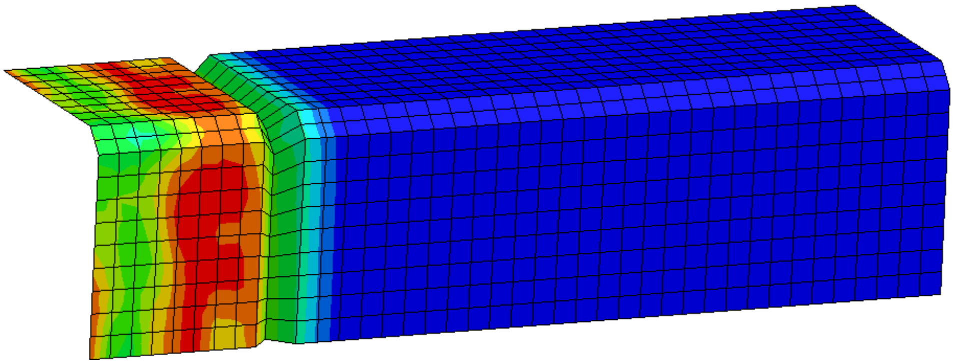
# Basic Tutorials

# LS-DYNA / LS-PrePost

## Ex. 3. Crash test



# Contents

# 1 Introduction

## 1.1 Purpose

- Learn to use mesh tools.
- Get familiar with contact definitions.
- Understand how to affect the time step

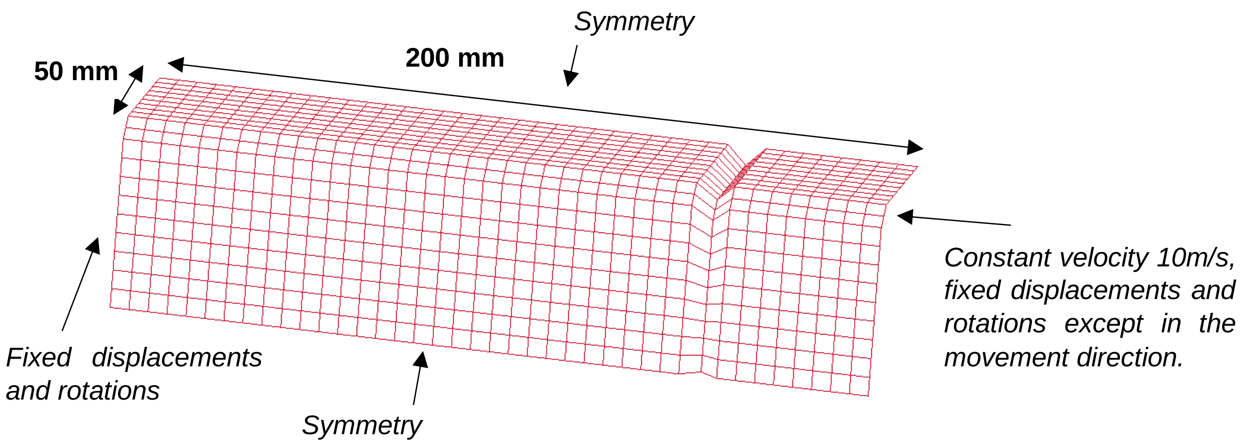
## 1.2 Prerequisites

- Basic knowledge in the finite element method.
- Understand the steps in tutorial **1 - Getting Started.**

## 1.3 Problem Description

The task is to crush a quarter model of a so called crash box (100x100x200 mm) used in cars to absorb impact energy to see how contacts are defined and behave in LS-DYNA. A quarter of the box will be modeled. Boundary conditions and dimensions of the box is shown in the figure.

| **Material properties** | |
|---|---|
| Density, $\rho$ | 7850 kg/m3 |
| Young's modulus, $E$ | 210 GPa |
| Poisson's Ratio, $\nu$ | 0.3 |
| Yield limit | 230 MPa |
| Tangent modulus | 500 MPa |

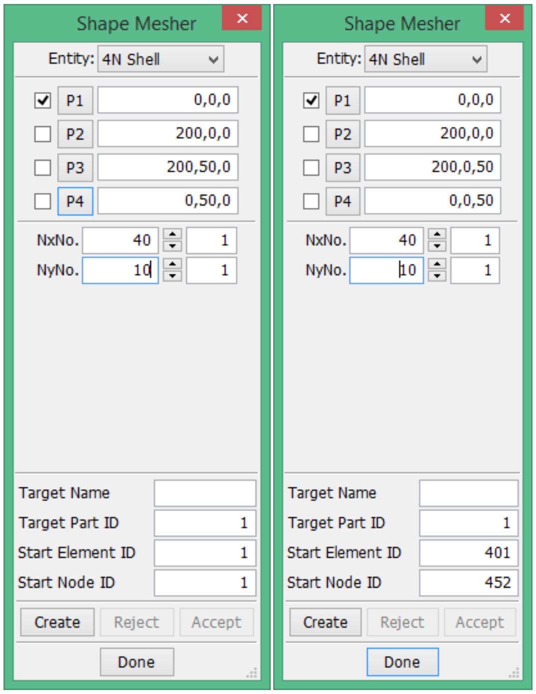



## 1.4 Data files

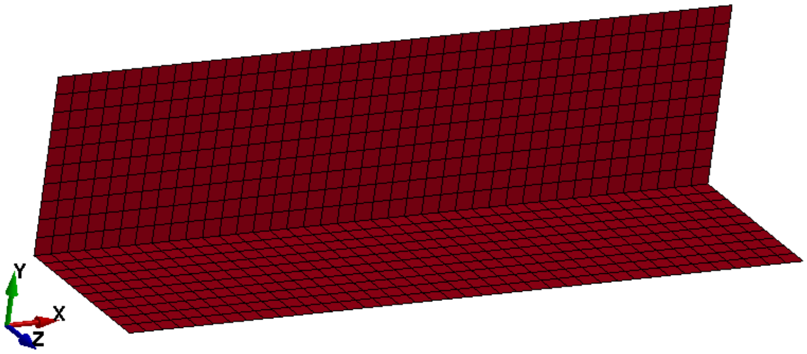
The final resulting keyword model is available as **crashbox_results.k**.

