

A”

Aalto University
School of Engineering

COE-C2004 - Materials Science and Engineering

Exercise 3

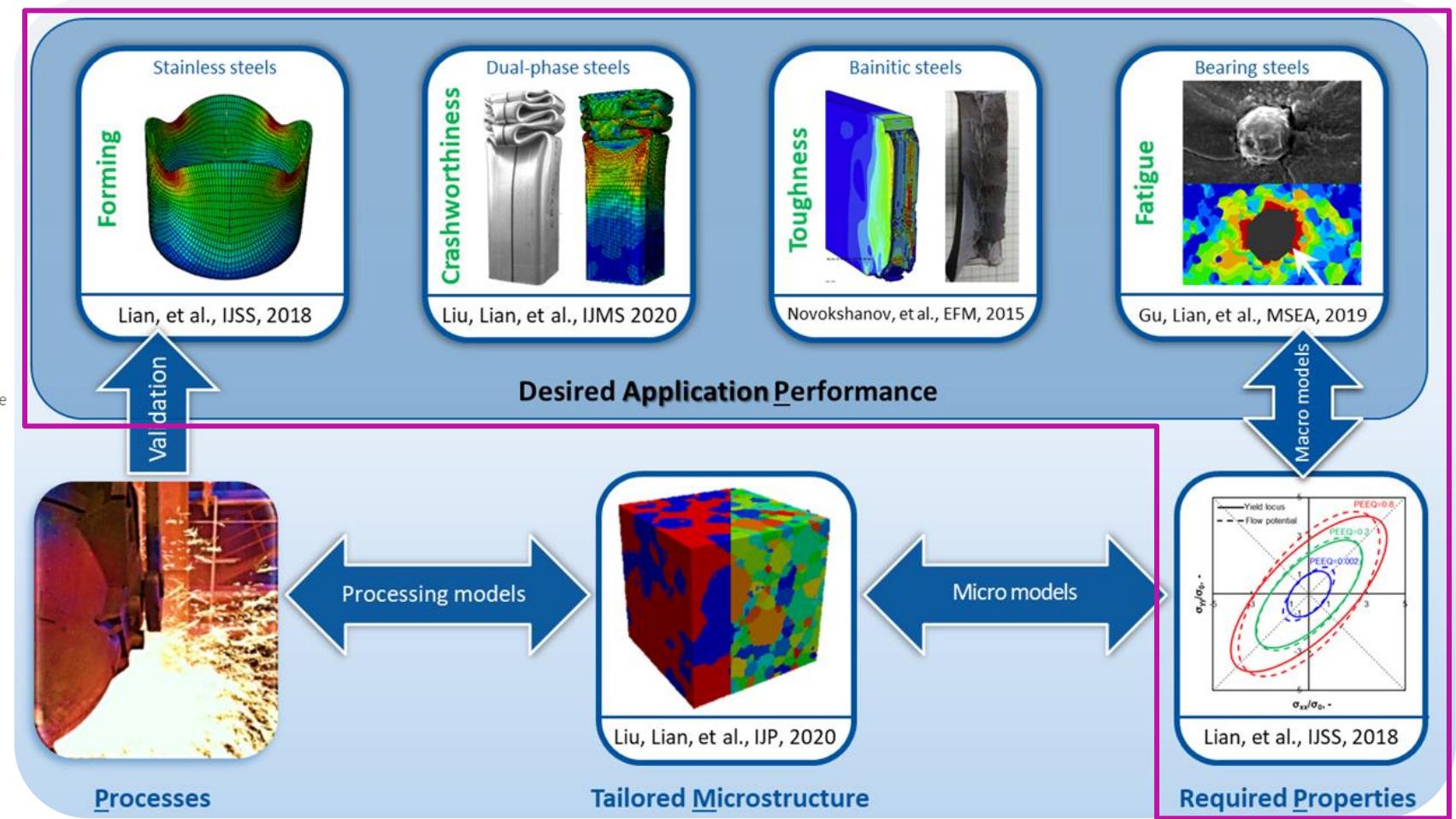
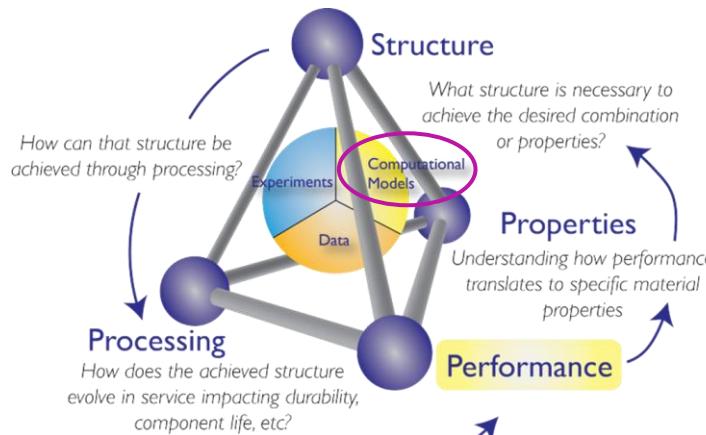
Prof. Junhe Lian

Wenqi Liu, Rongfei Juan(Teaching assistant)

Outline

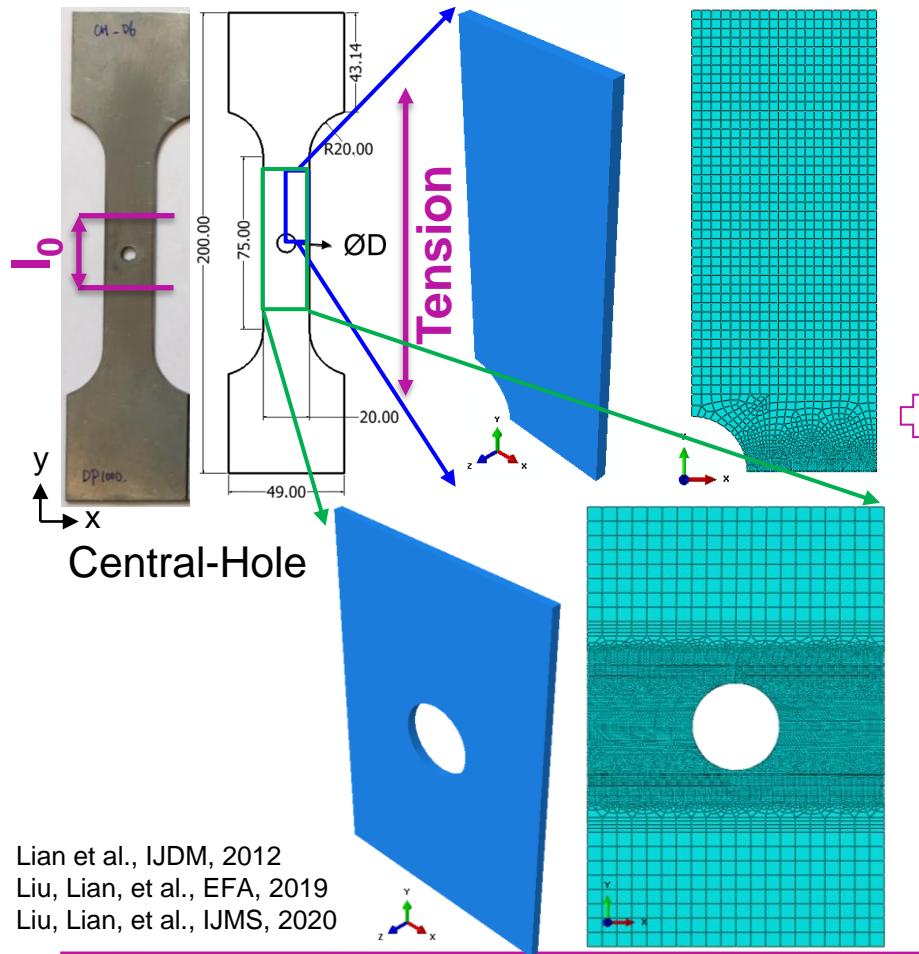
- Introduction
 - Macromechanics modeling
 - Tasks
- Software operations – Abaqus
- Feedback on Assignment
- Feedback & Questions

Integrated computational materials engineering (ICME)



Macromechanics modeling

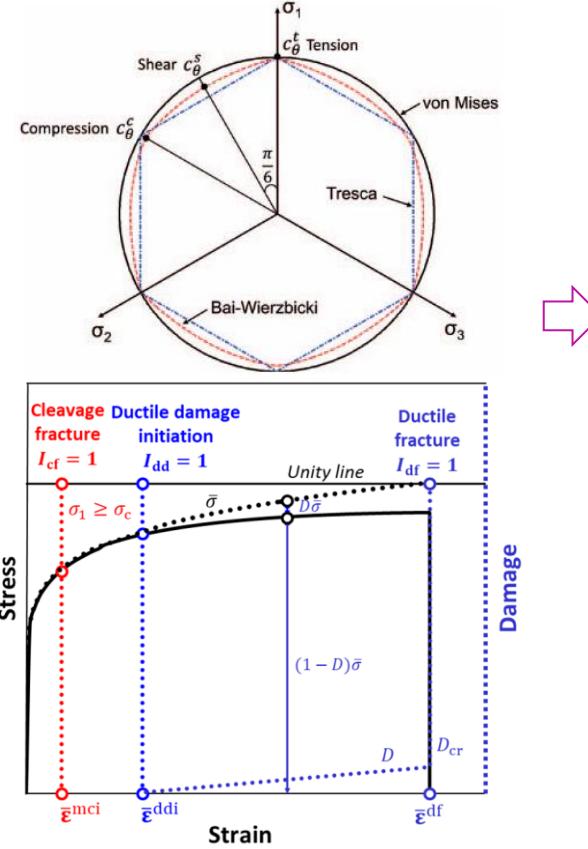
- Structure finite element modeling



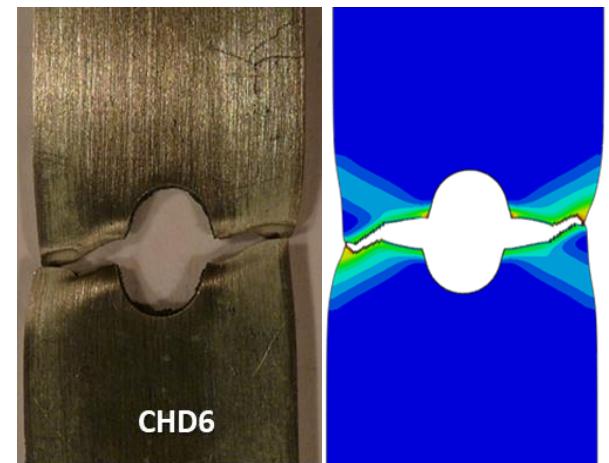
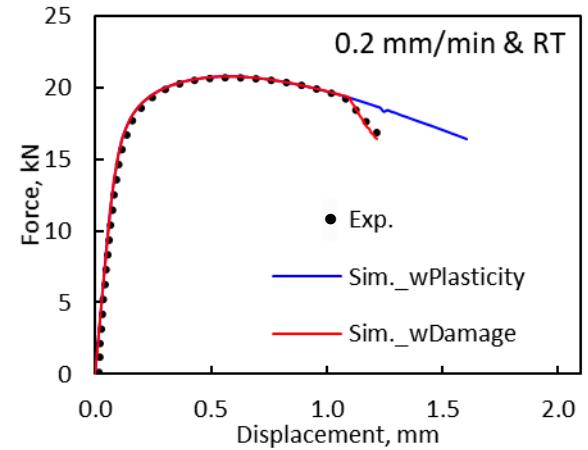
* Lian et al., IJDM, 2012
* Liu, Lian, et al., EFA, 2019
* Liu, Lian, et al., IJMS, 2020

- Material behavior models

- Plasticity (von Mises)
- Damage (eMBW/GTN)



- Results

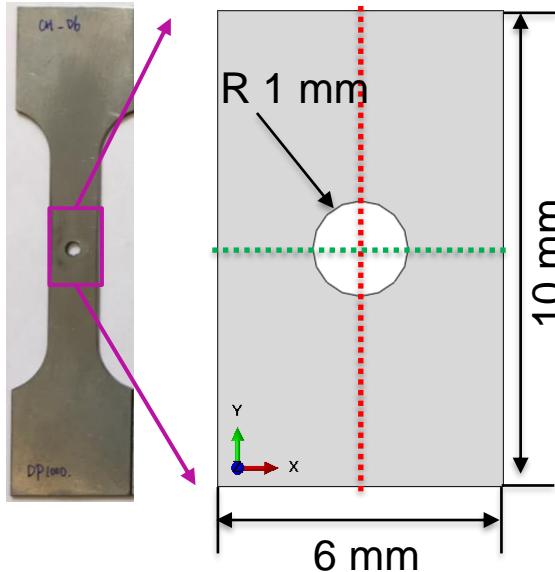


Tasks

Tasks

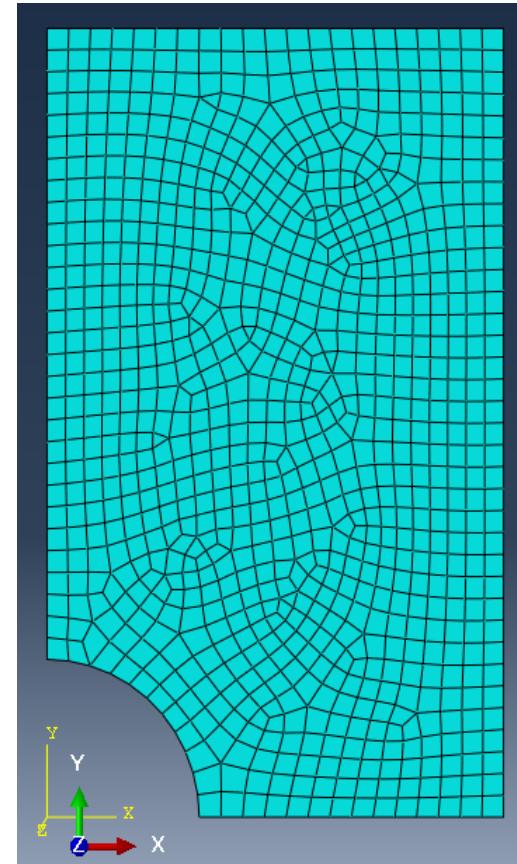
Central-Hole specimen

RD

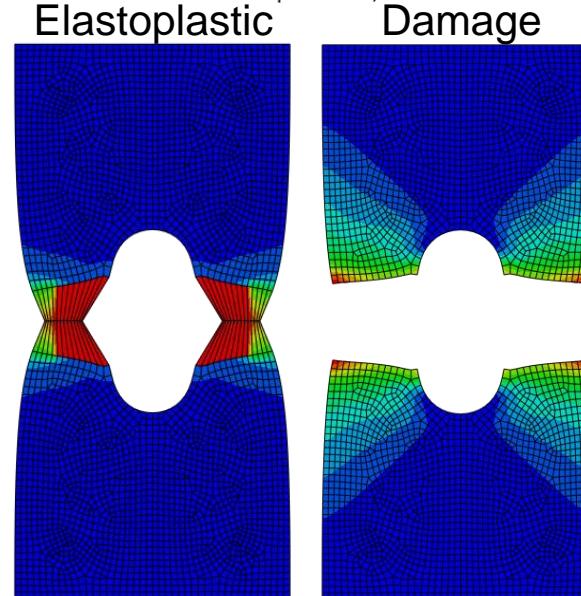
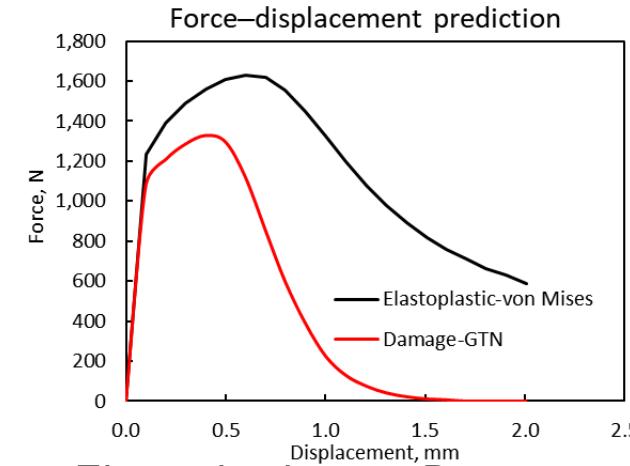


<https://edu.3ds.com/en/software/abaqus-student-edition>

- 2D 1/4 symmetry finite element model
- Uniaxial tension along RD with a pre-defined global strain of 20%



- Task1: Plasticity model - von Mises
Task2: Damage model - GTN (self learning)



Abaqus preparation

Software preparation

Input data download: 'E3flowcurve.txt' on MyCourses

Abaqus: free student version (<https://edu.3ds.com/en/software/abaqus-student-edition>)

Aalto VDI system: **mfavdi.aalto.fi**, or VMware Horizon Client **vdi.aalto.fi**, for more information, please refer to [Remote access to Windows classroom computers](#).

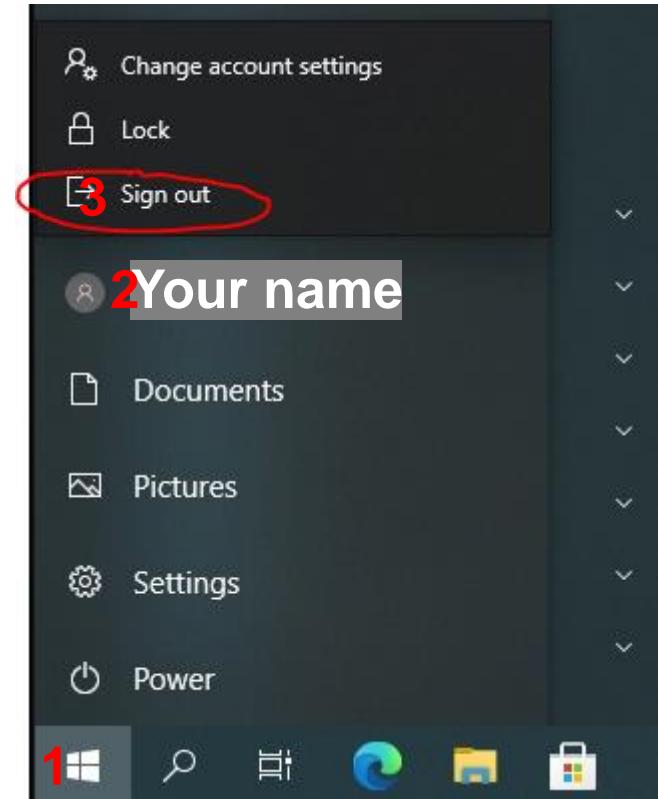
IMPORTANT! Please remember to do '**Sign Out**' after the session (NOT Disconnect). Click your username in Start and click 'Sign Out'.

Basic Rule: Please use DOT as the decimal separator, **NO COMMA!**

Contribution from Mr Binh Nguyen: For those who use win 10, go to control panel > region > additional settings > choose dot . as decimal separator.

The Abaqus student version has been installed in the VDI windows 10 3D, but it would be too slow if too many people are using VDI at the same time. You can also install the software on your own computer.

Make sure that you have downloaded the data file 'E3flowcurve.txt' from MyCourses.



Abaqus student version installation

← → C ⌂ edu.3ds.com/en/software/abaqus-student-edition ⌂ G ⌂ V ⌂ ...

Apps (<https://edu.3ds.com/en/software/abaqus-student-edition>)

3DEXPERIENCE® Edu

LEARN ONLINE GET SOFTWARE EXPLORE THE HUB BE RECOGNIZED PROJECT SHOWROOM JOB PLACE



ABAQUS STUDENT EDITION
COMPLETE SOLUTION FOR REALISTIC SIMULATION

FREE DOWNLOAD

f
Twitter

3DEXPERIENCE Edu ABAQUS Student Edition

The Abaqus Student Edition is available free of charge to students, educators, and researchers for personal and educational use. The Abaqus SE is available on Windows platform only and supports structural models up to 1000 nodes. The full documentation collection in HTML format makes this the perfect Abaqus learning tool both on campus or on the move.

Now you can have your own personal finite element analysis tool to use on or away from campus. Abaqus Student Edition is ideal for those using Abaqus as part of their coursework as well as for anyone wishing to become more proficient with Abaqus. All Students, Researchers, and Educators with a 3DEXPERIENCE ID associated with an academic institution are eligible for immediate download and access to tutorials and courseware... **free of charge!**

Abaqus student version installation



(<https://edu.3ds.com/en/software/abaqus-student-edition>)^E



ABAQUS Install instructions

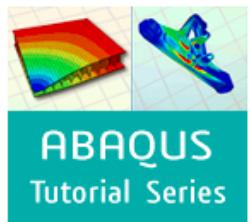


PDF 1.26 MB
Abaqus 2020 Student Edition Installation Instructions & known issues



Download

Install Abaqus according to the 'Install instructions', if you have questions, contact Abaqus service or Aalto IT.



ABAQUS Tutorials

To get started, ABAQUS Tutorials are available here



See more

Tutorials

Learning Resources

Tutorials and learning resources for Abaqus and other SIMULIA products are available at the

► [SIMULIA Learning Community](#)

Contact Abaqus for download problem

Download Issues

For download issues only (no other support for Abaqus), please contact us [here](#)

System requirements

ABAQUS Student Edition is not available on 32bits configurations

Note: The Microsoft Visual C++ 2010 SP1 Redistributable Package (x64) is required for successful execution of the Abaqus Student Editions.

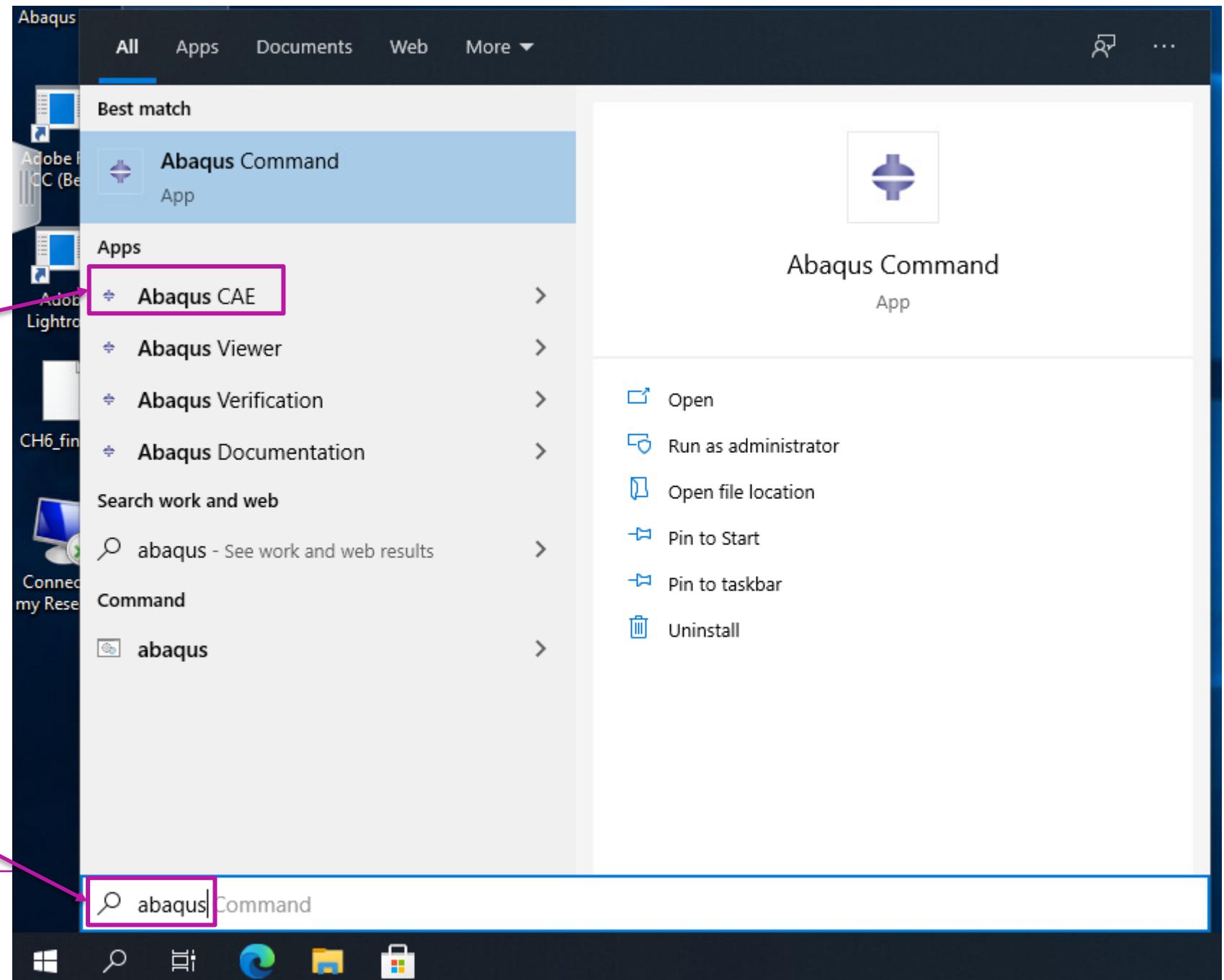
- Abaqus Student Edition 2020 (latest release): This version installs this package automatically and no additional steps are required.
- Abaqus Student Edition 2019: This release does not install this package automatically, and the user must download and install the Microsoft Visual C++ 2010 SP1 Redistributable Package (x64) using this link:
<https://www.microsoft.com/en-us/download/details.aspx?id=13523>. Failure to install this package will produce the following fatal runtime error with

Abaqus - Elastoplastic analysis (von Mises)

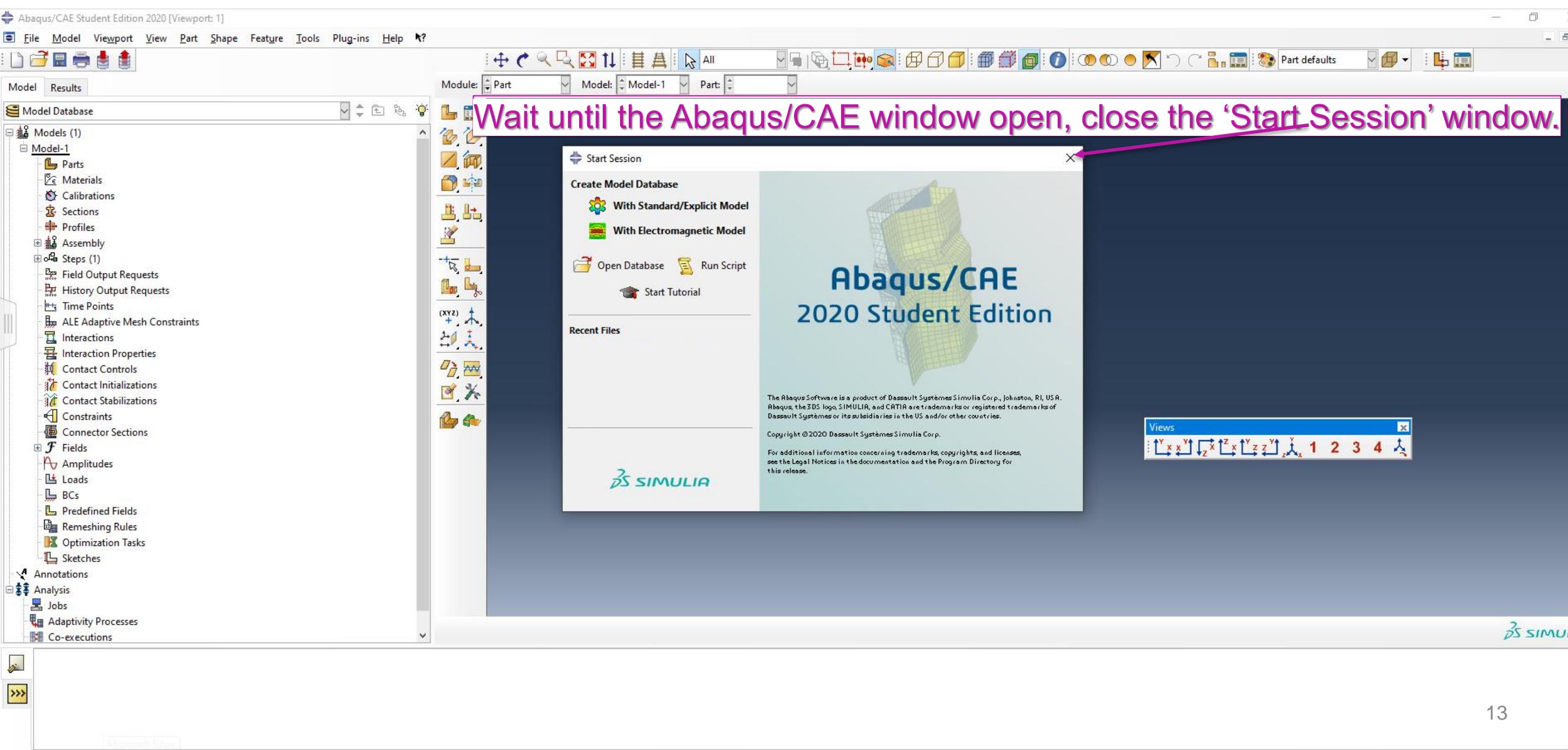
Start Abaqus

2. Choose and click 'Abaqus CAE'

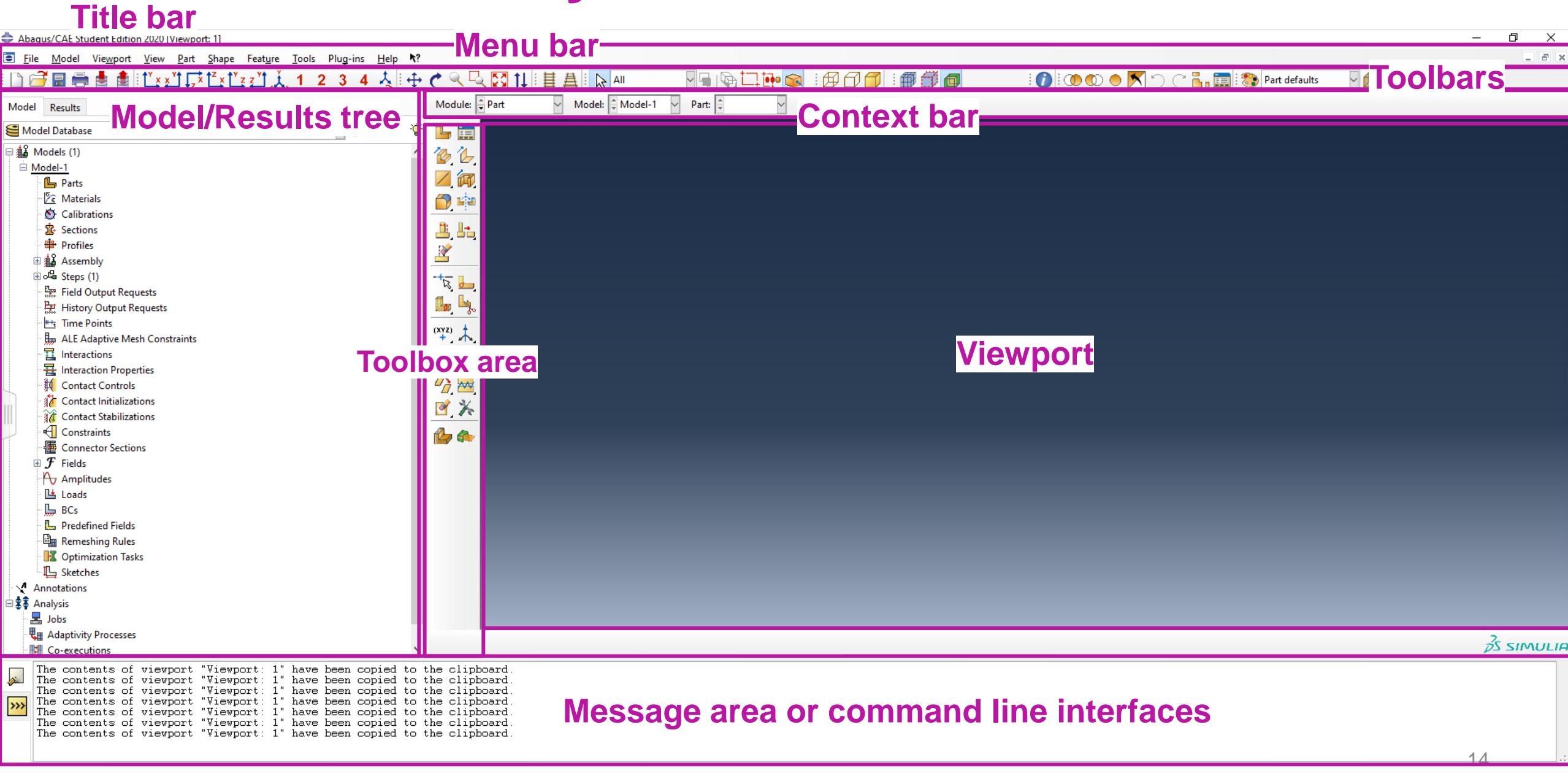
1. Search for
'abaqus'



Start Abaqus

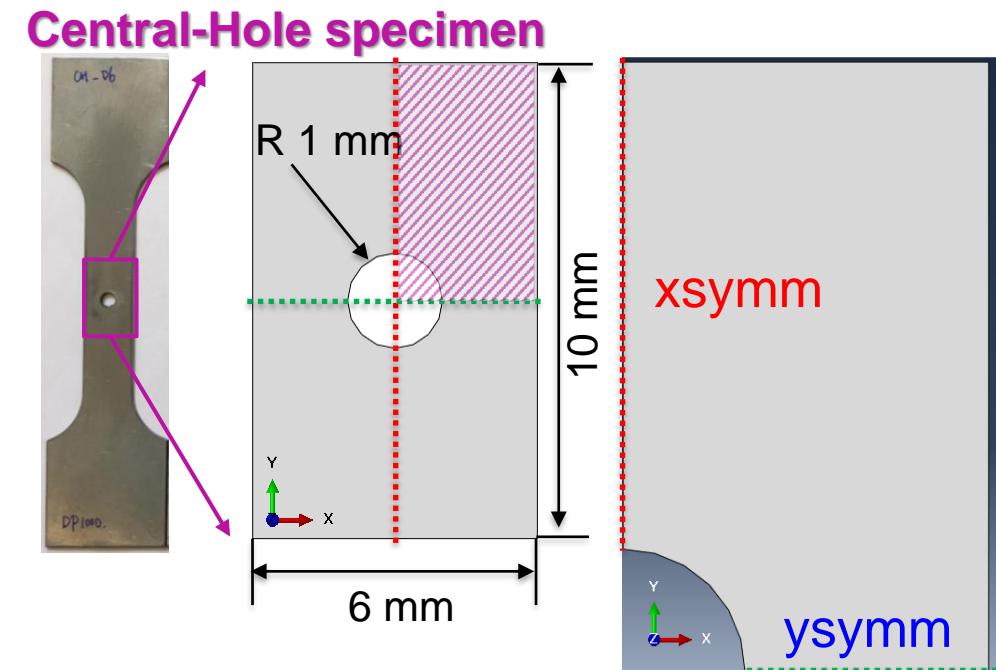


Layout of ABAQUS



Flow chart for establishing and operating a model in ABAQUS

- 1 Part building up according to the experimental specimen
- 2 Input material properties and assign them to the part
- 3 Assemble parts and set a proper step time for calculation
- 4 Assign loading and boundary conditions
- 5 Mesh the study region of the part
- 6 Create an input file and send it to calculate
- 7 Obtain an ODB file and extract useful data



only model the critical deformation zone:
a rectangle with length of 10 mm, width of 6 mm
a central hole with radius of 1 mm
1/4 symmetry specimen on 2D

Step 0: Set Abaqus work directory

1. Abaqus/CAE 2017 [Viewport: 1]

File Model Viewport View Part Shape Feature Tools Plug-ins Help ?

