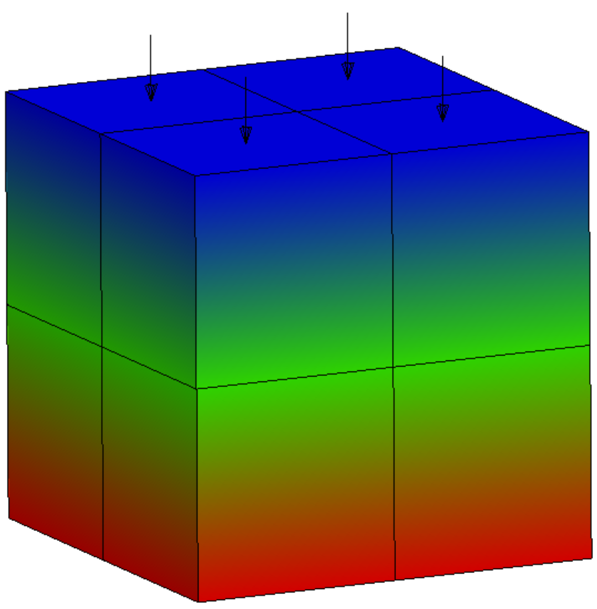
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 1. Getting started



Contents

1 Introduction

1.1 Purpose

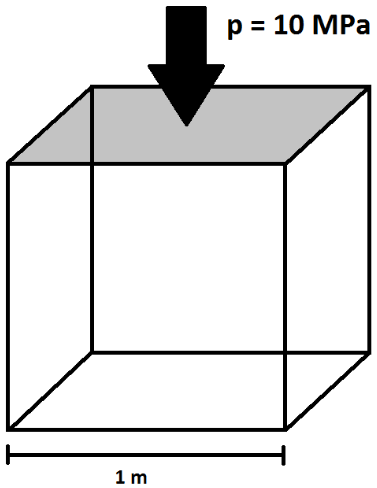
- The purpose with this tutorial is to get familiar with the pre- and post-processing tools in LS-PrePost 4.3 and also the basics of the LS-DYNA solver.

1.2 Prerequisites

- Basic knowledge in the finite element method

1.3 Problem Description

Consider the deformation of a cube on the ground with an applied pressure on the top surface. The task is to compute the vertical displacement of the cube due to this pressure.



Material properties		
Density, ρ	7850	kg/m3
Young's modulus, E	210	GPa
Poisson's Ratio, ν	0.3	

1.4 Data files

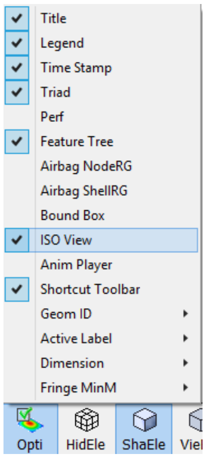
The solution can be found in **cube_results.k**

2 Preparation

2.1 Open LS-PrePost

Start LS-PrePost. In Microsoft Windows, you can find it in the start menu in the folder LSTC. Note: the exact location may differ depending on your installation.
The most common way to work with/open LS-PrePost is to have a short-cut on the desktop directly. This gives you control over which version of LS-PrePost you would like to use and you can easily update LS-PrePost separately.

2.2 View settings

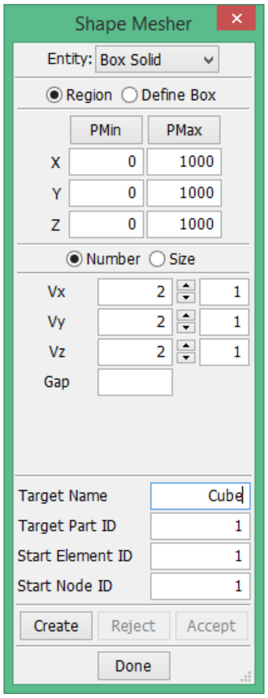


In LS-PrePost, go to **View > Toolbar** and activate **Text and Icon (Right)** and **Text and Icon (Bottom).** This is done to easier navigate through the different toolbars.
Check so you can see your Floating Toolbar in the LS-PrePost window.

If not, activate it by clicking on **Opti > ISO View** in the bottom toolbar.

