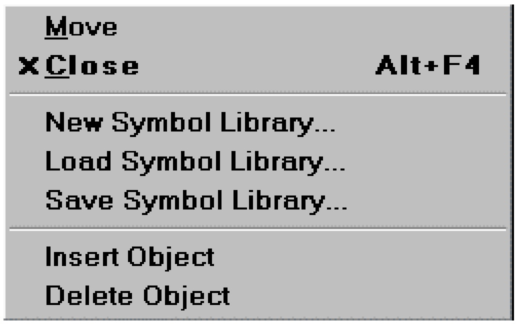
Delete Object 

All selected objects can be deleted with this menu option, the Delete Selected tool or by pushing the DELETE key.

Symbol Library 

Drawing symbols from a library can be loaded. This opens a window in which different drawings are contained. The desired drawing can be loaded as a symbol by selecting it with the left mouse button and pushing the Copy button at the bottom of this window. The symbols are active after loading and can be moved immediately.





Right clicking in the title bar of the Symbol Library window accesses a menu that enables the following:

Move	Move the Symbol window
Close	Close the Symbol window
New Symbol library	Create a new symbol library
Load Symbol library	Load another symbol library
Save Symbol library	Save the symbol library under a specified file name
Insert Object	Add a drawing to the symbol library
Delete Object	Delete the selected drawing from the symbol library

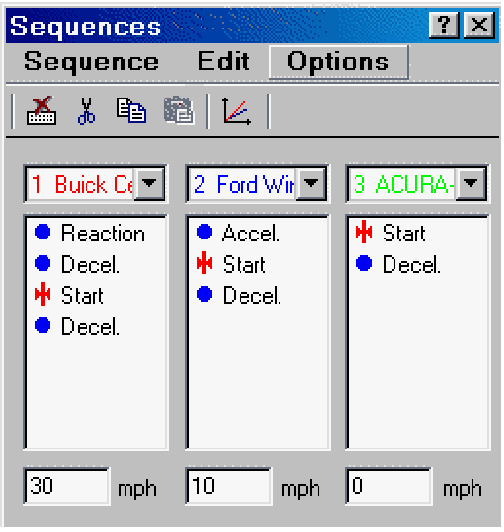
Delete Drawing

Deletes the loaded DXF drawing(s).

Chapter 4
Programming Sequences

General

One of the major advantages of PC-Crash lies in its ability to define driving situations dynamically. This is done using the menu option **Dynamics** - **Sequences** (F6), which opens the Sequences dialog box.



The Start icons for each vehicle in the Sequences dialog box represent time zero. As a default condition, Reaction and Braking sequences are placed before Start (which is usually the time of impact), with one braking sequence after Start (as shown for Vehicle 1, above). All sequences above Start are used for a backward simulation (before time = 0). All sequences below Start are used for a forward simulation (after time = 0).

Any number of sequences can be selected for each vehicle in the simulation. When a new sequence is selected from the menu at the top of the sequences dialog box, it is inserted below the current active (highlighted) sequence.

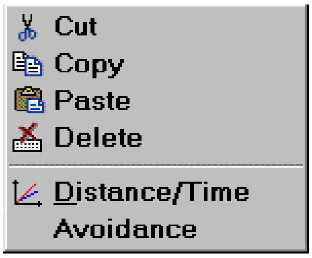
To view and alter the parameters of a particular sequence, open its dialog box by double-clicking on the sequence's icon.

Note that if a sequence's dialog box is open, that sequence cannot be moved or deleted.

In cases where there are multiple sequences of the same type it is possible to switch between the different sequences by double-clicking on the corresponding icon. The values in the window will change to the new sequence and can be modified.

Sequence Toolbar

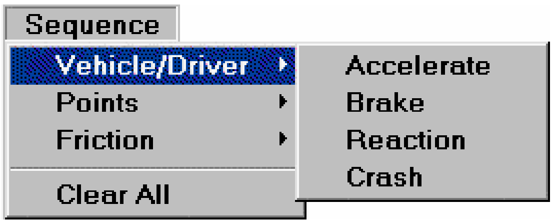
The sequence toolbar contains tools for deleting, cutting, copying and inserting sequences, and for accessing the Kinematics diagram feature. These functions can also be accessed by right-clicking on a sequence to bring up this menu.



Following are detailed descriptions of all sequence dialog box menu items.

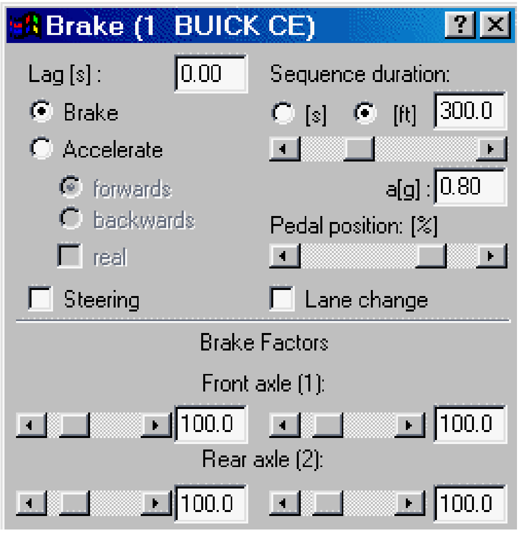
Sequence

Vehicle/Driver



Accelerate and Brake/Accelerate

The Accelerate and Brake sequence dialog boxes are the same, and are used to define all values associated with acceleration, braking or steering.



Brake or Accelerator Lag Time

This is the time it takes to reach the specified brake or acceleration level, from the previous brake or acceleration level. Either a constant level of the average between the previous and new level, or a linear increase from the previous to the new level occurs, depending on the setting in **Options – Options – Default Settings**. The default lag time is 0.20 seconds, a reasonable pressure build-up time for a panic stop with a modern hydraulic brake system.

Time/Distance

These option buttons are for specifying whether the length of the present sequence is to be measured in time or distance units. The scroll bar and edit box are for specifying the value of the time (in seconds) or the distance (in feet or meters).

Brake

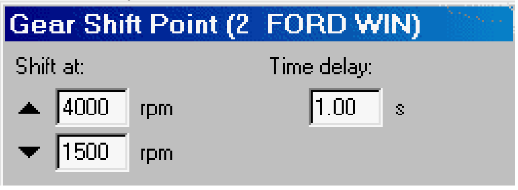
The Brake and Accelerate dialog boxes can be changed from one to the other at any time by choosing the respective option button.

The vehicle deceleration or acceleration can be defined by either specifying the individual factors at the wheels or by specifying the acceleration with the Pedal position slider bar or Acceleration text box.

If the individual brake factors are defined, the acceleration will be calculated and displayed in the Acceleration text box.

If the acceleration is specified the necessary brake or acceleration force is evenly distributed among all wheels except as follows:

- When braking is specified in a 3D simulation, the brake force distribution specified in **Vehicle - Vehicle Settings - Rear Brake Force** will be used.
- When ☑ Real Acceleration is activated, the parameters selected in **Vehicle - Engine/Drivetrain** will be used. The gear shift rpm and the shift delay will also have to be specified in this case, in the Gear Shift Point dialog box.



  The Shift Down text box (the lower one) applies when a vehicle is going up a slope steep enough to cause it to decelerate. This is useful for large trucks ascending steep grades.

The Brake and Accelerate dialog boxes are closed with the close symbol in the title bar.

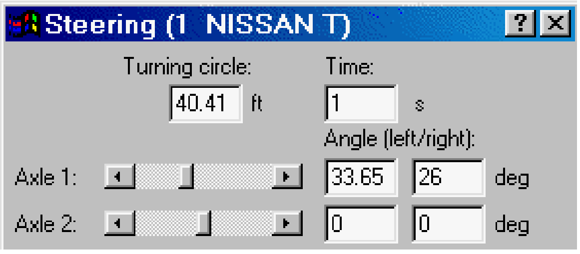
Note: *The wheel brake and acceleration forces are shown in percent of static vertical wheel load. In a 2D simulation a brake factor of 100% represents a fully locked wheel for coefficients of friction up to 1.0. In a 3D simulation the dynamic weight shift will increase the vertical loads on some wheels, in which case a brake factor of greater than 100% may be required to lock the wheel, depending on the coefficient of friction.*

For a forward traveling vehicle in a 3D simulation with a fairly high coefficient of friction, a brake factor of 100% will not lock the front wheels, but will lock the rear wheels. This causes an unstable driving condition in PC-Crash (and in the real world), in which the vehicle will tend to "switch ends".

To avoid this situation when it is known the leading wheels of the actual incident vehicle are locked, the 3D simulation vehicle's leading wheels must be "over-braked". This is done by specifying a brake factor of more than 100% for the front wheels (PC-Crash allows values as high as 500%). This can also be done by simply using the Pedal position slider bar to specify braking. This assigns front/rear braking power *according to **Vehicles – Vehicle Settings - Rear Brake Force**, which applies higher brake forces to the front and lower brake forces to the rear to prevent an unstable condition, similar to a real vehicle's brake proportioning valve.*

☑ Steering

This opens the Steering dialog box, in which the steering application time and steering angle can be specified. De-activating this check box will cancel all steering parameters.



Turning circle

This is the turning circle diameter of the center of the outside front wheel. If the steering angle of the outside front wheel is entered, the turning circle will be displayed. If the turning circle is entered, the steering angle of both front wheels is displayed. In this case, the steering angles for axles 2 and up are assumed to be zero. The Ackermann steering angles of the front two wheels are based on the geometry of the vehicle.

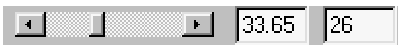
Steering Time

The steering time represents the time it takes to steer the vehicle from the last steering angle specified in a previous sequence (or zero if there was no previous sequence with steering) to the new angle specified.

A linear change of the steering angle over the specified time is assumed.

Steering Angle

Definition of the steering angle can be specified individually for each wheel so that wheels steered due to impact damage can be modeled. Normal Ackermann front wheel steering can be specified in one of two ways:

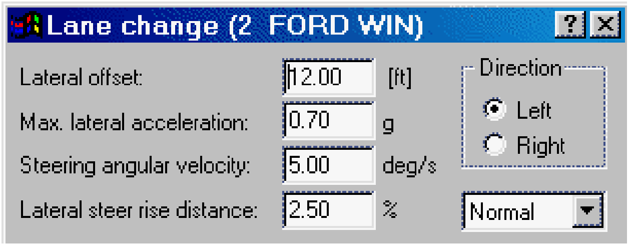
1. Use the slider bar 

. Clicking once in the space between the arrow buttons changes the outside wheel angle by 1° and clicking on an arrow button changes it by 0.1°.
2. Specify the angle of the outside front wheel and then hit the TAB key until the text box for the inside wheel is reached. Its angle will then change automatically to the correct value.

The wheels of the vehicle on the main screen and in the 3D window will turn to match the entered angle.

☑ Lane Change

This opens the Lane Change dialog box, in which a lane change maneuver can be specified. De-activating this check box will cancel the lane change maneuver.



Lateral offset

This is the total lateral movement required to complete the lane change.

Max. lateral acceleration

The specified lateral acceleration will not be exceeded while the steering wheel angle is being increased at the start of the lane change maneuver. The Steering Angular Velocity will drop to zero when this lateral acceleration is reached, and the specified Lateral Steer Rise Distance will be shortened.

Steering angular velocity

This is the angular velocity at which the front wheel steering angle is being changed.

Lateral steer rise distance

The Lateral Steer Rise Distance, expressed as a percentage of the Lateral Offset, is the lateral distance the vehicle moves while the steering angle of the front wheels is being increased.

Default settings for the Lateral Steer Rise Distance can be specified in the drop down list next to this text box as follows:

- Abrupt 4.0%
- Normal 2.5%
- Smooth 1.0%

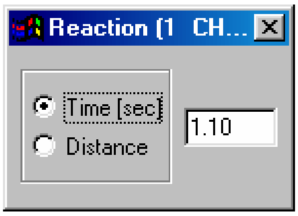
Direction

This specifies a lane change to the vehicle's left or right.

Refer to the **Technical Manual** for more information on the lane change feature.

Reaction

The Reaction sequence is for the definition of a perception-reaction time or distance.

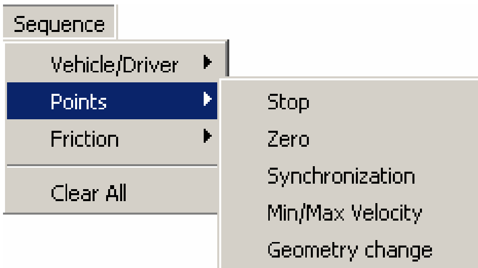


During the reaction sequence the vehicle is moving with its previously defined brake force or acceleration. Also the steering angle from the previous sequence is kept.

Crash

The Crash sequence allows a speed change to be specified anywhere along a vehicle's path, without using the program's crash model. This is useful for specifying small impacts (such as curb impacts) for which the impact speed change is estimated from vehicle damage.

Points



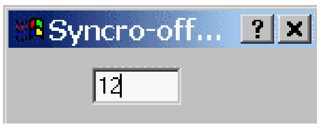
Stop

The Stop sequence stops a vehicle at the end of the previous sequence.

Zero

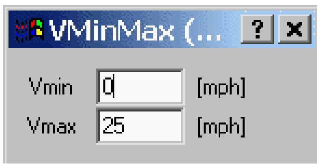
Time and distance are set to zero at the position where the Null point Sequence is placed. This sequence influences only the diagrams.

Synchronization

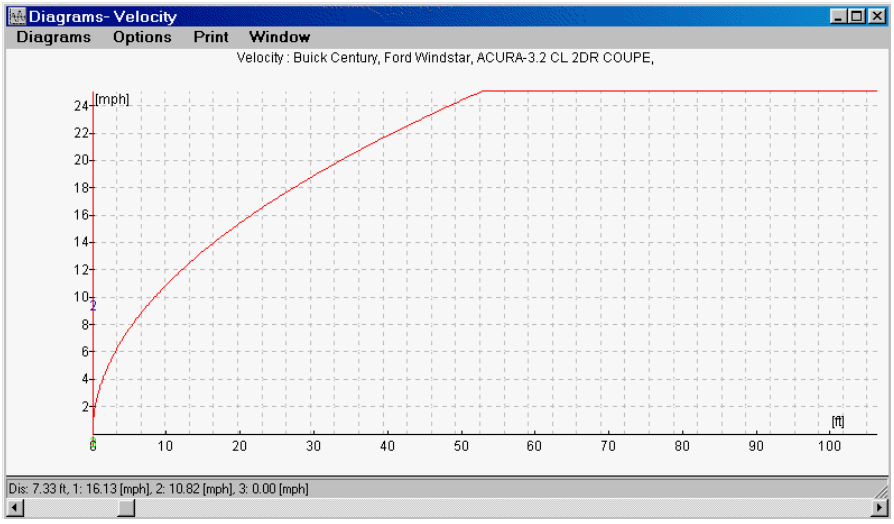


This offsets the distance graph for one vehicle from another in the Diagrams window. This sequence should be the first sequence listed under the Start sequence.

