



AIM

The aim of the Finite Element Method - FEM, Lab session is to provide to the attendants the ability of understanding and implementing simulations of laser mater interaction problems.

1. LS-DYNA

LS-DYNA from Livermore Software Technology Corporation (LSTC), is a highly advanced general purpose nonlinear finite element program that is capable of simulating complex real-world problems. The distributed and shared memory solver provides very short turnaround times on desktop computers and clusters operated using Linux, Windows and UNIX.

LS-DYNA is suitable to investigate phenomena involving large deformations, sophisticated material models and complex contact conditions for structural dynamic problems. LS-DYNA allows switching between explicit and different implicit time stepping schemes. Disparate disciplines, such as coupled thermal analyses, Computational Fluid Dynamics (CFD), fluid-structure interaction, Smooth Particle Hydrodynamics (SPH), Element Free Galerkin (EFG), Corpuscular Method (CPM), Discrete Element Method (DEM) and the Boundary Element Method (BEM) can be combined with structural dynamics. For pre- and post-processing, LS-DYNA comes with the LS-PrePost tool. LS-PrePost can be utilized to generate inputs and visualize numerical results.

1.1 Keyword format input files

An LS-DYNA input file is a text-file in so called Keyword format usually with a *.k, *.key or *.dyn suffix, e.g. laser.k. A finite element model in LS-DYNA is built up by different keywords, which is defined for all ingoing definitions and parameters in a model (e.g. *PART, *NODE). A short overview of the basic structure of such an input file for a basic 1 element finite element model is provided.

Consider a cube consisting of one element with eight node points as shown in Figure 1.1.

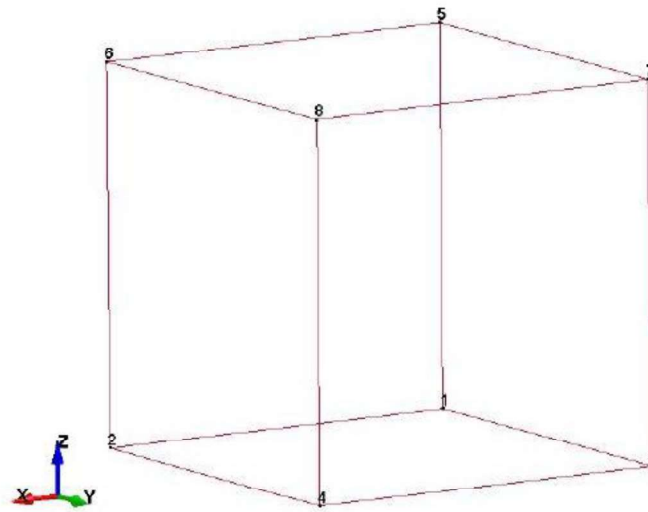


Figure 1.1 Cube consisting of one element with eight node points

The `*PART` keyword is used to begin the definition of the finite element model. The keyword `*PART` contains data that points to other attributes of this part, e.g. material properties. Keywords for these other attributes, in turn, point elsewhere to additional attribute definitions. The organization of the keyword input for the cube looks like this:

```

*PART
$# pid 1 secid 1 mid 1
*SECTION SOLID
$# secid 1 elform 1
*MAT_ELASTIC
$# mid 1 ro 7850.0 e 2.10E11 pr 0.3
*ELEMENT SOLID
$# eid 1 pid 1 n1 1 n2 2 n3 4 n4 3 n5 5 n6 6 n7 8 n8 7
*NODE
$# nid 1 x 0.0 y 0.0 z 0.0
    2 x 1.0 y 0.0 z 0.0
    3 x 0.0 y 1.0 z 0.0
    4 x 1.0 y 1.0 z 0.0
    5 x 0.0 y 0.0 z 1.0
    6 x 1.0 y 0.0 z 1.0
    7 x 0.0 y 1.0 z 1.0
    8 x 1.0 y 1.0 z 1.0
  
```

A brief description follows:

***PART:** We have one part with identification `pid=1`. This part has attributes identified by section identification `secid=1` and material identification `mid=1`.

***SECTION_SOLID:** Parts definitions that reference secid=1 are defined as constant stress 8 node brick elements (elform=1).

***MAT_ELASTIC:** Parts definitions that reference mid=1 are defined as an elastic material with density, Young's modulus and Poisson's ratio.

***ELEMENT SOLID:** The element with identification eid=1 are defined by nid=1 to nid=8 and belongs to pid=1.

***NODE:** The node identified by nid has coordinates x,y,z.

Boundary conditions and time dependent loads are also set by keywords and are usually applied on nodes, elements, segments or parts. Set definitions are often used to define groups of these entities. Since all loads are time dependent, curves need to be defined that states time vs load unit (force, pressure etc.).

One example of defining boundary conditions and loads on the cube looks like this:

```

*BOUNDARY_SPC_SET
$#  nsid      cid      dofz      dofry      dofrz
   1         0         0         1         0         1         0         1
*SET_NODE_LIST
$#   sid
   1
$#  nid1  nid2  nid3  nid4
   1     2     5     6
*LOAD_SEGMENT
$#  lcid      sf      at      n1      n2      n3      n4      n5
   1         1.0    0.0    4       8       7       3       0
*DEFINE_CURVE
$#  lcid
   1
$#          a1          o1
          0.0          0.0
          1.0          10.0
  
```

About the keywords above:

***BOUNDARY_SPC_SET:** The node set with identification nsid=1 are constrained in y-translation and x- and z-rotations.

***SET_NODE_LIST:** This keyword defines that node 1, 2, 5 and 6 belongs to node set sid=1.

***LOAD_SEGMENT:** A pressure load is applied on a segment that are defined by node 4, 8, 7 and 3.

***DEFINE_CURVE:** The curve consists of two points that defines the time vs pressure. This curve with identification lcid=1 is used for the load.



