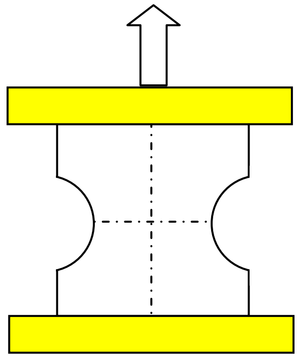
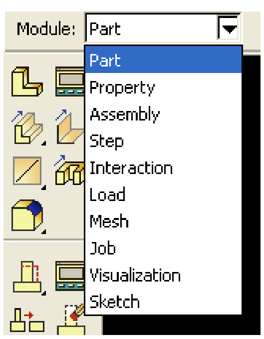
Abaqus Tutorial

Axi-symmetric steel notched bar Tension

This procedure demonstrates how to generate components, material properties, boundary conditions and forces to create a Finite Element model using Abaqus CAE software. The model to be created here is a section through a notched bar (an axisymmetric model), i.e.

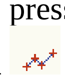


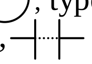
Only the right hand half of the cross section is created in the model.



1. Select **create part**  in the **Part** module. In the dialogue box that appears, name the part **workpiece**. Select **Axisymmetric, deformable, shell,** type the approximate size of 1. Click **continue**.

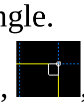
2. Select the **line icon** . Below the display window, type **(0, 0.030)**, press enter and **(0.0175, 0.030),** press enter and **(0.0175, 0.010)**, press enter, and then press escape.

   Select the **line icon**  again, type **(0, 0.030)**, press enter and **(0, -0.030),** press enter and **(0.0175, -0.030),** press enter and **(0.0175, -0.010)**, press enter, and then press escape.

Select the **circle icon**, type **(0.0175, 0)**, press enter and **(0.0175, 0.010),** press enter. Select the **trim icon**, select the right half of the circle, the press escape. Select **done** .

(Note that all of the dimensions are in metre.)

If the sketch of the section contains a small rectangle at any of the corners, it should be deleted as this will constrain the deformation of the section to maintain the same angle.

Select the **delete** icon , then select the feature to be deleted, e.g. the rectangle, , and then **Done**.

(Note, selecting the **red cross**  below the window will end the current procedure.)

3. Choose **create part** in the **Part** module. In the box that appears, name the part **die**. Select **Axisymmetric**. Select '**Analytical rigid**'. Click **continue**.

4. Select '**Create lines**' icon. Type in **(0,0.030)** and **(0.020,0.030)** to create a line. Click the **red cross** below the window to end the current procedure. Select **done**.

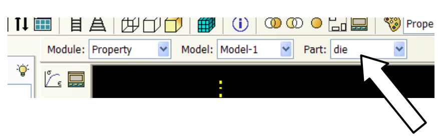
5. Select **Tools** from the list across the top of the screen. In the pull-down menu, select **reference point**. Click the **right end of the die**.

6. Proceed to the **Property** Module. Choose **create material** icon. Name the material-1 as **workpiece**. Select **General** and **Density**. Enter 7600 Kg/m^3. Select **Mechanical**. Choose **elasticity** and **elastic**. Enter 210 GPa (210E9) for Young's modulus and 0.3 for the Poisson ratio. Select **mechanical**. Choose **plasticity** and **plastic**. Enter the values as shown below. Click **OK**.

Flow stress (Pa)	Plastic strain
208E6	0
492E6	0.1
550E6	0.5
580E6	1.0

7. Choose **create section** . Name section-1 as **workpiece**, accept the other defaults. Click **continue**. Click **OK** and again **OK** in the next box that appears.

8. In the **Part** box,

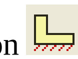


select **workpiece**. Go to **Assign section** option . Click on the image of the workpiece on the screen. Click **done**. In the **edit section assignment** box select **workpiece**. Click **OK**. Save the model.

9. Proceed to the **Assembly** module. Choose **Instance part** . In the **Create Instance** dialogue box, choose **workpiece** and click **apply**. Then choose **die** and click **OK**.

10. Proceed to **Step** module. Choose **create step** icon . Select **Dynamic Explicit**. Click **continue**. In the **Edit Step** dialogue box, under **Basic**, name the description as **Deform**. Turn **Nlgeom** on. Click **OK**.

11. Continue to **Interaction** module. Double click **constraints** function in the left hand side tree. Choose **tie**. Name it **constrain**. Click **continue**. On the screen select the **die** as the master surface. Click **done**. Choose **yellow** to indicate the bottom surface of the die. Choose **surface**. Select the **show/hide selection options** icon  below the window. In the box that appears, click the **select the entity closet to the screen** option. Click the top edge of the workpiece. Click **OK** and **done**.

12. Proceed to **Load** module. Choose the **create boundary condition** icon  (BC) for each of the following constraints.

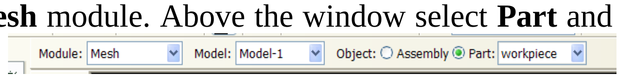
    a. Name the BC as **fixed_y**. Choose step: **Initial**; type: **Displacement/Rotation**. Click **continue**. Select **the bottom line** of the workpiece. Click **done**. Select **U1, U2, UR3** and click **OK**.