Min/Max Velocity



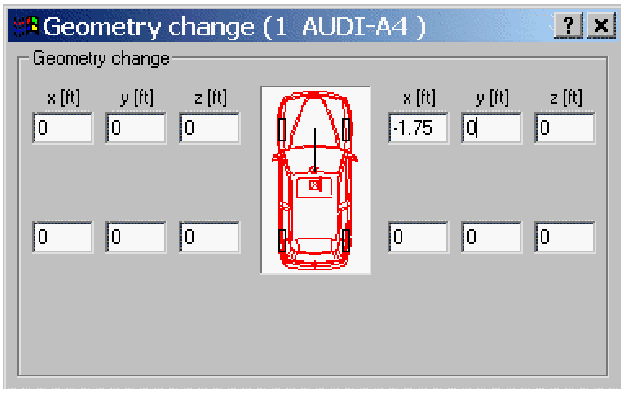
This enables minimum and maximum velocities for the sequences following this sequence to be specified. For example, the following diagram shows the effect of a maximum specified velocity of 25 mph on a vehicle acceleration sequence.

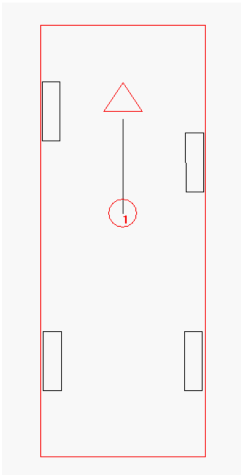


Alternatively, the effect of a minimum specified speed greater than zero will prevent a decelerating vehicle from going below that speed, regardless of the length of the braking sequence.

Geometry Change

The individual location of a vehicle's wheels can be changed at any time with this sequence. This is useful for modeling horizontal and vertical wheel movement due to damage and for modeling vertical movement due to tire blowouts.





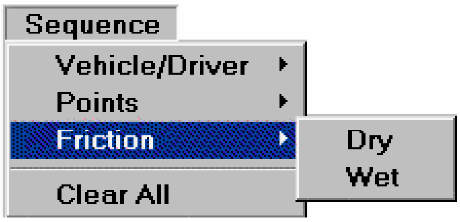
Flat tires can be modeled by raising the vertical position of the wheel by the extra distance the tire compresses due to its flat condition. This geometry change can be combined with a change in tire friction (see next item) and a change in tire slip angle (see **Vehicle – Tire Model**). The slip angle change must, however, occur at the start of a particular simulation.

Trailer disconnection

If this point is assigned to a trailer in a truck trailer combination the trailer decouples from the towing car at that point in the sequence. This point allows the simulation of malfunctioning or loosening trailer hitches that decouple without external force.

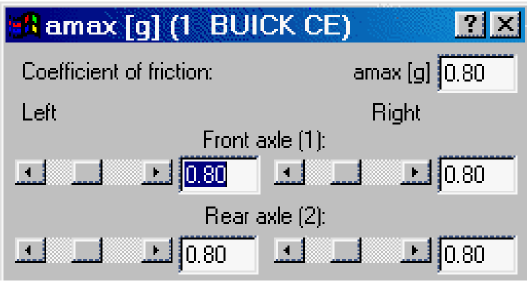
To simulate the decoupling of trailers due to external force, refer to section Vehicle Settings on page 106ff.

Friction



Friction sequences can be used to define friction coefficients. Friction sequences are valid from the sequence position to the start of a new friction sequence (dry or wet).

Dry Friction

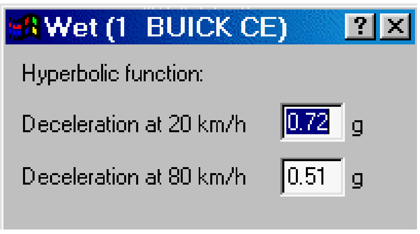


The "dry" friction may be defined for each wheel individually. When the left front wheel friction is changed, the other three wheels change with it. For control, the maximum possible acceleration of the car under these conditions is also displayed.

If no friction sequence is defined, the program will use the coefficient of friction specified in **Options - Options – Default Settings**.

Note: *A friction sequence placed immediately after the start will apply for the whole forward simulation, while one place immediately before the start will apply for the whole backward simulation.*

Wet Friction



The Wet Friction sequence allows the user to define a speed-sensitive friction coefficient, which is commonly the case when braking on wet roads. The characteristic parameters are calculated using a hyperbolic function, based on the specification of the acceleration at 20 km/h and at 80 km/h (below 20 km/h a constant acceleration is assumed). Typical values are in the table below.

$$\mu = \frac{0.2 \cdot A}{n} \cdot v^{2-n}$$ *, where v is in m/s²*.

	Partly Wet	Damp Wet	Wet	Very Wet	Extremely Wet
µ20	0.8	0.7	0.6	0.5	0.4
µ80	0.6	0.5	0.4	0.3	0.2
A	12.6	11.9	11.36	11.14	11.79
n	2.21	2.24	2.29	2.37	2.5

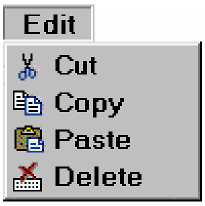
(Schimmelpfennig: Verkehrsunfall 3/85)

Note: *This relation should only be used if a vehicle is tracking normally while braking. If the vehicle slides sideways, it is very difficult to define the relationship between friction and velocity, and it should thus not be used under these conditions. For this reason, it is recommended this sequence not be used for the post-impact phase. In this case it is better to use the dry friction sequence with a reduced friction coefficient.*

Clear All

Deletes all previously defined sequences.

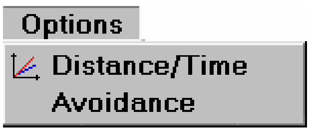
Edit



This menu option allows the user to perform following functions with the sequences:

Cut	Cuts the active sequence to the clipboard
Copy	Copies the active sequence to clipboard
Paste	Pastes the clipboard sequence below the active sequence
Delete	Deletes the active sequence

Options



The Options menu allows the user to select the Distance/Time calculation feature or perform avoidance calculations.

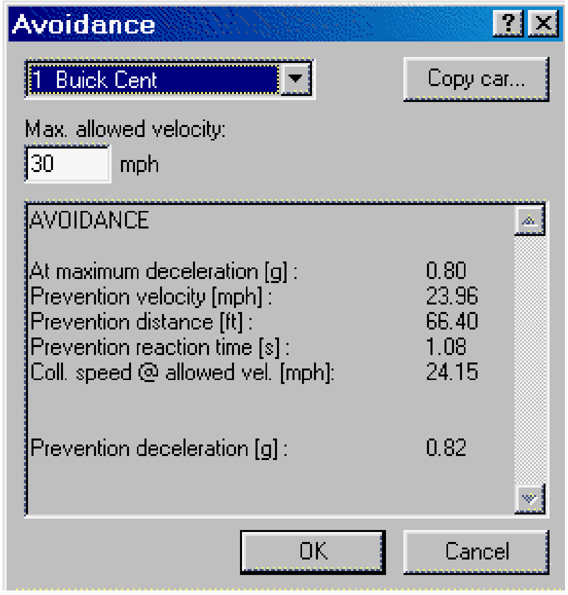
Distance/Time Calculations

This tool performs a kinematic distance/time calculation and displays a diagram of the results, based on the speeds and brake/acceleration sequences for each vehicle. Impacts and yaw are ignored for the kinematic simulation. See the **Technical Manual** for a complete description of the kinetic and kinematic models. The Kinematic model is also used in Dynamics – Kinematic Calculations (F10) or if the user switches to it from the Kinetics model using the Simulation Model tool .

Avoidance

This feature allows the user to easily perform an accident avoidance analysis of the current project.

This analysis is automatically done after solving for vehicle speeds with a PC-Crash simulation and determining the point of initial perception for a driver based on a specified perception/reaction time and pre-impact braking distance. The evasion (prevention) speed, reaction time, deceleration and distance are calculated independently, based on the available distance between initial perception and impact.

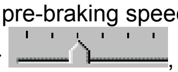


The Avoidance dialog box shows the following under the Avoidance heading:

- **At maximum deceleration** – The braking deceleration that the avoidance calculations are based on.
- **Prevention velocity** – The maximum speed which the selected vehicle could have been going and still have avoided the impact in the overall distance defined by the actual reaction time and pre-impact braking distance.
- **Prevention distance** – The overall pre-impact perception and braking distance required to avoid the impact, at the actual pre-braking speed.
- **Prevention reaction time** – The maximum reaction time that the impact could have been avoided at, at the actual pre-braking speed.
- **Collision speed at allowed velocity** – This is the collision speed that would have occurred if the vehicle had the pre-braking speed defined in the Max. Allowed velocity text box. The maximum allowed velocity is usually the speed limit at the accident scene.
- **Prevention deceleration** – The required pre-impact braking deceleration to avoid the impact, from the actual pre-braking speed.

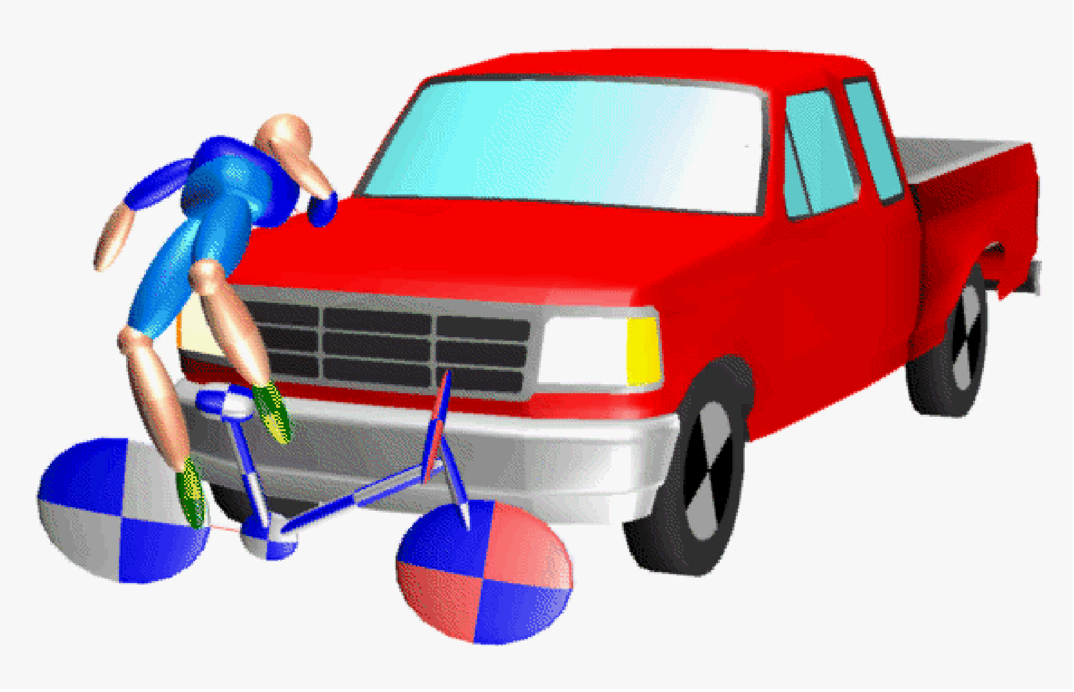
Copy car

If a more detailed avoidance analysis is to be done - particularly for the avoidance of a moving vehicle, the Copy car button should be used.

This button copies the selected vehicle's sequences to a new vehicle, but moves all the sequences to after the start sequence. The new vehicle is positioned at the point of reaction, calculated earlier. With the new order of sequences a forward simulation can be done starting at the point of reaction. The avoidance speed is easily determined by checking for contact between the selected vehicle and the other vehicle while trying different pre-braking speeds. The vehicles must be moved with the Simulation toolbar slider bar , after performing the new simulation with either of the Forward button(s) .

The additional vehicle can be deleted using **Vehicle - Erase Vehicle** after finishing the avoidance calculations.

Chapter 5
Multibody Model



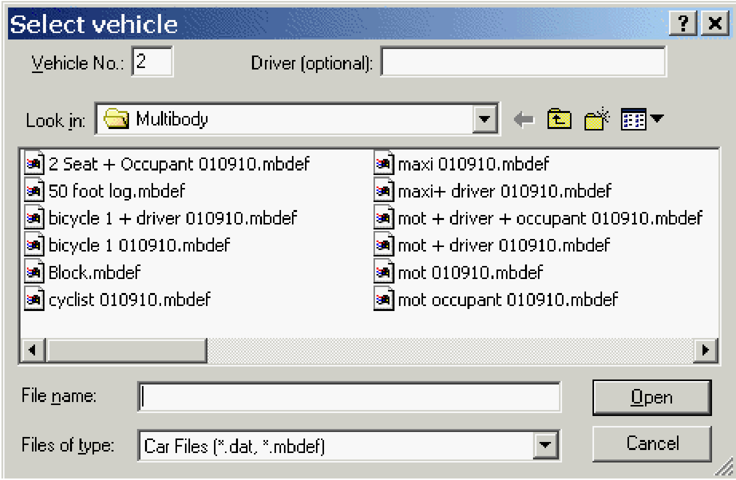
Vehicles are generally regarded as rigid bodies for the simulation of traffic accident collisions with kinetic 3D simulation programs. This simplification is reasonable for the simulation of vehicle - vehicle collisions, vehicle - trailer collisions and vehicle collisions with rigid obstacles such as trees or walls. However, for vehicle collisions with pedestrians and two-wheeled vehicles, this simplification does not allow the motion to be accurately modeled. The introduction of multibody modeling provides a powerful tool for the reconstructionist to study incident dynamics further, such as correlating pedestrian injuries to vehicle damage areas.

PC-Crash was extended to allow simulation of multibody pedestrians in version 5.1. During the development, special attention was paid to having reasonable computing time in conjunction with realistic motion. The present version of PC-Crash has been extended further to allow other multibody objects such as two-wheeled vehicles, multiple multibody objects in one simulation, and multibody objects on 3D ground surfaces.

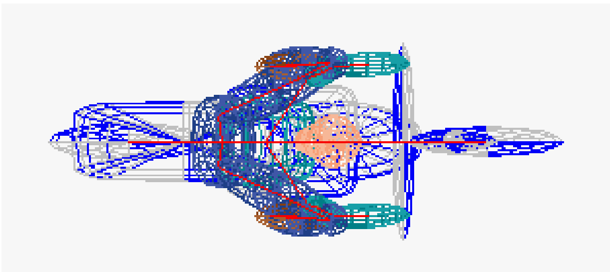
Operation

Loading a Multibody

The multibody is loaded as a custom vehicle with **File – Import - Custom Vehicle** . This opens the Select vehicle dialog box. A number of multibody files (designated by the file extension *.mbdef) are provided in the Multibody subdirectory of the PC-Crash directory.



Select the desired multibody file and press the Open button to load it. It appears on the main screen as shown below.



Saving a Multibody

Loaded multibody systems can be modified by the user and then saved for future use. After modifying the multibody as desired (see the section Changing Multibody Properties in this chapter), save the multibody under the desired name using **File – Export - Custom Vehicle**. This opens the Select vehicle dialog box with a Save button instead of the Open button. Make sure to select the *.mbdef file in the Files of Type drop down list.

