



Course in finite element simulations using Abaqus

Topic : Modelling and viewing in Abaqus/CAE

Esmail Tohidlou

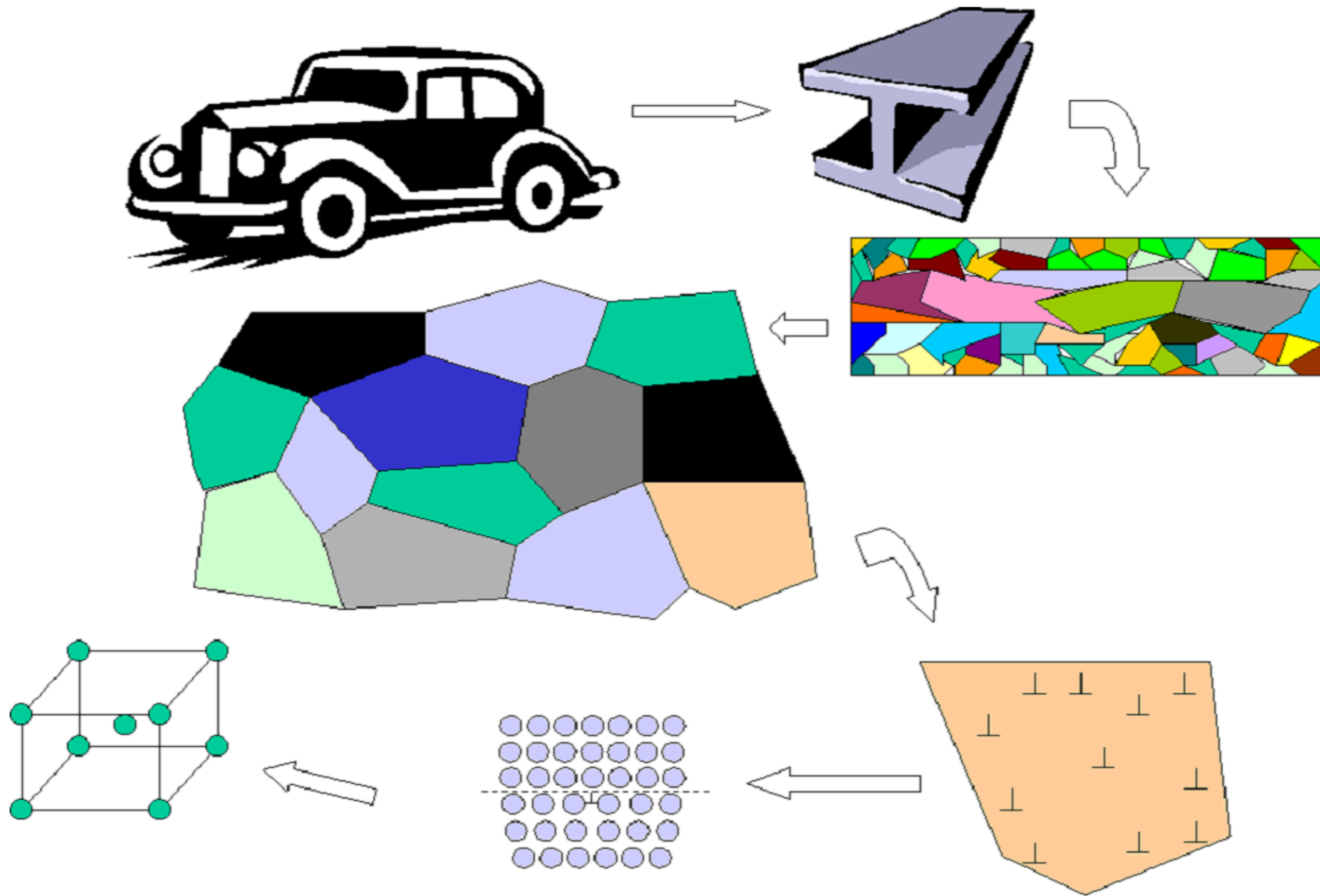
Department of Materials Engineering

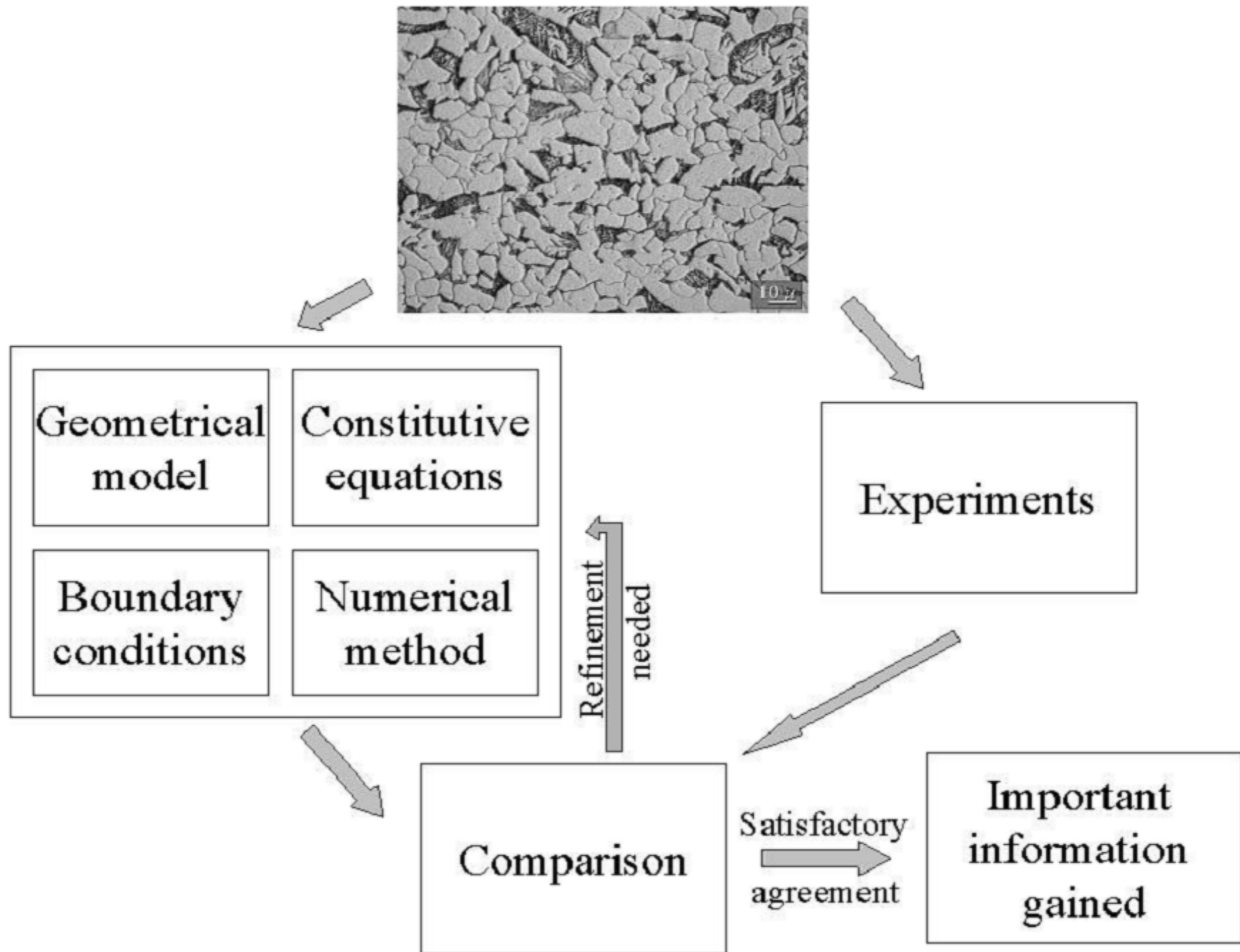
University of Sistan and Baluchestan

etohidlou@gmail.com

2017

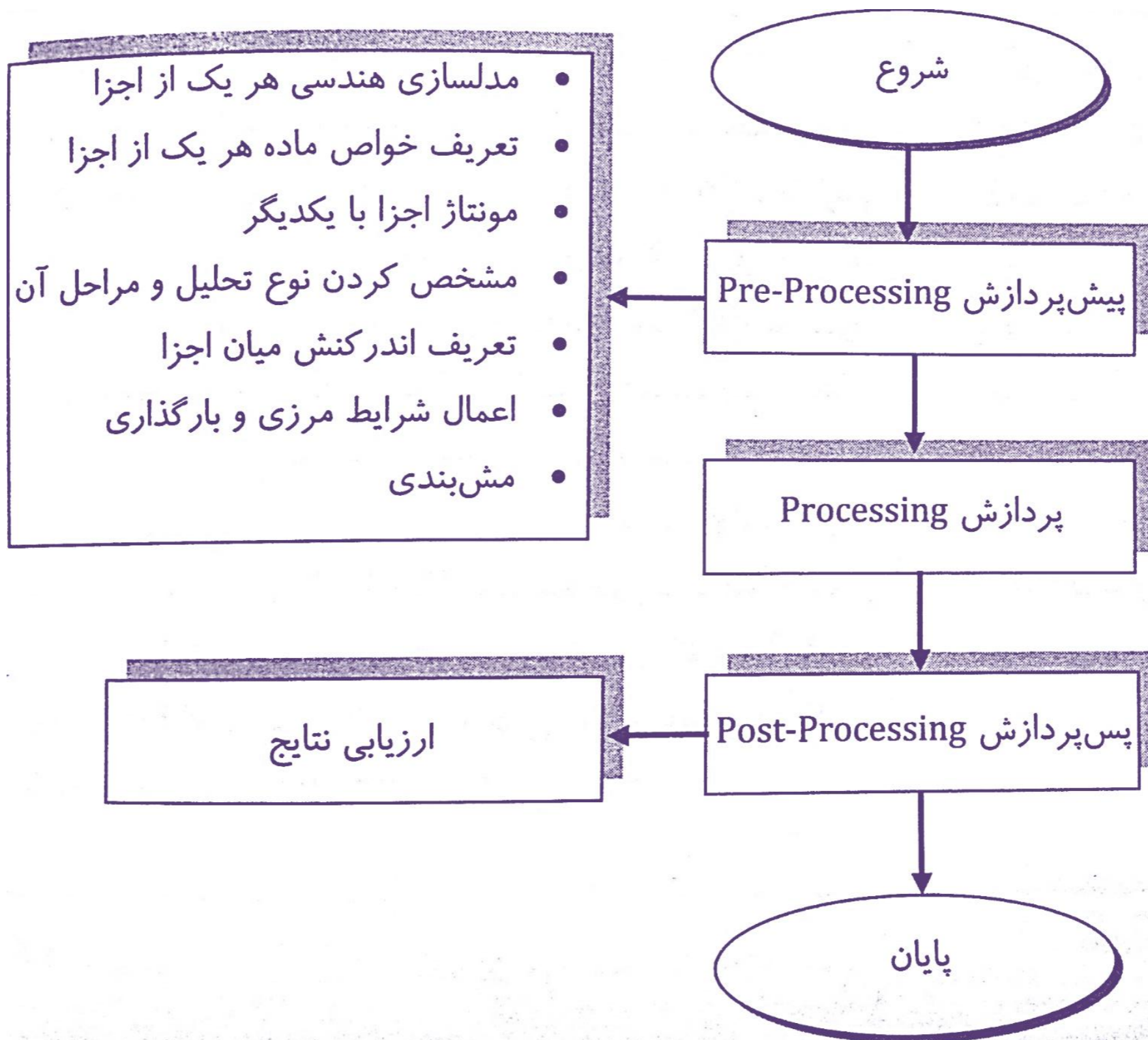
Modeling of materials may be performed at different length scales, depending on the area of interest. Even though there are orders of magnitude between the different scales in modeling of materials, there are still several similarities regarding mechanical properties and modeling approaches.