1.2 Consistent units

On the table below are presented consistent units for using LS-DYNA. The inappropriate selection of consistent units is one of the most common error done by the user.

Consistent units

Definition of a consistent system of units (required for LS-DYNA):

- 1 force unit = 1 mass unit * 1 acceleration unit
- 1 acceleration unit = 1 length unit / (1 time unit)²
- 1 density unit = 1 mass unit / (1 length unit)³

The following table provides examples of consistent systems of units. As points of reference, the mass density and Young's Modulus of steel are provided in each system of units. "GRAVITY" is gravitational acceleration.

MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	DENSITY	YOUNG'S	35MPH 56.33KMPH	GRAVITY
kg	m	s	N	Pa	J	7.83e+03	2.07e+11	15.65	9.806
kg	cm	s	1.0e-02 N			7.83e-03	2.07e+09	1.56e+03	9.806e+02
kg	cm	ms	1.0e+04 N			7.83e-03	2.07e+03	1.56	9.806e-04
kg	cm	us	1.0e+10 N			7.83e-03	2.07e-03	1.56e-03	9.806e-10
kg	mm	ms	kN	GPa	kN-mm	7.83e-06	2.07e+02	15.65	9.806e-03
g	cm	s	dyne	dyne/cm ²	erg	7.83e+00	2.07e+12	1.56e+03	9.806e+02
g	cm	us	1.0e+07 N	mbar	1.0e+07 Ncm	7.83e+00	2.07e+00	1.56e-03	9.806e-10
g	mm	s	1.0e-06 N	Pa		7.83e-03	2.07e+11	1.56e+04	9.806e+03
g	mm	ms	N	MPa	N-mm	7.83e-03	2.07e+05	15.65	9.806e-03
ton	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	1.56e+04	9.806e+03
lbf-s ² /in	in	s	lbf	psi	lbf-in	7.33e-04	3.00e+07	6.16e+02	386
slug	ft	s	lbf	psf	lbf-ft	1.52e+01	4.32e+09	51.33	32.17
kgf-s ² /mmmm		s	kgf	kgf/mm ²	kgf-mm	7.98e-10	2.11e+04	1.56e+04	9.806e+03
kg	mm	s	mN	1.0e+03 Pa		7.83e-06	2.07e+08		9.806e+03
g	cm	ms	1.0e+1 N	1.0e+05 Pa		7.83e+00	2.07e+06		9.806e-04

Table 1.1 Consistent units using LS-DYNA

2. LS-PrePost

LS-PrePost is an advanced pre- and post-processor designed specifically for LS-DYNA. It is developed for Windows, Linux and Apple and it is free to download from the web link <http://ftp.lstc.com/anonymous/outgoing/lsprepost/4.5/>.

LS-Prepost main functions contain:

- Full support of LS-DYNA keyword files
- Full support of LS-DYNA result files
- Robust handling of geometry data (new CAD engine)
- Pre-processing (meshing, model clean-up, entity creation)



- Post-processing (animation, fringe plotting, curve plotting)

2.1 Input and output files

Input

FEM: LS-DYNA Keyword, Nastran, I-DEAS Universal, PAM-CRASH, RADIOSS, ABAQUS

CAD: IGES, STEP

ASCII: glstat, matsum, etc.

Binary: d3plot, binout, etc.

Output

FEM: LS-DYNA Keyword, Nastran

Image: PNG, TIFF, BMP, GIF, JPG, PostScript

Movie: AVI, MPEG, Animated GIF, JPEG

XY Data: CRV, CSV, XML o CAD: IGES, STEP, STL

Other: Post.db, Project File

2.2 Mouse and Keyboard

Dynamic Model Operation

Rotate: Shift + Left-click

Translate: Shift + Middle-click

Zoom: Shift + Right-click/Scroll-wheel

Graphics Selection

Pick (single): Left Click o Area (rectangle): Left-click + Drag

Poly (polygon): Left-click at corners / Right-click to finish

List Selection

Multi-Select: Left-click + Drag / Ctrl + Left-click

2.3 Graphical user interface

The graphical interface of LS-PrePost is shown in Figure 2.1. On the right hand side, you can see the main toolbar. When clicking on one of these, a sub-toolbar just to the left will be shown. That is the location where you'll find most of the tools needed to create/modify/delete entities in your model.

In the bottom toolbar, are found the tools which are the most commonly used to determine how LS-PrePost should render mesh/surfaces, orient the model, etc. There are a couple of

the drop-down menus on the top left corner that you will use more or less frequently: File/View/Application/Settings. The Floating Toolbar is used to toggle between different views.