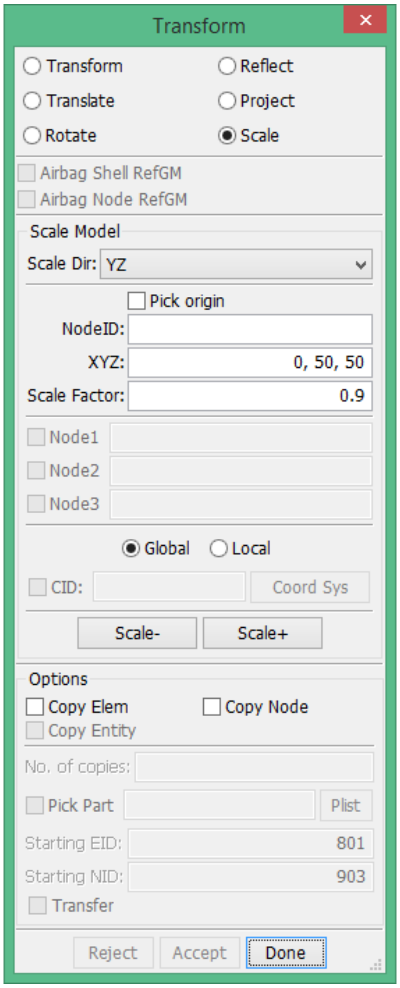
# 2 Create the model

## 2.1 Mesh

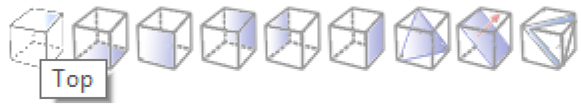



- Click **Mesh > ShapeM.**
  - Set **Entity: 4N Shell.**
  - Write the coordinates as in the left figure.
  - Set **NxNo = 40** and **NyNo = 10**.
- Click **Create** and **Accept.**
- Now change the coordinates to the ones in the right figure and change **Target Part ID** to **1**.
- Click **Create** and **Accept.**

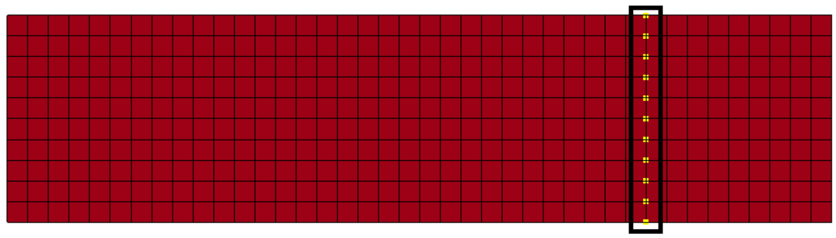




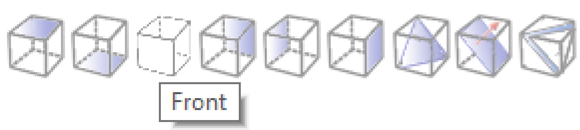

- Click **EleTol > Transf** and select **Scale**.
- Set **Scale Dir:** to **YZ**.
- Write **0, 50, 50** in **XYZ:** and **Scale Factor = 0.9**.
- Click on the **Top** view in the Floating Toolbar.



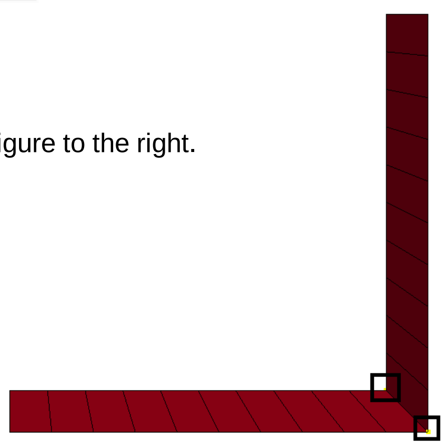

- In the node selection box, select **Area**.

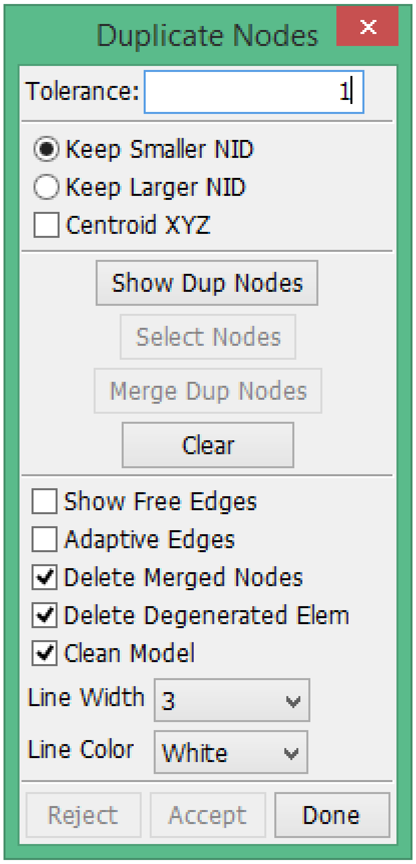


- Select the 22 nodes in the **tenth** (for example) node row, counted from the right.
- Click **Scale+**.
- Click **Accept**.
- Click on the **Front** view in the Floating Toolbar:

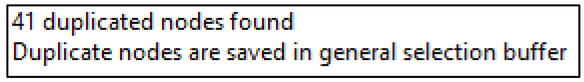



- Change **Scale Factor** to **0.95**.
- Select the **82** nodes in the corner, see the figure to the right.
- Click **Scale+**.
- Click **Accept**, then **Done**.



