



# Simulation Tutorial

## Modeling the Crushing Behavior of a Tubular Structure using the Ansys LS-DYNA Software

Axel Hallén, Marcus Gustavsson, Thomas Magnac, and János Plocher

Edited and curated by the Ansys Academic Development Team

[education@ansys.com](mailto:education@ansys.com)

## Ansys Software Used

This resource uses Ansys LS-DYNA®, the nonlinear dynamics structural simulation software and Ansys LS-PrePost®, a capability within the Ansys LS-DYNA tool.

## Summary

This simulation tutorial aims to provide a basic introduction into running Ansys LS-DYNA software simulations with the help of Ansys LS-PrePost and Ansys LS-Run capabilities. The tutorial features an example of a tubular structure with a square cross-section that is crushed against a flat, rigid plate. The tutorial is divided into four distinct topics, each with an accompanying video. It begins with the topic of meshing with the purpose of creating the structure for the analysis. The second part covers the analysis setup of creating the necessary keywords, wherein some of the basic modeling aspects of the Ansys LS-DYNA tool are touched upon. For the third part of the tutorial, you perform model checking prior to running the analysis, after which you also learn how to perform the simulation using the Ansys LS-Run capability. The fourth and final part of the tutorial concerns post processing, wherein you learn how to analyze the results using some of the functionality available in the Ansys LS-PrePost capability. An accompanying YouTube Tutorial series for all four parts can be found on the [Ansys Learning Channel here](#).

## Table of Contents

1. Meshing and Various Tools within the Ansys LS-PrePost tool .....	5
1.1 Importing and Meshing of Tube Geometry .....	5
1.1.1 Importing the CAD-file.....	5
1.1.2 Meshing the Geometry with the N-Line Mesher and Auto Mesher Tools.....	5
1.2 Creating the Rigid Plate .....	8
1.2.1 Using the Shape Mesher Tool to Create the Rigid Plate .....	8
1.3 Creating the End Plate .....	8
1.3.1 Using the Shape Mesher Tool to Create Half the End Plate .....	8
1.3.2 Using the Transform Tool to Create the Rest of the End Plate by Copying and Rotating the Elements.....	9
2. Analysis Setup with Ansys LS-PrePost tool .....	10
2.1 Creating and Assigning Materials .....	10
2.1.1 Creating a Rigid Material for the Flat Plate.....	10
2.1.2 Creating an Elasto-Plastic Material for the Tube .....	11
2.1.3 Assigning Materials to Parts .....	12
2.2 Creating and Assigning Element Formulations .....	12
2.2.1 Creating Element Formulations .....	12
2.2.2 Assigning Element Formulations to Parts .....	13
2.3 Creating Symmetry Boundary Conditions .....	13
2.3.1 Creating node sets.....	13
2.3.2 Creating Boundary Conditions.....	14
2.4 Creating Boundary Conditions for the End Plate .....	15
2.4.1 Constraining the End Plate .....	15
2.4.2 Prescribing a Motion for the End Plate .....	16
2.5 Setting up Contact Definitions .....	17
2.5.1 Creating a tied contact between the end plate and the tube .....	17
2.5.2 Defining Contact between the Tube and the Rigid Plate.....	17
2.5.3 Defining Self-Contact for The Square Tube .....	18
2.6 Giving your Simulation a Title.....	18
2.7 Setting the End Time for your Simulation.....	19
2.8 Time-Step Control .....	19
2.9 Hourglass Control .....	19
2.10 Defining Output from the Simulation .....	20
2.10.1 *DATABASE_ASCII_OUTPUT .....	20
2.10.2 *DATABASE_EXTENT_BINARY .....	21
2.10.3 *DATABASE_CROSS_SECTION_PLANE.....	21

3. Model Checking and Ansys LS-Run tool.....	21
3.1 Performing a Model Check to Find Errors.....	21
3.1.1 Keyword Check.....	21
3.1.2 Contact Check.....	23
3.2 Saving the Keyword File .....	24
3.3 Running the Simulation using LS-RUN .....	24
3.3.1 Specifying Input File .....	24
3.3.2 Specifying Solver Version.....	24
3.3.3 Specifying Number of CPUs and Memory .....	25
3.3.4 Submitting the Model for Analysis .....	25
4. Analyzing Results Using Ansys LS-PrePost tool .....	26
4.1 Opening the Results .....	26
4.2 Animating the Results .....	26
4.2.1 Animating the Quarter Model .....	26
4.3 Fringe Plotting .....	27
4.3.1 Fringe Plotting von Mises Stress.....	27
4.3.2 Fringe Plotting Effective Plastic Strain .....	27
4.3.3 Fringe Plotting Z-Displacement .....	28
4.4 The History Plot Tool .....	29
4.5 Creating Cross-Plots with the XY-Plot Tool .....	30
4.5.1 Using History Plot to Save Data .....	30
4.5.2 Creating the Cross-Plot.....	31
4.6 Reading Binary Output Data from Binout .....	32
4.6.1 Plotting Model Energies from Global Statistics.....	32
4.6.2 Plotting the Critical Time Step from Global Statistics .....	33
4.6.3 Plotting the Contact Force on the Tube .....	34
4.6.4 Plotting the Cross-Section Force .....	34
5. Concluding Remarks .....	36

# 1. Meshing and Various Tools within the Ansys LS-PrePost tool

## 1.1 Importing and Meshing of Tube Geometry<sup>1</sup>

### 1.1.1 Importing the CAD-file

With Ansys LS-PrePost tool open, start by opening the CAD-geometry of the tube which is contained in the file called **sym\_tube.igs**. **Do this by going to File** and then **Open**, located in the top menu. Use the default settings for the import.

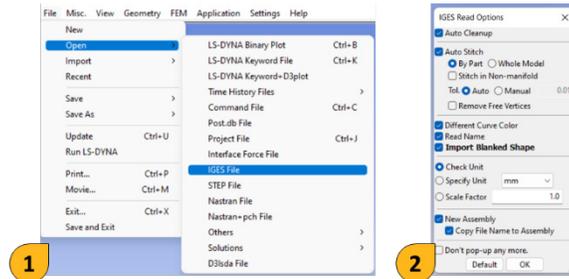


Figure 1: Importing the igs-format CAD-file of tubular structure.

### 1.1.2 Meshing the Geometry with the N-Line Mesher and Auto Mesher Tools

To mesh the tube, we will be using the *N-Line Mesher tool as well as the Auto Mesher tool*<sup>2</sup>. Select the N-Line Mesher from under the Mesh menu and choose the “4 Lines Shell” Type in the context menu. Then start at the left end of the tube by selecting the four edges defining the rectangular face as shown in Figure 2. Note if the description of the icons in the toolbar is not visible, you can include via the View Ribbon.

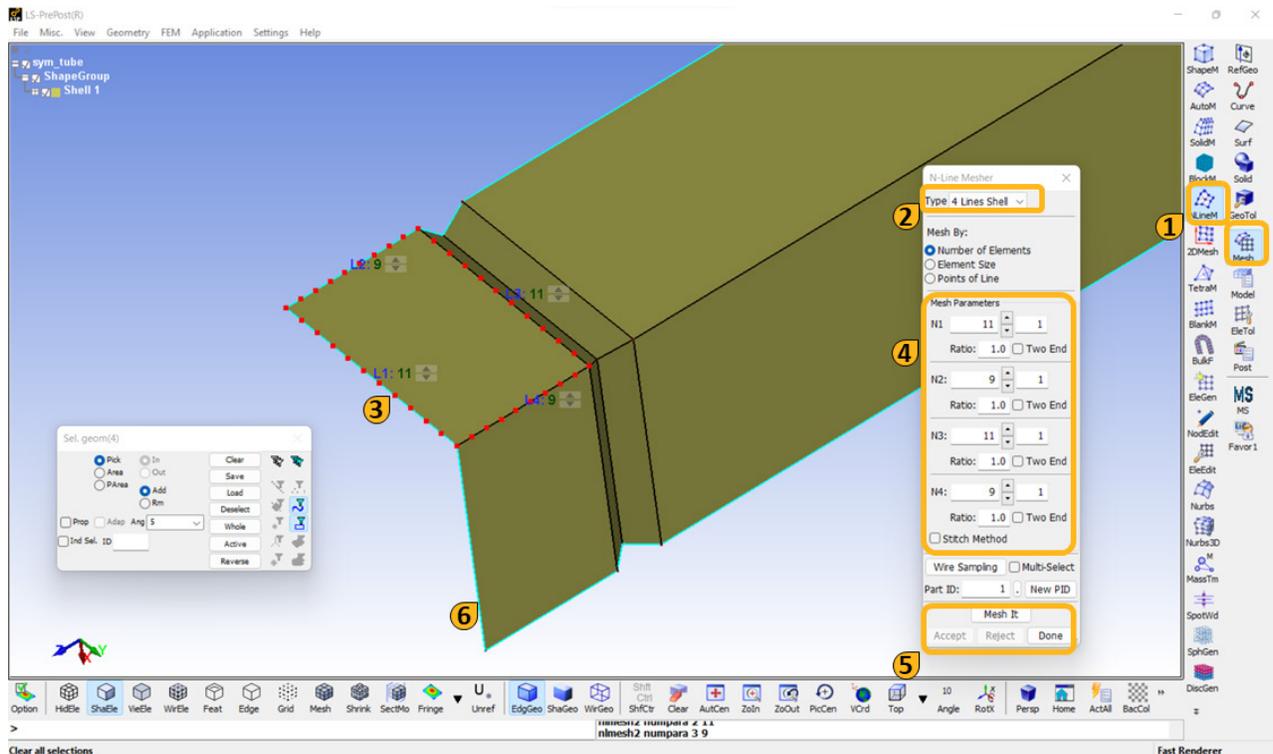


Figure 2: Employing N-Line Mesher and front tube section.

1 Please refer to the [tutorial YouTube video](#) for a walkthrough of Part 1

2 Please refer to the [Ansys LS-PrePost capability documentation](#) for more information

Set the number of elements in the width direction (global y-axis) to be 11 and the number of elements in the longitudinal direction (global z-axis) to be 9 and select “Mesh It” and “Accept” Repeat the process for the neighboring face (as indicated by step 6 in Figure 2) of the same size, and make sure the elements are assigned to the same part id as the previously created ones (here Part ID: 1). When meshing the inclined faces, maintain 11 elements in the width direction but reduce the number of elements in the longitudinal direction to 2. When meshing these faces, try assigning the elements to a new part id and then use the tool called *Move or Copy (MovCop)* to move them to the previous part. This tool is located under *Element Tools* (see Figure 3). Select the elements in the inclined by choosing the appropriate selection option and subsequently Part ID (PID) as indicated in Figure 3. To visualize the elements, use the Mesh Icon in the bottom toolbar.

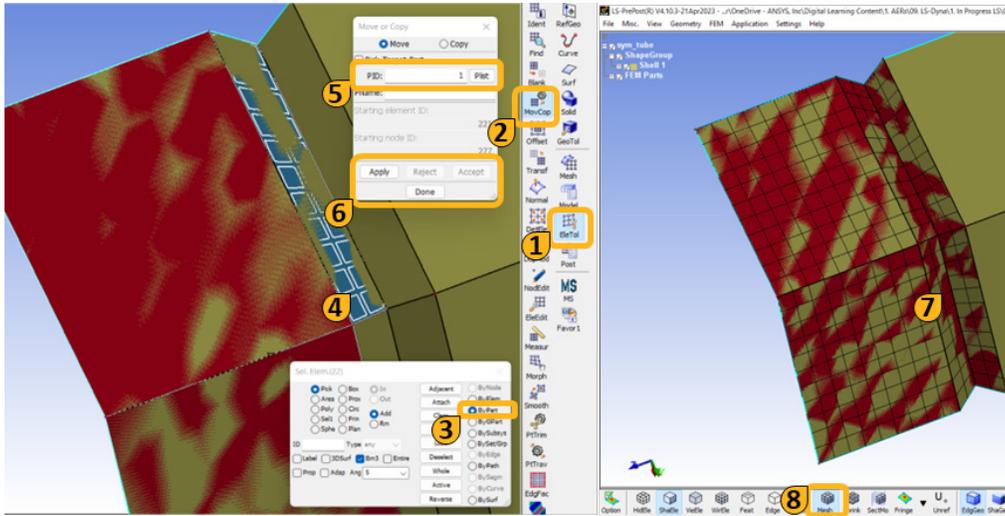


Figure 3: Meshing of inclined faces using the MovCop tool and visualizing final mesh.

When meshing the two remaining faces, the *Auto Mesher* tool will be used. With the tool selected, click on the two faces and use *size* for the mesh mode option. Set the element size to be 4.5 mm and assign the elements to the same part id as the rest of the elements (see Figure 4).

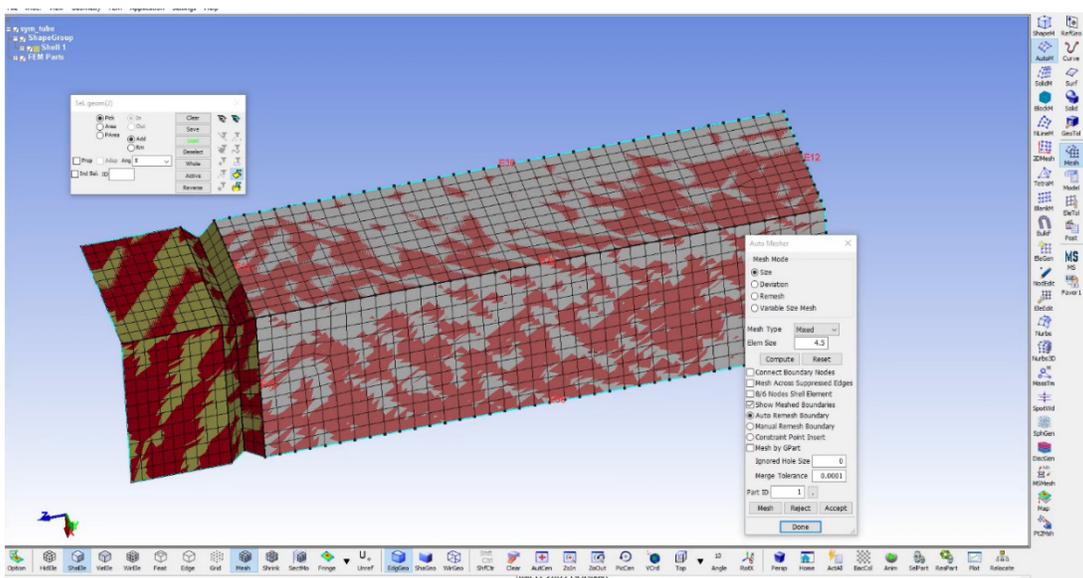


Figure 4: Use the Auto Mesh (AutoM) tool to mesh the remainder of the tube structure with element size 4.5.

Ensure node-to-node connectivity with the previously created mesh by adjusting the number of elements along the short edges to be 11 (see Figure 5), in order to get rid of any triangular elements.

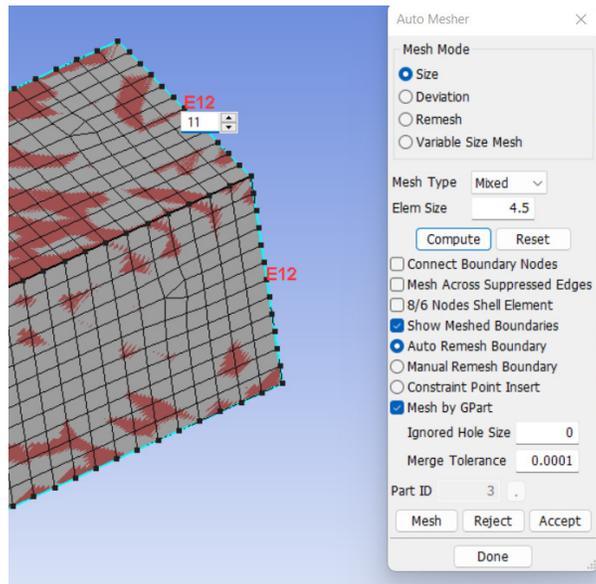


Figure 5: Adjusting edge element number from 12 to 11.

Because the mesh was created in separate stages, there will be duplicate node definitions in the boundaries between the different faces. Get rid of these extra nodes by using the Duplicate Nodes tool located under Element Tools (see Figure 6). Select all nodes in the model by choosing the area selection and box selecting the entire structure. Next, click on show duplicate nodes followed by merge duplicate nodes and accepting it by clicking done.