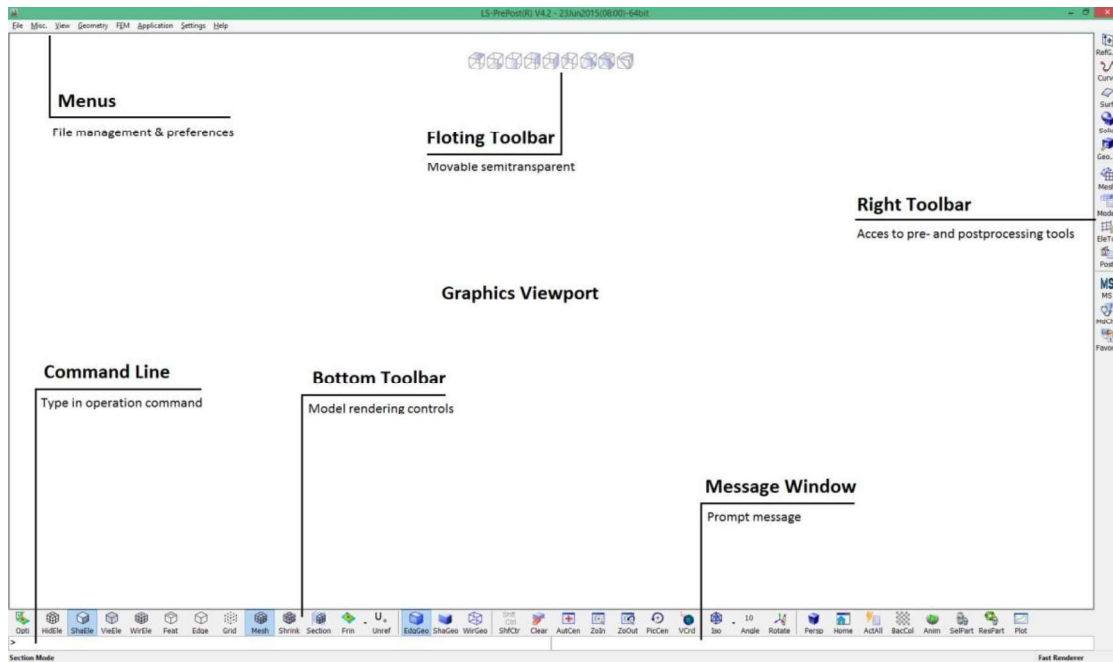
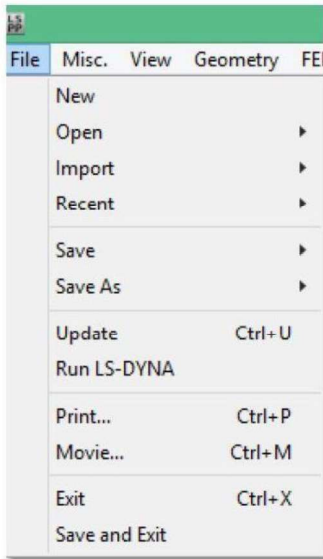


Figure 2.1 Graphical interface of the LS-PREPOST

2.4 Menus

In this section are presented the useful menus for the user, the File Menu, the Geometry Menu and the FEM menu.

File Menu



New – Launch a new session of LS-PrePost, all model/data will be closed

Open – Open file (new model created for each file opened)

Import – Import file (adds keyword data to current model)

Recent – Open recent files (stored in /user/.lspg_recent)

Save – Over-write current *Keyword* or *Project* file

Save As – Save any of the following file formats using advanced options: *Keyword*, *Active Keyword* (visible data), *Project*, *Post.db* (condensed d3plot data), *Geometry*, *Keyword* and *Project* (using the same file name).

Update – Load new d3plots for run in progress

Run LS-DYNA – Pop up LS-DYNA job submission dialog, currently only limited to the same local machine LS-PrePost is running

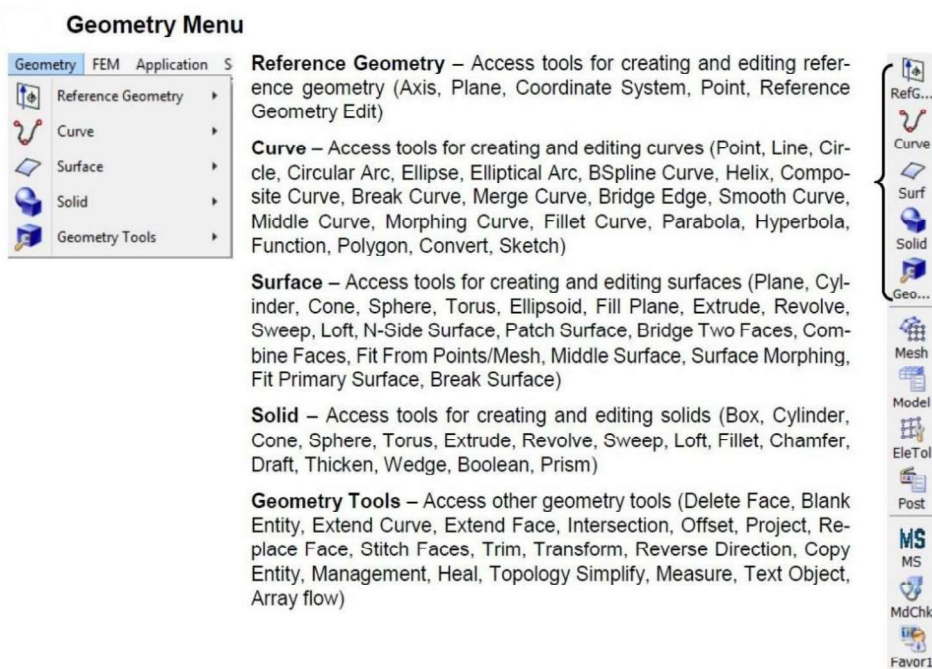
Print... – Launch printing interface (send to printer or image file)

Movie... – Launch movie generation interface

Exit – Exit LS-PrePost

Save and Exit – Save data to current file and exit LS-PrePost

Figure 2.2 File menu



Reference Geometry – Access tools for creating and editing reference geometry (Axis, Plane, Coordinate System, Point, Reference Geometry Edit)

Curve – Access tools for creating and editing curves (Point, Line, Circle, Circular Arc, Ellipse, Elliptical Arc, BSpline Curve, Helix, Composite Curve, Break Curve, Merge Curve, Bridge Edge, Smooth Curve, Middle Curve, Morphing Curve, Fillet Curve, Parabola, Hyperbola, Function, Polygon, Convert, Sketch)

Surface – Access tools for creating and editing surfaces (Plane, Cylinder, Cone, Sphere, Torus, Ellipsoid, Fill Plane, Extrude, Revolve, Sweep, Loft, N-Side Surface, Patch Surface, Bridge Two Faces, Combine Faces, Fit From Points/Mesh, Middle Surface, Surface Morphing, Fit Primary Surface, Break Surface)

Solid – Access tools for creating and editing solids (Box, Cylinder, Cone, Sphere, Torus, Extrude, Revolve, Sweep, Loft, Fillet, Chamfer, Draft, Thicken, Wedge, Boolean, Prism)