New Model Database Open... Network ODB Connector Close ODB Ctrl+O

2. Set Work Directory... Save Save As... Compress MDB... Save Display Options... Save Session Objects... Load Session Objects... Import Export Run Script... Macro Manager... Print... Ctrl+P Abaqus PDE... 1 Z:/.../MSEcourse2020/MSEE3.cae 2 C:/.../abaqus_temp/Job-GTN.odb 3 C:/.../abaqus_temp/Job-1.odb 4 C:/Users/.../31 Chao/8cubeRVE.cae Exit Ctrl+Q

Annotations Analysis Jobs Adaptivity Processes Co-executions Optimization Processes

Choose 'File' - 'Set Work Directory'

Set Work Directory

Current work directory: C:\programdata\abaqus_temp

New work directory: C:\programdata\abaqus_temp

Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.

OK Cancel

The default Work Directory, you can check your existed data file here.

3. You can also change it to your preferred work folder

Example:

Select a Work Directory

Directory: MSEcourse

MSE

4. OK

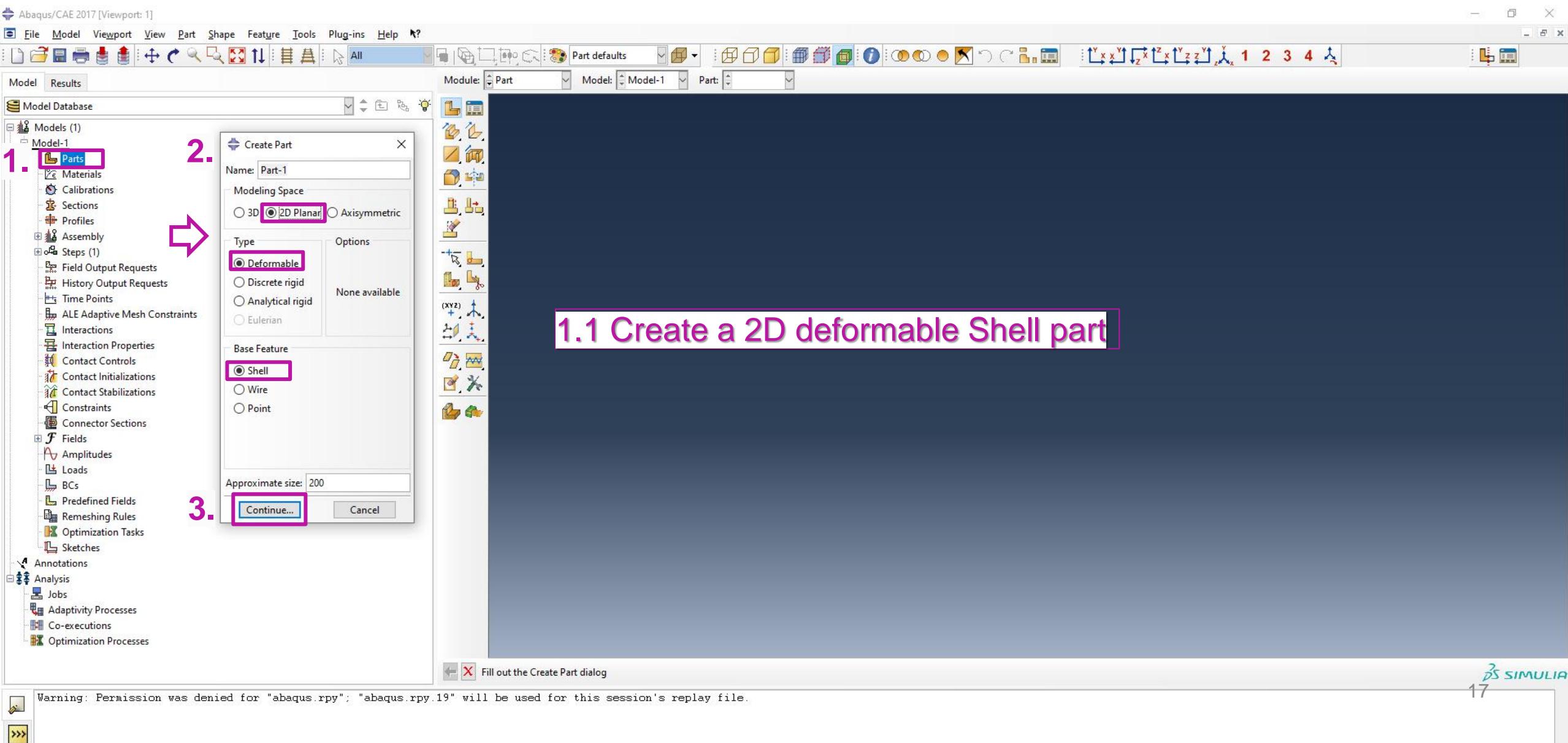
5. OK Cancel 16

Current work directory: C:\programdata\abaqus_temp

New work directory: /MSEcourse

Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.

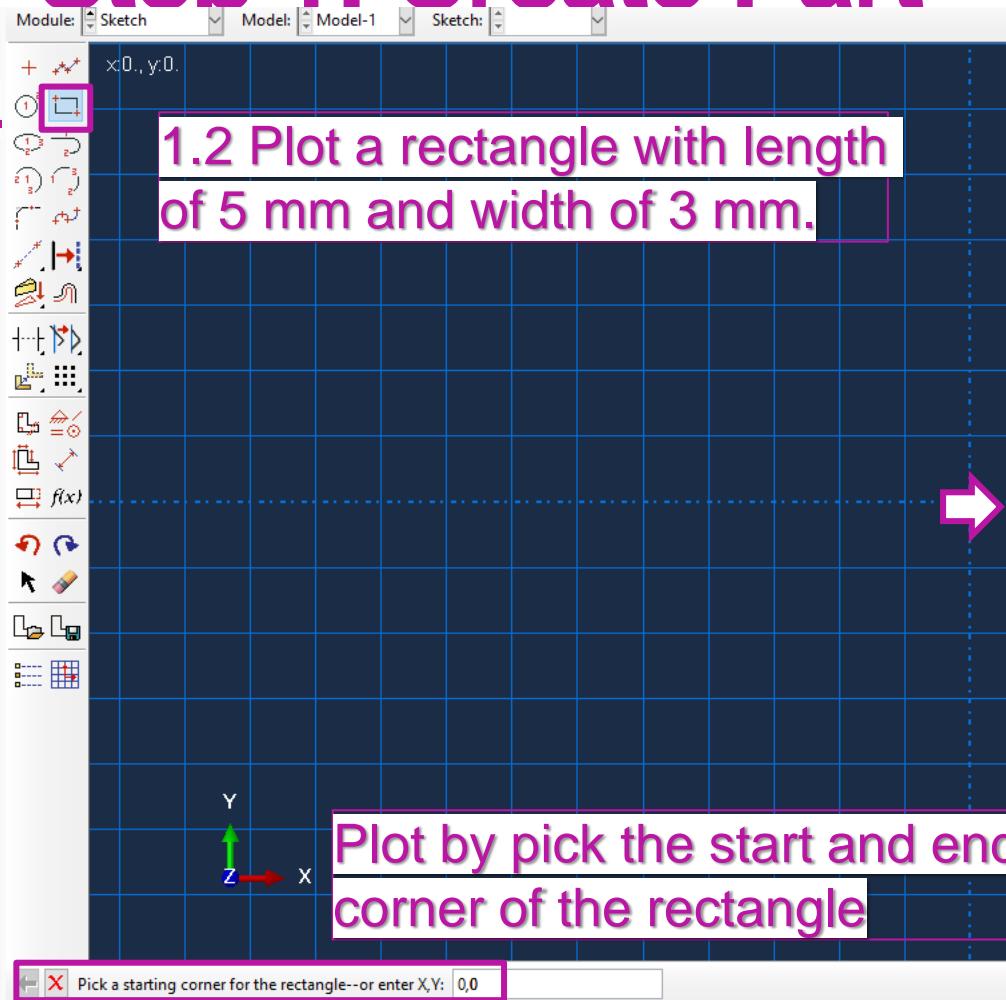
Step 1: Create Part



Step 1: Create Part

1.

1.2 Plot a rectangle with length of 5 mm and width of 3 mm.

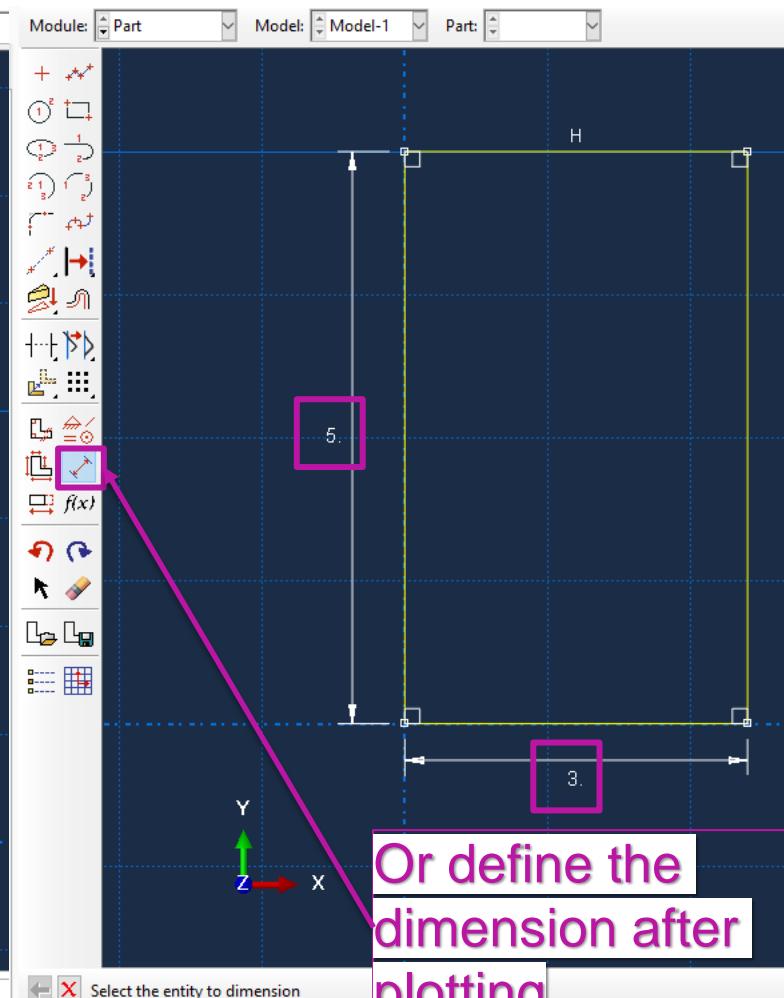
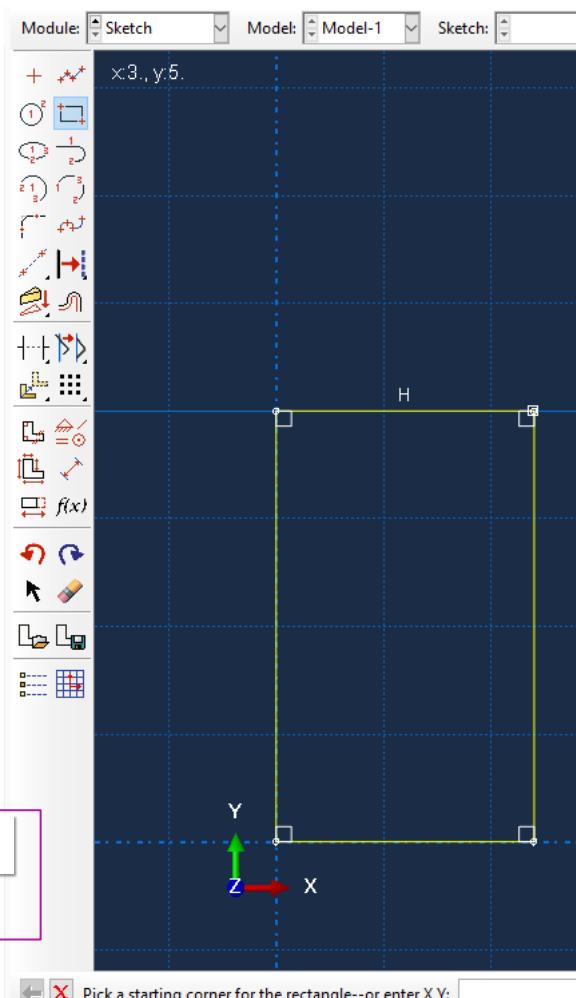


2.

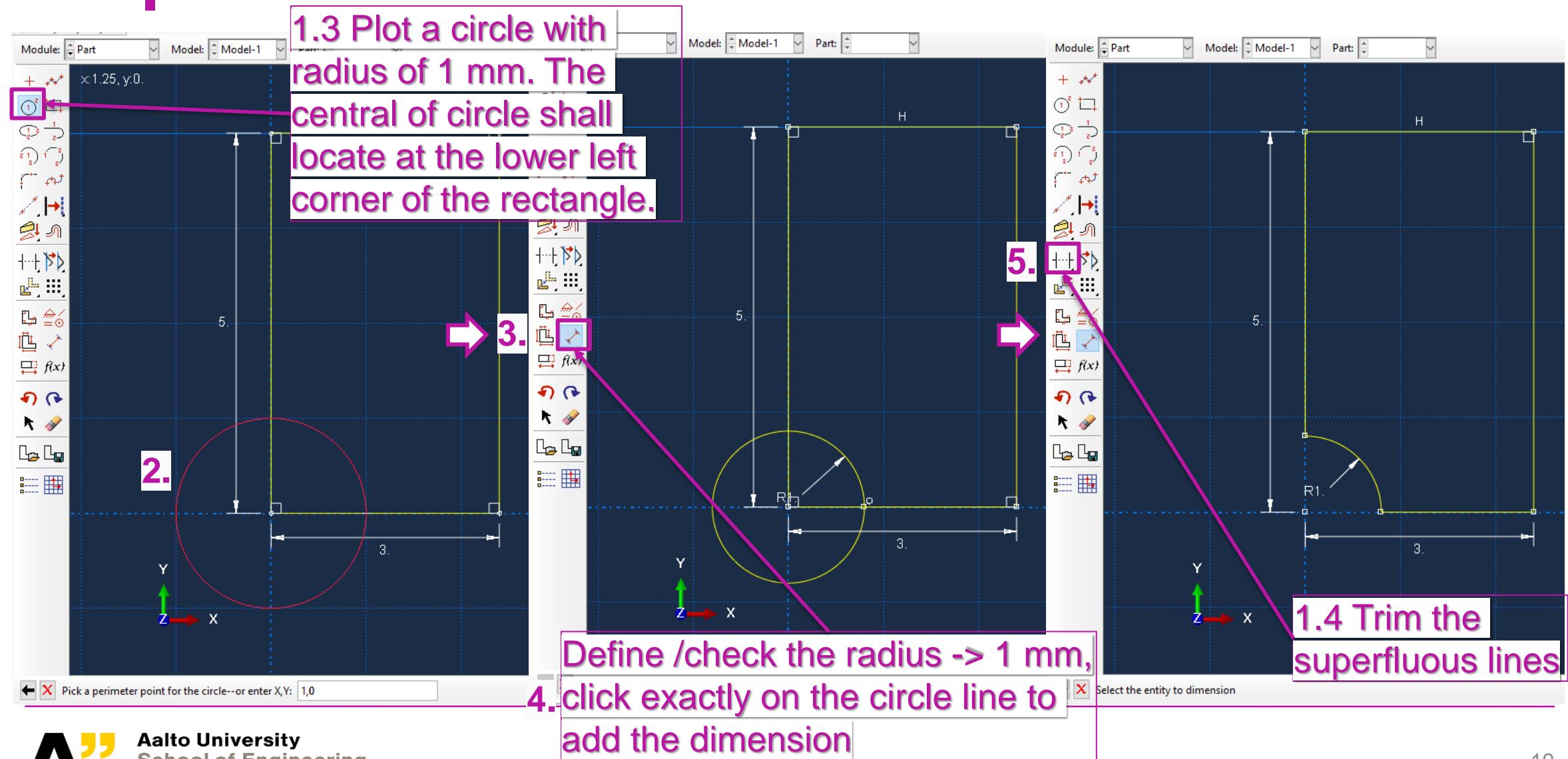
Pick a starting corner for the rectangle--or enter X,Y: 0,0

3.

Pick the opposite corner for the rectangle--or enter X,Y: 3,5

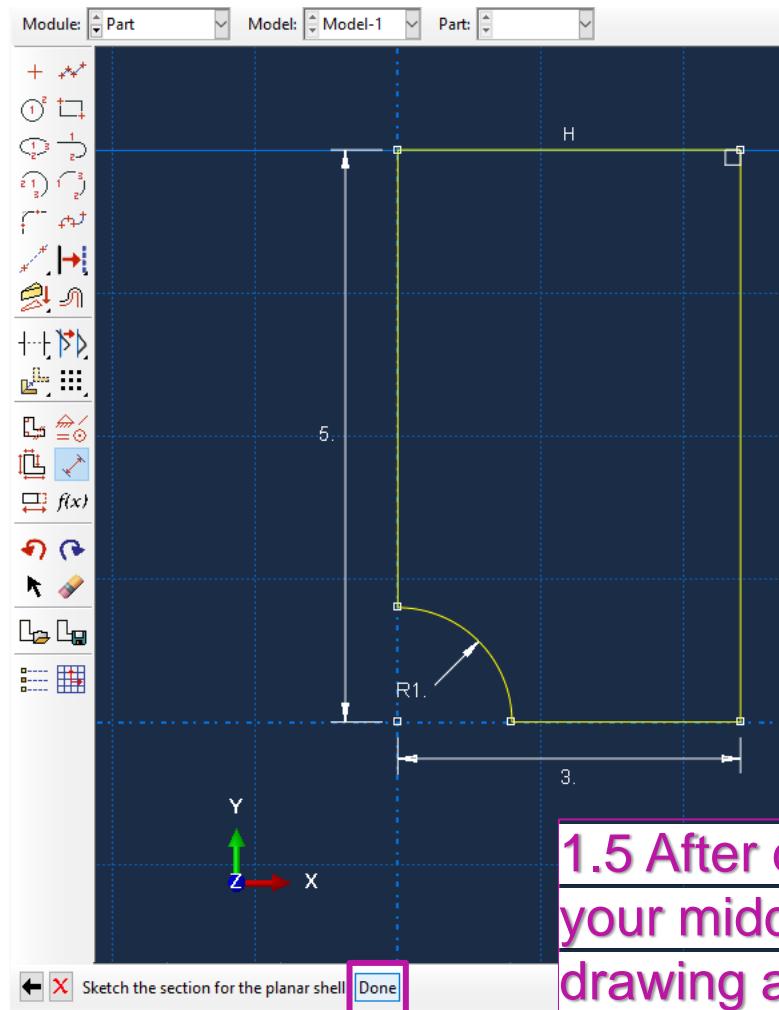


Step 1: Create Part

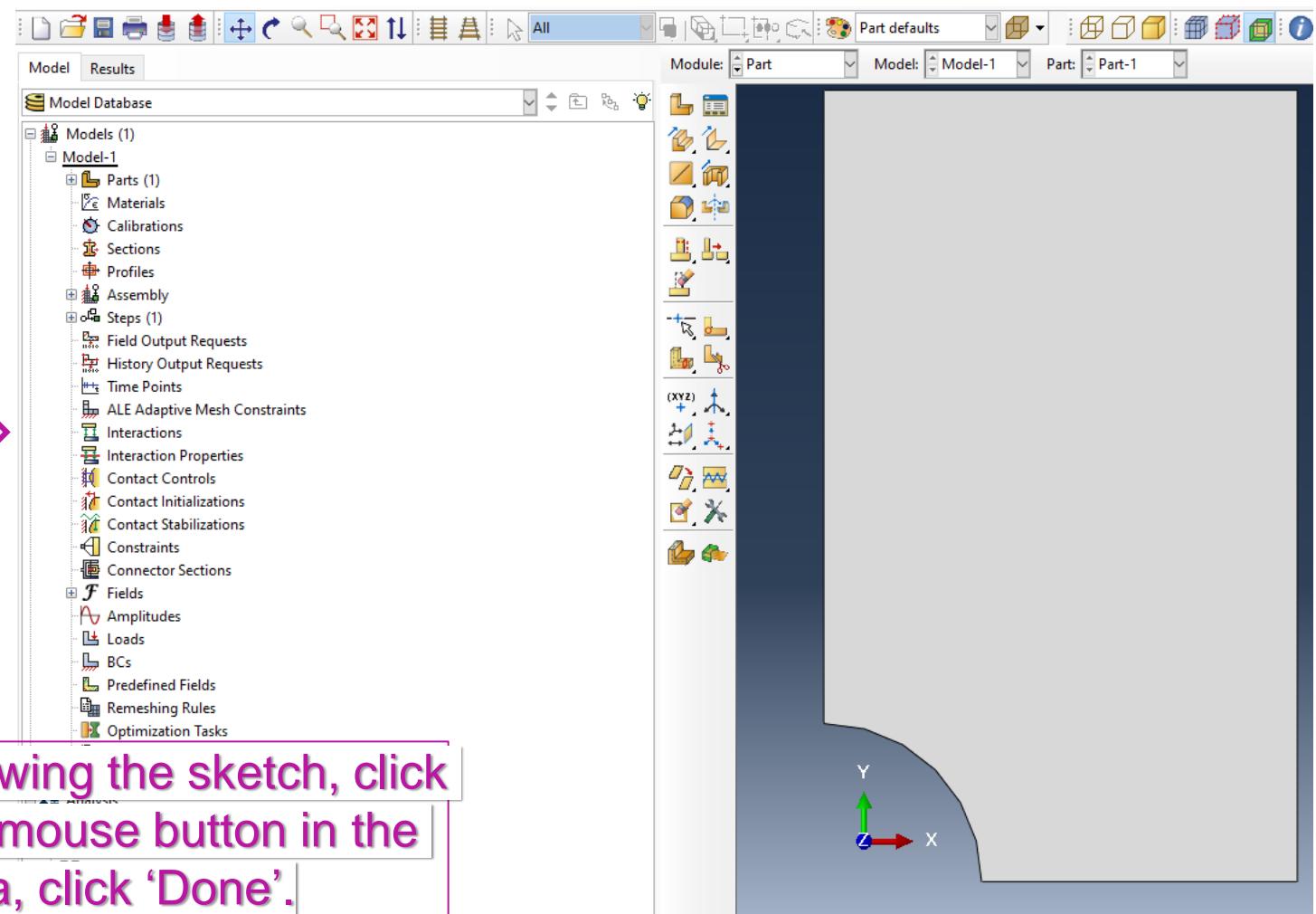


Step 1: Create Part

In Abaqus, middle click -> confirm/done

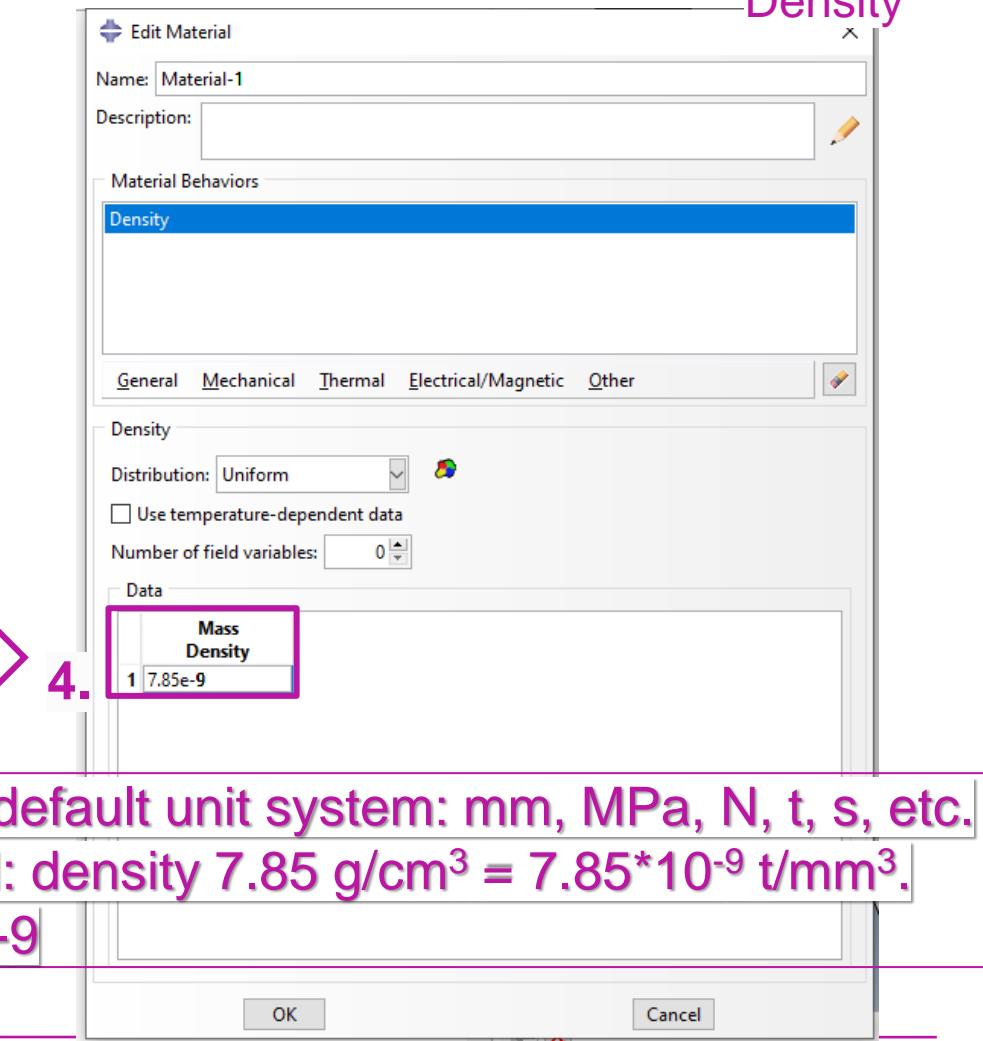
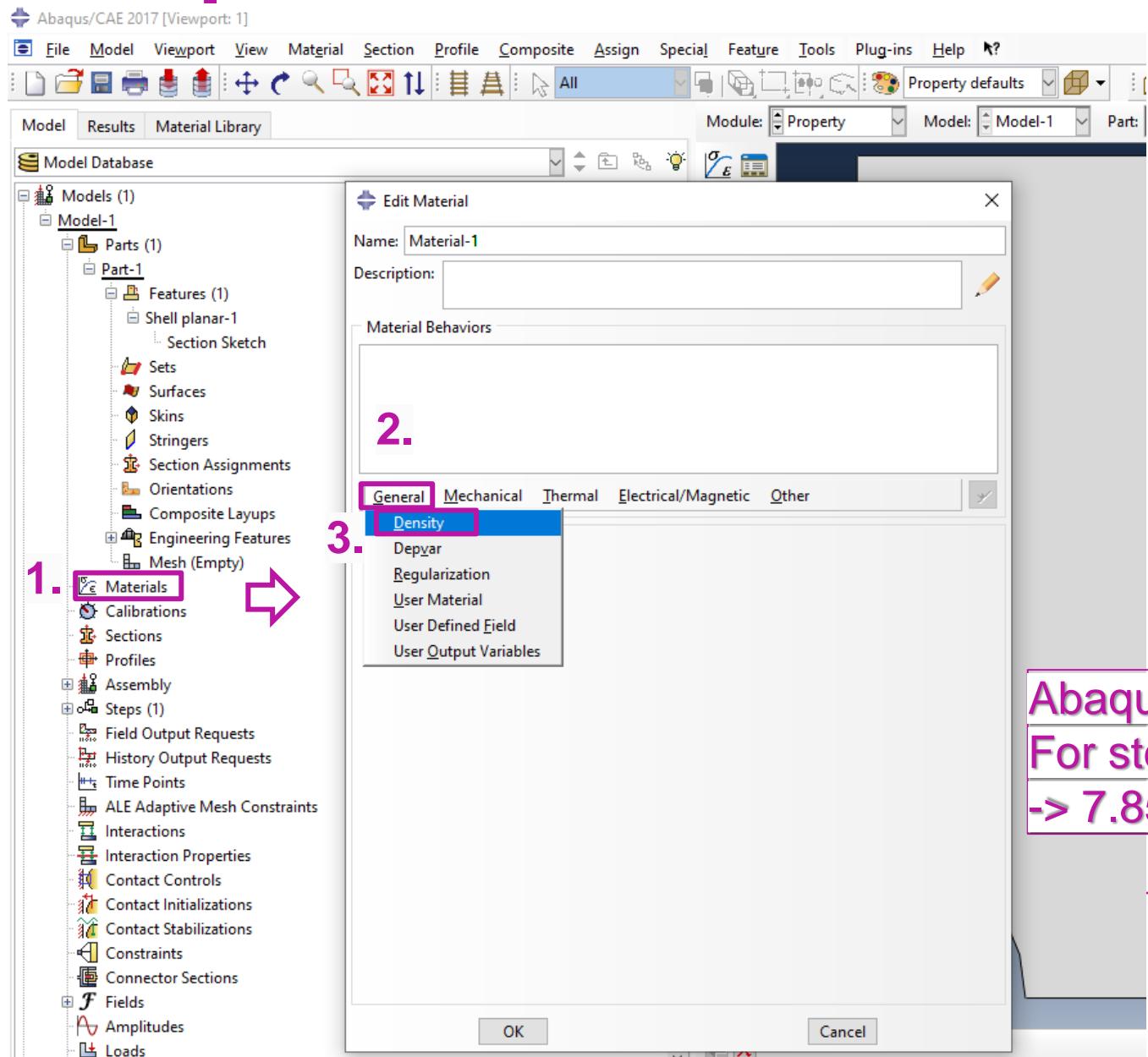


1.5 After drawing the sketch, click your middle mouse button in the drawing area, click 'Done'.



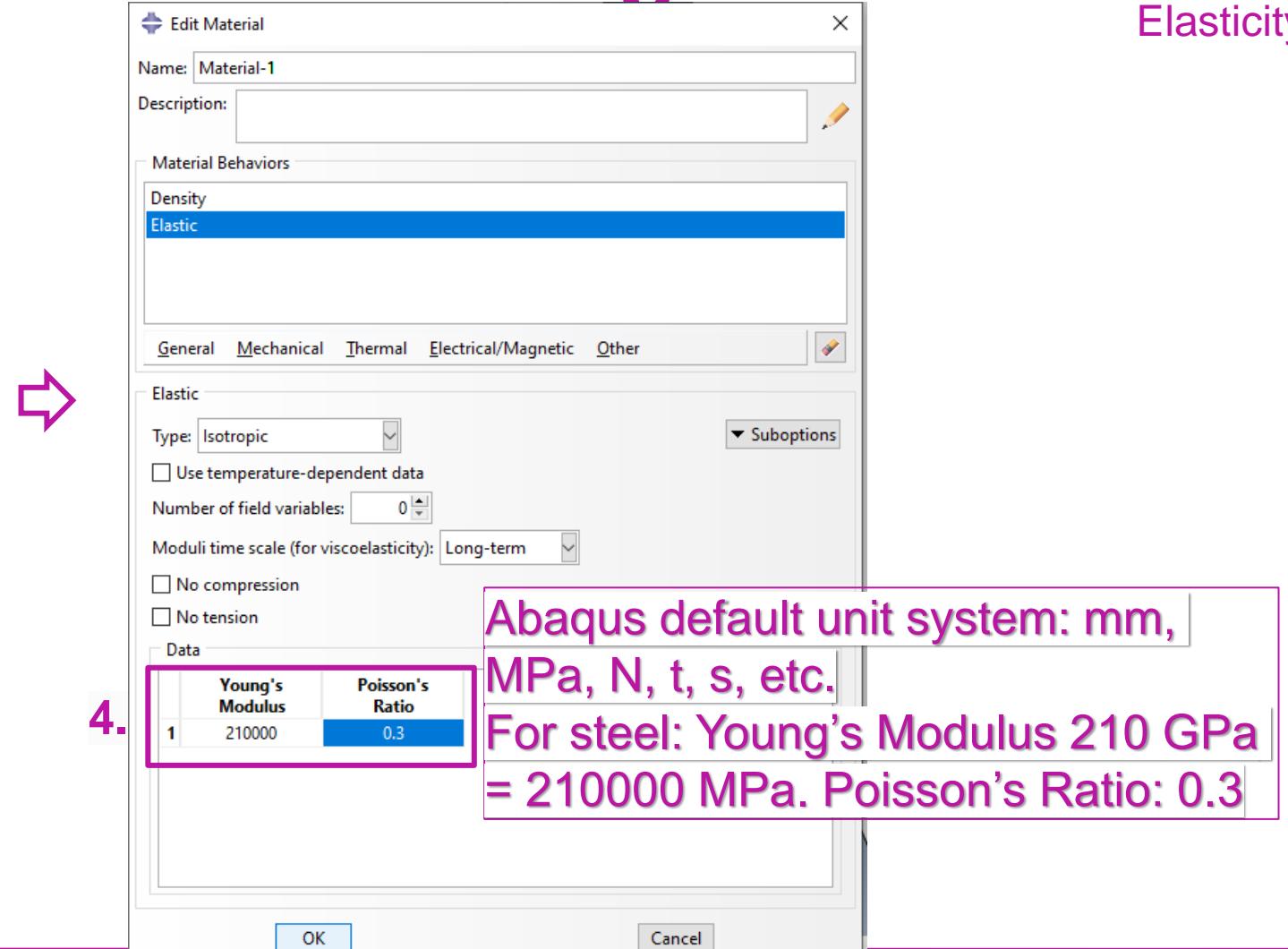
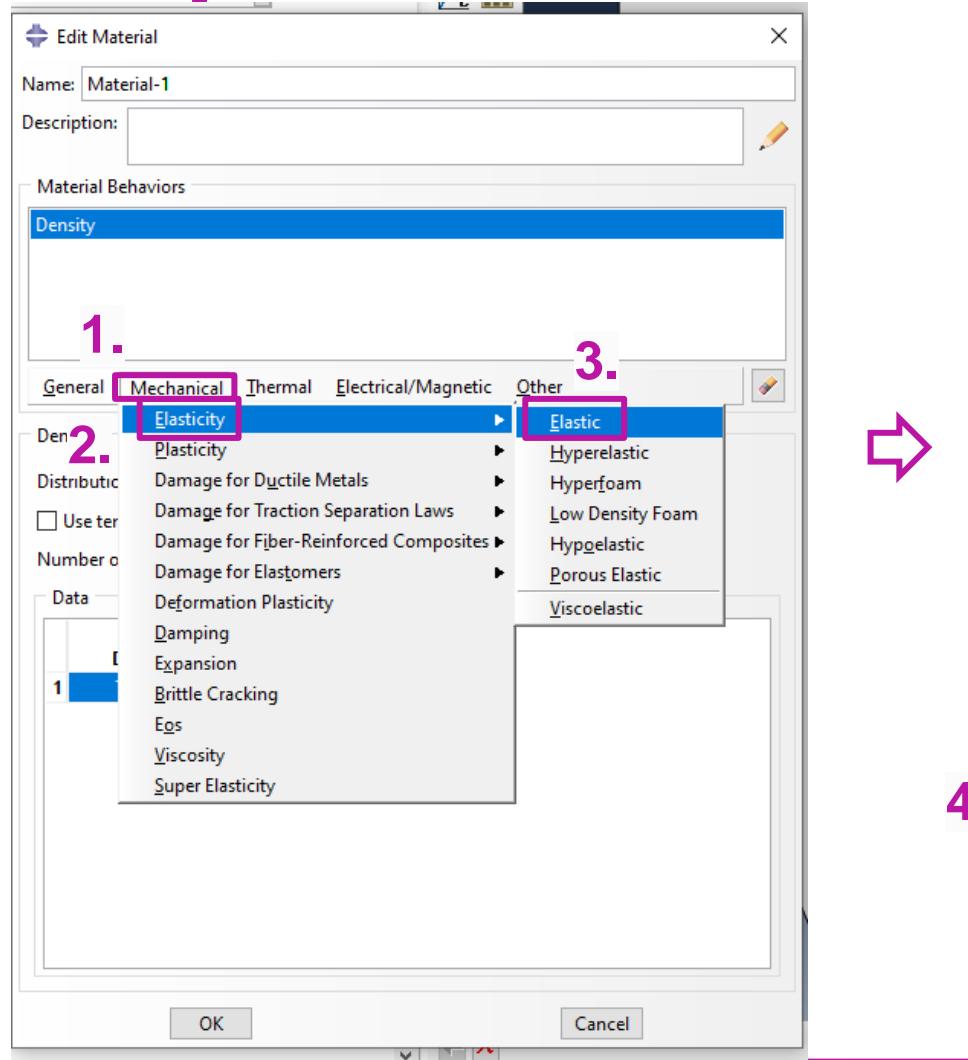
Step 2: Create Material and Assign to Part

• 2.1 Add Density



Step 2: Create Material and Assign to Part

- 2.2 Add Elasticity



Step 2: Create Material and Assign to Part

- 2.3 Add Plasticity

1.