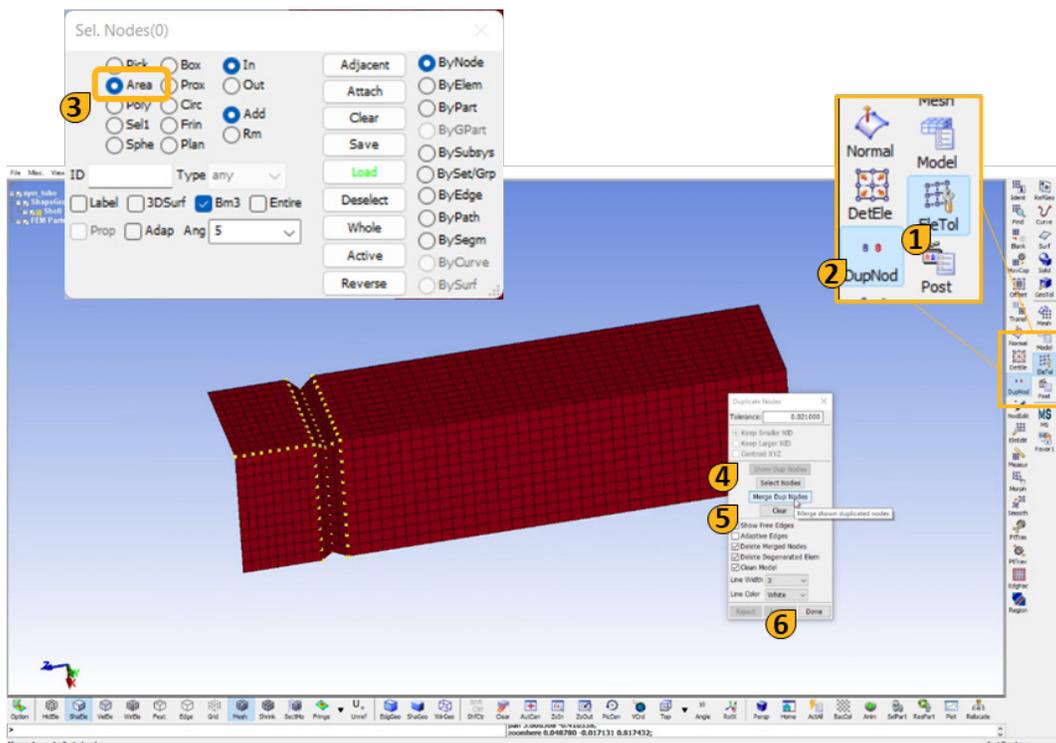


Figure 6: Removing duplicate nodes in the mesh.

## 1.2 Creating the Rigid Plate

### 1.2.1 Using the Shape Mesher Tool to Create the Rigid Plate

The Shape Mesher tool is located under Mesh (see Figure 7). Select it and set the entity to 4N Shell. In the fields for P1-P4, the coordinates of the corners of the plate are specified. First check where the origin of the model is located. Do this by using the Measure tool located under Element Tools. Measure the coordinate of the top left corner of the tube and check. Switch to measuring distance from one node to another and measure the height of the tube. The origin is thus located 50 mm below the top left corner of the tube.

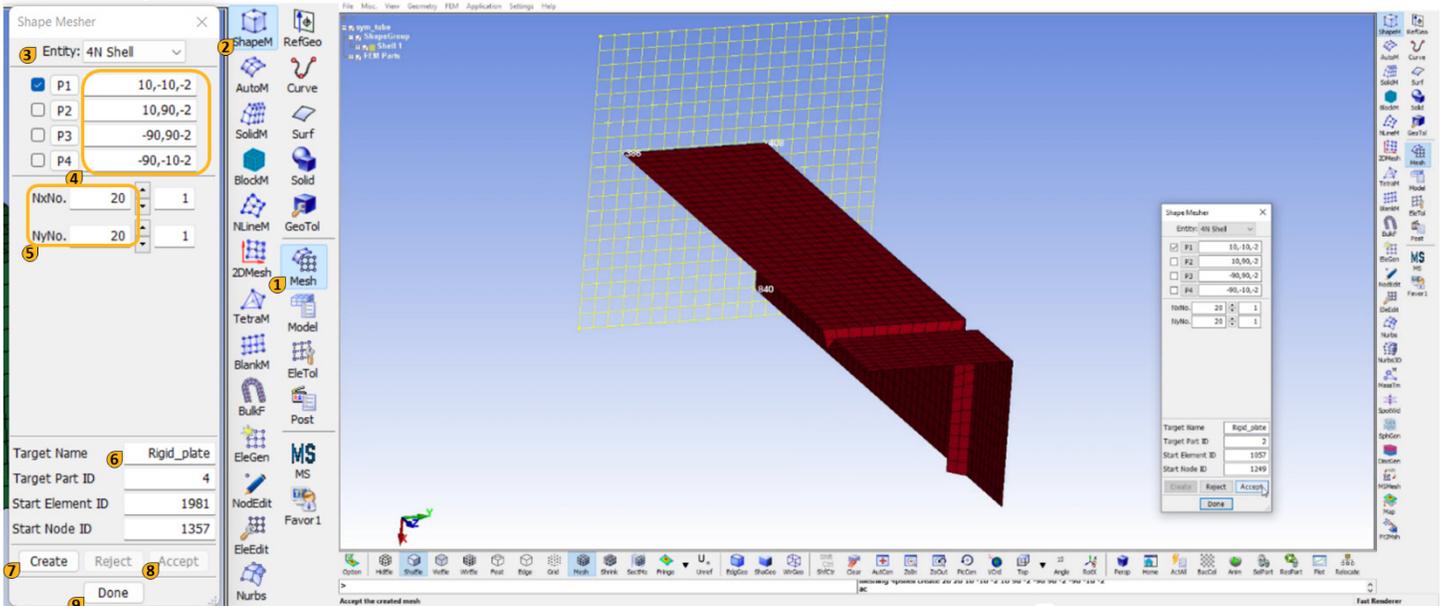


Figure 7: Creating the rigid end plate.

Go back to Shape Mesher and type in the coordinates of the corners of the plate, yielding a 100 by 100 mm large plate. Create 20 elements in both the x- and y-directions by using the NxNo and NyNo parameters, respectively. Make sure the elements are assigned to a new part id and give it a fitting name (see Figure 7).

## 1.3 Creating the End Plate

### 1.3.1 Using the Shape Mesher Tool to Create Half the End Plate

Select Shape Mesher and specify a rectangle that is 30 mm long in the x-direction and 51.75 mm long in the y-direction and let the first corner be located at the origin. Create 9 elements in the x-direction and 11 elements in the y-direction and make sure that the first corner of the end plate is located at the origin. Make sure the elements are assigned to a new part id and give it a fitting name.

The end plate is to be placed at the rear end of the tube. To place it there, we shall use the Transform tool which is located under Element Tools. With Transform selected, select all the elements of the newly created part and rotate them positive 90 degrees around the x-axis.

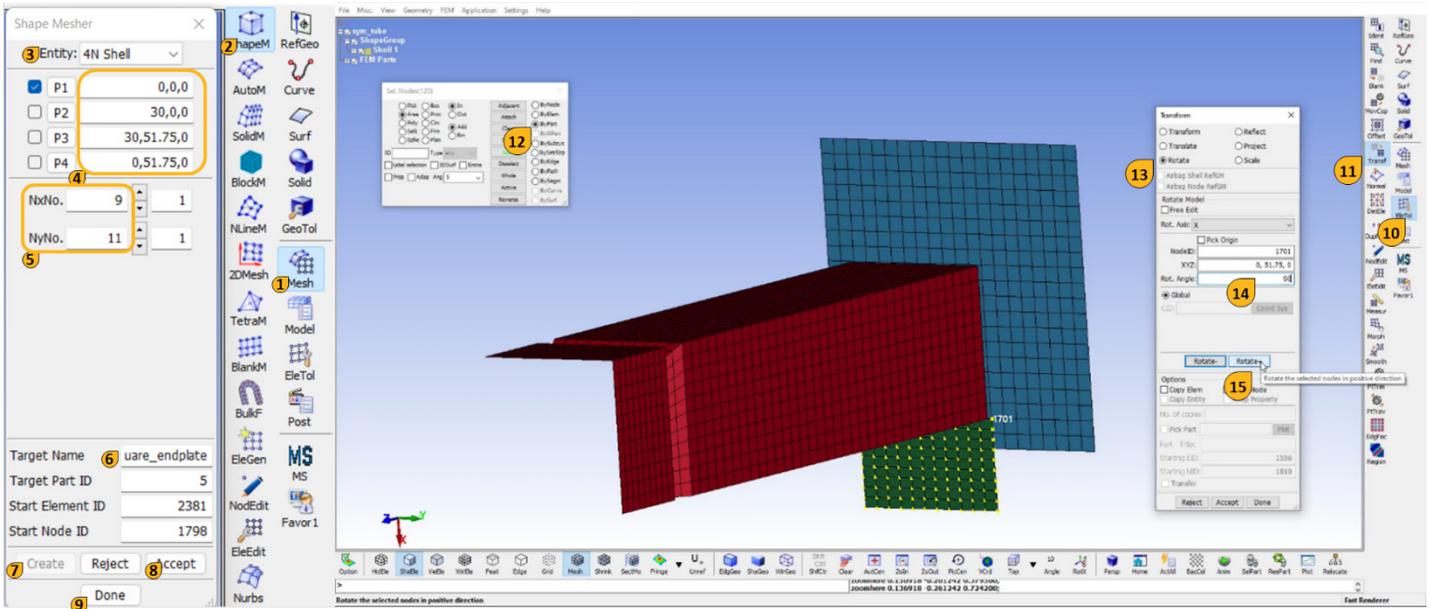


Figure 8: Steps for creating the endplate via the Shape Mesher tool and conducting rotations and translations to orientate it as desired.

Repeat the process but this time rotate about the y-axis instead. Finally, to move the end plate in place, stay in the Transform tool and translate the elements 228 mm in the z-direction.

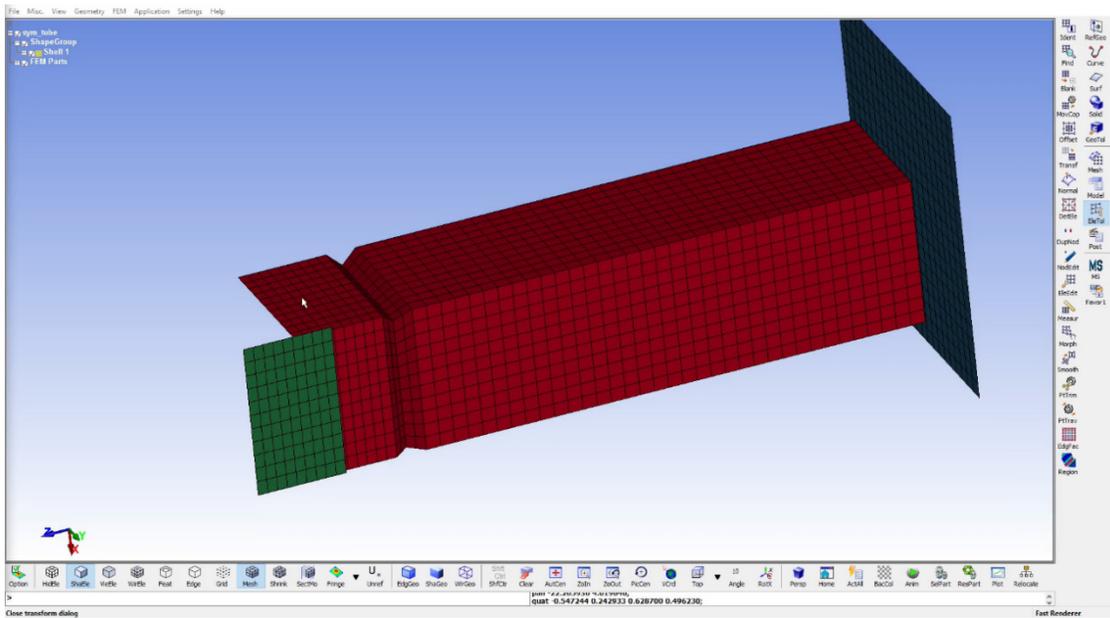


Figure 9: Final position of the end plate (green).

### 1.3.2 Using the Transform Tool to Create the Rest of the End Plate by Copying and Rotating the Elements

The end plate is only half finished and to create the remaining part that will cover the top of the tube we will again be using Transform. Select the elements of the end plate, choose rotate as operation and set the z-axis as the axis of rotation (see Figure 10). Choose the top right corner of the end plate as the origin about which to rotate. Under options, check the box for copy elements and set the number of

copies to be 1. Make sure that the elements that we are about to create are assigned to the same part id as the already existing elements of the end plate. Rotate negative 90 degrees.

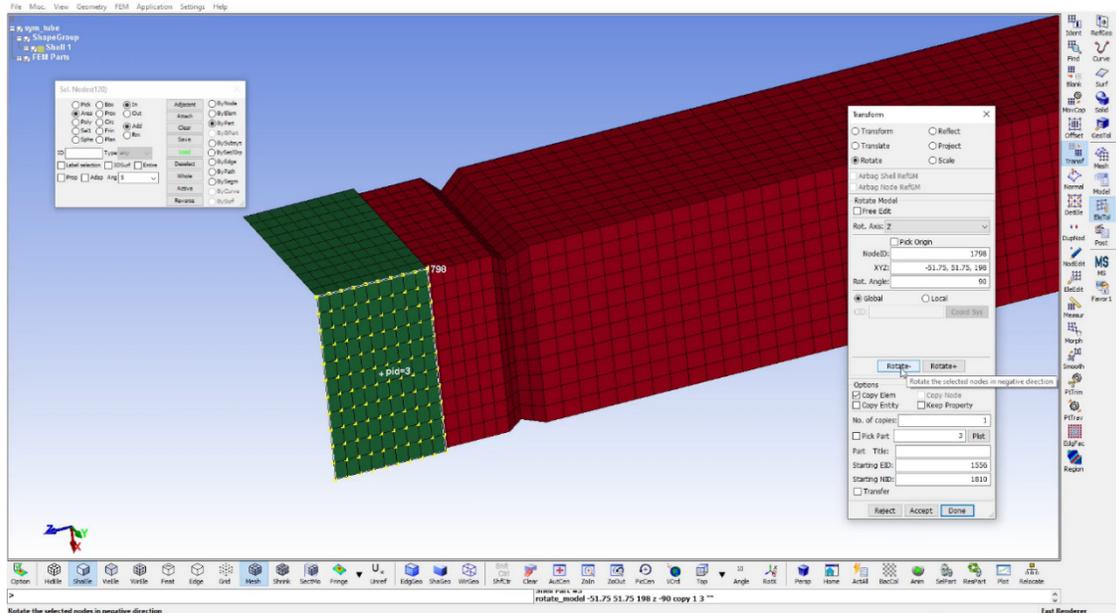


Figure 10: Steps for copying and moving the second part of the end plate.

Again, repeat the process from before of eliminating the duplicate node definitions in the boundary by using the tool called Duplicate Nodes.

## 2. Analysis Setup with Ansys LS-PrePost tool<sup>3</sup>

### 2.1 Creating and Assigning Materials

#### 2.1.1 Creating a Rigid Material for the Flat Plate

Keywords are created using the Keyword Manager which is found in the toolbar on the right-hand side of the graphical user interface, within the menu called Model (see Figure 11). In Keyword Manager, you can toggle between Model, to see the keywords currently defined in your model, or All, to see all available keywords in the Ansys LS-DYNA software. In The Ansys LS-DYNA software, rigid bodies are created by assigning a rigid material to them. This material keyword is called \*MAT\_RIGID, which is material model no. 20. It is good practice to assign steel like mechanical properties to the rigid material since even if they are not used to calculate deformations of the material, they are used in contact definitions when the rigid body comes into contact with other bodies in the model. On the second card of the keyword are three options that let you constrain rigid bodies using this material. Constrain the rigid body in all degrees of freedom, translational as well as rotational ones.

<sup>3</sup> Please refer to the [tutorial YouTube video](#) for a walkthrough of Part 2

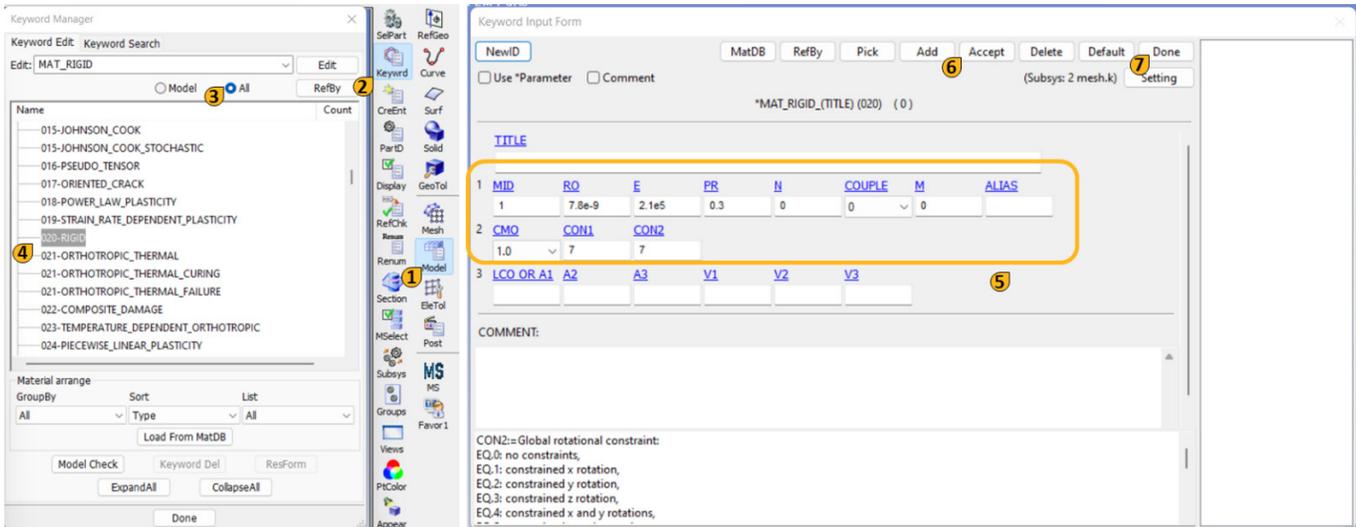


Figure 11: Steps for assigning the material properties of the flat plate.

When creating a keyword, you need to press Accept and then Done for it to be saved. Please note that when working with The Ansys LS-DYNA software, you do not input any units to the program. All numbers you input are unitless and it is up to you as the user to make sure that you are using a consistent system of units when you input values. Take a look at this website to see which choices you have for consistent unit systems: [lsdyna.ansys.com/consistent-units](http://lsdyna.ansys.com/consistent-units).