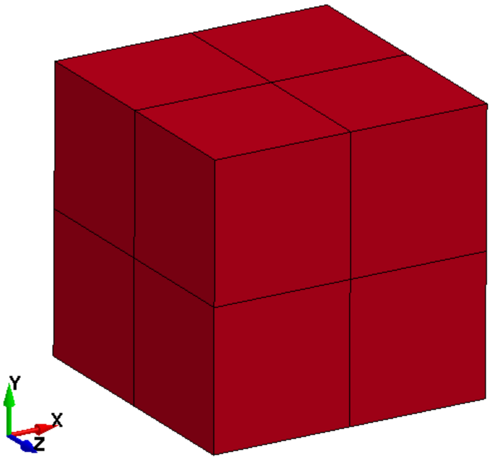
3 Create model

3.1 Geometry and mesh

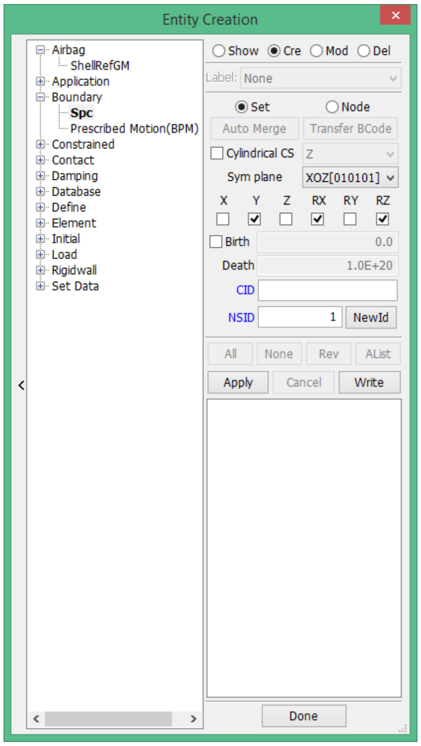


- Click **Mesh > ShapeM** (always use the menu on the right side, if nothing else is mentioned).
- Enter the values as in the picture to create a 1000x1000x1000 mm solid cube with two elements in the x-, y- and z-directions.
- Set **Target Name** to **Cube**.
- Click **Create, Accept** and then **Done.**

If you can't see your mesh, activate **Mesh** in the bottom toolbar.

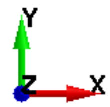


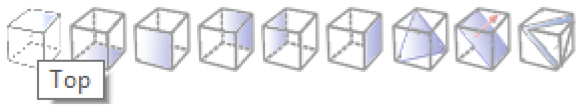
3.2 Boundary conditions



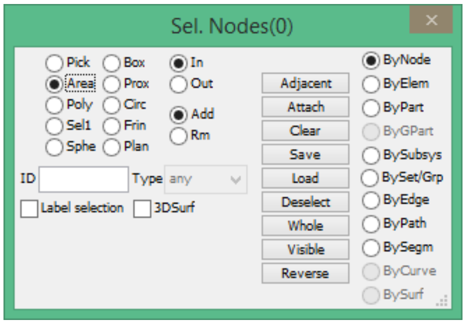
Apply boundary conditions to fix one side of the cube:

- Click **Model > CreEnt**.
- In the Entity Creation box, double-click on **Boundary** and click **Spc** in the dropdown that appears**.**
- Select **Cre**.
- **Set** shall be activated.
- Select **XOZ** as **Sym Plane**. **Y**, **RX** and **RZ** will then be activated. The boundary conditions will then have a translational constraint in global y-direction and rotational constraints about x- and z-axis.
- From the ISO-views on the top of the screen, click at the one called **Top**.

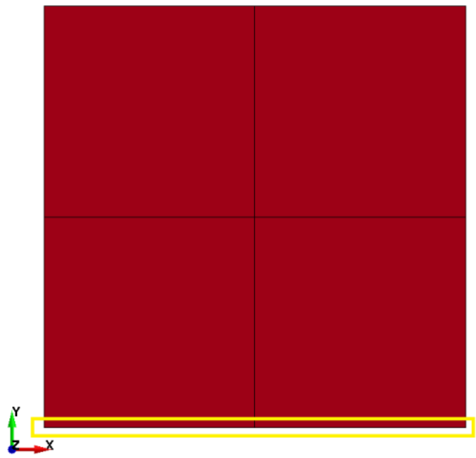




Check that your coordinate system looks like the one to the right. To show the coordinate system, click **Opti** and activate **Triad**.



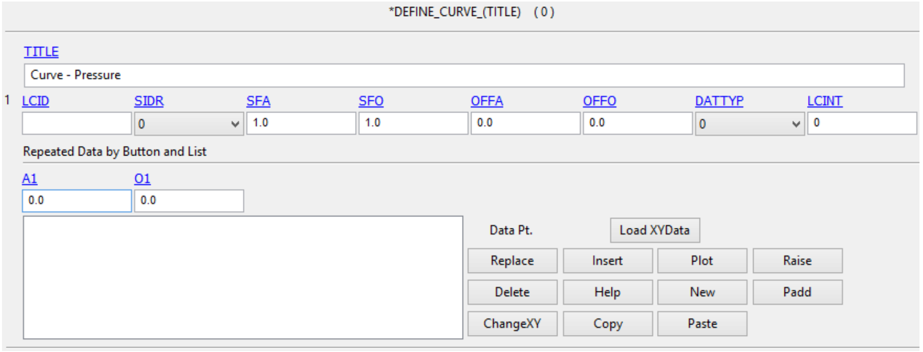
This box shows alternatives to select the nodes. Select **Area**.



- Select the nodes in the yellow square by making a box with the mouse.
- **Note** that the title of the selection box now indicates that 9  nodes have been picked.
- A node set will be created from the nodes that were chosen, **NSID = 1** in **Entity Creation** indicates that it will get an Id = 1.
- Click **Apply,** then **Done** in the Entity Creation box. The nodes are now constrained.

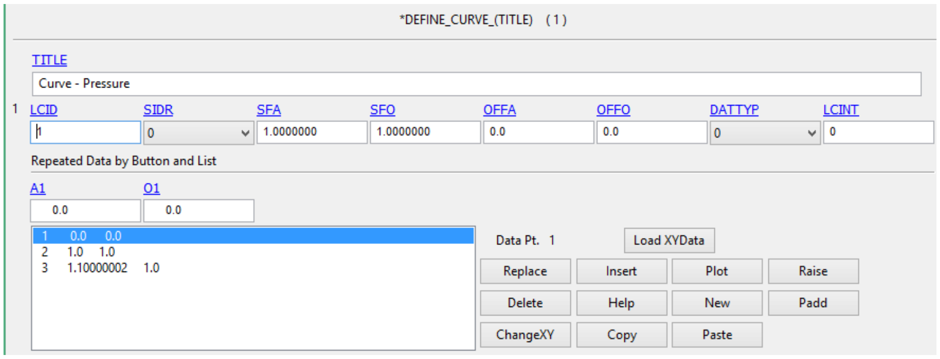
3.3 Apply the load

For loads, a curve must be defined that states the variation of the load over time. Click **Model > Keywrd**. In the dialog window that opens, there one can change between **Model** and **All. Model** shows the keywords that already have been created. **All** shows all possible keywords that are available in LS-DYNA.
Select **All** at the top of Keyword Manager window. Double-click **DEFINE > CURVE.** Name the curve **Curve – Pressure** for example. Note that all titles are optional, but it is good practice to make use of them to make the model clear and structured.
The points for the curve will be written in **A1** and **O1:**

