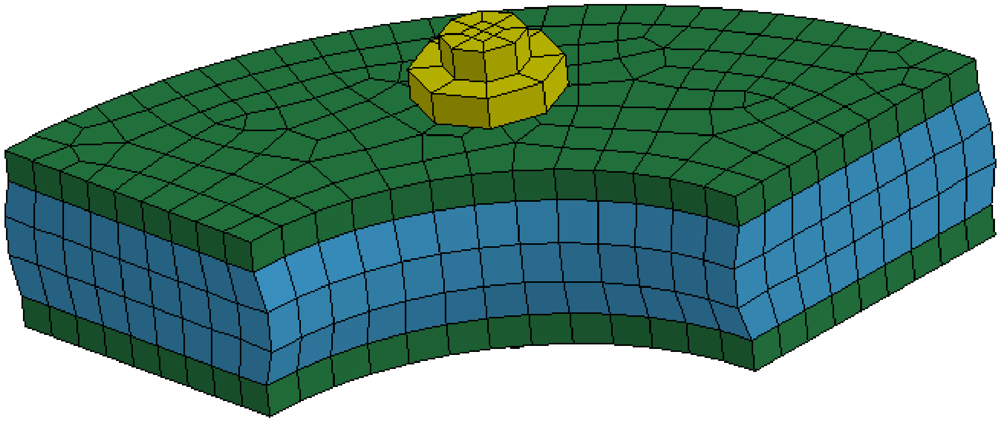
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 6. Mesh tool and dynamic relaxation



Contents

1 Introduction

1.1 Purpose

- Learn how to use the geometry and mesh tools in LS-PrePost.
- Get familiar with dynamic relaxation in LS-DYNA.

1.2 Prerequisites

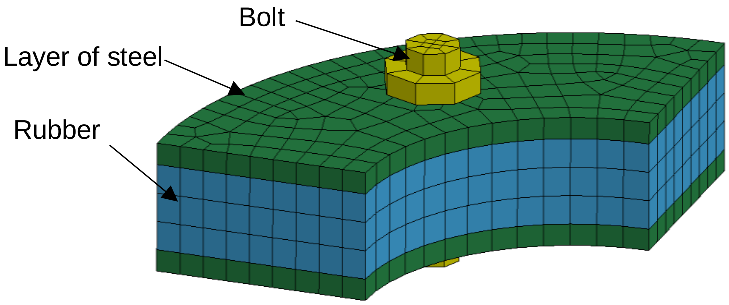
- Basic knowledge in the finite element method.
- Understand the steps in tutorial **1 - Getting Started.**

1.3 Problem Description

The task is to preload a bolt to compress a quarter of a disk using dynamic relaxation.

The tutorial consists of two parts

1. Create the geometry and mesh of the model.
2. Apply all keywords to perform the dynamic relaxation analysis.



2 Dynamic relaxation

It can sometimes be important to induce a steady state preload (gravity, stress in bolts etc.) before performing a transient dynamic analysis. This is usually done by dynamic relaxation in LS-DYNA, which is a transient analysis that takes place in pseudo-time (t<0). Each time step, the computed nodal velocities are reduced be a factor, so the solution undergoes a form of damping during the dynamic relaxation. When steady state is reached, i.e. the convergence factor has become sufficiently small, the transient analysis starts (t>0).

2.1 Data files

- Keyword file containing the mesh: **prestress_geometry.k**.
- The solution to the exercise: **prestress_results.k**.