Positioning the Multibody

The position and orientation of the pedestrian multibody system can be changed in the following three ways:

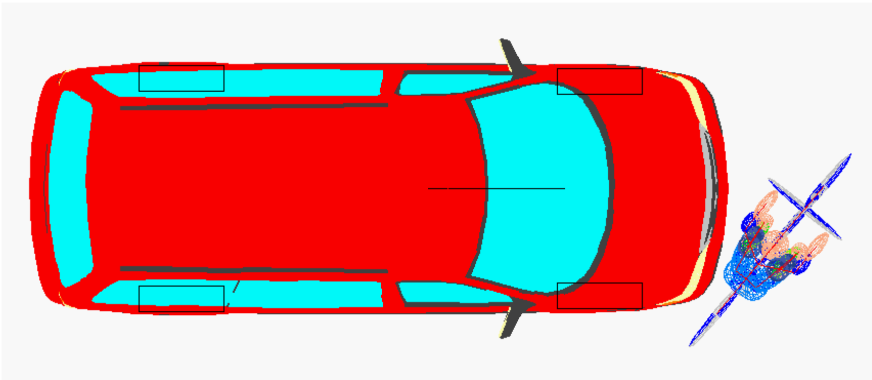
1. Using **Dynamics - Position & Velocity** (F7), all the loaded multibodies can be positioned and rotated as a group.

2. The **Tow Truck** tool can be used to position and rotate multibody systems individually, similar to the way a vehicle is. When positioning a

loaded motorcycle and rider, it is difficult to maintain the same relationship of the two with this tool.

3. Using **Vehicle – Multibody System – Settings**, the multibody systems and even the individual bodies can be moved and rotated together or separately. See the menu description of this menu item for more detail.

If an impact is to be performed with a vehicle, another multibody or the ground, ensure there is no overlap at initial placement. At least a small space must be left or unrealistically high contact forces will occur in the first integration step of the calculations.



Applying an Initial Multibody Velocity

The initial velocity of a multibody system can be changed as a whole, but the velocities of individual bodies within each system have to be the same. Thus, if two pedestrian multibody systems are loaded they can have different initial velocities, but different parts of one pedestrian's body cannot have different velocities. Multibody initial velocity can be defined in the following two ways:

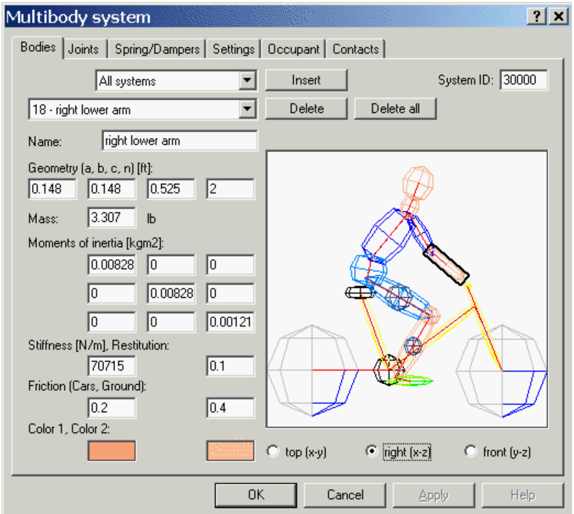
1. Using **Dynamics - Position & Velocity** (F7), all the loaded multibody systems can be given an initial velocity as a group.
2. Using **Vehicle – Multibody System – Settings**, the multibody systems can be given initial velocities together as a group or separately. See the description of this menu item in the following section for more detail.

Changing Multibody Properties

The multibody properties and initial conditions can be reviewed and changed using the menu item **Vehicle – Multibody System**. This accesses a window containing the following folders:

- Bodies
- Joints
- Spring/Dampers
- Settings
- Occupant
- Contacts

Bodies



In this folder multibody bodies can be modified, created or deleted. In the top drop down list box individual systems or "All systems" can be selected. In the second box, individual bodies can be selected. A body is defined by the name, the appropriate mass and the geometry. Each body is an ellipsoid, with ***a***, ***b***, ***c*** as semi-axes and order ***n***:

$$\left(\frac{|x|}{a}\right)^n + \left(\frac{|y|}{b}\right)^n + \left(\frac{|z|}{c}\right)^n = 1$$

The moments of inertia are calculated automatically. However, these values can also be manually entered.

Stiffness, restitution and friction for each body can be defined. In the case of friction, there can be different values for car and ground contacts. The multibody – multibody friction is the same as that chosen for multibody – ground contact.

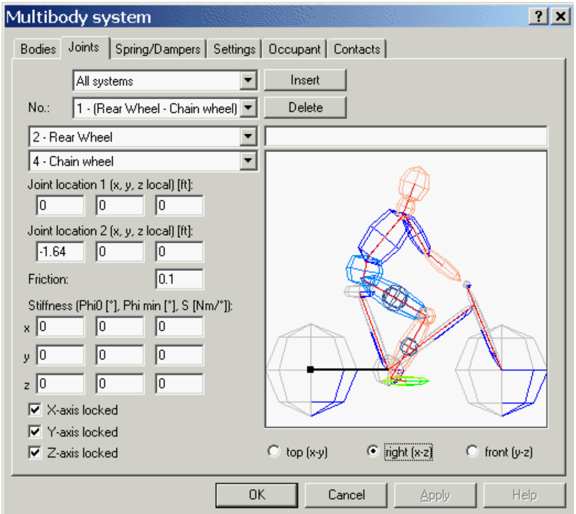
Two colors can be assigned to each body, which appear on opposing quadrants of each half ellipsoid. When a body is contacted in a simulation, the second color will turn red to illustrate the contact.

There is a preview window with option buttons for view selection:

- **Top** Projection in x-y plane,
- **Front** Projection in x-z plane,
- **Right** Projection in y-z plane.

In the preview window the individual bodies of a multibody system can be selected with the mouse. The appropriate body is then shown in the drop down list box.

Joints

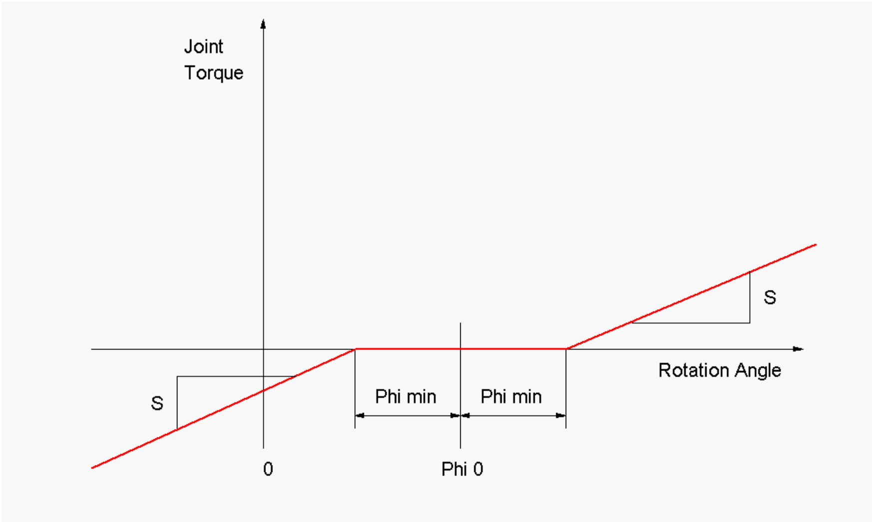


The individual bodies of a multibody system are interconnected by joints. These joints can be fully locked, fully free (similar to a ball joint), or can have stiffnesses applied about the x, y or z rotational axes.

In the top drop down list box of the Joints folder individual systems or "All systems" can be selected. In the second box, the joint can be selected by its number. The two bodies connected by the selected joint are shown in the next two boxes automatically. These three drop down list boxes can also be used to define new joints or change or remove existing joints. The joint location is defined in the local coordinate system of each body. The position is displayed in the preview window, where it can also be selected by clicking on it. The joint location is illustrated with a small black circle. A black line runs from the circle to the center of each connected body.

The stiffness of each joint can be specified as a constant frictional moment or a moment that increases with twisting angle. The frictional moment is defined with a coefficient of friction, which is applied at a radius of 1cm about the joint. The definition of Stiffness **Phi 0, Phi min** and **S** are similar to those for the trailer couplings, as follows:

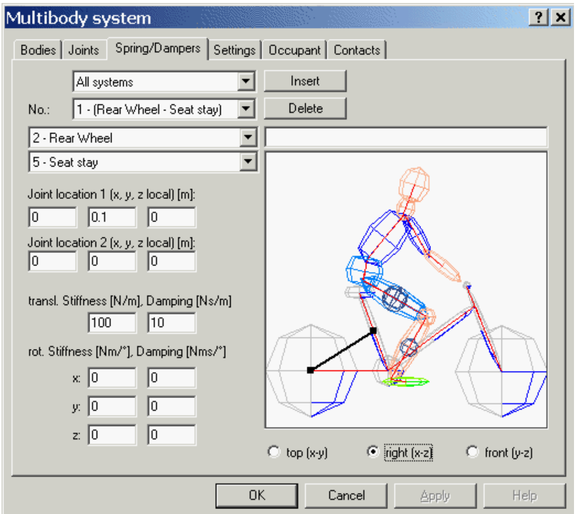
- **Phi 0** - This is an offset value from 0°. If the relative rotation is more than Phi 0 + Phi min or less than Phi 0 – Phi min, the linearly increasing torque S is applied. If Phi 0 = 0°, the torque S is symmetrical for positive and negative rotations.
- **Phi min** - Relative rotation with respect to Phi 0 at which the rotational torque S starts.
- **S** - Linearly increasing resistive torque about the specified axis, starting at Phi 0 + Phi min.



Note: *If very high stiffness values are used, the calculations may become unstable. The maximum stiffness that can be used depends on the body weights and the selected time step. Reduce the stiffness or the time step if multibody motion appears incorrect.*

The individual joints can also be locked by checking the x-axis locked check box. The joints cannot be locked individually about the x, y or z axes.

Spring/Dampers



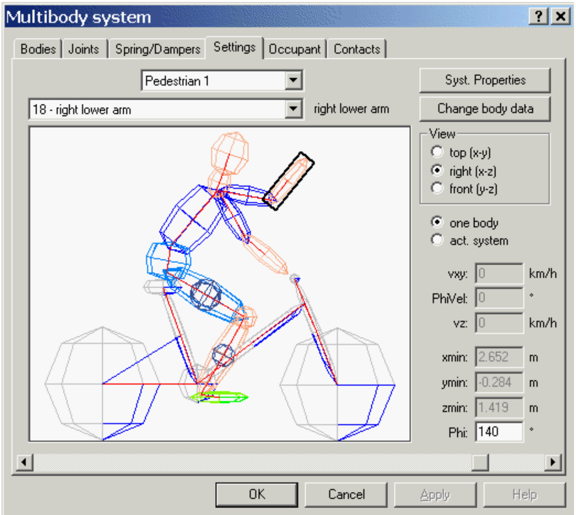
The individual bodies of a multibody system can be interconnected by spring/dampers systems.

In the top drop down list box of the Spring/dampers folder individual systems or "All systems" can be selected. In the second box, the spring/damper can be selected by its number, or it can be selected by clicking on the respective area in the preview window. The two bodies connected by the selected spring/damper are shown in the next two boxes automatically. These three drop down list boxes can also be used to define new spring/dampers or change or remove existing spring/dampers.

The endpoints of the spring/damper are defined in the local body coordinate system of the respective multibody. Translational stiffness/damping and rotational stiffness/damping coefficients about the x, y and/or z axes can be defined.

The first spring/damper is created by selecting the Insert button, which inserts a spring/damper between bodies 1 and 2. The bodies can then be changed to the desired ones in the body name drop down list boxes.

Settings



The functions in this folder allow rotation and positioning of the whole multibody or individual bodies.

In the top drop down list box the desired multibody system can be selected. Individual bodies can be selected n the second box, or by clicking on the respective body in the preview window.

After selection of the appropriate view button and option button (One Body or Act. System), the chosen body or system of bodies can be rotated. This can be done with the slide control beneath the preview window or by typing in the desired value in the Phi text box. If the body or system is to be rotated about more than one axis, the rotations should be done about the x, y and z axes in that order.

One or all systems can be positioned. The option button Act. System must be selected. Positioning is then done by typing the desired values in the xmin, ymin and zmin text boxes. These are the minimum positions for the system(s) with respect to the global coordinate system. For example, a value of 0 would be chosen for zmin if the system is to start out with its lowest point on the ground. The multibody system as a whole can also be positioned in the x-y plane in the Position & Velocity window.

One or all systems can be given an initial velocity by typing the desired values in the Vxy, PhiVel and Vz text boxes. The option button Act. System must be selected first.

Syst. Properties

The overall properties of each multibody system can be changed using the Syst. Properties button. Before this feature is used, select the particular multibody system to be modified in the drop down list box at the top of the Settings folder. In addition to the overall dimensions and weight, the coefficient of restitution of all contacts, and the coefficient of friction of ground and vehicle contacts can be changed. The multibody – multibody friction is the same as that chosen for multibody – ground contact.

Length, Width, Height

Only the height of the multibody is user-definable here. The “length” and “width” are scaled automatically, based on the selected height.

Weight

The weight of the multibody can be changed here. The body component weights are then calculated automatically, based on the selected weight.

Restitution

The coefficient of restitution for the multibody impacts with the striking vehicle and the ground is selected here.

Friction Ground

The coefficient of friction between the multibody and the ground is selected here. The lower of the specified multibody to ground friction or Friction Polygon friction coefficients will be used. The “Friction Ground” is also used for multibody-multibody contacts.

Friction Cars

The coefficient of friction between the multibody and the striking vehicle is selected here.

3D Dxf Car Contact

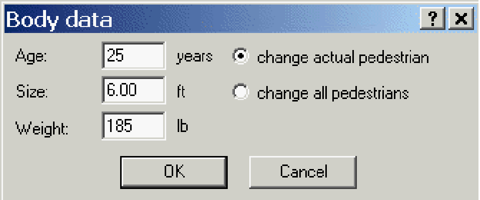
This enables the calculation of contacts with the 3D DXF vehicle shape, rather than the default shape. This will increase calculation time significantly, however. Also, care must be taken with DXF shapes to ensure they have no holes or inward pointing faces that could cause the striking multibody to be caught unrealistically. For these reasons, the default car shape, modified to the correct dimensions using **Vehicle – Vehicle Settings –Vehicle Shape**, is generally the better choice.

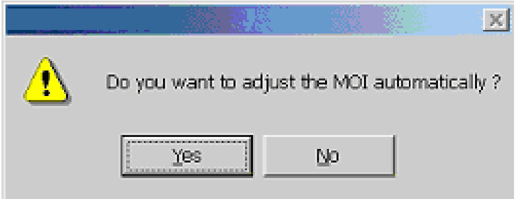
Occupant

This enables the multibody to have contacts with a vehicle’s interior, rather than it’s exterior. Selection of this will change the direction of the faces on the vehicle shape or attached DXF to be pointing inwards (towards the C.G.) rather than outwards. For occupants in seats, it may instead be easier to load the Occupant + Seat mbdef file and the features in the Occupant folder (see next folder description), which allow an occupant calculation to be made after the vehicle simulation has been completed.