### 2.1.2 Creating an Elasto-Plastic Material for the Tube

For the tube, create a material that has steel-like mechanical properties, and which can undergo plastic deformation. Use material model no.24, i.e., \*MAT\_PIECEWISE\_LINEAR\_PLASTICITY (see Figure 12). It is a commonly used and versatile material model for metal plasticity. In this case, simply define the hardening-behavior of the metal with a yield limit and a tangent modulus, meaning that we will define linear work hardening. This is the simplest way to describe plasticity. Normally, with this material model, you input hardening curves directly instead of defining the yield stress and tangent modulus.

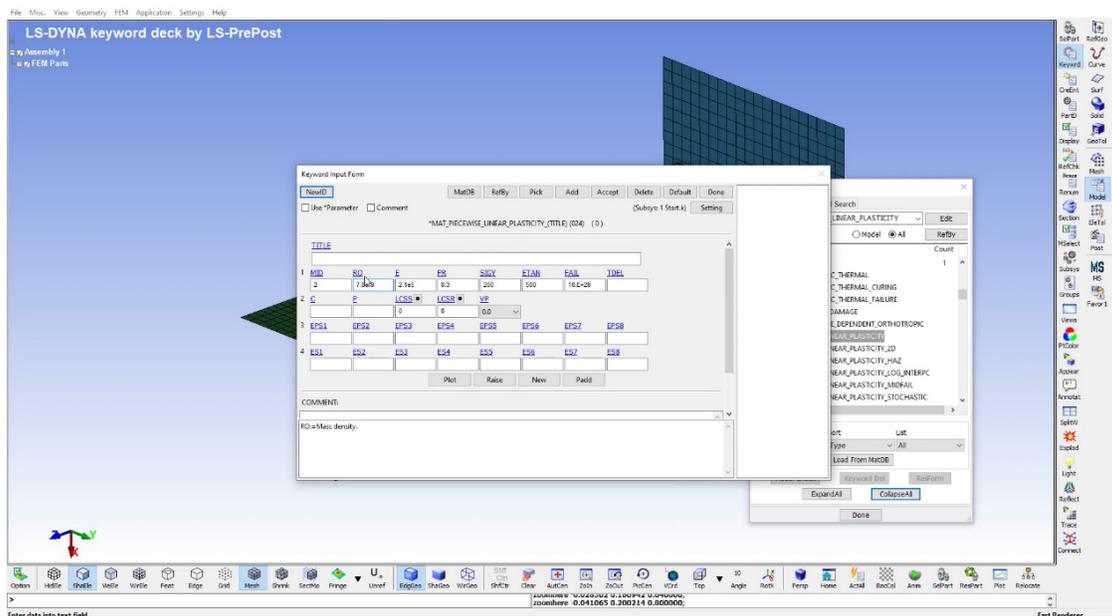


Figure 12: Assigning steel-like plasticity material properties to the tube.

Please note that in the Ansys LS-DYNA software, measures of stress and strain are always true stress and true strain. If you should input values of engineering stress and engineering strain, it will be stated explicitly.

### 2.1.3 Assigning Materials to Parts

Materials are assigned to parts on the \*PART keyword. In Keyword Manager, Switch to Model, and locate the part keywords. For each of the three parts, assign them the correct material using the third option called MID which is the ID you set when creating the material keywords. Assign the rigid material to the plate, and the elasto-plastic material to the tube and the end plate (see Figure 13).

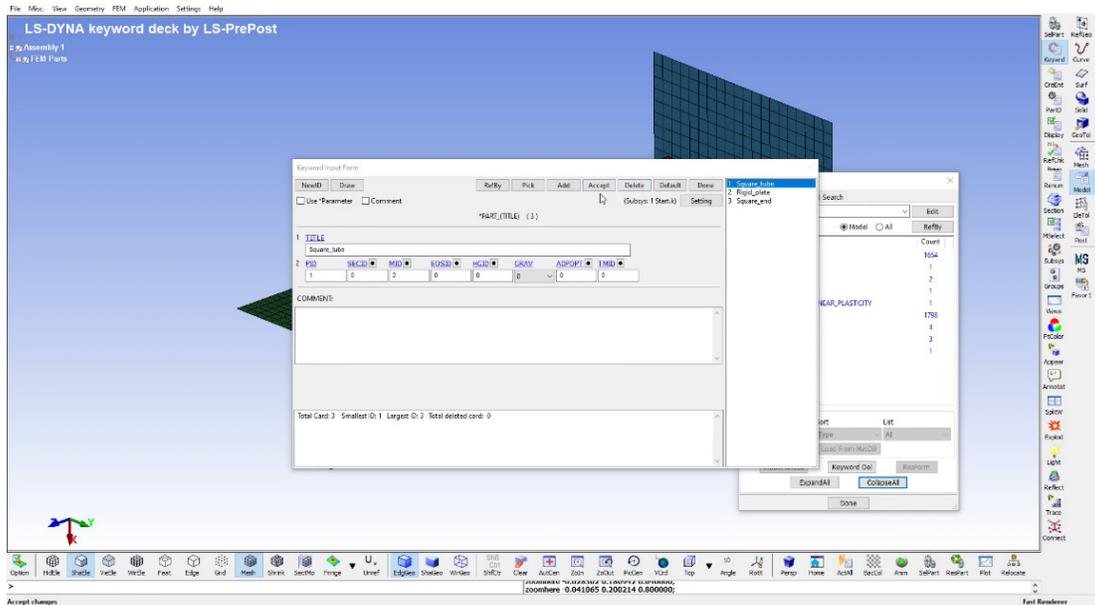


Figure 13: Assignment of material properties to parts.

## 2.2 Creating and Assigning Element Formulations

### 2.2.1 Creating Element Formulations

Element formulations are specified with the keywords beginning with \*SECTION. Since the mesh being used in this tutorial consists of shell elements, naturally, we must use \*SECTION\_SHELL. Set the element formulation to 2, and the thickness to 1.5 mm (see ). The number of through-thickness integration points should always be set to an odd number to ensure an even distribution around mid-section of the shell. A shell element requires at least 5 integration points through its thickness to be able to capture elasto-plastic behavior. Note that in the video, it was set to 3 by mistake. Create another \*SECTION\_SHELL with the same parameters as the previous one but with a thickness of 2 mm.

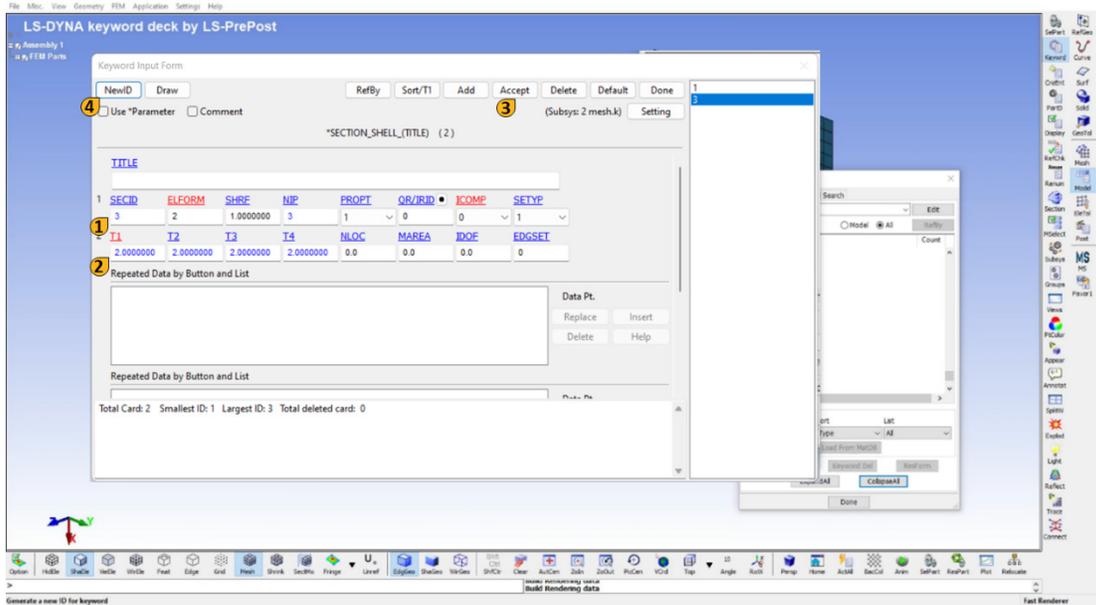


Figure 14: Creating the element formulation for the shells.

### 2.2.2 Assigning Element Formulations to Parts

Assign the newly created element formulations to the three parts in the model in the same way as was previously done when assigning materials. Element formulation is specified with the parameter called SECID, as shown in Figure 15. Assign the element formulation with a thickness of 1.5 mm to the tube and the rigid plate (i.e. SECID = 1), and the one with a thickness of 2 mm to the end plate (i.e. SECID = 3).

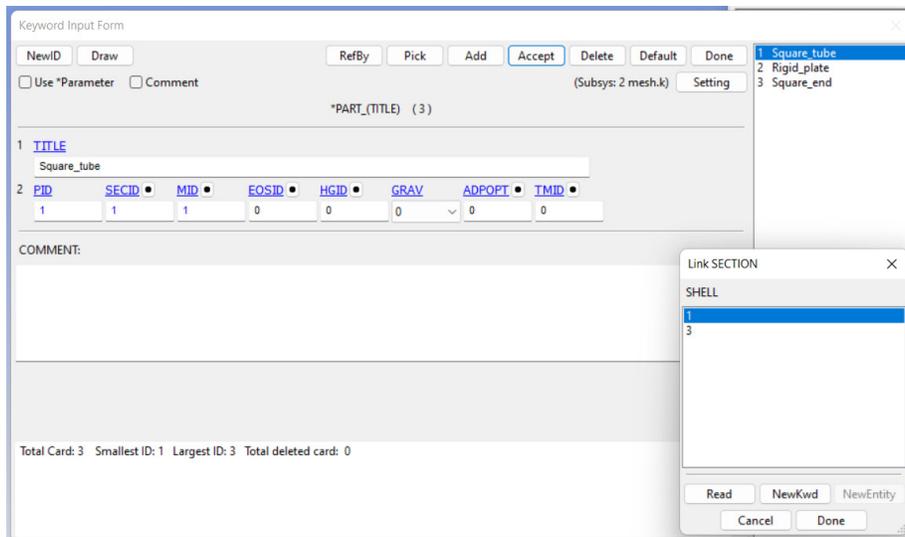


Figure 15: Section ID assignment to the parts.

## 2.3 Creating Symmetry Boundary Conditions

### 2.3.1 Creating node sets

The square tube is symmetrical both in the x- and in the y-direction so we are only modeling one quarter of the full tube to reduce computational cost. Symmetry conditions will have to be defined, and this will be done by applying boundary conditions on the nodes belonging to the edges of the model. The easiest way to create node sets is to use the Create Entity tool which is found under Model

(see Figure 16). With Create Entity selected, toggle on create mode. In the selection window, check the by edge option as well as the option to propagate the selection. Then select a node on the tube, on the edge in the y-direction. Select the whole edge as well as the edge of the end plate and give the node set a fitting name. Repeat the process to create a node set for the symmetry condition in the x-direction.

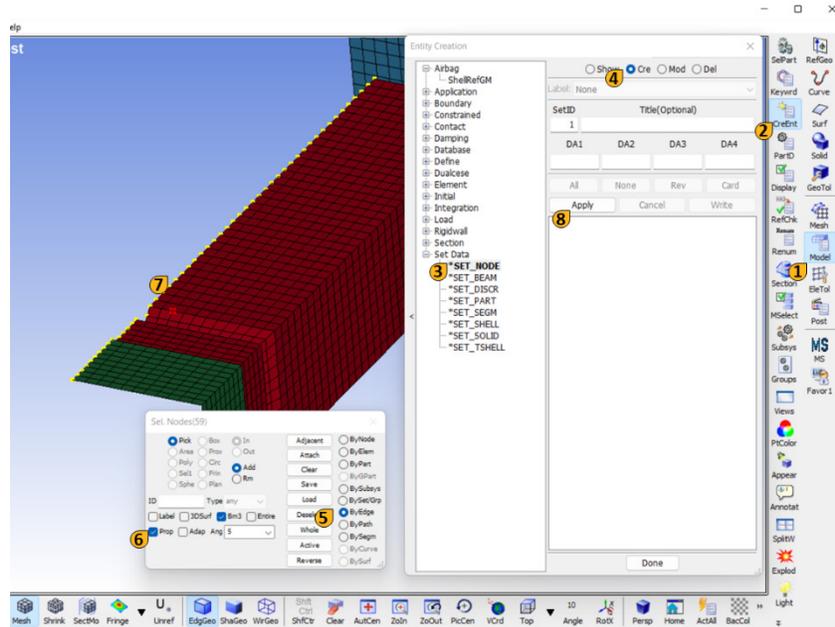


Figure 16: Employ the Create Entity tool to create a node set at the symmetry planes.

### 2.3.2 Creating Boundary Conditions

To create the boundary conditions to define the symmetry, we will use the keyword called `*BOUNDARY_SPC_SET`, where SPC stands for single point constraint. The syntax of this keyword is quite straightforward. Specify the node set that the constraints are to be applied on with the option `NSID`. Then apply constraints of the rotational degrees of freedom for those nodes with the remaining options, as shown in Table 1. Create another keyword to define the symmetry condition in the x-direction. That means that you need to constrain translation in the x-direction as well as rotation about the y- and z-axes.

Table 1: Single Point Constraint Input for the edges defining the symmetry condition of the quarter model.

NSID	CID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
1	0	0	1	0	1	0	1
2	0	1	0	0	0	1	1

The option `CID` lets you control in respect to which coordinate system the constraints are applied. For this tutorial you can ignore that option since it is set to the global coordinate system by default. Since the elements in this model are shell elements, each node has six degrees of freedom, three translational ones and three rotational ones. For the symmetry condition in the y-direction, set constraints on y-translation as well as rotation about the x- and z-axes.

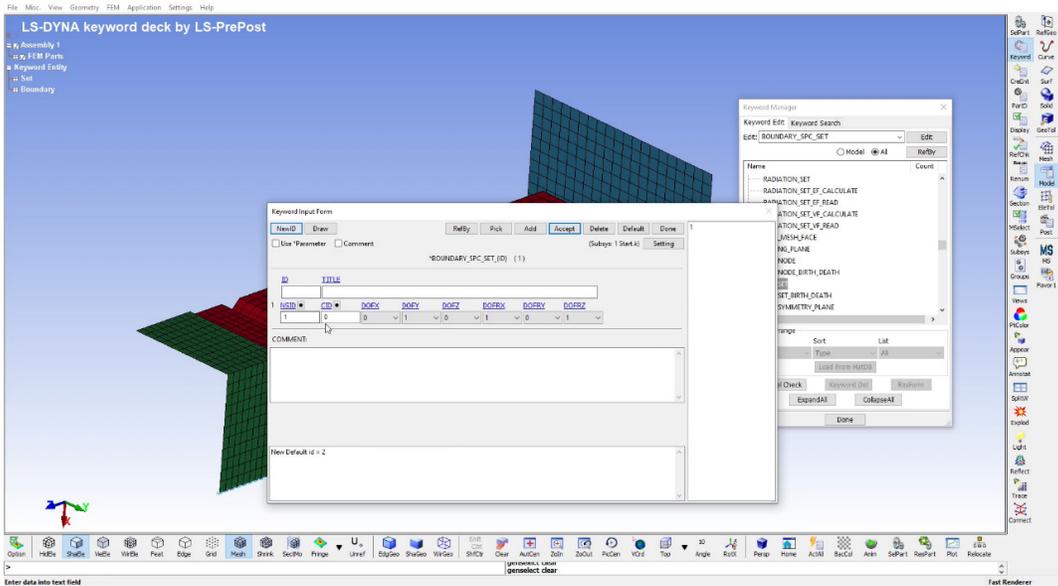


Figure 17: Creating single point boundary constraints.

## 2.4 Creating Boundary Conditions for the End Plate

### 2.4.1 Constraining the End Plate

The end plate will be fixed so that it can only translate in the direction of the motion, i.e., the z-direction. Create a node set containing the nodes belonging to the end plate on the two edges which are furthest in the z-direction, as shown in Figure 18. Constrain all degrees of freedom except for translation in the z-direction for this newly created node set, using the keyword `*BOUNDARY_SPC_SET`, as outlined in Table 2

Table 2: Single Point Constrain Input for the end plate edge of the quarter model.

NSID	CID	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
1	0	1	1	0	1	1	1

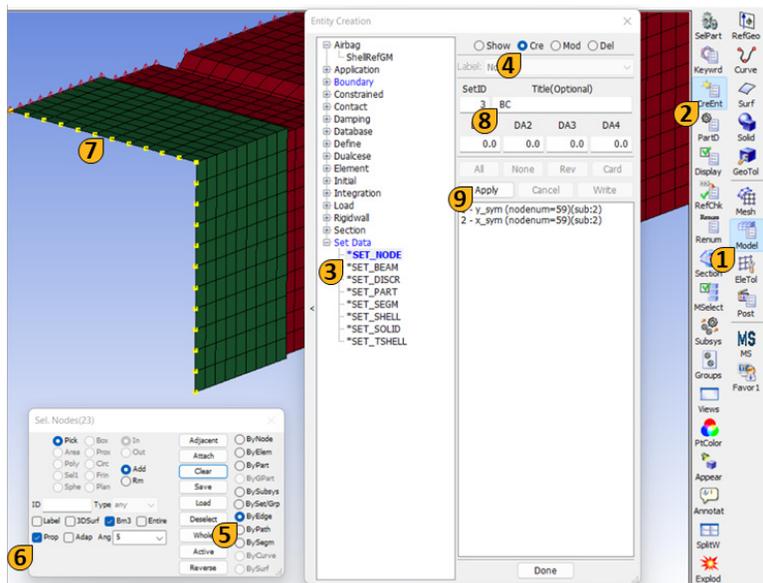


Figure 18: Node set of the end plate edge.

