# 4. Solution and Postprocessing of ABAQUS CAE

When you finish preprocessing of your finite element analysis, the finite element model should be written into finite element code and submitted to the solver. Meanwhile, we also would like to monitor or manage the solution processing. Finally, after the finite element analysis is completed successfully, the visualization module can be used for users to check the results in different approaches.

4.1 Solution—Job Module Module: Job
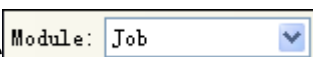
Once you have finished the finite element modeling such as geometry, material property, boundary condition (load and constraint), you can use the job module to analyze your model.

In this module, you can **create a job** which includes all the information of your finite element model. And then you can submit the job to the solver (ABAQUS/Standard or ABAQUS/Explicit). When you submit a job associated with a model for analysis, ABAQUS/CAE first generates an input file representing your model and then the solver performs the analysis using the contents of this file.

Tip: you can generate only the input file alternatively. And then open and edit preprocessing information in your working directory using notebook.

You can use **Job Manager** to **submit** job for analysis. Then use **monitor** function to monitor the analysis process. Use **result** function to view the results in visualization module and use the **kill** function to terminate the analysis before it is completed.

Job editor: During creating new job, you can edit the job attributes in **job editor** like follows:
Description: you can write down the description about your job, which will remind you the objective of this job.
Submission Page: configure the submission attributes of your job, such as job type, run mode, and submit time.
General Page: configure job settings such as the analysis input file processor printout and the name of the directory used for scratch files.
Memory Page: configure the amount of memory allocated to an ABAQUS analysis.
Parallelization Page: configure the parallel execution of an ABAQUS analysis job, such as the number of processors to use and the parallelization method.
Precision Page: configure either single or double precision for ABAQUS/Explicit analysis.
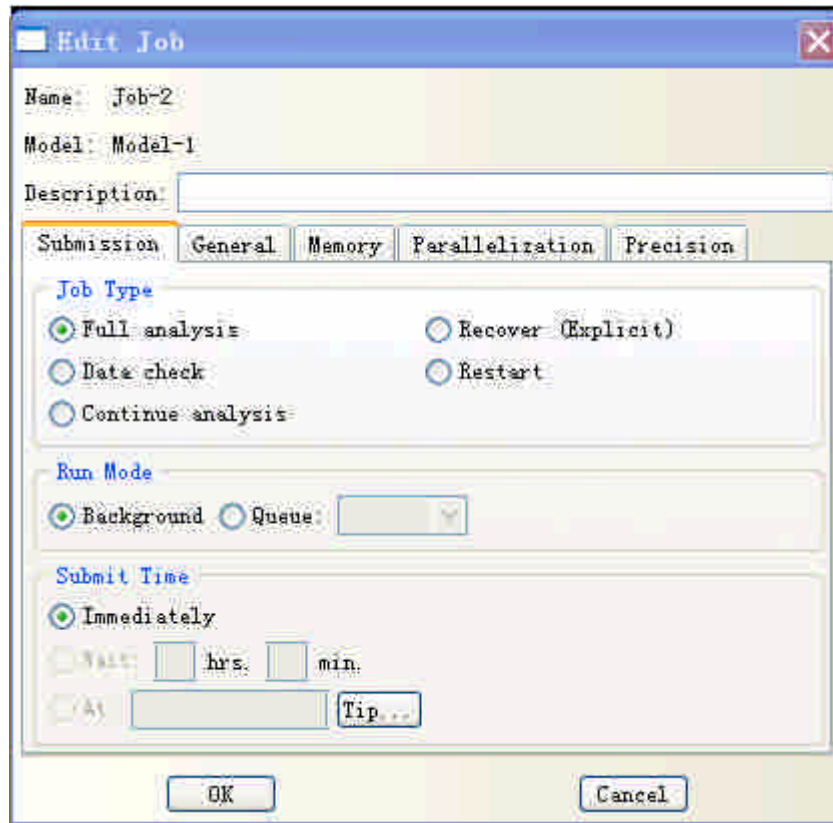
Figure 4.1 Edit job