Change Body Data

The body data of a loaded pedestrian multibody can be modified using the Change Body Data button. Before this feature is used, select the pedestrian in the drop down list box at the top of the Settings folder. The Change Body Data button opens a dialog box in which age, height and weight can be changed. The body component sizes and weights are then calculated automatically, based on the selected values. Option buttons allow the selected pedestrian or all pedestrians to be changed. After making changes in the Body Data dialog box, a message box appears asking if you want the MOI (moments of inertia) adjusted to suit the specified values.





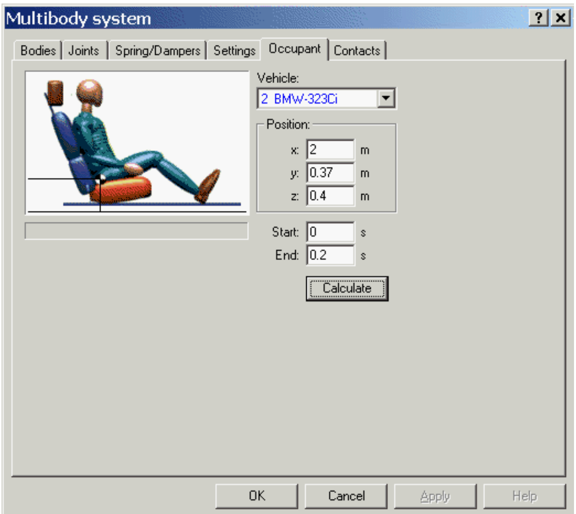
The body data is changed according to the research report "International Data on Anthropometry" by Hans W. Jurgens, Ivar A. Aune and Ursula Pieper, published by the Federal Institute for Occupational Safety and Health, Dortmund, Federal Republic of Germany and a study made by various scientiests in Slovakia[1].

Occupant

An unrestrained occupant and also a restraint occupant (belts are modeled using spring-damper elements) can be placed in a vehicle to examine motion in an impact or other event, after the vehicle simulation has been completed. This feature only functions when the 'Occupant + Seat' or 'seat + occupant + belt' mbdef file is imported, which can be done after the vehicle simulation is completed. This multibody system can also be loaded beforehand, but must be deactivated until the vehicle simulation is completed. This is done using the Simulation Model button to turn off ☑ Multibody System. Remember to turn it back on after the vehicle simulation is completed.

The position of the occupant is specified in the Occupant window. The occupant's H-point position is specified in the x, y and z text boxes, which define the distance from the vehicle's front; left of the longitudinal centerline (use a negative number for right side passengers), and above the ground. By default, a spring/damper joins the seat to the vehicle it is placed in. This spring/damper can be viewed and modified in the Spring/Dampers folder.

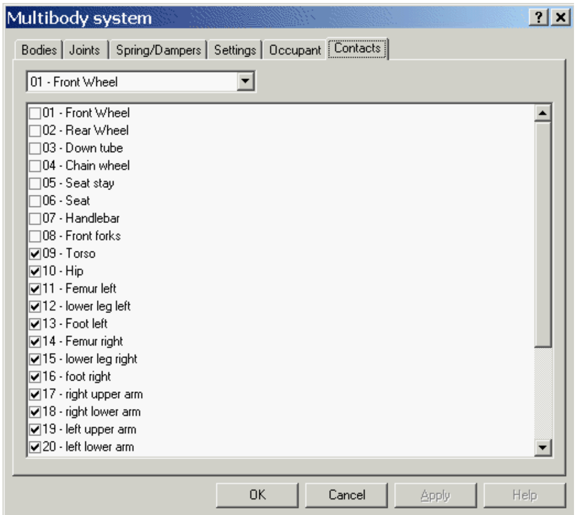
After the Start and End times for the multibody calculation are selected, the Calculate button is used to run the calculation. The calculation progress is shown beneath the occupant picture.



[1] Autori – kolektiv, Vademecummedici, Zilina Slovakia 1998

Contacts

In this folder the contact calculation choices can be made. This is done by selecting a body in the drop down list at the top and then using the check box beside every other body to define whether or not contact forces will be calculated if the two bodies come into contact. In the following example, the bicycle bodies are connected with fixed joints and there is some overlap of connecting bodies, so these contacts have been turned off by default.



Running the Simulation

The simulation is started using the Forward Simulation button (exception: occupant simulation is started within the multibody window). The multibody impacts are calculated automatically, without the use of the Crash dialog box.

The calculation for the multibody model takes place in 1ms steps, even if 5ms is specified in Simulation Model . However, if the integration step is reduced to 1ms, 0.5ms or 0.1ms, these values will be used for the calculation.

***Note**: If 5ms is specified, although the multibody's motion is calculated at 1ms intervals, the vehicle moves at 5ms intervals. At higher impact speeds (more than about 40 km/h or 25 mph) or with very heavy multibody objects, this can cause the multibody shape to go through the vehicle outline and then rebound unrealistically due to a force much higher than could normally occur. Reducing the integration step fixes this problem. To decrease calculation time, the simulation can be manually stopped after a collision, the integration step can be increased back to 5ms, and then the remainder of the simulation can be calculated. Also, check that the stiffness of the multibody is realistic if unusual rebounding takes place or bodies seem to go through the contacting surface.*

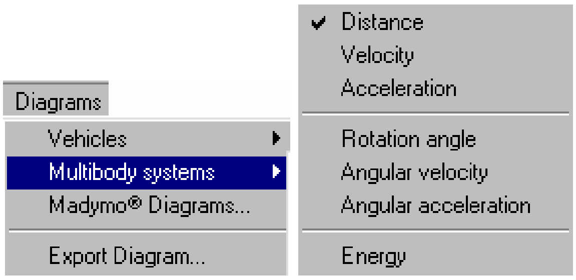
Impact forces are calculated at the point where the tangent of the multibody ellipsoid surface is parallel to and in contact with the contacting surface. If this is not possible, an impact point will not be located and there will be no contact force. This can occur if the multibody's tangent point is beyond the edge of the contacting plane, such as in the opening for a 3D vehicle DXF's grille when 3D DXF Car Contact is selected. In this case, the best solution is to fill the grille opening with a polygonal surface using a 3D drafting program such as AutoCAD. Another solution is to make the multibody system with more, smaller bodies.

Viewing the Multibody Results

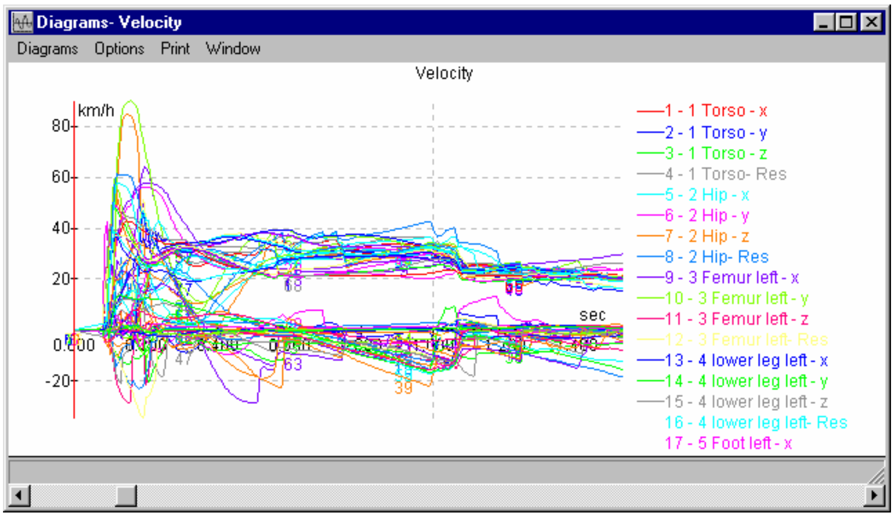
Dynamic values of the complete multibody simulation can be viewed in the form of diagrams. The Multibody model simulation motion can be viewed on the main screen or in the 3D window using the Simulation Toolbar slider bar. An animation can be rendered the same way an animation of a vehicle-vehicle incident can.

Viewing the Multibody Dynamics with Diagrams

The multibody diagrams can be viewed by selecting **Options – Diagrams – Diagrams – Multibody Systems**.

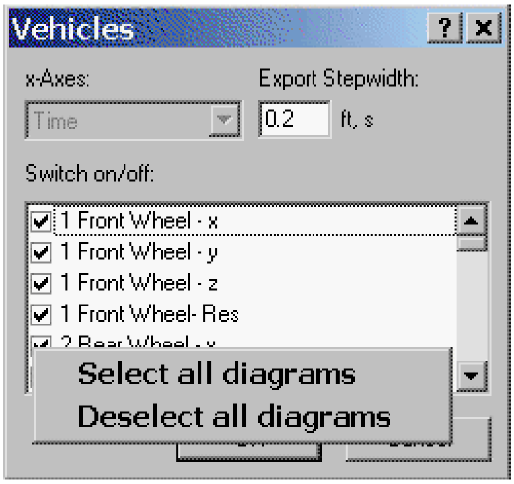


From the Multibody Systems menu, diagrams for distance, velocity, time etc. of each rigid body component of the multibody can be viewed. The following example shows the velocity diagrams of all of a pedestrian's components.

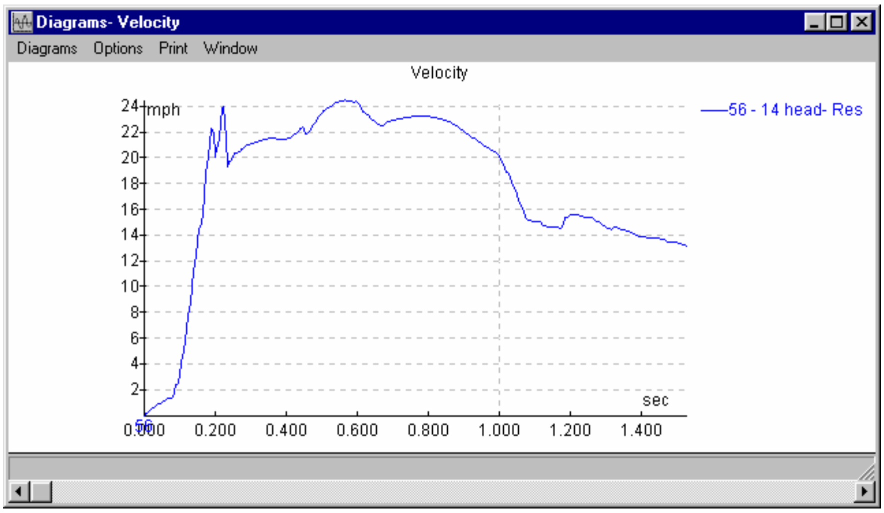


In order to view individual graphs clearly, it is possible to turn some of them off. This is done by selecting **Options – Diagrams/Axes** in the Diagrams window. This opens a window in which each item has a check box beside it. Click on each check box to turn it off or on.

If you want to view only one or a few of the diagrams, you can first turn them all off by right-clicking in this window. This allows all items to be selected or deselected. First deselect them, and then select only the ones you want to view.



For example, selecting only Head – Resultant Velocity allows this one graph to be viewed very clearly.



The diagram data can also be exported, for example to a spreadsheet program such as MS Excel®. This is done by selecting the Diagrams window menu option **Diagrams – Export Diagram**.

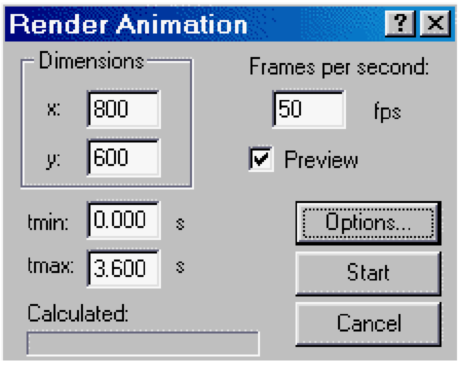
This opens a window in which the data can be saved as a diagram file (*.dia) or a DXF file (*.dxf). Note that the default time step width for the diagram data is 0.5 seconds, which should normally be changed to a much smaller value.

Viewing the Multibody Motion in 3D

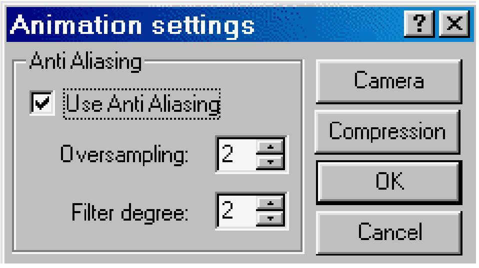
Multibody animations are made the same way that other PC-Crash animations are. First, to view the occupant in 3D, place the camera in the desired position on the main screen using the Camera button . The view can then be adjusted further in the 3D Visualization window; refer to the menu item description for **View – 3D Window** for more information.

Rendering the Multibody Animation

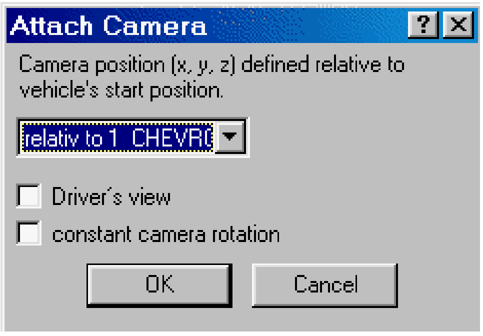
In the 3D window, select the menu option **Animation – Render**. After selecting a file name for the animation to be generated, the Render Animation dialog box opens. Since the pedestrian motion will vary greatly over a small time period, use a high number of frames per second (50 frames per second has been selected below).



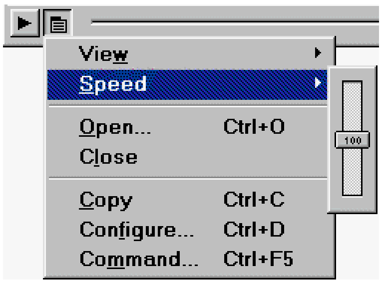
Generally, the best way to view pedestrian motion is with the camera beside the striking vehicle, attached to it. Press the Options button in this dialog box to open the Animation Settings dialog box.