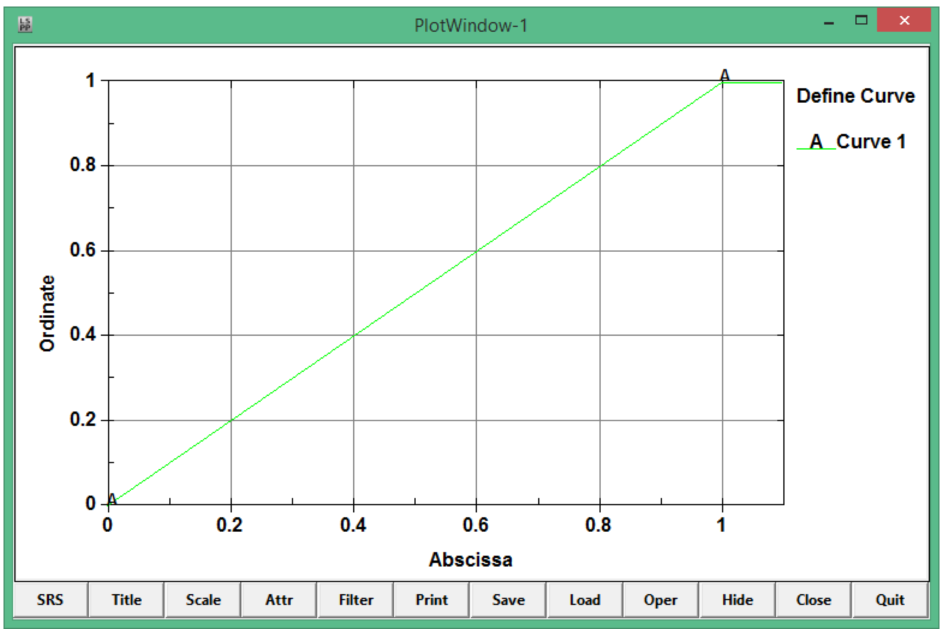


- Write **0** and **0,** Click **Insert.**
- Then **1** and **1, Insert.**
- Finally, **1.1** and **1, Insert.**
- Click **Accept**

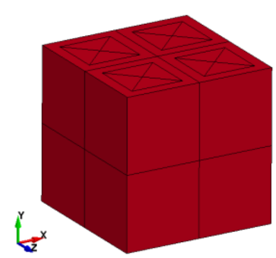
It is important that the curve extends beyond the end time of the simulation. The simulation will have the termination time 1 s (will be set later). Therefore, the last point of 1.1 was added.



To view the curve, Click **Plot.** Close the **PlotWindow (X or Quit)** and the ***DEFINE_CURVE (Done)** window



Now click **Model > CreEnt**:

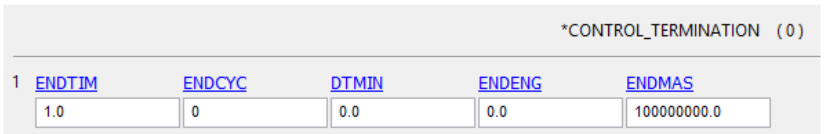


- In the Entity Creation box, double-click on **Load** and click on **Segment** in the dropdown that appears.
- Click **Cre**
- Change **Type:** to **LOAD_SEGMENT_SET**.
- Give the load the title **Pressure**.
- Click on **LCID** and select **1 Curve – Pressure**, press **Done**.
- To obtain a pressure of 10 MPa, the scale factor **SF** will be used. Set **SF** to **10** (the pressure unit is MPa for the S2 unit system).
- From the selection box, **Pick** can be activated.
- Click on the **four** segments on the top of the cube, as in the figure. If necessary, deactivate entities with right mouse button.
- Click **Apply**, then **Done**

3.4 Termination

The end time for the simulation needs to be set. This keyword is almost always mandatory for any simulation using LS-DYNA:

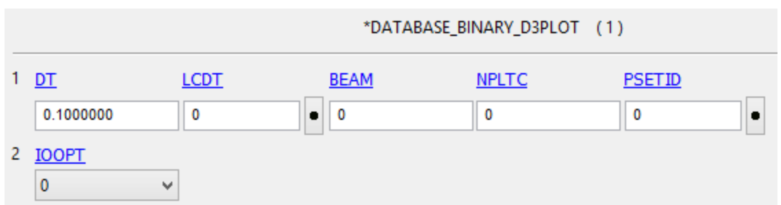
- Click **Model > Keywrd**.
- Double-click **CONTROL > TERMINATION**
- Set **ENDTIM** to **1**. The simulation will then last for 1 time unit, which is second in this case.
- **Accept**, then **Done**.