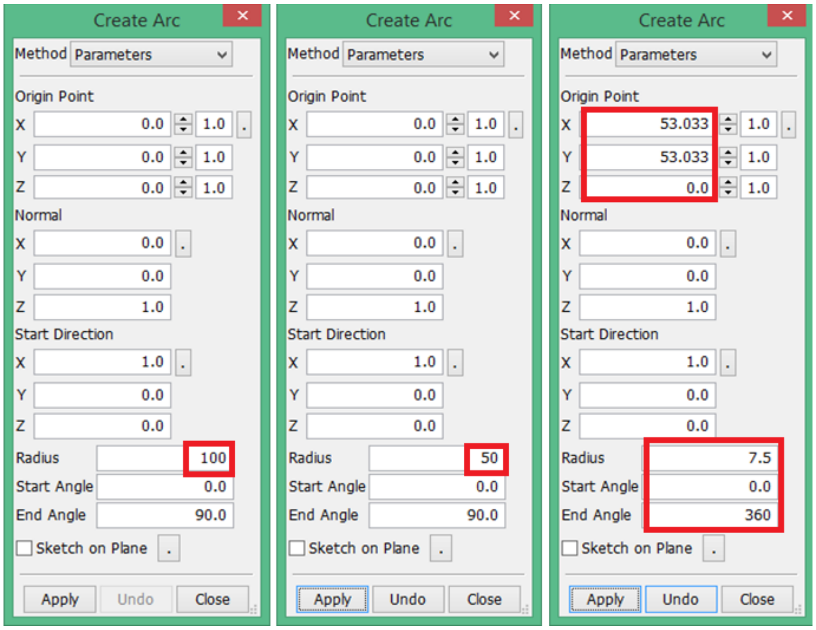
3 Part A - Create the mesh

In this first part of the tutorial, the geometry and mesh for the model in the picture will be created. If you want to skip this part, you can go directly to Part B (Section 4) and open the file prestress_geometry.k, containing the outcome of Part A, i.e. the mesh.

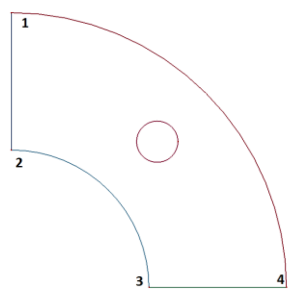
3.1.1 Create the geometry

Do as follows to create the geometry:

- Click **Curve > CirArc**. Create the three curves as in the images below:

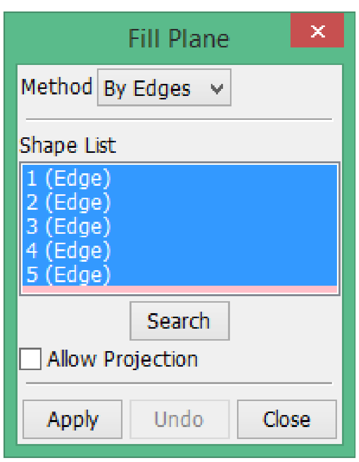


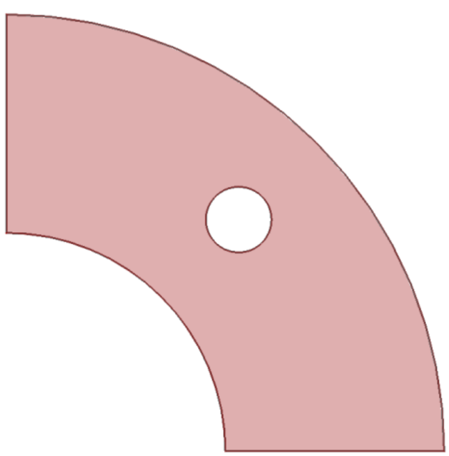
- Click **AutCen** in the bottom menu to auto center the lines.
- Click **Curve > Line**, change method to **Point/Point**.
- Click on point **1** and **2**, then **Apply**.
- Click on point **3** and **4**, then **Apply**.