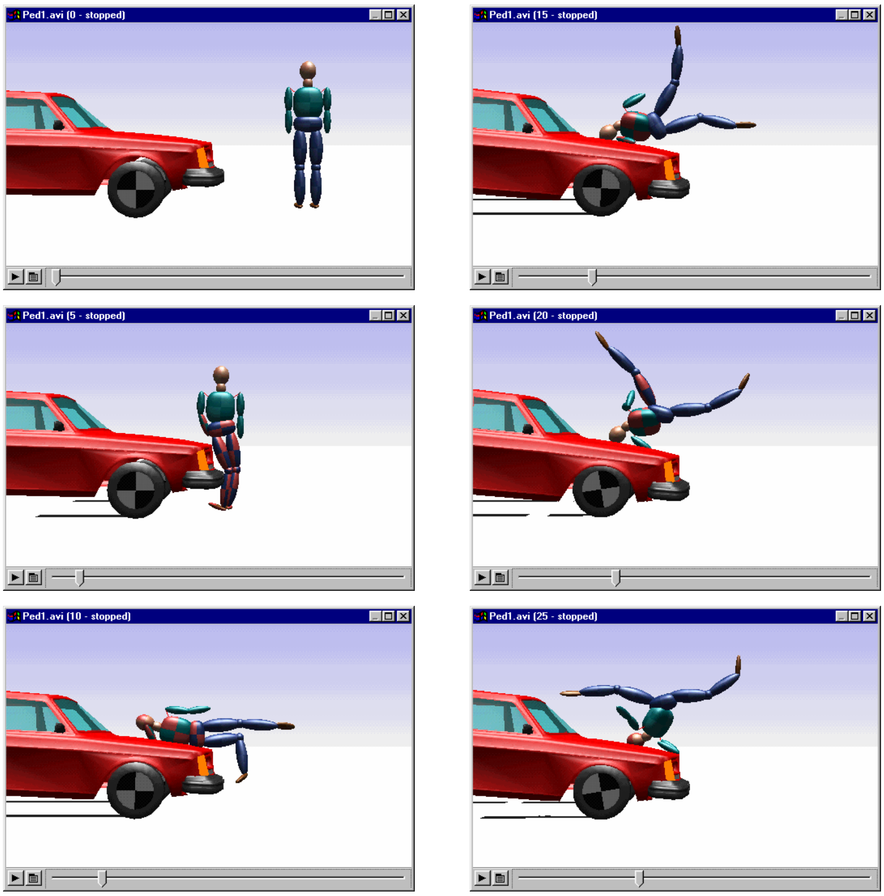
Then, press the Camera button to open the Attach Camera dialog box. Select "Relative to …" in the drop down list to select the correct vehicle to attach the camera to.



Once the camera is attached, return to the Render Animation dialog box and press the Start button to render the animation. After it is rendered, you may wish to view the animation in slow motion. Click on the Options button near the lower left corner of the Animation window to open the following menu.



Select the Speed menu item and slide the slider bar to a low speed such as 5% of actual. The animation can then be viewed in slow motion by clicking on the arrow button. The Animation window main slider bar can also be used to view each animation frame in detail. Following is a six frame pedestrian impact sequence taken at intervals of 100ms.



Viewing Multibody Occupant Motion

When a multibody occupant is used, the vehicle body often obscures the view of the occupant motion. Refer to the next chapter **Madymo Occupant Model** in the section **Viewing Madymo in 3D** for different viewing methods.

Chapter 6
Madymo Occupant Model

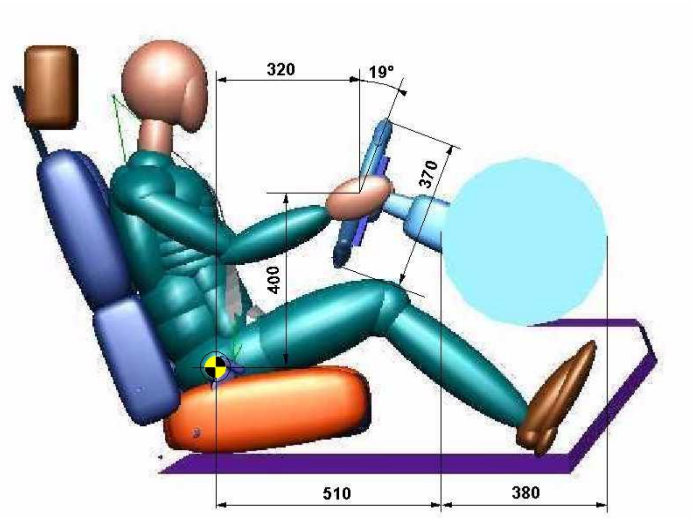
In PC-Crash it is possible to carry out detailed simulations of restrained or unrestrained occupant motion and loading, with the optional interface to Madymo (MAthematical DYnamic MOdel).

Madymo is an occupant modeler program developed by TNO Road-Vehicles Research Institute for use by vehicle manufacturers and developers. Madymo uses a multibody occupant model with a finite element seat belt and airbag. Also, a seat belt pre-tensioner can be defined.

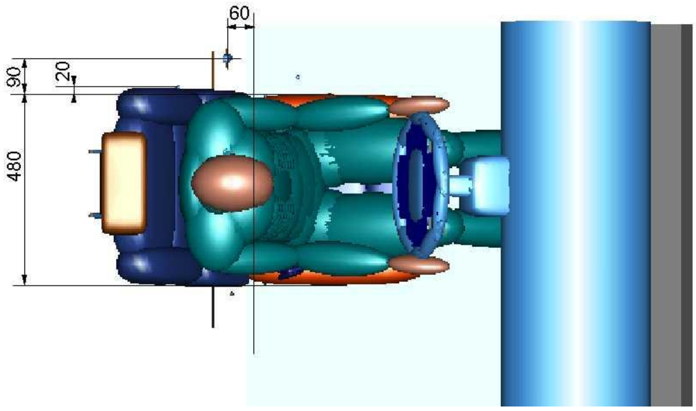
The provided PC-Crash CD contains the Madymo manuals. Refer to those manuals for a complete description of the theoretical models of Madymo. This chapter is confined to an explanation of the use of Madymo within PC-Crash.

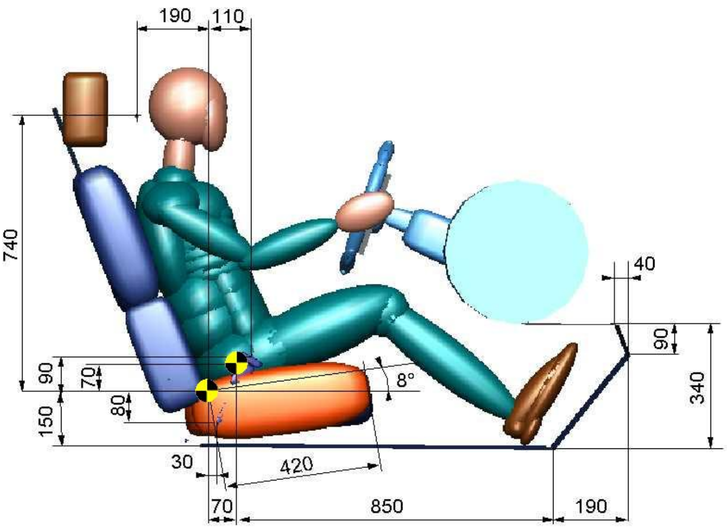
The PC-Crash version of Madymo has some limitations built into it compared with the full workstation version of Madymo, as follows:

- The occupant height is fixed at 50th percentile (5'-9" tall)
- The steering wheel and dash geometry is fixed, as follows:

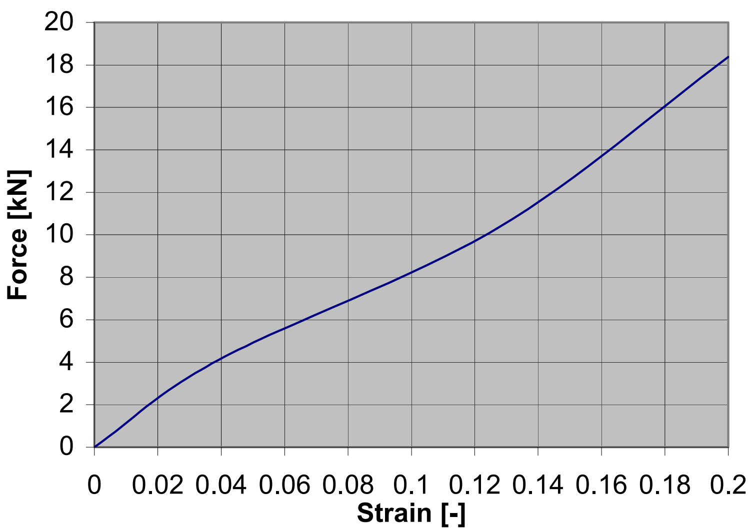


- The seat belt, seat cushion and foot well dimensions are fixed at the following values:



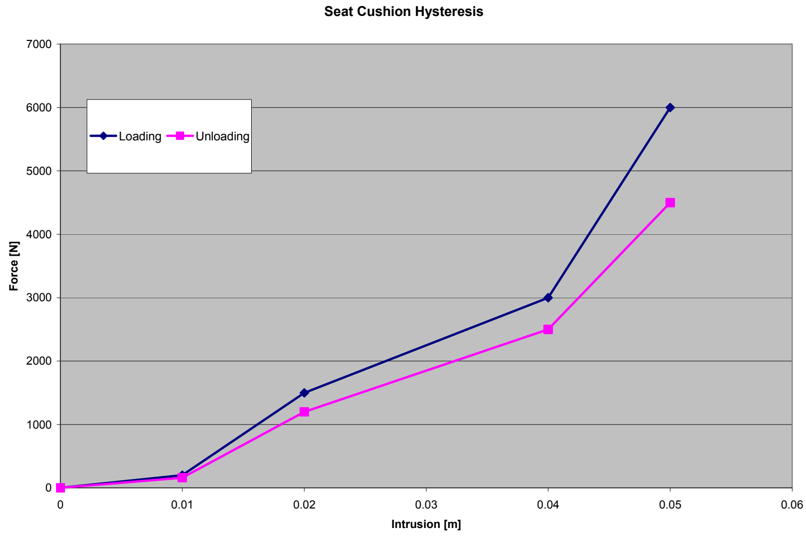


- The seat belt webbing elasticity is fixed (13% stretch @ 11 kN)

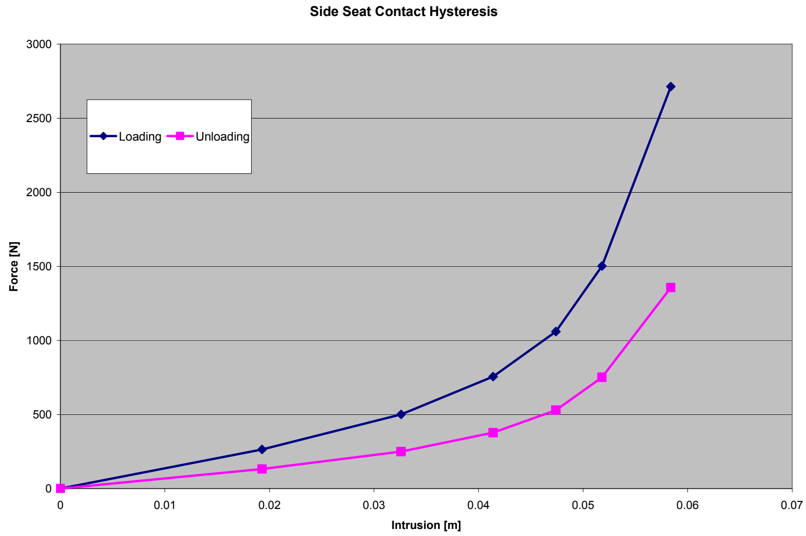


- The occupant cannot be out of a normal seated position at the start of the impact
- The maximum length of the occupant simulation is 2000ms after the impact

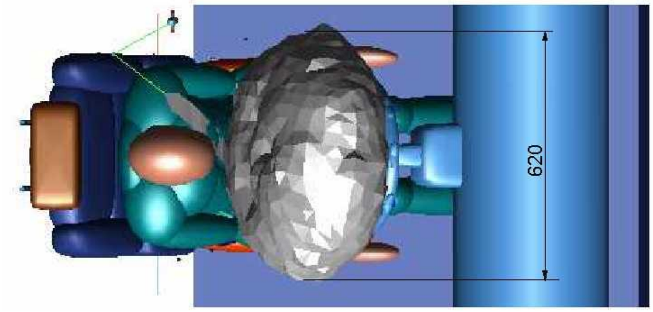
- The coefficient of friction for all occupant - interior contacts is 0.3, except for the seat cushion, which is 0.4.
- The seat middle cushion hysteresis is as follows:

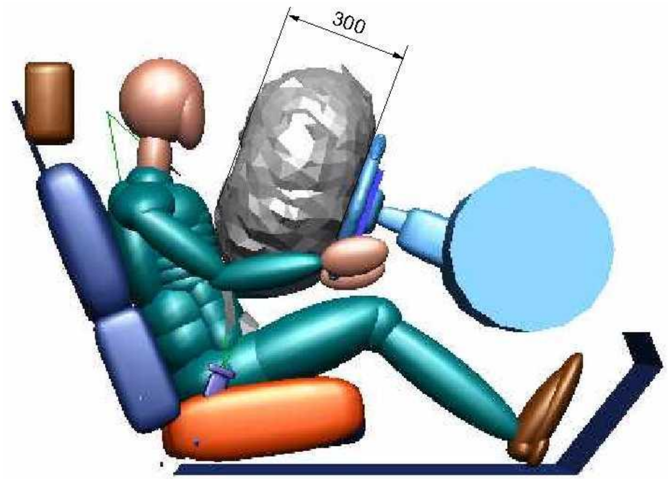


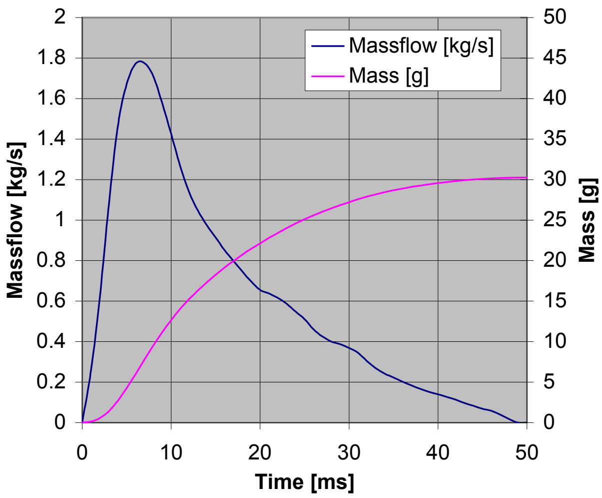
- The seat side cushion hysteresis is as follows:



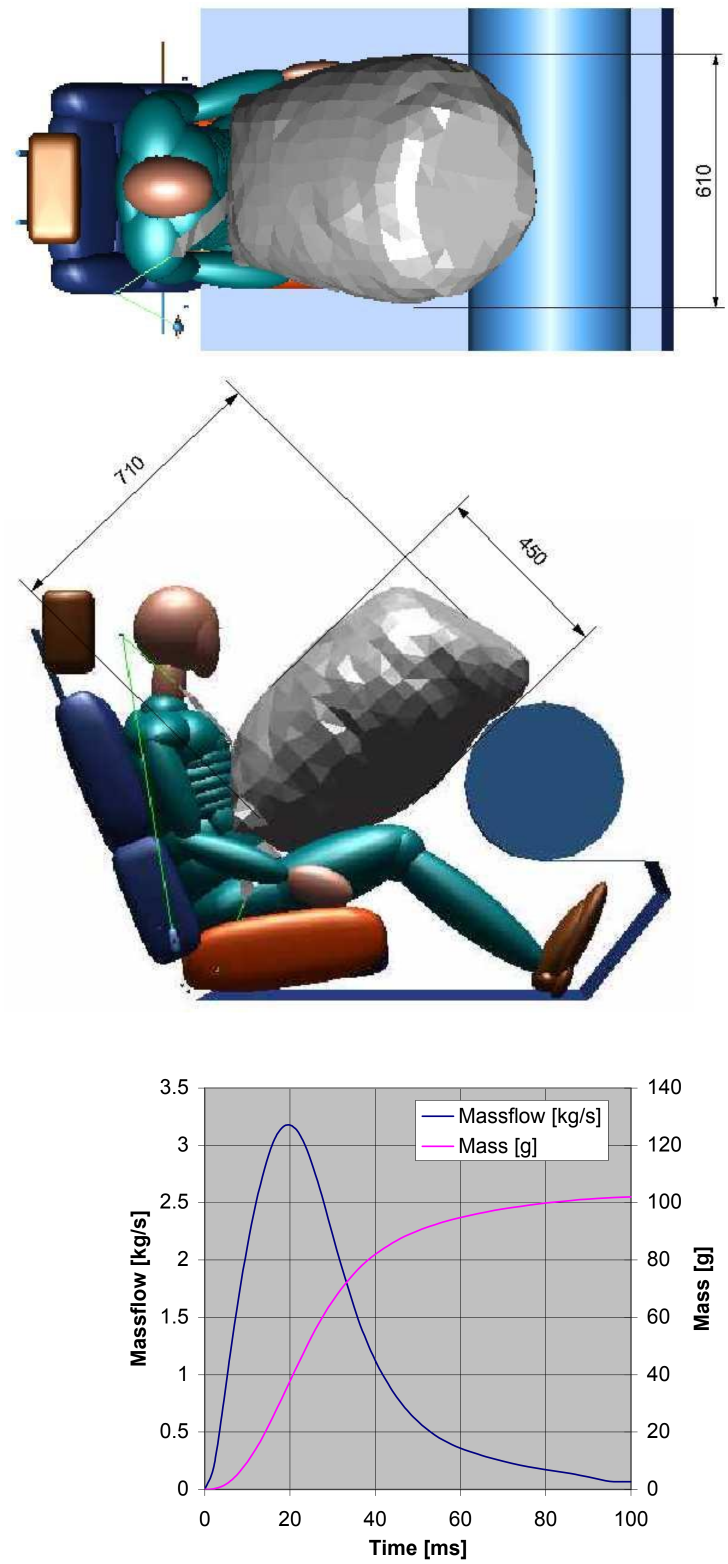
- The driver's and right front airbag properties are fixed. The driver's airbag volume is about 50 liters and the right front airbag volume is about 115 liters. Other properties are as follows:







Driver's Airbag Dimensions and Mass Flow



Right Front Airbag Dimensions and Mass Flow

Operation

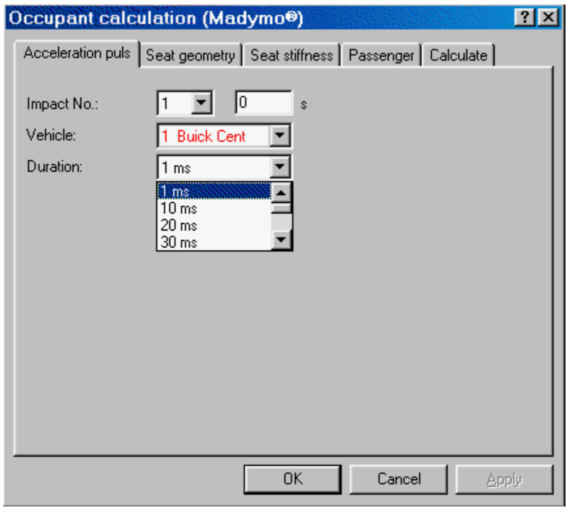
First, the PC-Crash simulation has to be completed and saved. A pre-impact simulation time of at least 250ms must be included in the simulation, to allow the Madymo occupant model to reach a steady state condition prior to impact. It is also needed to allow for the pulse build-up duration of a typical crash, compared to the infinitely small (time=0) pulse duration that the PC-Crash impact model is based on.

The PC-Crash impact and trajectory simulation, including the use of sequences, is described in the previous chapters.

For the Madymo simulation, select **Impact - Madymo® Occupant Simulation**. This opens the Occupant Calculation (Madymo®) window, which has five folders, as follows:

Acceleration Pulse

This folder allows the selection of the impact number and the vehicle for which the occupant simulation is to be calculated. For the default acceleration pulse (1ms), the instantaneous velocity change from the PC-Crash momentum-based crash simulation is used. Impact durations up to 150ms, using a square wave pulse, can be selected. This time is centered on the PC-Crash impact, so that half is before and half is after it.

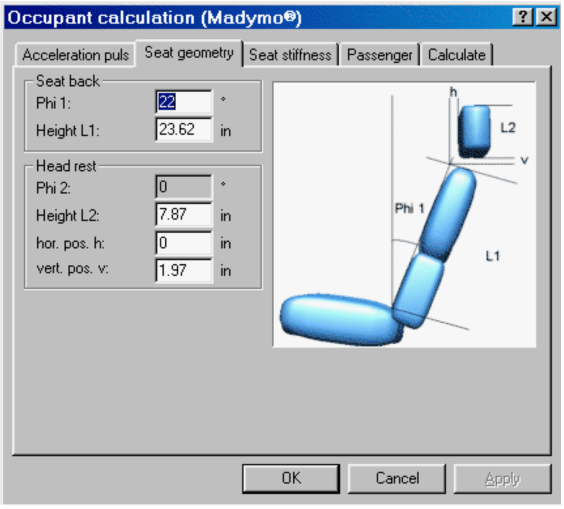


When an acceleration pulse other than 1ms is selected, the Madymo interior model and the PC-Crash vehicle outline will move with respect to one another in rendered animations during the selected impact time. This because the Madymo model's velocity change is being averaged over the selected acceleration pulse length, while the vehicle's velocity change is not.

Also, the maximum duration (2000ms) of the Madymo run will be shortened with longer acceleration pulse times. This is because the total number of calculations that are performed is limited to 200. These calculations are at 10ms intervals except during the crash pulse, when they are at 1ms intervals. For example, with a 100ms crash pulse duration, 100 calculations at 1ms intervals are used during the crash, leaving 100 calculations at 10ms intervals for the remainder of the Madymo run, which is a total time of 1100ms. The following table gives the maximum possible Madymo run length for the available crash pulse durations.

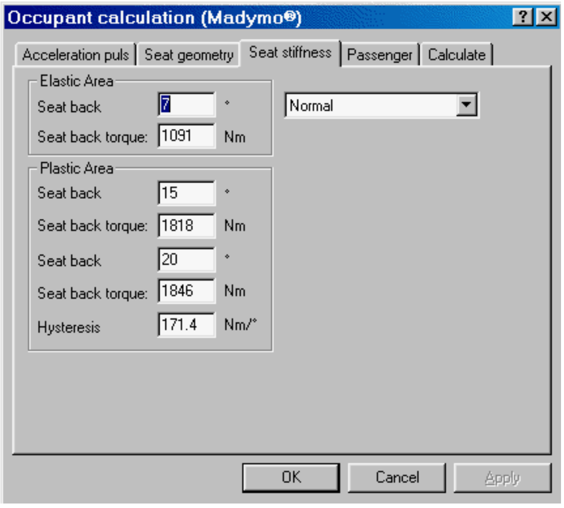
Crash Pulse Duration (ms)	Max. Madymo Sim. Time (ms)
1	2000
10	1910
20	1820
30	1730
40	1640
50	1550
60	1460
70	1370
80	1280
90	1190
100	1100
110	1010
120	920
130	830
140	740
150	650

Seat Geometry



The seat geometry folder is for entering the seat and head restraint dimensions. The size and angle of the seat cushion and the angle of the head restraint cannot be changed.

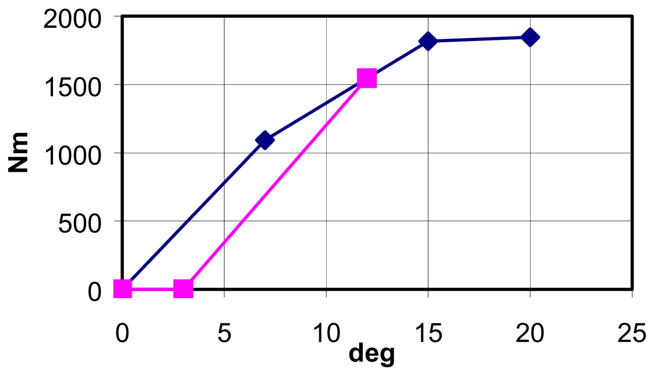
Seat Stiffness



The elastic and plastic stiffness properties can be varied here. This can be done by selecting Normal, Stiff or Soft in the drop down list, or by specifying actual values in the Elastic and Plastic areas.

Three seat back angles and seat back torques can be specified, along with a hysteresis slope to account for energy loss on rebound. The 1st seat back angle (in the Elastic Area of this folder) is the one at which the seat deformation changes from elastic to plastic. No damping is assumed below this angle and specified elastic seat back torque. The 2nd angle and torque specified define a point where the slope of the 1st plastic curve changes to the 2nd. A 3rd torque is specified at a 3rd angle to define the slope of the 2nd plastic curve.

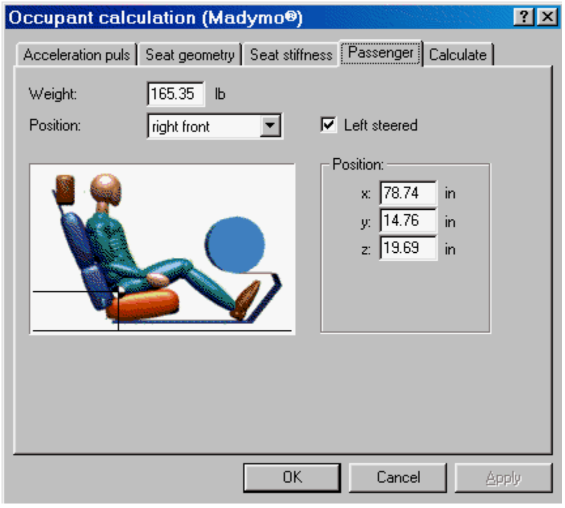
The following figure is a graph of seat back torque vs. angle, using the "Normal" default values. The small change in torque from 15° to 20° seat back angle indicates an almost completely plastic seat back pivot above 15°. The second line in the figure, nearly parallel with the elastic portion of the graph, is the hysteresis curve from a seat back angle of about 12°.



Seatback Pivot Characteristic

Passenger

This folder is for changing the occupant's seated position and weight. A normal seated posture is assumed - there is no option to model unusual postures. The occupant height is 50th percentile male (5'-9" tall). It cannot be changed. There can be only one occupant in the calculation at any one time.

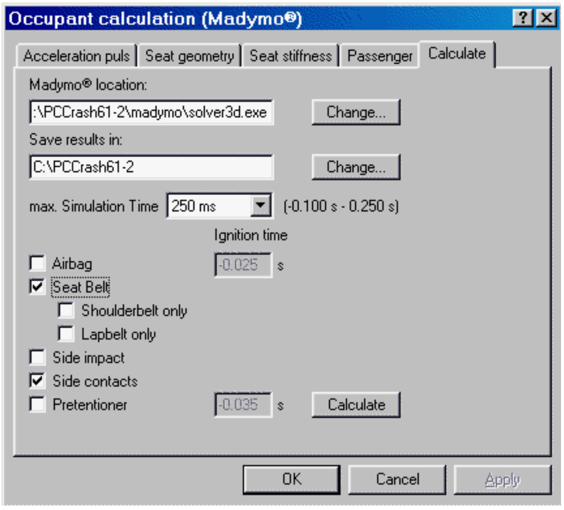


If  Left Steered is checked (default), the steering wheel will appear for the Left Front seated position. If unchecked, the steering wheel will appear for the Right Front seated position.

The occupant's seat position is the occupant's H-point in the vehicle coordinate system, as measured from the front center of the vehicle at ground level.

Calculate

This folder is for specifying directory locations, the simulation time and the type of restraint, and for initiating the Madymo calculation.



The Calculate Dialog Box contains the following:

Madymo® Location

In this text box, the location of the Solver.exe file must be specified. The Solver file is in the Madymo subdirectory of the directory PC-Crash is installed in.

Save Results In

This text box is for specifying the location where the user wishes to store the calculated Madymo results. The results are always stored in subdirectories that are created when the calculations are being performed. The selected directory name must conform to DOS rules (no spaces, maximum 8 characters).

Max. Simulation Time

This drop down list is for specifying the duration of the Madymo simulation after the impact (0 to 2000ms). A time of 100ms prior to impact is always used.

Airbag

An airbag can be used for the driver's or right front seating positions. The point in time when airbag ignition starts can be defined in the adjacent text box. A negative time can be used to compensate for the fact that the time=0 impact point in PC-Crash is at the mid-point of the selected crash acceleration pulse duration.

Note: *The operating system must be Windows NT or 2000 for the airbag calculation to function.*

Seat Belt

This specifies a lap and torso seat belt. The latch plate is modeled as a sliding type unless Side Impact is activated, in which case the lap and torso belt has a fixed latch plate.

The frictional force at the sliding latch plate is modeled as follows:

$$F_{Friction} = F_{Belt} \cdot e^{\mu\alpha}$$

where

$F_{Friction}$ = Belt tension with friction (on side of latch plate where webbing length is increasing)

F_{Belt} = Belt tension without friction (on side of latch plate where webbing length is decreasing)

μ = Latch plate friction coefficient = 0.1

α = Angle change of belt webbing at latch plate

Torso Belt Only

Specifies use of a torso belt with no lap belt.

Lap Belt Only

Specifies use of a lap belt only.

Note: *An unrestrained occupant is modeled by selecting none of the seat belt check boxes.*

Side Impact

This option and the following one are useful for side impacts and angled impacts. The selection of this option changes the sliding latch plate in a lap and torso belt to a fixed type. This is necessary to prevent the latch plate from releasing the belt when the finite element portion of the lap or torso belt reaches the latch plate.

Side Contacts

This defines a vertical side surface to model contact forces between the occupant and an adjacent door panel. The door panel is a full height (roof to floor) vertical plane fixed in position, 10cm (4") inside the outer width dimension of the vehicle.

Pretensioner

This defines a seat belt pretensioner. The point in time when pretensioning starts can be defined in the adjacent text box. A negative time can be used to compensate for the fact that the time=0 impact point in PC-Crash is at the mid-point of the selected crash acceleration pulse duration.

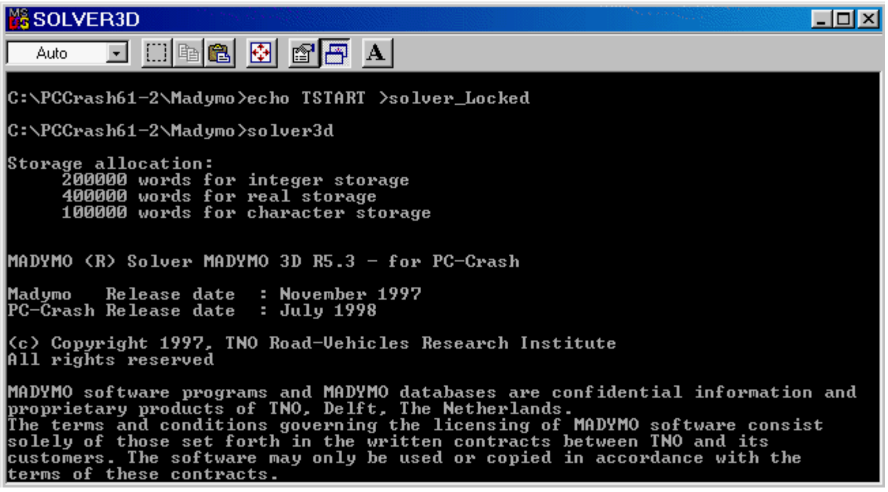
Activating the pretensioner feature introduces a retractor tension force of approximately 1500-1700 N up to a maximum of 12 cm webbing retraction. Retractor locking occurs when the webbing then starts to withdraw from the retractor due to occupant movement.