3.5 Output

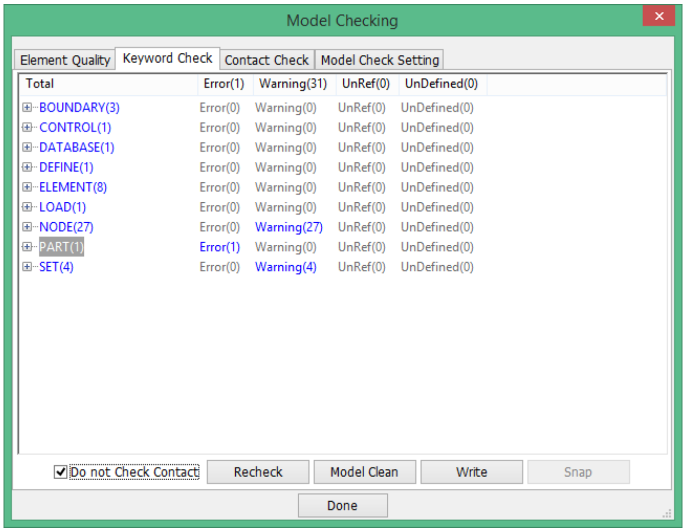
The user must request all the data needed to post-process an analysis using LS-DYNA, before starting the simulation. We will create something called d3plot, which gives complete output states of the simulation:

- Click **Model > Keywrd**.
- Double-click **DATABASE > BINARY_D3PLOT**
- Set **DT** to **0.1**. This implies that results will be printed every 0.1 time unit.
- **Accept**, then **Done**

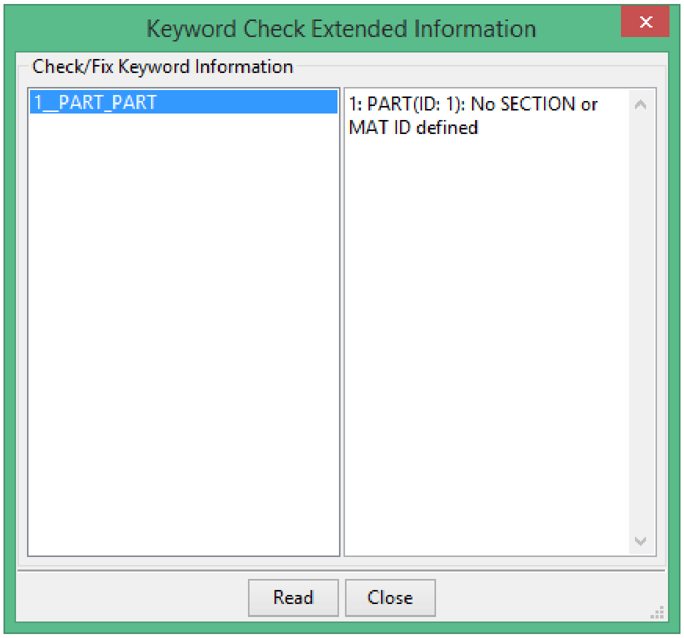


3.6 Model check

A good way to check your model is to use the built-in function **Model Check** in LS-PrePost: From the top menu, click **Application > Model Checking > General Checking**. Then switch to **Keyword Check**.



By double-click on **Warning** and **Error**, more information is shown in a pop-up window. From these warnings and/or errors, one can conclude that no element or material properties are defined for the cube. To close the window click **Done.**

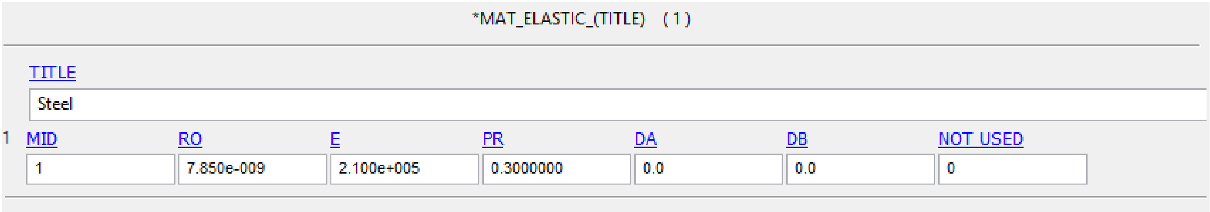


3.7 Material properties

To create a material card to define the material properties:

- Click **Model > Keywrd**
- Double-click **MAT > 001-ELASTIC**. This is an isotropic elastic material.
- Name the material to **Steel.**
- Set the material properties **RO**, **E** and **PR** as in the figure below (also stated in section 1.3).

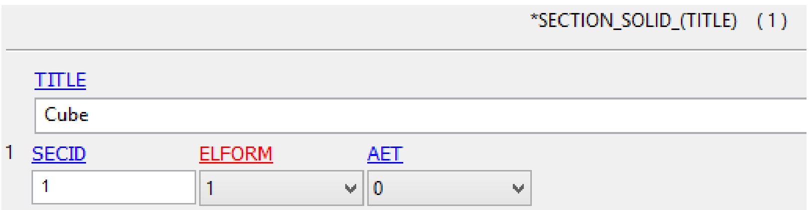
- Click **Accept**, then **Done**.



3.8 Element properties

The element type to be used:

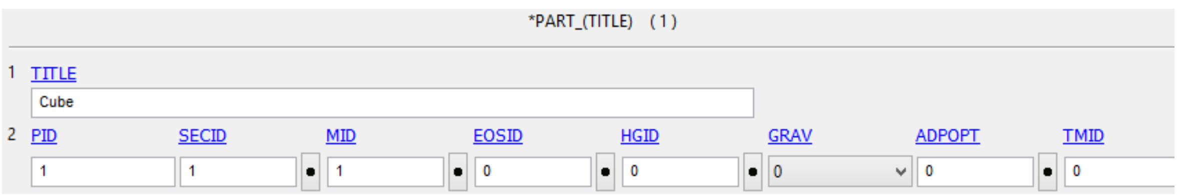
- From **Keyword Manager**, double-click **SECTION > SOLID**.
- Name the section to **Cube**
- Use **ELFORM = 1**, which is the default element formulation.
- Click **Accept**, then **Done**.