Create the surface:

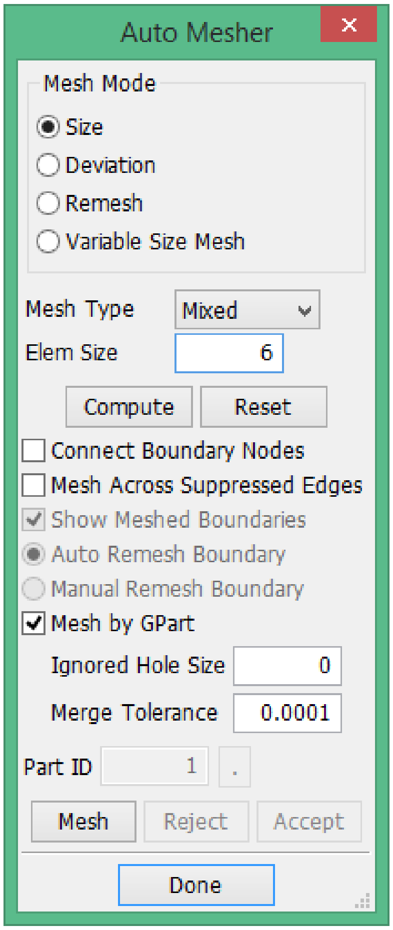
- Click **Surf > FillPln** and set method to **By Edges**.
- Click on all curves, then **Apply**. The result is shown below.



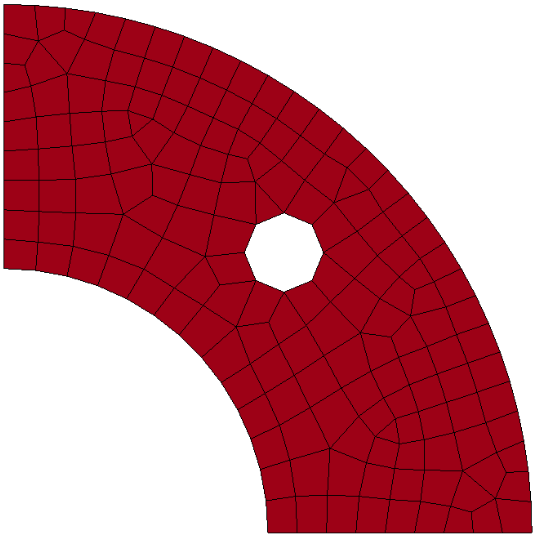


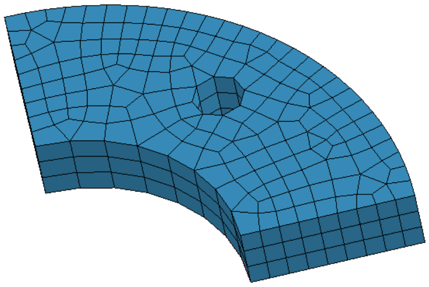
3.1.2 Create the mesh

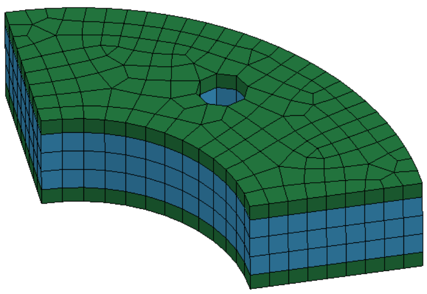
Now create the mesh as follows:



- Click **Mesh > AutoM**.
- Select **Size** as **Mesh Mode**.
- Select the surface, set the **Mesh Type** to **Mixed** and **Element Size** to **6**.
- Click **Mesh**. You can click on the red numbers if you want to change the mesh density on that specific edge. If you are satisfied with the mesh, click **Accept**.
- To hide the geometry (not the mesh), deselect **EdgGeo** in the bottom menu.