Calculate

Pushing 

starts the Madymo calculation. The Solver 3D window appears. Depending on the complexity and length of the simulation and the speed of the computer processor, the calculation can take from a few minutes to over an hour. The progress (in % completion) can be seen at the bottom of the Solver 3D window.

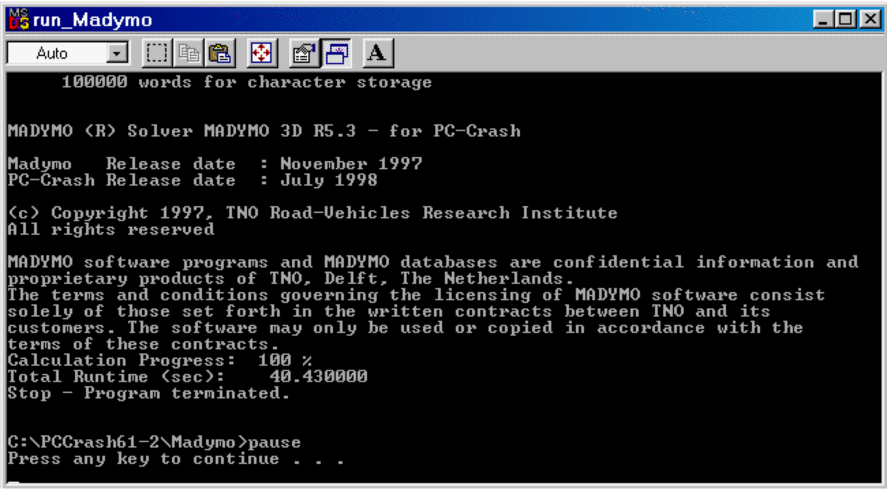
The calculation takes longer the first time a Madymo simulation is run, especially on Windows NT/2000 systems.



Note: *If the Calculate button is greyed out, check that the location of the Solver.exe file has been specified correctly. If it has, the probable cause is that the user aborted a previous Madymo calculation before completion. In this case, a file called "Solver_Locked" must be deleted from the Madymo subdirectory where the Solver.exe file is located.*

If the Calculate button is not greyed out, but the calculation progress at the bottom of the Solver 3D window does not show within a few minutes of when the Calculate button is pressed, check that the Madymo hardware lock has been installed properly. On Windows NT/2000 computers, it has to be installed when the Dongle Install program is run.

After the calculating process finishes, the following text appears:



When the user presses a key to continue, the calculated files are written to the selected directory.

Note: *If the calculation progress does not get to 99% before the program is terminated, this could be due to one of the following:*

1. *The combination of the acceleration pulse time and the selected maximum Madymo simulation time exceeds that shown in the table in the* ***Acceleration Pulse*** *section of this chapter.*
2. *The limits of motion for the occupant have been exceeded. This is indicated in the bottom of the reprint file (*****.rep****) in the directory the Madymo files were written to, by a line such as "Bryant angle for xxx exceeds pi/2". This may occur if there is a severe impact between the occupant and the interior.*

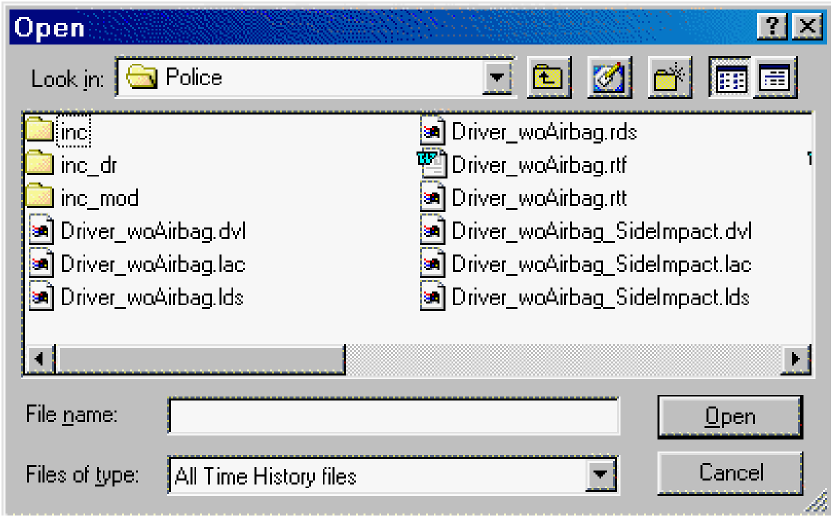
Viewing the Madymo Results

As with the Multibody model, the results of the Madymo occupant simulation can be viewed with the use of diagrams of the results and with 3D animations.

Viewing Madymo with Diagrams

The Madymo diagrams can be viewed by selecting the menu option **Options – Diagrams – Diagrams - Madymo® Diagrams**.

The following window appears, from which the desired diagram file can be opened:

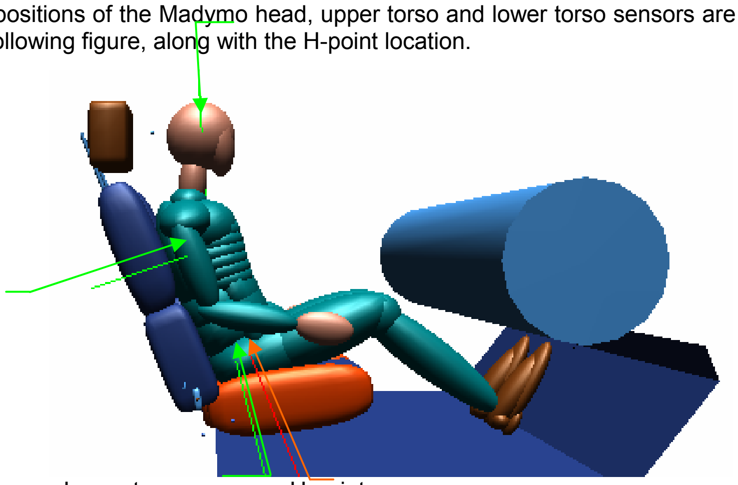


The diagram file types are as follows:

***.dvl**	Resultant displacement (m) and velocity (m/s)	Sternum Head/seat Upper torso/seat Lower torso/seat
***.lac**	Resultant, x, y and z accelerations (m/s²)	Head Upper torso Lower torso EMD/Retractor (same location)
***.lds**	Resultant, x, y and z linear displacements (m)	Head Upper Torso Lower Torso Left Knee Right Knee Buckle point Buckle anchor point Upper belt mounting point Car body (vehicle C.G.) EMD/Retractor (same location)
***.rds**	Resultant, x, y and z relative displacements (m)	Sternum Head/seat Upper torso/seat Lower torso/seat
***.rtf**	Resultant constraint force (N), Inertial X, Y and Z forces (N), Joint x, y and z forces, (N) Body x, y and z forces (N)	On lower lumbar from lower torso On lower torso from lower lumbar On Upper torso from upper lumbar On upper lumbar from upper torso On lower neck sensor from lower neck bracket On lower neck bracket from lower neck sensor On head from nodding plate On nodding plate from head On left knee from femur On left femur from knee On right knee from femur On right femur from knee

		On left middle tibia from upper tibia On left upper tibia from middle tibia On right middle tibia from upper tibia On right upper tibia from middle tibia On left lower tibia from middle tibia On left middle tibia from lower tibia On right lower tibia from middle tibia On right middle tibia from lower tibia
***.rtt**	Resultant constraint torque (Nm), Inertial X, Y and Z torques (Nm), Joint x, y and z torques (Nm), Body x, y and z torques (Nm)	Same as ***.rtf**

The positions of the Madymo head, upper torso and lower torso sensors are shown in the following figure, along with the H-point location.

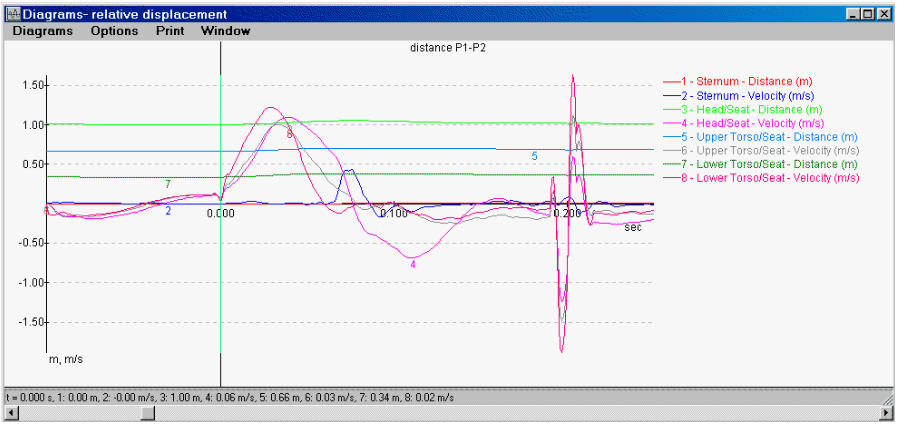


Lower torso H-point

Relative to the H-point of the Madymo dummy in its starting seated position, the sensor locations (in meters) are:

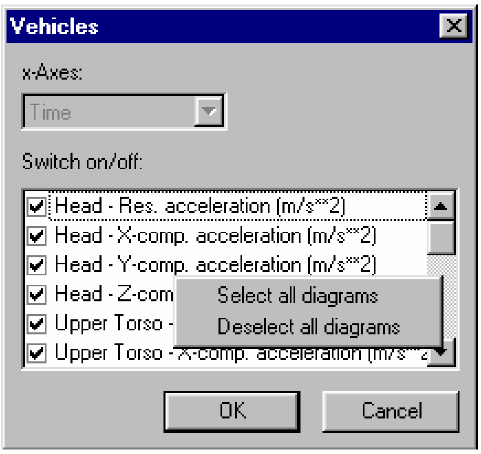
Sensor	x	y	z
Head	-0.14	0	0.65
Upper Torso	-0.18	0	0.28
Lower Torso	-0.04	0	-0.02

Once the desired diagram file is loaded, it will appear with all items graphed on it.

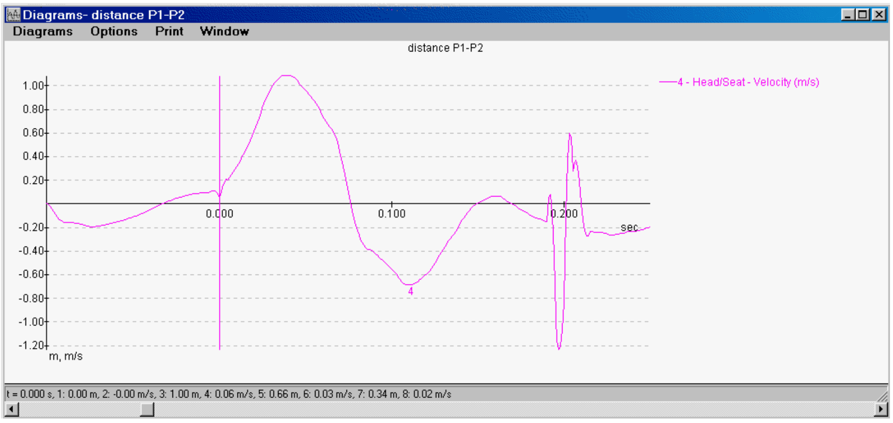


In order to view individual graphs clearly, it is possible to turn some of them off. This is done by selecting the Diagrams window menu option **Options – Diagrams/Axes**, which opens a window in which each item has a check box beside it. Click on each check box to turn it off or on.

If you want to view only one or a few of the diagrams, you can first turn them all off by right-clicking in this window. This brings up a menu which allows all items to be selected or deselected. First deselect them, and then select only the ones you want to view.



For example, selecting only Head/Seat Velocity from the preceding diagrams allows this one graph to be viewed clearly.



The diagram data can also be exported, for example to a spreadsheet program such as MS Excel®. This is done by selecting the Diagrams window menu option **Diagrams – Export Diagram**.

This opens a window in which the data can be saved as a diagram file (*.dia) or a DXF file (*.dxf). Note that the default step width for the diagram data is 0.5 seconds, which should normally be changed to a much smaller value.

***Note:** The ***.dvl** velocities relative to the seat (H-point) are calculated as follows:*

$$v = \frac{d}{dt}\sqrt{x^2 + y^2 + z^2}$$

This formula calculates the change in distance between two points with respect to time, which is not the same as the relative velocity of the selected item with respect to the seat or vehicle. For example, this formula would be equal to 0 if the selected item is rotating around the H-point of the seat, with no change in radius.

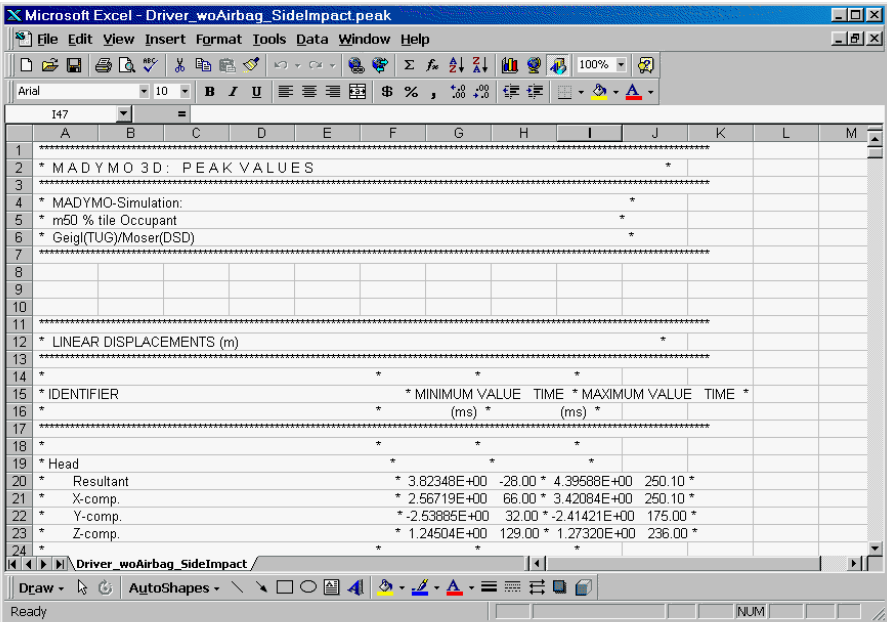
To obtain the relative velocity of the selected item with respect to the seat or vehicle, the following formula should be used:

$$v = \sqrt{\dot{x}^2 + \dot{y}^2 + \dot{z}^2}$$

*For example, to calculate the impact velocity of an occupant's head against the side window, use the relative x, y and z displacement data provided in the ***.rds** file to calculate the relative x, y and z velocities. This can be done easily by exporting this data to a spreadsheet.*

Peak Values

Peak values of the Madymo diagrams are outputted in tabular form to the file ***.peak**. These can be viewed in a text editor or spreadsheet program such as MS Excel®.