Now apply the material and element properties to the part. Since the part already is created, one can activate **Model**, instead of **All**, in the **Keyword Manager**. This makes it easier to navigate through the list of keywords.

- Double-click **PART > PART**
- Click on the black dot next to **SECID**, defined entities will then be shown
  Select your newly created section (**1 Cube**) and **Accept** and then **Done**
- Do the same thing for **MID**. Click **Accept**, then **Done**.

The result should be as shown below.

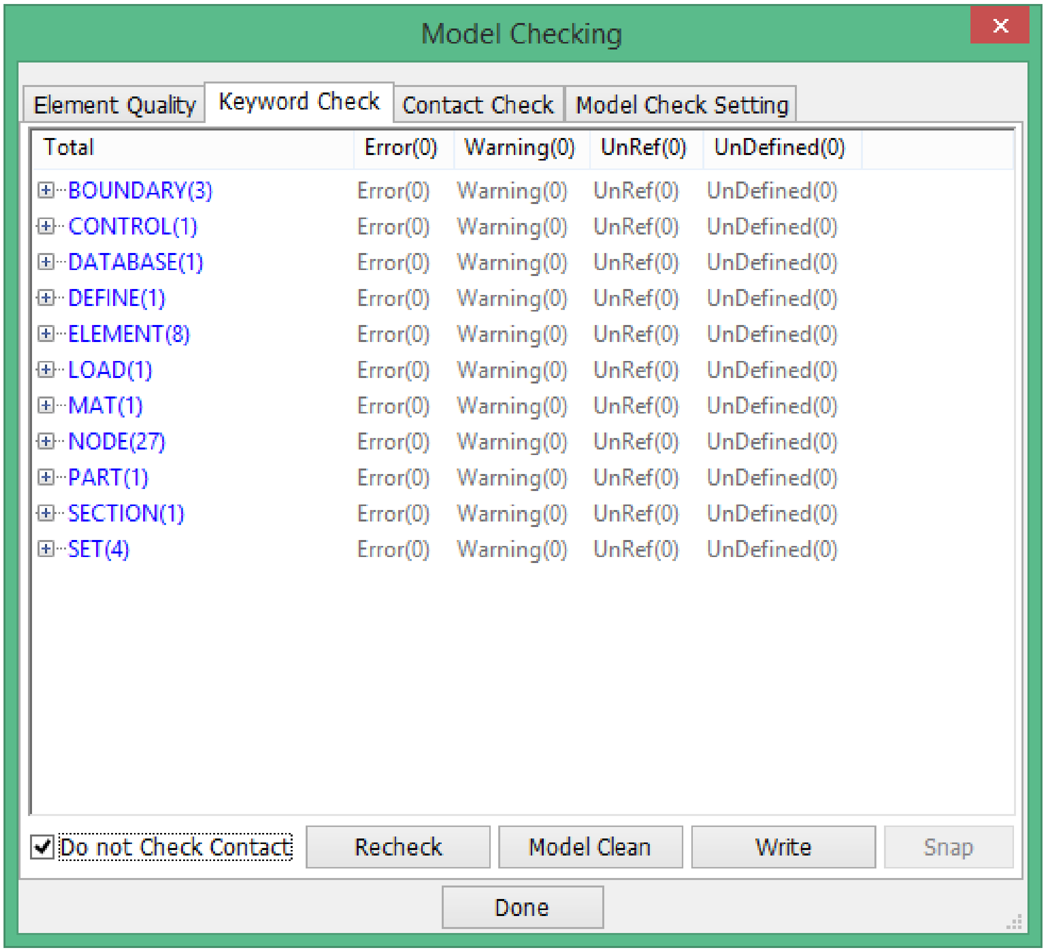


3.9 Check the model before running

Now check for errors using the **Model Check**:

- From the top menu, click **Application > Model Checking > General Checking**.
- Switch to **Keyword Check**. The warnings and error should now be gone.
- Click **Done**.

Note that even if no errors or warnings occurs, the model can still be incomplete or wrong. There is no way for any pre-processor to know your intended use of the model. Hence, the loadings and boundary conditions can only be checked if they make any sense, not if they are correct with respect to your load case.