2.

3.

4.

select all data and copy

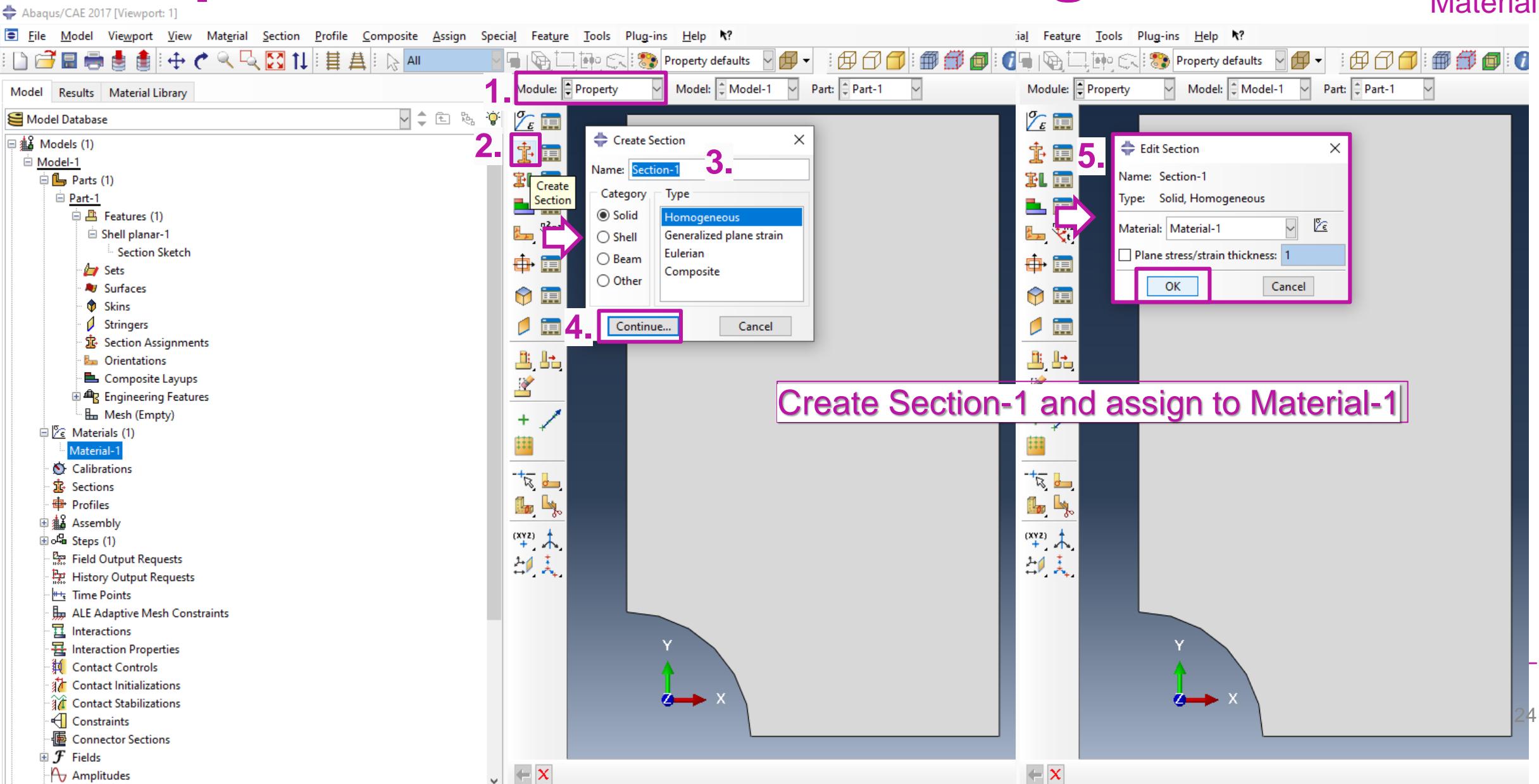
	Yield Stress	Plastic Strain
1	222.2154	0
2	224.6897161	0.0001
3	225.918863	0.0002
4	226.923468	0.0003
5	227.8094852	0.0004
6	228.6186665	0.0005
7	229.3726736	0.0006
8	230.0844894	0.0007
9	230.7626344	0.0008
10	231.4130632	0.0009
11	232.0401346	0.001
12	237.5093947	0.002
13	242.1738289	0.003
14	246.4023323	0.004
15	250.3404634	0.005
16	254.0638928	0.006
17	257.6180956	0.007
18	261.0330204	0.008
19	264.3296867	0.009
20	267.523558	0.01
21	270.6264321	0.011
22	273.647575	0.012
23	276.5944391	0.013
24	279.4731382	0.014
25	282.2887729	0.015
26	285.0456609	0.016
27	287.747503	0.017
28	290.397507	0.018
29	292.9984802	0.019
30	295.5529017	0.02
31	298.0629772	0.021
32	300.5306837	0.022
33	302.9578044	0.023
34	305.3459572	0.024
35	307.6966176	0.025
36	310.0111383	0.026

Use the fitted flow curve data for plastic parameters, make sure the first column is 'Stress', and the second one is 'Plastic Strain'.

For this exercise, use the data from 'E3flowcurve.txt' file.

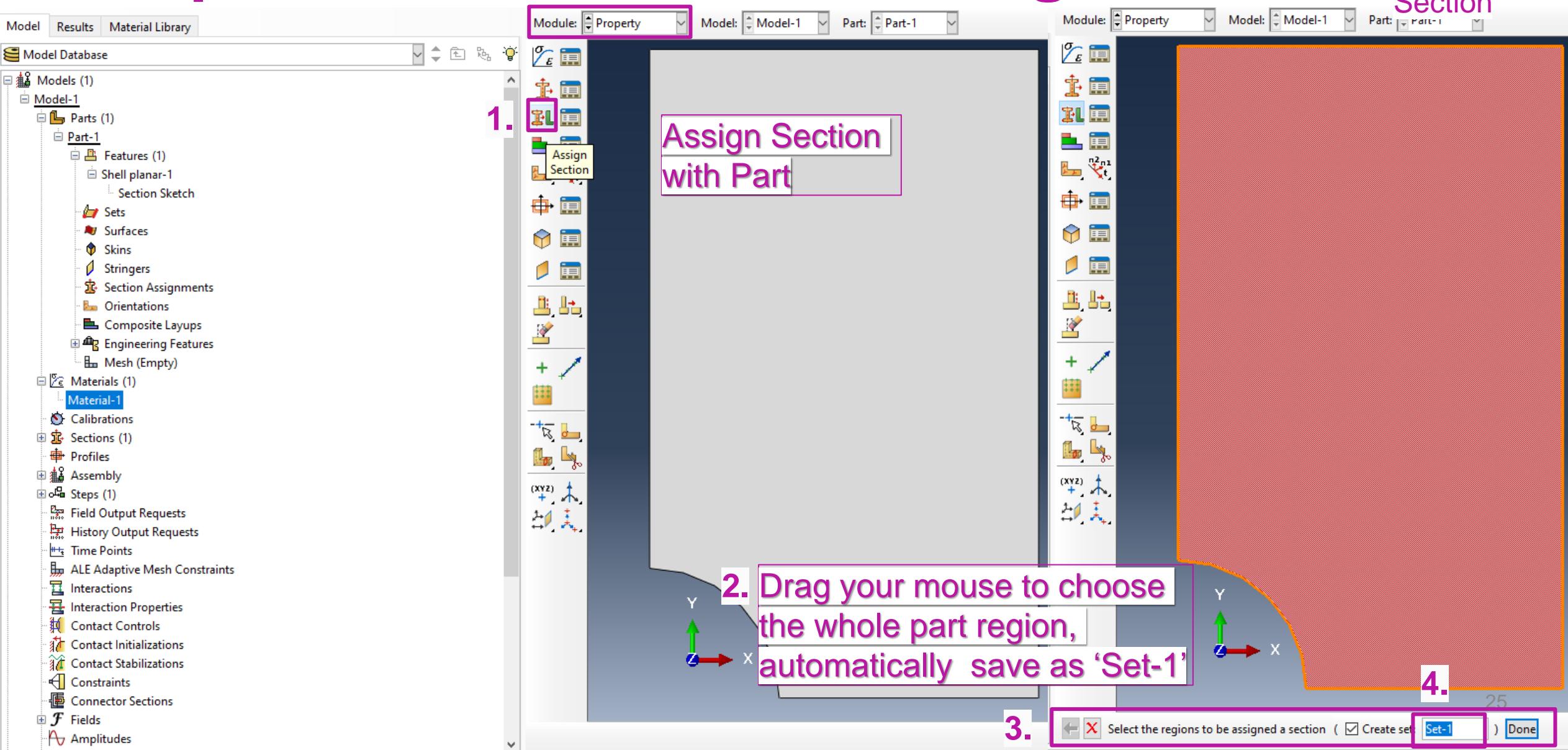
- 2.4 Assign Section to Material

Step 2: Create Material and Assign to Part



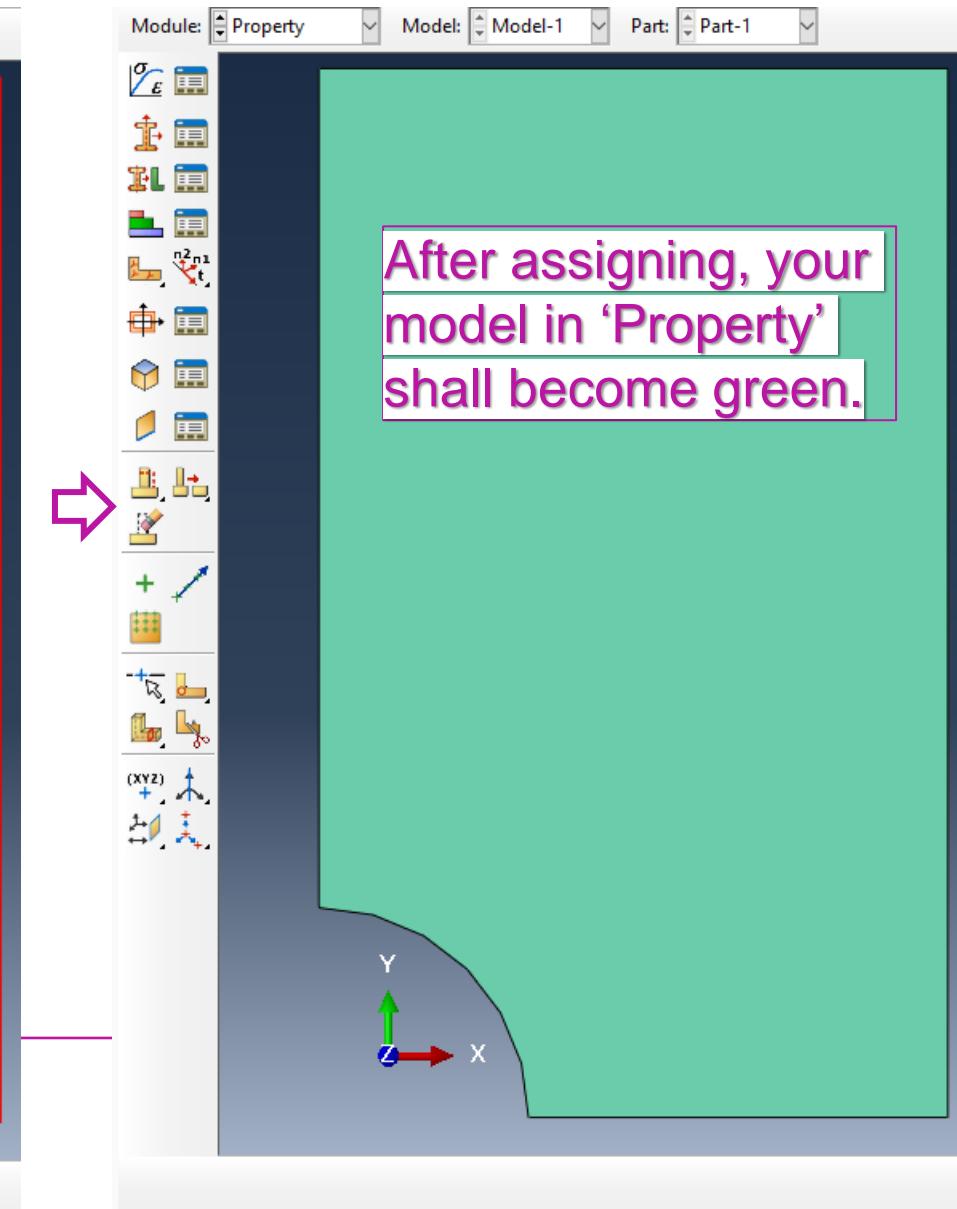
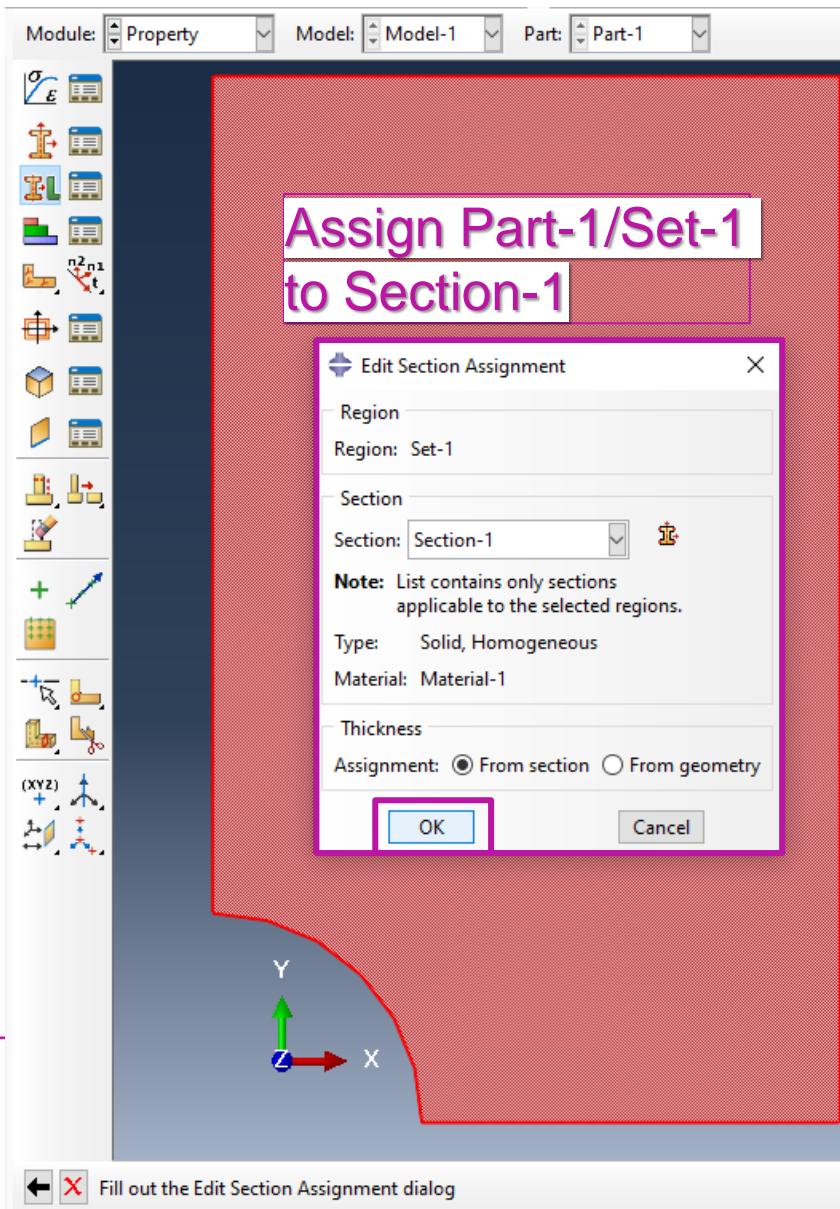
- 2.5 Assign Part to Section

Step 2: Create Material and Assign to Part

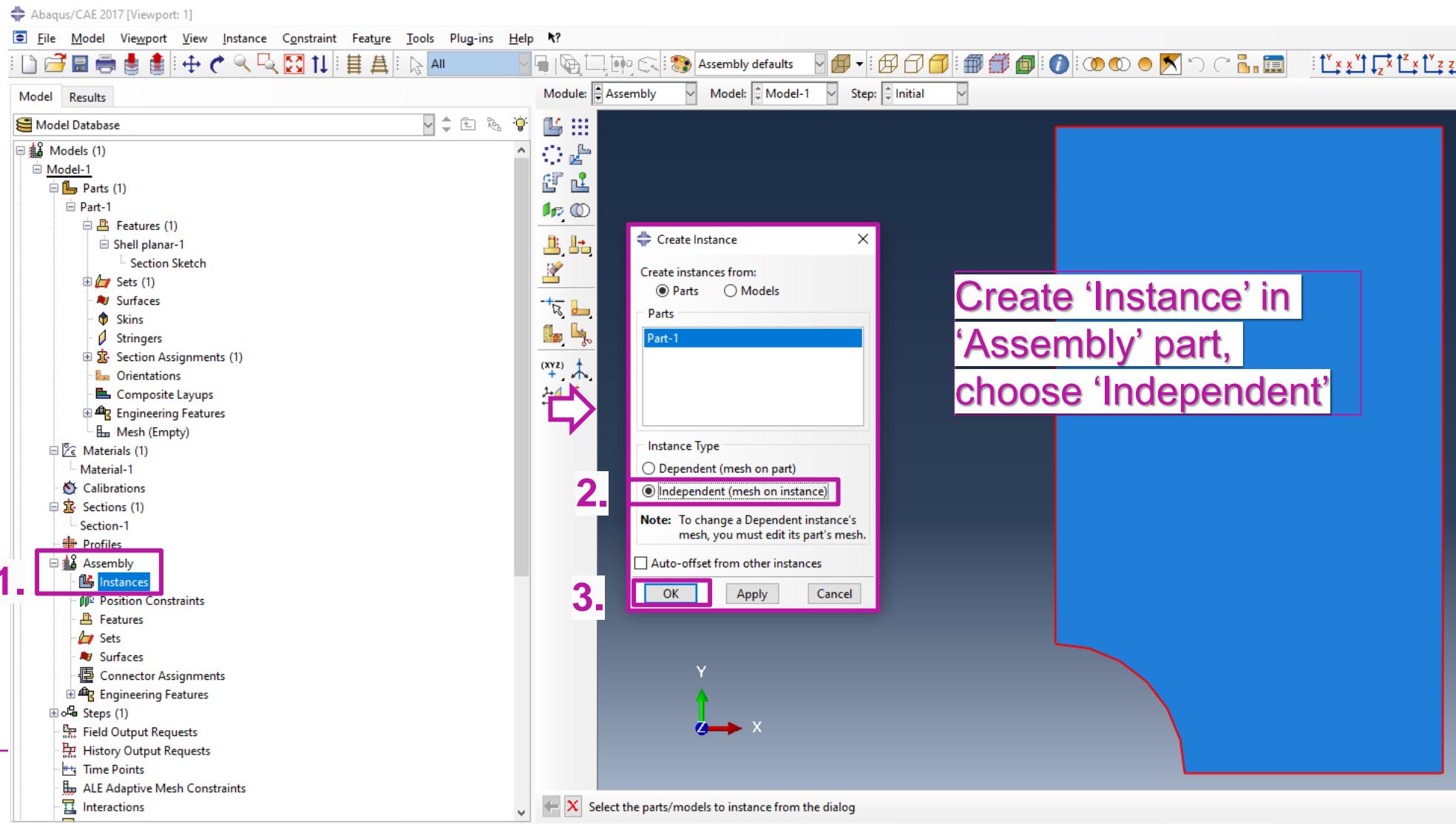


- 2.5 Assign Part to Section

Step 2: Create Material and Assign to Part



Step 3: Create Assembly



Step 4: Create Step and Boundary Conditions

• 4.1 Create analysis Step

The image shows the Abaqus/CAE software interface. The top menu bar includes File, Model, Viewport, View, Step, Output, Other, Tools, Plugins, Help, and a question mark icon. The toolbar has various icons for model creation and modification. The main window displays a 'Model Database' tree on the left with 'Model-1' selected, containing 'Parts (1)', 'Materials (1)', and 'Steps (1)'. A pink box highlights the 'Steps (1)' entry. The central workspace shows a 3D model of a shell planar-1 part with a coordinate system (X, Y, Z). A pink arrow points from the 'Steps (1)' entry to a 'Create Step' dialog box. The dialog box has a 'Name' field set to 'Step-1' and an 'Insert new step after' dropdown set to 'Initial'. A pink box highlights the 'Procedure type' dropdown, which is currently set to 'General'. Below it, the 'Dynamic, Explicit' option is highlighted with a pink box and a pink arrow pointing to it. The 'Continue...' button at the bottom right of the dialog box is also highlighted with a pink box. A pink arrow points from the 'Dynamic, Explicit' option to a 'Edit Step' dialog box on the right. The 'Edit Step' dialog box shows 'Name: Step-1' and 'Type: Dynamic, Explicit'. The 'Time period:' field is set to '0.01'. A pink box highlights this value. Below it, there are options for 'NLgeom' (Off or On) and 'Include adiabatic heating effects'. A pink box highlights the 'OK' button at the bottom right of the 'Edit Step' dialog box. A pink arrow points from the 'OK' button to a large pink box containing the text '5. Set Time Period to 0.01 s'. A pink arrow points from the 'Edit Step' dialog box to a 'Drag the mouse in a viewport to pan the view' instruction at the bottom.

1. Double click 'Steps' to add a new 'Step-1', choose 'Dynamic, Explicit'

2.

3.

4.

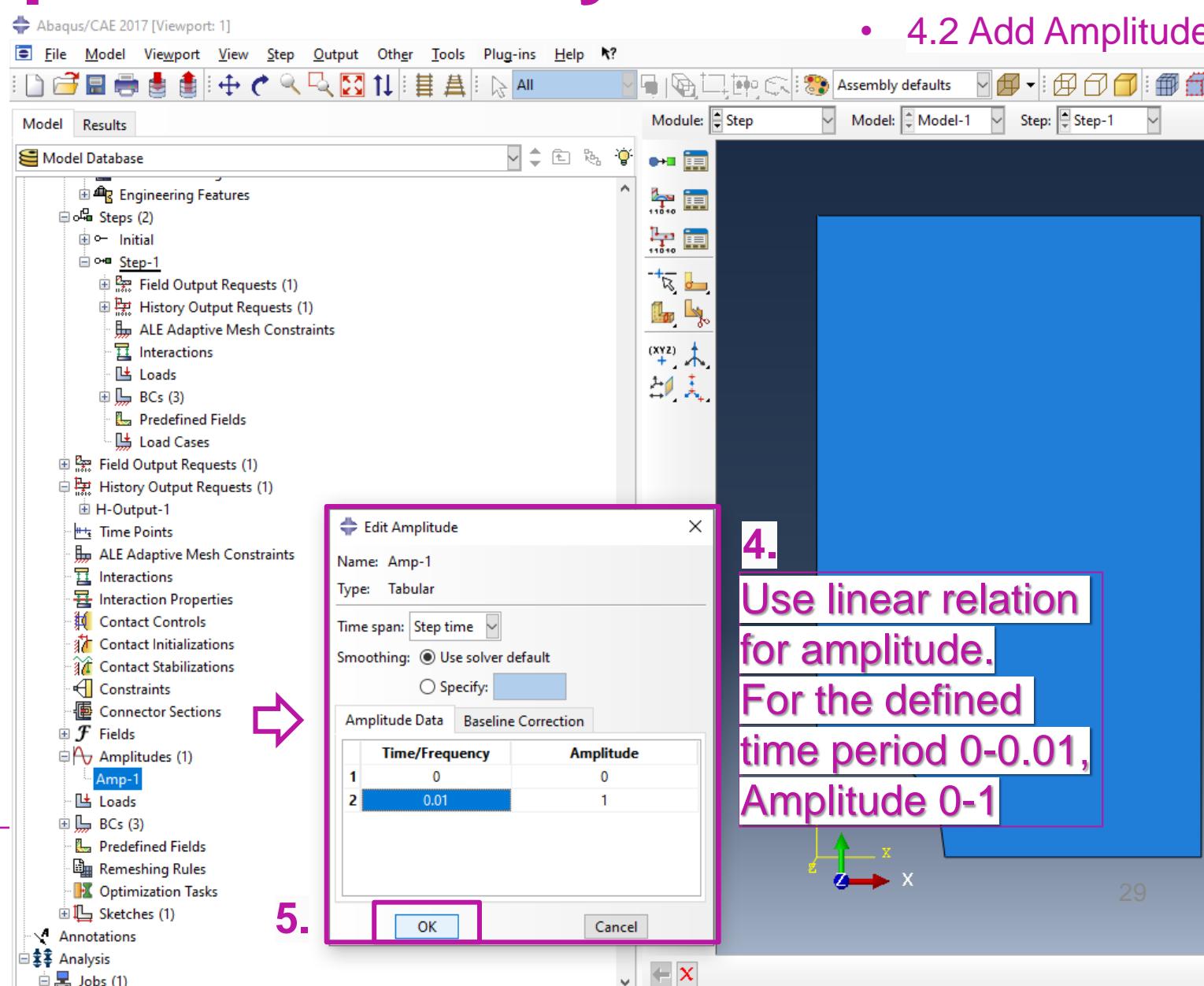
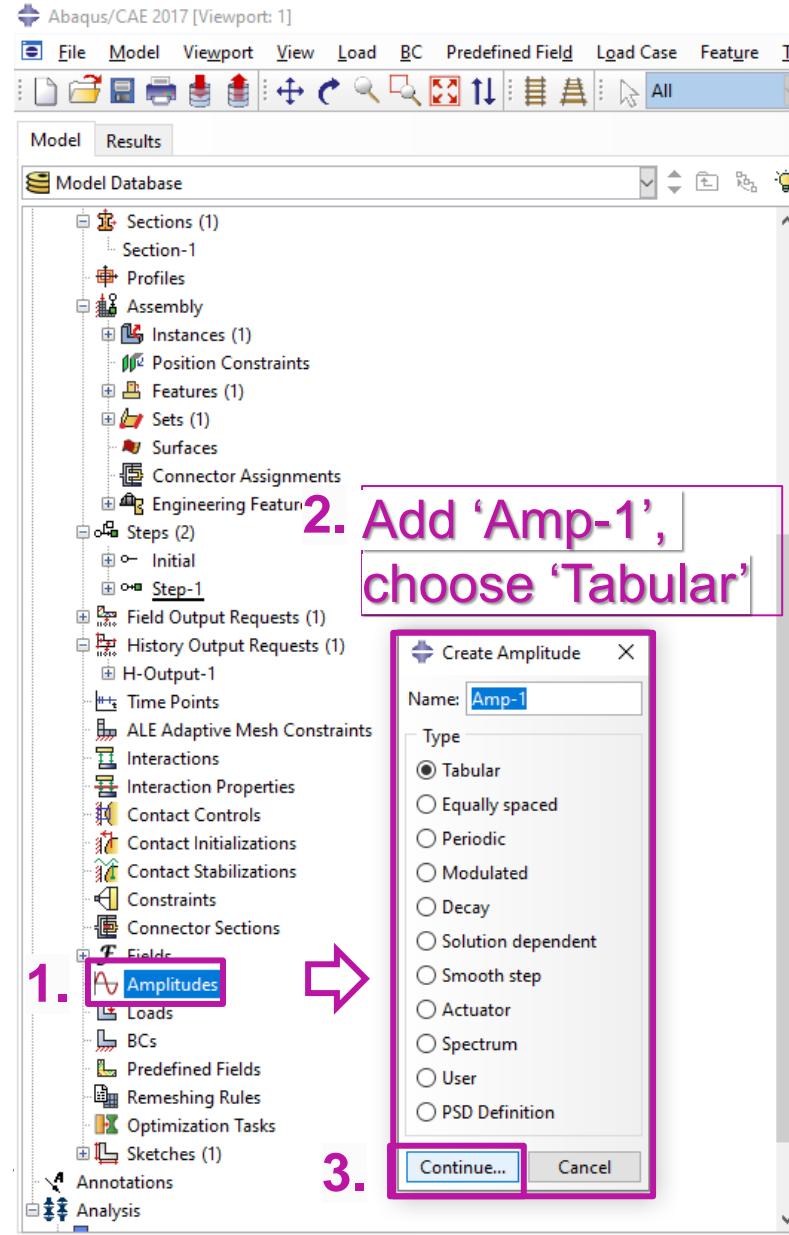
5. Set Time Period to 0.01 s

6.