Problems investigated

- Topic 1: Abaqus introduction
 - Example: investigation of stress concentration
- Topic 2: Material models:
 - Exercise: Elastic, Non-linear elastic, Elastic-plastic, Visco-elastic, Anisotropic elastic
- Topic 3: Build up a 2-D plane stress model
 - Exercise: Scripting, Plate with a hole
- Topic 4: Composite layup, buckling of structures
- Topic 5: Crack-tip simulations
- Topic 6: Cohesive laws, extended finite elements (XFEM).



مراحل مدل سازی و تحلیل مسائل در نرم افزارهای المان محدود

Pre-processing, solver, Post-processing

- Pre-processing: Abaqus CAE
 - Defining and build up the model (saved in .cae and .jnl file and abaqus.rpy)
 - Generate a job-name -> generate an .inp-file
 - Can be replayed/modified using a python script in the CAE or
 - command: `abaqus cae noGUI=PYTHONSCRIPTpre.py`
- Solver: Abaqus analysis
 - Solve the problem defined in the .inp-file
 - Information about the solving process can be found in the .dat, .msg and .sta file
 - The analysis can be performed using the submit in CAE or the command: `abaqus job=JOBNAME.inp cpus=12 interactive`
- Post-processing: Abaqus Viewer
 - Open the output database .odb
 - Can be replayed/modified in a python script in the CAE, Viewer or command: `abaqus viewer noGUI=PYTHONSCRIPTpost.py`

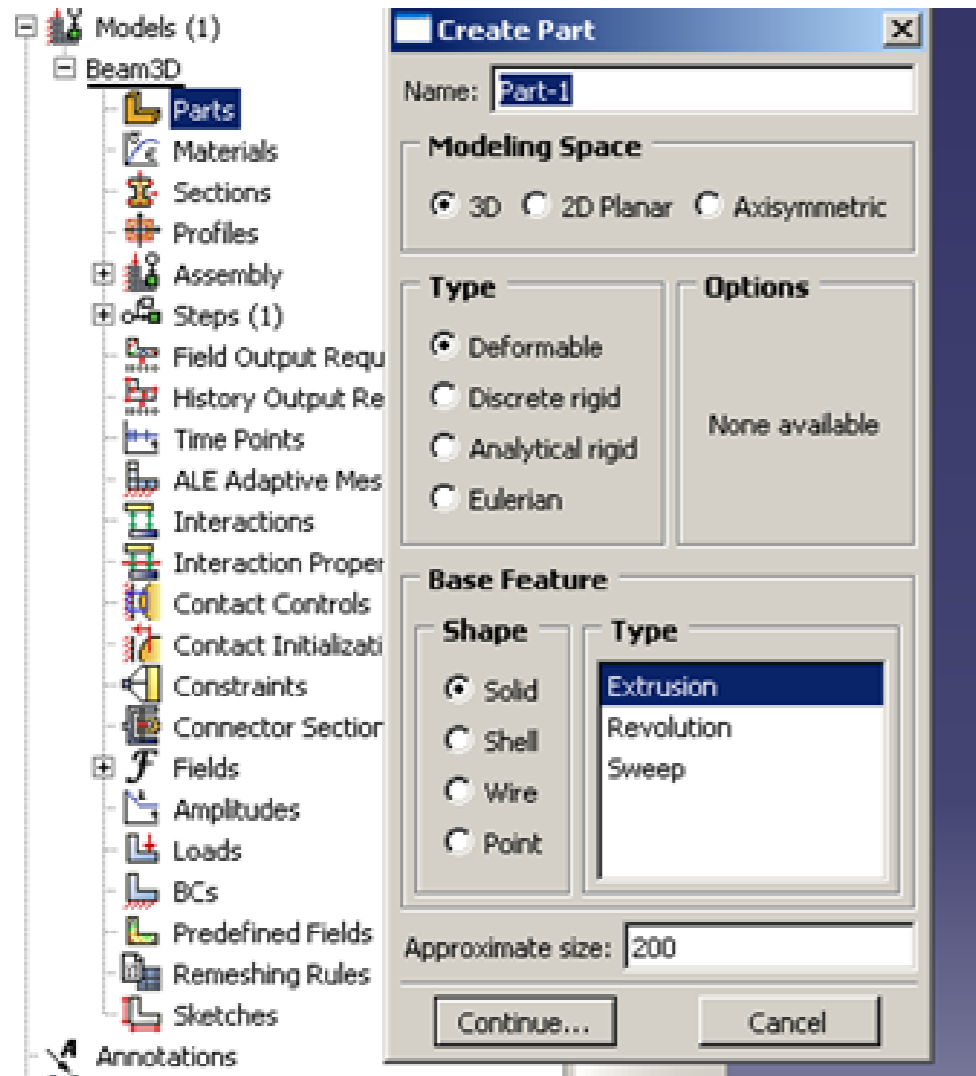
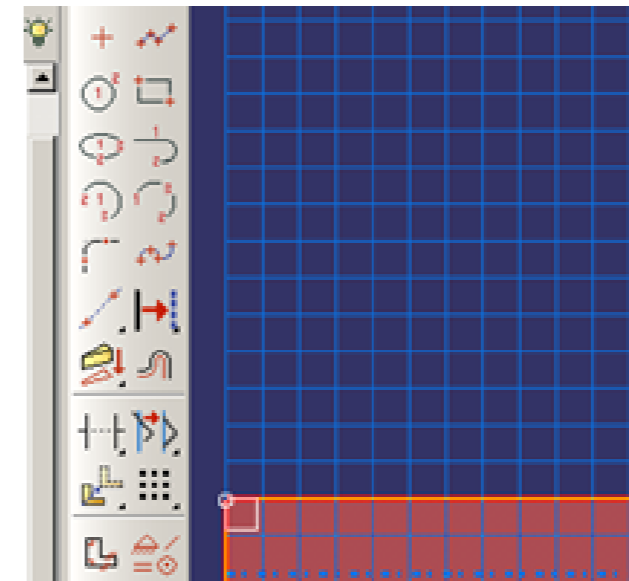
Introduction to Abaqus/CAE Pre-processing

“Getting Started with Abaqus: Interactive Edition”

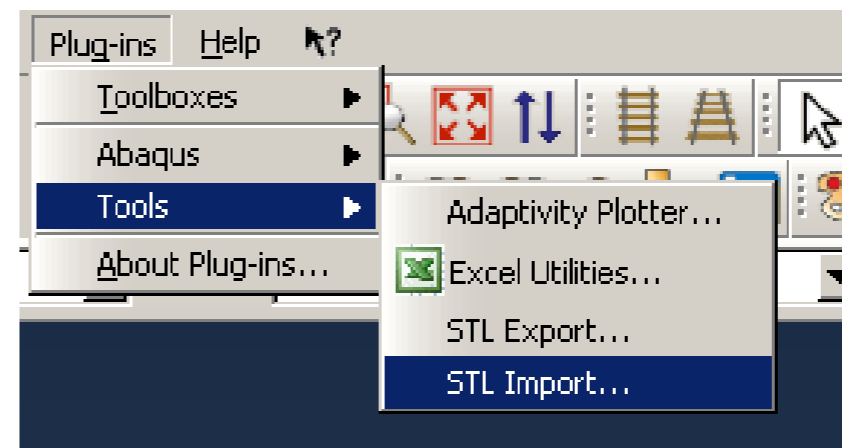
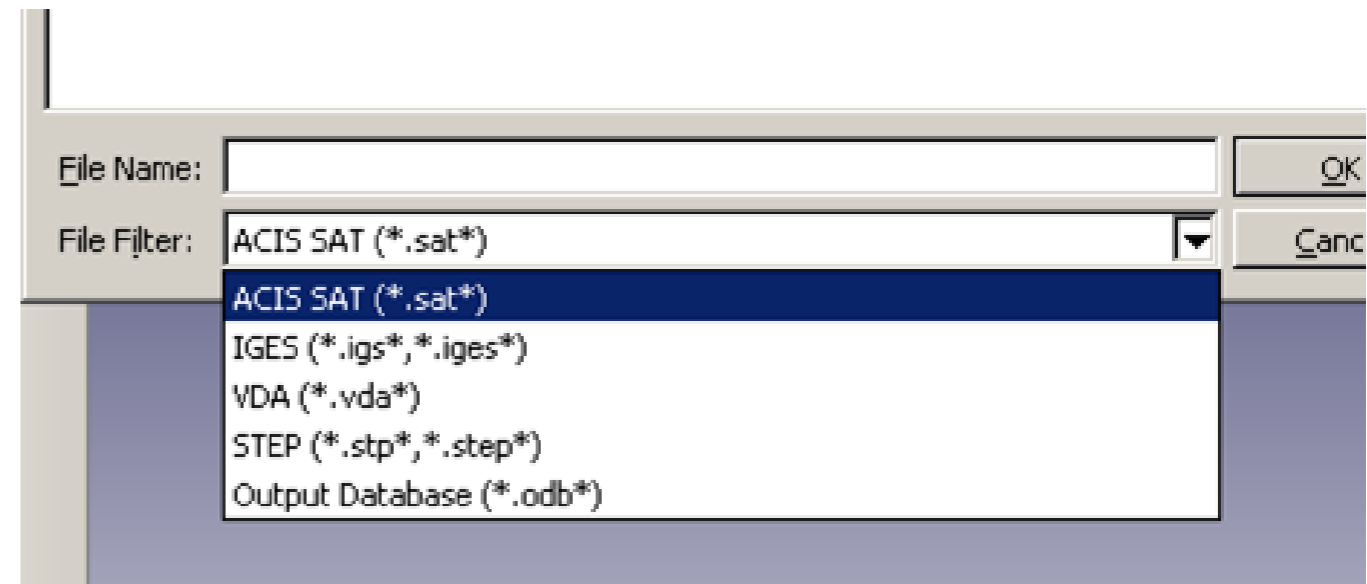
- Creating part (2D or 3D)
 - Sketch (or import) the geometry of a part
- Material definition
 - Select e.g. a linear elastic material
- Define a section with a material
- Assign the section to a part
- Assembling parts to a model
- Configure the analysis
 - Initial and step-1 (analysis type)
 - Boundary condition and load (e.g. by using predefined sets)
 - Define output: history and field (size of output and output variable)
- Mesh the model on part or assembly level (independent/dependent)
 - 1) mesh-control, 2) seed, 3) element-type, 4) meshing part
- Create and submit analysis job

Creating a parts

Make a sketch



"Import Part" from CAD etc.