Viewing Madymo in 3D

Madymo animations are made the same way that PC-Crash animations are – refer to the menu item description for View **– 3D Window**.

For the Madymo simulation, the first step is to load the *.kn3 file from the directory in which the Madymo simulation was saved. This is done by selecting the **Animation – Load Madymo® Kin3 File** menu item in the 3D Visualization window.

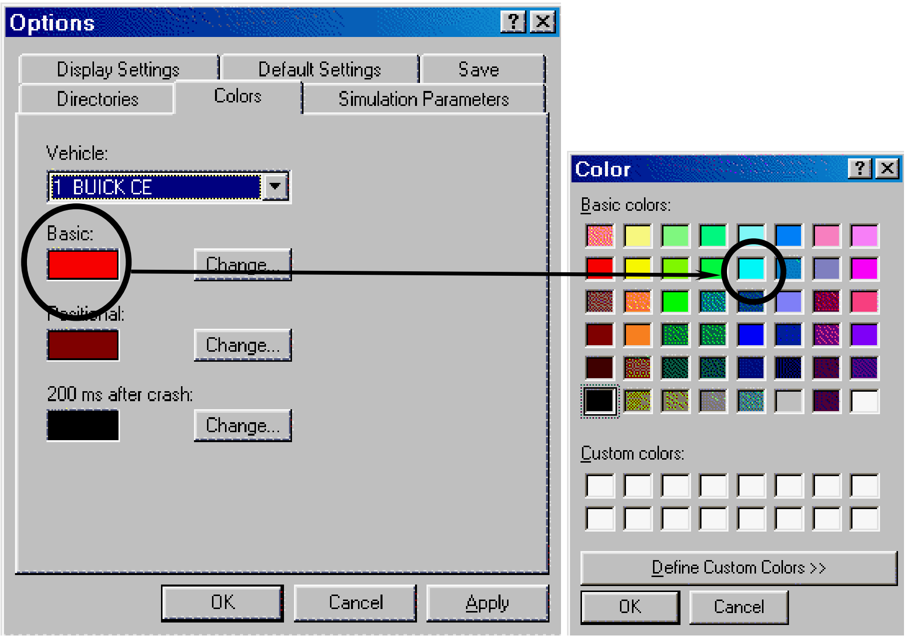
A window appears in which the ***.kn3** file for the desired Madymo run can be selected. Once the ***.kn3** has been loaded, the Madymo multibody occupant and interior will be visible in the 3D window and on the main screen.

If the PC-Crash project from which the Madymo run was calculated is open, the Madymo occupant and interior will be placed in the correct position in the correct vehicle. To view the occupant, first place the camera in the desired position on the main screen, using the Camera button . The camera parameters can then be adjusted further in the 3D Visualization window.

The vehicle body normally obstructs the view of the occupant. This can be overcome in several ways, as follows:

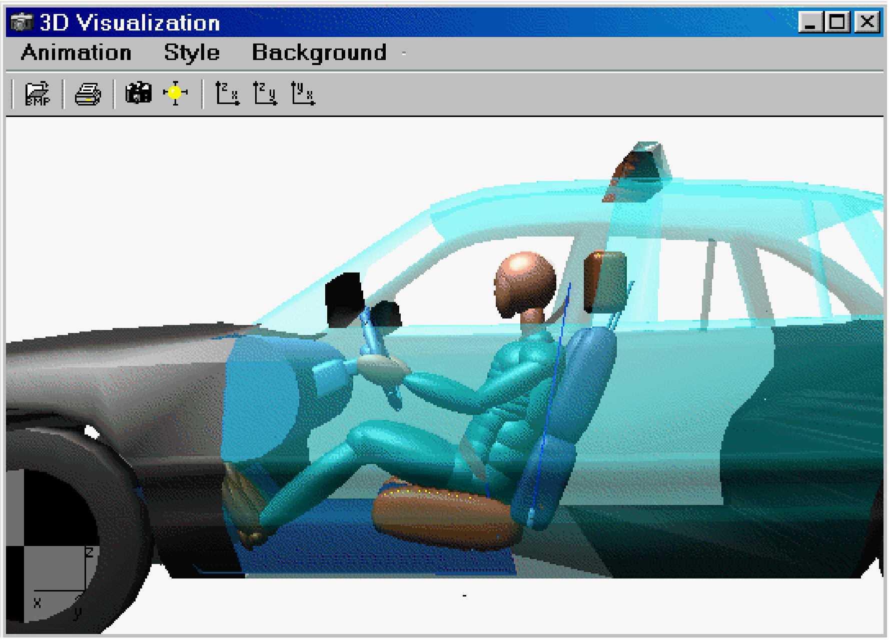
Making the Vehicle Body Transparent

The vehicle body color can be changed to make it transparent. This is done by firstly selecting **Options – Options – Colors**. Then, click on the Change button for the basic color of the vehicle in question. Change the color to the transparent aqua blue in the second row.



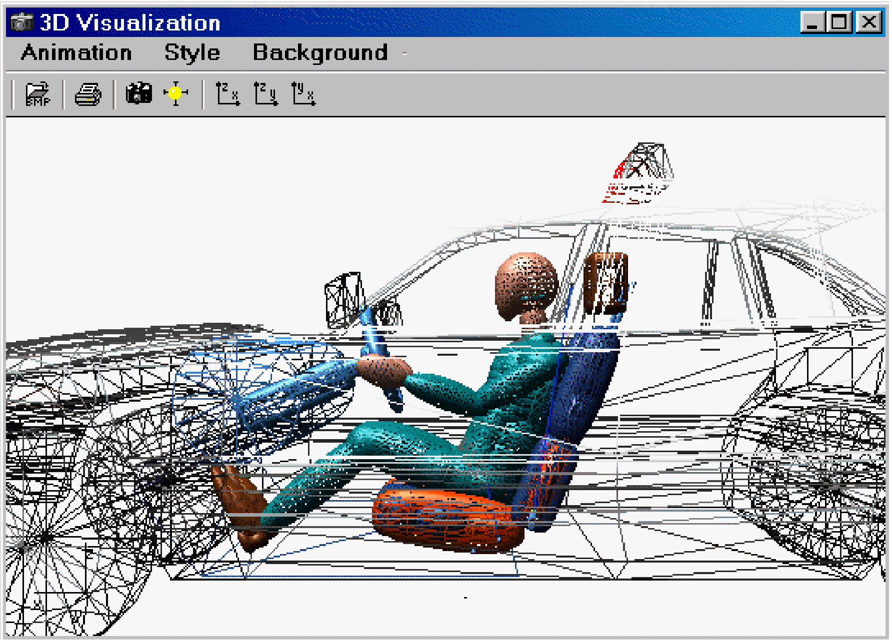
If a vehicle DXF shape is attached to the vehicle, only the portion of the vehicle that is in front of the occupant needs to be changed to the transparent color. This can be done by selecting **Vehicle – Vehicle DXF – ☑ Edit Drawing** and using the Change Line Style button to change the color of selected components.

The occupant will now be visible through the transparent body.



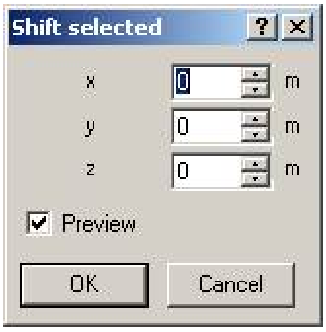
Making the Vehicle Body a Wire Frame

The vehicle body can be changed to appear as a wire frame in the 3D window. This is done by selecting the 3D window menu option **Style – Display Options** and checking Wireframe View. Note that this also changes the Madymo model to wire frame.

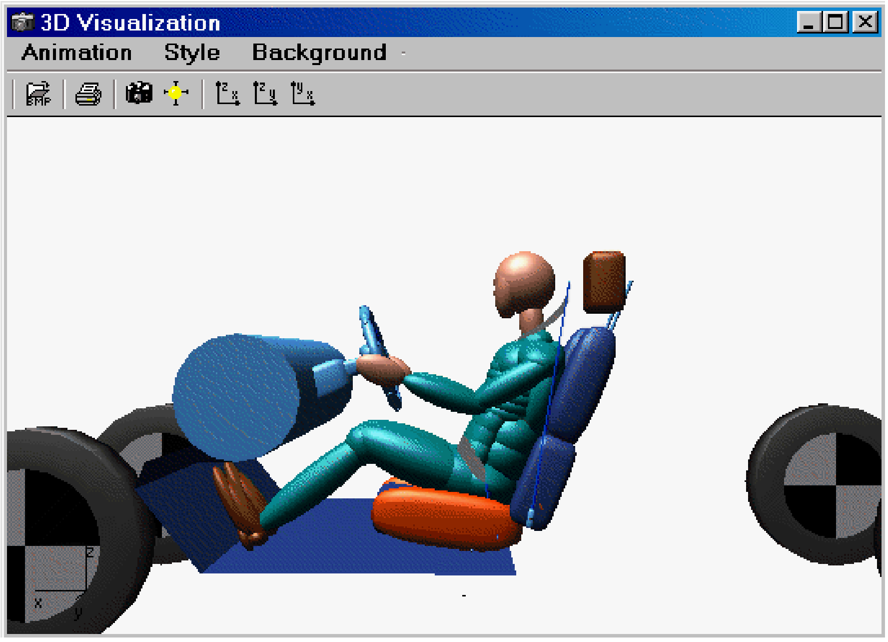


Shifting a DXF Vehicle Body

If a 3D vehicle shape has been attached to the vehicle, it can be shifted out of the way for viewing of the occupant. This is done by selecting **Vehicle – Vehicle DXF** - **Edit Drawing**. Then choose Shift Selected (3D) in the **Draw Toolbar**.

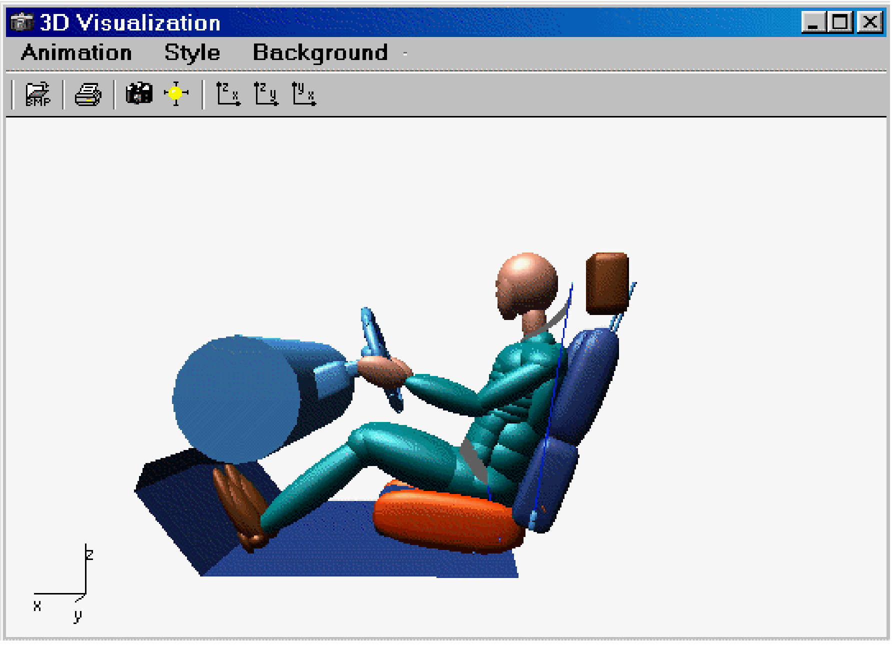


Shift the vehicle body vertically (in the z direction) a sufficient distance so that it is not visible in the 3D camera view of the occupant (usually 5m or 16 feet is adequate). The Madymo occupant and seat will then be visible without any vehicle bodywork in the way.



Turning Off Vehicles in the 3D Window

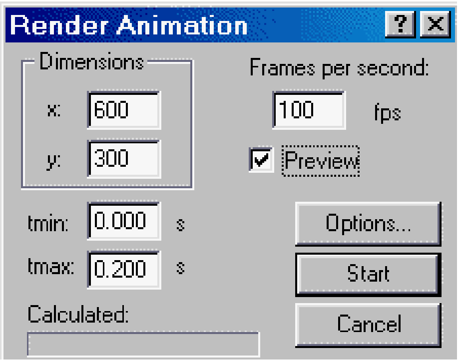
The 3D view of the vehicles can be turned off by de-selecting ☑ Vehicles in the **Style – Display Options** menu item of the 3D window. As a result, only the Madymo occupant and interior components are visible in the scene.



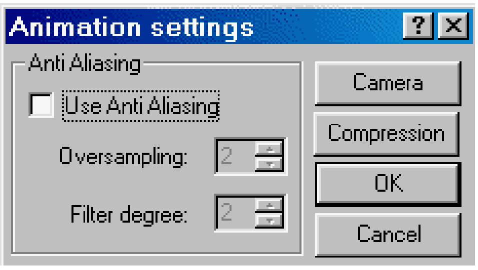
Rendering the Madymo Animation

In the 3D Visualization window, select the menu option **Animation – Render**. You will be asked for a file name under which to save the animation.

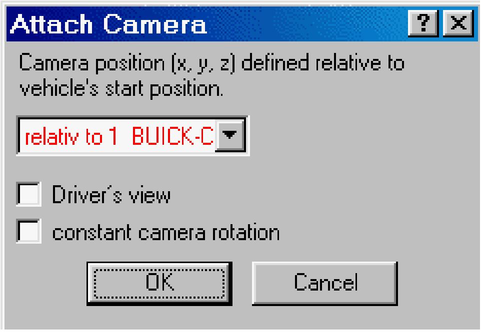
After entering and saving a file name, the Render Animation dialog box opens. Since the length of the occupant motion simulation will typically be only a few hundred milliseconds, use a high number of frames per second (100 frames per second has been selected in the following example).



Generally, the best way to view occupant motion is with the camera attached to the occupant's vehicle. Press the Options button in this dialog box to open the Animation Settings dialog box.

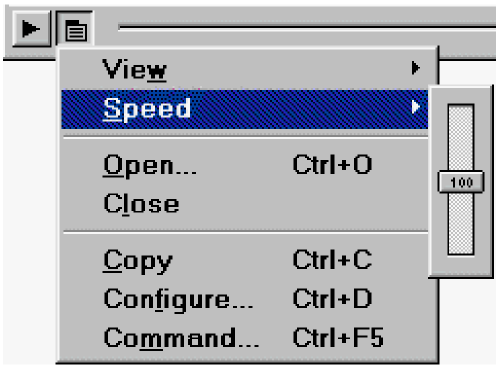


Then, press the Camera button to open the Attach Camera dialog box. Select "Relative to …" in the drop down list to select the correct vehicle to attach the camera to.



Once the camera is attached, return to the Render Animation dialog box and press the Start button to render the animation.

After it is rendered, you may wish to view the animation in slow motion. Click on the Options button near the lower left corner of the Animation window to open the following menu.



Select the Speed menu item and slide the slider bar to a low speed such as 5% of actual. The occupant animation can then be viewed in slow motion by clicking on the arrow button. The Animation window main slider bar can also be used to view each animation frame in detail.

Index