Job Monitor provides necessary information during solution process such as log, errors, warnings and output. It continually updates the information presented in the job monitor dialog box as the analysis progresses.
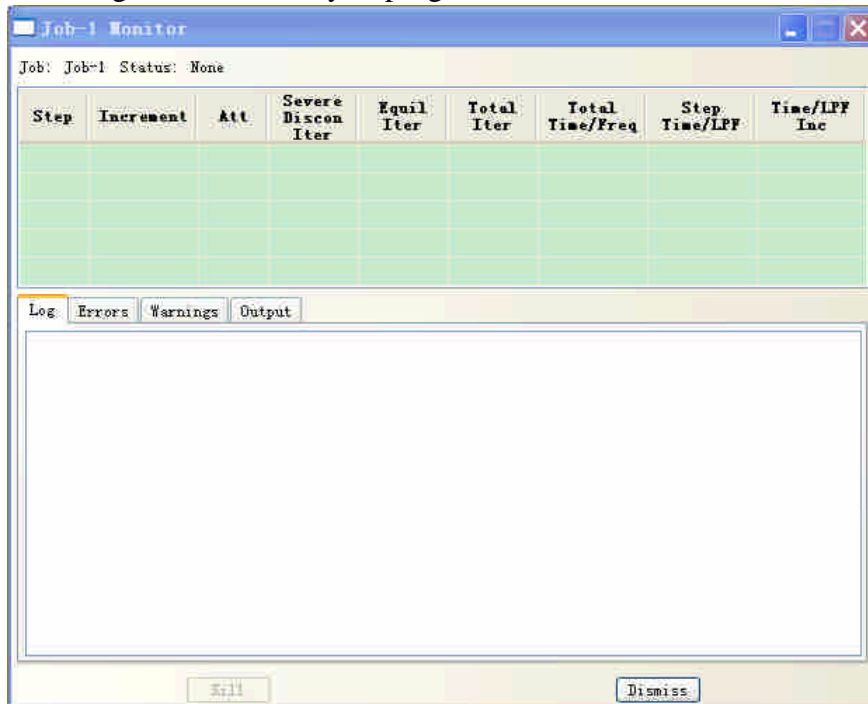

Figure 4.2 Job monitor

Tip: more details see *18. The Job module of ABAQUS/CAE User's Manual in ABAQUS Documentation*

4.2 Postprocessing--Visualization module Module: Visualization ∨

Once you complete your finite element analysis successfully according to the prompt in the log tag of job monitor, you can choose the **results** button in **job manager** window to enter **visualization module** to view the result in different methods. The **Visualization module** provides graphical display of finite element models and results. It obtains model and result information from the output database; you can control what information is placed in the output database by modifying output requests in the **Step module**.

Tip: Another method to enter the **Visualization module** is **Start Session→ Open Database** choose the *.odb file.

Or ⬆**open database→File Filter Output Database** (*.odb*)

You can view your model and results by producing any of the following plots:

Fast plot: is a quickly drawn representation of your model.

Undeformed shape: displays the initial shape or the base state of your model.

Deformed shape: displays the shape of your model according to the values of a nodal variable such as displacement.

Contours: displays the values of an analysis variable such as stress or strain at a specified step and frame of your analysis. The **Visualization module** represents the values as customized colored lines, colored bands, or colored faces on your model.

Symbols: displays the magnitude and direction of a particular vector or tensor variable at a specified step and frame of your analysis. The **Visualization module** represents the values as symbols (for example, arrows) at locations on your model.

Material orientations: displays the material directions of elements in your model at a specified step and frame of your analysis. The **Visualization module** represents the material directions as material orientation triads at the element integration points.

X–Y data: is a two-dimensional graph of one variable versus another.

Time history animation: displays a series of plots in rapid succession, giving a movie-like effect. The individual plots vary according to actual result values over time.

Scale factor animation: displays a series of plots in rapid succession, giving a movie-like effect. The individual plots vary in the scale factor applied to a particular deformation.

Harmonic animation: displays a series of plots in rapid succession, giving a movie-like effect. The individual plots vary according to the angle applied to the complex number results being displayed

Important Visualization options

Tools→Job Diagnostics: Diagnostic information helps you determine the causes of nonconvergence in a model when your analysis completed with warnings or can not be completed successfully. You can view information for each stage of the analysis and use ABAQUS/CAE to highlight problematic areas on the model in the viewport.