Units in Abaqus

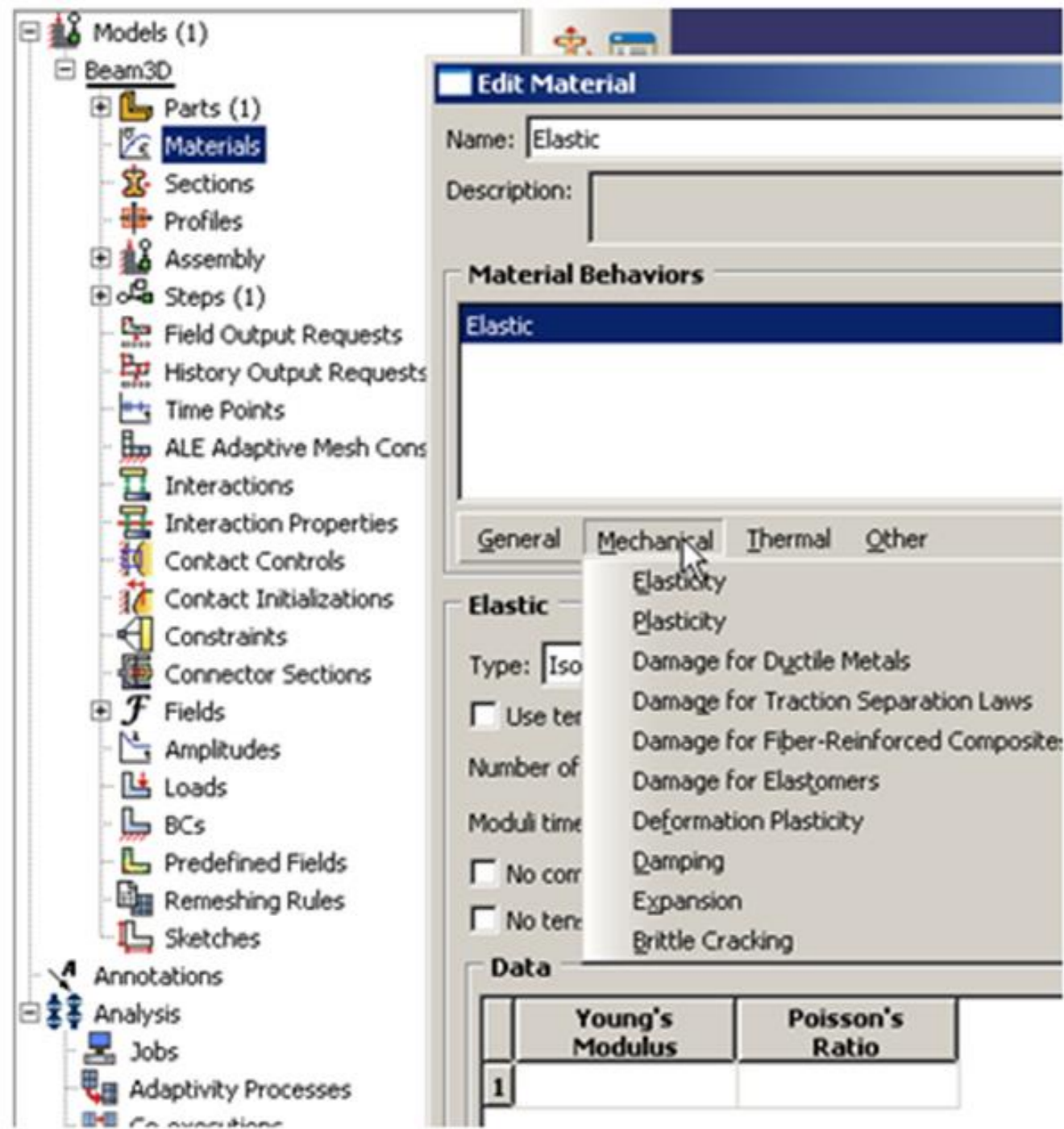
2.3.1 Units

Before starting to define this or any model, you need to decide which system of units you will use. ABAQUS has no built-in system of units. All input data must be specified in consistent units. Some common systems of consistent units are shown in Table 2–1.

Table 2–1 Consistent units.

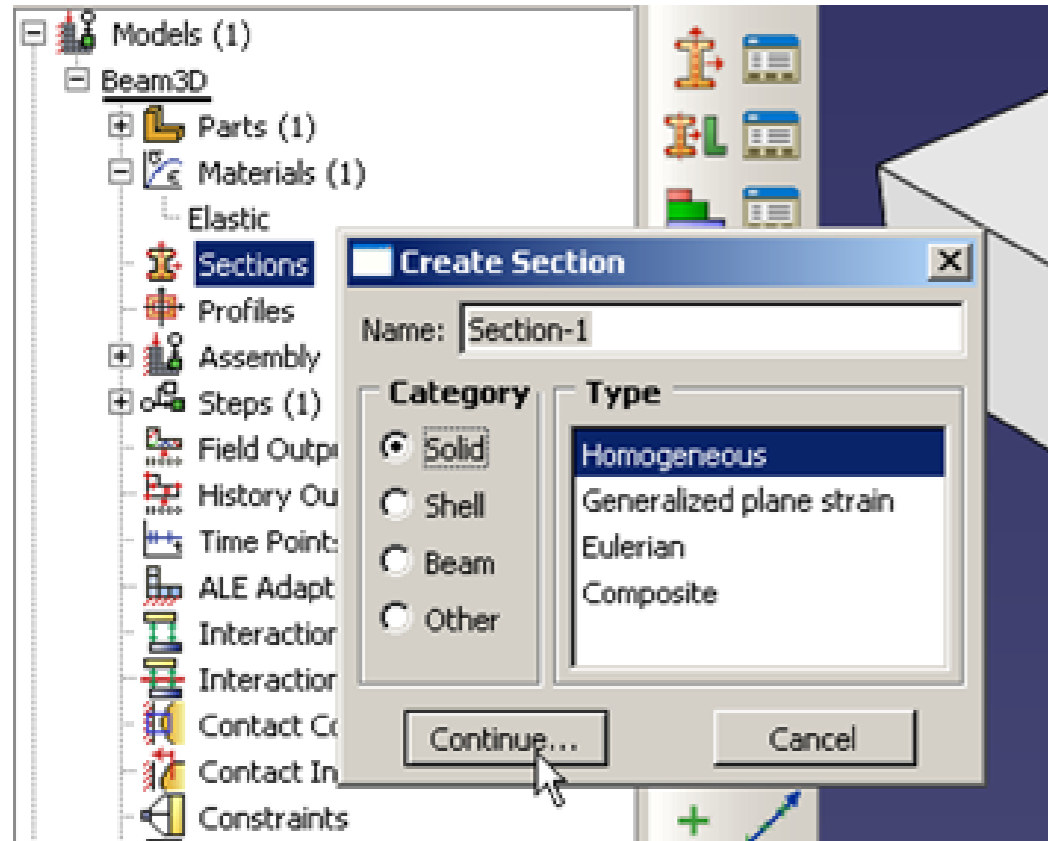
Quantity	SI	SI (mm)	US Unit (ft)	US Unit (inch)
Length	m	mm	ft	in
Force	N	N	lbf	lbf
Mass	kg	tonne (10^3 kg)	slug	$\text{lbf s}^2/\text{in}$
Time	s	s	s	s
Stress	Pa (N/m^2)	MPa (N/mm^2)	lbf/ft^2	psi (lbf/in^2)
Energy	J	mJ (10^{-3} J)	ft lbf	in lbf
Density	kg/m^3	tonne/mm^3	slug/ft^3	$\text{lbf s}^2/\text{in}^4$

Material definition

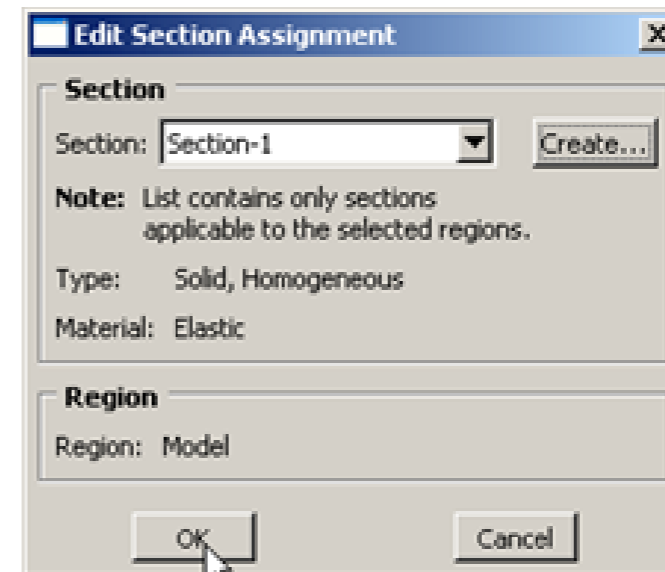
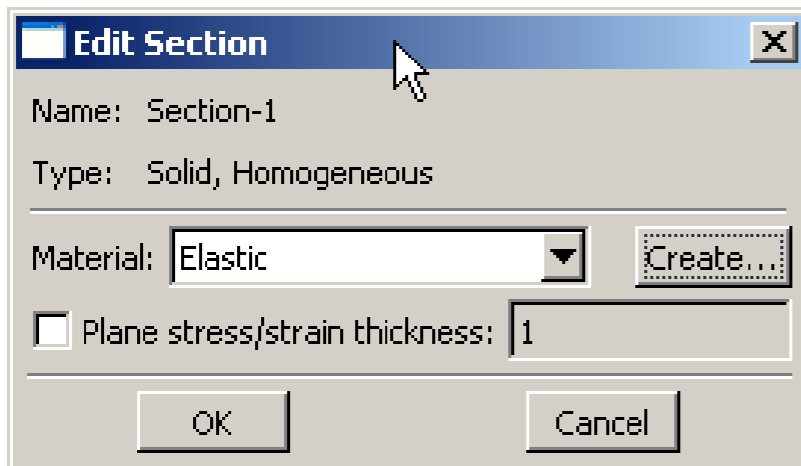
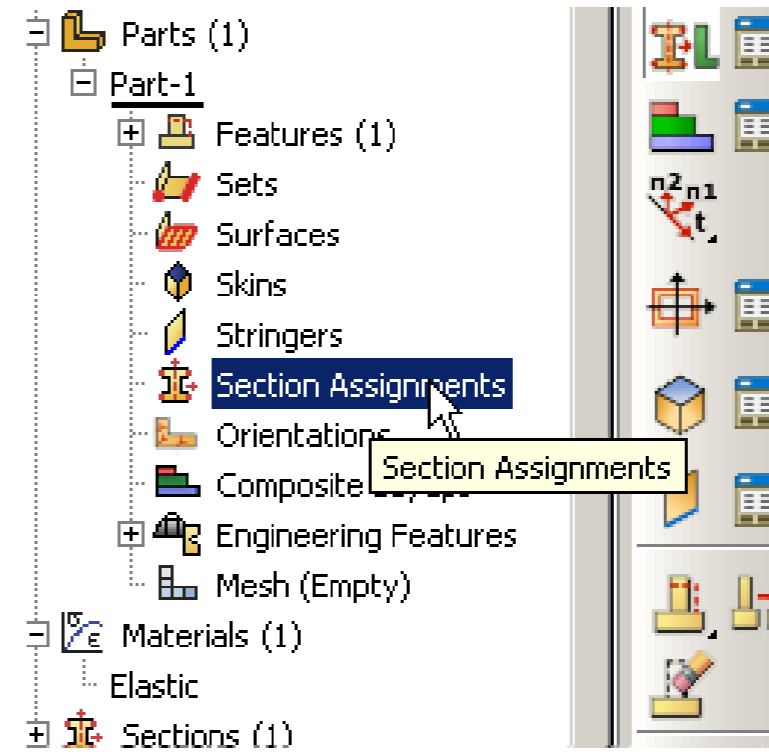