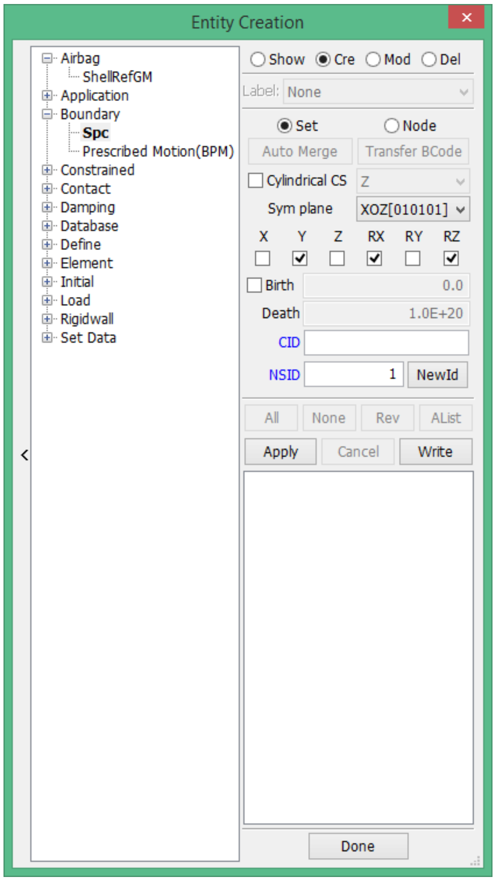



- Click **EleTol > DupNod**.
- This tool can be used to find nodes that are within a certain distance of each other and merge them together.
- Set **Tolerance = 1**.
- Click **Show Dup Nodes**.
- Following message will be shown in the message box:



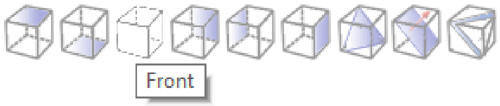

- The duplicated nodes will be highlighted.
- Click **Merge Dup Nodes**. Since **Keep Smaller NID** is activated, the smaller node ID will be retained when the nodes are merged.
- Click **Accept**, then **Done**.

## 2.2 Boundary conditions



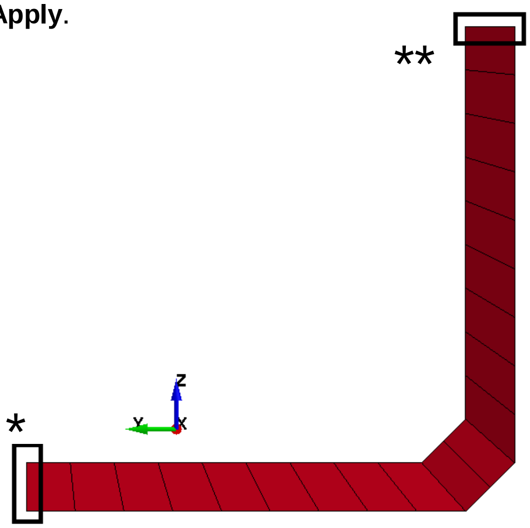

Apply symmetry boundary conditions:

- Click on the **Front** view in the Floating Toolbar.

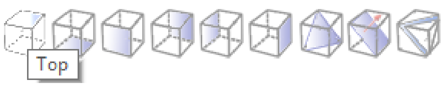



- Click **Model > CreEnt.**
- Double-click **Boundary > Spc.**
- Select **Cre** and activate **Set.**
- Set **Sym plane** to **XOZ**
- In node selection box, activate **Area** and select the 41 nodes in *.
- Click **Apply**.

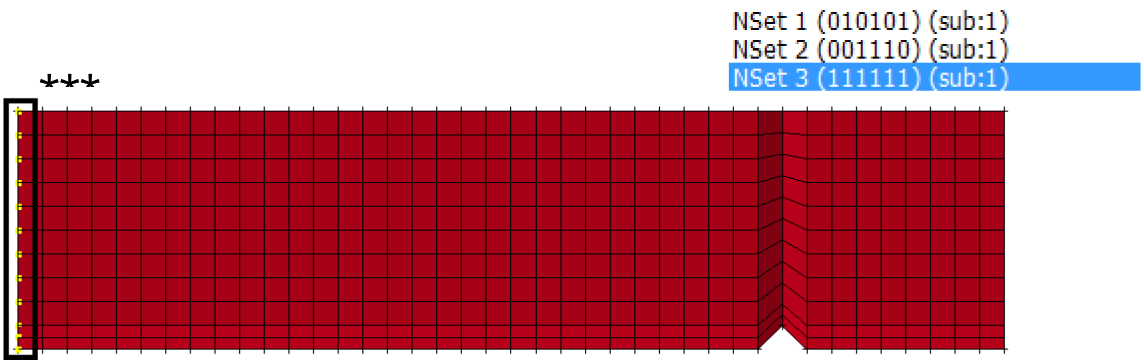
- Select the nodes in ** and set **Sym plane** to **XOY**.
- Click **Apply**.