Tools→Query: Query option includes General Queries and Visualization Module Queries.
General Queries can display the results on Node, Distance, Element and Mesh.
Visualization Module Queries includes Probe values and Stress linearization.
Probing values: displays model data and analysis results as you move the cursor around a model plot;
Stress linearization is the separation of stresses through a section into constant membrane and linear bending stresses. You specify the section as a path through your model, and the Visualization module displays the linearized stresses in the form of an X–Y plot.

Tools→Path: is a line you define by specifying a series of points through your model. You can view results along the path in the form of an X–Y plot.

Tools→View cut: allow you to slice through a model so that you can visualize the interior or selected sections of the model. You can define planar, cylindrical, or spherical view cuts. In addition, you can define a view cut along a constant contour variable value.

Tools→X-Y data: is a tabular listing of X- and Y-data values;

Tools→display group: is a collection of selected model components and can contain the entire model or combinations of part instances, geometry (cells, faces, or edges), elements, nodes, and surfaces. Display groups allow you to reduce clutter on your screen and focus on areas of interest within your model, to access "hidden" components in complex models, and to decrease the amount of time needed to refresh the display in the current viewport.

Report→field output: is a tabular listing of field output values.

4.3 The meaning of output in ABAQUS

In **Visualization Module**, users should know which output is the required one or you expect. It is necessary to understand the relationship of variables between engineering and ABAQUS output in **Visualization Module**, which will help new users get a correct analysis data and carry out validation.

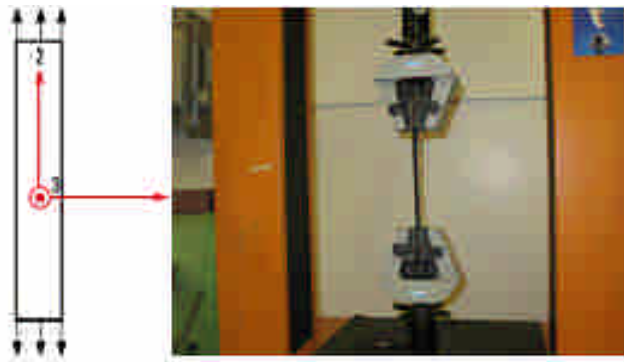To take an uni-axial tension as an example shown in Fig.4.3

Figure 4.3 Uni-axial tension

In engineering viz. the results you can from testing,
The engineering strain (nominal strain) and engineering stress (nominal stress) can be expressed as

$$\varepsilon_{eng} = \frac{\Delta l}{l_0} \quad \text{and} \quad \sigma_{eng} = \frac{F}{A_0}$$

Where Δl is the extended length, the l_0 is the original length, F is the applied

force and A_0 is the original cross-section.

In ABAQUS simulation, the results are usually true strain (LE) and true stress (Mises stress). Engineering stress and strain can be converted to true stress - log strain by:

$$\varepsilon_{true} = \ln\left(1 + \varepsilon_{eng}\right) \quad \text{and} \quad \sigma_{true} = \sigma_{eng}\left(1 + \varepsilon\right)$$

You can also define the strain output as nominal strain (NE) which is the same as engineering strain.

4.4 File type of ABAQUS

During the whole finite element analysis process, there are different types of files generated with different functions. The following are some important files you should know

5

✓ When you select **File→Sav**e from the main menu bar and save the model database, ABAQUS/CAE saves the following files:
<div align="center">

The model database file (model_database_ name.cae)
</div>

✓ ABAQUS/CAE generates an input file that is read by ABAQUS/Standard or ABAQUS/Explicit when you submit a job for analysis.
<div align="center">

Input files (job_name.inp)
</div>

✓ Output database files contain the results from your analysis.
<div align="center">

Output database files (job_name.odb)
</div>

✓ The lock file (job_name.lck) is written whenever an output database file is opened with write access, including when an analysis is running and writing output to an output database file. It is deleted automatically when the output database file is closed or when the analysis that creates it ends.
<div align="center">

The output database lock file (job_name.lck)
</div>

✓ The restart file is used to continue an analysis that stopped before it was complete.
<div align="center">

The restart file (job_name.res)
</div>

✓ The data file contains printed output from the analysis input file processor, as well as printed output of selected results written during the analysis. It also contains the warning and error information if the analysis has.
<div align="center">

The data file (job_name.dat)
</div>

✓ The message file contains diagnostic or informative messages about the progress of the solution. You can control the diagnostic information that is output to the message file using the Step module.
<div align="center">

The message file (job_name.msg)
</div>

✓ The status file (job_name.sta) contains information about the progress of the analysis.
<div align="center">

The status file (job_name.sta)
</div>

Tip: More details see "File extensions used by ABAQUS," Section 3.6.1 of the ABAQUS Analysis User's Manual.

