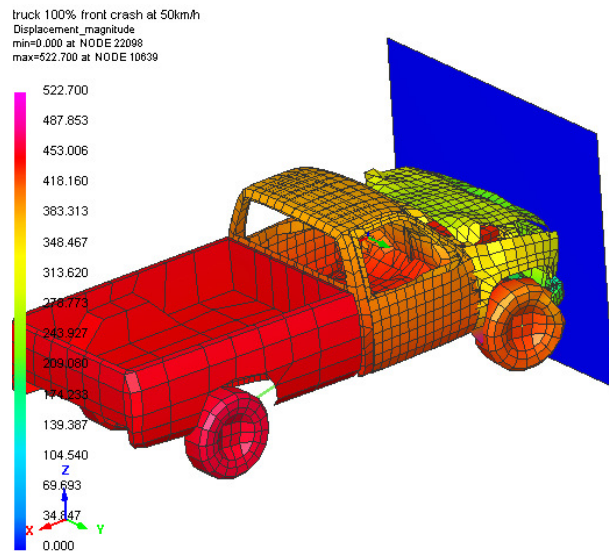


# Tutorial 8

## Frontal Crash of a Simplified Truck Model



### Problem description

**Outline** Frontal crash of a simplified truck model against a rigid wall is performed. Many of the basic elements needed to perform a full vehicle crash simulation are presented and typical results from such a simulation investigated.

**Analysis type(s):** Explicit

**Element type(s):** 3D Shell elements, Rigid Bodies, Beams and Bars

**Materials law(s):** Elasto-plastic

**Model options:** Boundary conditions, Spotwelds, Contact

**Key results:** Stress distributions and Impact forces

**Prepared by:** Anthony Pickett, ESI GmbH/Institute for Aircraft Design, Stuttgart  
**Date:** January 2008  
**Version:** V4 (Updated May 2012 for Visual-Crash PAM V8.0)

## Background information

---

### Pre-processor, Solver and Post-processor used:

- **Visual Crash PAM:** To examine the truck model.
- **Analysis (PAM-CRASH Explicit):** To perform the explicit Finite Element analysis.
- **Visual Viewer:** Evaluating the results for contour plots, deformations and time history information.

### Prior knowledge for the exercise

This exercise is not worked through in detail and it is assumed that the reader has gained a good working knowledge of using **VCP** from tutorials 1 through 7. The completed model of the simplified truck is provided and is examined using **VCP**.

Note the model is for teaching purposes and is a very simplified representation of a crash model using circa 10,000 elements; today's models can typically use several millions of elements and are highly complex.

---

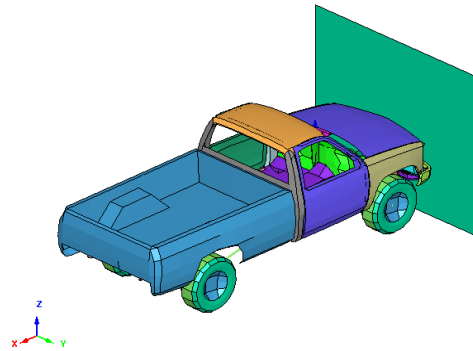
### Problem data and description

Units: kN, mm, kg, ms

Description: The simplified truck model has approximately 10,000 nodes and elements in 48 material groups.

Loading: Impact mass of the truck is 1300kg which impacts the rigid wall at 13,88m/sec.

Material: The materials used are mainly elasto-plastic mild steel.



### Supplied datasets

The completed truck model, including all entities, loadings, contacts and material/part properties is provided and has the name:

**Truck\_Front.pc**

---

## Contents

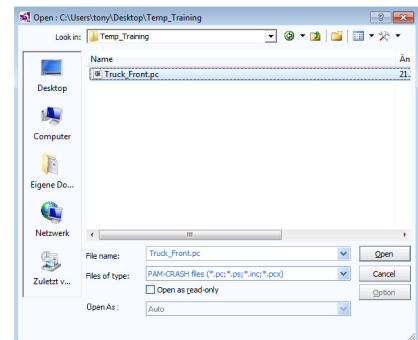
Tutorial 8.....	1
Frontal Crash of a Simplified Truck Model .....	1
Problem description.....	1
Background information.....	2
Part 1: Using Visual Crash PAM to examine the simplified truck model.....	3
Part 2: Examining and visualising some analysis results .....	5