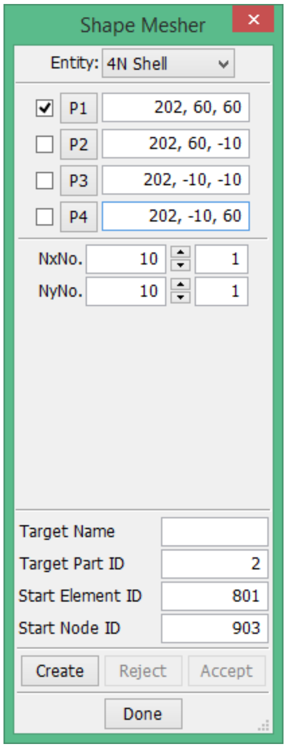

- Click on the **Top** view in the Floating Toolbar



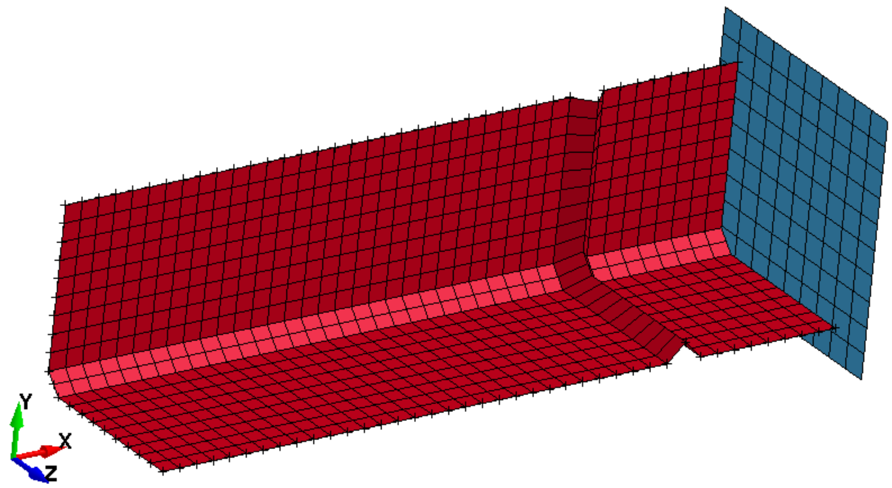

- Select the nodes in ***, change **Sym Plane** to **All Fix** and click **Apply**
- The constraints to the left should now have been created.
- Click **Done**



## 2.3 Rigid plate



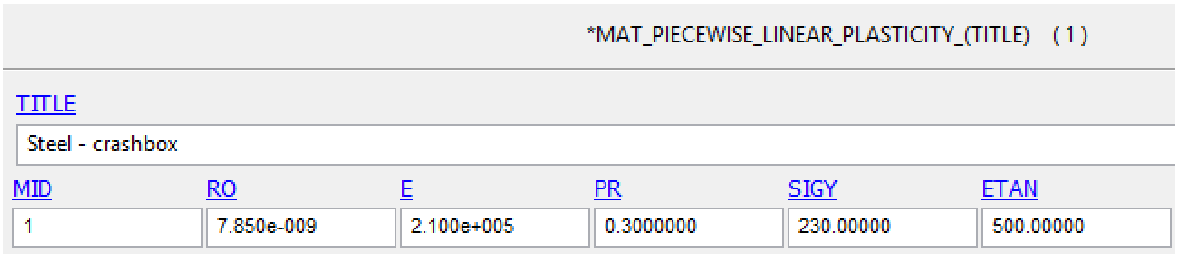

- Click **Mesh > ShapeM**.
- Set **Entity: 4N Shell**.
- Enter the vales as in the figure.
- Click **Create**, **Accept** then **Done**.



## 2.4 Material properties

To create the material card for the crash box, do as follows.

- Click **Model > Keywrd**. Make sure that **All** is activated.
- Double-click **MAT > 024-PIECEWISE_LINEAR_PLASTICTY**.
- Write in the data in the figure below and then click **Accept** and **Done**.



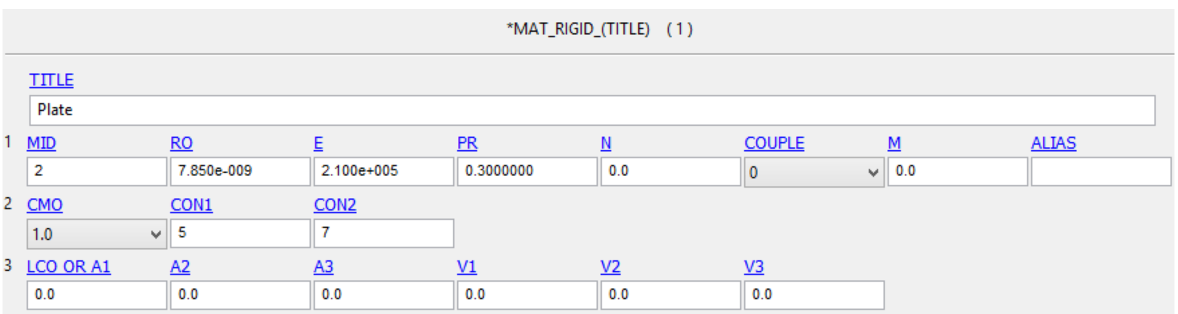
*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (1)

TITLE: Steel - crashbox

| MID | RO | E | PR | SIGY | ETAN |
|---|---|---|---|---|---|
| 1 | 7.850e-009 | 2.100e+005 | 0.3000000 | 230.00000 | 500.00000 |

The plate is to be rigid, thus define also a rigid material:

- Double-click on **MAT > 020-RIGID** in the Keyword Manager.
- The newly created plate will be considered as rigid, which means that it cannot be deformed.
- Enter the values as in the figure below. Then click **Accept** and **Done**.



*MAT_RIGID_(TITLE) (1)

TITLE: Plate

| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| 1 | MID | RO | E | PR | N | COUPLE | M | ALIAS |
| | 2 | 7.850e-009 | 2.100e+005 | 0.3000000 | 0.0 | 0 | 0.0 | |
| 2 | CMO | CON1 | CON2 | | | | | |
| | 1.0 | 5 | 7 | | | | | |
| 3 | LCO OR A1 | A2 | A3 | V1 | V2 | V3 | | |
| | 0.0 | 0.0 | 0.0 | 0.0 | 0.0 | 0.0 | | |

Even though the part cannot be deformed, the Young's modulus **E** and Poison's ratio **PR** should be set. This is because they are needed to compute the contact stiffness, if a contact definition is used.