## 2.4.2 Prescribing a Motion for the End Plate

We will crush the square tube by prescribing a displacement to the end plate which is going to be connected to the tube. This displacement will be prescribed to the edge of the end plate, meaning we can use the same node set as we used to constrain the edge just before. Use the keyword called `*BOUNDARY_PRESCRIBED_MOTION_SET` to prescribe the displacement. This keyword will require you to input a curve which defines the motion, so create that first. Use the keyword `*DEFINE_CURVE` and create a ramped curve from 0 to 0.02 seconds (see Figure 19). Best practice is to not input values on curves directly, instead let the ordinate value go up to 1. The curve will later be scaled to get the correct value for the prescribed displacement. The end time of the simulation is going to be 0.02 seconds, but for numerical reasons, if you have a curve which is used up until the end of a simulation, always extend it slightly beyond the termination time. This is to avoid problems with the Ansys LS-DYNA software extrapolating that curve and which could happen, and then the curve is extrapolated in the tangential direction of the last data point.

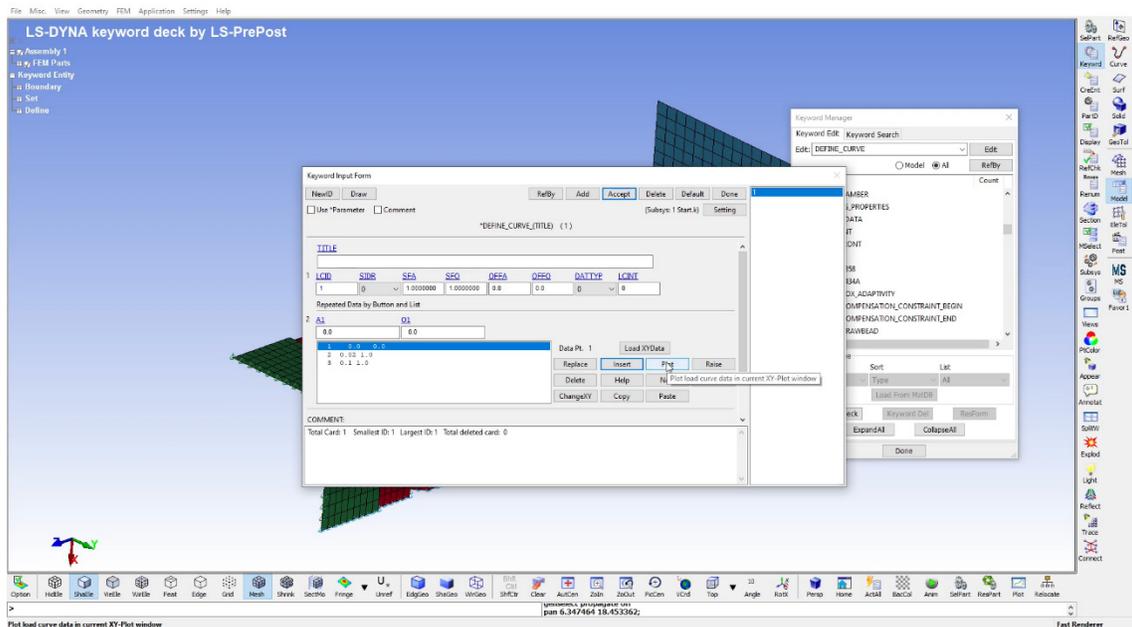


Figure 19: Ramped curve definition.

Create a `*BOUNDARY_PRESCRIBED_MOTION` keyword wherein you specify the newly created curve at the option for load curve id (LCID = 1). Next, use the scale factor (SF) option to multiply the ordinate value of the load curve by a factor of -80, meaning that the end plate is going to be prescribed a displacement of 80 mm in the negative z-direction.

Please note that there is a curve generation tool in Ansys LS-PrePost tool, located at the top menu under Application, Tools, CurveGen. This tool lets you create smooth curves containing many points with ease (see Figure 20).

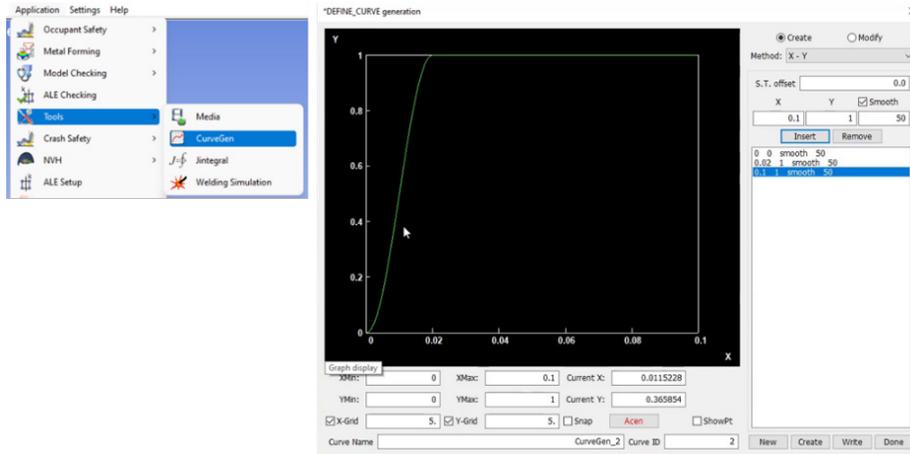


Figure 20: The Curve Generation tools allows for a much smoother curve creation.

## 2.5 Setting up Contact Definitions

### 2.5.1 Creating a tied contact between the end plate and the tube

The end plate can be connected to the tube by means of a tied contact. One option is to use the keyword called `*CONTACT_TIED_NODES_TO_SURFACE_OFFSET`. Surface A and surface B refer to the two sides of a contact. Set the options `SURFATYP` and `SURFBTYP` to 3 to indicate that you will define the two sides of the contact by their part definitions. Then proceed to set the tube as surface A and the end plate to surface B, meaning that the nodes of the tube will be tied to the end plate, as shown in Figure 21.

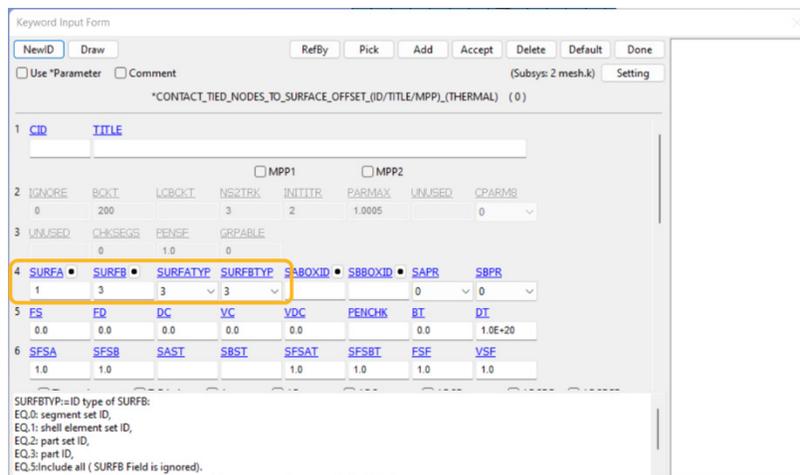


Figure 21: Tied contact definition between end plate and tube.

### 2.5.2 Defining Contact between the Tube and the Rigid Plate

Contact between the tube and the rigid plate is necessary, otherwise the tube will simply pass through the plate. Use the keyword `*CONTACT_AUTOMATIC_NODES_TO_SURFACE`, and again define the surfaces on the part level. Set the square tube as surface A and the rigid plate as surface B, meaning that the contact algorithm will check if the nodes of the tube penetrate the segments of the rigid plate. Set the static coefficient of friction to 0.1, as shown in Figure 22

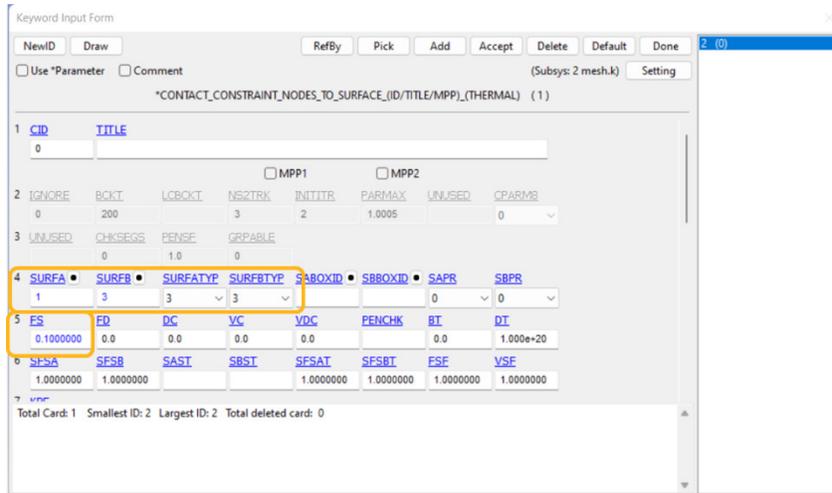


Figure 22: Contact definition between tube and the rigid plate.

### 2.5.3 Defining Self-Contact for The Square Tube

As the tube is crushed, it will start to buckle and come into contact with itself. It is therefore necessary to define a contact to allow for this behavior. Use `*CONTACT_AUTOMATIC_SINGLE_SURFACE`, wherein you only specify surface A (see Figure 23). Set the square tube as surface A and set the static coefficient of friction to 0.1.

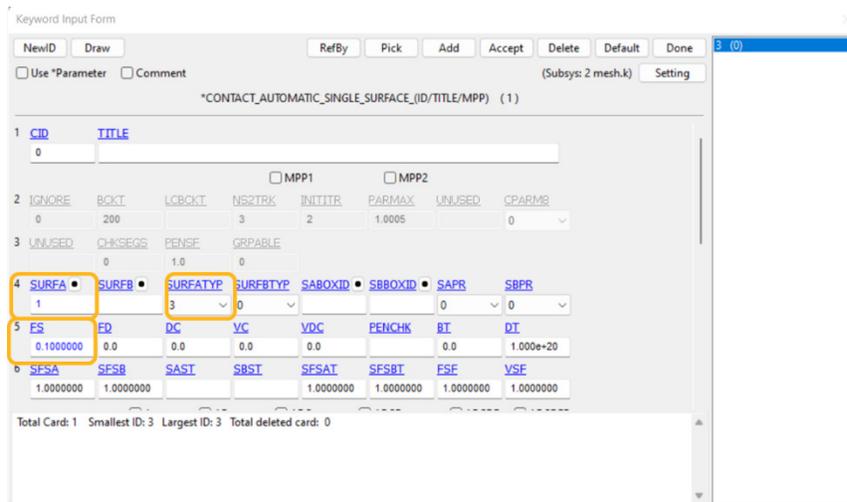


Figure 23: Self-contact definition for tube.

## 2.6 Giving your Simulation a Title

By using the keyword `*TITLE`, give your simulation a descriptive name which is then shown at the top left corner of the screen (see Figure 24).

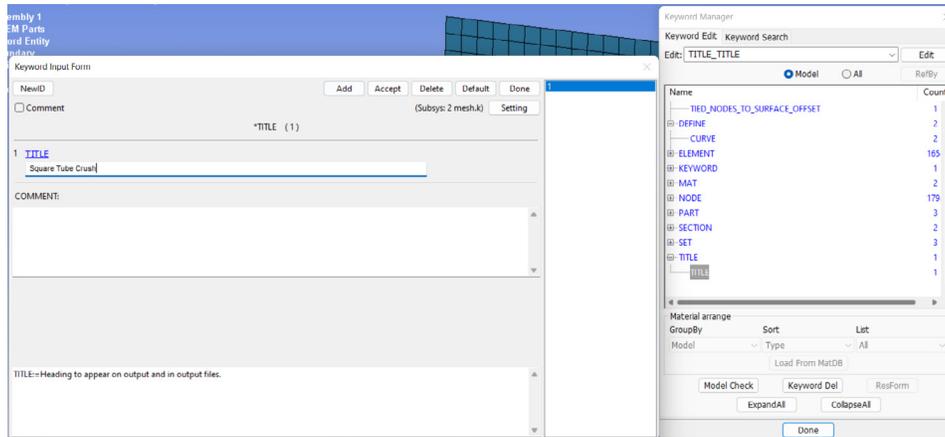


Figure 24: Assigning simulation a title.

## 2.7 Setting the End Time for your Simulation

You must always specify the end time of an analysis with the Ansys LS-DYNA software. This is done using `*CONTROL_TERMINATION`. Set `ENDTIM` to 0.02 seconds, as shown in Figure 25

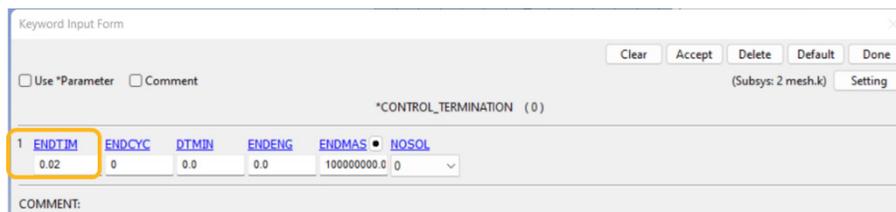


Figure 25: Setting the end time for the simulation.

## 2.8 Time-Step Control

The simulation in this tutorial is done using explicit time integration. Explicit time integration schemes are conditionally stable, and the Ansys LS-DYNA software uses a scale factor for the critical time-step to ensure numerical stability. This is specified using `*CONTROL_TIMESTEP`. Set the option `TSSFAC` to 0.9 (see Figure 26), meaning that the time-step used in the analysis is 90 % of the critical time-step. This is sufficient for most applications, but in some cases a lower value for the time-step scale factor might be necessary.

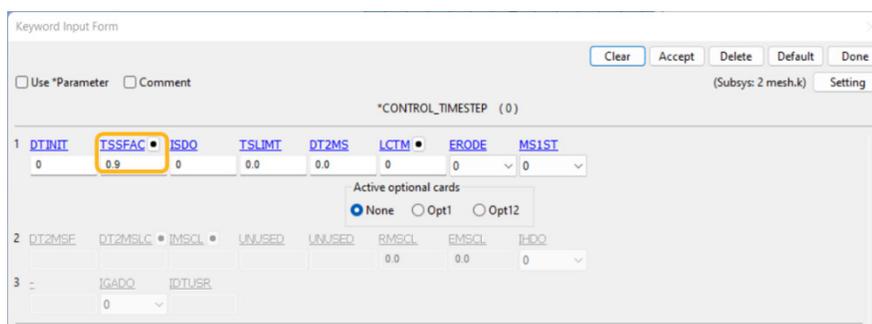


Figure 26: Time step control definition.

## 2.9 Hourglass Control

When using under-integrated element formulations such as in this tutorial, there is always a risk that you will encounter hour-glassing which is when elements deform in zero-energy modes. This is an entirely unphysical behavior and must be prevented. To combat this issue, hourglass control is used which is a scheme that applies forces to prevent hourglass deformation. Hourglass control can be

specified either for all parts in a model or for an individual part. Use `*CONTROL_HOURLGLASS` to define it for all parts in the model. Set the hourglass formulation (IHQ9 to 4 (i.e. stiffness form of type 2 (Flanagan-Belytschko))) and the factor (QH) to 0.1 (i.e. default), as shown in Figure 27.

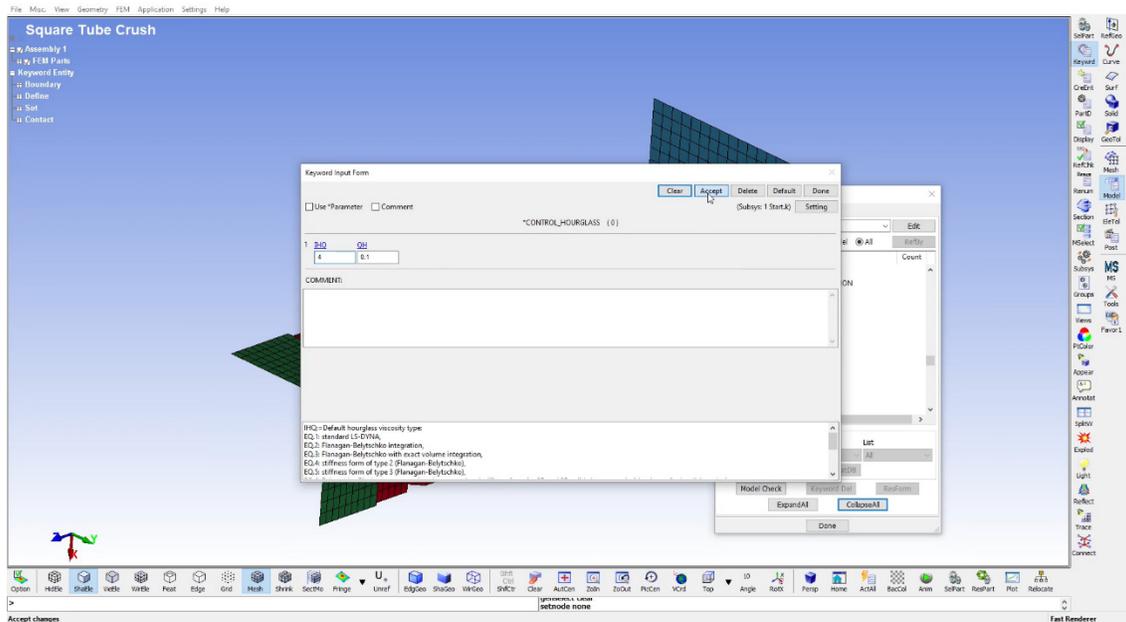


Figure 27: Hourglass controls

If you would like to learn more about hour-glassing, please see the following resources:

- [LSDYNA Hourglass \(ansys.com\)](https://ansys.com/learning/courses/ls-dyna-hourglassing)
- [Introduction to Explicit Dynamics using LS Dyna \(ansys.com\)](https://ansys.com/learning/courses/introduction-to-explicit-dynamics-using-ls-dyna)
- [LSDYNA Hourglass — Welcome to LS-DYNA Examples \(ansys.com\)](https://ansys.com/learning/courses/ls-dyna-examples)

## 2.10 Defining Output from the Simulation

### 2.10.1 \*DATABASE\_ASCII\_OUTPUT

To receive output of interest, you will often need to request it before running a simulation. Go to Keyword Manager and find the keyword called `*DATABASE_ASCII_OUTPUT`. Check the boxes for global statistics (GLSTAT) which provides information on model. In addition, check the box for the resultant interface (RCFORC) and cross-section (SECFORC) forces. For convenience, specify the output frequency of  $1e-5$  at the top with the field for Default DT and press enter (see Figure 28). This will autofill the output frequency for all your checked outputs.

