QUICK GUIDE TO ABAQUS/ CAE

The Abaqus/ CAE workspace

The Abaqus/ CAE workspace (Fig. 1) consists of the following subsections that are going to be mentioned throughout this guide, namely:

- The Viewport: The main output screen of Abaqus/ CAE where pre and post processing data is visualized.
- The model Tree View: All the basic modelling steps are presented in this section in the form of tree nodes. Each node is subdivided into several subnodes offering corresponding functionalities.
- The Toolbar Section: Each node of the model Tree View has a corresponding toolbar from where the user can access relevant forms.
- The prompt region: Upon selecting to perform a certain function, tips and or command buttons appear on the prompt region.

Fig. 1 Abaqus CAE workspace overview
Step 1: File definition

Start Abaqus and choose to create a new model database. Remember to save in regular time intervals throughout your session.

Step 2: Geometry Definition

In the Model Tree double click on the Parts node (or right click on Parts and select Create) - Fig. 2a.

In the Create Part form (Fig. 2b) check the appropriate parameters depending on your structure (in this case a 2D wire structure). Pressing Continue the sketch area appears that is basically a CAD interface where you can draw the geometry of your model.

Fig. 2 (a) Model Tree view (b) Create Part form
Step 3: Material Definition

Double click on the Materials node in the Model Tree (Fig. 3)

Fig. 3 Material Node in Model Tree view

Name the new material and give it a description. To define an Elastic Material click on the Mechanical tab → Elasticity → Elastic and give appropriate values for the Young’s Modulus and the Poisson’s ratio (Fig. 4)

Fig. 4 Edit Material Form – Elastic Material Definition

To add a nonlinear plastic law to the elastic material (e.g. consider the case of an elastic perfectly plastic Von-mises type material model), after defining the elastic properties on the “Mechanical” tab → Plasticity → Plastic and give appropriate values for the initial yield stress and initial plastic deformation. By pressing Enter, another row appears at the table where you should define more points in the stress plastic strain curve (i.e a failure stress and the corresponding plastic strain) as presented in Fig. 5.
Step 4: Section Definition

Double click on the **Sections** node in the Model Tree (Fig. 6a). Name the section and select **Beam** for the category to define a linear element Section and **Truss** for the type (Fig. 6b). Click **Continue...**, select the material created from the dropbox and set the cross sectional area. Click **OK**.
**Step 5: Section Assignment**

On the **Parts** node in the Model Tree, double click on **Section Assignments**. Select the elements from the model viewport, then the Section created earlier from the combo box and click **OK**. Thus, each line entity is attributed with a section property.

**Step 6: Model Instance**

Under the **Assembly** node in the Model Tree double click on **Instances**. Select **Dependent** for the instance type and then click **OK**.

**Step 7: Model Mesh**

Under the Part node, double click on **Mesh** (Fig. 7).

Click on the **Assign Element Type** in the toolbar.

![Assign Element Type](image)

![Seed Edges](image)

Fig. 8(a) Assign Element Type menu button (b) Seed Edges menu button

This is the element library of Abaqus (Fig. 9). Select **Standard** for element type, **Truss** for the truss element and **Linear** to define the two node Isoparametric Element. Click **OK**.
In the toolbox area click on the Seed Edge: By Number icon (hold down icon to bring up the other options). Select the entire geometry and click Done in the prompt area. Define the number of elements along the edges as 1 and click Enter in the prompt region then Done in response to the next prompt. In the toolbox area click on the Mesh Part icon. Click Yes in the prompt area to mesh the instance.

**Step 8: Analysis Type definition**

Double click on the Steps node in the Model Tree. The Create Step (Fig. 10) form appears where you can define the analysis type. For a linear static analysis or a NR scheme choose Static, General from the list box. For an Arc-length Riks type method choose Static Riks. Click Continue, name the step then click OK to close the form. Please be careful of the different parameters one should consider for the type of analysis s/he wishes to perform. In case of a large displacement analysis, remember to check the Nlgeom radio button (Fig. 11).
Fig. 10 Create Step form

Fig. 11 Nlgeom Parameter
Step 9: Boundary Conditions

Double click on the BCs node in the Model Tree. The Create Boundary Condition form appears (Fig. 11). Name the boundary conditioned and select Displacement/Rotation for the type. Click Continue then select the nodes at which Bcs should be assigned. Please note the messages appearing on the prompt area (Fig. 12). Press Done in the prompt area. In the form choose the degrees of freedom you wish to restrain and set their corresponding value to zero. Then click OK.

Fig. 12 Create Boundary condition form

Fig. 13 Boundary condition definition prompt message
Step 10: Loads

Double click on the Loads node in the Model Tree. Name the load, and select Concentrated force as the type. Click Continue... and then select the points where loads should be assigned and press Done in the prompt area. In the next form define the values of the assigned loads and then click OK.

Warning: Every time you decide to delete a STEP the corresponding loads and boundary conditions will be also deleted.

Step 11: Run Analysis

In the Model Tree double click on the Job node, name the job, then click Continue and then OK. In the toolbar click on Job Manager (Fig. 13). Press Submit to start the analysis. While Abaqus is solving the problem right click on the job submitted, and select Monitor to get info on the analysis procedure. After the analysis is finished press on Results to open the Visualization module of Abaqus.