Geometry Tools – Access other geometry tools (Delete Face, Blank Entity, Extend Curve, Extend Face, Intersection, Offset, Project, Replace Face, Stitch Faces, Trim, Transform, Reverse Direction, Copy Entity, Management, Heal, Topology Simplify, Measure, Text Object, Array flow)

Figure 2.3 Geometry menu

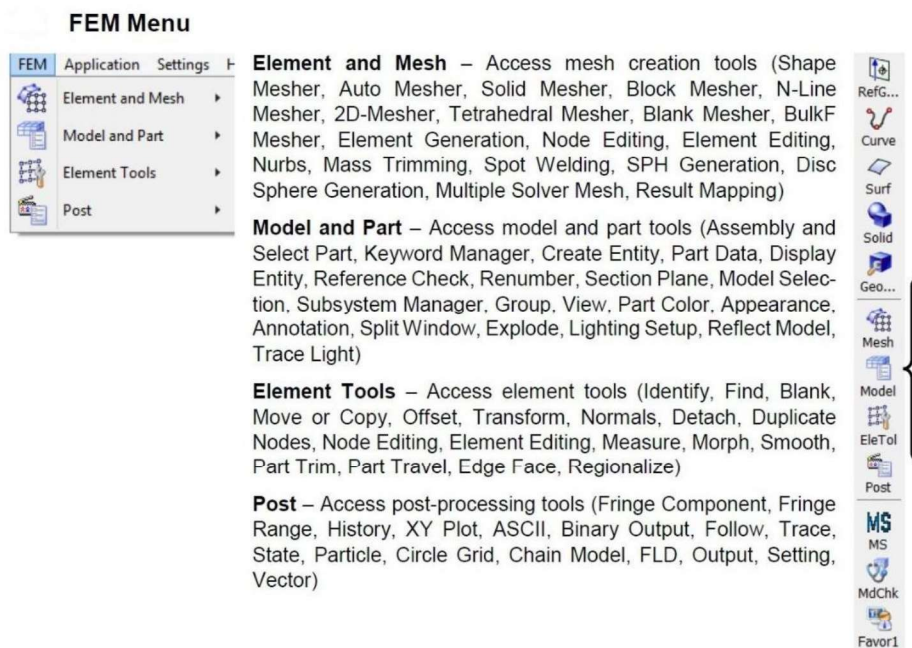


Figure 2.4 FEM Menu

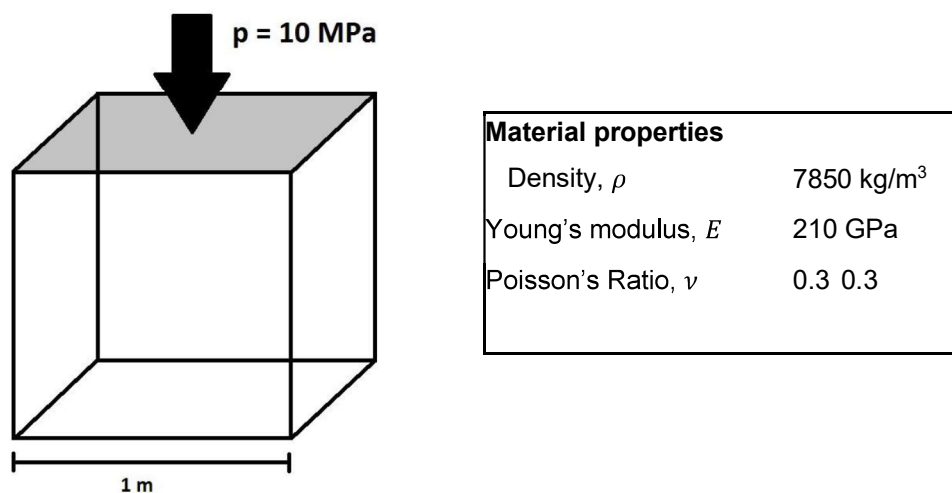
3. Getting started

3.1 Purpose

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

3.2 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.





Erasmus+

3.3 Data files and unit system

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

The S2 unit system is used in this tutorial

	S1	S2
length	meter	millimeter
time	second	second
mass	kilogram	tonne
force	Newton	Newton
Young's modulus of steel	210.0E+09	210.0E+03
density of steel	7.85E+03	7.85E-09
gravitation	9.81	9.81E+03

3.4 Preparation LS-PrePost

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.

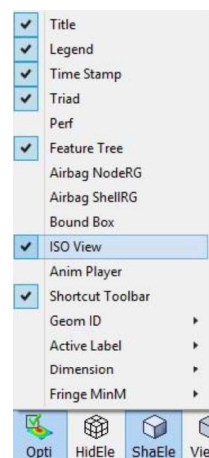
3.4.1 View settings

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-

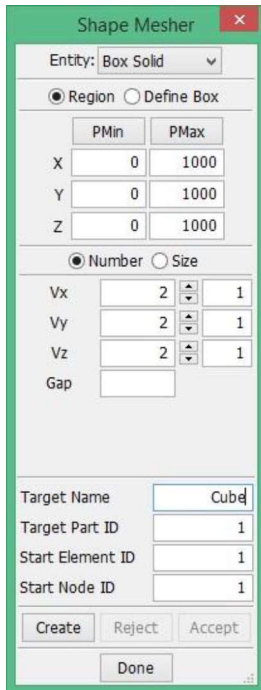
PrePost window.