Topic 2:
Will focus on this

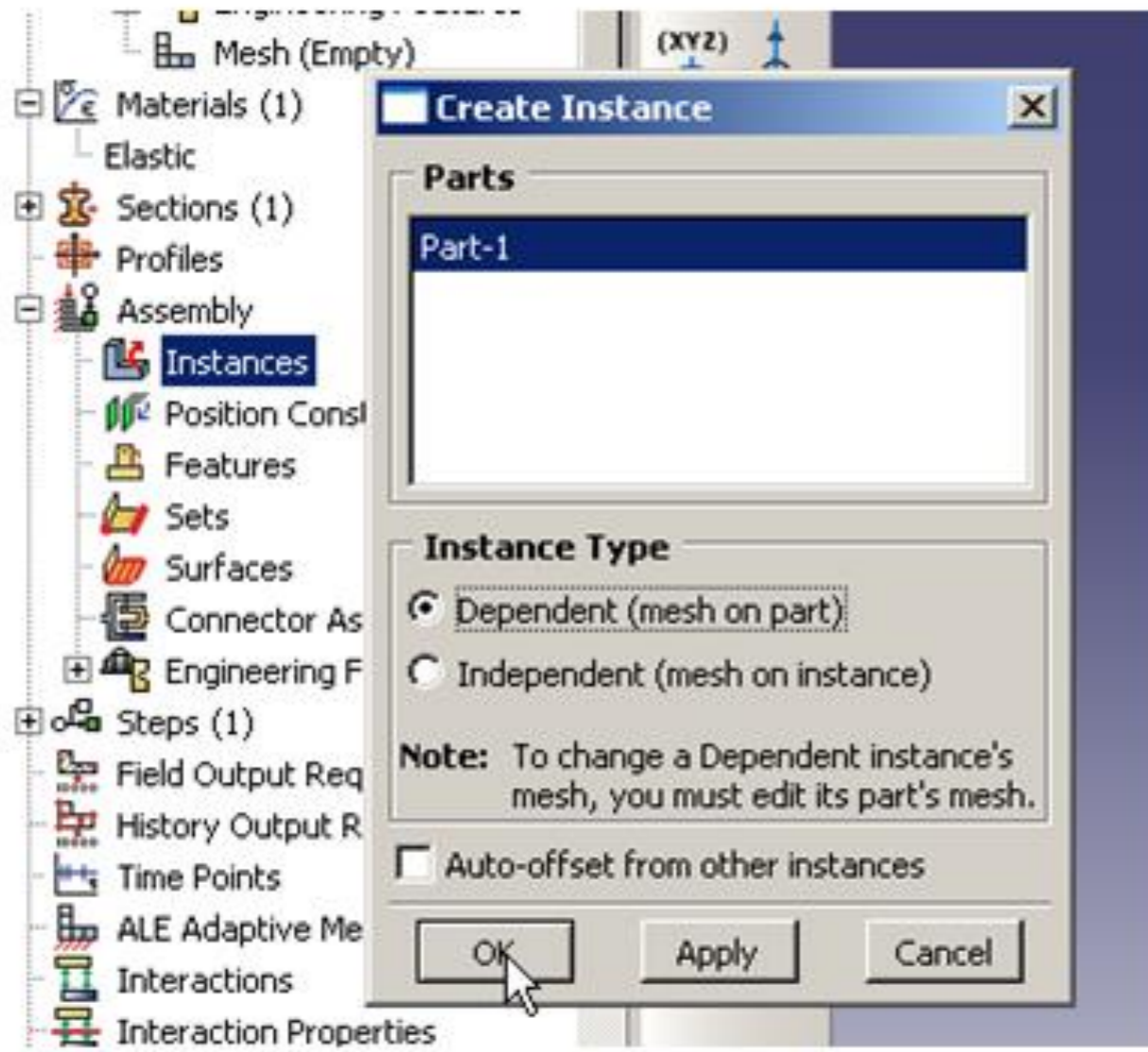
Create section



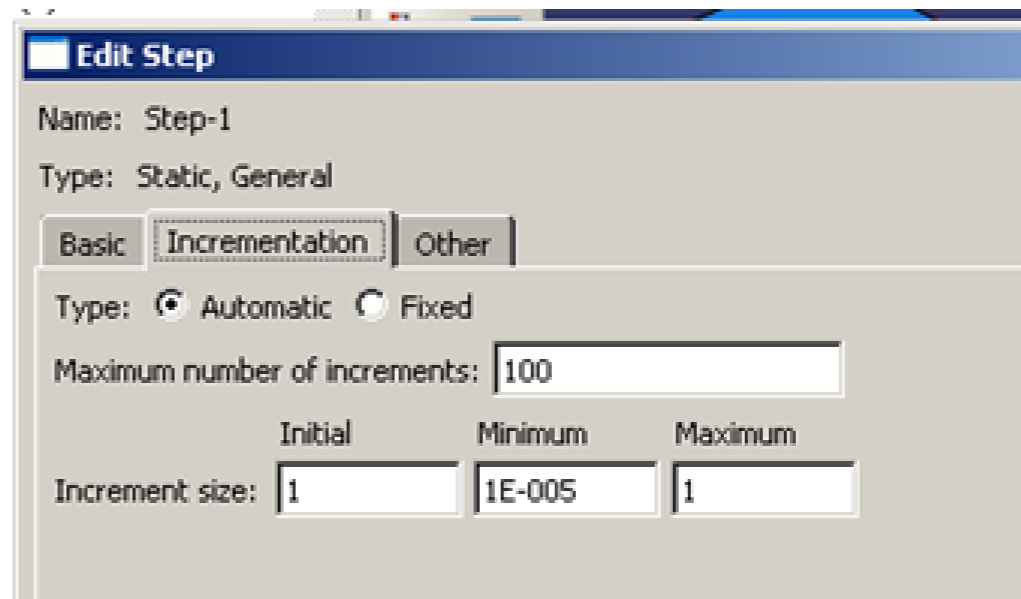
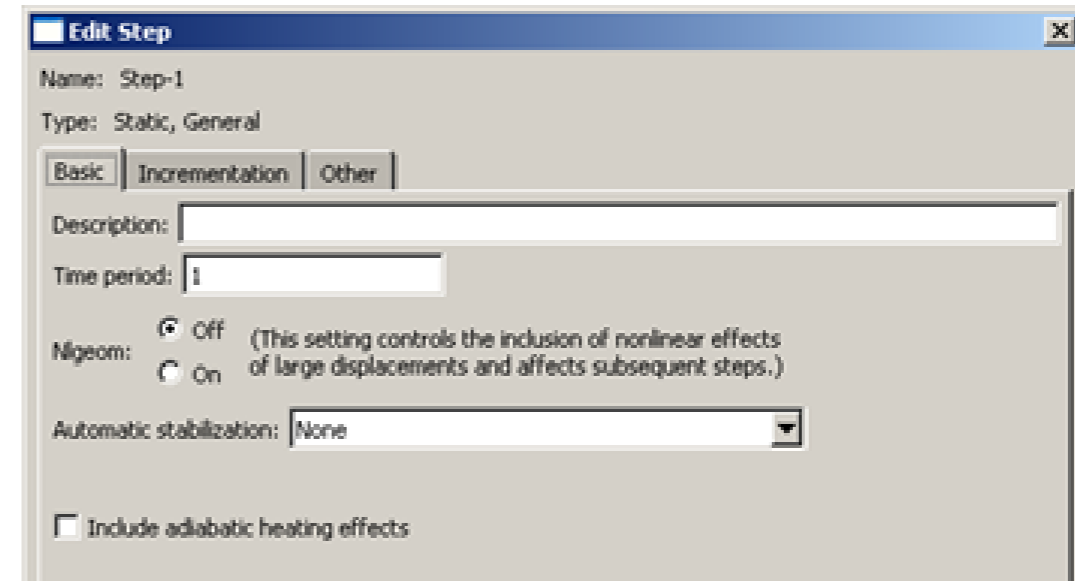
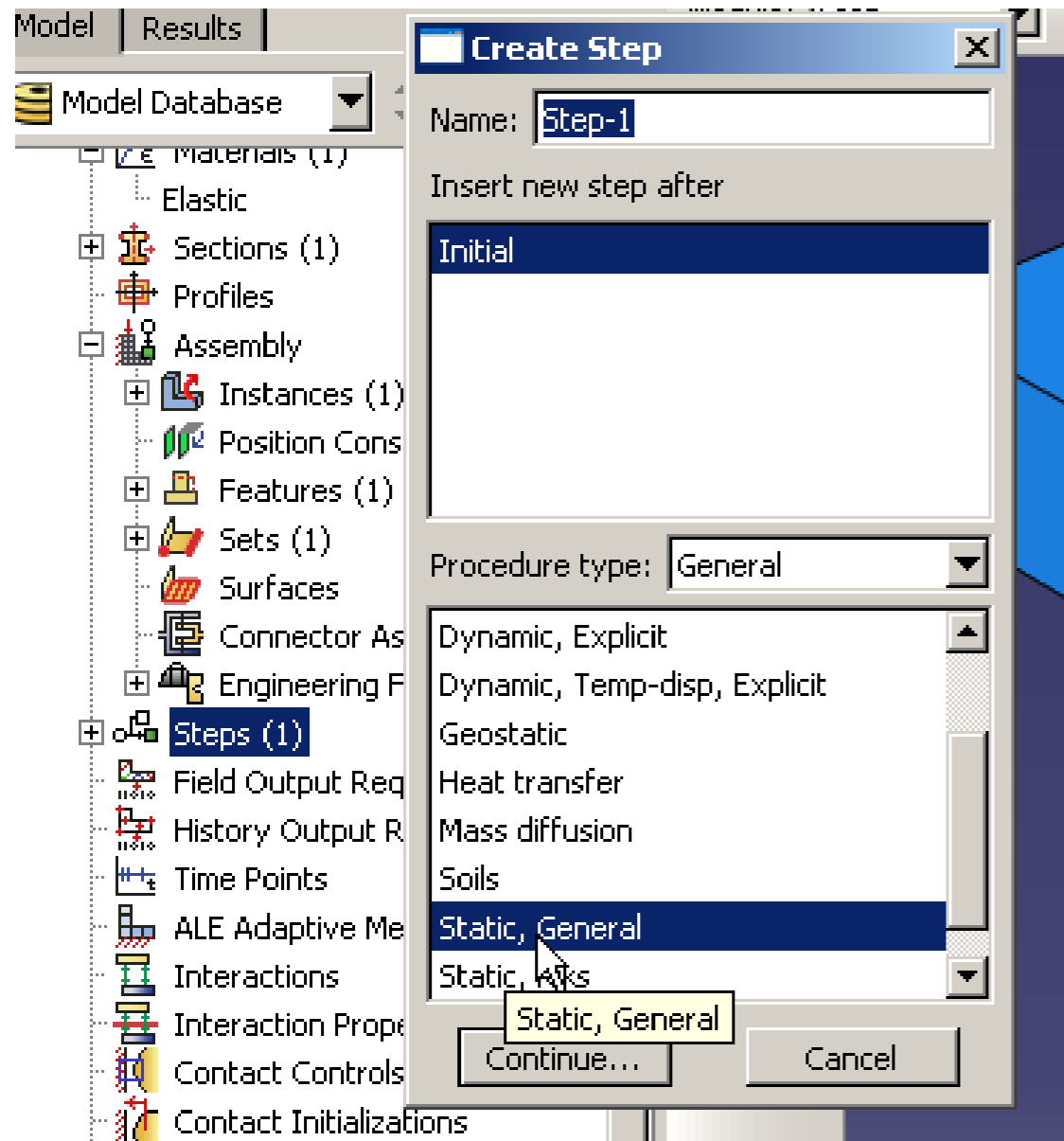
Assign section to part