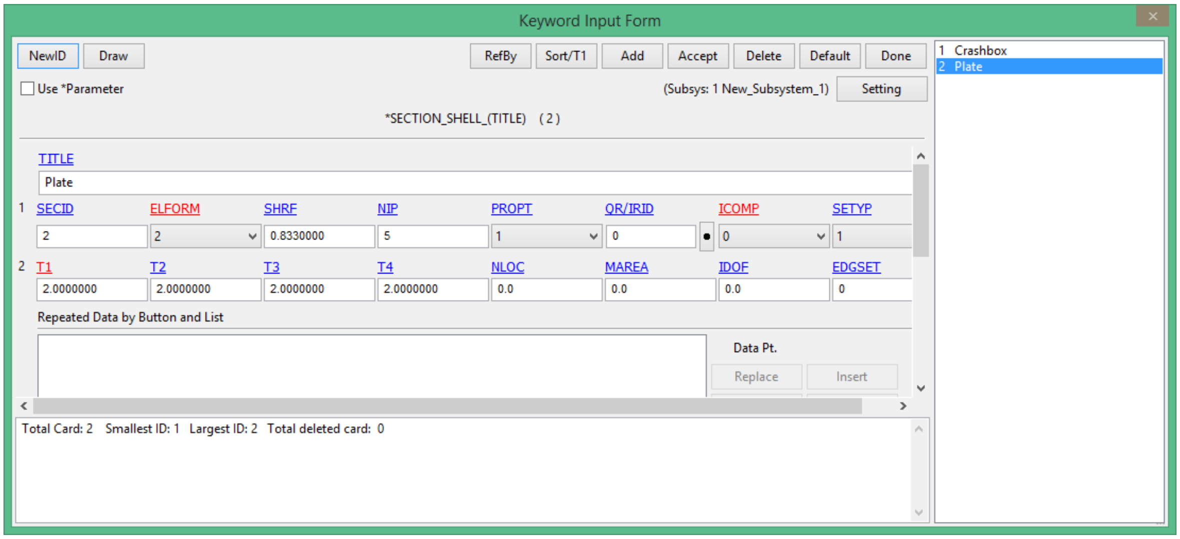
The parameters **CMO**, **CON1** and **CON2** are used to constrain the center of mass of the rigid body in translational and rotational degrees of freedom:

- **CMO = 1**: constraints are applied in global directions.
- **CON1 = 5**: translational constraints in Y and Z.
- **CON2 = 7**: constrained in all rotations.

## 2.5 Element properties

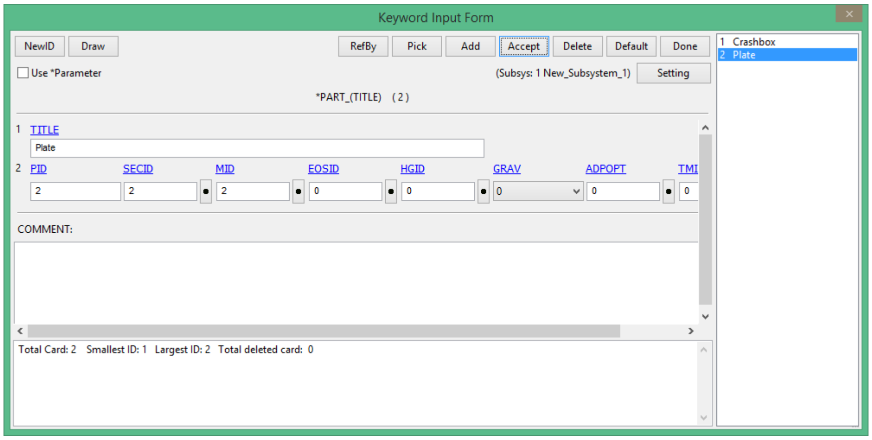
To set the element properties for the crash box and rigid plate, define 2 cards:

- In the Keyword Manager, double-click **SECTION > SHELL**.
- Write **Crashbox** as title.
- Set **SHRF = 0.8333** and **NIP = 5** (recommended, but not default in LS-DYNA).
- **T1-T4 = 1.5**. (activates after setting T1=1.5, then press Enter)
- Click **Accept**, then **NewID**.
- Change the title to **Plate**.
- Change the thickness to **2** instead.
- Click **Accept**, then **Done**.



Now attach the created element cards to the parts:

- In the Keyword Manager, double-click **PART > PART**.
- Select **1 shell_4p** (Crashbox) in the top right corner and assign the newly created material and section. There one can also change the name of the part**.**
- Assign material and section to **2 shell_4p** (Plate).
- Click **Accept,** then **Done**.



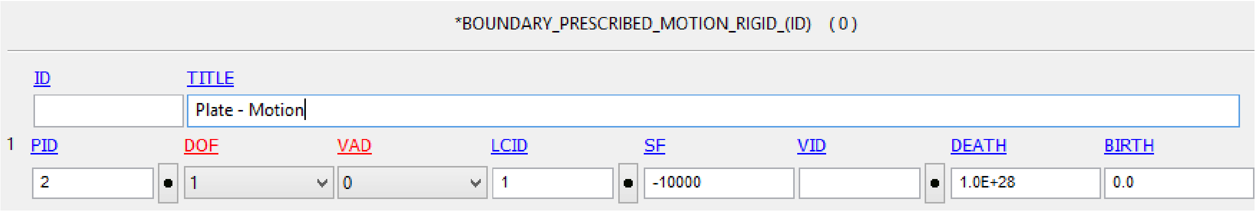
## 2.6 Motion

The rigid plate is to crush the crash box. Apply the motion of the plates as follows. First create a curve defining the motion:

- Click **Model > Keywrd**.
- Double-click **DEFINE > CURVE.**
- Name the curve, e.g. to **Velocity**.
- The points for the curve will be written in **A1** and **O1.**
    - Write **0** and **1,** Click **Insert.**
    - Then **1** and **1, Insert.**
- Click **Accept,** then **Done.**

Now use the defined curve to apply a motion to the rigid plate:

- Double-click **BOUNDARY > PRESCRIED_MOTION_RIGID** in the Keyword Manager**.**
- Enter the values as in the figure below.