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

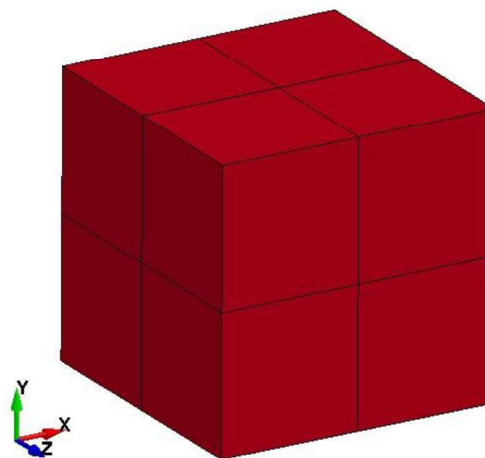
3.5 Create model

3.5.1 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**.

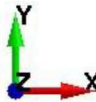
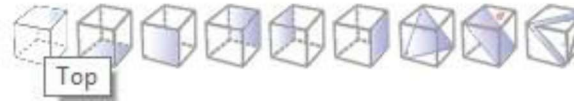
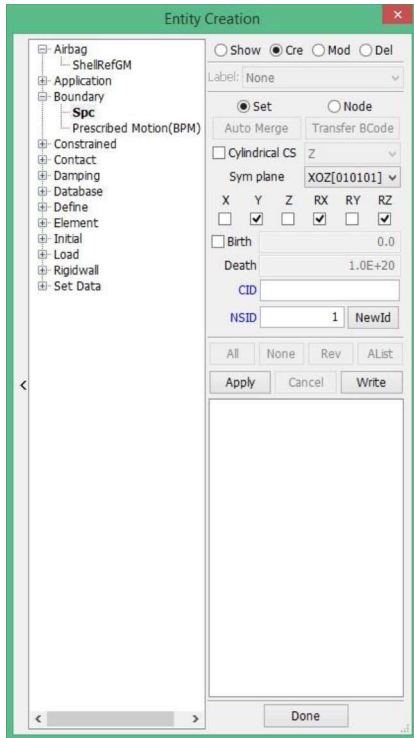
If you can't see your mesh, activate **Mesh** in the bottom toolbar.



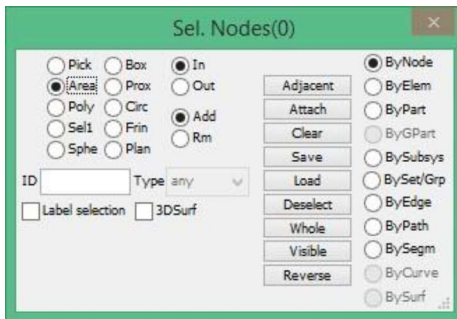
3.5.2 Boundary conditions

Apply boundary conditions to fix one side of the cube:

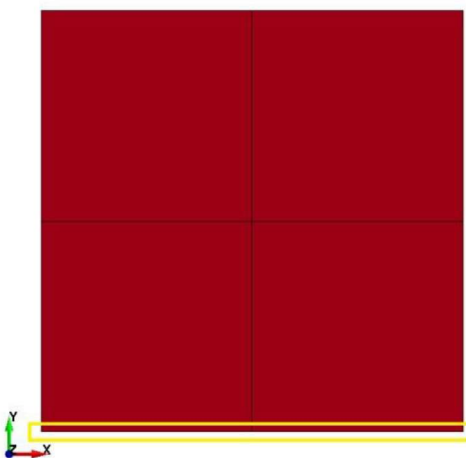
- Click **Model > CreEnt**
- In the Entity Creation box, double-click on **Boundary** and click **Spc** in the dropdown that appears.
- Select **Cre**
- **Set** shall be activated.
- Select **XOZ** as **Sym Plane**, Y, RX and RZ will then be activated. The boundary conditions will then have a translational 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 above. To show the coordinate system, click **Opti** and activate **Triad**.



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

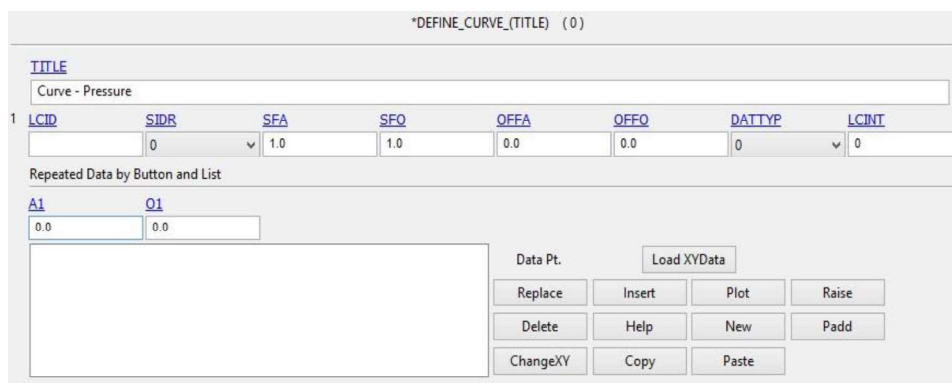


- Select the nodes in the yellow square by making a box with the mouse.
 - A node set will be created from the nodes that were chosen, **NSID = 1** in **Entity Creation** indicates that it will get an Id = 1.
 - Click **Apply**, then **Done** in the Entity Creation box.
- The nodes are now constrained.

3.5.3 Apply the load

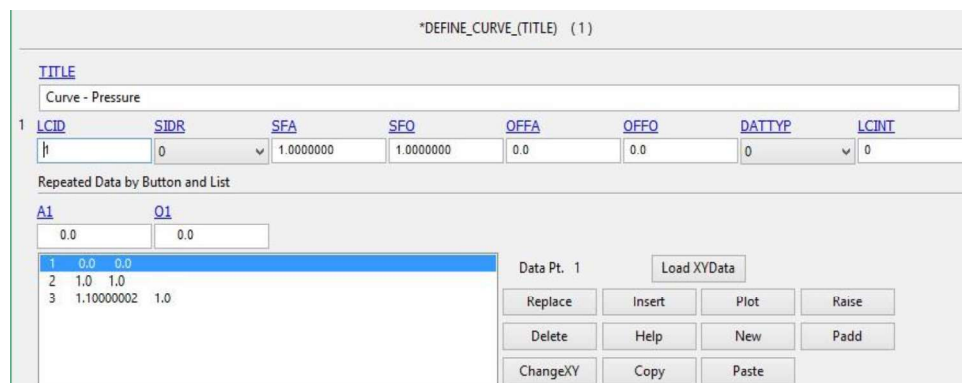
For loads, a curve must be defined that states the variation of the load over time. Click **Model > Keywrd**. In the dialog window that opens, there one can change between **Model** and **All**. **Model** shows the keywords that already have been created. **All** shows all possible keywords that are available in LS-DYNA.