Drag the mouse in a viewport to pan the view

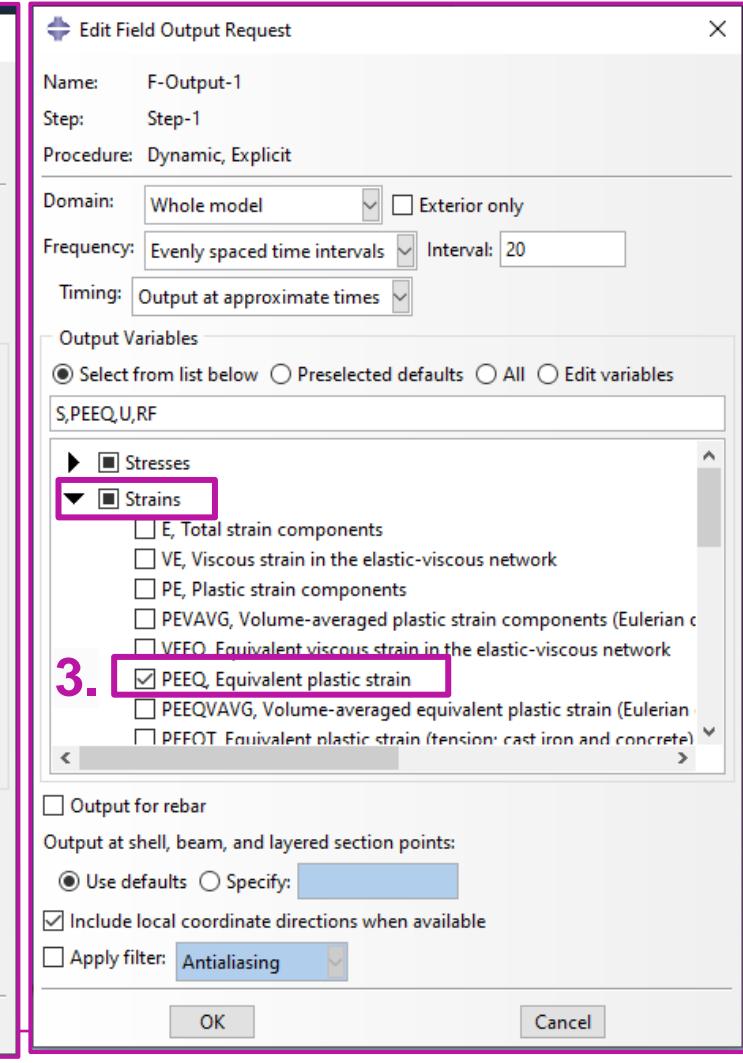
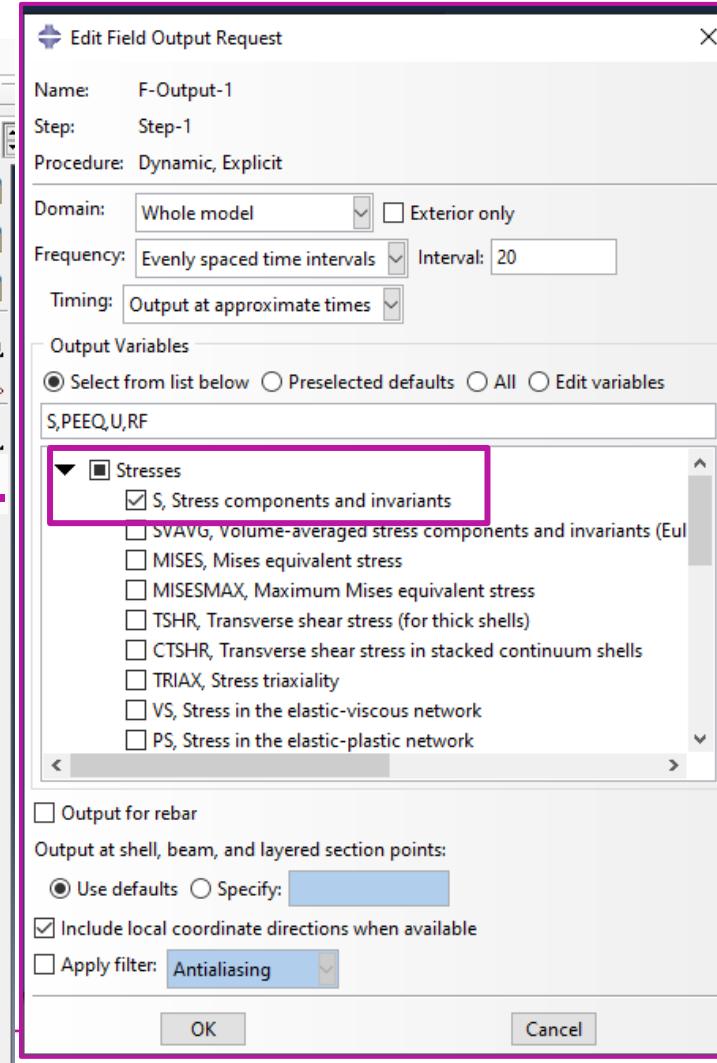
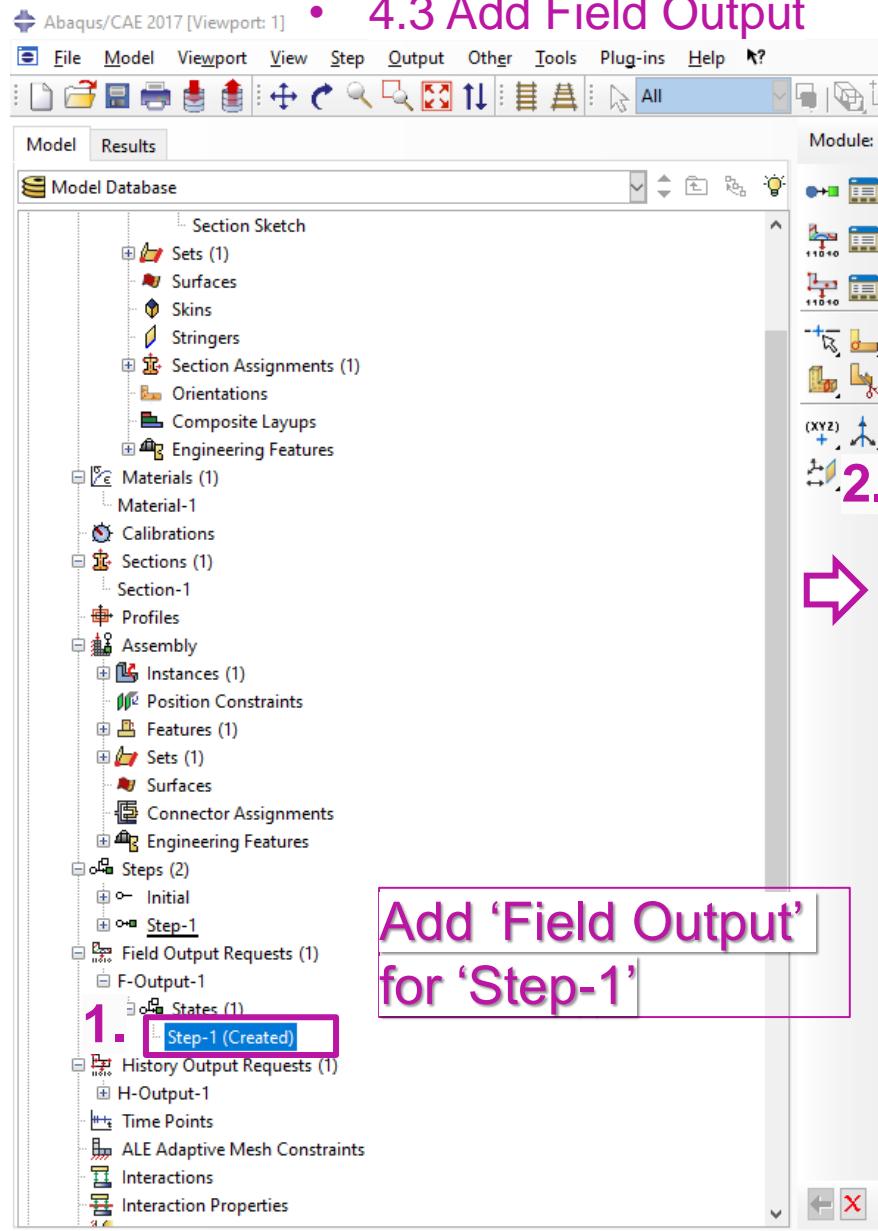
28

Step 4: Create Step and Boundary Conditions



Step 4: Create Step and Boundary Conditions

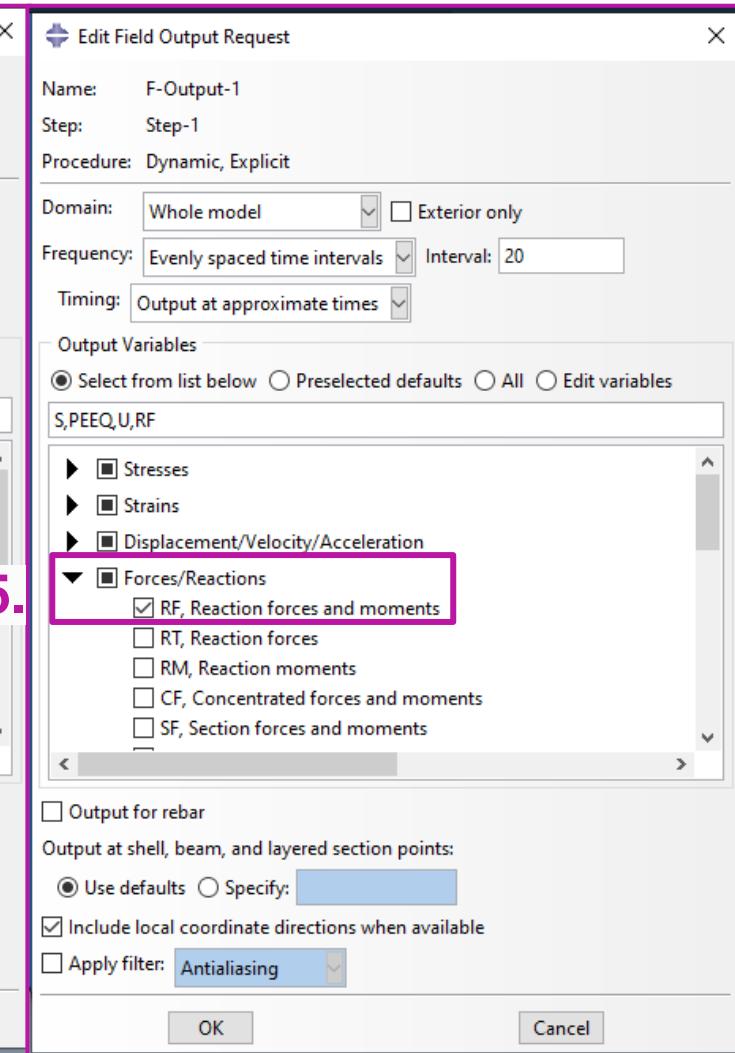
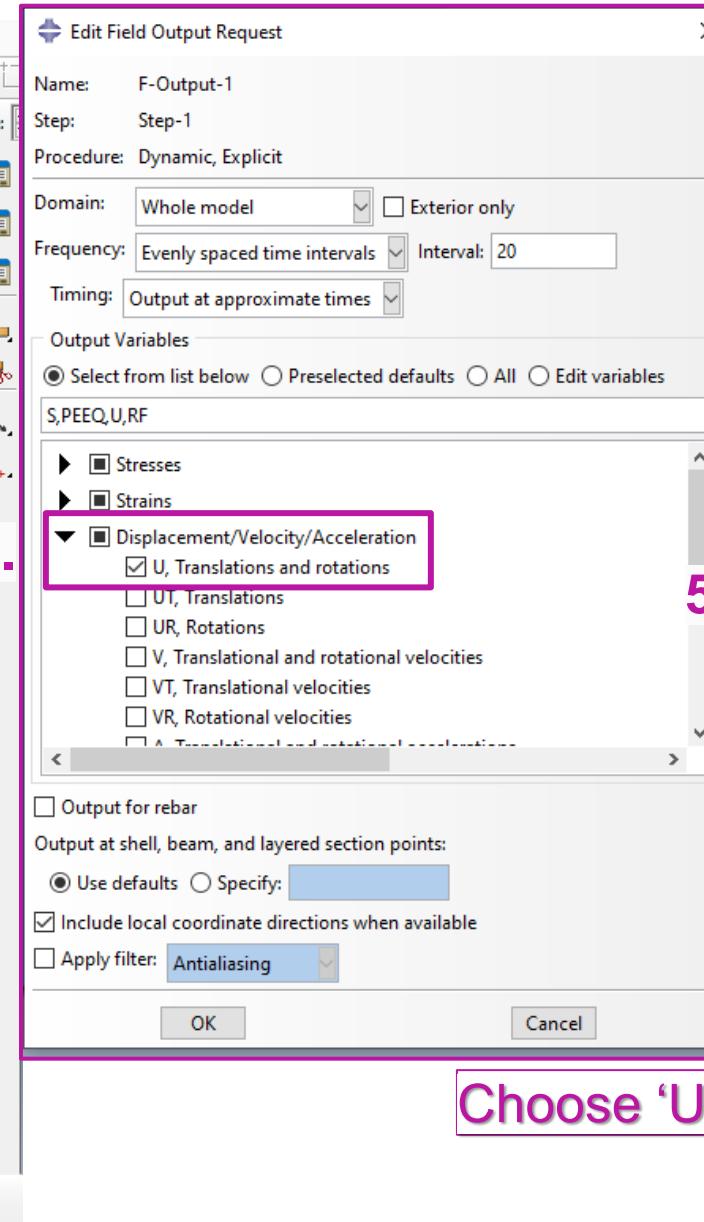
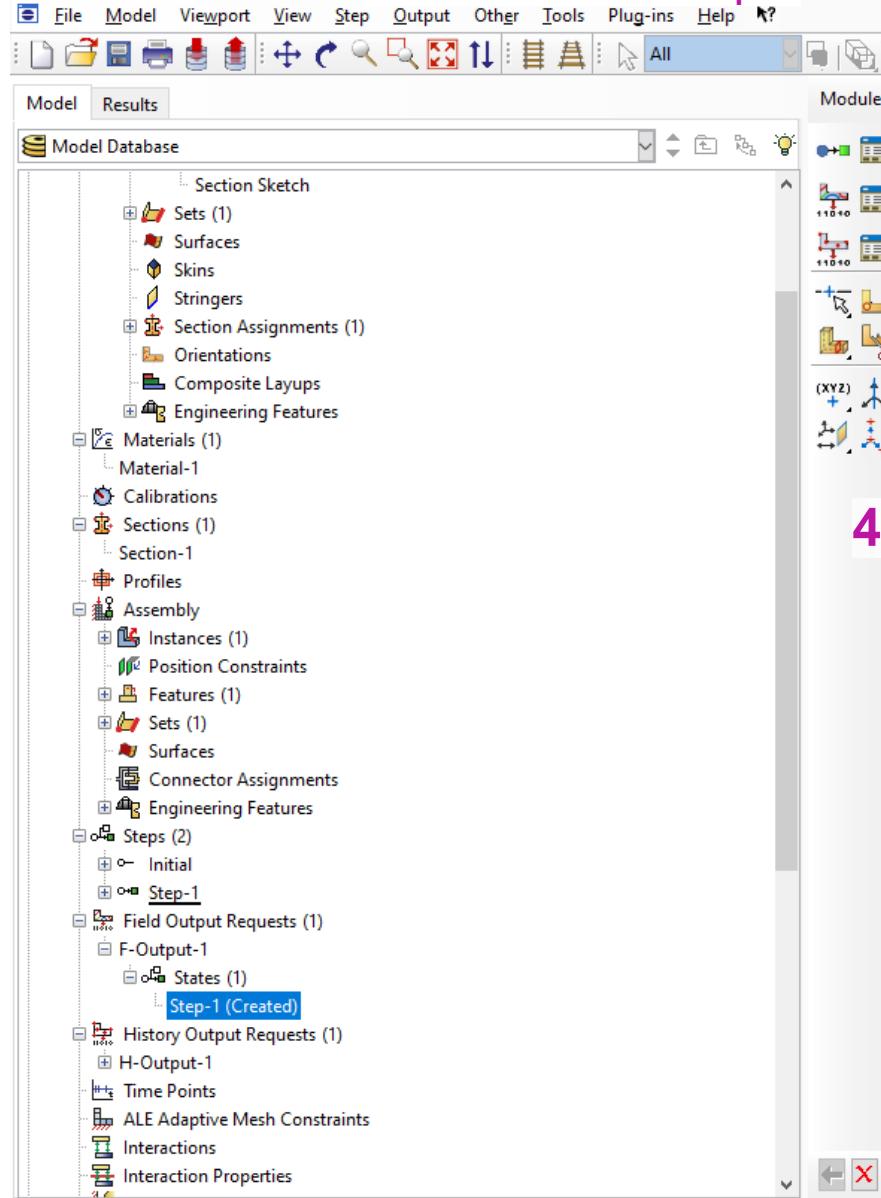
• 4.3 Add Field Output



Choose 'S', 'PEEQ'

Step 4: Create Step and Boundary Conditions

• 4.3 Add Field Output



Choose 'U', 'RF'

Step 4: Create Step and Boundary Conditions

1. Add 'BCs', choose 'Symmetry/...'

2. Create Boundary Condition

3. Continue...

4. Pick up the base line,
i.e. y symmetry line

5. Create a 'Ysymm' set

6. choose 'YSYMM'

7. OK

click exactly on the aimed line

32

The image shows the Abaqus CAE interface with the following steps highlighted:

1. In the left sidebar, under "BCs", select "Symmetry/...".
2. In the "Create Boundary Condition" dialog, enter "BC-1" for Name, "Step-1" for Step, and "Dynamic, Explicit" for Procedure. Under Category, select "Mechanical". Under Types for Selected Step, select "Symmetry/Antisymmetry/Encastre". Click "Continue...".
3. In the "Edit Boundary Condition" dialog, set Name to "BC-1", Type to "Symmetry/Antisymmetry/Encastre", Step to "Step-1 (Dynamic, Explicit)", and Region to "Ysymm". Under CSYS, click the "Global" button. Under Type, select "YSYMM (U2 = UR1 = UR3 = 0)". Click "OK".
4. In the 3D view, pick up the base line (y symmetry line).
5. Create a "Ysymm" set.
6. Choose "YSYMM".
7. Click "OK".

A callout points to the Y-axis in the 3D view, indicating where to click. The 3D view shows a model with a blue base and a grey top surface, with a coordinate system (X, Y, Z) and a boundary condition set on the Y-axis.

Step 4: Create Step and Boundary Conditions

- 4.5 Add 'X symmetry' boundary condition

The image shows the Abaqus CAE interface with the following steps highlighted:

1. Double click 'BCs' again, choose 'Symmtry/...'.
2. Create Boundary Condition dialog: Name: BC-2, Step: Step-1, Procedure: Dynamic, Explicit, Category: Mechanical, Types for Selected Step: Symmetry/Antisymmetry/Encastre.
3. Continue... button.
4. Pick up the left boundary line, i.e. x symmetry line.
5. Create a 'Xsymm' set.
6. choose 'XSYMM'.
7. OK button.

A callout text "click exactly on the aimed line" points to the boundary line being selected. A callout text "Select regions for the boundary condition (Create set: Xsymm) Done" is at the bottom right. A coordinate system (X, Y, Z) is shown at the bottom center.

Step 4: Create Step and Boundary Conditions

- 4.6 Add displacement-controlled boundary condition as loading

1. Double click 'BCs' again, choose 'Displacement/...'.

2. Create Boundary Condition

Name: BC-3
Step: Step-1
Procedure: Dynamic, Explicit

3. Continue...

4. Pick up the top line, i.e. displacement line,
5. Create a 'Disp' set

6. Click exactly on the aimed line

6. Edit Boundary Condition

Name: BC-3
Type: Displacement/Rotation
Step: Step-1 (Dynamic, Explicit)
Region: Disp

CSYS: (Global)

Distribution: Uniform
 U1:
 U2: 1
 UR3: radians
 Amplitude: Amp-1

Note: The displacement boundary condition will be reapplied in subsequent steps.

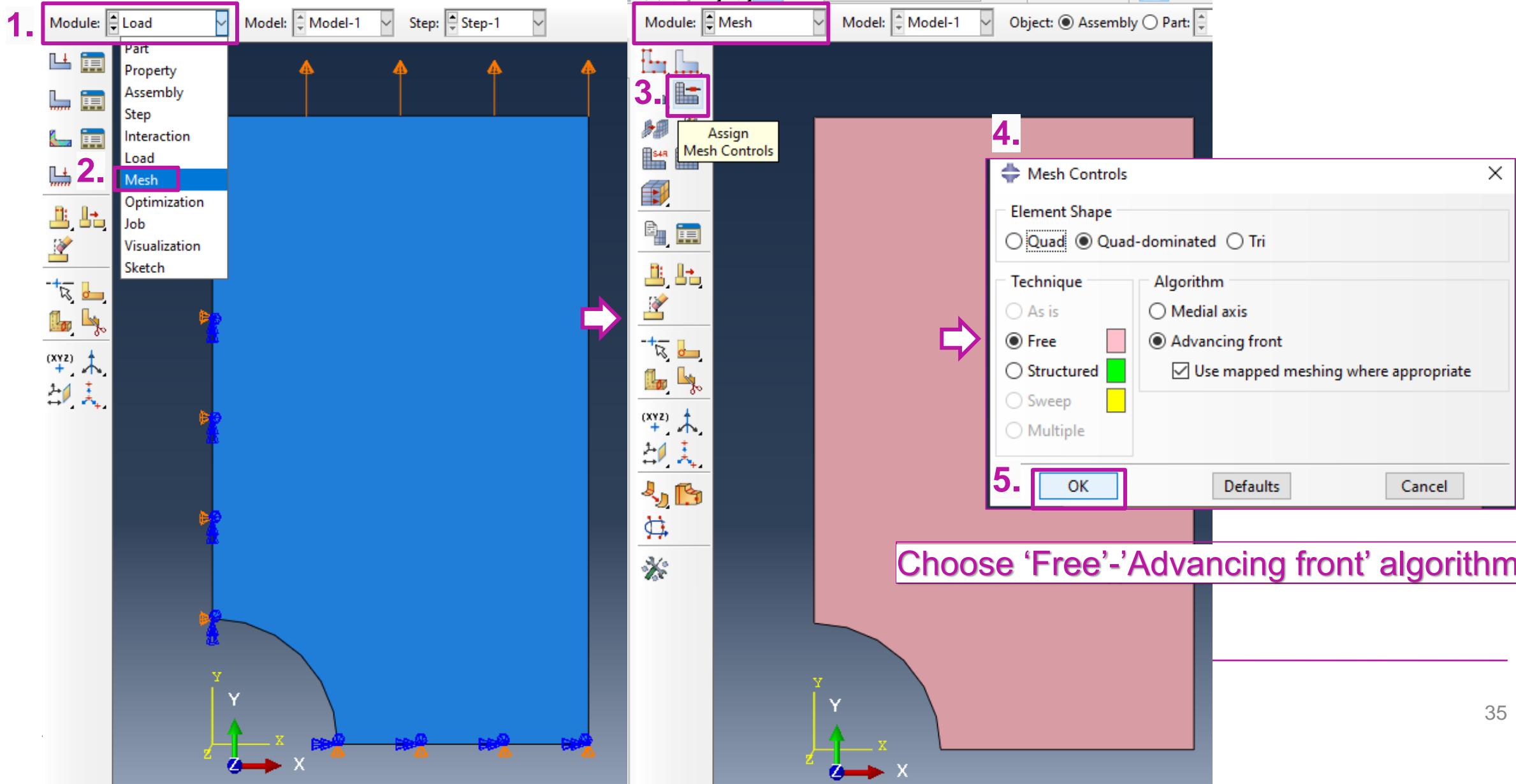
7. OK

8. Add 1 mm displacement along U2, Add 'Amp-1' for a global strain of 20%

Step 5: Create Mesh

- 5.1 Set Mesh type

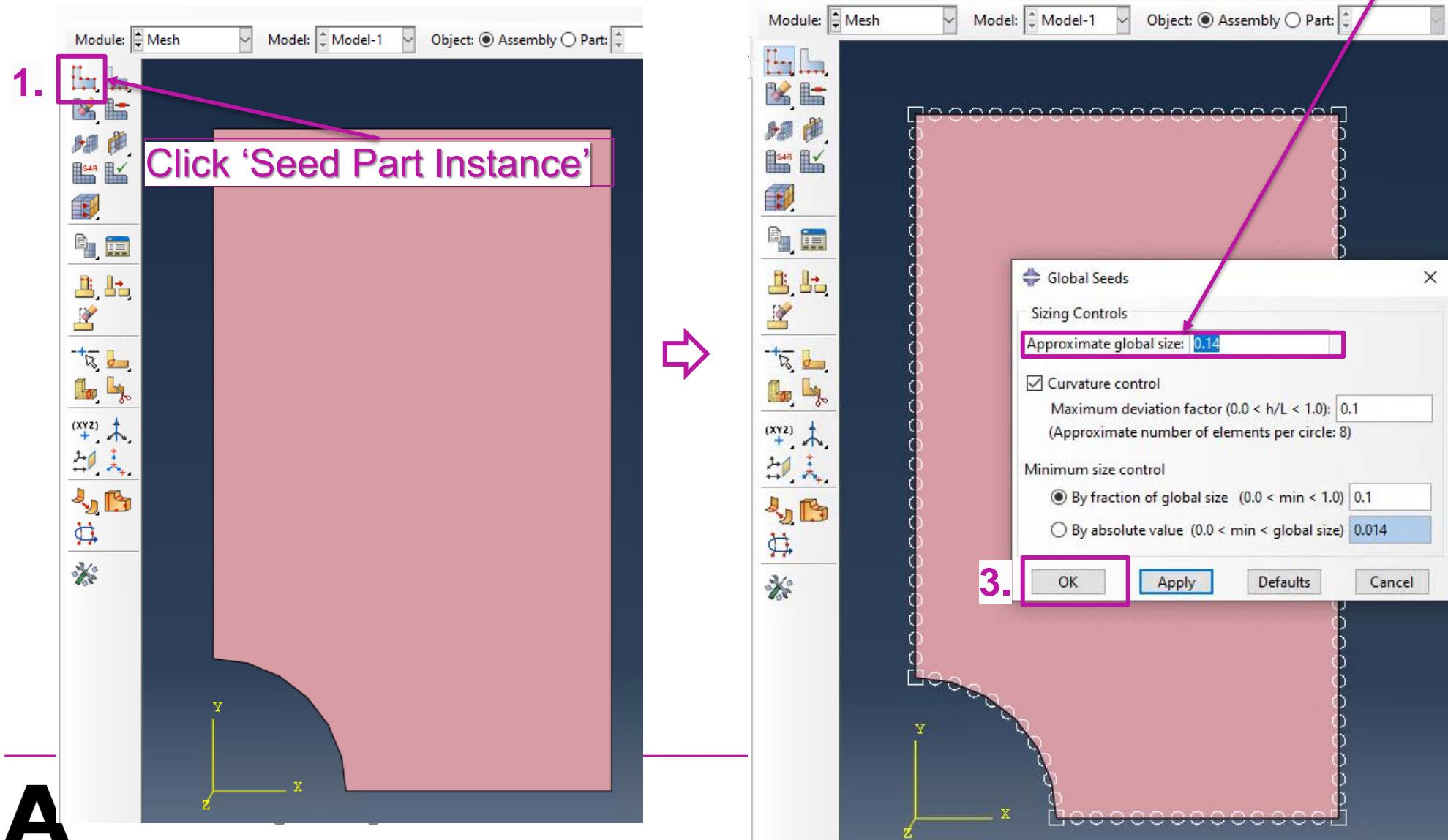
Change to 'Mesh' Module, Click 'Assign Mesh Control'



Step 5: Create Mesh

- 5.2 Set Mesh seed size

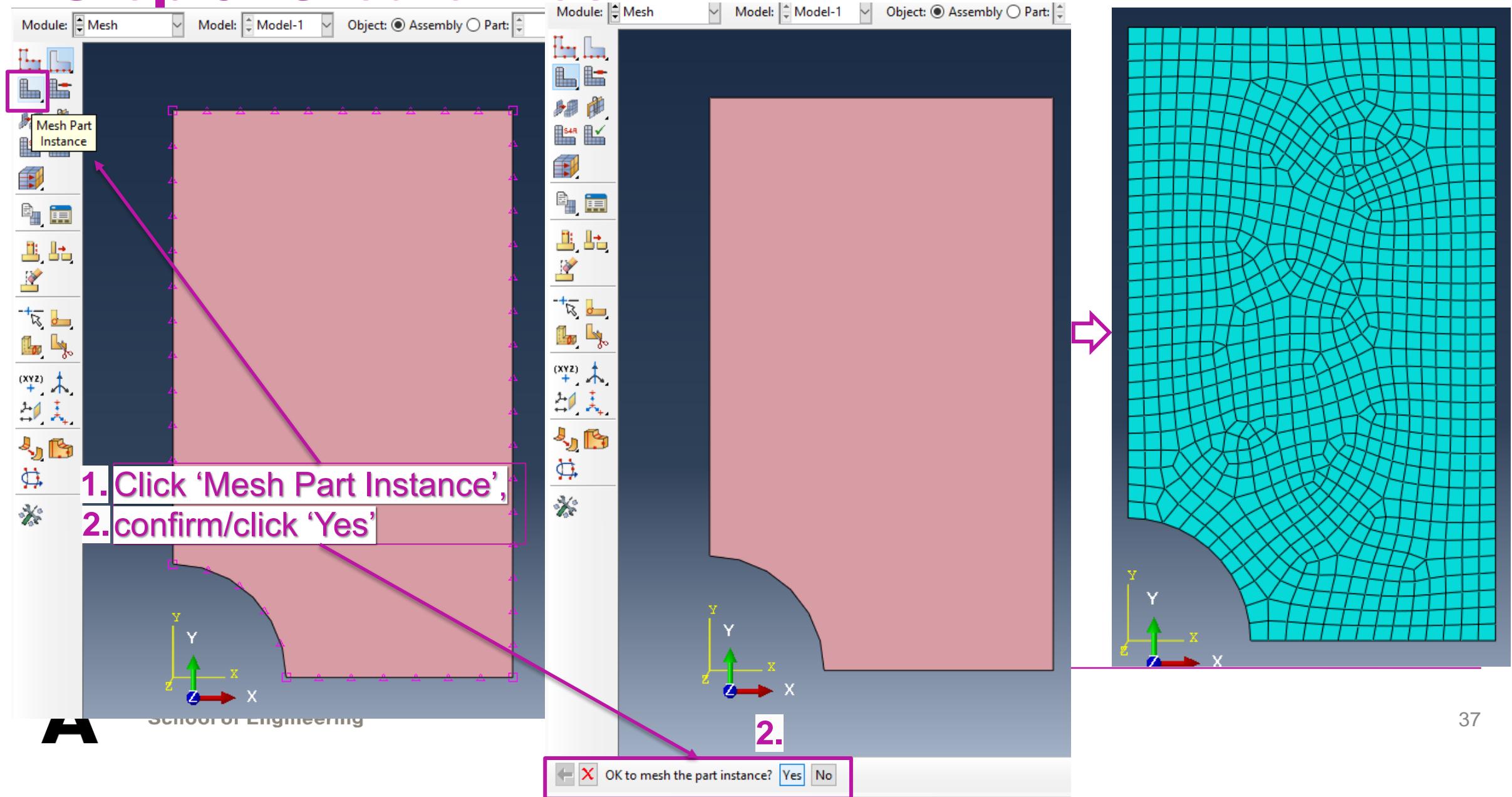
2. Set approximate element size as 0.14 mm



A

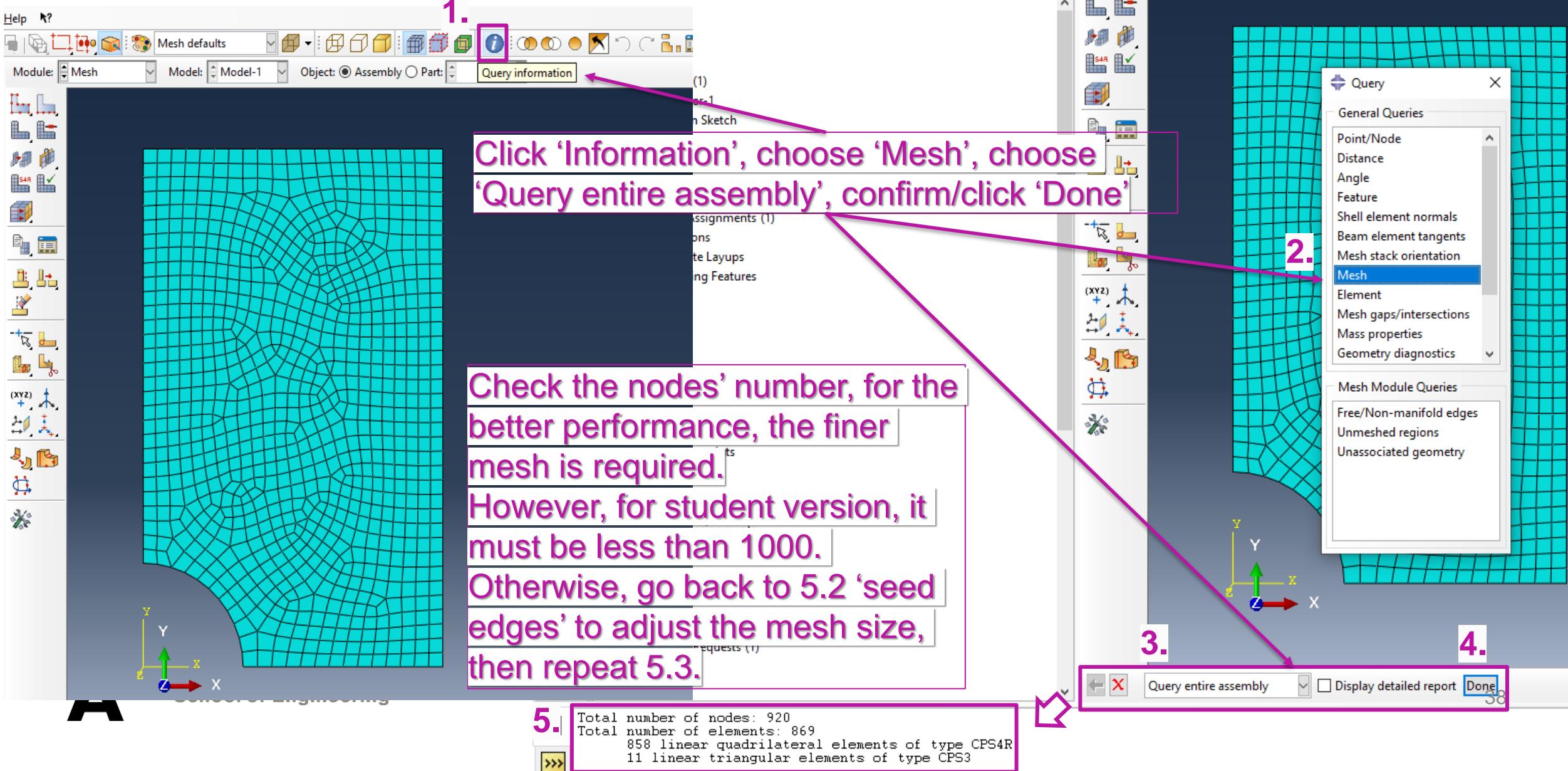
- 5.3 Mesh part whole

Step 5: Create Mesh



Step 5: Create Mesh

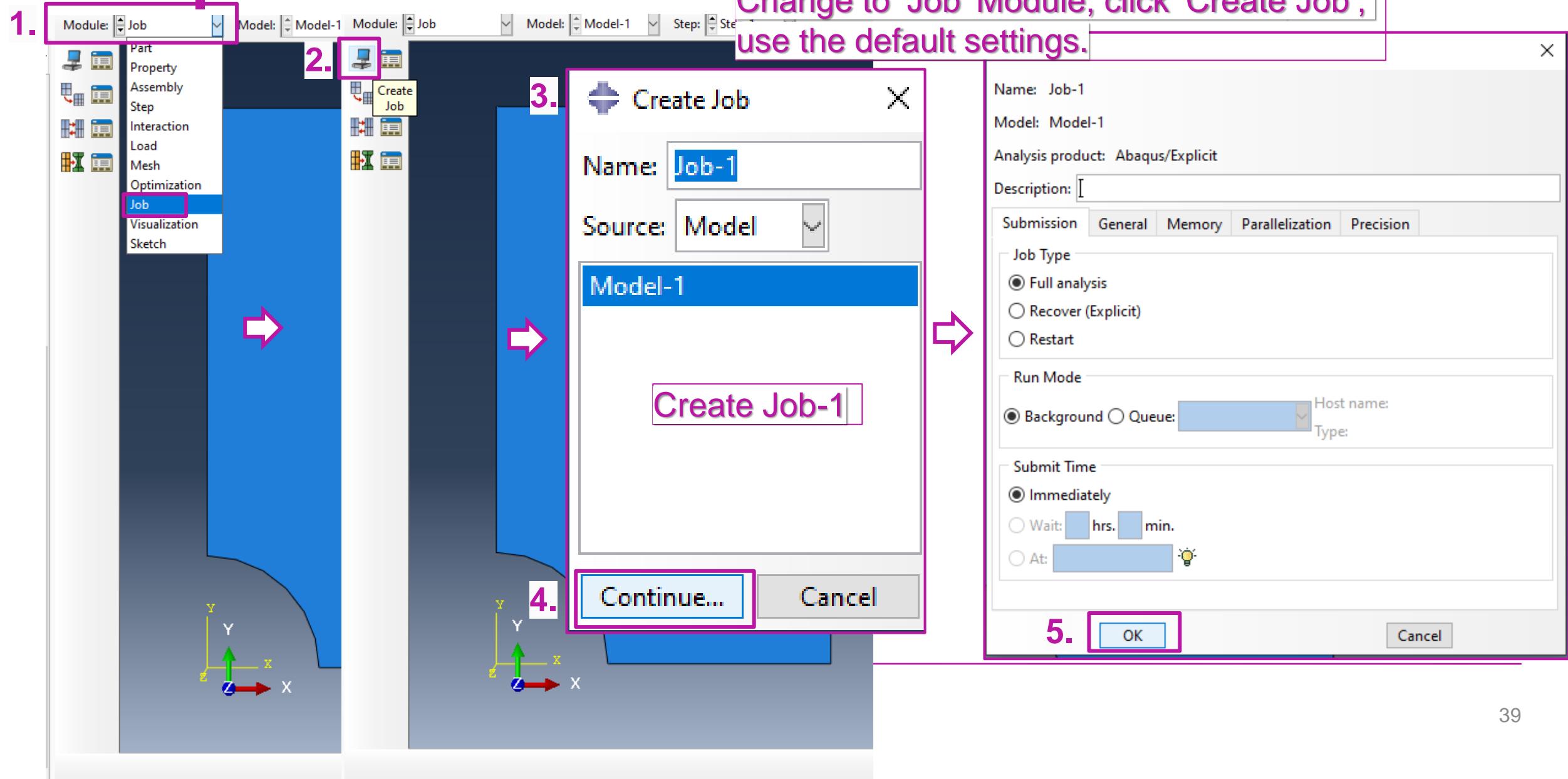
- 5.4 Check number of nodes



Step 6: Create Job

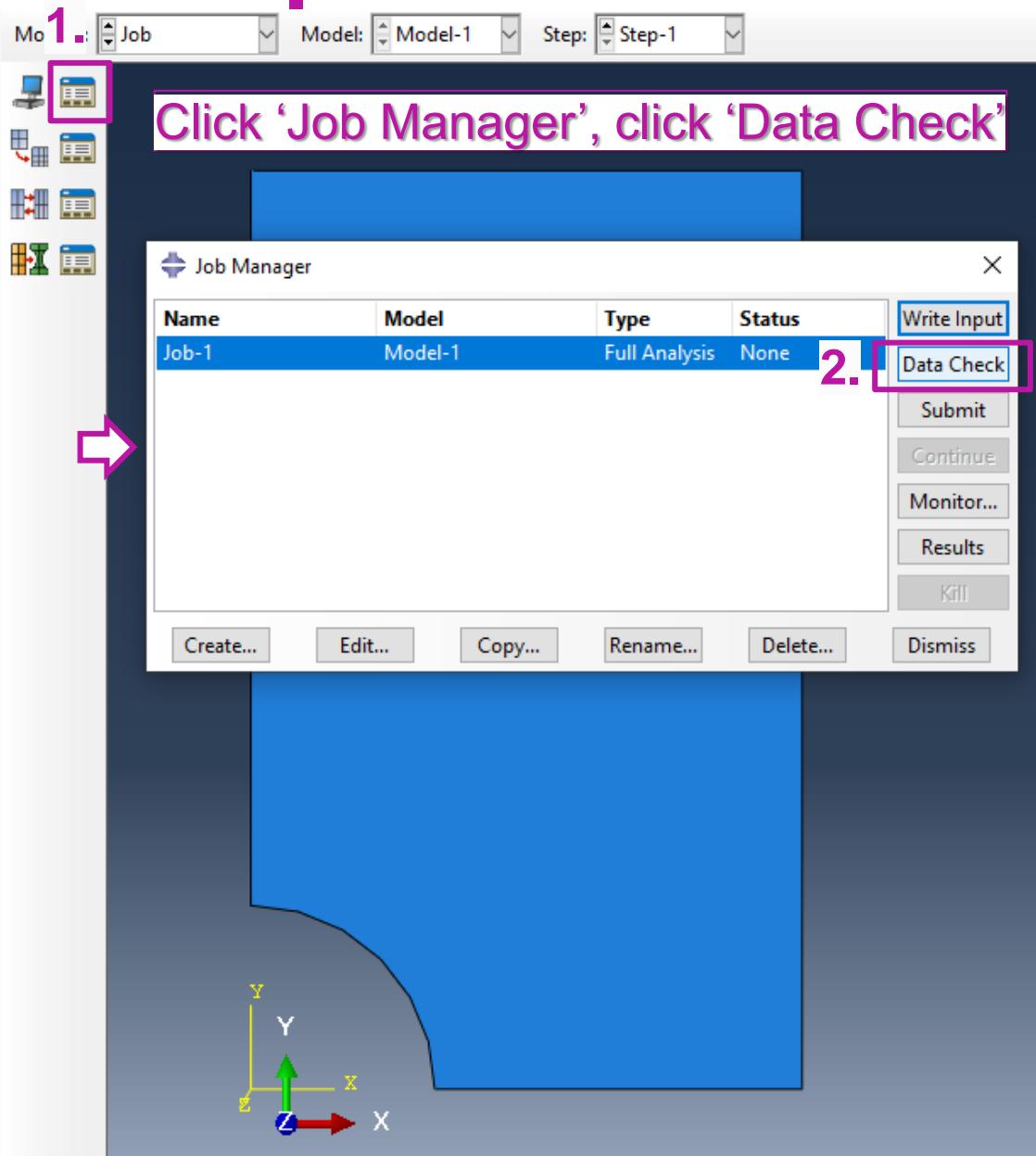
- 6.1 Create Job

Change to 'Job' Module, click 'Create Job', use the default settings.