## Part 1: Using Visual Crash PAM to examine the simplified truck model

### Starting Visual-Crash PAM and examining the model

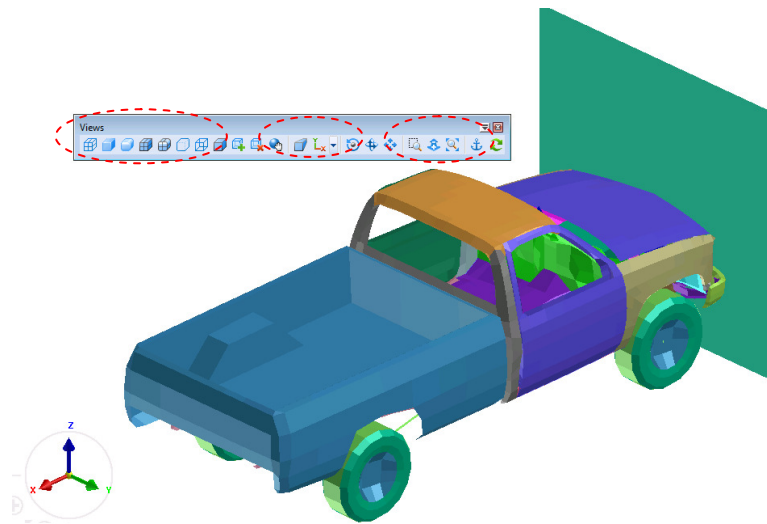
Start **Visual Crash PAM** and open the truck model using **Open File**, then select the file:

**Truck\_Front.pc**



### Positioning and centering of the model

As in tutorials 1 through 7 the various viewing tabs can be used, together with the mouse, to change viewing angles, position and type of viewing.



In the same manner as done previously in Tutorial 7 we shall look at the different model entities. Again, this is most easily done via the explorer window by clicking on the entity of interest and opening it in Edit mode with the right mouse key.

The following gives some brief details of the entities assigned to this truck model:

**Parts and Materials:** There are 48 parts and 48 associated materials. Note that this is not usual and normally one material with specific mechanical properties is assigned to many different parts each having a different thickness. The shell elements all use the 103 'iterative' elasto-plastic law and plasticity is defined using a power law function. You will also see no material strain rate effects are defined in this simplified model.

The Belytschko-Tsay shell element is used throughout with 'stiffness and plastic modulus' hourglass control (parameter ISHG=1 on the material cards). This control is effective and absorbs less energy than the alternative 'elastic modulus' option. Viscous hourglass controls are also available, but are rarely used in PAM-CRASH. Reduced integration elements (ISINT=1) are CPU more expensive but could be used if hourglassing was detected to be a problem (NB it rarely is!).

## Tutorial 8: Frontal crash of a simplified truck model

- Initial Velocities:** Two initial velocity groups are defined for the car and the wall. The wall is modelled using a single shell element and its velocities are set to zero by this option. This shell element is used here for visualisation purposes and as a contact surface in the segmented rigid wall definition (see later). All nodes in the truck are given an initial impact velocity of 13.89m/sec.
- Pressure faces:** Constant internal pressure is applied to the inner surfaces of the tires. This is reasonably accurate for tyre pressure, but the modern approach is to use an airbag definition so pressure increases with tyre volume change during impact.
- Contacts:** The entire truck is placed in one single self contact definition (Type 36). Today, this is computationally efficient and reliable; but care must be taken to make sure there is no internal energy increase due to initial penetration of the elements. This is especially necessary in coarse or approximately meshed models like this training model. Notice in this model the contact thickness is only 0.5mm in order to prevent (minimise) shell element initial penetration problems.
- A type 34 contact is used to define contact between the vehicle (slave nodes) and the wall (master elements). The wall is a fully fixed single shell element.
- Rigid wall:** A rigid wall could be used to define contact between the car and the wall impact surface. However, this kinematic constraint can cause problems with other kinematic constraints such as rigid bodies. Recent versions of PAM-CRASH do data checks to avoid possible conflicts. The wall contact is therefore modelled as a Type 34 penalty contact.
- Rigid bodies:** In total of 40 conventional rigid bodies have been defined to provide simple rigid connection between different parts of the model.
- Tied contacts:** Two tied contacts have been used to connect coarse to fine meshes and avoid the need for local remeshing. This can be a useful option, but it should not be used in areas where important load transfer, or deformation mechanisms, may occur.
- Outputs:** In total 11 nodal time history outputs and 6 sections have been defined. These give time history information of individual nodes and force transfer through sections of the structure. The information is saved on the **.ERF** file.
- Pam Controls:** The usual controls have been specified; however, in order to reduce CPU time some timestep control options have been imposed (see control option TCTRL). These timestep controls add some mass and/or modify local element mechanical properties to meet the user specified minimum timestep limits. These options must be used with care and some experience is needed.