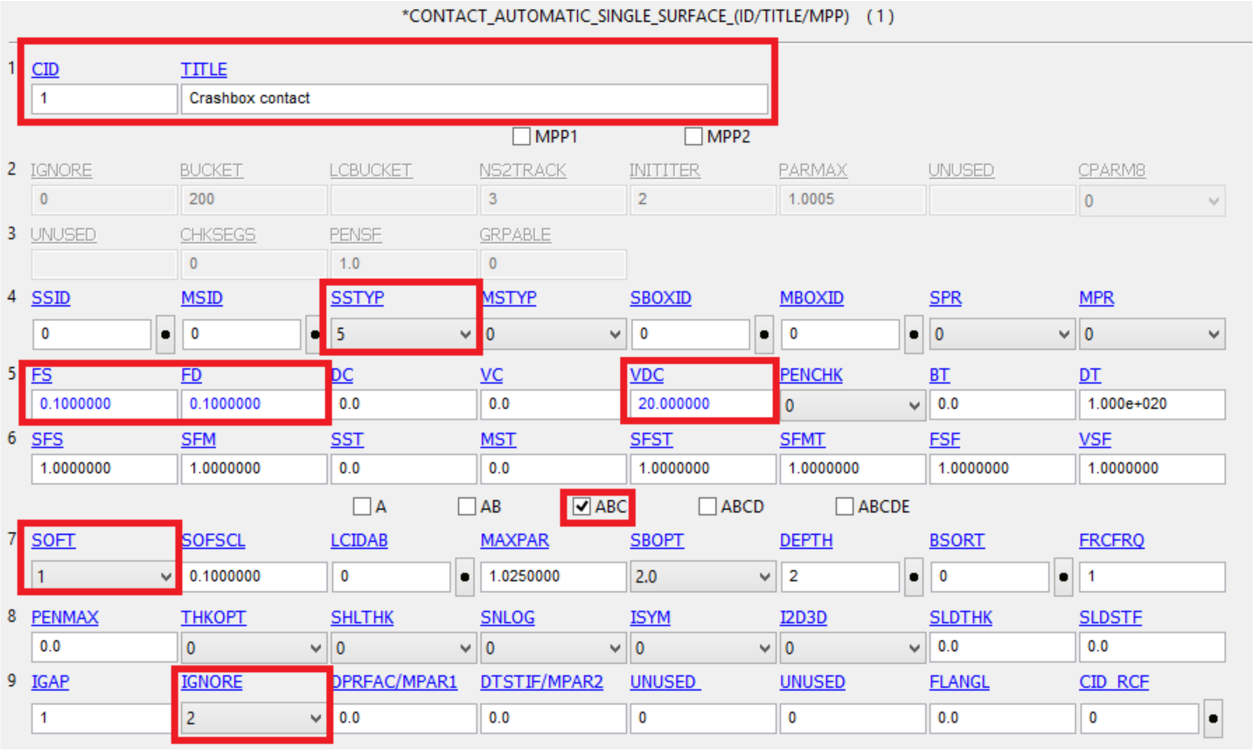



Note: The options above imply that a constant velocity motion will be applied in the negative (negative scale factor) x-direction for the rigid Plate.

## 2.7 Contact

By using contact definitions, the structural domains will interact instead of just passing through each other. For self-contact, meaning that nodes and elements from the same part are able to contact each other, a single surface type of contact in LS-DYNA has to be chosen:

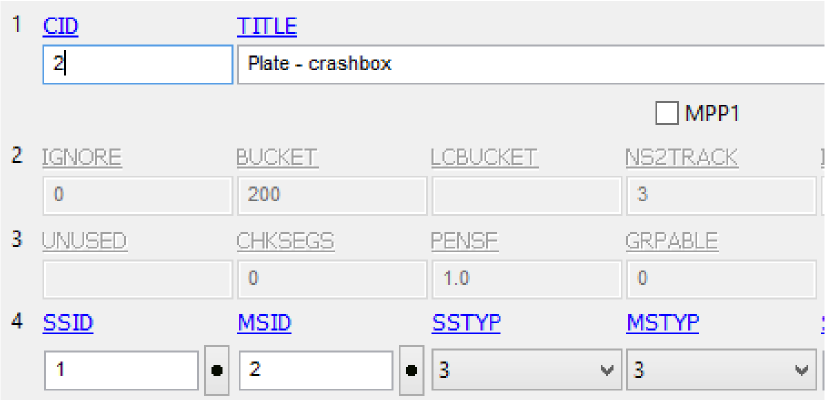
- Click **Model > Keywrd**, double-click **CONTACT > AUTOMATIC_SINGLE_SURFACE**. For a single surface contact, contact is considered between all parts in the slave list.
- Set **SSTYP = 5**, all parts in the model will then be included in the single surface contact. (Another way is to set SSTYP = 3, then create a part set of the parts that will be included in the contact and assign that to SSID.)
  Set **FS** and **FD** to **0.1**.
- Other settings that we recommend to use are **VDC = 20, SOFT = 1** and **IGNORE = 2**. Read more about contacts in section **5. Summary**.
- Click **Accept**, then **Done**.



## 2.8 Tied contact

The end of the crash box will be tied “welded” to the rigid plate, using a so called tied contact:

- Double-click **CONTACT > TIED_SHELL_EDGE_TO_SURFACE_OFFSET** in the Keyword Manager.
- The tied contact will be used to tie the top nodes of the crashbox to the plate.
- Enter the values as in the figure below.
- Click **Accept**, then **Done**.