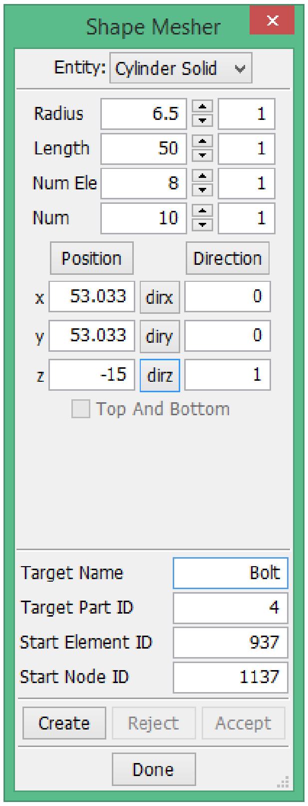






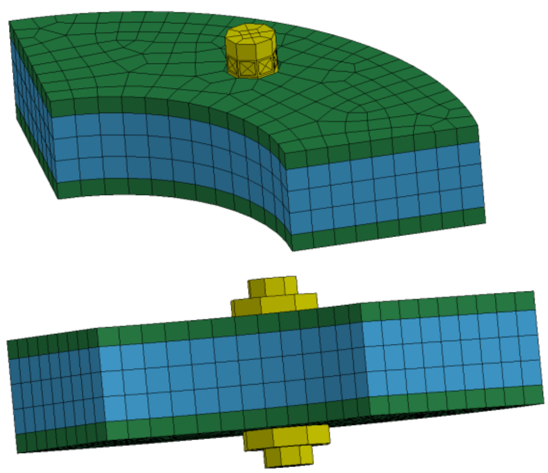
- Click **Mesh > EleGen**.
- Select **Solid**, and set **Solid By** to **Solid_Face_Drag**.
- Set **Thickness = 20**, **Segment = 3** and **Direction** to **Z** by clicking on **Z**.
- Select **Whole** in the Segment selection dialog.
- Click **Create** and **Accept.**
- While still in the Element Generation dialog, change **Thickness** to **5** and **Segment** to **1**.
- Activate **Prop** in the Segment selection dialog and click on an element at the top of the solid to activate all top segments.
- Click **Create** and **Accept**.
- While still in the Element Generation dialog, set **Part ID** to **3** and select the segments of the red part (or at the bottom of the blue part).
- Change the direction to **0, 0, -1**.
- Click **Create, Accept** and **Done**.

We will not need the shell mesh any longer, so delete **Part 1** in the Keyword Manager.



- Click **Mesh > ShapeM**.
- Set **Entity** to **Cylinder Solid.**
- Enter the values as in the figure. Do not change **Start Element ID** or **Start Node ID**, they will be set automatically and may differ from the figure.
- Click **Create**, **Accept** and **Done**. Note that your cylinder may have become green instead of yellow as in the figures. That doesn't matter, just check so that the cylinder is a different part compared to the outer layers of the disc (using **Model > SelPart** for example).
- To create the nuts, click **Mesh > EleGen.** Set **Part ID** to **4.** Change **Solid By** to **Solid_Face_Offset. Thickness = 5 and Segment = 1.** Select the 16 segments (both over and under the disc) as in the figure to create the nuts.
- Click **Create, Accept** and **Done**.

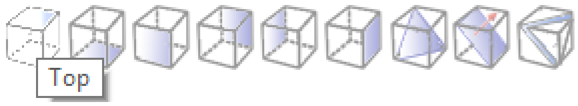
The mesh is now completed, save the keyword file, which is to be used in Part B.