Step 6: Create Job

- 6.2 Data check



3.
Checking process running...

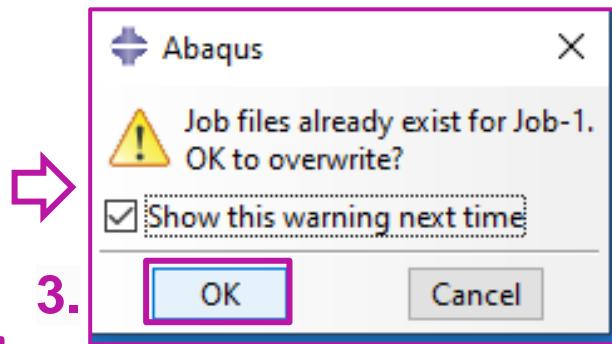
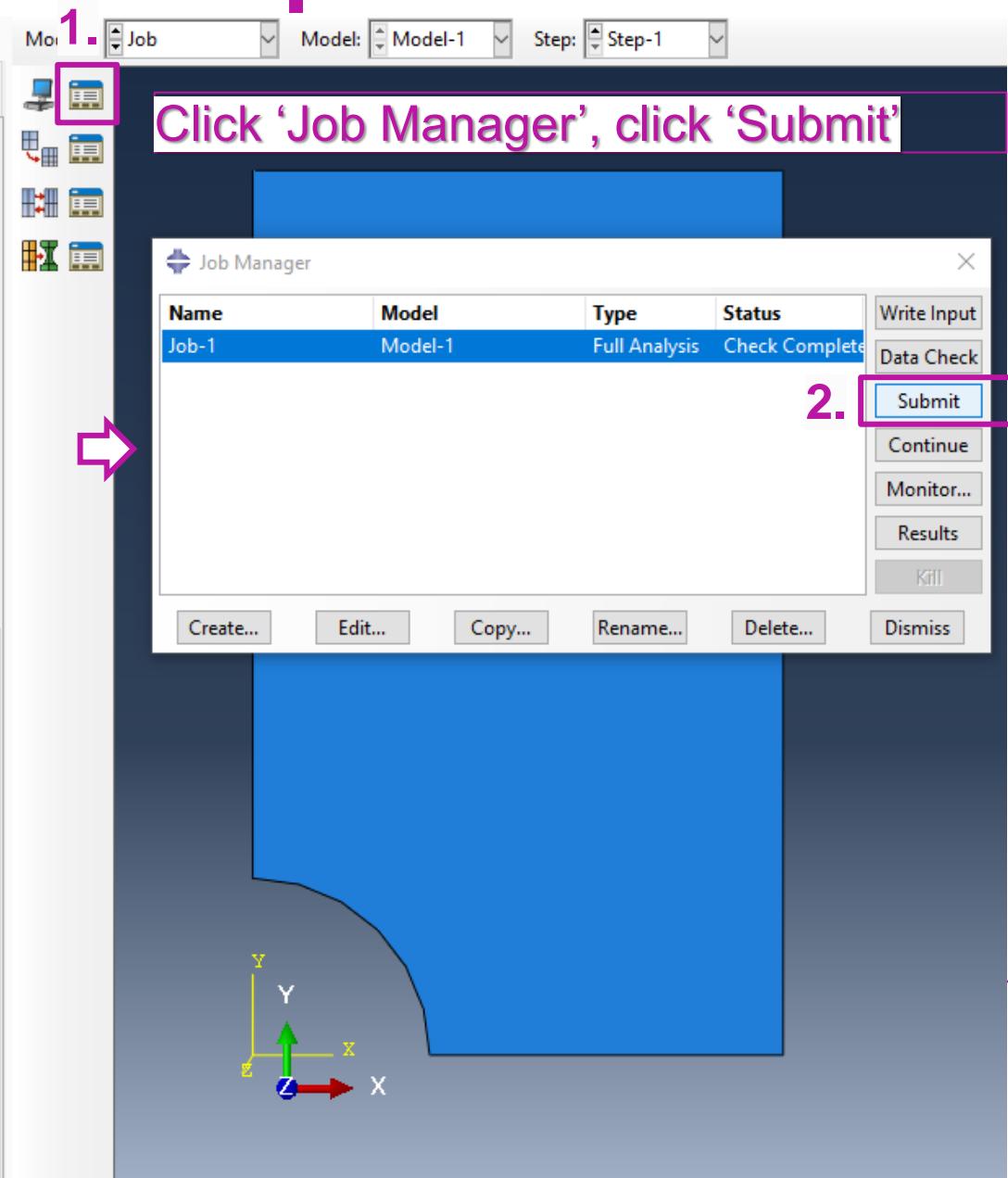


4.
'Check completed' / 'completed successfully'

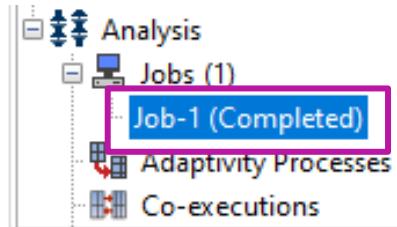
The job "Job-1" has been created.
The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Explicit Packager completed successfully.
Job Job-1: Abaqus/Explicit completed successfully.
Job Job-1 completed successfully.

Step 6: Create Job

- 6.3 Submit a job



3.



4.

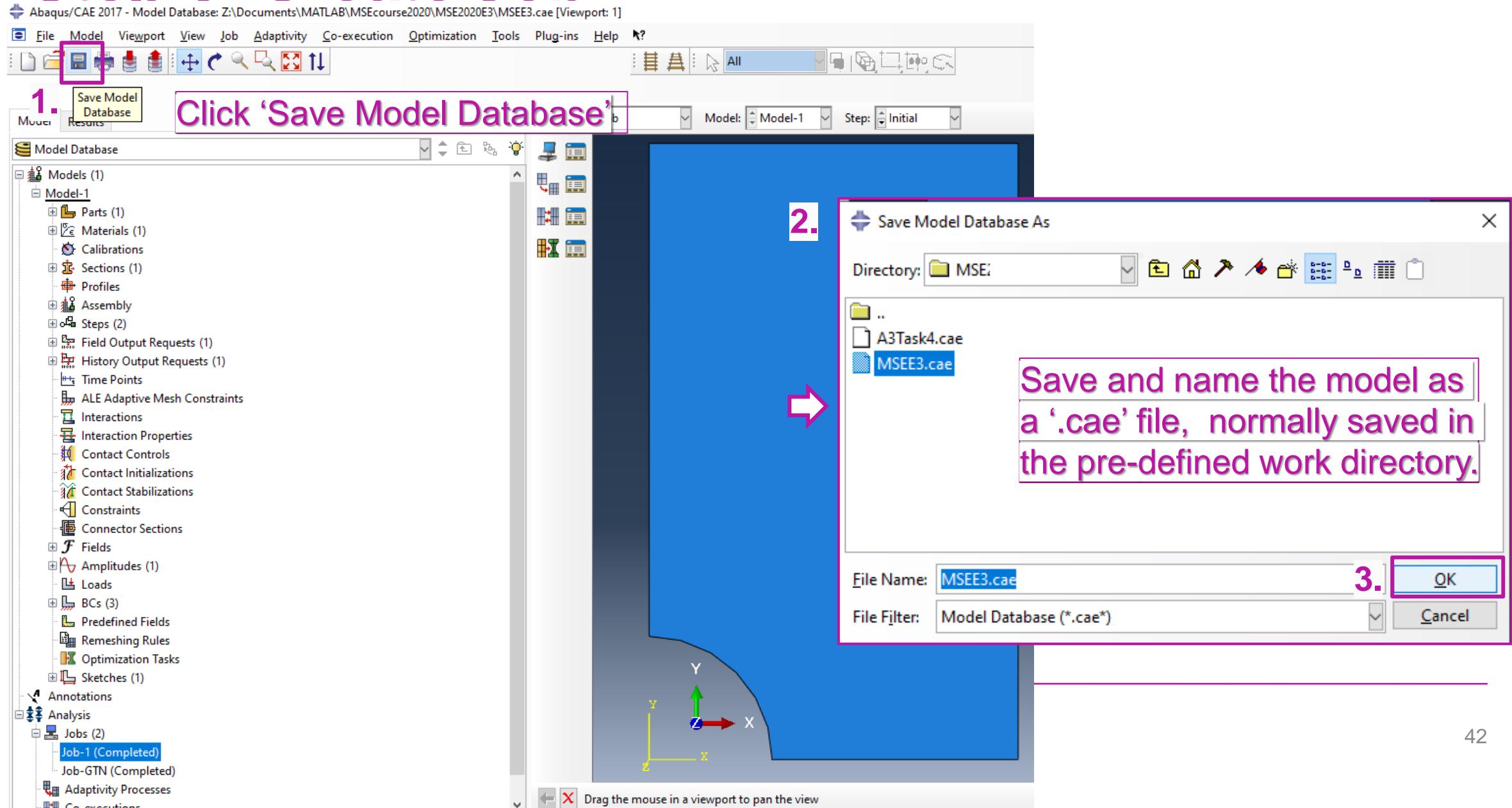
Job running...



The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Explicit Packager completed successfully.
Job Job-1: Abaqus/Explicit completed successfully.
Job Job-1 completed successfully