Note: The slave nodes in SSID (crash box) will be tied to a master surface in MSID (for shell elements) if the distance between them are smaller than $\delta$.

$$\delta_1 = 0.6\ x\ (thickness\ of\ slave\ node + thickness\ of\ master\ segment)$$
$$\delta_2 = 0.05\ x\ \min(master\ segment\ diagonals)$$
$$\delta = \max(\delta_1, \delta_2)$$

In this case:

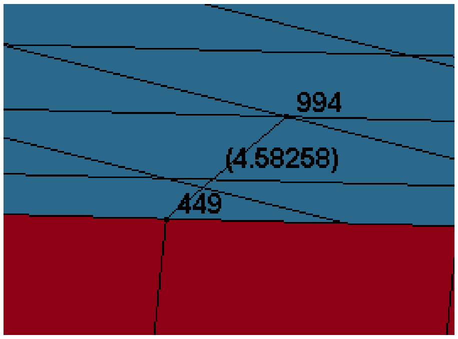
$$\delta_1 = 0.6\ x\ (1.5 + 2.0) = 2.16$$

$\delta_2 = 0.05\ x\ 9.8995 = 0.495$ (very easy to calculate by hand in this example, since all elements in the plate have the same size.)

$$\delta = 2.16$$

Thus, for the tying to occur, the distance between the slave node and the master surface must therefore be smaller than 2.16 mm. Check this as follows:

- Click **EleTol > Measure**.
- Item should be **Dist N2N**.
- Click on a node on the plate and at a node at the top of the crashbox.




dx=2 dy=-1 dz=4 dist=4.58258

The distance showed in the model is the length between the nodes. To get the closest distance between the crashbox and the plate, check the **Message Window**. The distance in x-direction (shortest in this case) are **2 mm**. The slave nodes at the top of the crashbox will then be tied to the master surface (shorter than 2.16 mm).

## 2.9 Termination time

Set the termination time of the simulation:

- Click **Model > Keywrd**.
- Double-click **CONTROL > TERMINATION**.
- Set **ENDTIM** to **0.015**.
- **Accept**, then **Done**

## 2.10 Output

Set the output of d3plot data from the simulation:

- Click **Model > Keywrd**.
- Double-click **DATABASE > BINARY_D3PLOT**.
- Set **DT = 2e-4**.
- **Accept**, then **Done**.

Also add output of global statistics:

- In the Keyword Manager, double-click **DATABASE > ASCII_option.**
- Activate **GLSTAT** (global data) and set **DT = 1e-6.**
- Click **Accept**, then **Done**.

## 2.11 Control cards

Set the following control cards:

- In the Keyword Manager, double-click **CONTROL > ENERGY**.
- Set all parameters to **2**. This will calculate the stated energies and include them in the energy balance.
- Click **Accept**, then **Done**.

Other control cards that are recommended to use:

- ***CONTROL_ACCURACY**: Set **OSU = 1** and **INN = 4** (2 would also work in this case)
- ***CONTROL_SHELL**: Set **ESORT = 1**.

## 2.12 Save

The model is now ready to be saved, use **File > Save As > Save Keyword As**. Chose a folder path and name your file **crashbox.k** for example. **Note** that the folder path cannot contain any spaces.

Close **LS-PrePost.**

## 3 Run the simulation

Run the simulation using **LS-Run**, see exercise **1. Getting Started** for more information on how to do this. The simulation runs in less than a minute to completion. Use the “SMP Single” **Preset**.

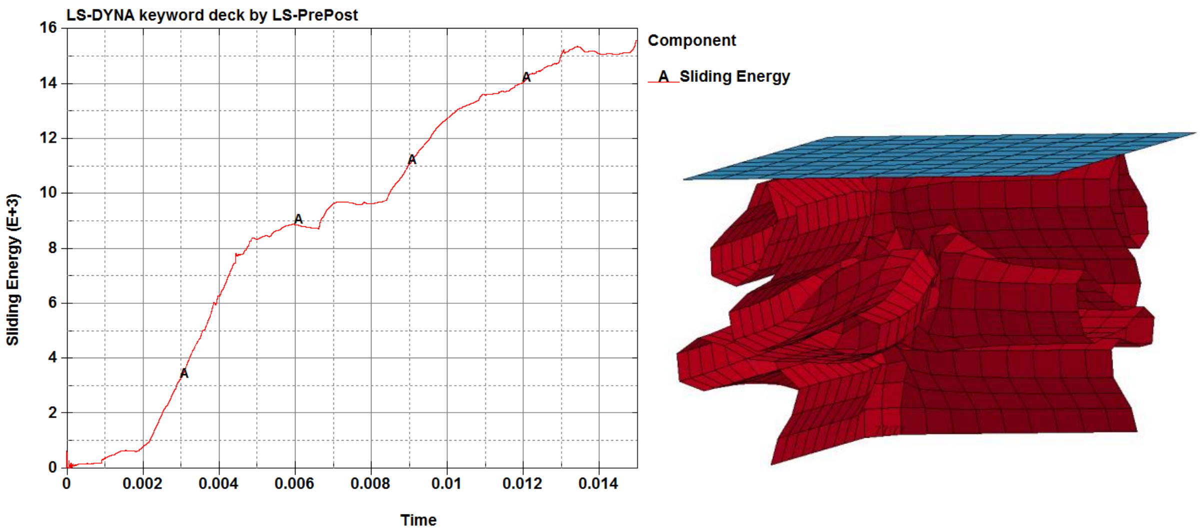
## 4 Post processing

Open the **d3plot**. Click **Forward** in the **Animation Toolbar** to see what happens.