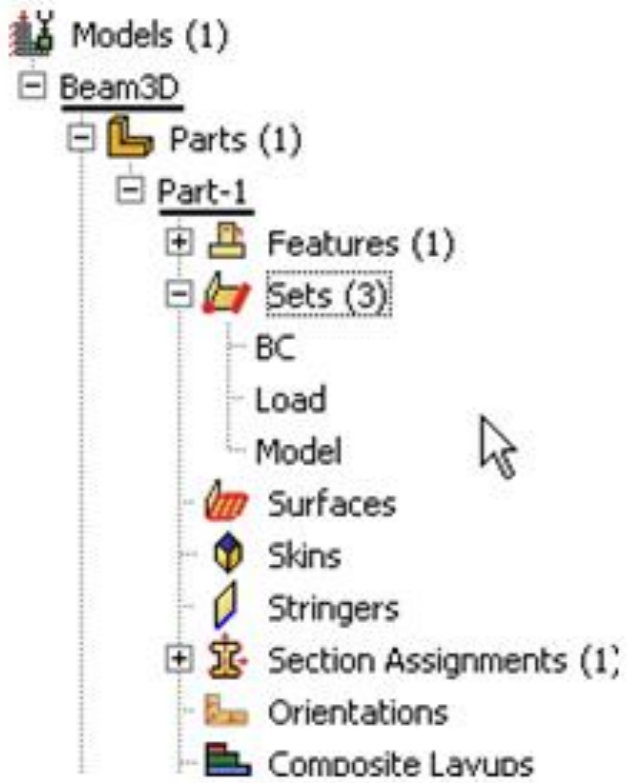
Assembly model



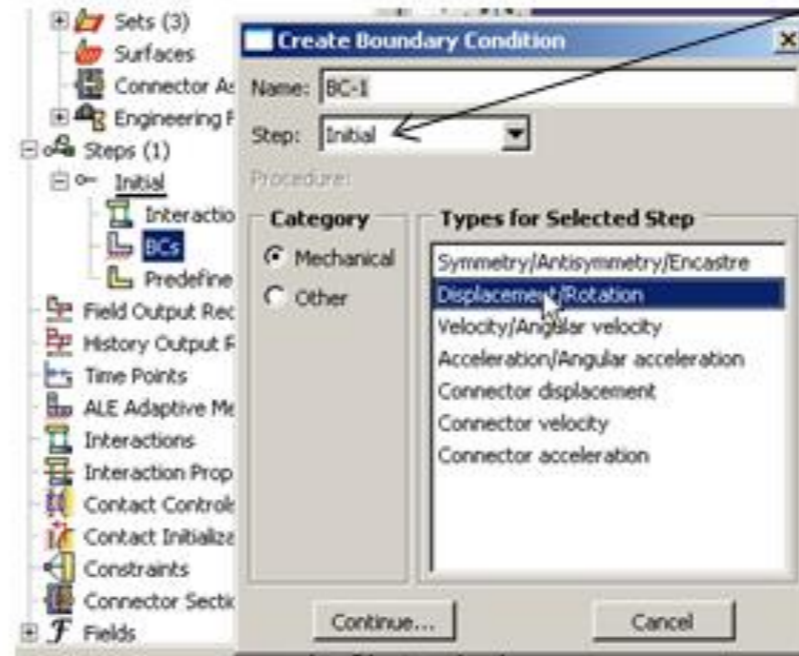
Configure step



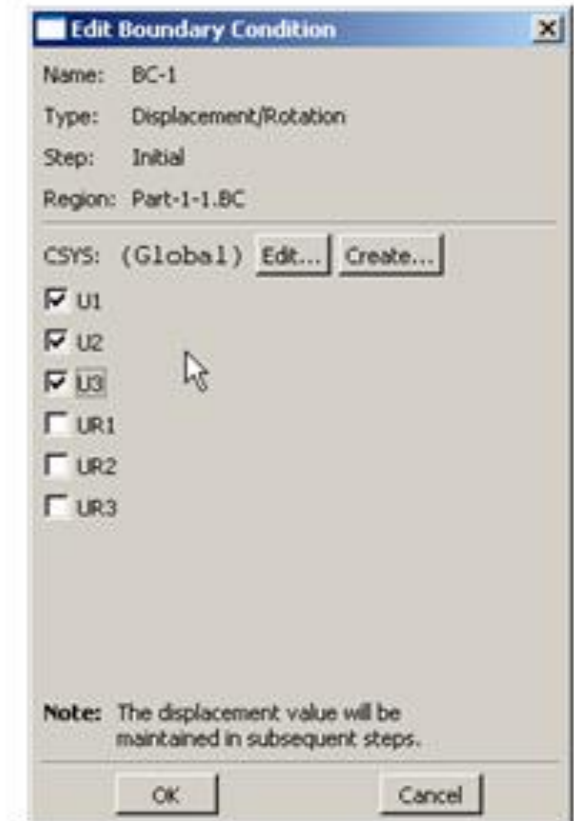
Define sets



Boundary

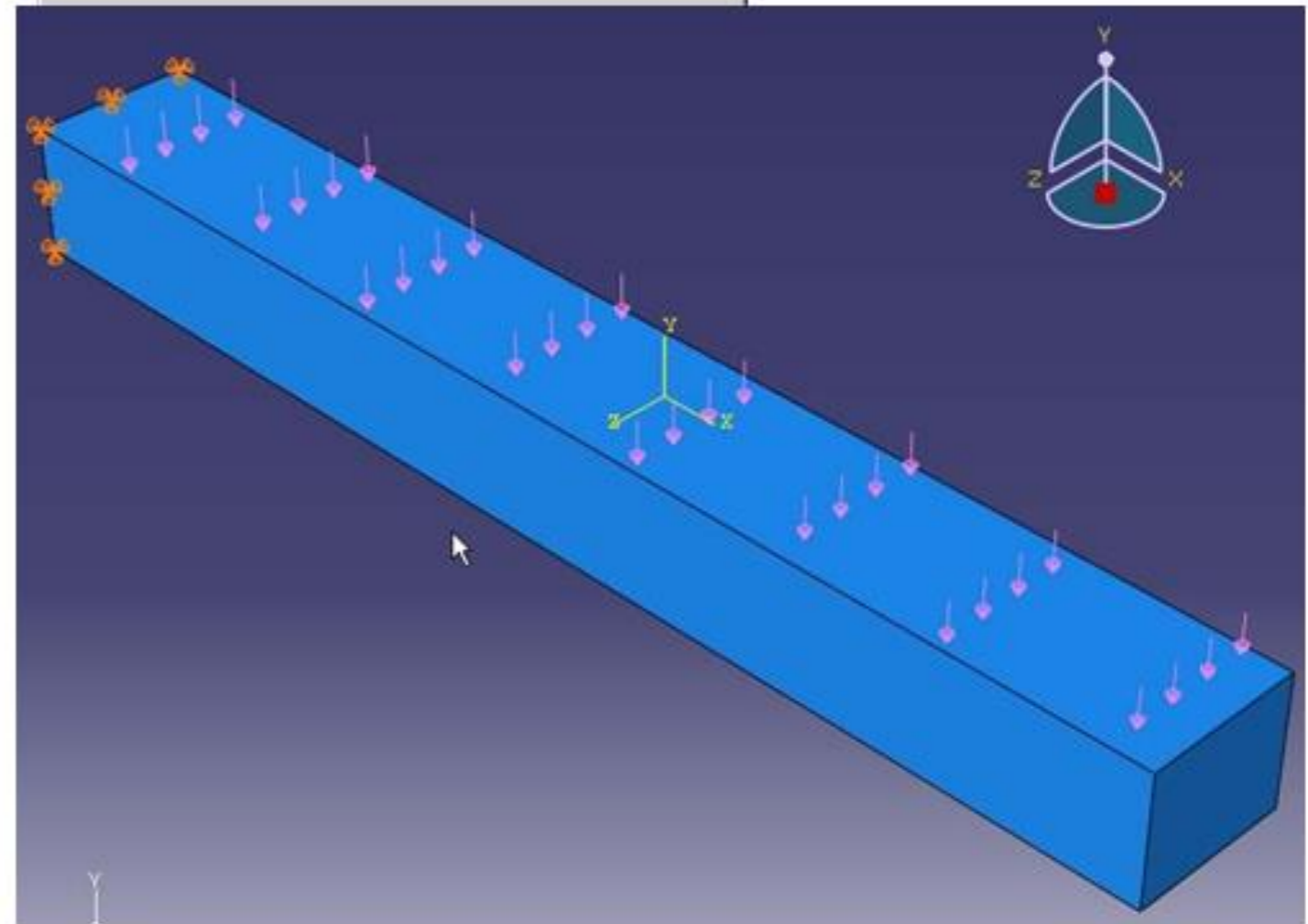