4 Part B – Set up the analysis given the mesh

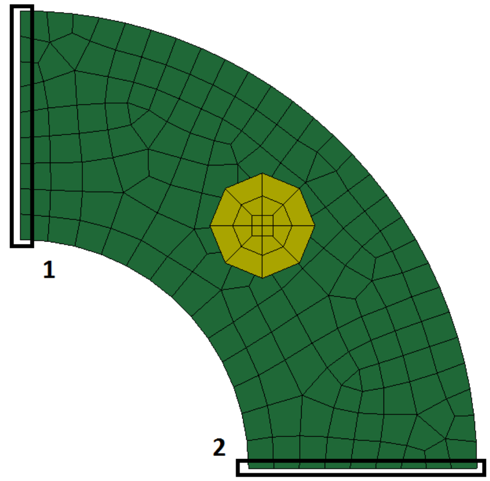
4.1 Boundary conditions

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



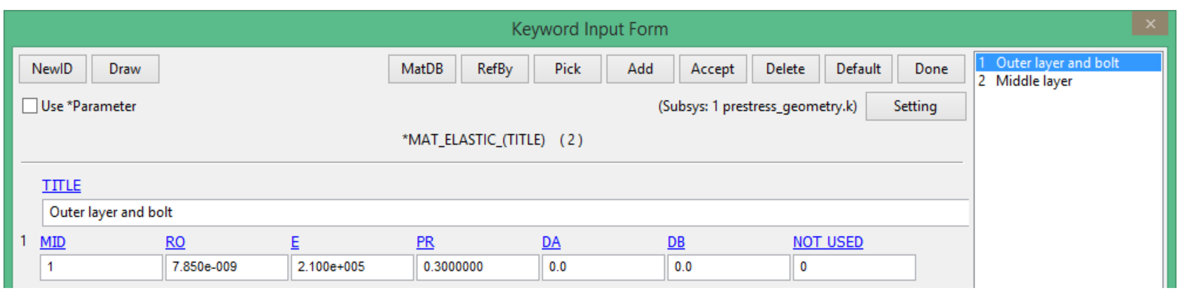
Create the constraints in the figure with **Model > CreEnt**, using the following symmetry planes **(**in **SPC_SET)**:

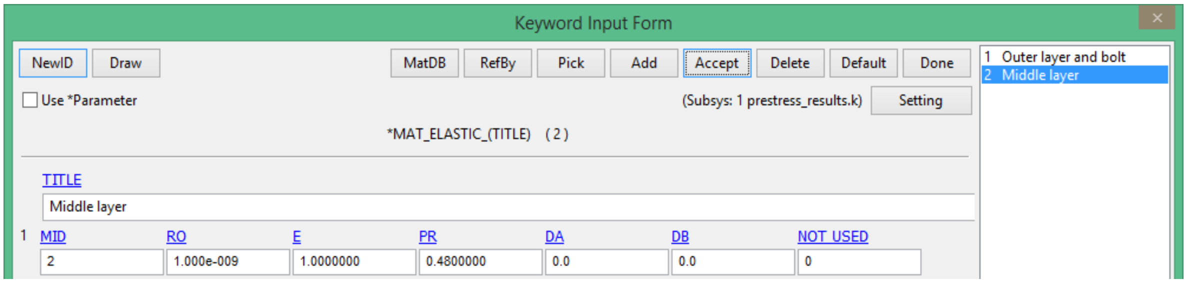
1. YOZ
2. XOZ



4.2 Material properties

Create two materials of type **MAT_ELASTIC**. One steel material for the bolt and outer layer, another soft rubber material for the middle layer.