Plot the contact energy, i.e. frictionally dissipated energy and penalty spring energy:

- Click **Post > ASCII**.
- Select **glstat*** and click **Load**.
- Select **8-Sliding Energy** and click **Plot**.
- Check so your sliding energy is positive, since negative sliding energy indicates contact problems.

Also plot all energies from **1-Kinetic Energy** to **9-External Work** to get an idea of the size of the energies and work involved.

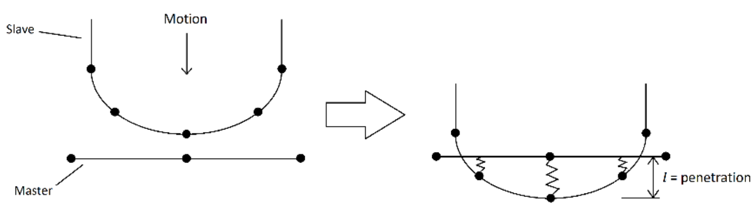


# 5 Summary and comments

The focus in this exercise was to set up a model containing contact definitions. Below follow some comments to the exercise and recommendations.

## 5.1 Contact definition

In contacts, there are generally a master side and a slave side. Most recommended contacts are based on the penalty method. The penalty method is used in contacts to prevent penetrations of a slave node through a master segment. This is accomplished by applying interface springs between penetrating slave nodes and corresponding master segments.




The contact force applied from the interface springs are calculated as:

$$f_s = -lk_c n_c$$
$$f_s = applied\ force$$
$$l = length\ that\ slave\ node\ has\ penetrated$$
$$k_c = contact\ interface\ stiffness$$
$$n_c = normal\ direction\ in\ which\ the\ force\ is\ applied$$

This contact interface stiffness $k_c$ will be calculated in different ways, depending on the used formulation. Which formulation to be used are set by the SOFT parameter in the CONTACT keyword

## 5.2 Contact parameters

In this tutorial, we used several parameters in the contact definitions, these are explained below.

- **FS and FD:** Static and dynamic coefficient of friction
- **VDC**: The viscous contact damping parameter. Originally, contact damping was implemented to damp out the oscillations that existed normal to the contact surfaces in sheet metal forming simulations. It has been found that contact damping is often beneficial in reducing high-frequency oscillation of contact forces in crash or impact simulations.
- **SOFT:** This non-default method calculates the stiffness of the linear contact springs based on the nodal masses that come into contact and the global time step size. The resulting contact stiffness is independent of the material constants and is well suited for treating contact between bodies of dissimilar materials. The **SOFT = 1** option is recommended for impact analysis where dissimilar materials come into contact.

- **IGNORE**: Ignore initial penetrations. “Initial” in this context refers to the first timestep that a penetration is encountered. By setting **IGNORE = 2**, LS-DYNA allow initial penetrations to exist by tracking the initial penetrations. However, penetration warning messages are printed with the original coordinates and the recommended coordinates of each slave node given.

## 5.3 Contact recommendations

Both single surface and tied contact have been used. For self-contact and contact between different parts, AUTOMATIC_SINGLE_SURFACE is recommended. Some recommendations for contacts:

- Uniform meshes improve result in all contact parts
- Avoid sharp corners
- Do not double define contacts
- Avoid initial penetrations (check in LS-PrePost using Contact Check under Model Checking)

Avoid very small contact thicknesses

## 5.4 Time step

Earlier, we checked that the time step was smaller than the contact time step. If the time step wouldn’t be smaller than the contact time step, or if we were unsatisfied with the time step for any other reason e.g. long simulation times, it would be desirable to change the size of the time step.

For explicit simulations, the time step is determined for all deformable elements in the model. The expression varies depending on if which type of element that are used. The general idea is:

$$\Delta t_e \approx \frac{L_e\sqrt{\rho}}{\sqrt{E}}$$

$$L_e = Charteristic\ length\ of\ an\ element$$
$$\rho = Density$$
$$E = Young's\ modulus$$

The critical time step is determined as the minimum of all calculated time steps:

$$\Delta t_c = \min(\Delta t_e)$$

The actual time step $\Delta \boldsymbol{t}$ is calculated by scaling the critical time step, default in LS-DYNA is $\Delta \boldsymbol{t} = \mathbf{0.9}\Delta \boldsymbol{t_c}$**.** The scale factor **TSSFAC** (0.9 as default) can be changed in **CONTROL_TIMESTEP**. To get more detailed information about the critical time step, see section “Time Step Control” in the Theory Manual.

By observing the time step calculation, one can see that by changing certain material properties, the time step can be increased or decreased. A regular approach is mass scaling i.e. modify the mass of the parts in the model. One simple way is to change the density of the material, which is not recommended since it in some cases require a large increase in mass in order to get a noticeable change in the time step. A more stable and more commonly used mass scaling method is applied in **Optional exercise no. 2**.

# 6 Optional exercises

Optional exercises:

1. At which thickness of the plate will the crashbox NOT be tied to the plate? Use the following equations.

$$\delta_1 = 0.6\ x\ (thickness\ of\ slave\ node + thickness\ of\ master\ segment)$$
$$\delta_2 = 0.05\ x\ \min(master\ segment\ diagonals)$$
$$\delta = \max(\delta_1, \delta_2)$$

   Change the thickness of the plate to check your calculations.

2. Add the keyword **CONTROL_TIMESTEP** and set **DT2MS = -5e-7**. If the time step for an element becomes smaller than **|DT2MS|**, mass will be added to those elements so the time step becomes equal to **|DT2MS|**. The minimum time step allowed is then **TSSFAC x |DT2MS|**. In this case, 0.9 x |-5e-7| = **4.5e-7**.

3. While running the simulation, notice that the time step never becomes lower than 4.5e-7. Use **glstat** to see how much mass that have been added to the model, both in mass unit and percentage. To obtain information about the added mass for the different parts, add **matsum** in **DATABASE_ASCII_option**.