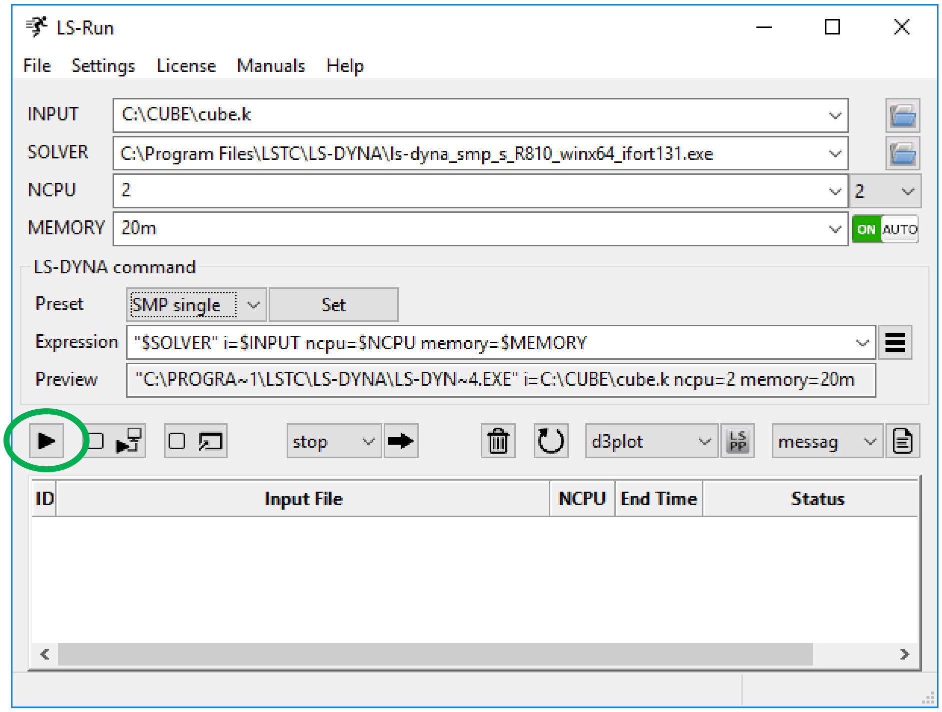

Model Checking
Element Quality
Keyword Check
Contact Check
Model Check Setting
Total
Error(0)
Warning(0)
UnRef(0)
UnDefined(0)
BOUNDARY(3)
CONTROL(1)
DATABASE(1)
DEFINE(1)
ELEMENT(8)
LOAD(1)
MAT(1)
NODE(27)
PART(1)
SECTION(1)
SET(4)
Do not Check Contact
Recheck
Model Clean
Write
Snap
Done

3.10 Save and run the simulation

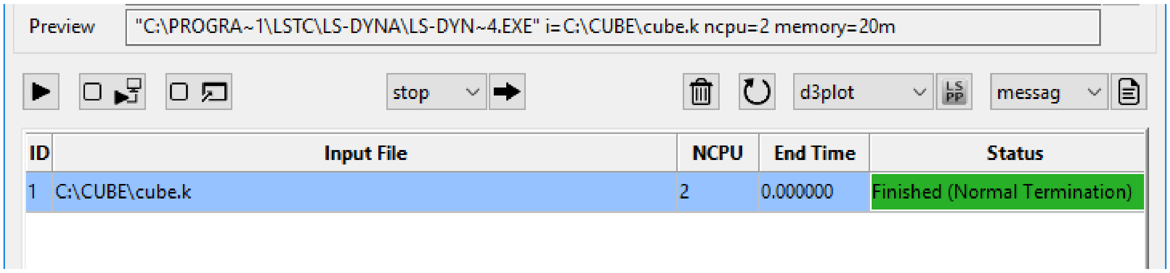
It is preferable to run each simulation in a separate folder, thus create one before saving if you have not done so, e.g. CUBE. First save the finished keyword model from LS-PrePost in the new “CUBE” folder on the computer using **File > Save As > Save Keyword As…** Use the file name **cube.k,** note the **.k** suffix.

To run the model first open **LS-Run**  from the start menu in Windows.

Select the **cube.k** file on the INPUT file using the file menu button to the right of the INPUT field or by drag-and-dropping the file **cube.k** onto the INPUT field. Select solver “SMP Single” and set NCPU to 2. This should yield the result below. Note: To learn more about **LS-Run**, see the Help menu.

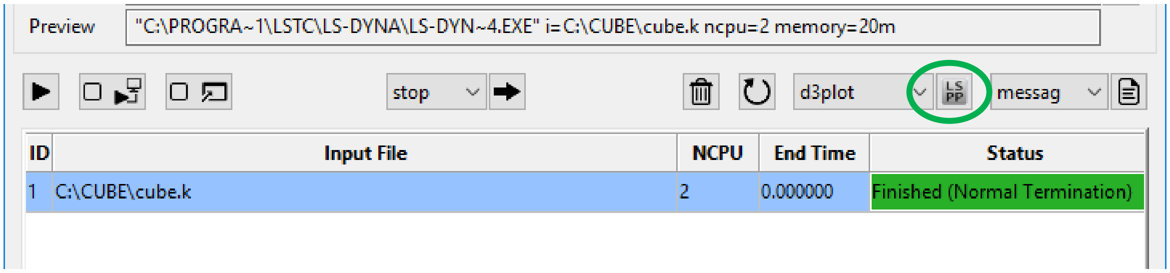


To start the simulation, press the play button circled in green in the image above. In just a few second the simulation should be done yielding “Finished (Normal Termination)”.

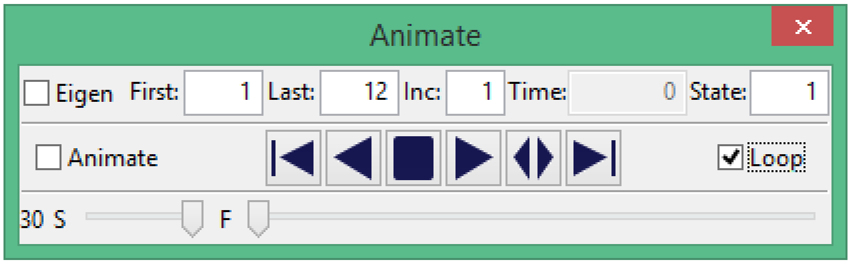


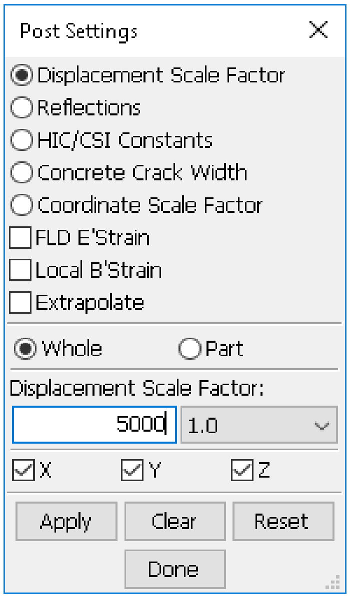
3.11 Post processing

To visualize the results, you need to open the **d3plot** result file. This is done by selecting (left click) the finished simulation in **LS-Run** and then left-clicking the LS-PrePost button circled in green below.