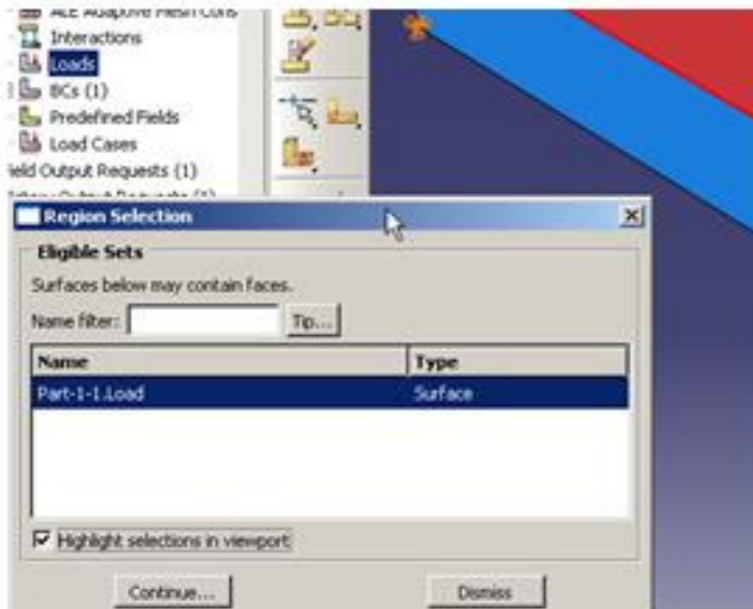
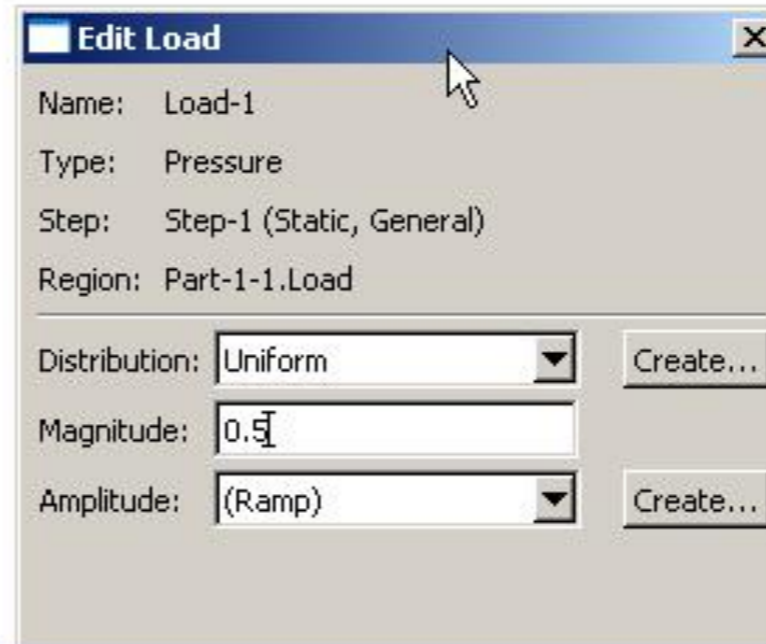
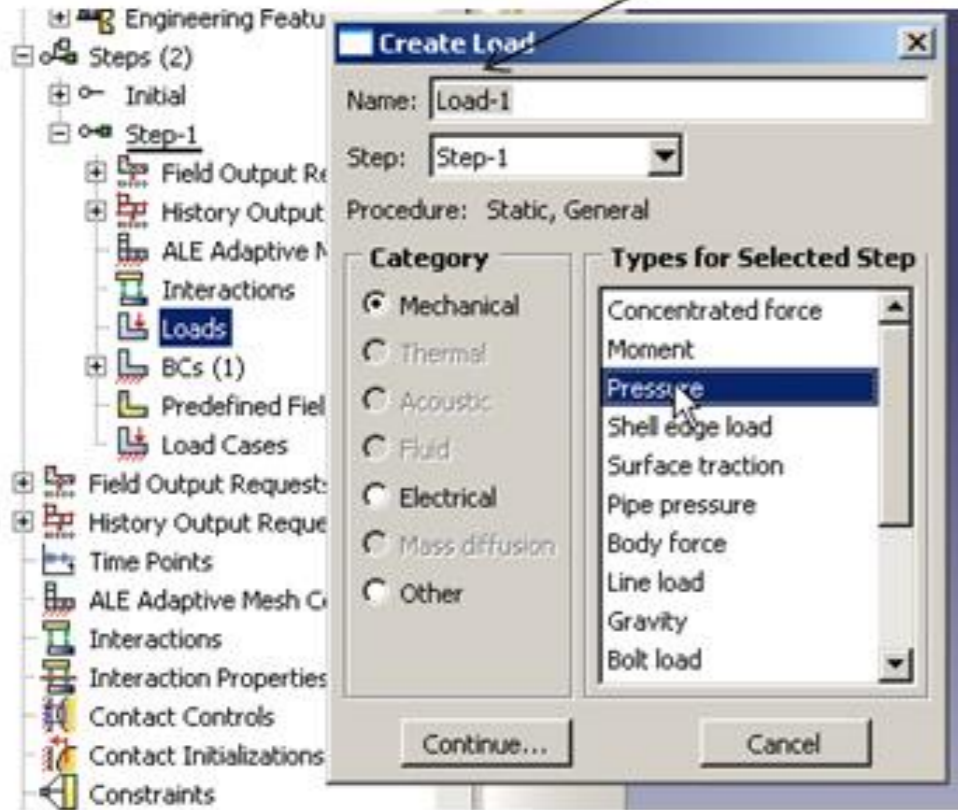


Note the step for the BC



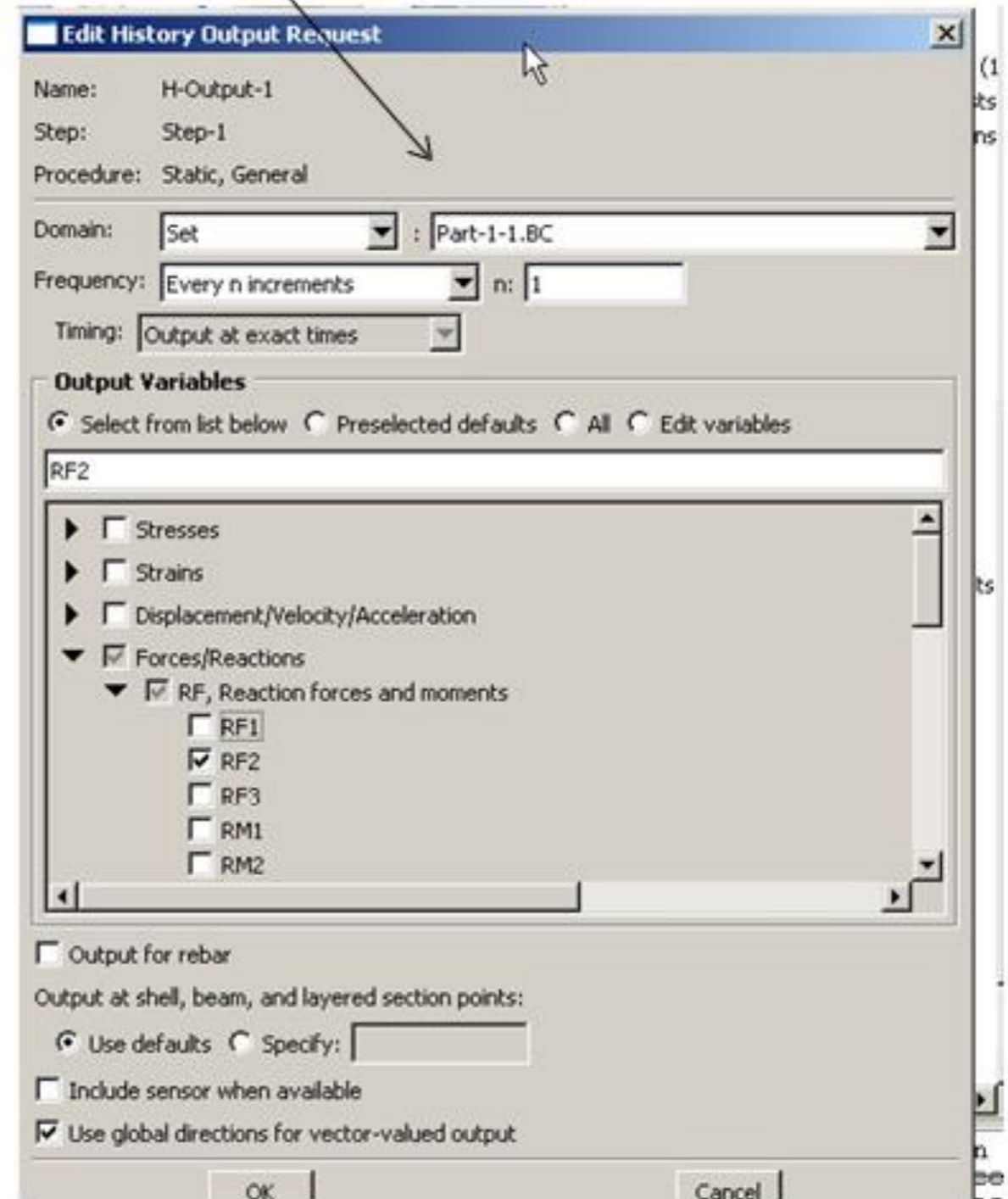
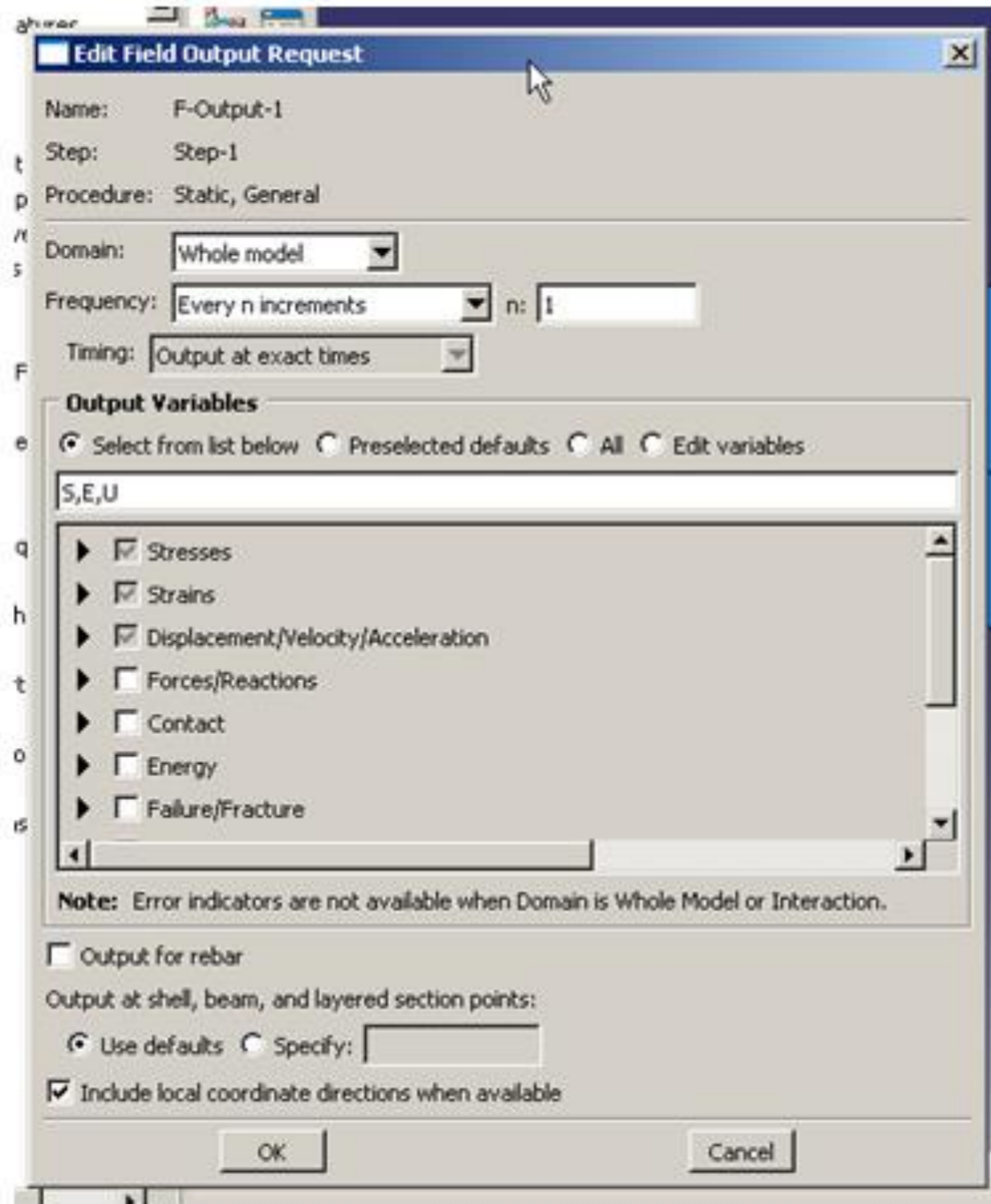
Load