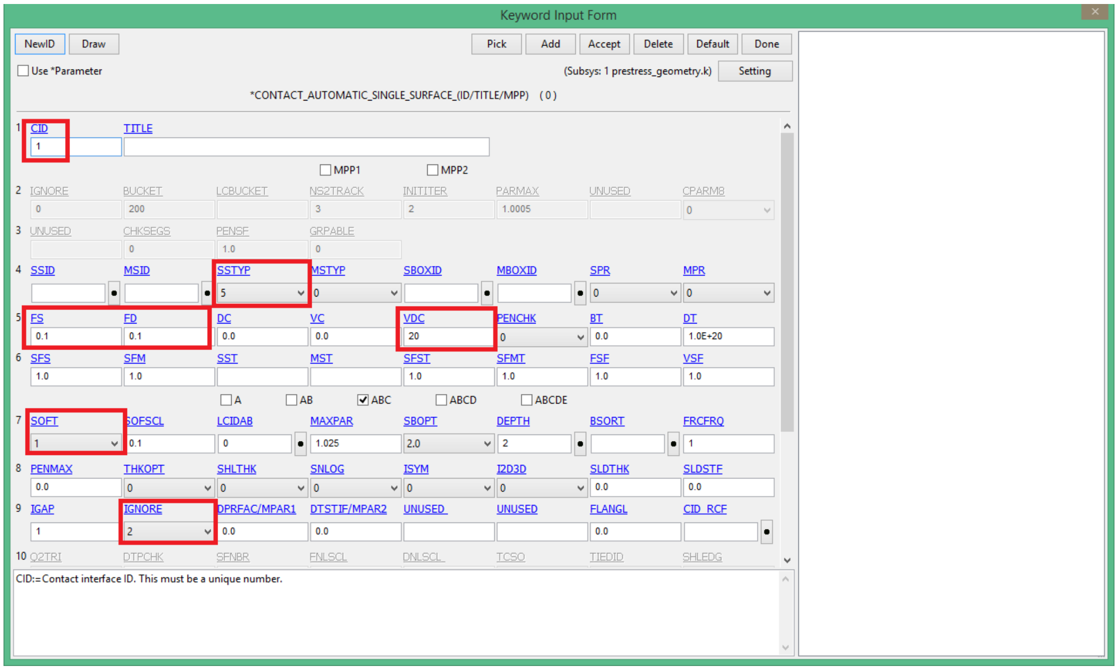
4.3 Element properties

Create a **SECTION_SOLID** using **ELFORM = -2**. Apply the first material to the bolt (part 4) and the outer layers (part 3). Apply the second material to the middle layer (part 2).

Apply the same section/element formulation to all parts.

4.4 Contact definition

Add an **AUTOMATIC_SINGLE_SURFACE** contact with the following options.



4.5 Preload bolt

To preload a bolt to a prescribed stress value, use the keyword **INITIAL_STRESS_SECTION**. As input we need:

1. A ramped curve defining the stress versus time (to get the size of the stress).
2. Cross-section definition.
3. Part set. This set together with the cross-section will define on which solid elements the stress will be applied.

Do as follows:

1. **CURVE**
   Click **Application > Tools > CurveGen** in the top menu.
   Change Method to **X – Y**.
   Make sure that **Smooth** is activated and has a value of **50**.

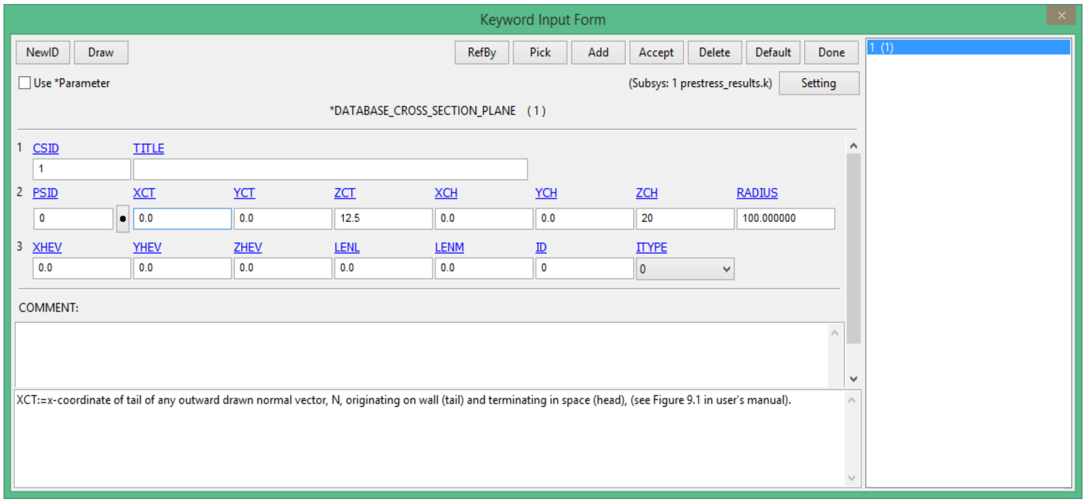
   Enter the values as in the table and then press **Create**.