Select **All** at the top of Keyword Manager window. Double-click **DEFINE > CURVE**. Name the curve **Curve – Pressure** for example. 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**:

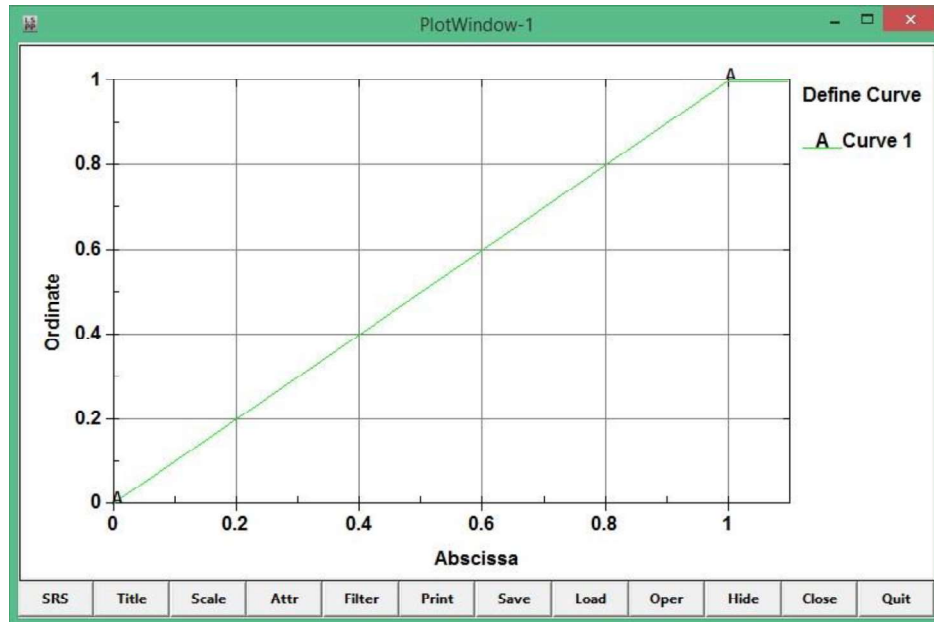


- Write **0** and **0**, Click **Insert**.
- Then **1** and **1**, **Insert**.
- Finally, **1.1** and **1**, **Insert**.
- Click **Accept**

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.

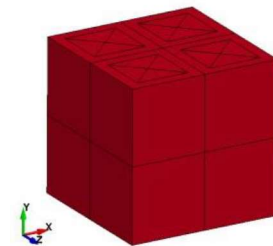


To view the curve, Click **Plot**. Close the **PlotWindow (X or Quit)** and the ***DEFINE_CURVE (Done)** window.



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 selected 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**



3.5.4. 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**.

*CONTROL_TERMINATION (0)					
1	ENDTIM	ENDCYC	DTMIN	ENDENG	ENDMAS
	1.0	0	0.0	0.0	100000000.0

3.5.5 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**

*DATABASE_BINARY_D3PLOT (1)					
1	DT	LCDT	BEAM	NPLTC	PSETID
	0.1000000	0	<input checked="" type="checkbox"/>	0	0
2	IOOPT				
	0				

3.5.6 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 materia
- 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**.

*MAT_ELASTIC_(TITLE) (1)

TITLE
Steel

1	MID	RO	E	PR	DA	DB	NOT USED
1		7.850e-009	2.100e+005	0.3000000	0.0	0.0	0

3.5.7 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**.

*SECTION_SOLID_(TITLE) (1)

TITLE
Cube

1	SECID	ELFORM	AET
1	1	1	0

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.

*PART_(TITLE) (1)

1 TITLE
Cube

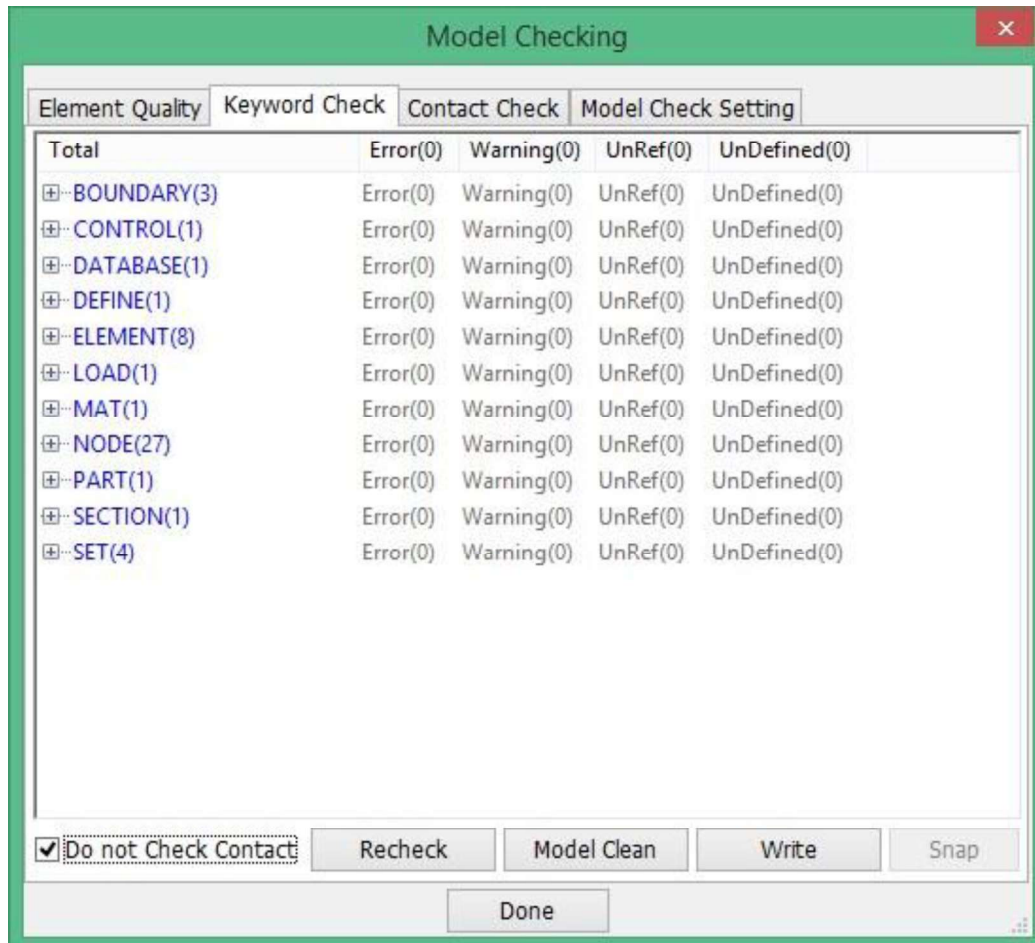
2	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
1		1	1	0	0	0	0	0