Note the step for load or prescribed displacements

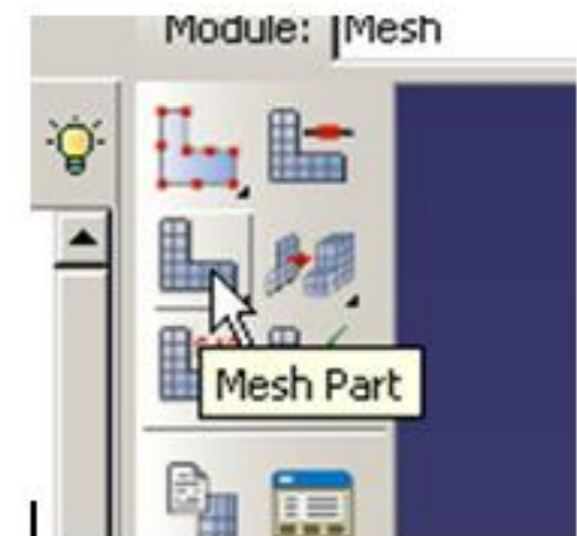
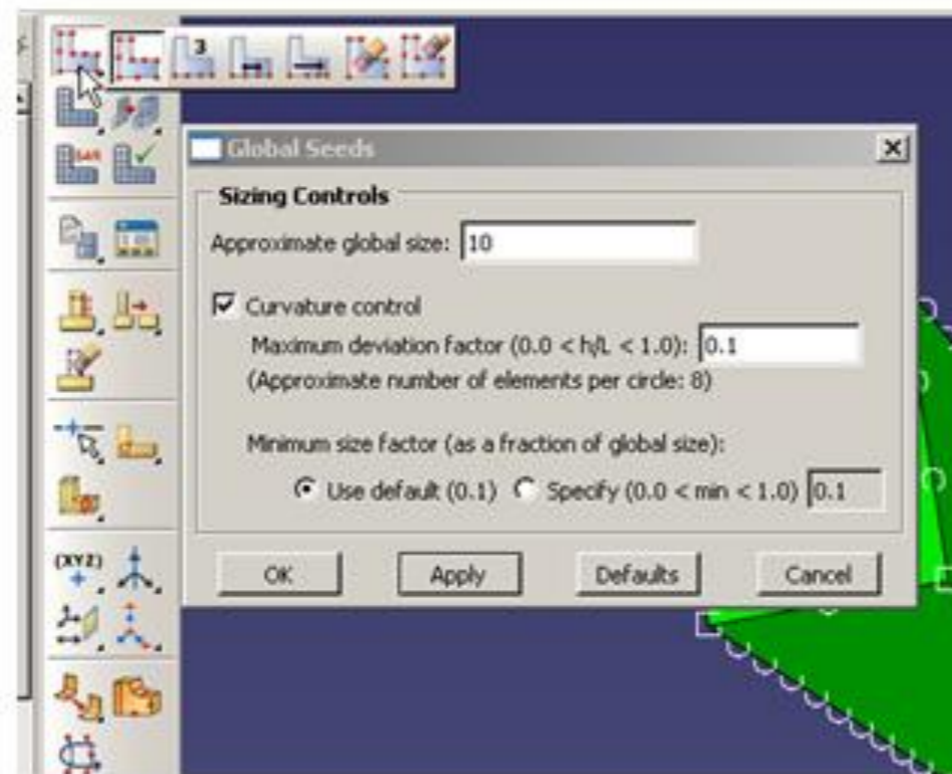
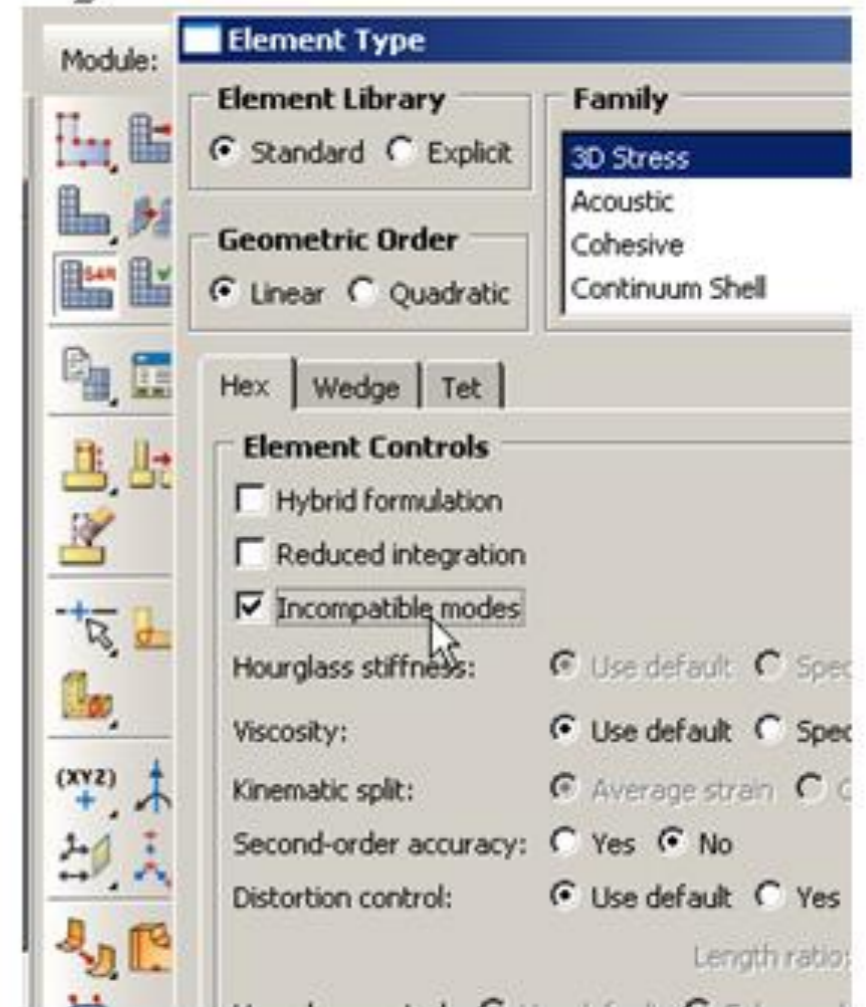
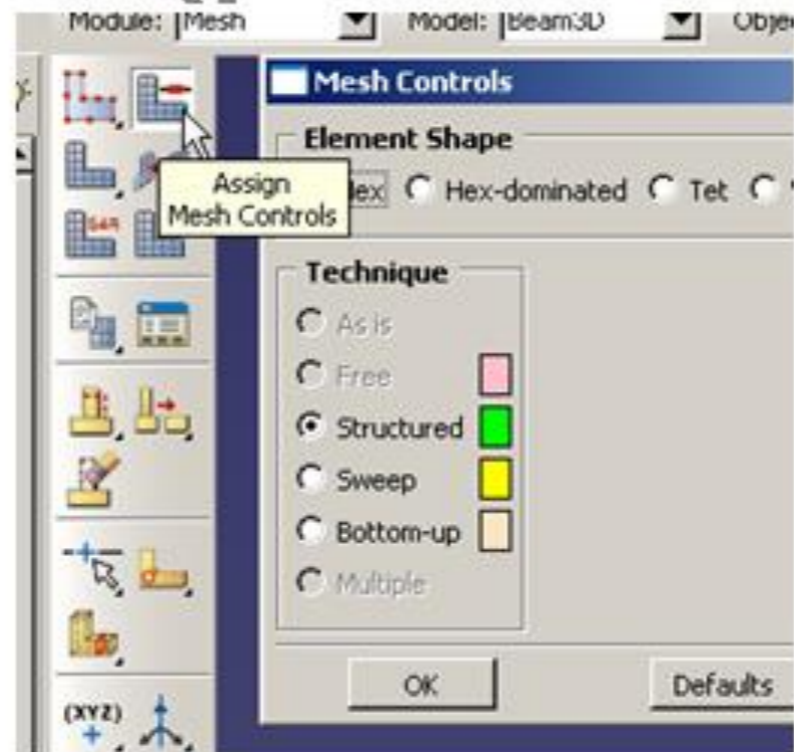
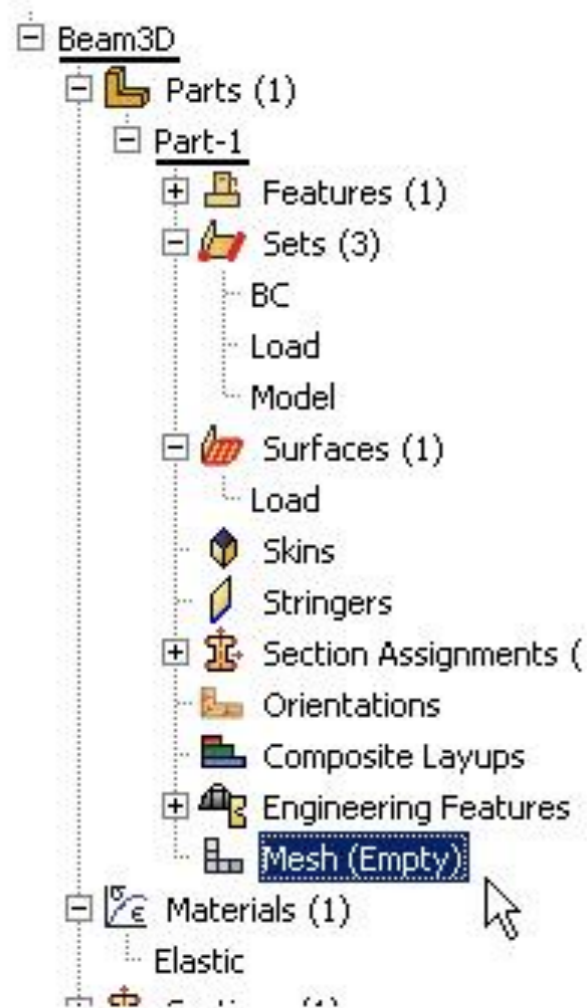