Once LS-PrePost is opened, the animate toolbar is the tool that enables you to step through the different states of the simulation. Hold the mouse over the different buttons, a text box will pop up and show information about the different possibilities using the animator toolbar. Play around with the buttons to see what happens. Note that the deformations are very small, therefore you will probably not notice that anything happens with the cube.

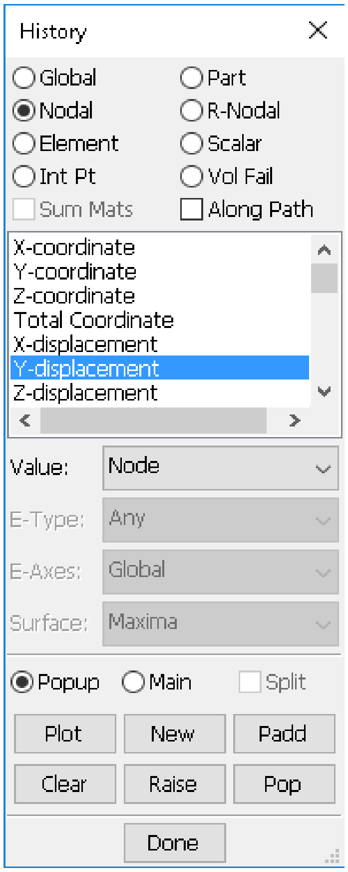




To easier see what happens, you can scale up the deformations:

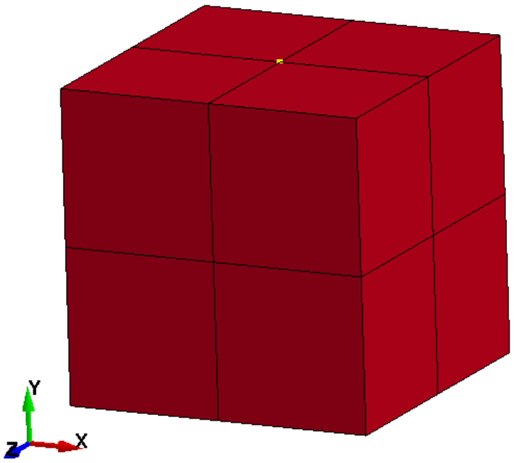
- Click **Settings > Post Settings** in the top menu.
- Select **Displacement Scale Factor,** write **5000** as the factor. X, Y and Z shall be activated, which implies that the displacements will be scaled in all directions.
- Click **Apply**, then **Done**.

Play around with the animate toolbar again and see how the cube deforms.



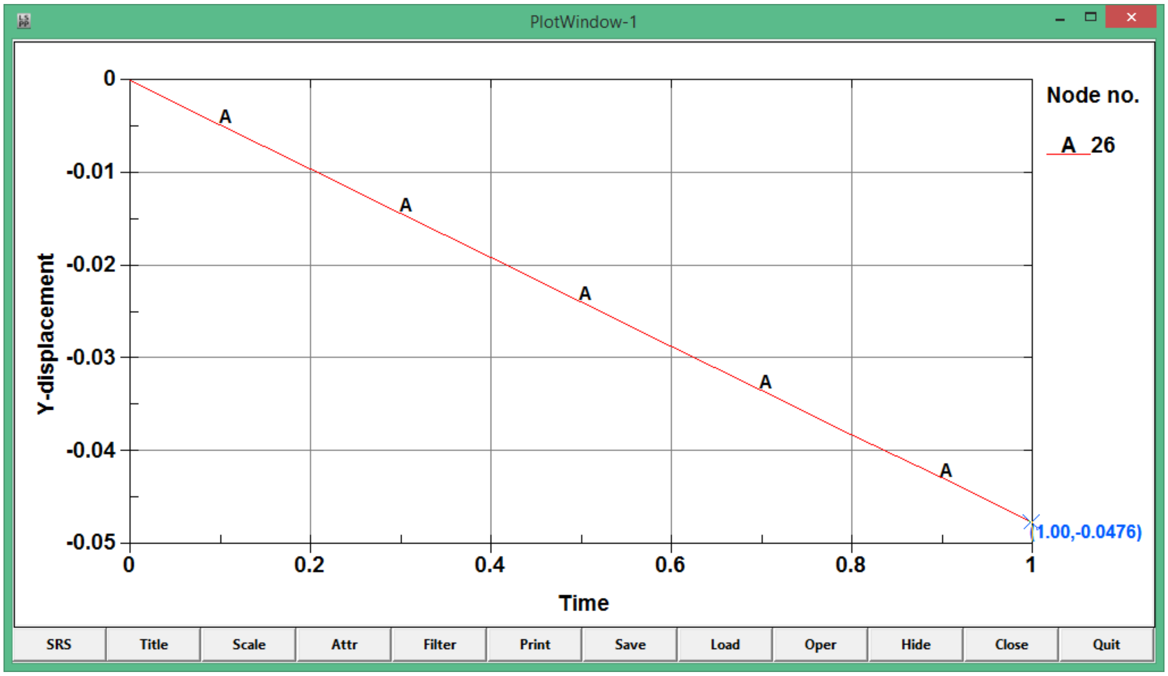
Then plot the deformation history as a curve:

- Click **Post > History**.
- In the **History Box**, select **Nodal > Y-displacement.** Select a node on the top of the box.
- Click **Plot**.