4.5 Input file and command usage

The analysis objective of ABAQUS/Standard and ABAQUS/Explicit is the *.inp file rather than the *.cae file. Thus the *.inp file is the bridge between solver and model. When you submit your job to solver, the *.inp will be generated automatically in the

directory. Alternatively, you can use the button **write input** in **Job manager** windows to generate *.inp file. Users can modify the input file to change parameters of the whole analysis process. Meanwhile, the *.inp file also support some special functions (keywords) that ABAQUS/CAE does not support.

ABAQUS provides an alternative way to submit job and carry out analysis using **ABAQUS command**

ABAQUS 6.9-1→ABAQUS command
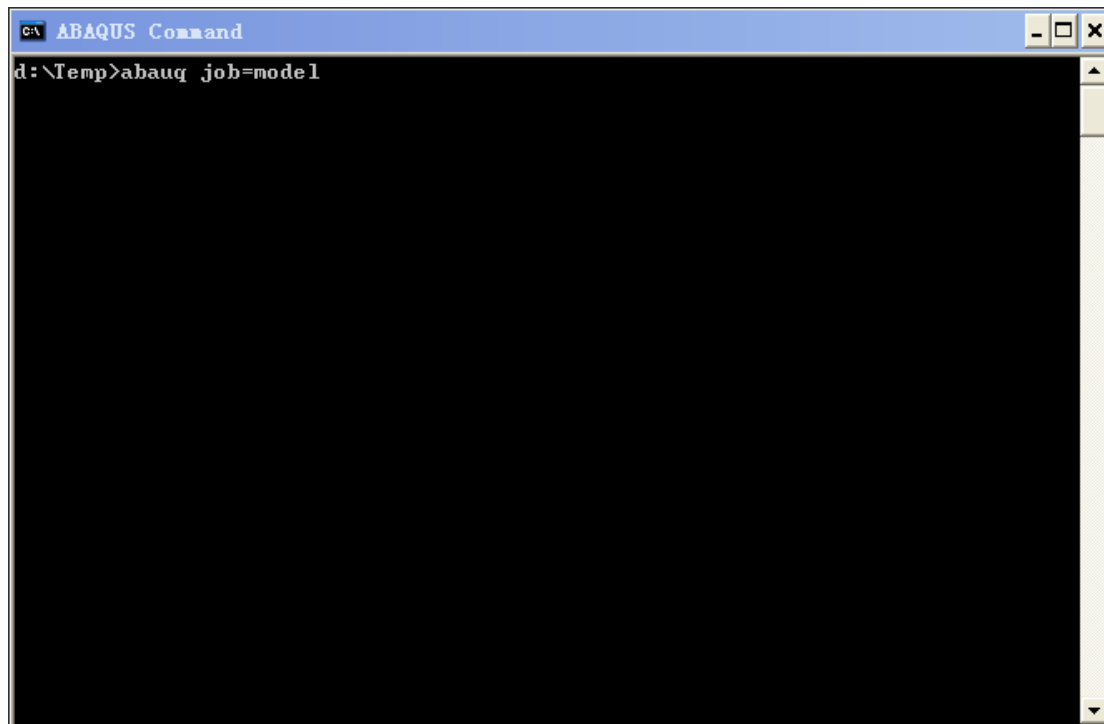
Tip: more details see 3. Execution procedure in ABAQUS Analysis User's Manual.

Furthermore, for the users who need to calculate complex structure analysis and need a lot computational source, as well as the users who do not have the license of ABAQUS solution, Bluebear cluster in the University campus provide the ABAQUS application using University high performance computational source.
Details see **http://www.bear.bham.ac.uk/bluebear/applications/abaqus_691.shtml**

Tip: **in order to have a sustainable career using ABAQUS, you can use the crutch from My ABAQUS. It is an online system where all the information is from ABAQUS technician including FAQ about engineering practice.**

Start→All Programs→ABAQUS 6.9-1→My ABAQUS