X	Y
0	0
1e-4	10
1	10

   Open your newly created curve in the Keyword Manager (**DEFINE_CURVE**). Change **SIDR** to **1**, this will activate a dynamic relaxation analysis.

2. **CROSS-SECTION**

   Open the keyword **DATABASE_CROSS_SECTION_PLANE**. Define the coordinates of the tail and head of the normal vector to the plane (**XCT** to **ZCH**), see figure. You can either define the plane as finite rectangular, infinite or as a circle. We will use the circle option by defining **RADIUS = 100**.



   Normally, this keyword is used to define a plane where one wants to get section forces (output to file secforc). One can also set a Part ID, to tell LS-DYNA for which parts the cross-section forces are to be obtained. In this case, we will leave this blank.

   Use the display function to see where the cross-section plane is localized.

3. **PART SET**
   Create a Part set, containing only the bolt (part number 4)

When these three keywords are created, open **INITIAL_STRESS_SECTION** and set **CSID**, **LCID** and **PSID** to your keywords that you just created.

As mentioned before, the parameter **SIDR = 1** automatically activates dynamic relaxation analysis. To set some other options for the relaxation, open **CONTROL_DYNAMIC_RELAXATION:**

- Change the number of iterations between convergence checks (**NRCYCK**) to **100**.
- **DRTOL** is the convergence tolerance, there we will use the default value.
- Click **Accept**.

4.6 Output

To create a binary output database for the dynamic relaxation analysis, we will use **DATABASE_BINARY_D3DRLF**. **CYCL** can be defined to set how often d3drlf plots will be written. Set **CYCL = 1**, this means that a plot will be written at every convergence check.

Save the keyword and start the analysis using the single precision SMP version of LS-DYNA.