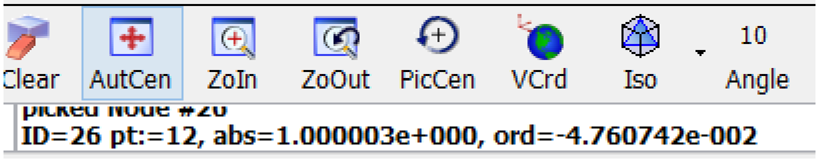
In the Plot Window, Y-displacement vs Time are stated. Zoom by pressing Ctrl and make a box with the mouse. A right-click will reset the window to original. Zoom in on the curve around Time=1, click on the final state.

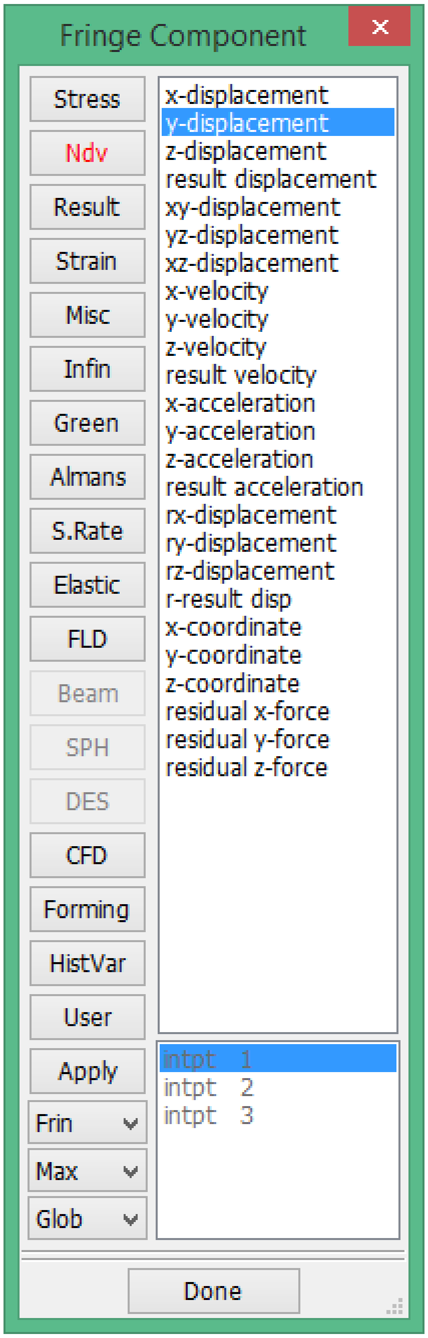
Note that the Y-displacement is **-0.0476 mm**.



When you click on the curve, information about the picked point will be stated in the **Message box**, which is located at the bottom of the LS-PrePost window. If you double-click in this box, a bigger box will pop up.

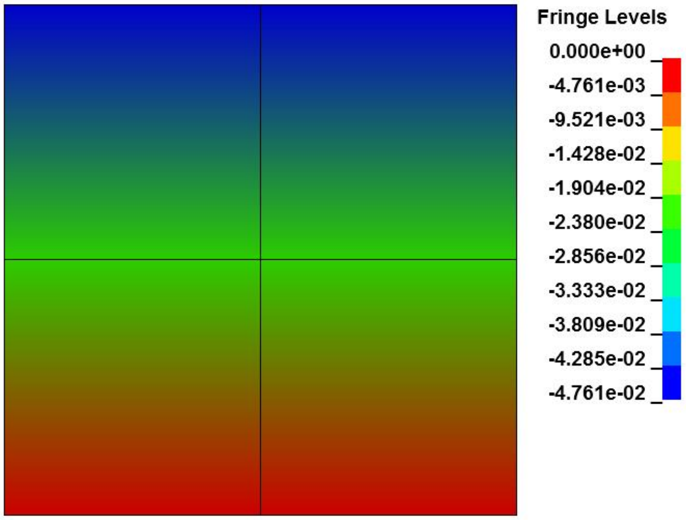
Close the **Message box** and **Plot Window.**





Now, click **Post > FriComp > Ndv > y-displacement.** Use the **Animate toolbar** and go to the last state.

The values in the **Fringe Level** shows that the maximum y-displacement is **-4.761e-02 mm**.



4 Analytical solution

The analytical solution of the vertical displacement due to a 10 MPa pressure load is derived from Hooke's law.

$$Displacement = \frac{pL}{E} = \frac{10e^{6} \cdot 1}{210e^{9}} = 4.76e^{-5}\ m = \mathbf{0.0476\ mm}$$

The simulation result should be nearly identical to the analytical solution.

5 Summary

In this exercise, we have defined a simulation model for LS-DYNA. The small example illustrates how the different entities are connected as well as how you set up boundary conditions and loads in LS-DYNA.

6 Optional exercises

There are several possibilities to try out different features in LS-DYNA with this simple model. What happens if you change:

- material?
- boundary conditions?
- element formulations?
- load level?