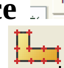
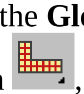
    b. Name the BC as **fixed_x**. Choose step: **Initial**; type: **Symmetry/Antisymmetry/Encastre**. Click **continue**. Select the **left border line** of the workpiece. Click **done**. Select **XSYMM** and click **OK**.

    c. Name the BC as **fixed_ref**. Choose step: **Initial**; type: **Displacement/Rotation**. Click **continue**. Select **reference point**. Click **done**. Select **U1** and **UR3** and click **OK**.

    d. Name the BC as **deform_up**. Choose step: **Step-1**; type: **Displacement/Rotation**. Click **continue**. Select **reference point**. Click **done**. Select **U2** and input '**0.001**' in the adjacent box. For **Amplitude** box, Select **Create**. For the amplitude type, select **Tabular**. Then enter 0,0 in the first row and 1,1 in the second row and click **OK**.

    e.

13. Proceed to **Mesh** module. Above the window select **Part** and in the **Part** box, select **workpiece** 

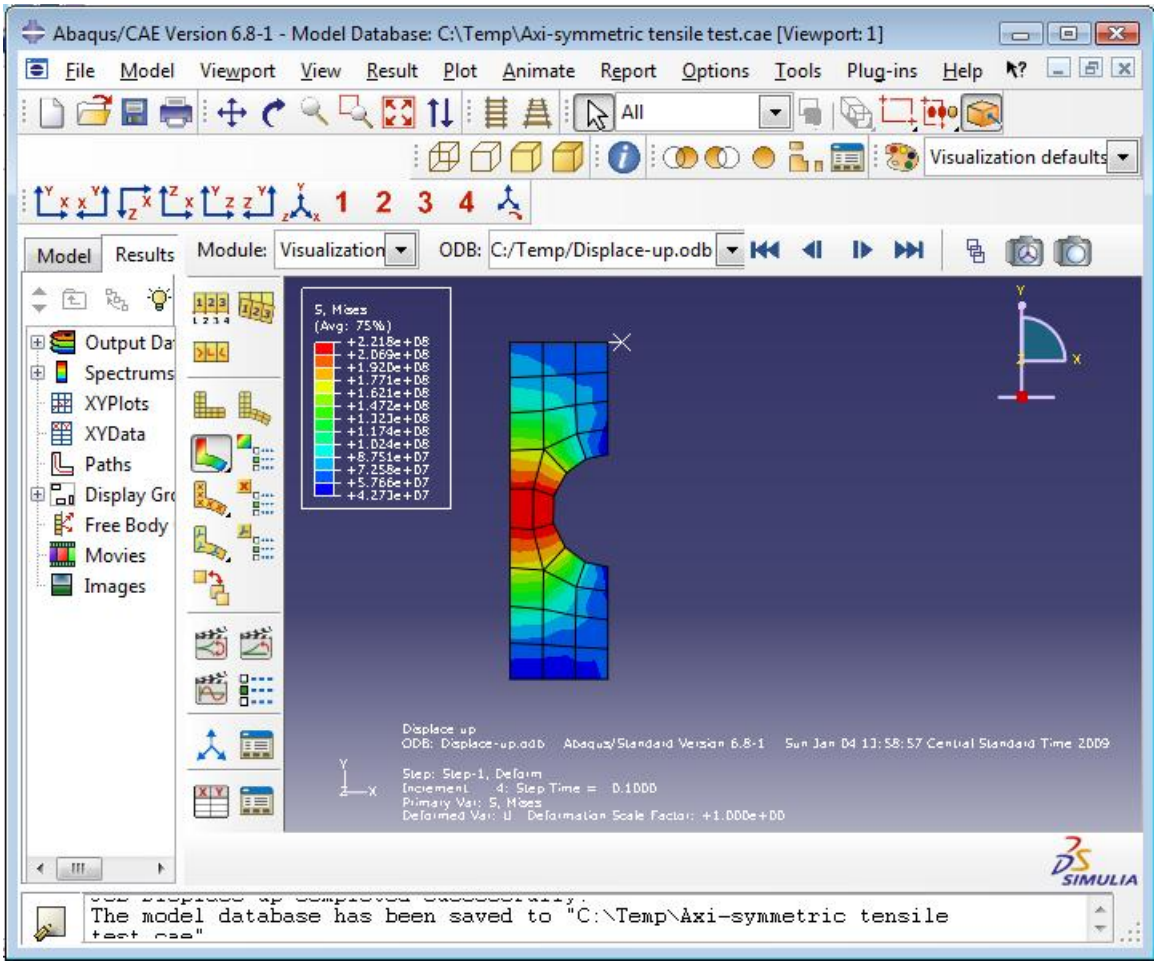
. Click on the **Seed Part** icon . Accept the defaults in the **Global Seeds** dialogue box. Click **done**. Click on the **Mesh Part Instance** icon , at the bottom of the screen select **yes**.

14. Next, go to **Job** module. Click on '**create job**' icon . Name: **Job-1**, click **continue**. Name description as '**Displace up**'. Click **OK**. Save the model.

15. Click on **Job manager** . Click '**Submit**' to submit the job to the solver. Click **monitor** to check the analysis progress.

16. Once the job is complete, click **results** to check the results. Select the icon  to display contours. To examine different stresses etc, select **Result, Field Output** from the main menu bar.

Below is a typical illustration of the results that you should obtain, von Mises stress is displayed.



Student must re submit the jop_1 for different deform_up displacement (0.002m, 0.005m). You must use your judgement as to whether the mesh is appropriate (e.g. are the elements small enough).