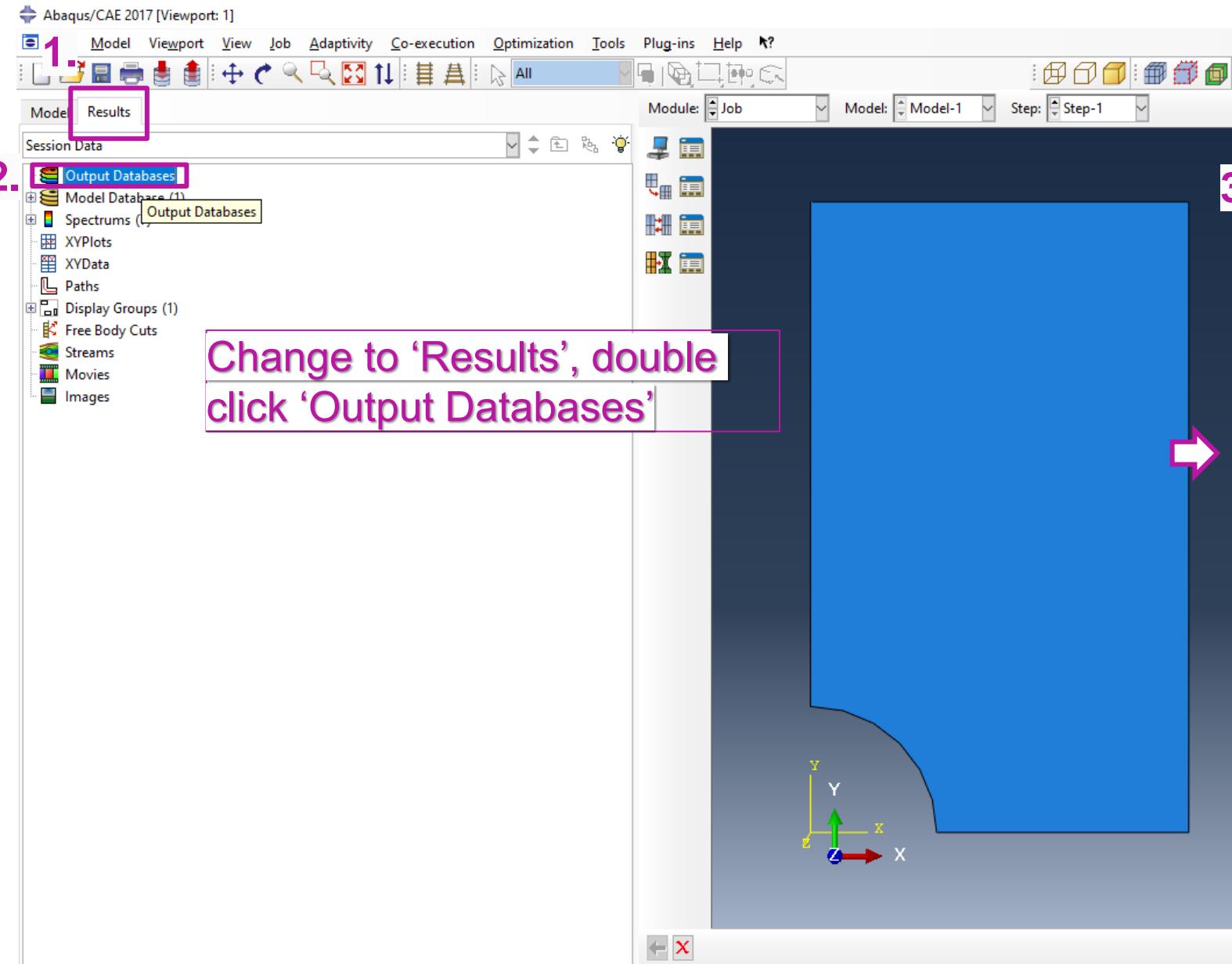
Step 6: Create Job

- 6.4 Save the model

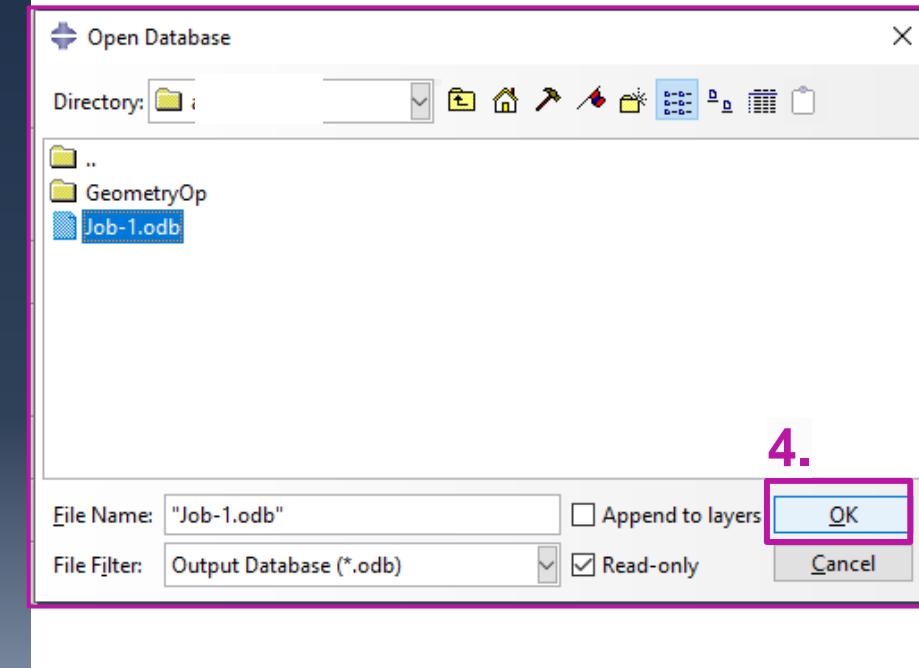


Step 7: Results

- 7.1 Open result file: .odb file

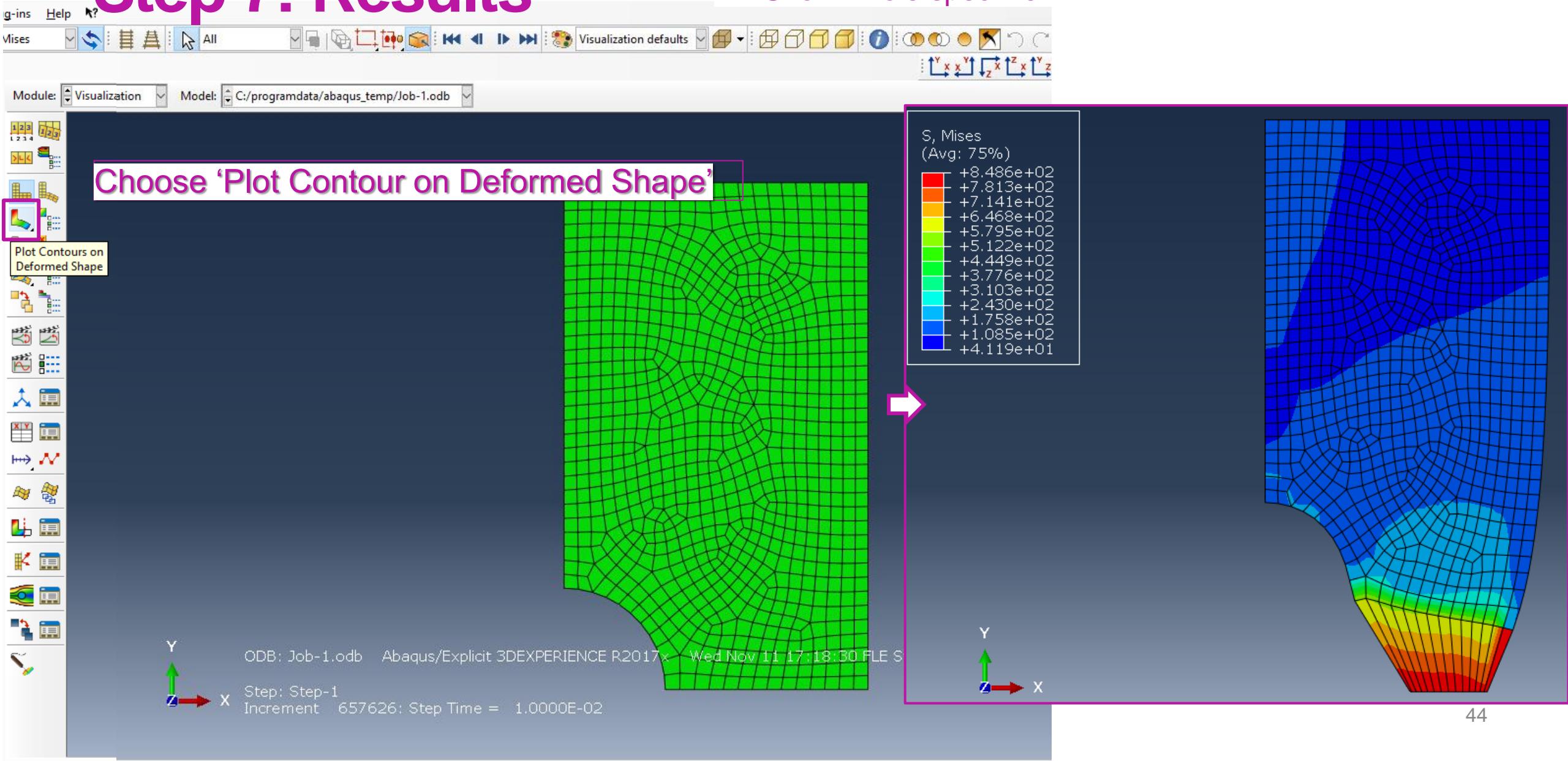


3. Change path to your work folder, select the '.odb' type file



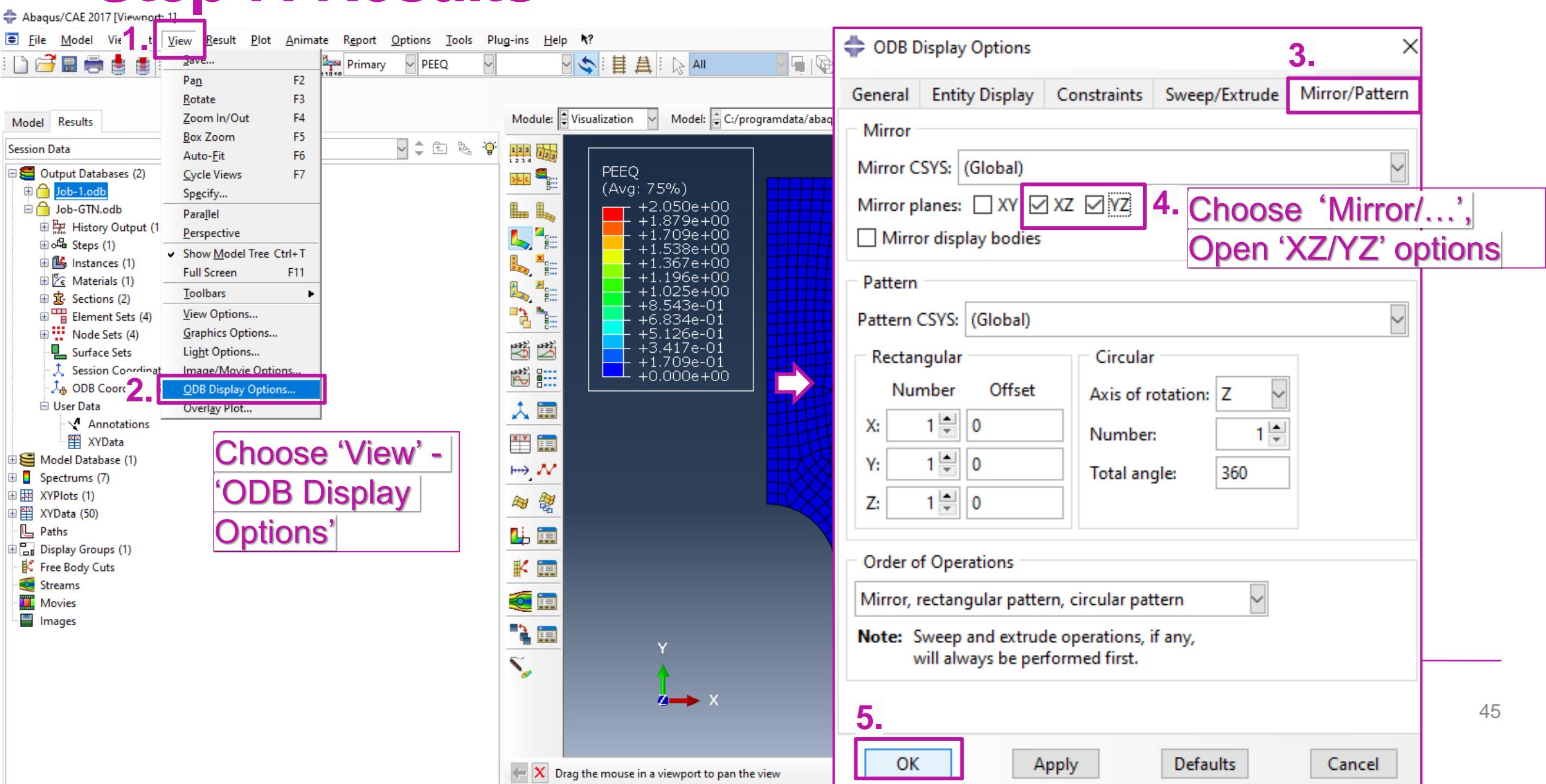
Step 7: Results

- 7.2 Show whole specimen



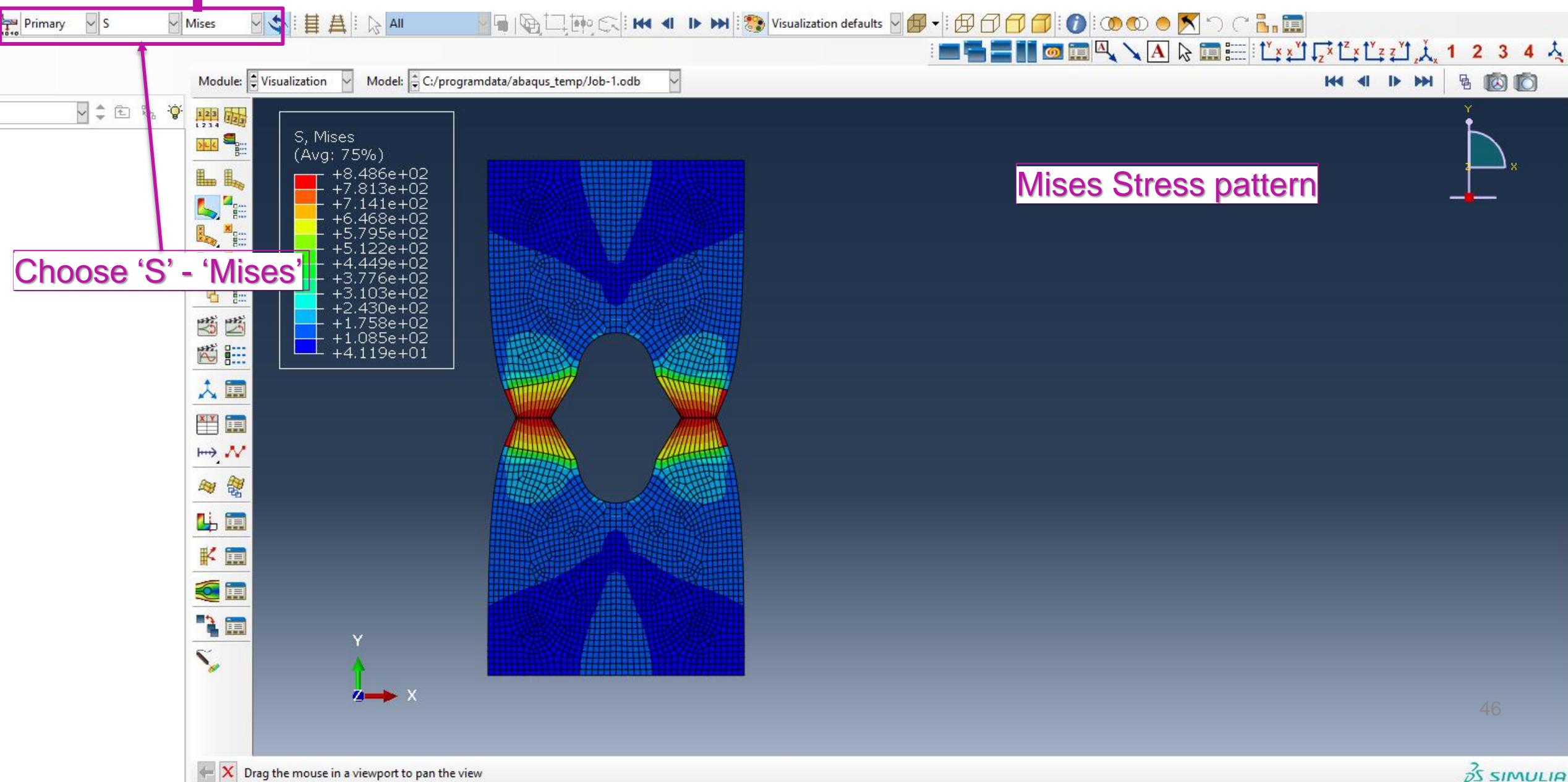
Step 7: Results

- 7.2 Show whole specimen



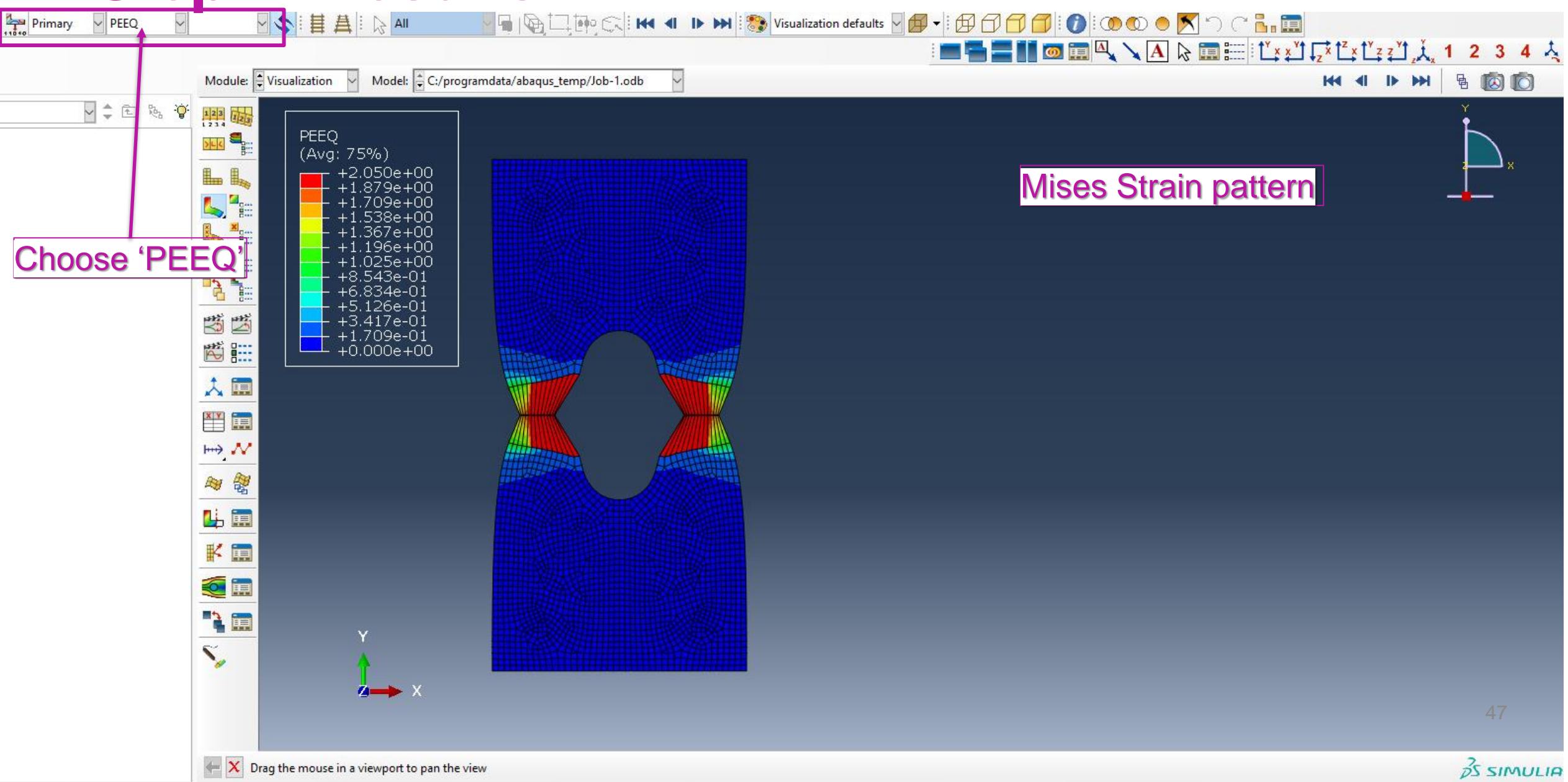
Step 7: Results

- 7.3 Mises stress pattern



Step 7: Results

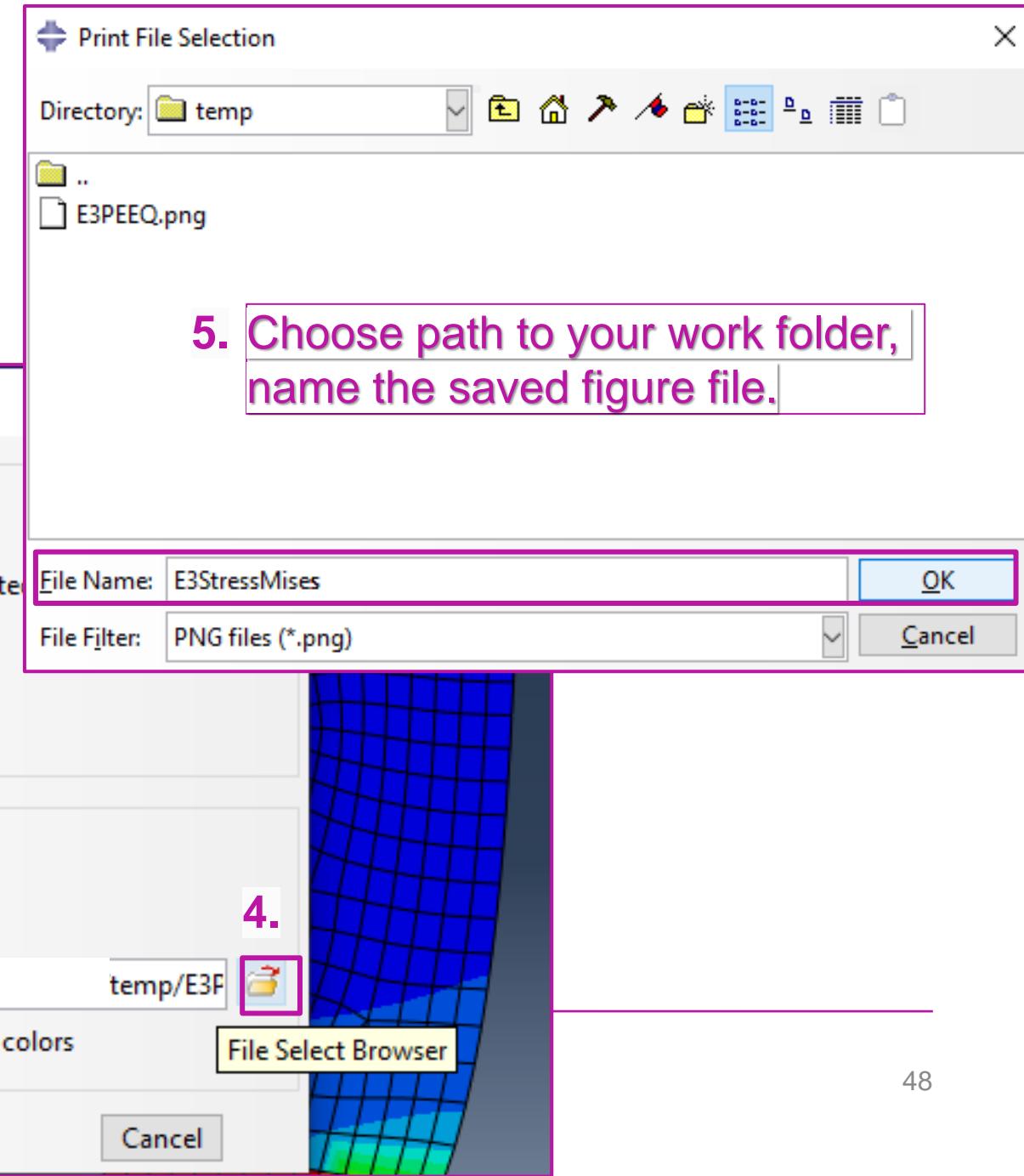
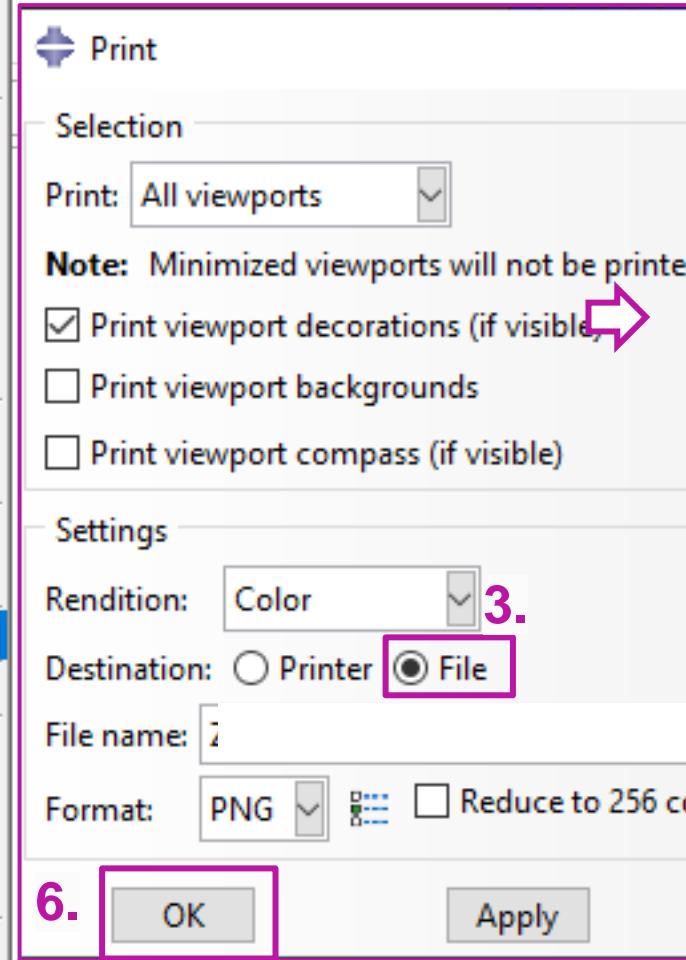
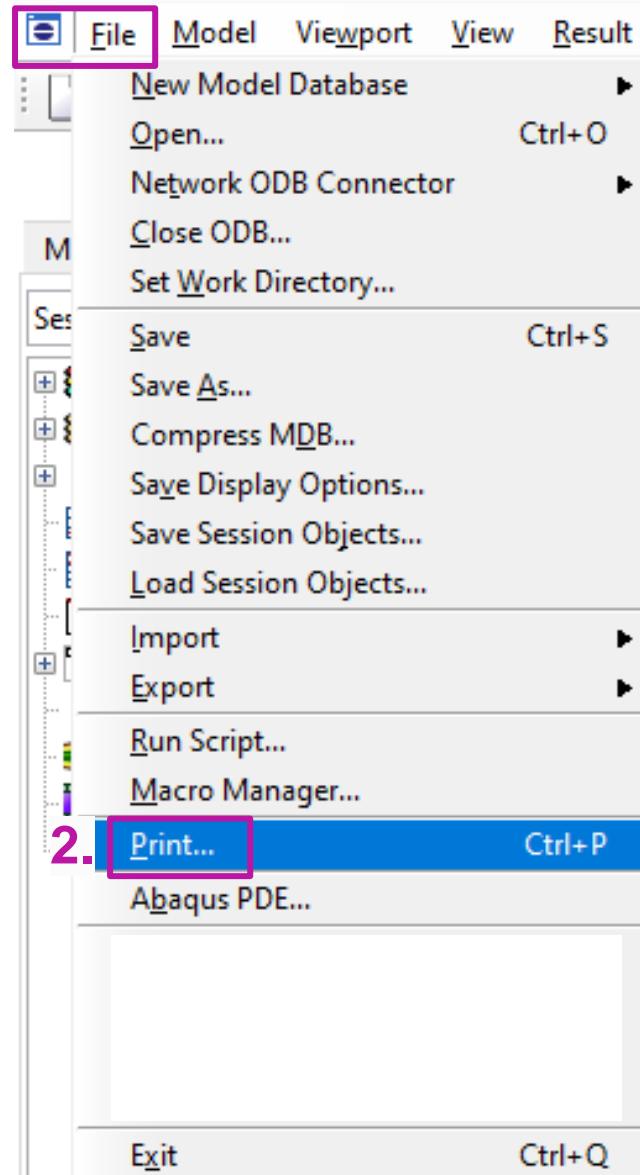
- 7.4 Mises strain pattern



Step 7: Results

• 7.5 Save figures

1. Abaqus/CAE 2017 [Viewport: 1]



Step 7: Results

- 7.6 Extract force and displacement data

Choose 'Create XY Data',
Choose 'ODB field output',
Choose 'Unique Nodal'

Module:

1. **2.** **3.** Continue...

4. **5.** Note: XY Data will be extracted from the active steps/frames Active Steps/Frames... **6.** Select 'RF:RF2', 'U:U2' to extract data along Y direction

XY Data from ODB Field Output

Steps/Frames

Note: XY Data will be extracted from the active steps/frames Active Steps/Frames...

Variables Elements/Nodes

Output Variables

Position: Unique Nodal

Click checkboxes or edit the identifiers shown next to Edit below.

PEEQ: Equivalent plastic strain

RF: Reaction force

Magnitude

RF1

RF2

S: Stress comp

U: Spatial displacement

Magnitude

U1

U2

Edit: U.U2,RF.RF2

Section point: All Select Settings...

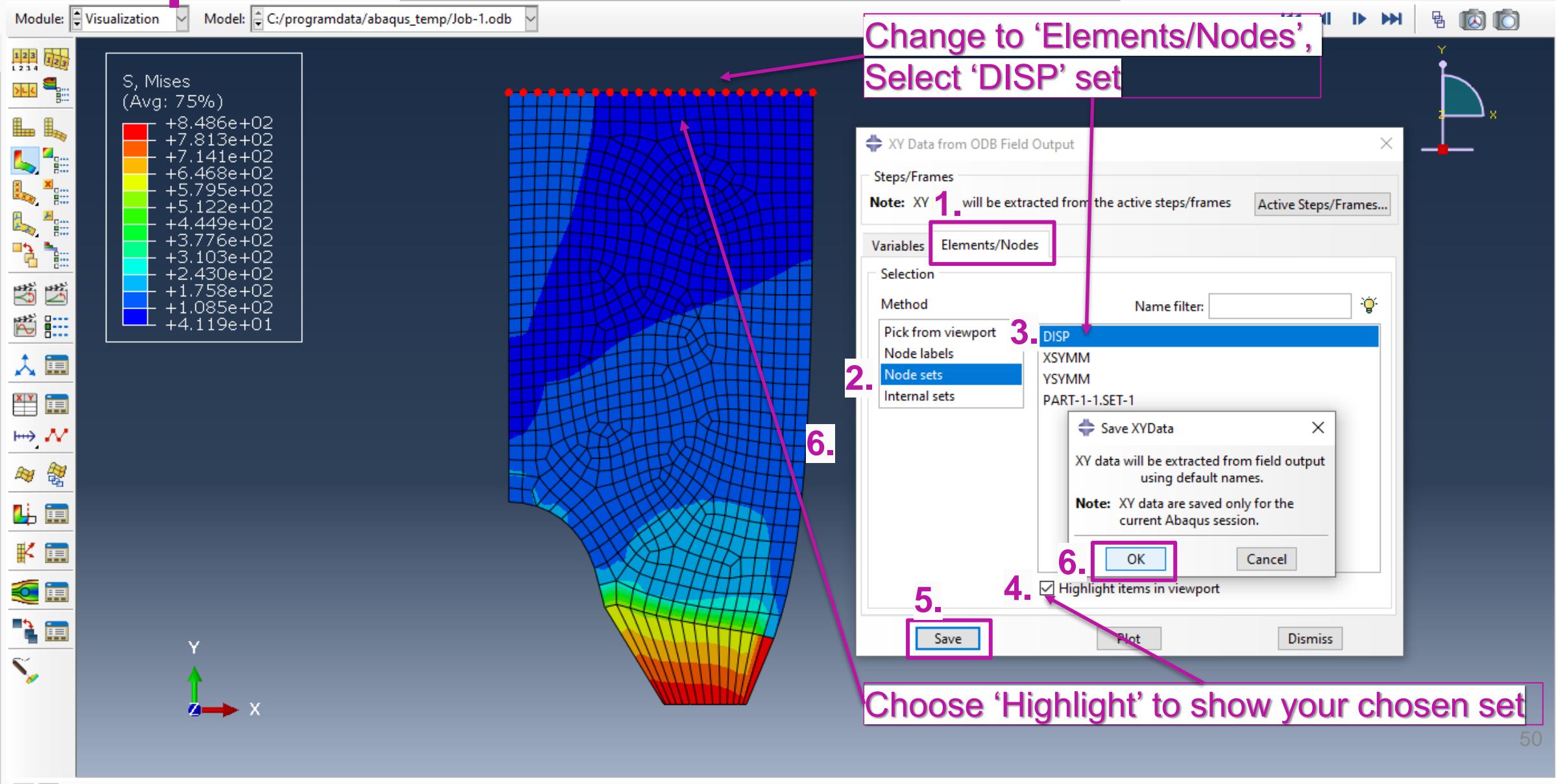
Save Plot Dismiss

Aalto University School of Engineering

49

Step 7: Results

- 7.6 Extract force and displacement data

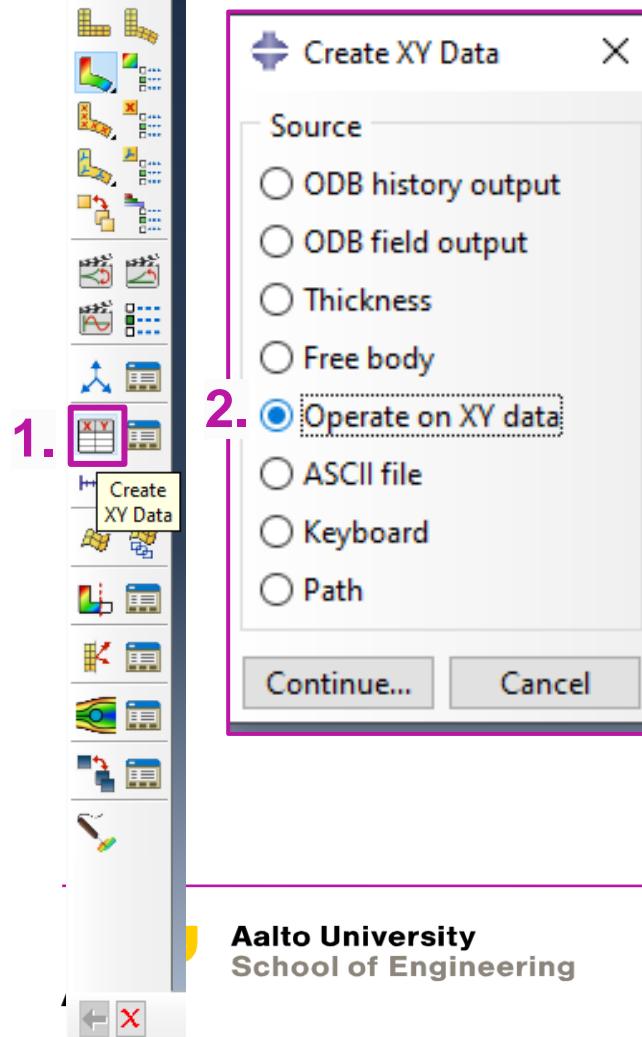


Step 7: Results

- 7.5 Calculate force

Total Force=(sum on all nodes)*2 due to symmetry

**Choose ‘Create XY Data’,
Choose ‘Operate on XY data’.**



1

- 7.5 Calculate force

Step 7: Results

Operate on XY Data

Enter an expression by typing and selecting XY Data and Operators below.

Example: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5"

sum (I)

avg(())

1. Check the position of the cursor

XY Data

Name filter:

Name	Description
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 PI: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
U:U2 PI: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 2	
U:U2 PI: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 10	
U:U2 PI: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 9:	
U:U2 PI: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 9:	

2. Add to Expression Skip checks

Total Force=(sum on all nodes)*2 due to symmetry

X

Operators

A - XYData, float, or integer
X - XYData
I - integer
F - float
sineButterworthFilter(X,F)
sin(A)
sinh(A)
smooth(X,I)
smooth2(X,F)
sqrt(A)
srss((X,X,...))
sum((A,A,...))
swap(X)
tan(A)
tanh(A)
truncate(X,F)
vectorMagnitude(X,X,X)

Select all RF:RF2 data, (use Shift),
then click 'Add to Expression'

- 7.5 Calculate force

Step 7: Results

Operate on XY Data

Enter an expression by typing and selecting XY Data and Operators below.

Example: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5"

PART-1-1 N: 104", "RF:RF2 Pl: PART-1-1 N: 105", "RF:RF2 Pl: PART-1-1 N: 106", "RF:RF2 Pl: PART-1-1 N: 107",
 "RF:RF2 Pl: PART-1-1 N: 108", "RF:RF2 Pl: PART-1-1 N: 109", "RF:RF2 Pl: PART-1-1 N: 110", "RF:RF2 Pl: PART-1-1
 N: 111")*2

1. add '*2' to expression

XY Data

Name filter:

Name	Description
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 2	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 10	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 9	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 1	

Operators

A - XYData, float, or integer
 X - XYData
 I - integer
 F - float

0
 +
 -
 *
 /
 1/A
 abs(A)
 acos(A)
 append((X,X,...))
 asin(A)

Add to Expression Skip checks

Create XY Data... Plot Expression Clear Expression Cancel

Click 'Save as...', name as 'Force', 'OK'

2.

Total Force=(sum on all nodes)*2 due to symmetry

Operate on XY Data

Enter an expression by typing and selecting XY Data and Operators below.

Example: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5"

PART-1-1 N: 104", "RF:RF2 Pl: PART-1-1 N: 105", "RF:RF2 Pl: PART-1-1 N: 106", "RF:RF2 Pl: PART-1-1 N: 107",
 "RF:RF2 Pl: PART-1-1 N: 108", "RF:RF2 Pl: PART-1-1 N: 109", "RF:RF2 Pl: PART-1-1 N: 110", "RF:RF2 Pl: PART-1-1
 N: 111")*2

XY Data

Name filter:

Name	Description
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
RF:RF2 Pl: PART-1- From Field Data: RF:RF2 at part instance PART-1-1 node	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 2	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 10	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 9	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 1	

Operators

A - XYData, float, or integer
 X - XYData
 I - integer
 F - float

0
 +
 -
 *
 /
 1/A
 abs(A)
 acos(A)
 append((X,X,...))
 asin(A)
 atan(A)
 avg((A,A,...))
 butterworthFilter(X,F)

Save XY Data As

3. Name: Force

Note: The created XY Data object is saved only for the current Abaqus session.

4. OK Cancel

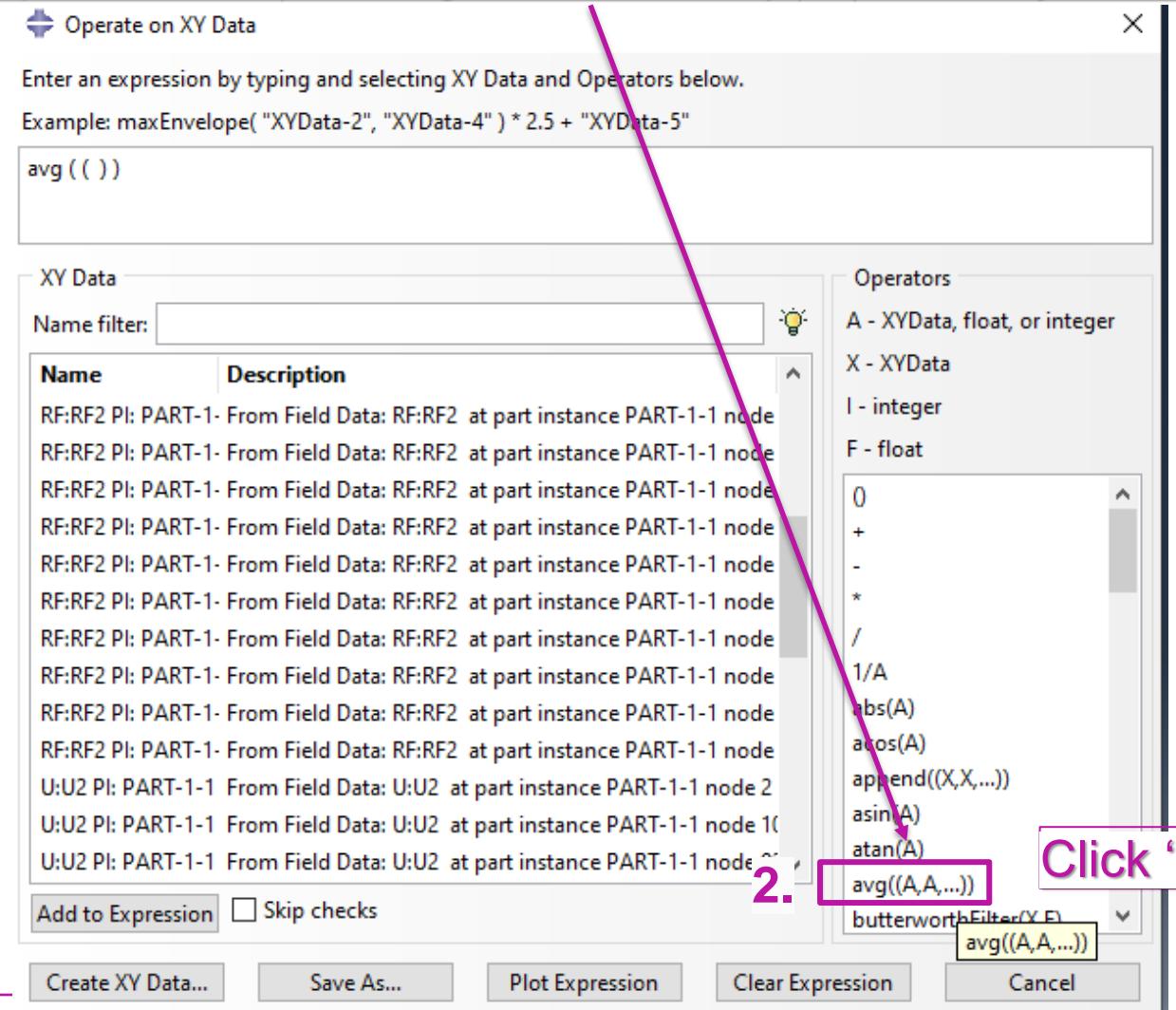
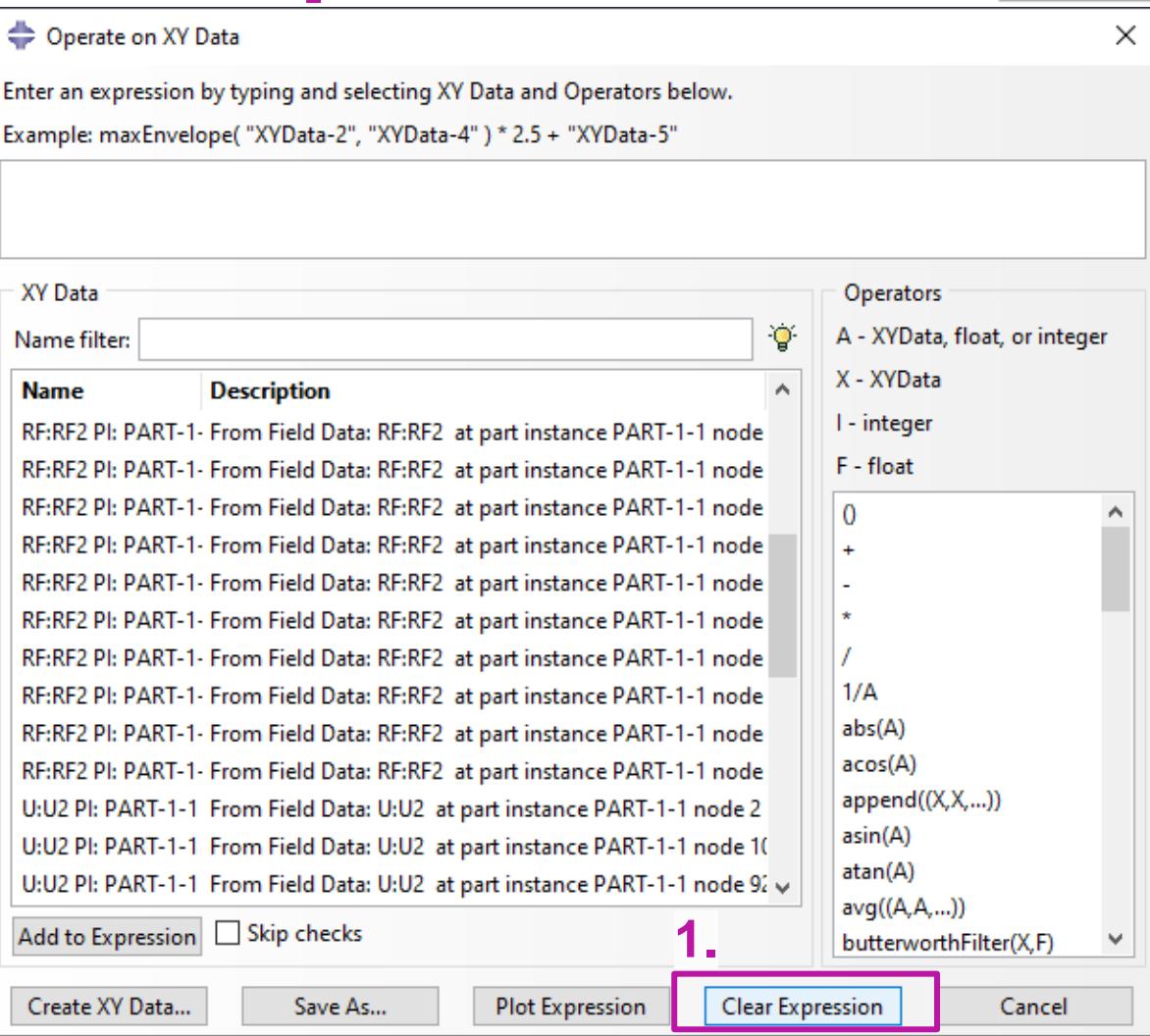
Add to Expression Skip checks

Create XY Data... Save As... Plot Expression Clear Expression Cancel

Step 7: Results

- 7.6 Calculate displacement

Total Displacement=(average of all nodes)*2 due to symmetry



Click 'avg'

Step 7: Results

- 7.6 Calculate displacement

Total Displacement = (average of all nodes) * 2 due to symmetry

Operate on XY Data

Enter an expression by typing and selecting XY Data and Operators below.

Example: maxEnvelope("XYData-2", "XYData-4") * 2.5 + "XYData-5"

"U:U2 Pl: PART-1-1 N: 102", "U:U2 Pl: PART-1-1 N: 103", "U:U2 Pl: PART-1-1 N: 104", "U:U2 Pl: PART-1-1 N: 105", "U:U2 Pl: PART-1-1 N: 106", "U:U2 Pl: PART-1-1 N: 107", "U:U2 Pl: PART-1-1 N: 108", "U:U2 Pl: PART-1-1 N: 109", "U:U2 Pl: PART-1-1 N: 110", "U:U2 Pl: PART-1-1 N: 111"))*2

XY Data

1. add '*2' to expression

Name filter:

Name	Description
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 99	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 100	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 101	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 102	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 103	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 104	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 105	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 106	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 107	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 108	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 109	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 110	
U:U2 Pl: PART-1-1 From Field Data: U:U2 at part instance PART-1-1 node 111	

Add to Expression Skip checks

Create XY Data...

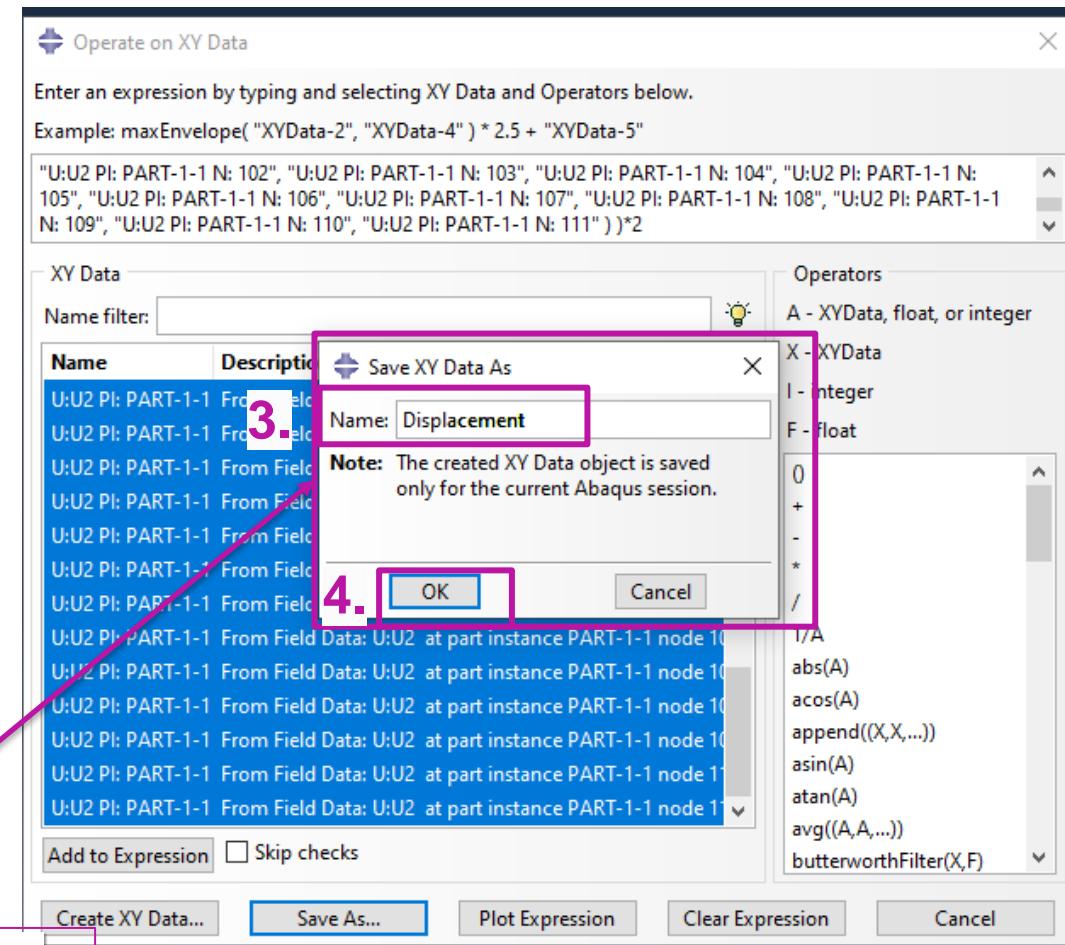
2. Save As...

