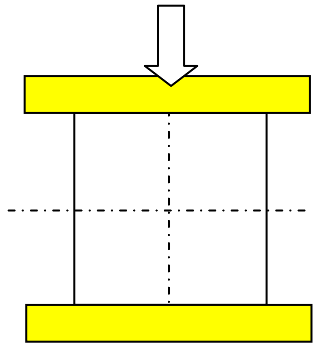
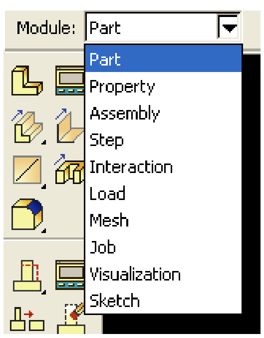
Abaqus Tutorial

Axi-symmetric Upsetting

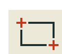
This procedure demonstrates how to generate components, material properties, boundary conditions and forces to create a Finite Element model using Abaqus CAE software. The model to be created here is a section through a cylinder (an axisymmetric model), i.e.



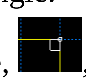
Only the upper-right hand quadrant of the cross section is created in the model.

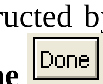


1. Select **create part**  in the **Part** module. In the dialogue box that appears, name the part **workpiece**. Select **Axisymmetric, deformable, shell,** accept the approximate size of 200. Click **continue**.

2. Select the **box icon** . Below the display window, type **(0,0)**, press enter and **(20,20),** press enter, to create a box. Note that all of the dimensions are in mm.

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**.

The box (or cross section of the workpiece) may also be constructed by specifying the individual lines by selecting the **create lines** icon . Select **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 **(0,20)** and **(30,20)** to create a line. Click the **red cross** below the window to end the current procedure. Select **done**.

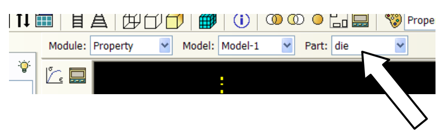
5. Select **Tools** from the list across the top of the screen. In the pull-down menu, select **reference point**. Click the **right end of the die**.

6. Proceed to the **Property** Module. Choose **create material**  icon. Name the material-1 as **workpiece**. Select **General** and **Density**. Enter 7.6e-9 ton/mm^3. Select **mechanical**. Choose **elasticity** and **elastic**. Enter 210 GPa for Young's modulus (be sure to convert this to N/mm^2, due to the dimensions of the box being in mm's) and 0.3 for the Poisson ratio. Select **mechanical**. Choose **plasticity** and **plastic**. Enter the values as shown below. Click **OK**.

Flow stress (MPa)	Plastic strain
404	0
965	0.1
1209	0.5

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.

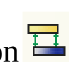
8. In the **Part** box,

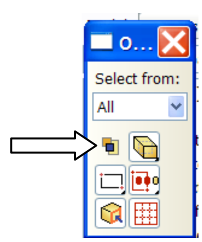


select **workpiece**. Go to **Assign section** option . Click on the image of the workpiece on the screen. Click **done**. In the **edit section assignment** box select **workpiece**. Click **OK**. Save the model.

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**.

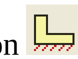
10. Proceed to **Step** module. Choose **create step** icon . Select **Dynamic Explicit.** Click **continue**. In the **Edit Step** dialogue box, under **Basic**, name the description as **Deform**. Turn **Nlgeom** on.

11. Continue to **Interaction** module. Choose **create interaction** icon . Name it **contact**. Choose **Initial** in the **step** list and **Surface to Surface Contact** for the contact type. Click **continue**. On the screen select the **die**. 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. In the **Edit Interaction** window that should appear, select **create**, next to the **contact interaction property** box. Name it '**Friction**'. Click **continue**. In the next dialogue box choose **Mechanical** and select **tangential behaviour**. Select **Penalty** in the **Friction Formulation** area and enter **0.3** in the friction coefficient box. Click **OK**.

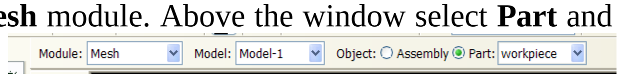
13. Proceed to **Load** module. Choose the **create boundary condition** icon  (BC) for each of the following constraints.

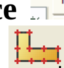
    a. Name the BC as **fixed_y**. Choose step: **Initial**; type: **Symmetry/Antisymmetry/Encastre**. Click **continue**. Select **the bottom line** of the workpiece. Click **done**. Select **YSYMM** and click **OK**.