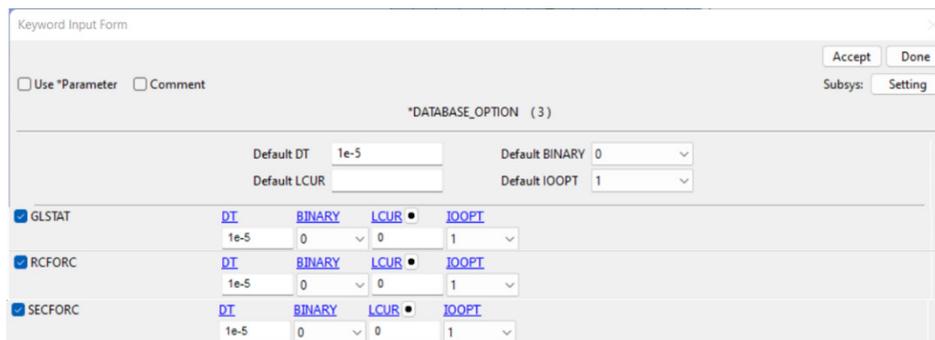


Figure 28: Defining output databases for the simulation.

### 2.10.2 \*DATABASE\_EXTENT\_BINARY

By default, plastic strain is not available in the output from an Ansys LS-DYNA software simulation. Instead, the user must request it by using the keyword \*DATABASE\_EXTENT\_BINARY and through setting the option STRFLG to 1 (see Figure 29).

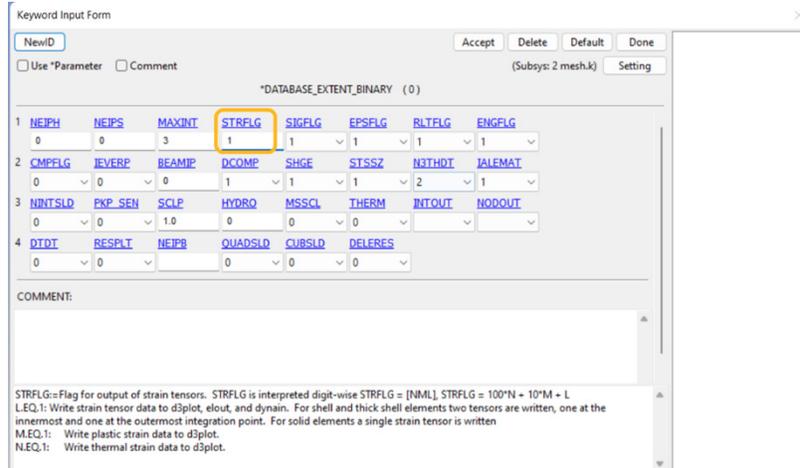


Figure 29: Activate the writing of strain tensor data.

### 2.10.3 \*DATABASE\_CROSS\_SECTION\_PLANE

By defining a cross-section plane through the tube, you will be able to get data of the forces acting on that cross-section, since you previously checked the box for cross-section forces data with \*DATABASE\_ASCII\_OUTPUT. Create a \*DATABASE\_CROSS\_SECTION\_PLANE 12 mm in from the end of the tube facing the rigid plate. Do this by setting ZCT (z-coordinate of tail of normal vector) to 12 and ZCH (z-coordinate of head of normal vector) to an arbitrary value larger than 12. You can also use an arbitrary value for XHEV (x-coordinate of head of edge vector).

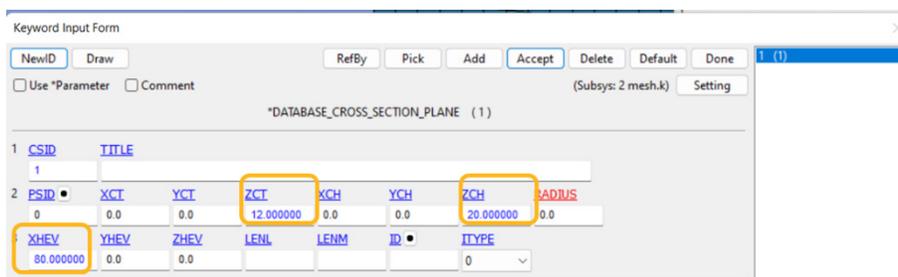


Figure 30: Defining a cross-section plane on which force data shall be recorded.

## 3. Model Checking and Ansys LS-Run tool<sup>4</sup>

### 3.1 Performing a Model Check to Find Errors

#### 3.1.1 Keyword Check

Open Model Check either from the top menu from Application, Model Checking, General Checking or by going to Keyword Manager and pressing Model Check. The Keyword Check finds problems in the syntax of the keyword deck which you have created. There are different types of issues which are

<sup>4</sup> Please refer to the [YouTube tutorial video](#) for a walkthrough of Part 3

highlighted. Errors are fatal and will result in an error termination if you try to run the model. Warnings are not as severe and will not result in error termination. They are instead an indication that something in the setup is potentially problematic (see Figure 31). If you are watching the video for this tutorial, you will see that there is an error found under one of the \*PART keywords. This was caused by neglecting to assign a material to one of the parts. This is something that will result in an error termination, and which must be fixed before running the model.

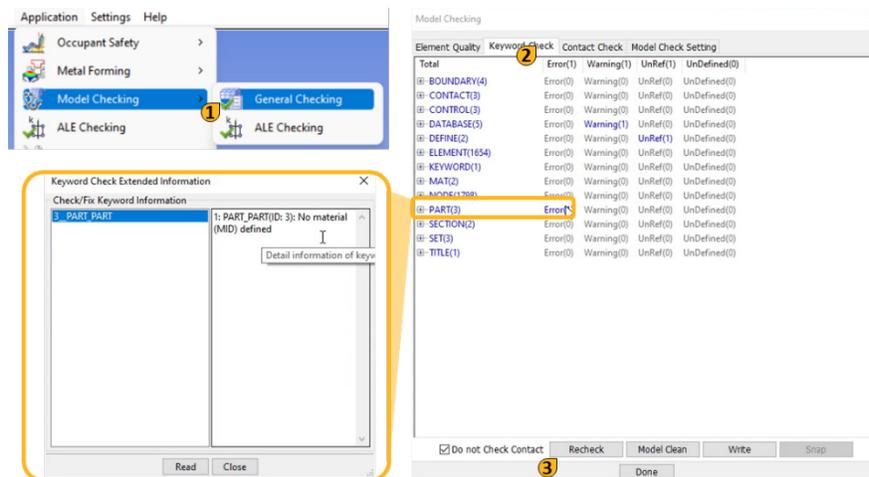


Figure 31: Checking the model, issues an error associated with a missing material definition.

If you have this error in your model, go to the \*PART keywords and assign the correct material to the part in question (see Figure 32), and then go back to the keyword checker and do a recheck to find that the error is gone.

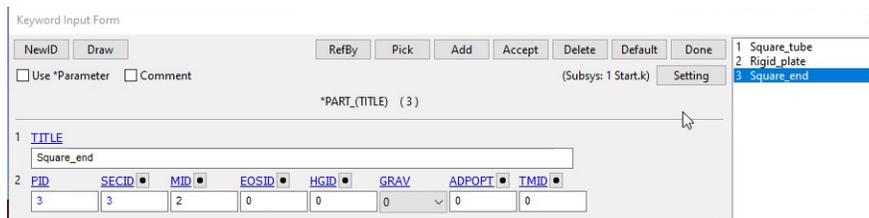


Figure 32: Assigning the square end part a material (MID = 2).

Moreover, you might have a warning regarding the previously defined cross-section plane stating that it has lengths L and M which are less than or equal to zero.

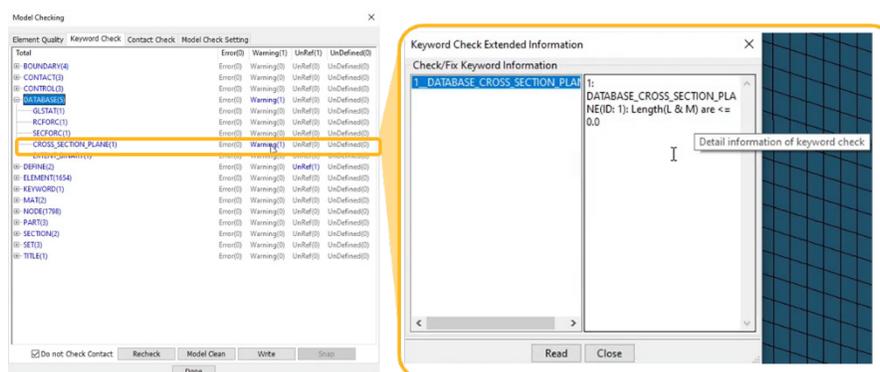


Figure 33: Warning message associated with the cross-section plane.

Go to the keyword in question and inspect these two options. Also read the documentation for this particular keyword in the Ansys LS-DYNA software manual ([ANSYS LS-DYNA Keyword Reference Manual](#)). When working with Ansys LS-DYNA software, in general, inputting a value of zero to an option, results in a default value being used. In this case, the default value for L (LENL) and M (LENM) is infinity. L and M dictate the width and height of the cross-section plane. In the case of this model, that is not a problem, since the cross-section will not include any parts other than the square tube. Imagine a scenario where there are other parts above, below or to the sides of the tube. In that case, this will be a problem, since the cross-section plane which is meant to give output data for the force in the tube will cut through other parts and collect data from them as well. In such a case, you would need to have a plane with finite lengths.

This highlights the difference between an error and a warning very well. An error is always a problem whereas a warning is usually only a problem under certain circumstances.

Furthermore, in the tutorial video there is also a message about a keyword being unreferenced (UnRef(1)). This happens when you have created a keyword that is not used anywhere, and this does not have to be a problem. It will not disturb a simulation in any way, but it could indicate that you have missed to use something.

### 3.1.2 Contact Check

Proceed by performing a contact check to verify that the contact definitions are set up correctly. First inspect the penalty-based contacts of which there are two, one between the tube and the rigid plate and one for the tube itself. Start with the former and verify that there are no initial penetrations between the two parts (see Figure 34). Also verify that there are no penetrations for the self-contact of the square tube.

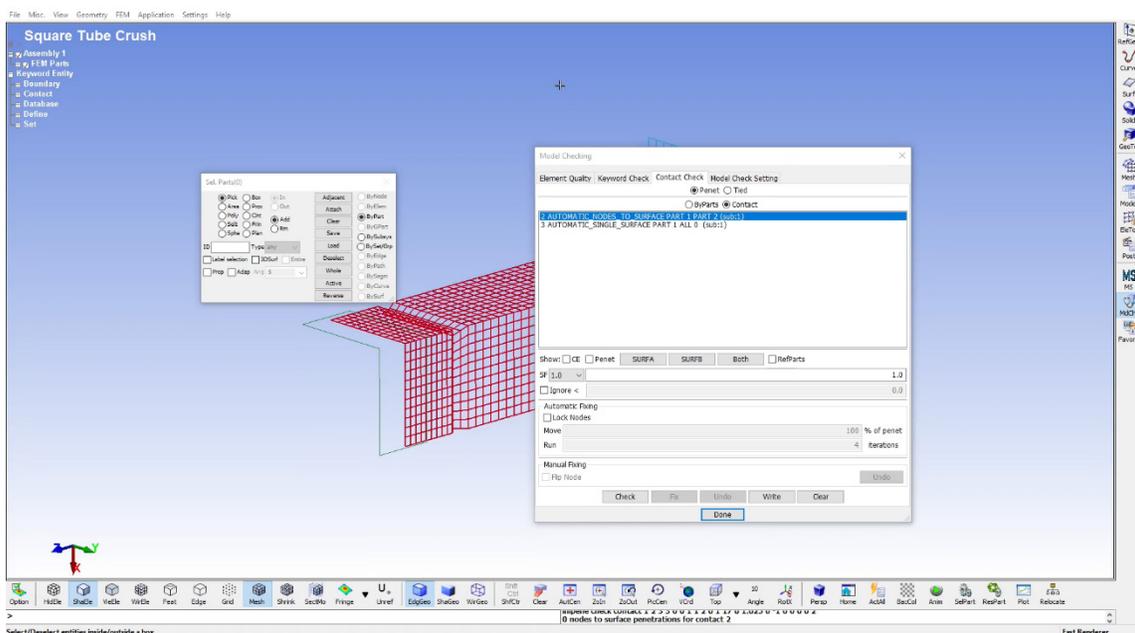


Figure 34: Contact definition check window for penetration.

Switch to checking tied contacts and check the box for tied nodes and do the check to verify that the nodes on the square tube have been tied properly to the end plate, as displayed in Figure 35.

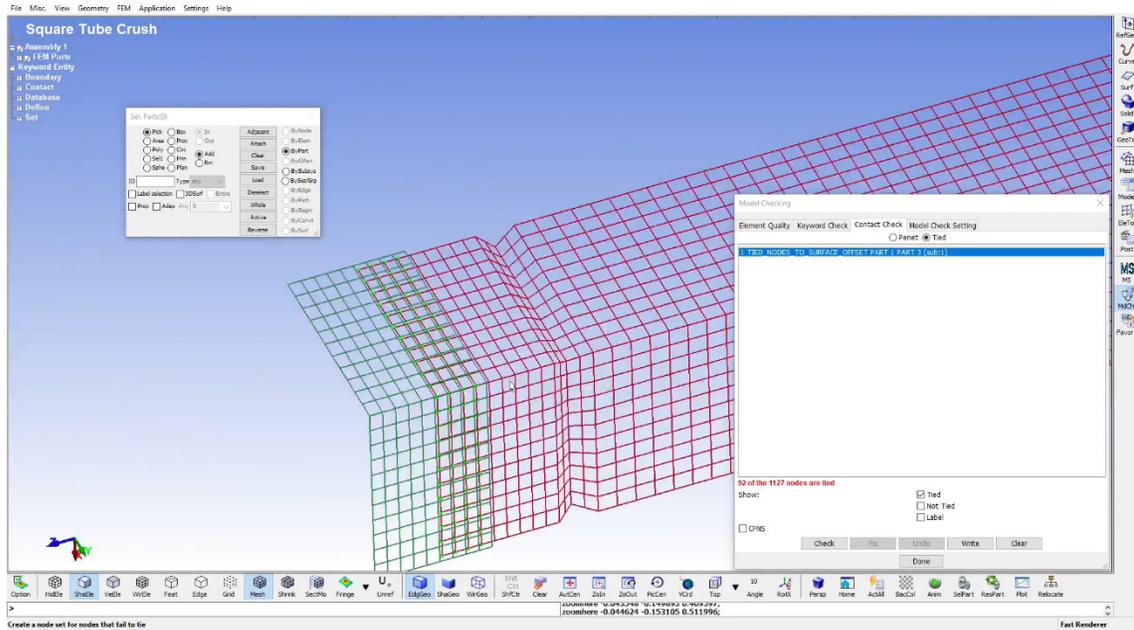


Figure 35: Contact definition check window for tied contacts between the tube and end plate.

## 3.2 Saving the Keyword File

Now that you have performed a model check, you can be confident that the model will run as intended when you submit your job. Save the keyword file with an appropriate name to a directory where you intend to run your simulation and note that each simulation requires a separate directory or folder. If you first run a simulation and then run another simulation in the same folder, the results of the first simulation will be overwritten.

## 3.3 Running the Simulation using LS-RUN

### 3.3.1 Specifying Input File

For the first field called *INPUT*, click the folder icon on the right-hand side and navigate to the keyword file which you just saved or load the pre-made file from the attached document of the resource.

### 3.3.2 Specifying Solver Version

Now, when setting the solver version, first set the preset. First off, single and double precision refers to how many significant digits Ansys LS-DYNA software allows a number. Single precision allows a floating-point number 4 bytes of memory whereas double precision allows it 8 bytes of memory. This means that when using double precision, floating point numbers will have twice as many significant digits as compared to using single precision. The choice between single or double precision obviously impacts numerical accuracy in the form of round-off errors, but it also affects how much memory your analysis requires. When using the implicit solver of Ansys LS-DYNA software, the user is forced to use double precision to ensure numerical accuracy. If you try to use a single precision version of Ansys LS-DYNA software for an implicit simulation, you will be met with an error termination. For explicit analyses, we will typically use single precision for the reduced computational cost. There are circumstances where you might want to use a single precision version even for explicit analyses, for instance if you have a very high need of accuracy or if you have a very long simulation where the cumulative error due to numerical round-off errors will become problematic after a time. Bear in mind that double precision is in general 30 % more computationally expensive than single precision.