Plot Expression

Clear Expression

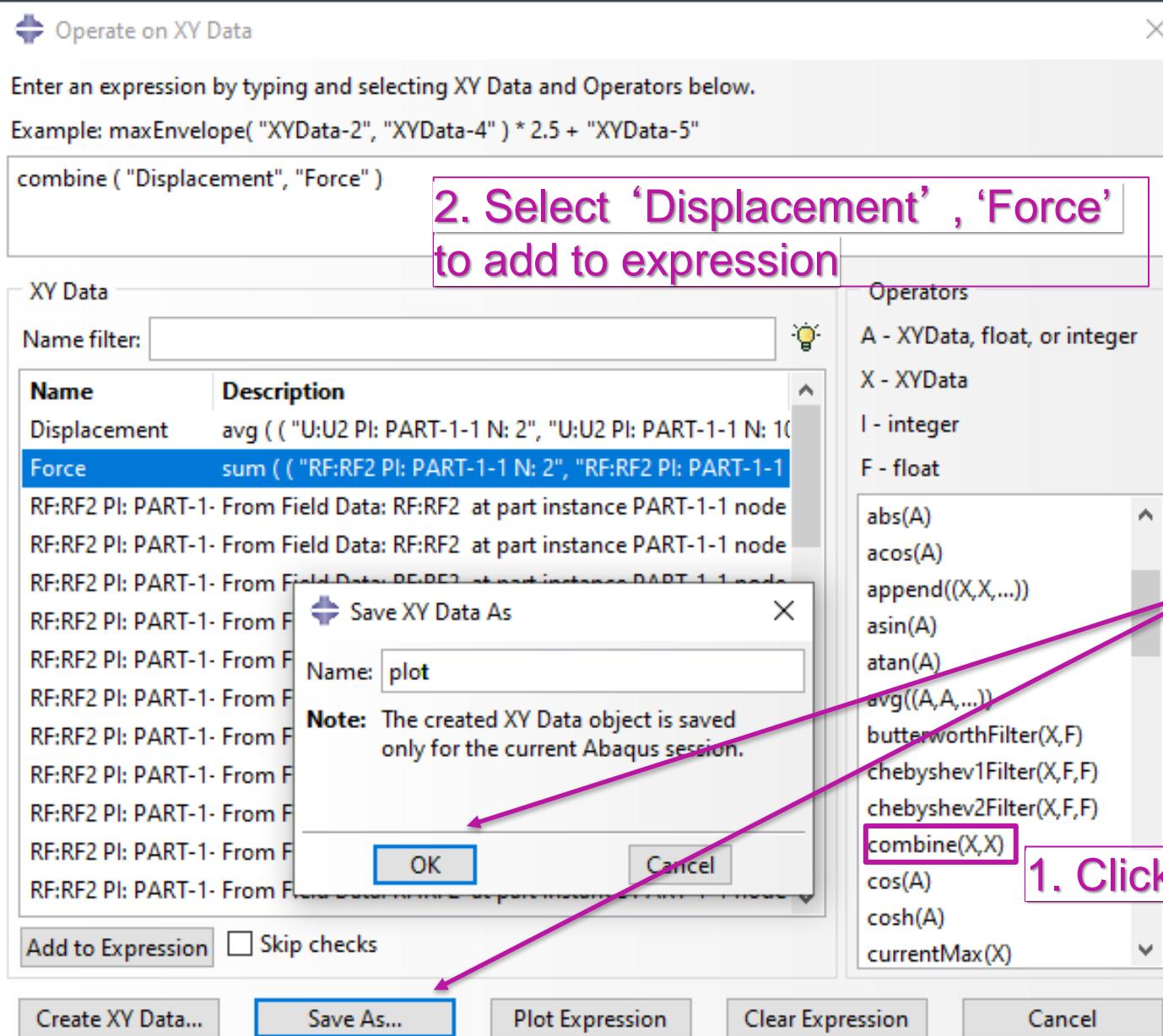
Cancel

Click 'Save as...', name as 'Displacement', 'OK'



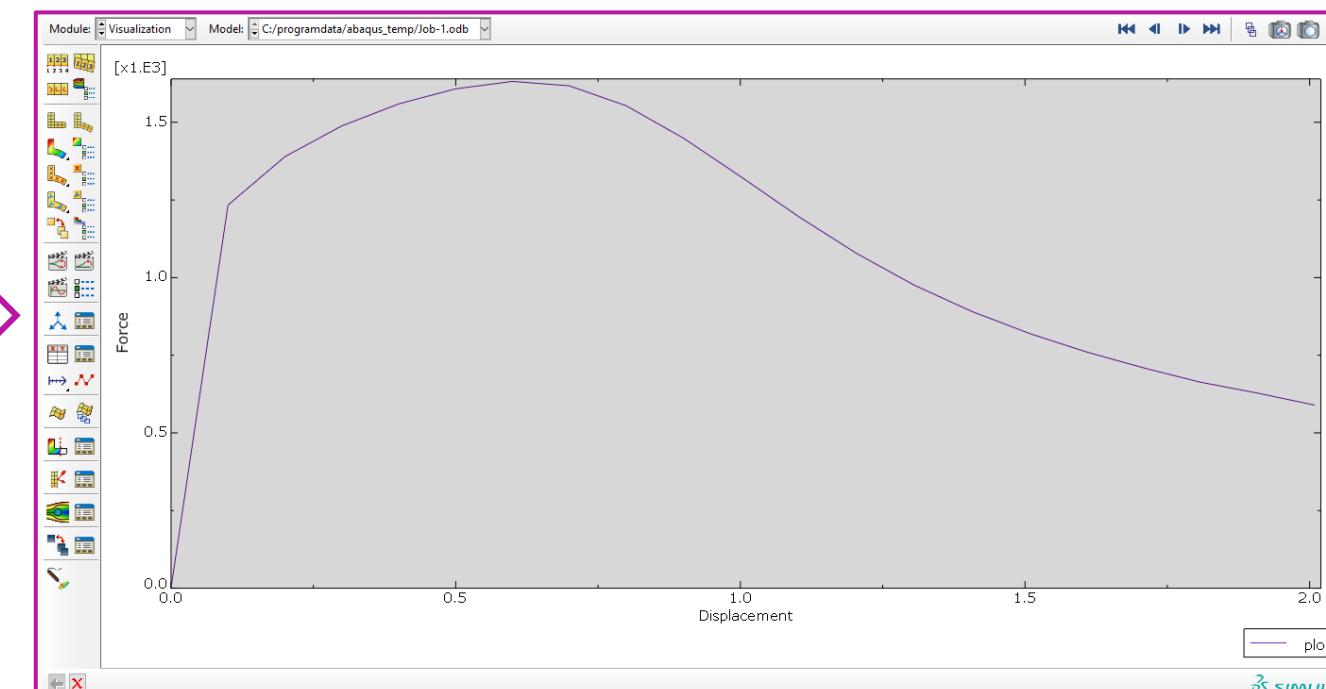
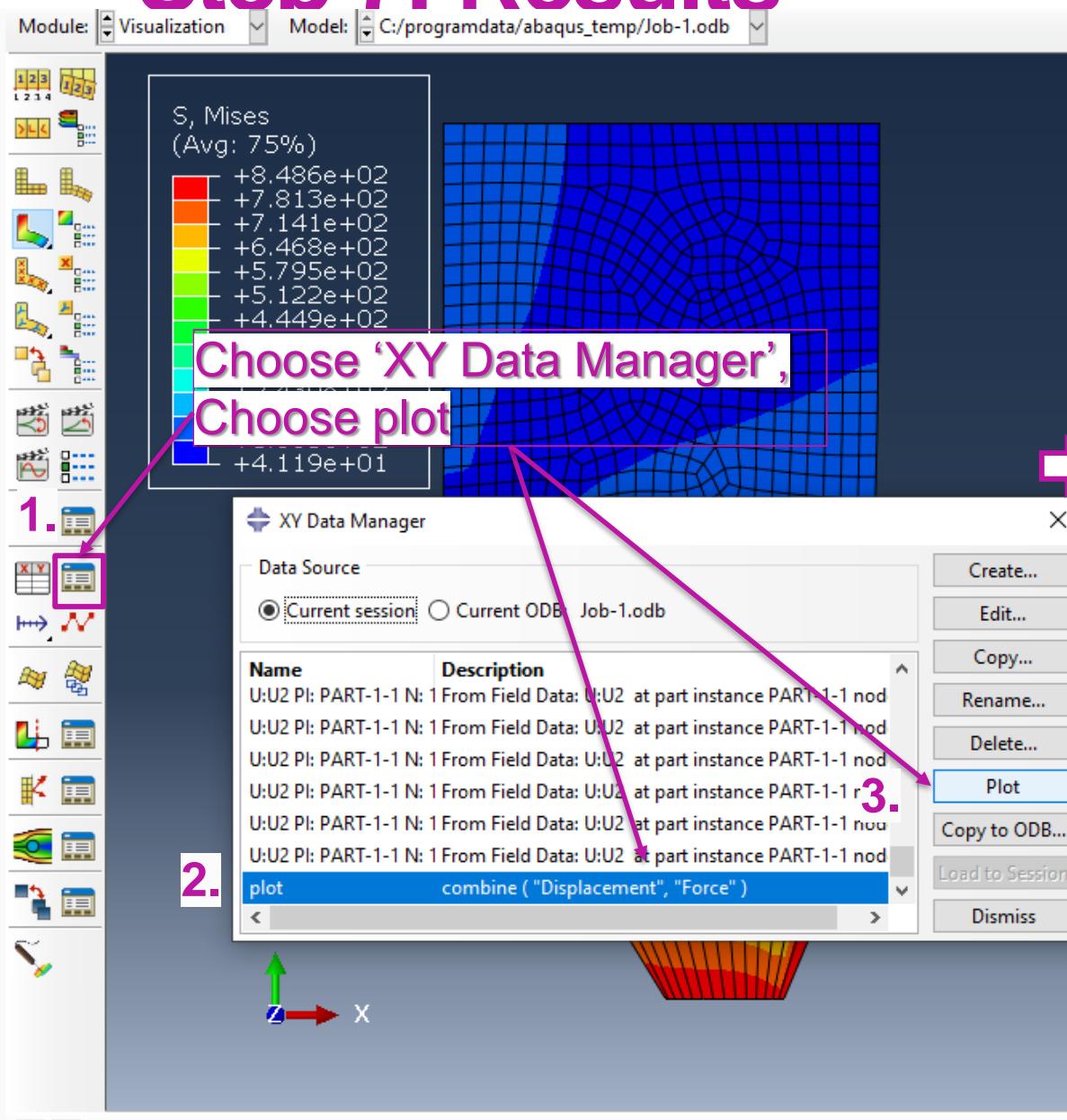
Step 7: Results

- 7.7 Plot force-displacement curve



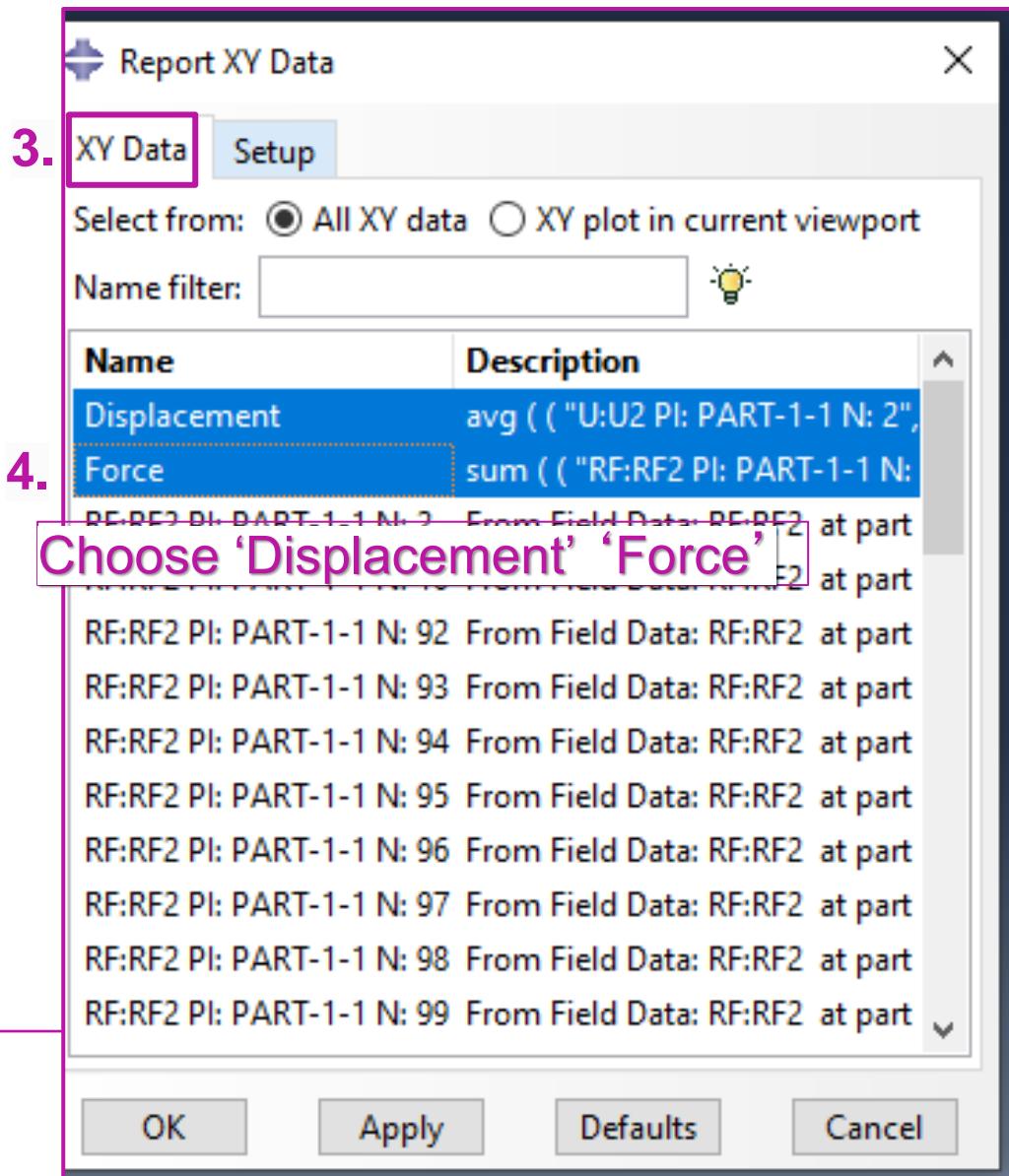
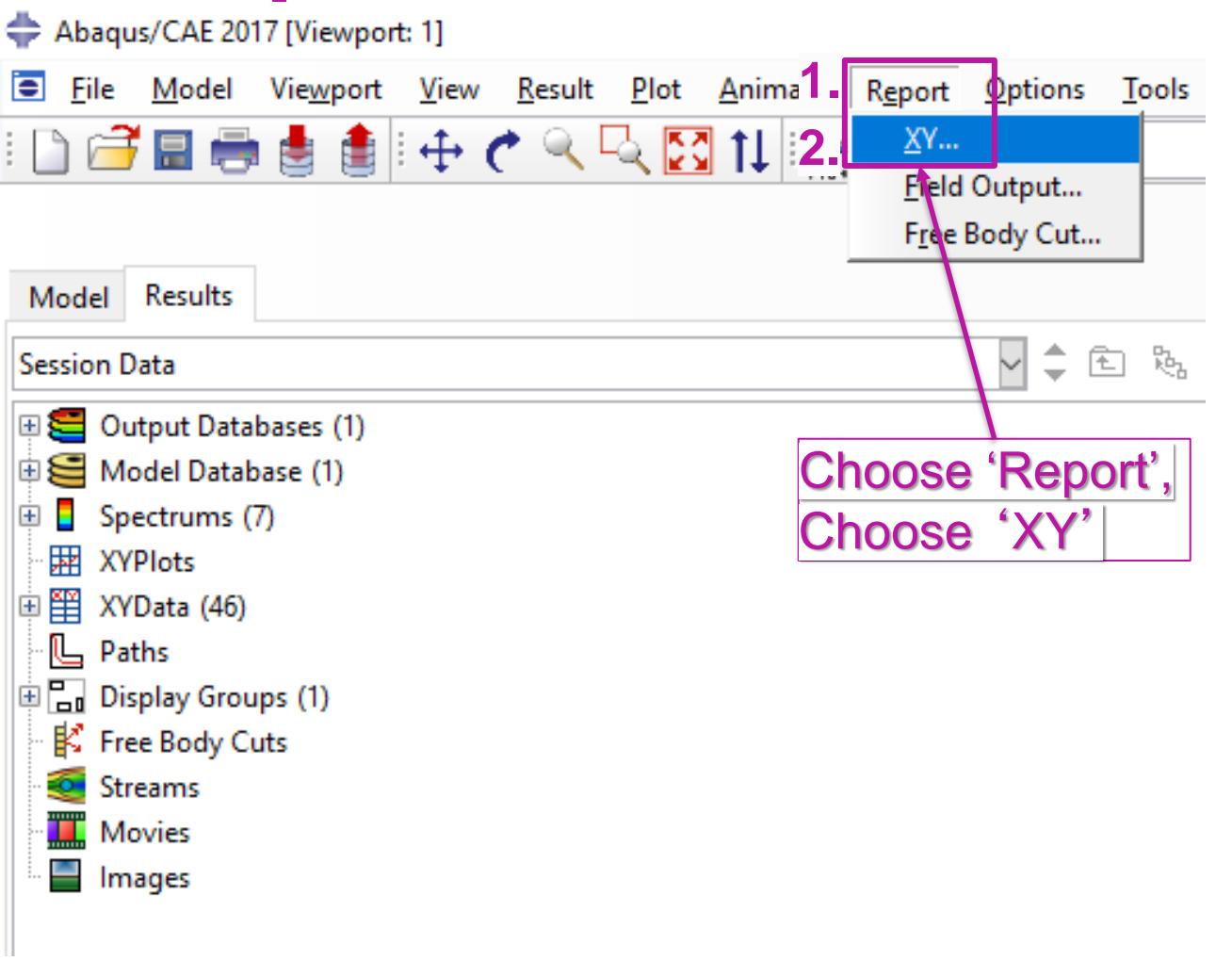
• 7.7 Plot force-displacement curve

Step 7: Results



Step 7: Results

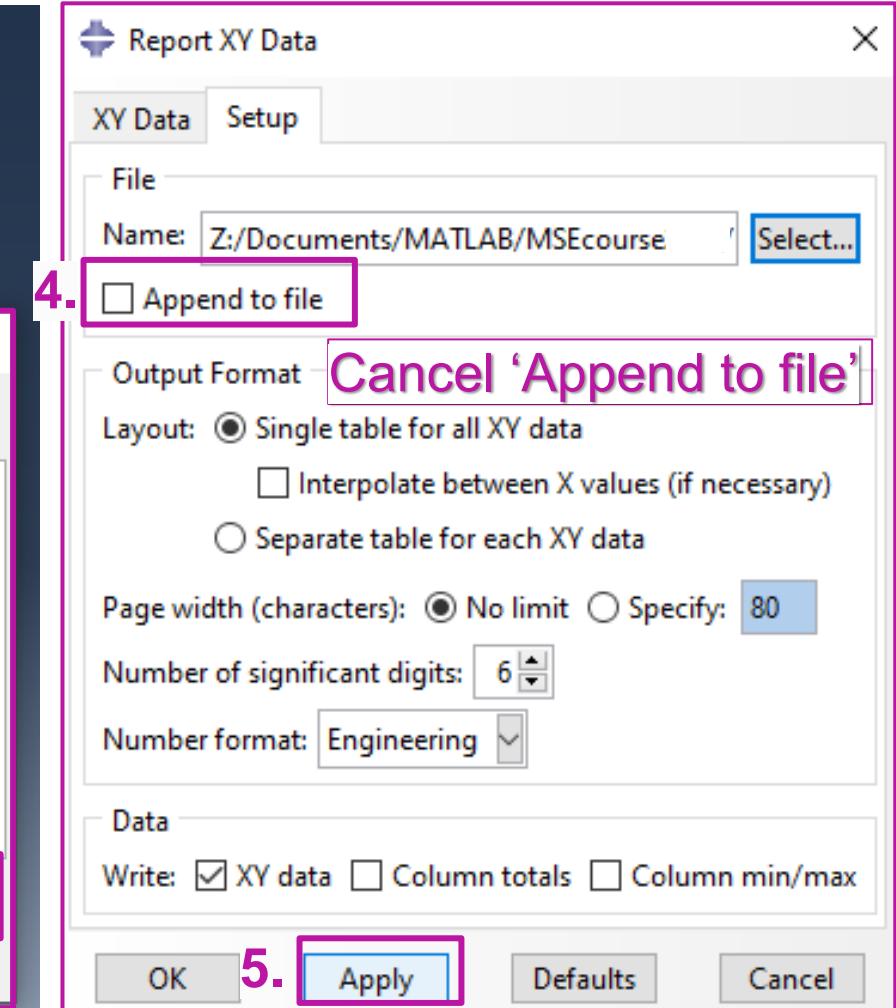
- 7.8 Export force-displacement data



- 7.8 Export force-displacement data

Step 7: Results

In 'Setup', choose path to your work folder, name the saved file.



The screenshot shows the 'Report XY Data' dialog box. The 'File' tab is selected. The 'Name' field contains 'Z:/Documents/MATLAB/MSEcourse'. The 'Select...' button is highlighted with a pink box. Below it, the 'Append to file' checkbox is checked and highlighted with a pink box. The 'Output Format' section includes 'Layout: Single table' (selected), 'Interpolate between X values (if necessary)' (unchecked), and 'Separate table for each XY data' (unchecked). The 'Page width (characters)' field is set to 'No limit'. The 'Number of significant digits' is set to 6. The 'Number format' dropdown is set to 'Engineering'. The 'Data' section has 'XY data' checked and 'Column totals' unchecked. Buttons at the bottom include 'OK', 'Apply', 'Defaults', and 'Cancel'.

1. In 'Setup', choose path to your work folder, name the saved file.

2. Select...

3. E3ForceDisp.rpt

4. Append to file

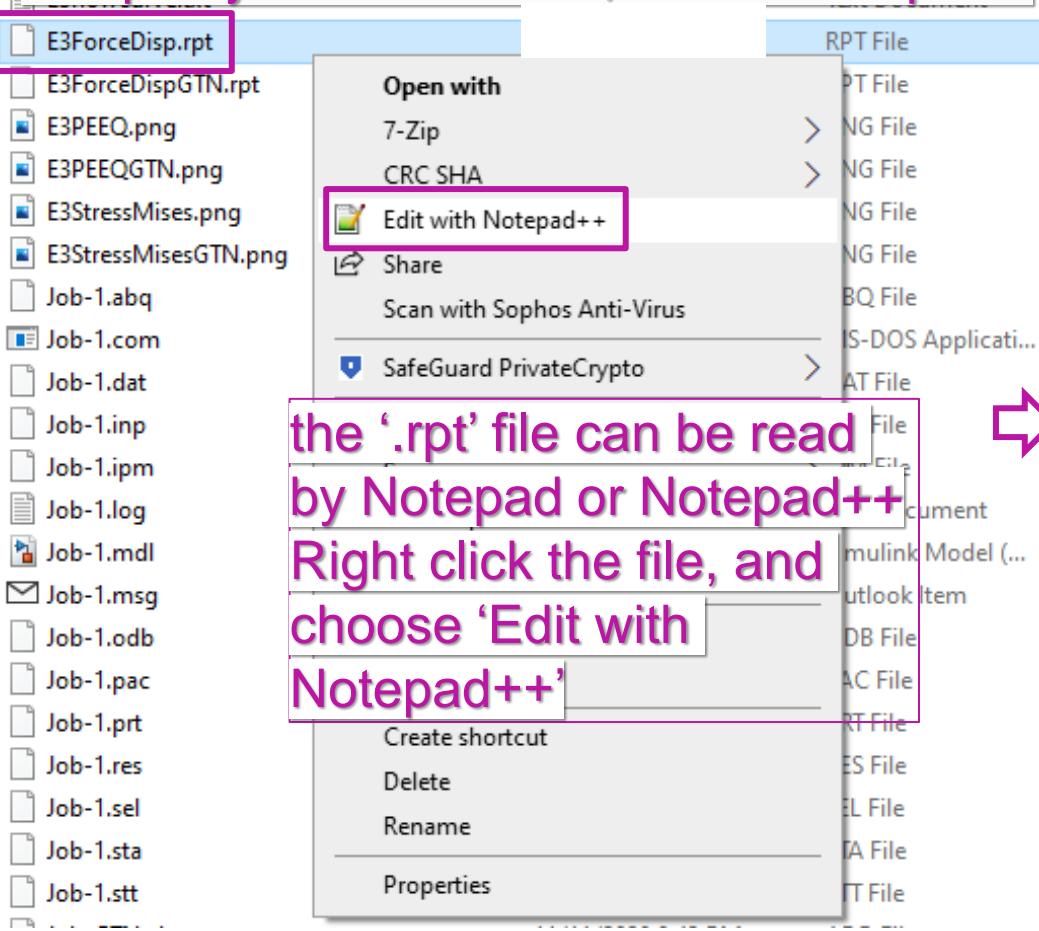
Cancel 'Append to file'

5. Apply

- 7.8 Export force-displacement data

Step 7: Results

1. Open your work folder, select the .rpt file



the '.rpt' file can be read
by Notepad or Notepad++
Right click the file, and
choose 'Edit with
Notepad++'

.rpt file for the exported data

NE3ForceDisp.rpt - Notepad++

File Edit Search View Encoding Language Settings Tools Macro Run TextFX Plugins Window ?

2nd Displacement 3rd Force

1st X: Step time

	X	Displacement	Force
1			
2			
3			
4	0.	0.	1.30984
5	500.003E-06	100.008E-03	1.23347E+03
6	1.00001E-03	200.187E-03	1.38968E+03
7	1.50001E-03	300.008E-03	1.48842E+03
8	2.00001E-03	399.829E-03	1.5592E+03
9	2.50001E-03	499.65E-03	1.60769E+03
10	3.00001E-03	599.47E-03	1.63157E+03
11	3.50001E-03	699.291E-03	1.61751E+03
12	4.00001E-03	799.396E-03	1.55331E+03
13	4.5E-03	900.727E-03	1.44815E+03
14	5.00001E-03	1.00206	1.32318E+03
15	5.50001E-03	1.1034	1.19512E+03
16	6.E-03	1.20473	1.07722E+03
17	6.50001E-03	1.30606	975.062
18	7.00001E-03	1.4074	890.342
19	7.5E-03	1.50873	819.569
20	8.00001E-03	1.60895	760.047
21	8.50001E-03	1.7073	709.873
22	9.00001E-03	1.80622	663.928
23	9.50001E-03	1.90793	628.27
24	10.E-03	2.00964	589.134
25			
26			

Normal text file length : 1,486 lines : 29 Ln : 1 Col : 1 Sel : 0 | 0 Windows (CR LF) UTF-8 INS

Abaqus - Damage modeling (GTN)

Abaqus - GTN model simulation

Based on the elastoplastic model, make the following changes:

- Step1: Add the material parameters in ‘Material’
- Step2: Change the element type in ‘Mesh’
- Step3: Add the field output variables in ‘Field Output’
- Step4: Create a new job and submit in ‘Job’
- Step5: Check new results in ‘Results’

Step 1: Edit Material

Model Results Material Library

Module: Property Model: Mo

Edit Material

Name: Material-1

Description:

Material Behaviors

Density

Elastic Plastic

Mechanical

General Mechanical Thermal Electrical/Magnetic Other

Elasticity

Plasticity

Damage for Ductile Metals

Damage for Traction Separation Laws

Damage for Fiber-Reinforced Composites

Damage for Elastomers

Deformation Plasticity

Damping

Expansion

Brittle Cracking

Eos

Viscosity

Super Elasticity

Plastic

Cap Plasticity

Cast Iron Plasticity

Clay Plasticity

Concrete Damaged Plasticity

Concrete Smeared Cracking

Crushable Foam

Drucker Prager

Mohr Coulomb Plasticity

Porous Metal Plasticity

Soft Rock Plasticity

Creep

Swelling

Viscous

Relative density: 0.95

Use temperature-dependent data

Number of field variables: 0

Data

	q1	q2	q3
1	1.5	1	2.25

Suboptions

Porous Failure Criteria

Void Nucleation

OK Cancel

OK Cancel

Model Database

Models (1)

Model-1

Part-1

Features (1)

Shell planar-1

Section Sketch

Sets (1)

Surfaces

Skins

Stringers

Section Assignments (1)

Orientations

Composite Layups

Engineering Features

Materials (1)

Material-1

Calibrations

Sections (1)

Section-1

Profiles

Assembly

Instances (1)

Position Constraints

Features (1)

Sets (4)

Surfaces

Connector Assignments

Engineering Features

Steps (2)

Initial

Step-1

Field Output Requests (1)

History Output Requests (1)

Material-1

Name: Material1

Description:

Step 1: Edit Material

Material Behaviors

Density

Elastic

Plastic

Porous Metal Plasticity

Void Nucleation

General Mechanical Thermal Electrical/Magnetic Other

Porous Metal Plasticity

Relative density: 0.95

 Use temperature-dependent data

Number of field variables: 0

Data

	q1	q2	q3
1	1.5	1	2.25

- ▼ Suboptions
- Porous Failure Criteria
 - Void Nucleation

Suboption Editor

Porous Failure Criteria

Total volume void fraction at total failure: 0.45

Critical void volume fraction: 0.015

Suboption Editor

Void Nucleation

Use temperature-dependent data

Number of field variables: 0

Data

	Mean	Standard Deviation	Volume Fraction
1	0.12	0.1	0.00215

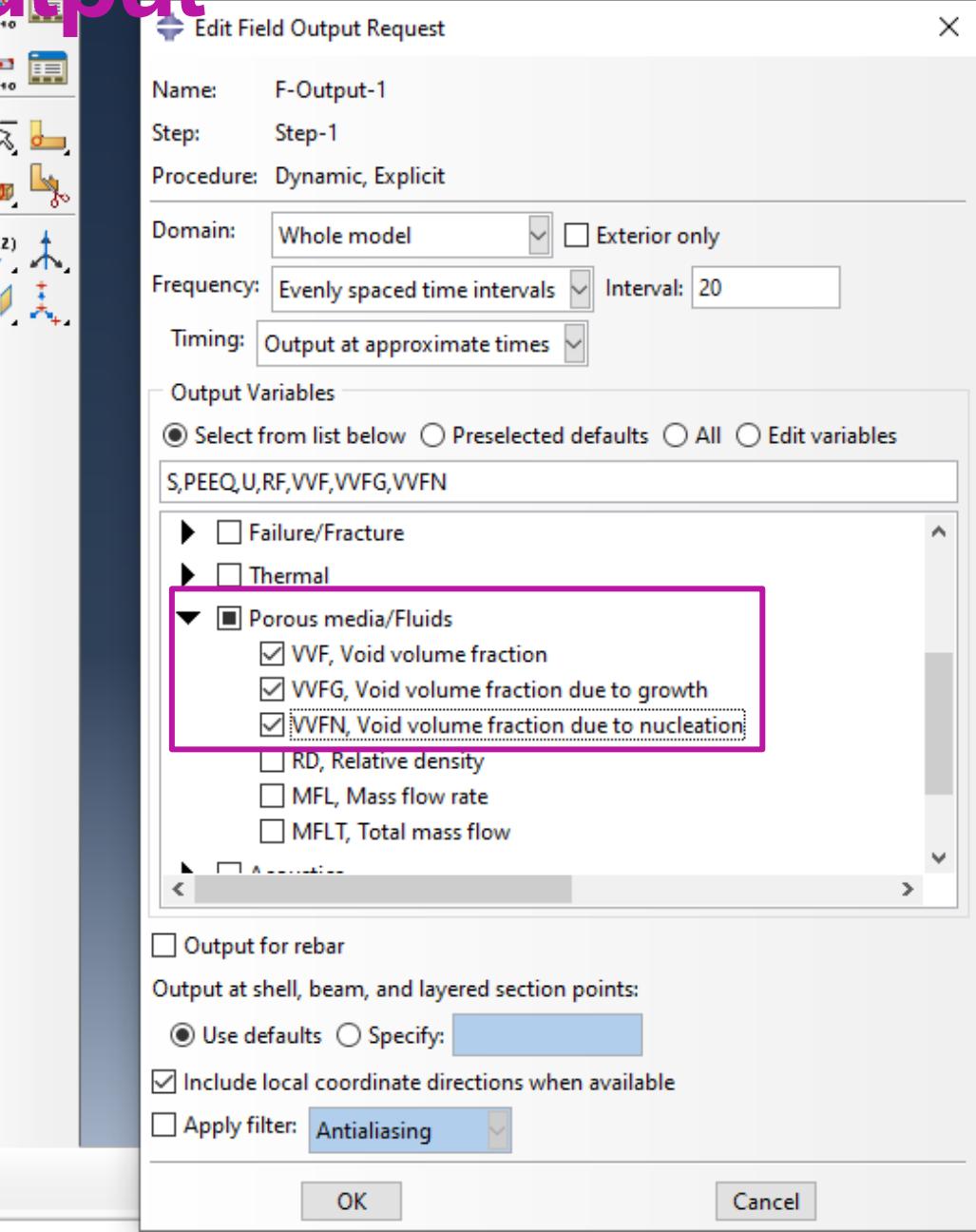
Model Results

Module: Step Model: Model-1 Step: Step-1

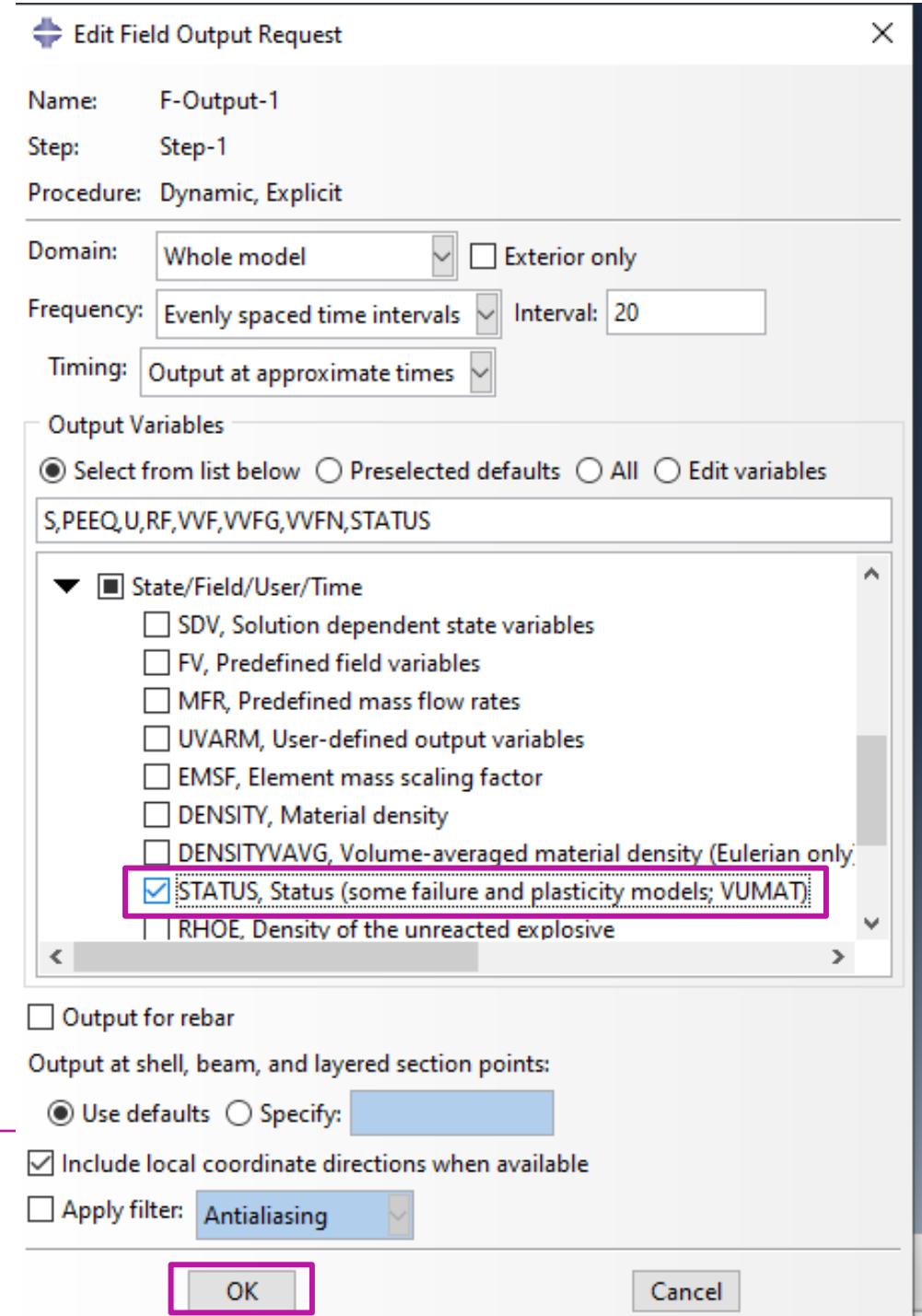
Step 2: Edit Field Output

Model Database

- + Root reference
- + Sets (4)
 - Surfaces
 - Connector Assignments
 - Engineering Features
- Steps (2)
 - Initial
 - Step-1
 - Field Output Requests (1)
 - History Output Requests (1)
 - ALE Adaptive Mesh Constraints
 - Interactions
 - Loads
 - BCs (3)
 - Predefined Fields
 - Load Cases
 - Field Output Requests (1)
 - F-Output-1
 - States (1)
 - Step-1 (Created)
 - History Output Requests (1)
 - H-Output-1
 - Time Points
 - ALE Adaptive Mesh Constraints
 - Interactions
 - Interaction Properties
 - Contact Controls
 - Contact Initializations
 - Contact Stabilizations
 - Constraints
 - Connector Sections
 - Fields
 - Amplitudes (1)
 - Amp-1
 - Loads