3.5.8 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 not exist.
- 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.



3.5.9 Save and run the simulation

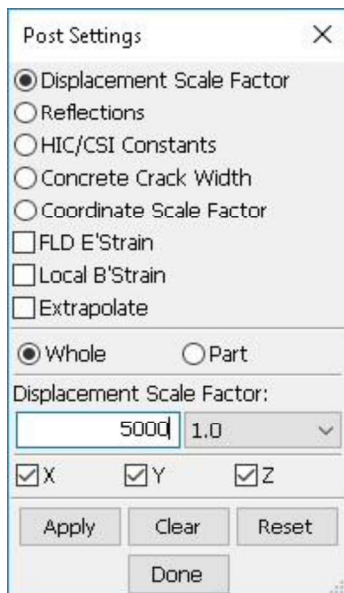
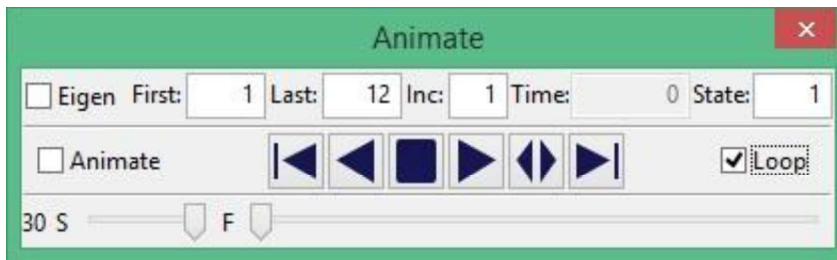
It is preferable to run each simulation in a separate folder, thus create one before saving if you have not done so, e.g. CUBE. First save the finished keyword model from LS-PrePost in the new “CUBE” folder that you have created on your computer using **File > Save As > Save Keyword As**. Use the file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

To how to run the simulation, you will be given orders from the instructor.

3.5.10 Post processing

To visualize the results, you need to open the **d3plot** result file. This is done by selecting **File>Open>LS DYNA Binary Plot**. Once Binart Plot is opened, the animate toolbar is the tool

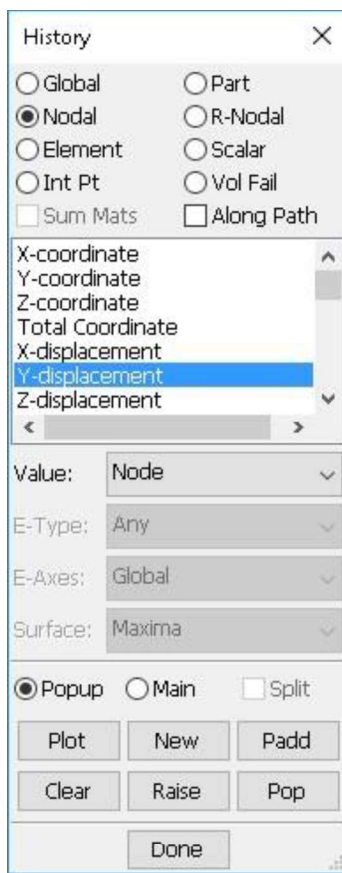
that enables you to step through the different states of the simulation. Hold the mouse over the different buttons, a text box will pop up and show information about the different possibilities using the animator toolbar. Play around with the buttons to see what happens. Note that the deformations are 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:

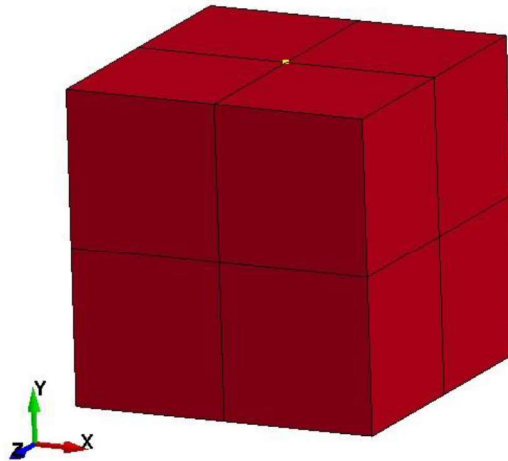
- Click **Settings > Post Settings** in the top menu.
- Select **Displacement Scale Factor**, write **5000** as the factor. X, Y and Z shall be activated, which implies that the displacements will be scaled in all directions.
- Click **Apply**, then **Done**.