There is also a difference between the SMP (Symmetrical Memory Processing) and the MPP (Massively Parallel Processing) variants of Ansys LS-DYNA software. A simplified explanation is that with SMP, you are limited to performing your simulation on a single computer. It can still be run on multiple cores, but all the cores have to be inside the same computer. Using MPP on the other hand, you can run simulations that are divided into several computers.

It is recommended to use MPP-variants since parallelization scales better for larger models and since most users use MPP, so the functionality will typically be further developed and tested, although the SMP and MPP variants should have more or less the same functionality. Set the preset to MPP single-precision, with MS-MPI and click the browse icon to the right of the solver field and navigate to the corresponding version of Ansys LS-DYNA software on your computer. It is a good idea to always try to use the latest version of Ansys LS-DYNA software since bug fixes and new functionality is continuously added to the program.

### 3.3.3 Specifying Number of CPUs and Memory

The number of CPUs or cores and memory is set in the top right corner of the Ansys LS-Run tool. By using more than one core, you utilize parallel computing to speed up your simulation. Ideally, you would want your simulation time to be cut in half by doubling the number of cores, but in reality, one is usually met by a diminishing returns behavior where the more cores one uses, the less simulation time each additional core saves you. To achieve the ideal scaling with parallelization, you can take it as a rule of thumb that there should be 10 000 elements in your model for every CPU that you run it on.

With the memory flag, you specify how much RAM (Random Access Memory) you allow the simulation. Remember that with single precision, a floating-point number is 4 bytes and in double precision 8 bytes. One floating-point number in your analysis is a word and with the memory flag, you specify the number of words. When specifying memory, we use the suffix M which stands for mega. Set the memory flag to for instance 20 M to allow the simulation 20 million words.

### 3.3.4 Submitting the Model for Analysis

With everything done, the only thing left is for you to submit the job by clicking on the play symbol on the left-hand side. When you do this, a command prompt will appear so that you can verify that the simulation starts running (see Figure 36).

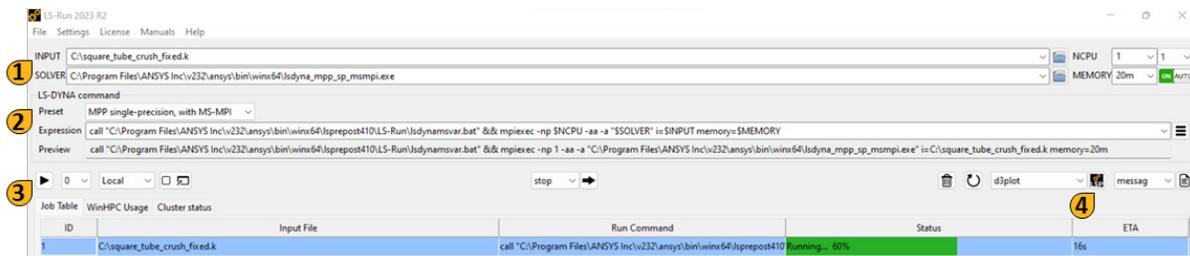


Figure 36: Steps for submitting the model for analysis.

## 4. Analyzing Results Using Ansys LS-PrePost tool<sup>5</sup>

### 4.1 Opening the Results

Open the results of the simulation either through Ansys LS-PrePost tool directly or through the Ansys LS-Run tool where there is a button that is a shortcut to opening the results (d3plot) with Ansys LS-PrePost tool (compare step 4 in Figure 36).

### 4.2 Animating the Results

#### 4.2.1 Animating the Quarter Model

When you open a d3plot file in Ansys LS-PrePost tool, the animation toolbar will appear automatically. It is also found in the bottom toolbar in the program. Click the play button to view the animation (see Figure 37). You can decrease the speed of the animation with the slider in the left corner. With the Animate checkbox ticked, you can drag the middle slider manually to go through the animation.

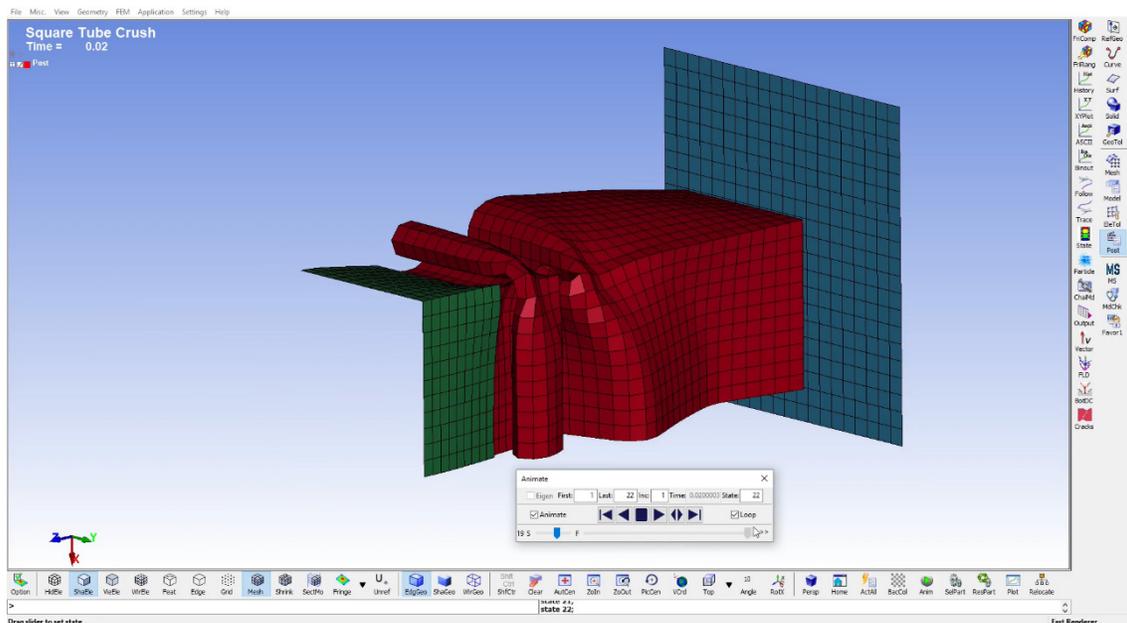


Figure 37: Animation of the tube quarter model being crushed via the Animation window.

#### 4.2.2 Reflecting the Model to Animate the Whole Geometry.

Since we only modeled a quarter of the tube's geometry to make use of symmetries, we are only seeing a portion of the full geometry. That can be changed by reflecting the model about its symmetry planes. Do this by using the Reflect Model tool found under Model (see Figure 38). Reflect the model about the YZ-plane as well as the XZ-plane to see the full geometry of the tube. Play the animation again.

<sup>5</sup> Please refer to the [YouTube tutorial video](#) for a walkthrough of Part 4

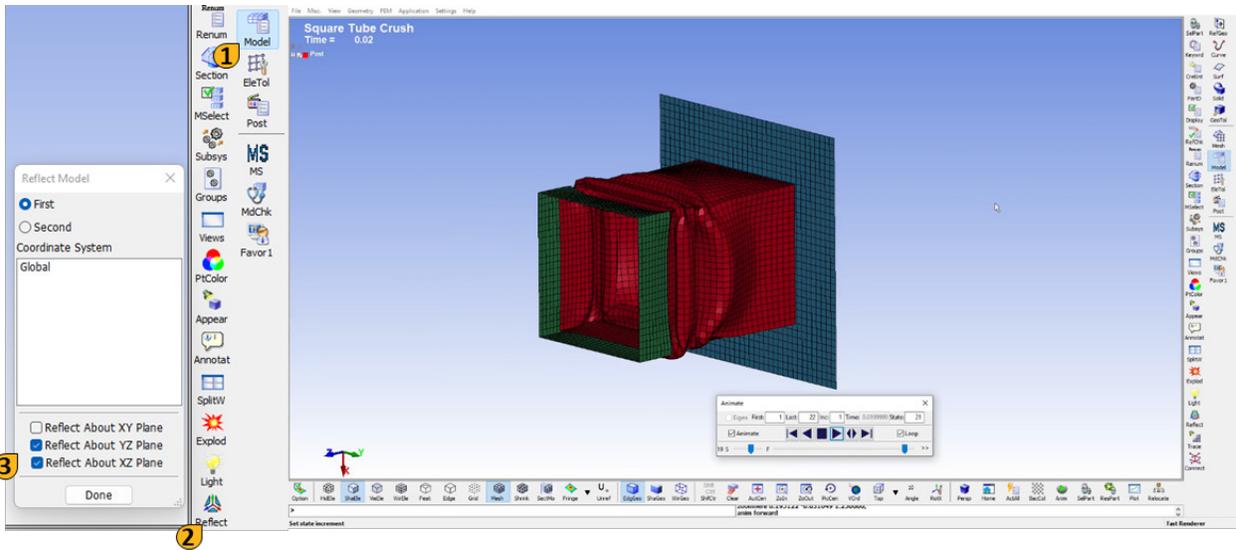


Figure 38: Applying the Reflect option to visualize the entire model instead only a quarter of it.

## 4.3 Fringe Plotting

### 4.3.1 Fringe Plotting von Mises Stress

The tool called Fringe Component lets you visualize results in a nice way (see Figure 39). Locate it under Post. In the left menu are categories of results such as stress, strain and miscellaneous. Locate and select von Mises stress and click apply. The color bar on the right-hand side will be updated. Play the animation once more.

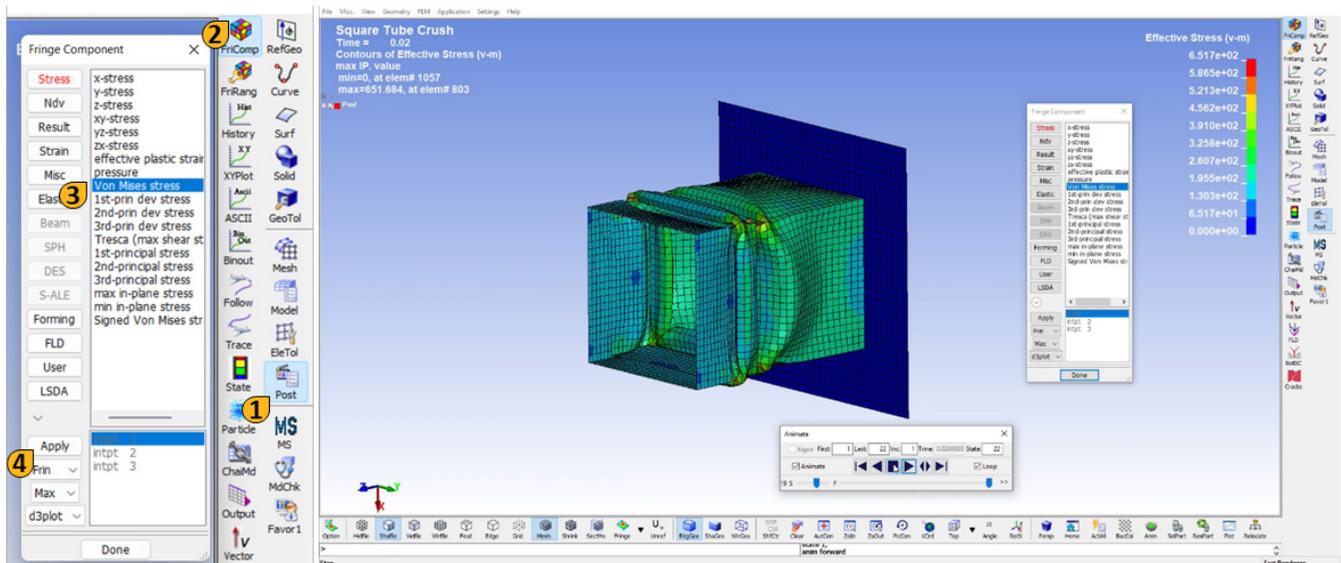


Figure 39: Visualizing the von Mises stress distribution in the tube by using the post-processing fringe component.

### 4.3.2 Fringe Plotting Effective Plastic Strain

Also visualize effective plastic strain which you previously requested as output with the keyword called \*DATABASE\_EXTENT\_BINARY. Note that effective plastic strain is in the stress category, as shown in Figure 40.

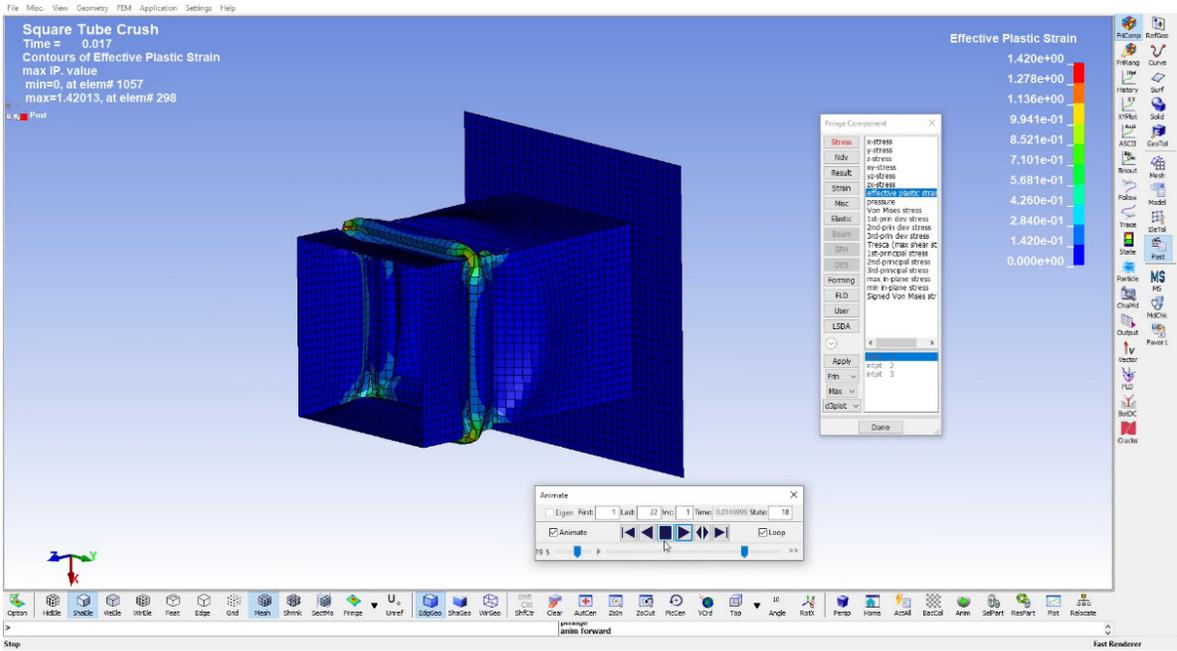


Figure 40: Effective plastic strain distribution.

### 4.3.3 Fringe Plotting Z-Displacement

Visualize z-displacement and play the animation, using the Ndv (Nodal Displacement/Velocity Contour) icon in the Fringe Component Window. Look at the color bar and notice that the values change with every state. This is because the color bar or the fringe bar, takes the minimum and maximum values of each state as the minimum and maximum values for the scale. This can make comparisons between states a bit difficult. Open the tool called Fringe Range (see Figure 41). By default, the behavior is set to dynamic, meaning that the color bar updates with each frame. Instead, set it to user defined and set the minimum value to -80 which is the prescribed motion of the tube. Set the maximum value to 0 and press update. Play the animation again and notice the difference.

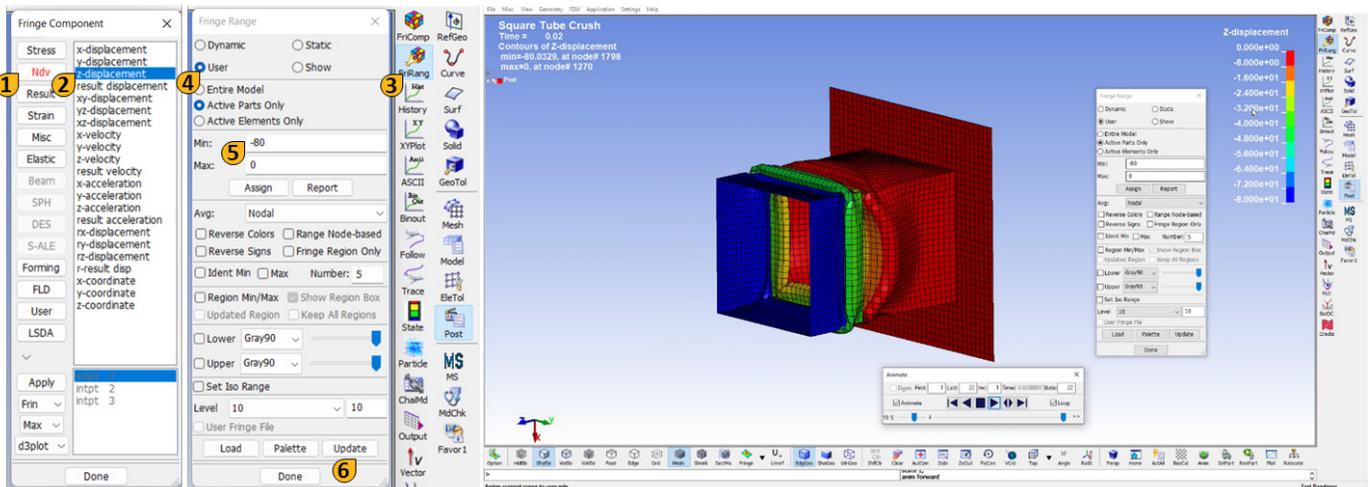


Figure 41: Fringe Range tool for keeping the state variable constant throughout.