Set for extracting reaction force

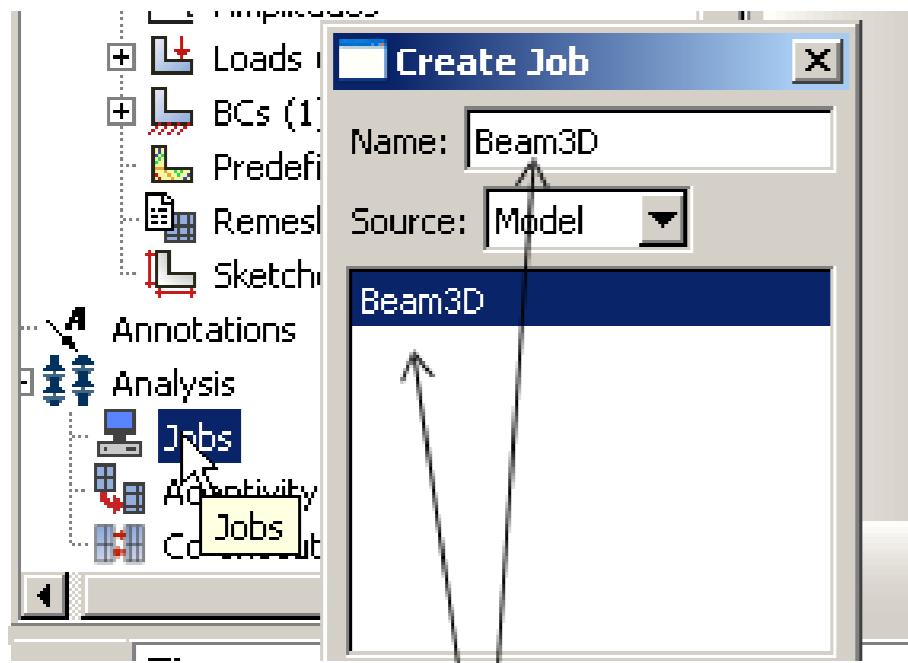
Define output



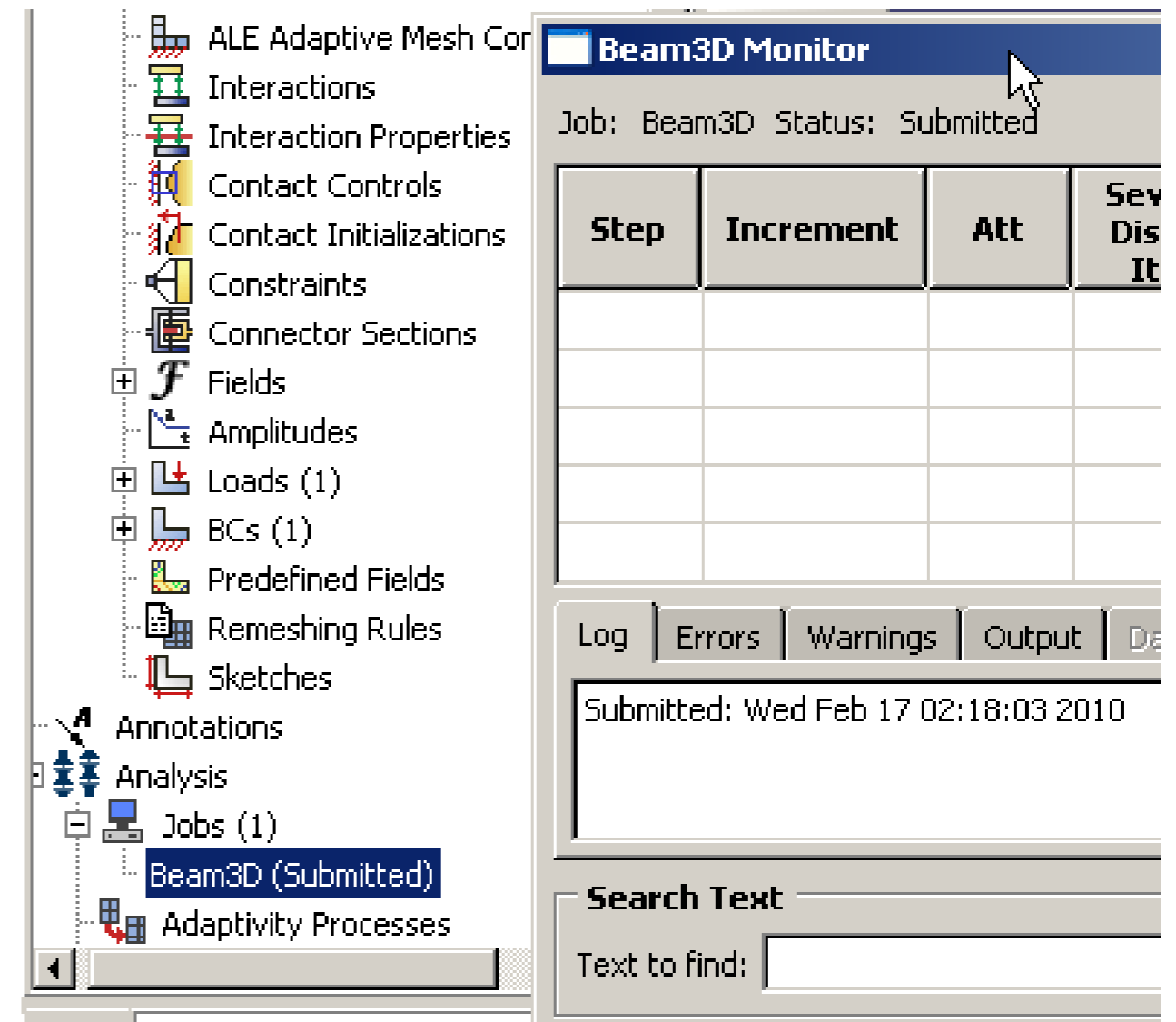
Mesh the model (part or instance)



Create job, submit and monitor



Use same name in order to link the output files with the model



Post-processing ...

Results Visualization

Types of plots available

- **Model plots** Fast plot mode
 Undeformed shape
 Deformed shape
- **Results plots** Contour
 Tickmark plots
 Symbol
 Material orientations
- **X-Y plots** Results versus analysis time
 Spatial variation (path plot)
 Operate on results
 Plot to screen or save to file

field data required

field or history data required

visualization toolbox



Modify the contour plot

The image displays the Abaqus/CAE interface with two dialog boxes open for modifying a contour plot. The 'Viewport Annotation Options' dialog is in the foreground, and the 'Contour Plot Options' dialog is in the background. Arrows indicate the specific settings being modified.

Viewport Annotation Options Dialog:

- General:** Show bounding box, Show min/max values
- Upper Left Corner:** % Viewport X: 2, % Viewport Y: 98
- Text:** Set Font..., Color: [white]
- Background:** Match viewport, Transparent, Other color
- Numbers:** Format: Fixed, Decimal places: 1

Contour Plot Options Dialog:

- Basic:** Note: User-defined interval values override the settings below.
- Min/Max:** Max: Specify: 100, Min: Specify: 0
- Auto-Computed Limits:** When auto-computing animation limits: Use first and last frame limits

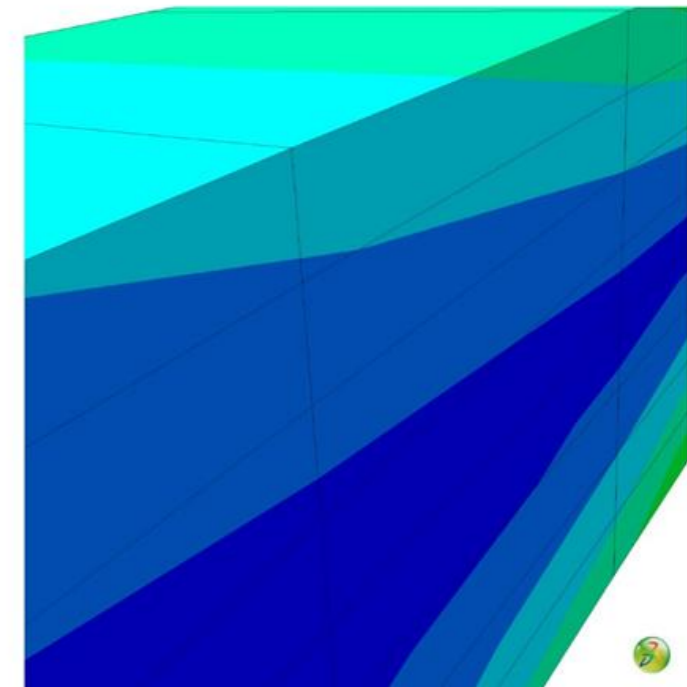
Color Scale Legend: S, Mises (Avg: 75%)

139.8
100.0
90.0
80.0
70.0
60.0
50.0
40.0
30.0
20.0
10.0
0.0

Tips and tricks

- Copy model before add new features
- Job-name = Model-name
- Keep related files in one folder e.g. by use the "start in" setup or open by click on cae-file in folder eventually by making an empty file with the cae-extension.
- After running the job, check the .dat, .msg, and .sta file or use monitor in job menu.
- Use screen dump and saving e.g. in PowerPoint for later reference
- Save x-y plot as rpt-files for later comparison plotting
- Use macros or scripting for repeating sequences.

**Saving models for word and powerpoints
-> File->Eport->3DXML**



Additional softwares

Notepad++: Good freeware program for editing files under windows

Matlab, Gnuplot or Python: For plotning xy-plots

Extra help

Simulia: <https://swym.3ds.com/#community:73/home>

Yahoo: https://groups.yahoo.com/neo/groups/ABAQUS_USERS/info

Facebook: <https://www.facebook.com/groups/2244871379/>

LinkedIn: <https://www.linkedin.com/groups?home=&gid=3164622>