## Tutorial 8: Frontal crash of a simplified truck model

### Part 2: Examining and visualising some analysis results

The dataset is run with **PAM-CRASH** and the results are post-processed with **Visual Viewer**.

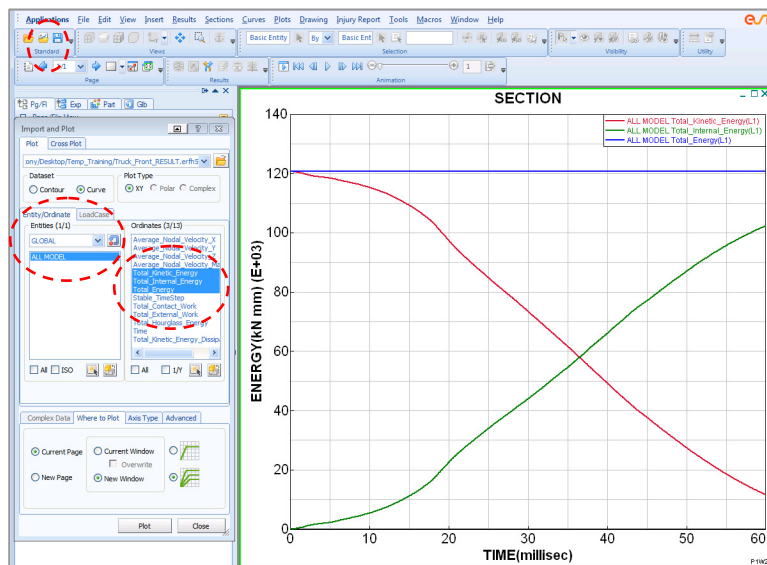
For results start **Visual Viewer** and open the results file, **Truck\_Front\_RESULT.erfh5**.

If information on the model units system is requested select MM, K, MS; this is important for plots and especially if any numerical filters are to be used.

#### **Global energy plots and numerical stability**

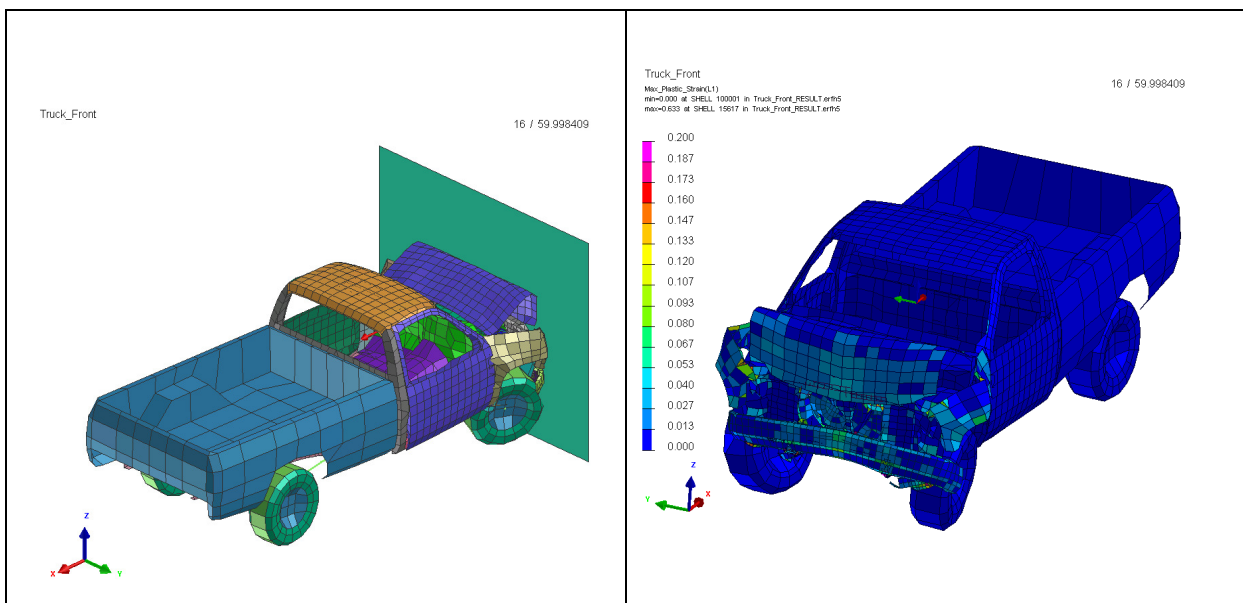
First, the solution stability should be checked by examining the system energy balance. For this study there is no energy input to the system during the crash (from external loads or imposed velocity/displacements) and therefore the total energy should be constant.