## 4.4 The History Plot Tool

History Plot lets you plot results on the global level, part level, elemental level or the nodal level. Select the elemental level and click on x-stress. Then pick an arbitrary element on the tube with the left mouse button and click plot (see Figure 42).

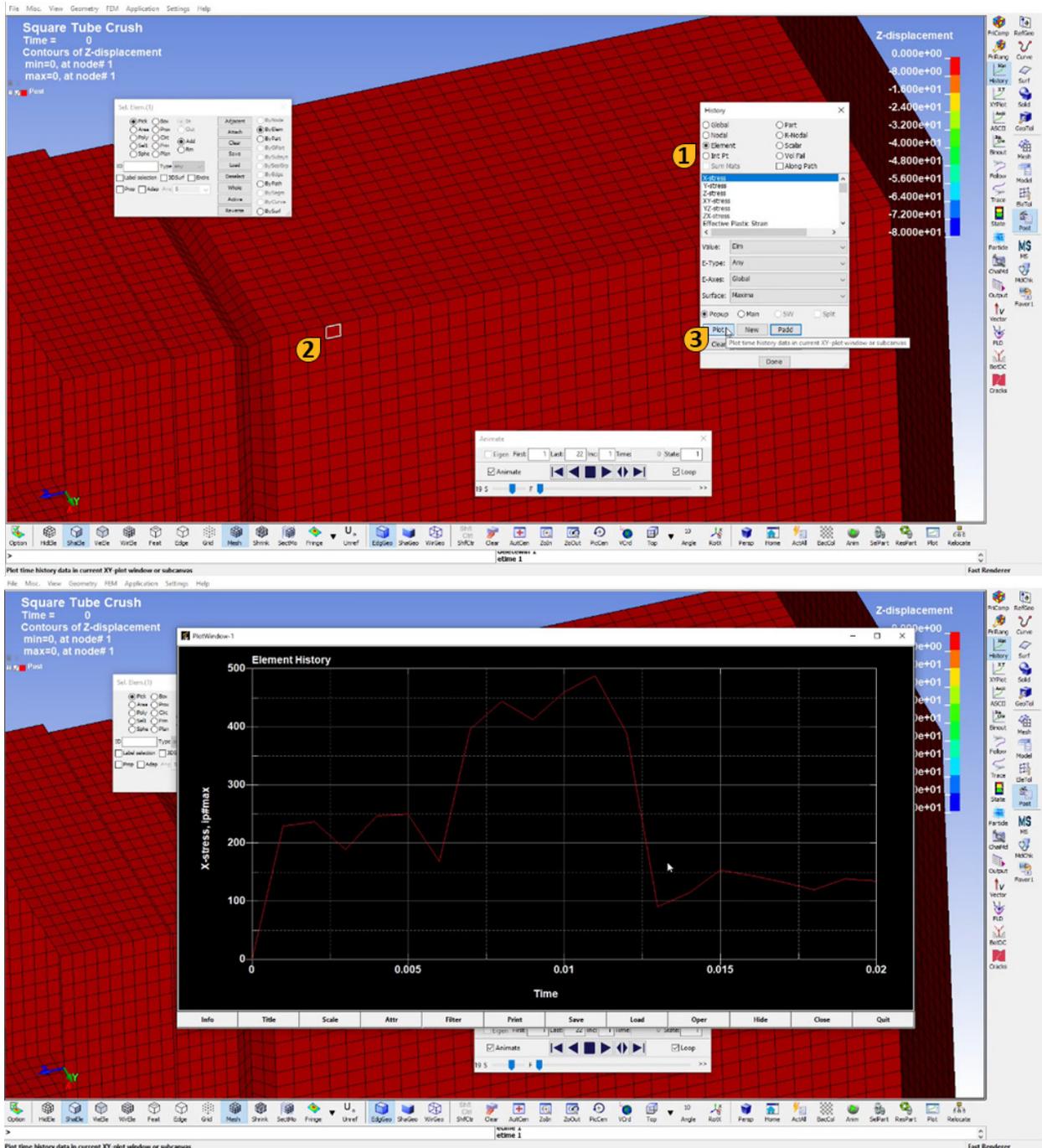


Figure 42: Plotting the x-stress in an arbitrary element of the tube.

You can add other components to an already existing plot using the plot add function. Minimize the plot window but do not close it, and then select the y- and z-stresses and click *Padd* to add them to the pot, as shown in Figure 44. You can also plot multiple things in the same plot by selecting them all from the beginning and clicking plot.

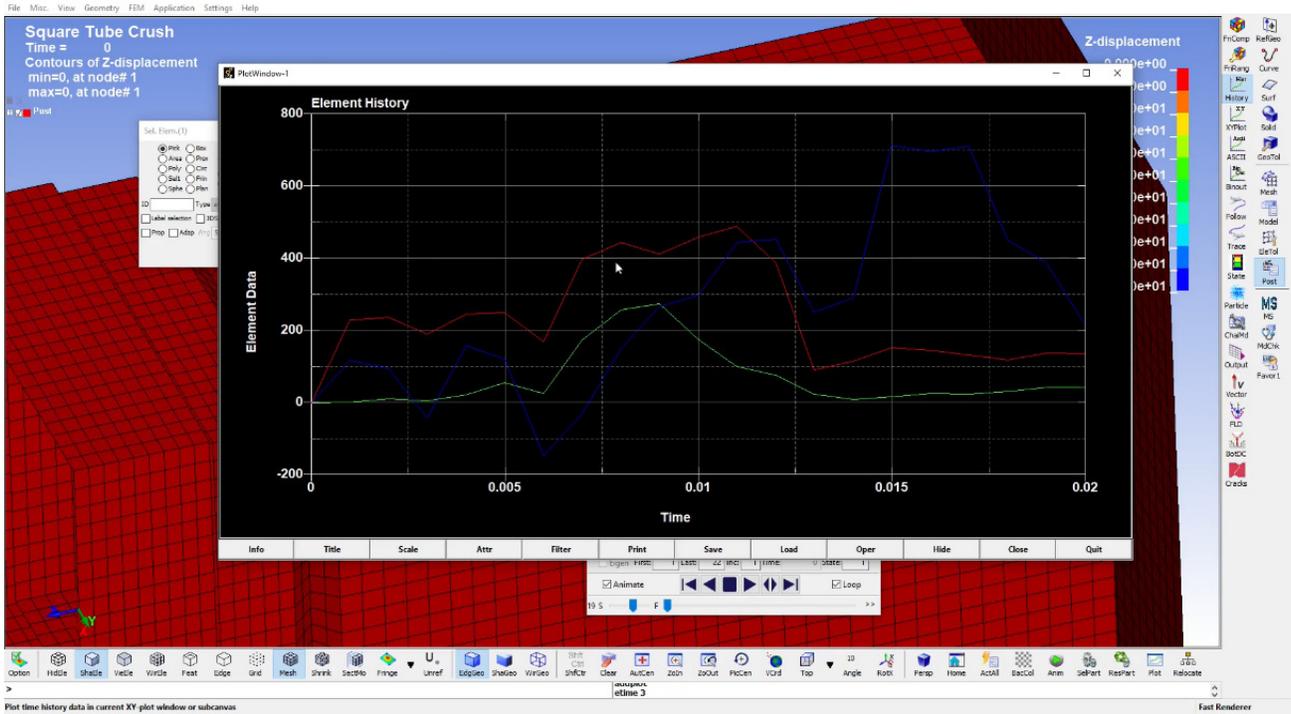


Figure 44: Overlay of x,y and z-stresses of the selected element.

## 4.5 Creating Cross-Plots with the XY-Plot Tool

### 4.5.1 Using History Plot to Save Data

To be able to create a cross-plot, you must first create the data for it. Let us create a plot of effective stress as a function of effective plastic strain. Go to History Plot and select an element on the square tube. Plot effective plastic strain and save the data to an appropriately named file (see Figure 45). Repeat the procedure to get the effective stress in the same element.

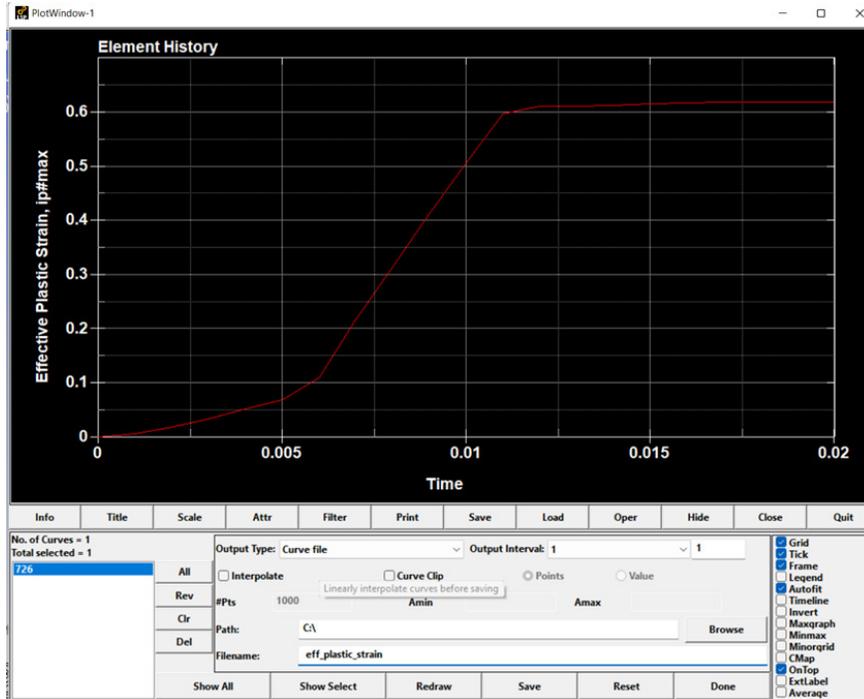


Figure 45: Saving the effective plastic strain plot of the selected element

#### 4.5.2 Creating the Cross-Plot

Select XYPlot and notice that the two files you just created show up automatically. If you want to add data from another file, simply click the add button and select the file. Select cross and first click on the file with the data for the x-axis and click on it once more where it appears under curve names. Do the same thing for the data you want to have on the y-axis (see Figure 46).

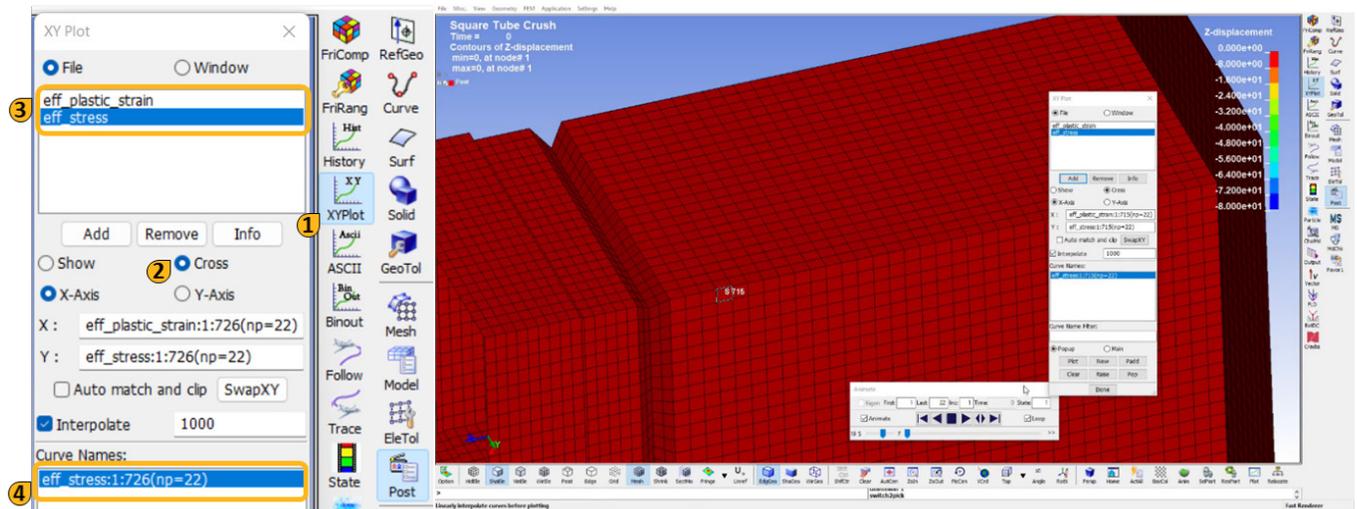


Figure 46: Creating a cross-plot from the effective plastic strain and effective stress recorded at the selected element.

Click on plot to view effective stress as a function of effective plastic strain. Notice that since the material model used for the square tube defines work-hardening by a tangent modulus, the element will exhibit linear hardening (see Figure 47).

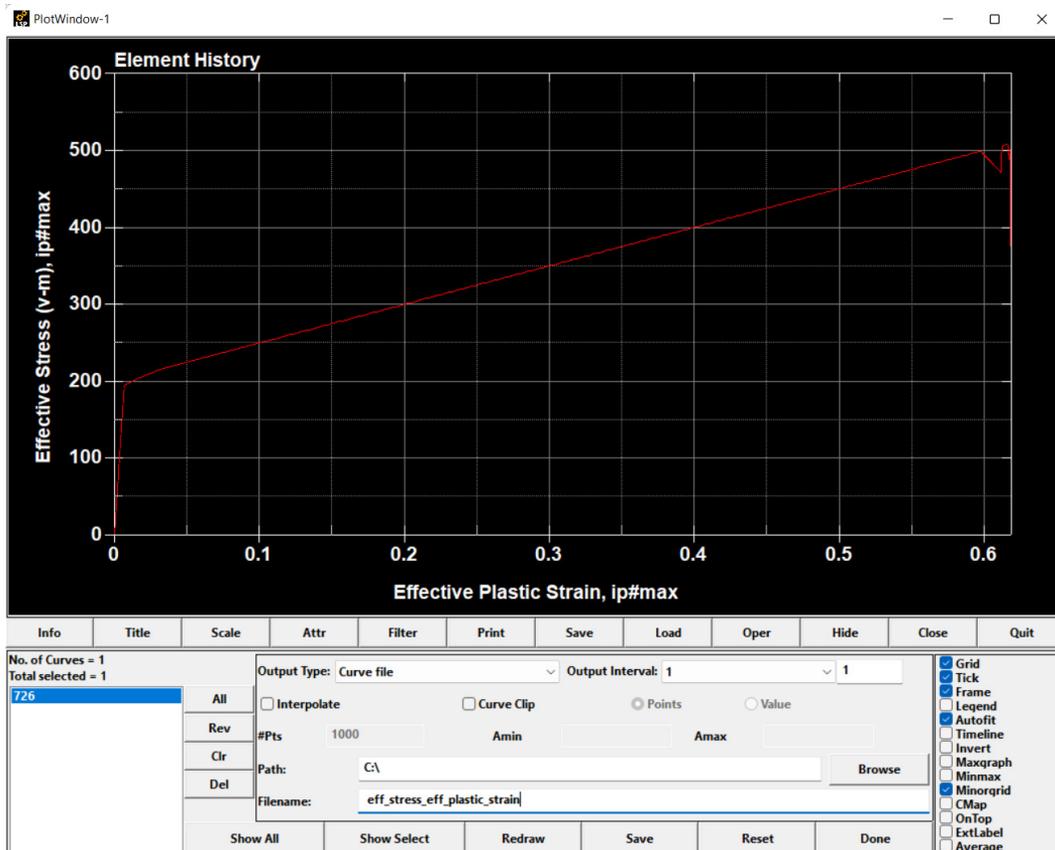


Figure 47: Effective plastic strain vs effective stress plot, exhibiting work-hardening.

## 4.6 Reading Binary Output Data from Binout

There are two remaining plot functions in Ansys LS-PrePost tool, which we have not yet covered, ASCII Plot and BINOUT Plot. Without going into detail, ASCII is an abbreviation of American Standard Code for information interchange and is a system for representing letters and other characters in a computer. ASCII-data is in human-readable form, meaning that you can open it in a text editor and inspect its contents. If you do the same thing with binary data, it will be unintelligible to you. The binary format allows for faster information processing in the computer which means that it will take less time for Ansys LS-DYNA software to write the output as compared to the ASCII format. Another advantage of the binary format is that it requires less memory to store the same amount of information.

Since the model in this example was run with an MPP-version of Ansys LS-DYNA software, the output will be in the binary format. Had it been run with an SMP-version instead, it would be in the ASCII format. Therefore, use the Binout tool instead of the ASCII tool. The two tools contain the same functionality. Begin by loading the binary output file which is in the directory the simulation was run in and is called binout0000 (see Figure 48). Select the file from the list on the right-hand side and notice that the results for global statistics, resultant contact forces and cross-section forces which you requested when setting up the model have become available.

### 4.6.1 Plotting Model Energies from Global Statistics

Plot internal energy, kinetic energy as well as total energy from glstat. Add a legend to your plot by clicking on the title button and checking the box for legend (see Figure 48).