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



Then plot the deformation history as a curve:

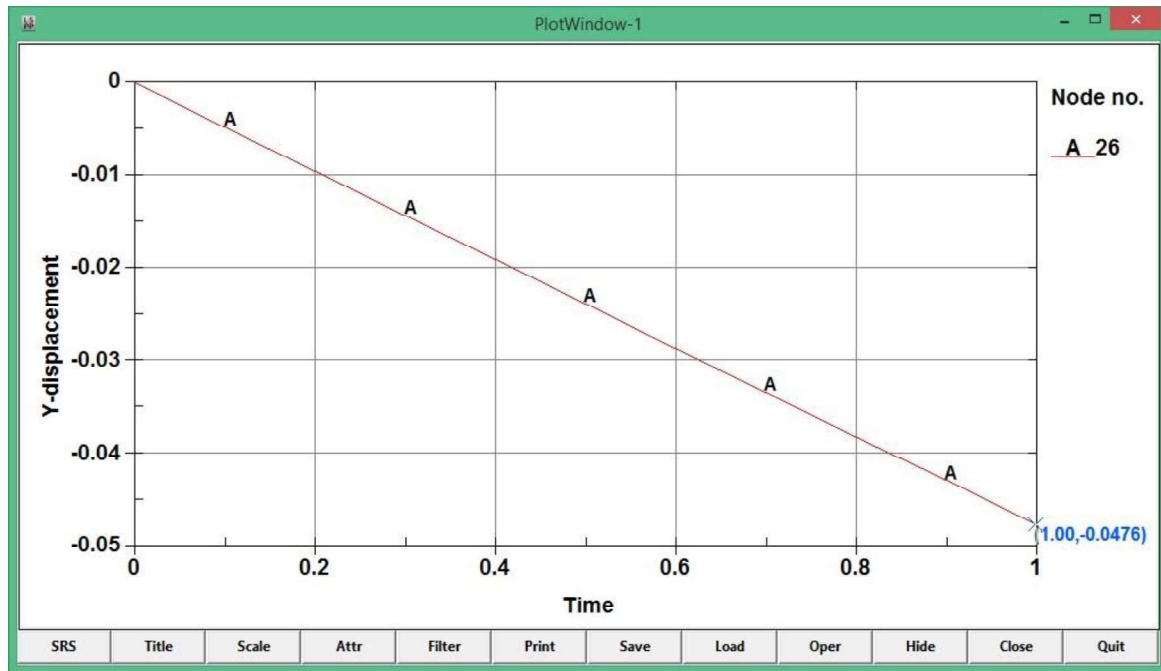
- Click **Post > History**.
- In the **History Box**, select **Nodal > Y-displacement**.



- 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**.

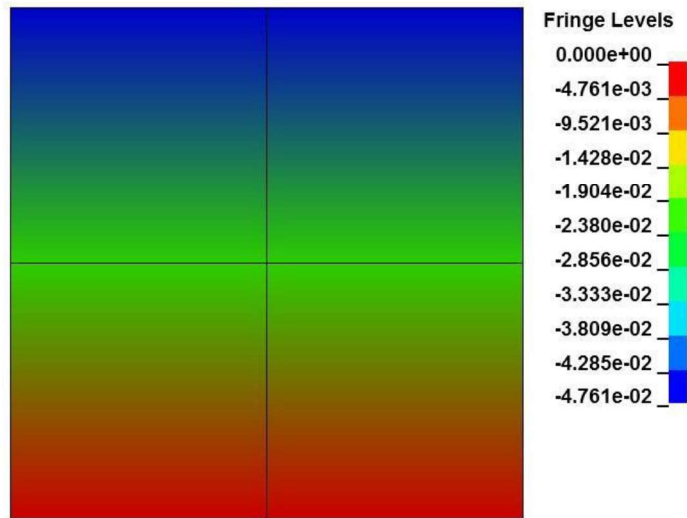
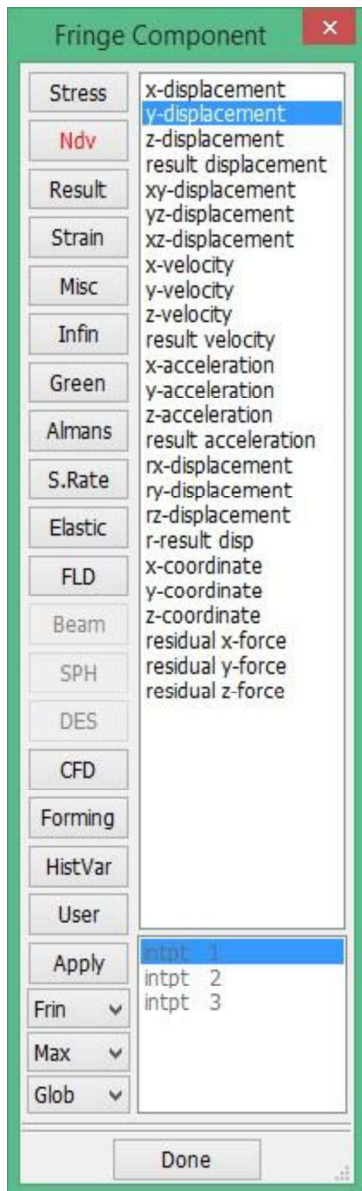


When you click on the curve, information about the picked point will be stated in the **Message box**, which is located at the bottom of the LS-PrePost window. If you double-click in this box, a bigger box will pop up.

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**.



3.5.11 Analytical solution

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

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

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

4. Exercises

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

- material?



Erasmus+

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

References

Tutorials taken from: <https://www.dynasupport.com/tutorial/introduction-ls-dyna-ls-prepost-for-explicit-and-implicit-analysis>

<https://www.dynasupport.com/manuals>