Plot the three global energy time histories (Internal, External and Total) using the options highlighted. The total energy is constant indicating a stable solution.



#### **Deformations**

Next it is best to check deformations and contacts to see if results are sensible. The **Truck\_Front\_RESULT.erfh5** file also contains deformation and contour plots (e.g. plastic strain) at regular states during the crash. Some example results are shown below.

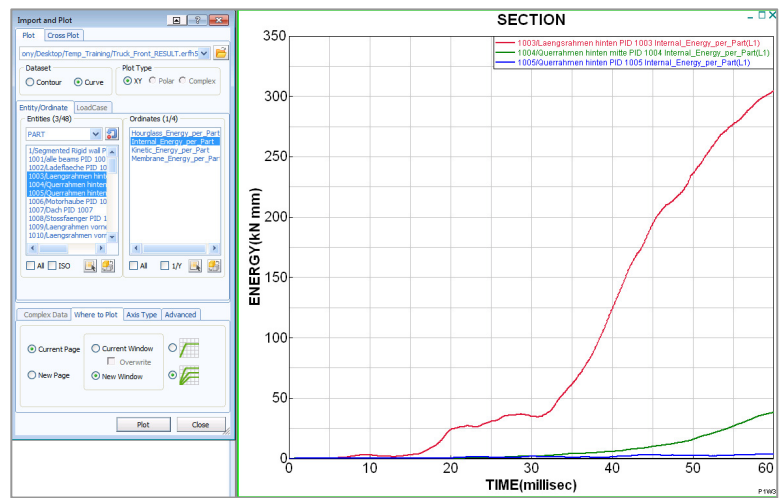


## Tutorial 8: Frontal crash of a simplified truck model

### Other time histories

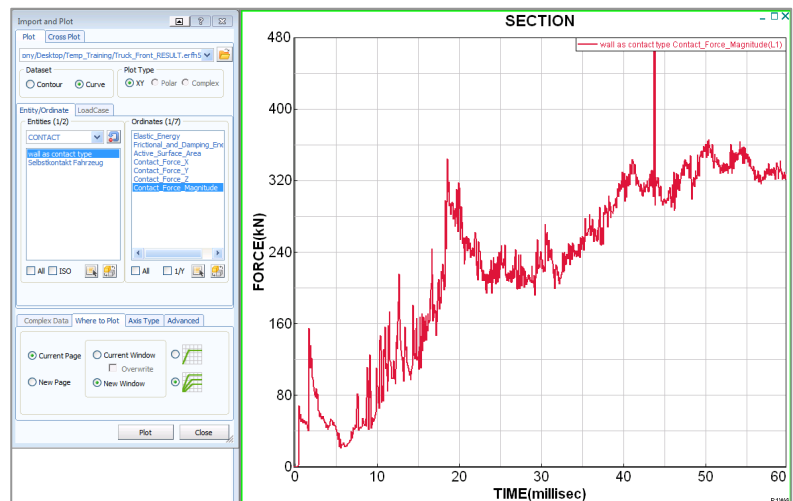
#### Material energies

A variety of other time history information is available. The adjacent plot shows typical internal energy time history plots (**Entity Type > PART > Mat. Internal energy**) for three selected materials.



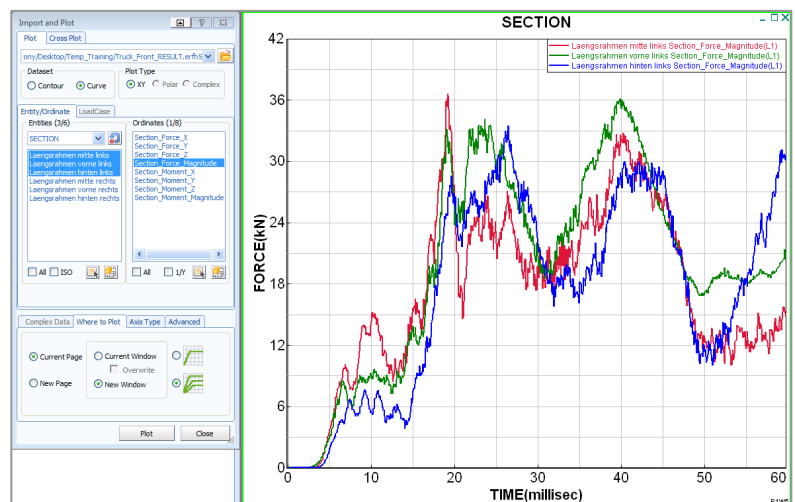
#### Wall impact force

The wall impact force time history is obtained from the appropriate contact force time history (wall contact).