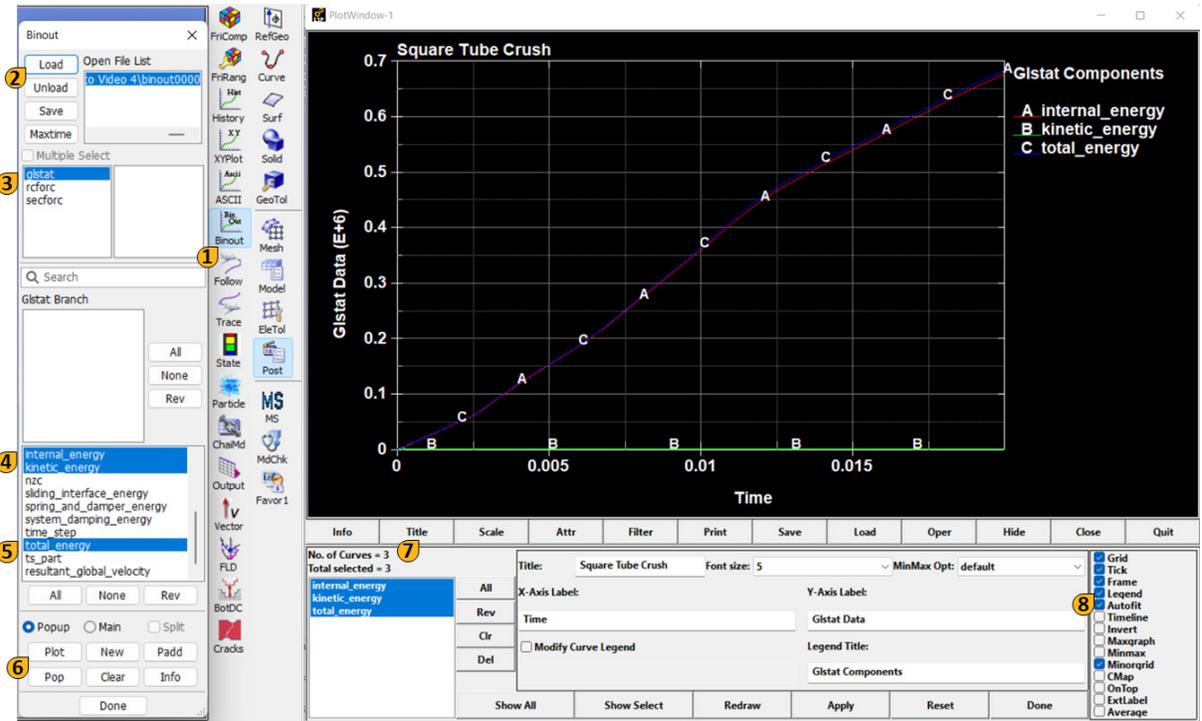


Figure 48: Plotting some internal, kinetic and total energy from the global statistics.

#### 4.6.2 Plotting the Critical Time Step from Global Statistics

Repeat the previous process but this time plot the critical time-step as a function of time to see how it decreases because of deformation as the simulation progresses, as shown in Figure 49

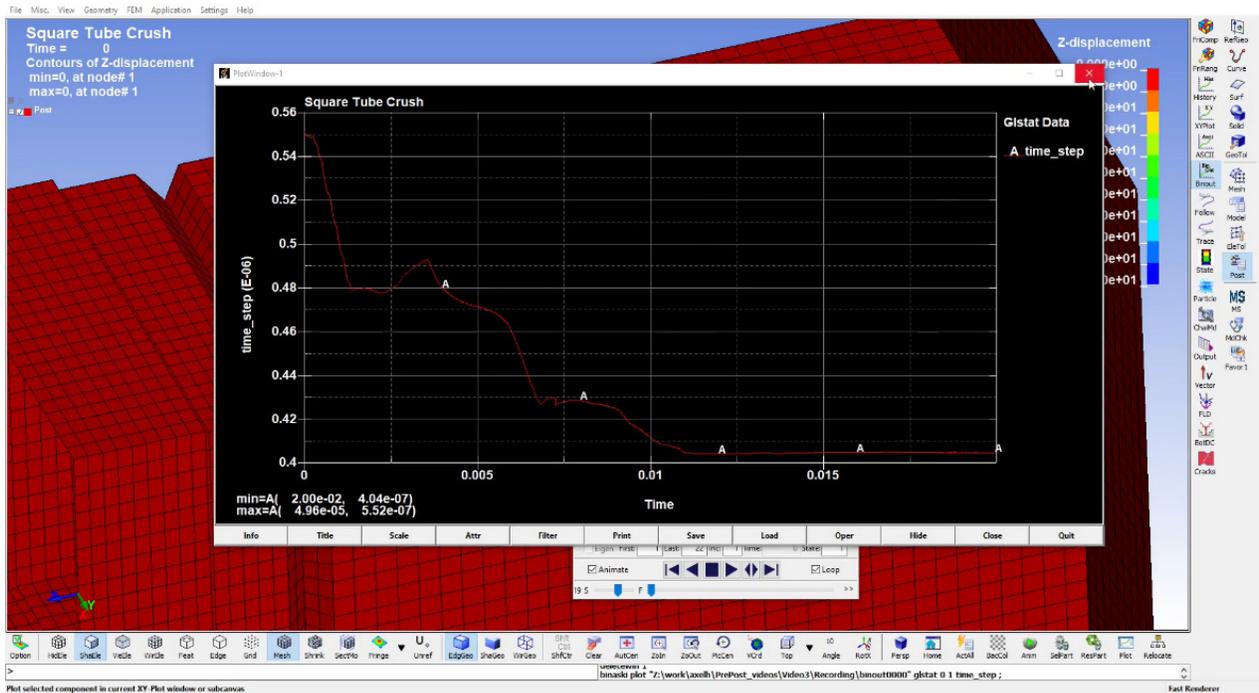


Figure 49: Time-step plot.

### 4.6.3 Plotting the Contact Force on the Tube

Inspect the resultant force experienced by the tube in the contact between the tube and the rigid plate. Do this by choosing rforc and selecting S-1, with S referring to Surface A and the number 1 to the id of the contact keyword. The resulting plot is displayed in Figure 50

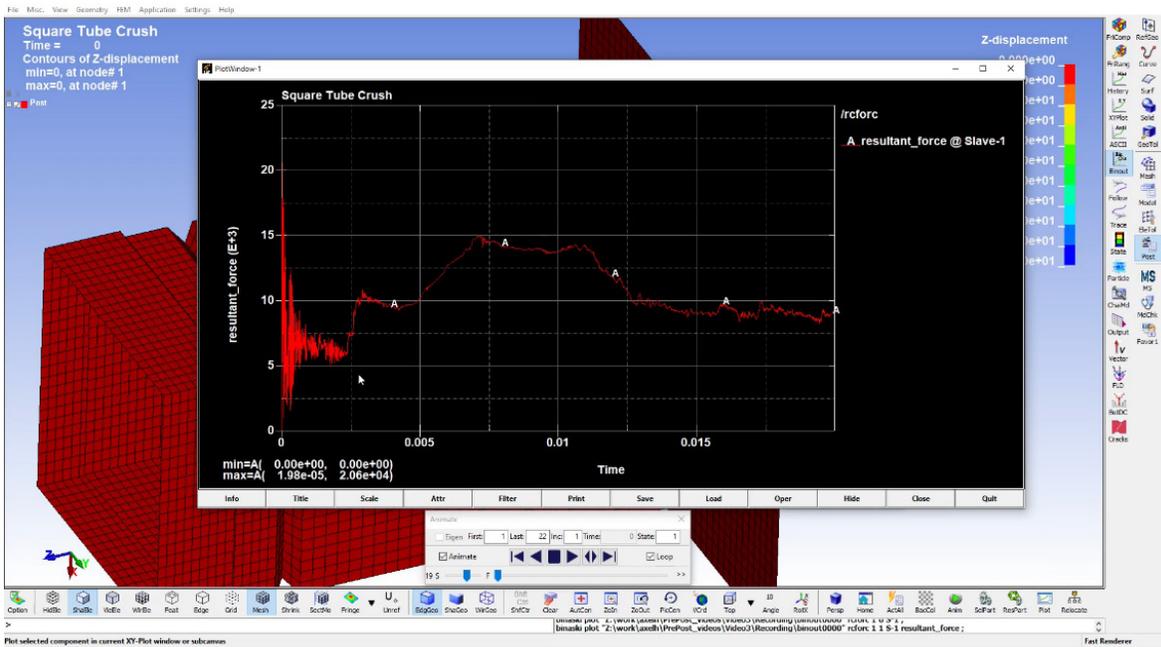


Figure 50: Plotting the resultant contact force on the tube.

### 4.6.4 Plotting the Cross-Section Force

Lastly, let us look at some of the data collected by the cross-section plane which we defined during the model setup. Its data is contained within secforc. Begin by plotting the total force, as shown in Figure 51

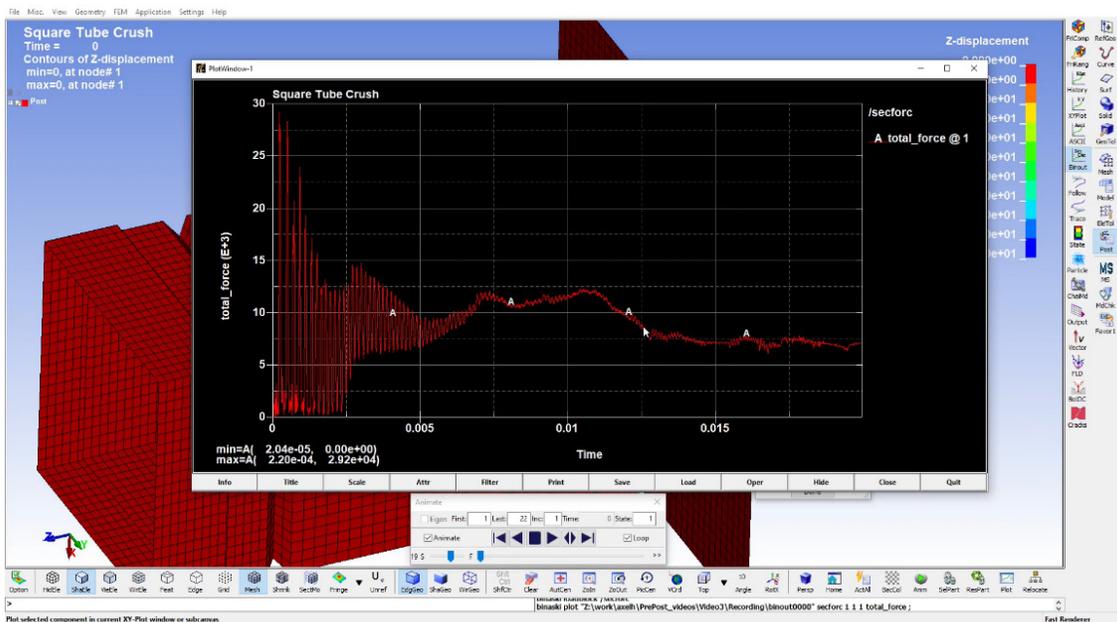


Figure 51: Plot of the cross-section forces.

You will probably notice that the data is quite oscillative. It is possible to apply numerical filters to your data. Locate the filters by clicking on the filter button. Let us apply a numerical filter for the sake of demonstration. The values we will use are quite arbitrary. Set the filter type to *bw* as in Butterworth and the frequency to 1000 Hz. Click apply and notice the difference (compare Figure 52).

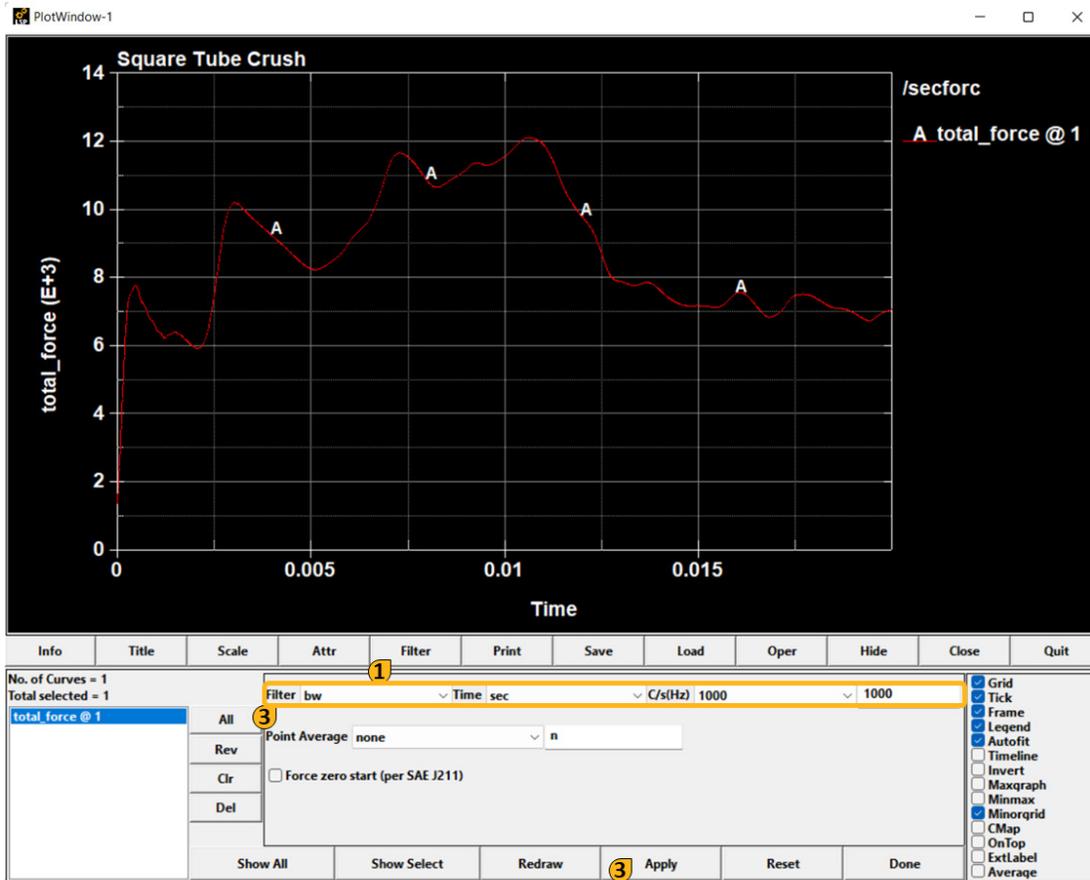


Figure 52: Applying filters to smoothen/clean oscillating data.

The current plot shows force. Let us convert it into stress instead. Minimize the plot window, select area and click *Padd* to add a plot of the cross-section's area as a function of time to the already existing plot, as shown in Figure 53. Stress is of course force divided by area. Perform this operation by going to Oper and select divide curves from the list. Select the force curve as curve1 so that it is in the numerator, and similarly select the area curve as curve2 to place it in the denominator. Click apply and notice that the curve now instead shows stress as a function of time. Also notice that because of the unit system we have used, the stress is in MPa.

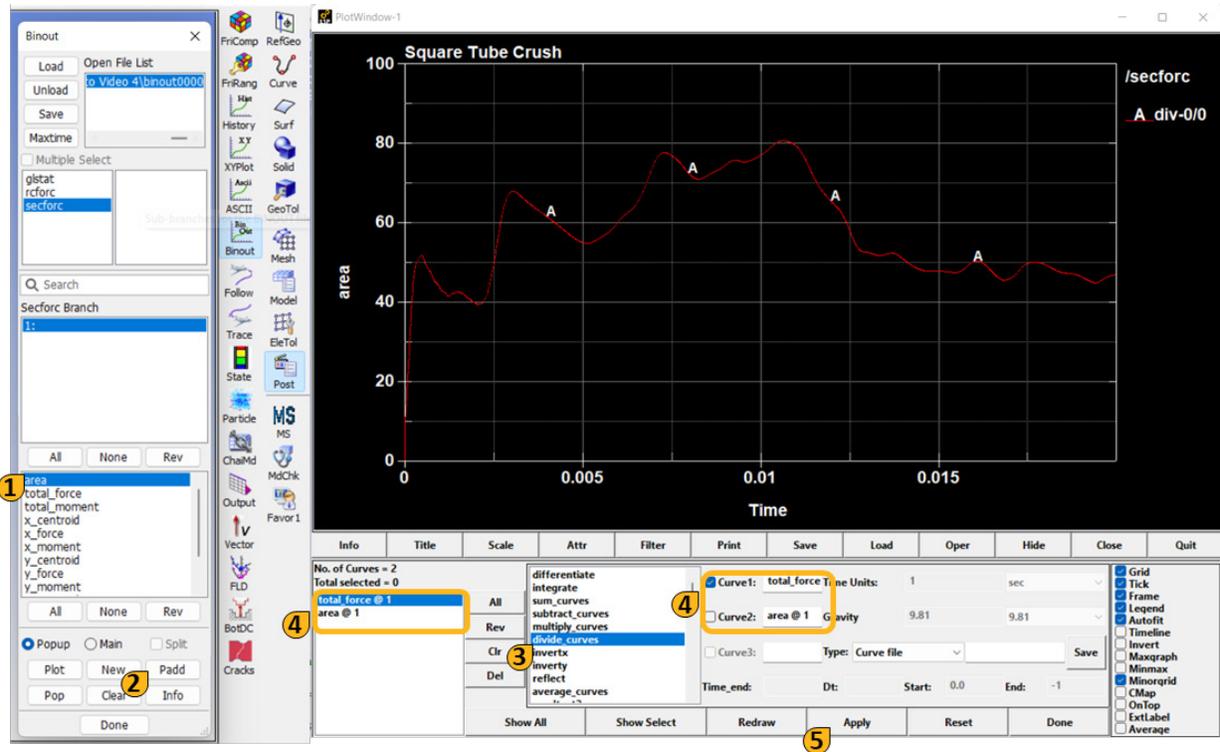


Figure 53: Plotting stress in Megapascal by combing force and area data.

## 5. Concluding Remarks

This marks the end of the tutorial. Good job on making it all the way here! To continue learning about Ansys LS-DYNA software, please visit [lsdyna.ansys.com](http://lsdyna.ansys.com) where you can find useful information, example models as well as conference papers presented at conferences.

Good luck on your continued learning!

© 2025 ANSYS, Inc. All rights reserved.

## Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

## Document Information

This case study is part of a set of teaching resources to help introduce students to topics focused on structures and structural simulations.

## Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, MicroProjects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

## Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2025 ANSYS, Inc. All Rights Reserved.