    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_down**. Choose step: **Step-1**; type: **Displacement/Rotation**. Click **continue**. Select **reference point**. Click **done**. Select **U2** and input '**-5**' 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**.

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

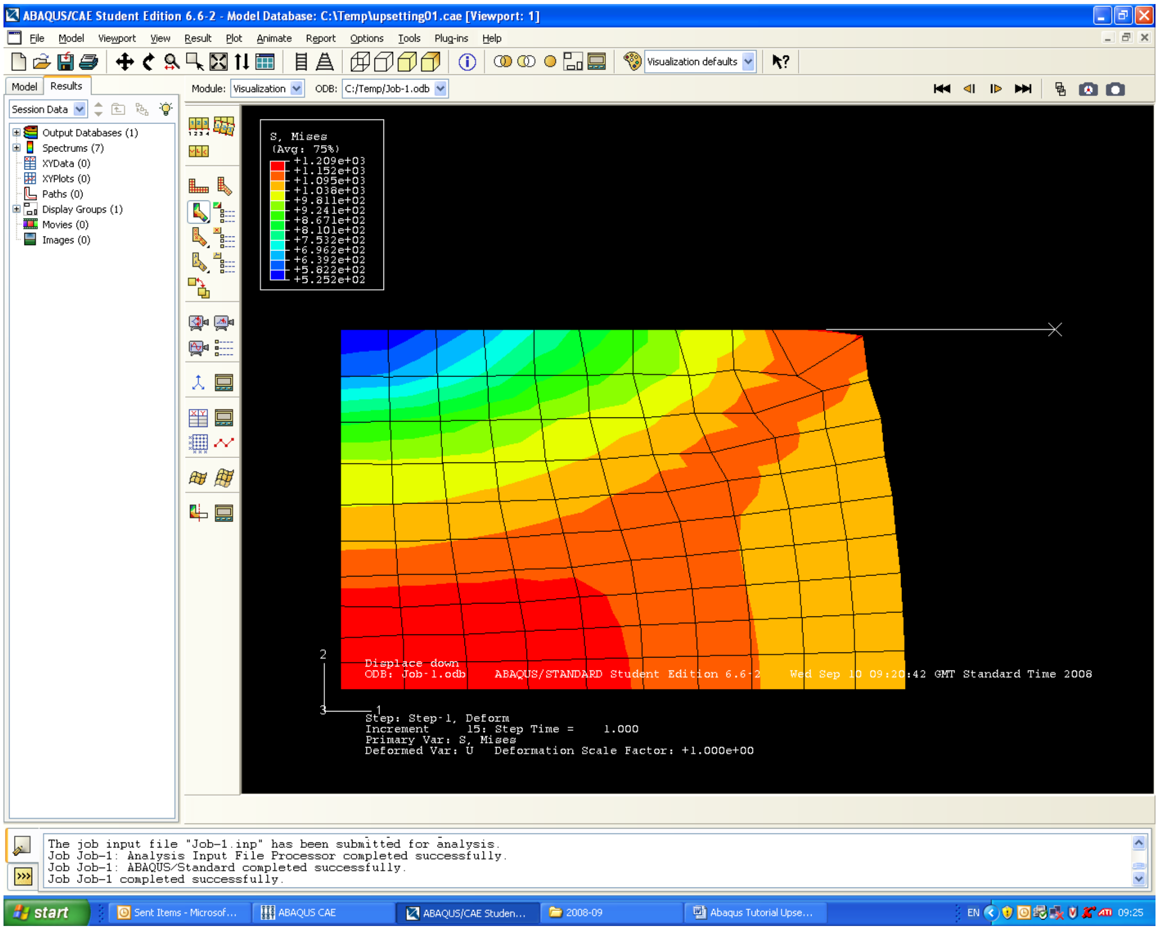
. Click on the **Seed Part** icon . Accept the defaults in the **Global Seeds** dialogue box. Click **done**. Click on the **Mesh Part Instance** icon , at the bottom of the screen select **yes**.

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

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

17. 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.



You must use your judgement as to whether the mesh is appropriate (e.g. are the elements small enough), and whether the increment of deformation per step is small enough (specified as -5 in the load module for this example).

If the right-hand free surface of the workpiece deforms enough to reach the die surface, you must specify an interaction between these two surfaces, otherwise the workpiece material will move into the die, which is not physically possible.