#### Section force time histories

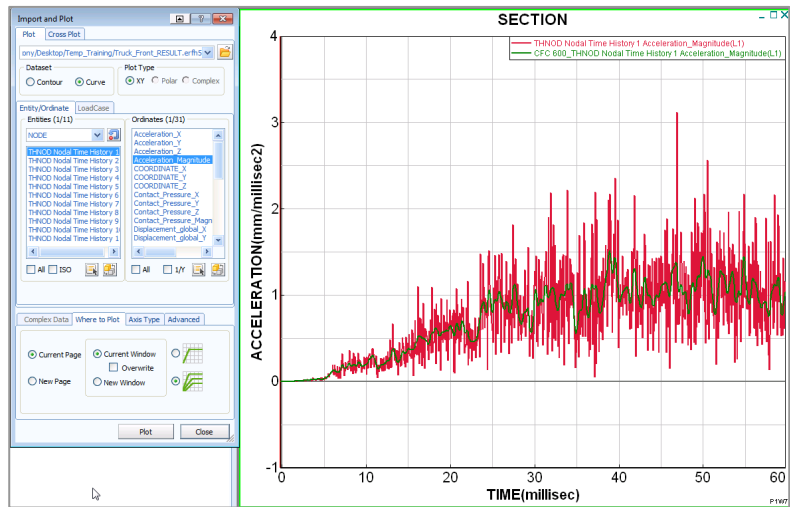
The adjacent view shows section force time histories for three sections (**Entity Type > SECTION**). These are especially useful to understand the load transfer mechanisms in the structure.



## Tutorial 8: Frontal crash of a simplified truck model

### Nodal time histories

Finally, time history information at nodes is shown with (**Entity Type > NODE**) and selection of the required node and component. The displacement and velocity time histories generally have a smooth shape; the acceleration signal, however, has a very high frequency that needs to be filtered to get a meaningful curve. The adjacent plot shows the unfiltered (Acceleration Magnitude - Component) for one node, together with the filtered version of the same data (in green).

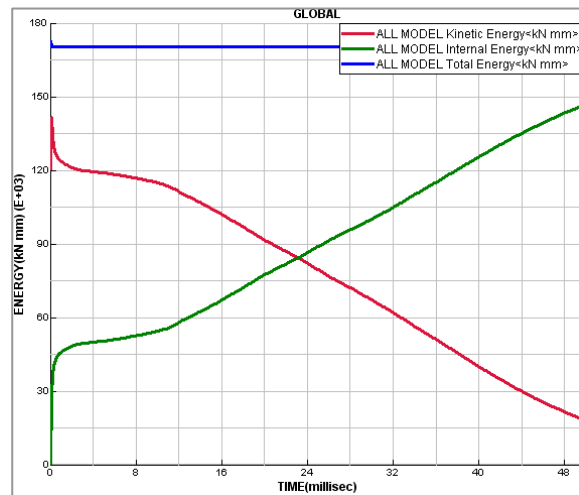


**Remark:** It is important to activate the **Pre-Filter** flag in the **Pam Controls** when using filters in the post-processing. Also, the correct filter should be selected when comparing results to filtered test results.

### An unstable analysis

As a useful exercise it is suggested to change the value of  $h_{\text{cont}}$  in the self contact for this model from 0.5 to 2.0 mm, export the new dataset (**Truck\_Front\_Hcont2.pc**) and rerun the analysis.

Investigating the **.THP** file in **Visual Viewer** it is seen that there are sudden jumps in energies at the start of the simulation. Plotting internal energies for individual materials can help to identify the location of this increase; alternatively, looking at contour plots of plastic strain is another possibility.



The instabilities are due to local element penetrations in the contact interface, causing temporary high contact forces and materials plastification. Either the mesh should be 'cleaned', or options available in **VCP** used to remove the initial penetrations (e.g. **Checks > Penetration/ Intersection**). The use of a small  $h_{\text{cont}}$  value can be effective, but is not really the best solution.