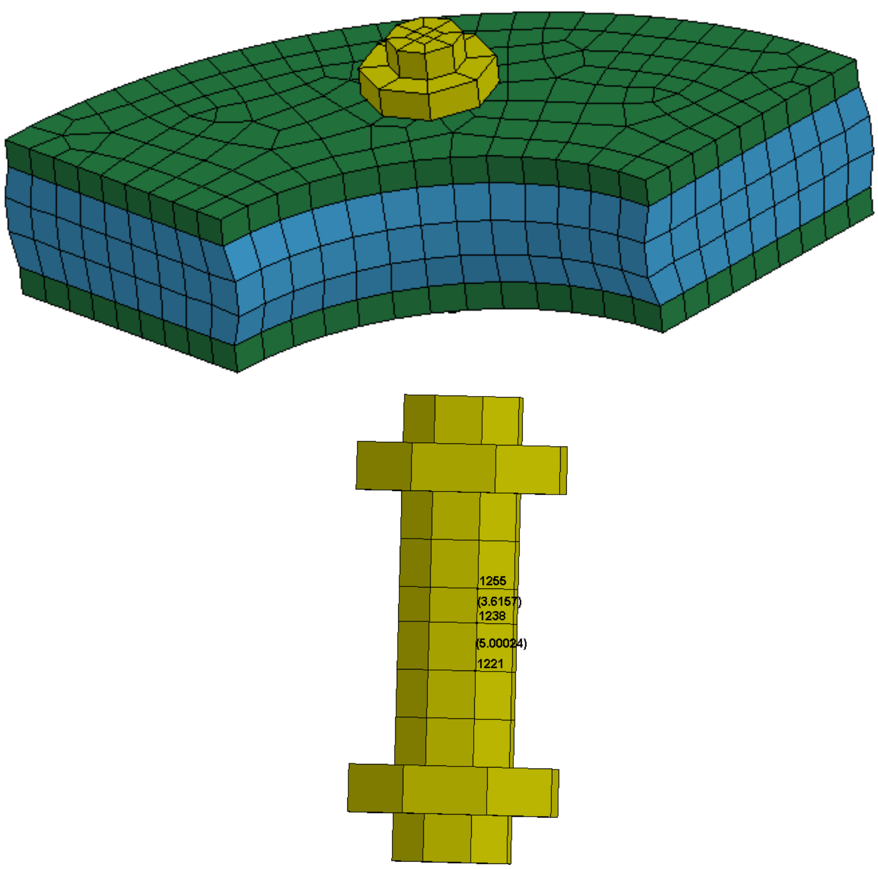
4.7 Post-process

Information about the dynamic relaxation is written in the solver window (to see the solver window in LS-Run make sure to toggle the button Note that when the convergence factor becomes lower than our specified value 1e-3 (default), the dynamic relaxation analysis is done.

```
cycle      time          current          maximum        convergence
                    distortional ke  distortional ke     factor
  100   3.21351E-05    1.03929E-01      1.03929E-01     1.00000E+00
  100 t 3.2135E-05 dt 3.25E-07 write d3drlf file             03/18/16 10:10:23
32900   1.06757E-02    2.04977E-02      1.98762E+01     1.03127E-03
32900 t 1.0676E-02 dt 3.24E-07 write d3drlf file             03/18/16 10:12:55
33000   1.07082E-02    2.01570E-02      1.98762E+01     1.01413E-03
33000 t 1.0708E-02 dt 3.24E-07 write d3drlf file             03/18/16 10:12:55
33100   1.07406E-02    1.97352E-02      1.98762E+01     9.92906E-04
33100 t 1.0741E-02 dt 3.24E-07 write d3drlf file             03/18/16 10:12:55
33100 t 1.0741E-02 dt 3.24E-07 write d3dump01 file           03/18/16 10:12:55
```

Open the **d3drlf** file in the same way that you usually open the d3plot and see what happens with the model during the analysis.

Notice that the elements in the specified cross-section become smaller when the stress is applied.



Now use **Post > History > Element** and plot the stress for a few elements in the specified cross-section and check that the prescribed stress is reached.

In **Post > ASCII**, there is a file called **relax**, which is created during a dynamic relaxation analysis. This file contains information about the dynamic relaxation, like **Convergence** and **Total Kinetic energy**.

Normally the transient analysis starts when the dynamic relaxation is done. In this case, we skipped the transient analysis, just to have focus on the dynamic relaxation. If you open the **d3plot**, this will contain data from the last dynamic relaxation phase. To perform a transient analysis after the dynamic relaxation, do the optional exercise in section 6.

5 Summary

In this exercise, a specified stress was applied on a cross section to preload a bolt. This is just one way to do it and preloads can also be applied in other ways. For example, by shrinking the bolt by cooling it using ***LOAD_THERMAL_LOAD_CURVE** or by prescribing an axial force to beam elements using ***INITIAL_AXIAL_FORCE_BEAM**. Which method that is most suitable depends on the application.

There are several parameters in ***CONTROL_DYNAMIC_RELAXATION** that affects the steady state solution. Some of them are discussed below:

- **DRTOL**
  Convergence tolerance. A smaller value will result in a converged solution nearer the steady state, but the simulation will on the other hand take a longer time.
- **DRFCTR**
  Dynamic relaxation factor. Reduction factor for the nodal velocities each time step. If the value is too small, the simulation will never reach steady state due to overdamping.
- **DRTERM**
  The dynamic relaxation will stop when **DRTERM** is reached, even if the convergence criteria are not satisfied
- **IDRFLG**
  Flag to activate dynamic relaxation. This is not required if dynamic relaxation is activated with **SIDR** in ***DEFINE_CURVE**. If **IDRFLAG = -999**, the dynamic relaxation will be skipped. The latter is a convenient way to turn of dynamic relaxation despite that it has been activated using **SIDR** on one or more DEFINE_CURVE cards.

6 Optional exercises

Perform a transient analysis after the dynamic relaxation. Apply a load to the disk for example. If you use a load curve, remember that **SIDR** for the new load curve should be **0**. Also remember to set a termination time and activate d3plot.