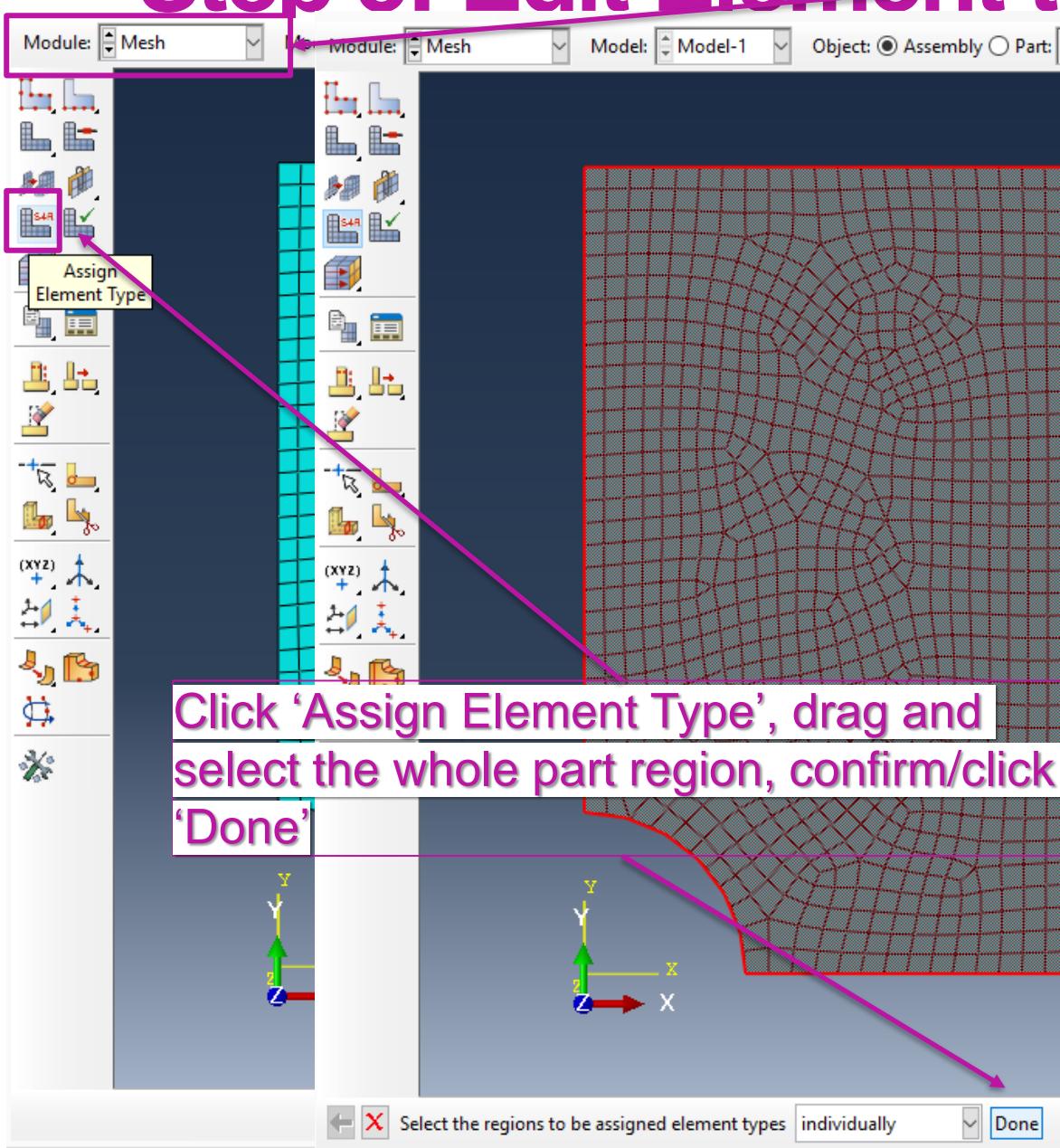


Step 2: Edit Field Output

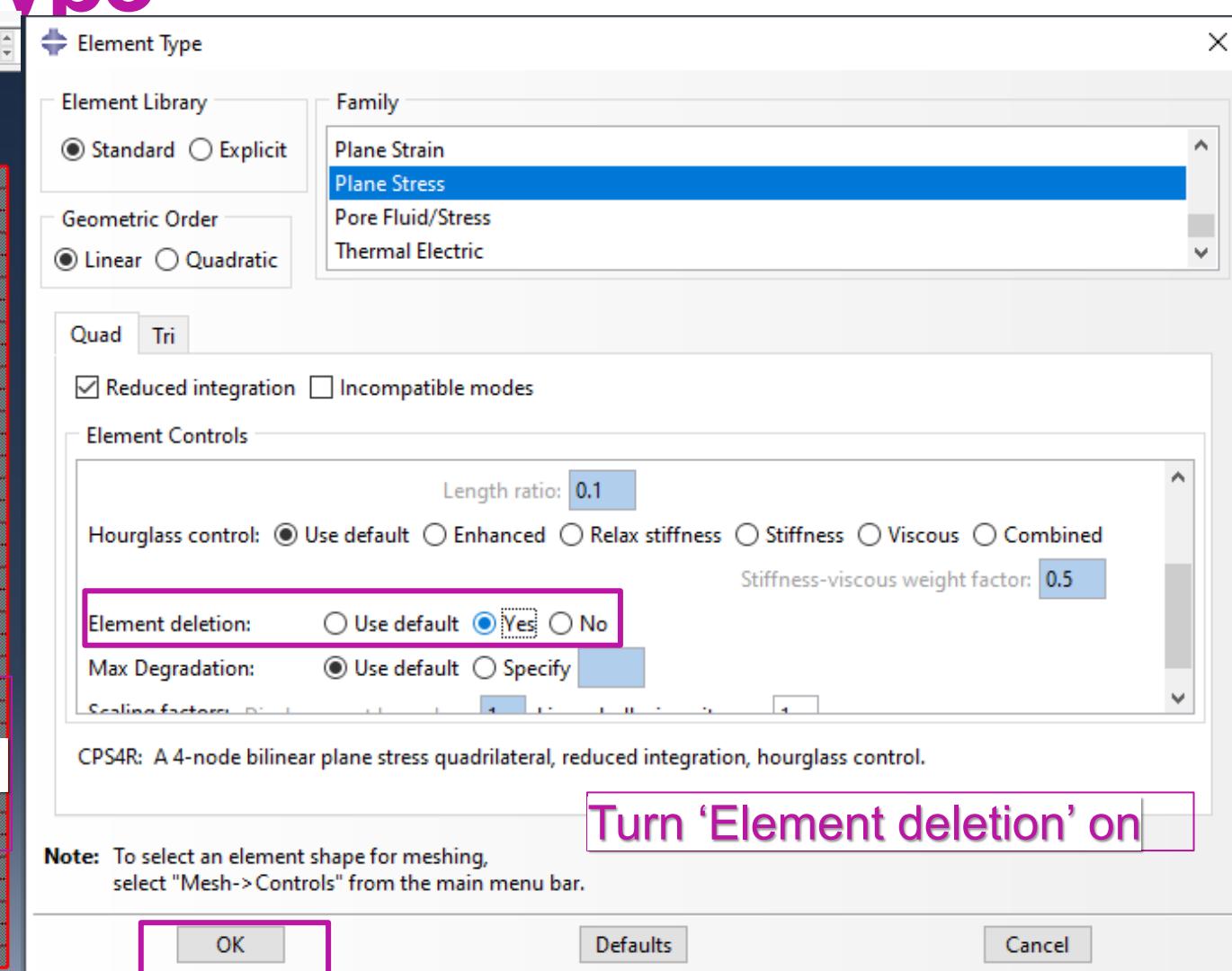


Step 3: Edit Element type

Change to 'Mesh' Module

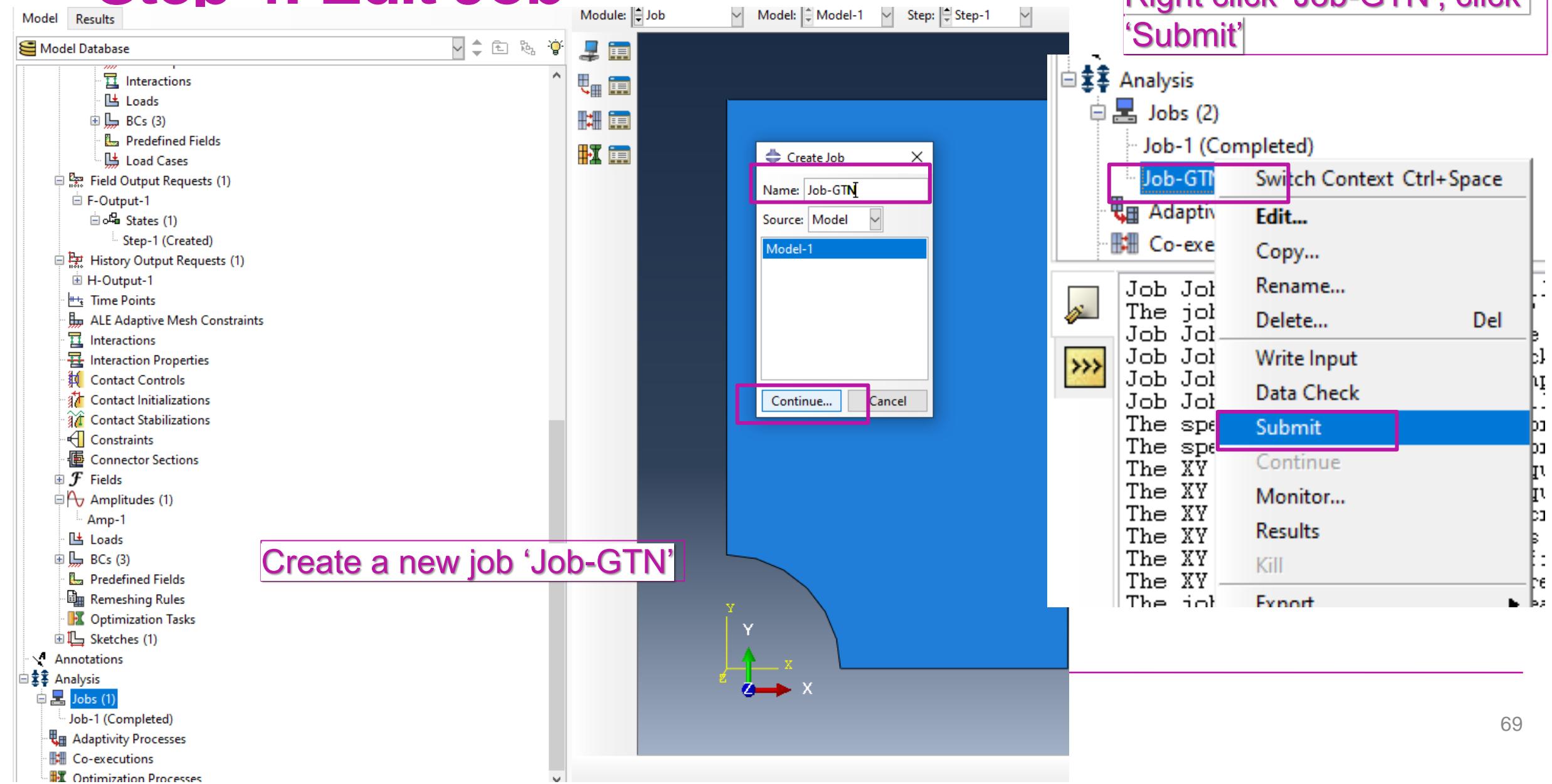


Click 'Assign Element Type', drag and select the whole part region, confirm/click 'Done'

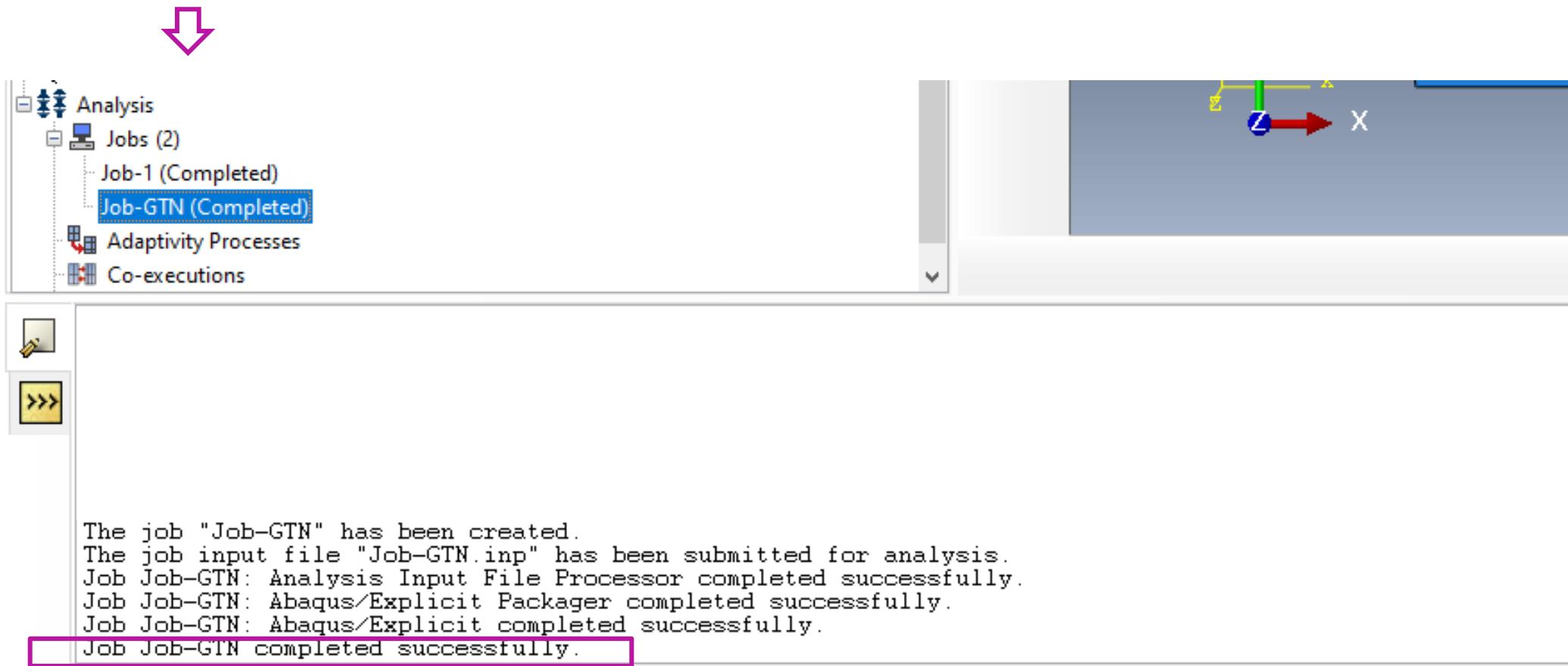


Turn 'Element deletion' on

Step 4: Edit Job



Running...



Step 5: Results

Abaqus/CAE 2017 [Viewport: 1]

File Model Viewport View Job Adaptivity Co-execution Optimizatic

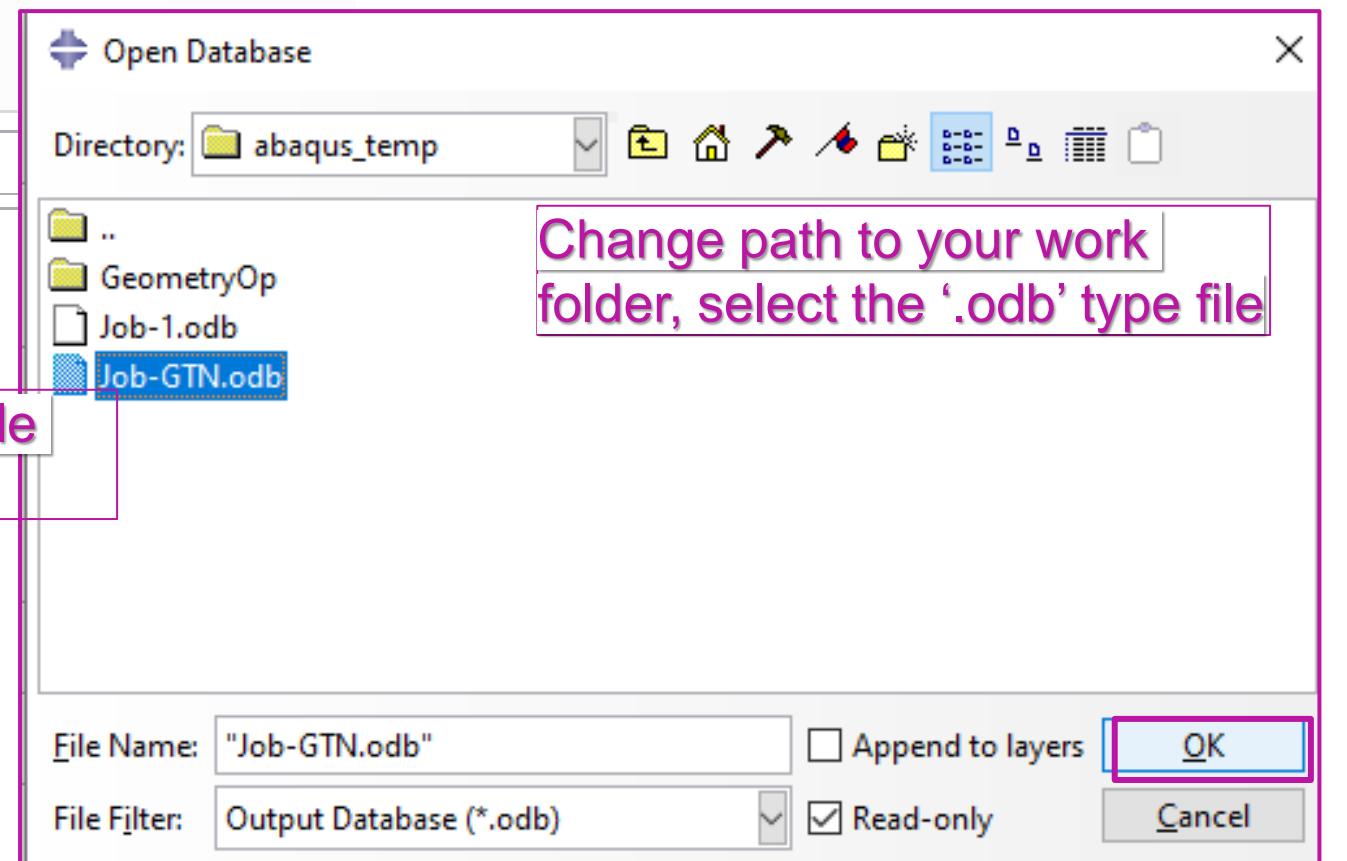


Model Results

Session Data

- + Output Databases (1)
- + Model Database (1)
- + Spectru Output Databases
- + XYPlots (1)
- + XYData (47)
- + Paths
- + Display Groups (1)
- + Free Body Cuts
- + Streams
- + Movies
- + Images

Change to 'Results', double click 'Output Databases'



Step 5: Results

e.g. 5.2 Show whole specimen

Abaqus/CAE 2017 [Viewport: 1]

File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help ?

Save... Pan F2 Rotate F3 Zoom In/Out F4 Box Zoom F5 Auto-Fit F6 Cycle Views F7 Parallel Perspective Show Model Tree Ctrl+T Full Screen F11 Toolbars View Options... Graphics Options... Light Options... Image/Movie Options... ODB Display Options... Overlay Plot...

Module: Visualization Model: C:/programdata/ab

Session Data

- Output Databases (2)
 - Job-1.odb
 - Job-GTN.odb
 - History Output (1)
 - Steps (1)
 - Instances (1)
 - Materials (1)
 - Sections (2)
 - Element Sets (4)
 - Node Sets (4)
 - Surface Sets
 - Session Coordinate
 - ODB Coordinate S
- User Data
 - Annotations
 - XYData
- Model Database (1)
- Spectrums (7)
- XYPlots (1)
- XYData (50)
- Paths
- Display Groups (1)
- Free Body Cuts
- Streams
- Movies
- Images

Choose 'View' - 'ODB Display Options'

Choose 'Mirror/...', Open 'XZ/YZ' options

ODB Display Options

General Entity Display Constraints Sweep/Extrude Mirror/Pattern

Mirror

Mirror CSYS: (Global)

Mirror planes: XY XZ YZ Mirror display bodies

Pattern

Pattern CSYS: (Global)

Rectangular

Number	Offset
X: 1	0
Y: 1	0
Z: 1	0

Circular

Axis of rotation: Z

Number: 1

Total angle: 360

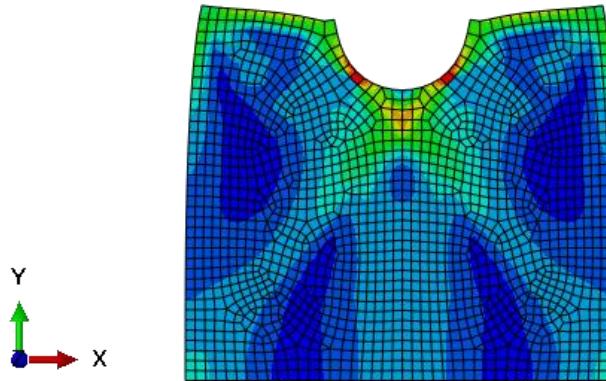
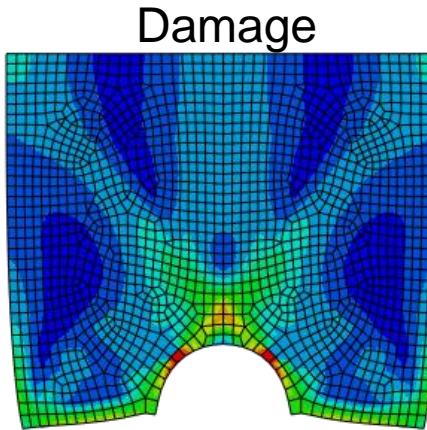
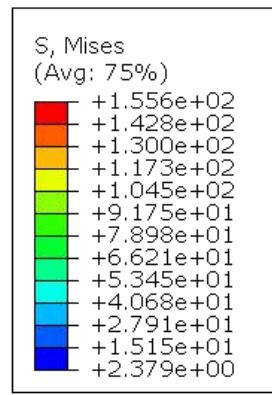
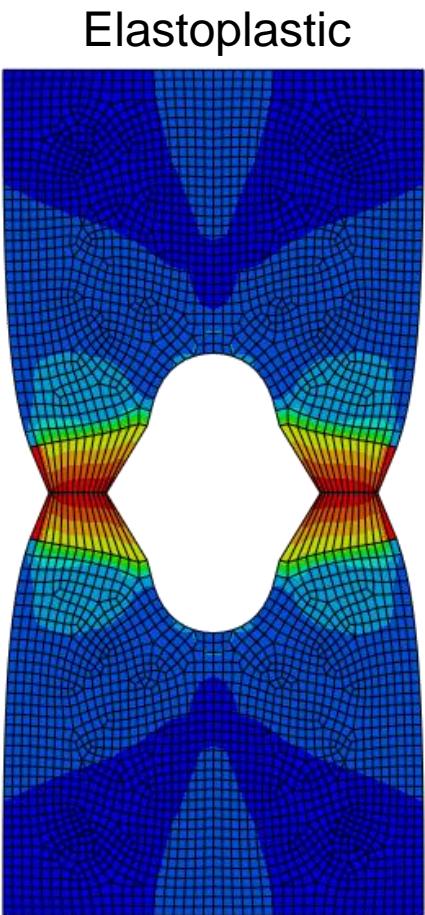
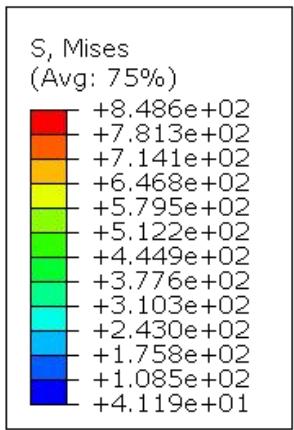
Order of Operations

Mirror, rectangular pattern, circular pattern

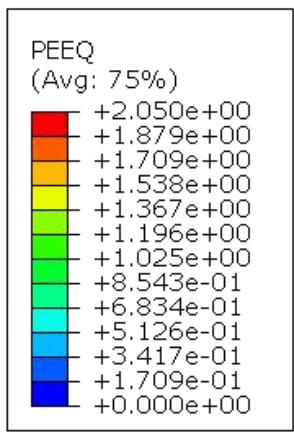
Note: Sweep and extrude operations, if any, will always be performed first.

OK Apply Defaults Cancel

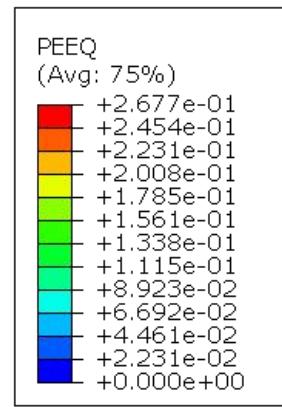
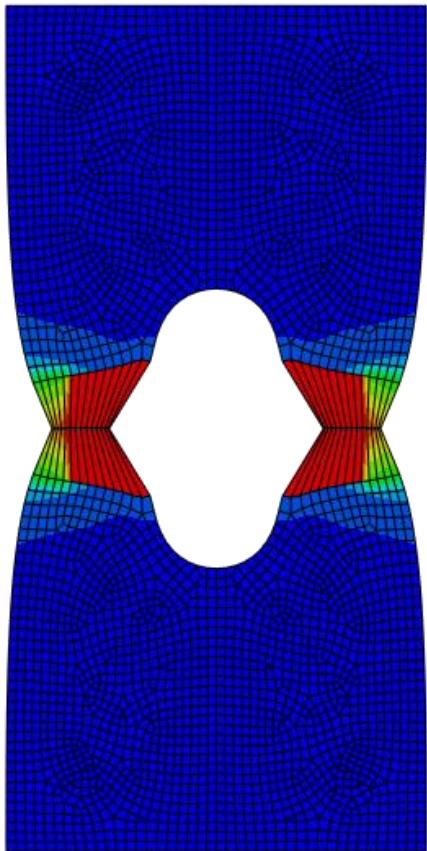
Step 5: Results



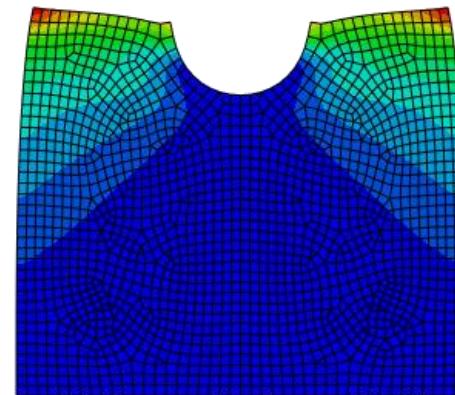
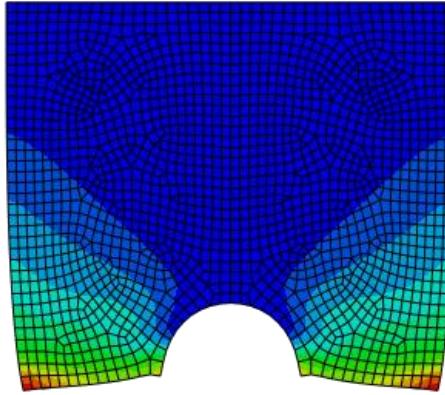
Step 5: Results



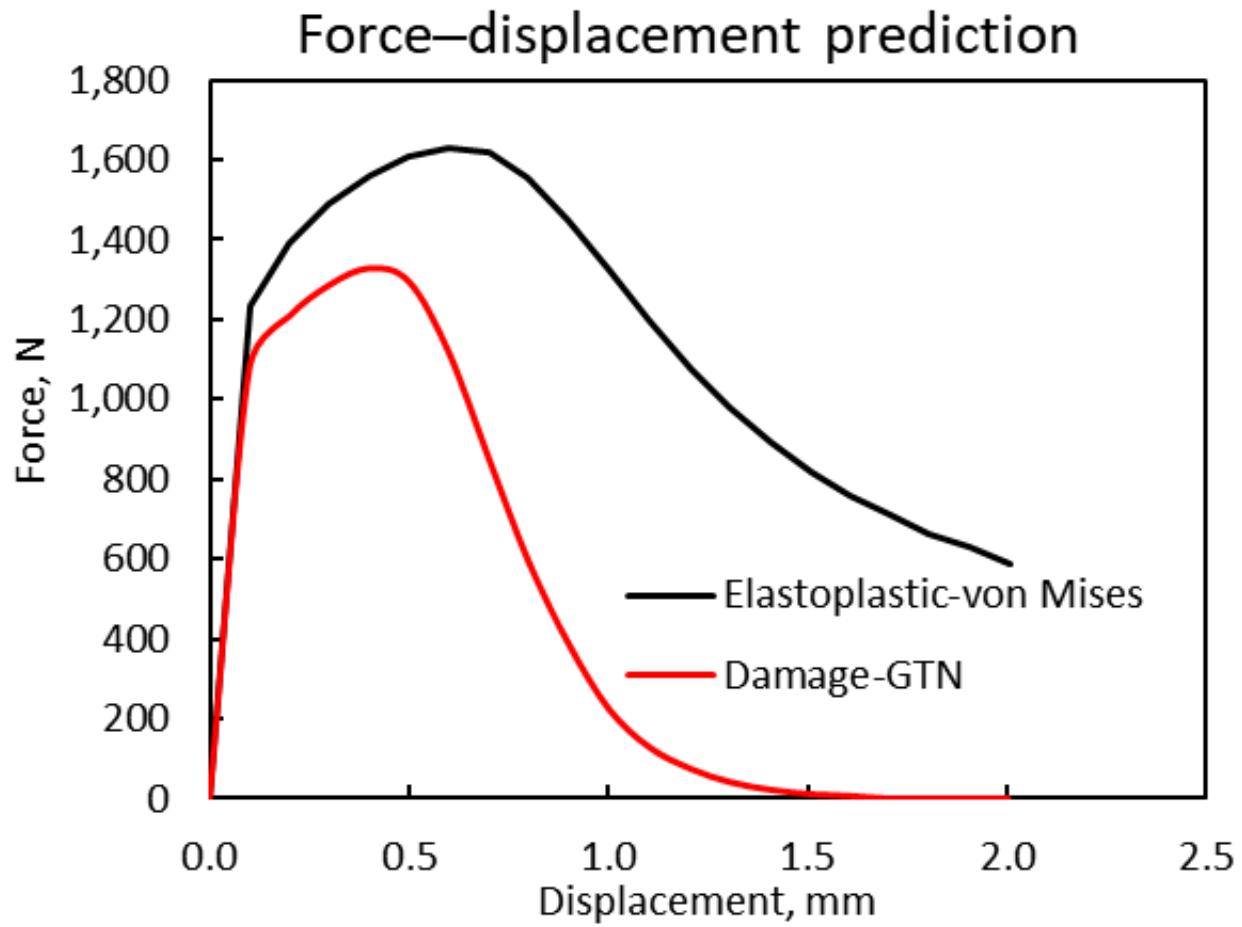
Elastoplastic



Damage



Step 8: Results



Abaqus Work directory/files

Step 0: Set Abaqus work directory

1. Abaqus/CAE 2017 [Viewport: 1]

File Model Viewport View Part Shape Feature Tools Plug-ins Help ?

New Model Database Open... Network ODB Connector Close ODB Ctrl+O

2. Set Work Directory... Save Save As... Compress MDB... Save Display Options... Save Session Objects... Load Session Objects... Import Export Run Script... Macro Manager... Print... Ctrl+P Abaqus PDE... 1 Z:/.../MSEcourse2020/MSEE3.cae 2 C:/.../abaqus_temp/Job-GTN.odb 3 C:/.../abaqus_temp/Job-1.odb 4 C:/Users/.../31 Chao/8cubeRVE.cae Exit Ctrl+Q

Annotations Analysis Jobs Adaptivity Processes Co-executions Optimization Processes

Choose 'File' - 'Set Work Directory'

Set Work Directory

Current work directory: C:\programdata\abaqus_temp

New work directory: C:\programdata\abaqus_temp

Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.

OK Cancel

The default Work Directory, you can check your existed data file here.

3. You can also change it to your preferred work folder

Example:

Select a Work Directory

Directory: MSEcourse

MSE

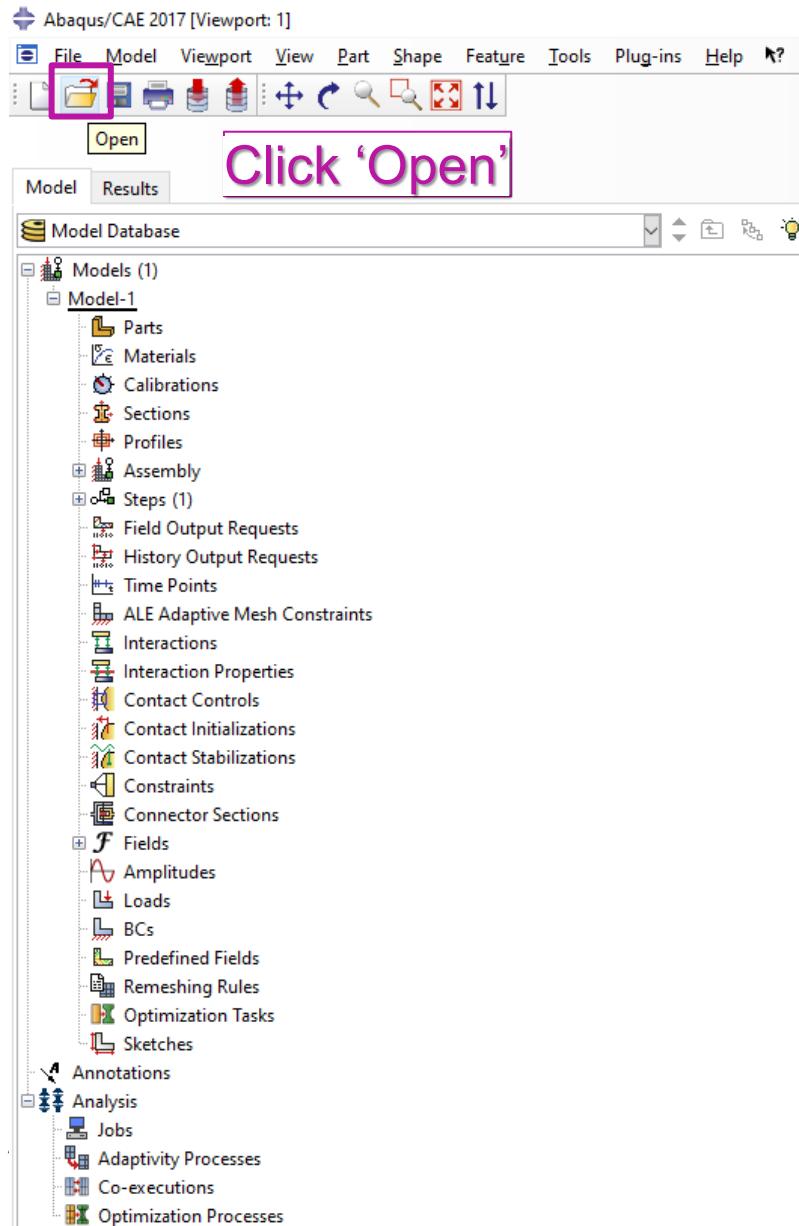
4. OK

5. OK Cancel

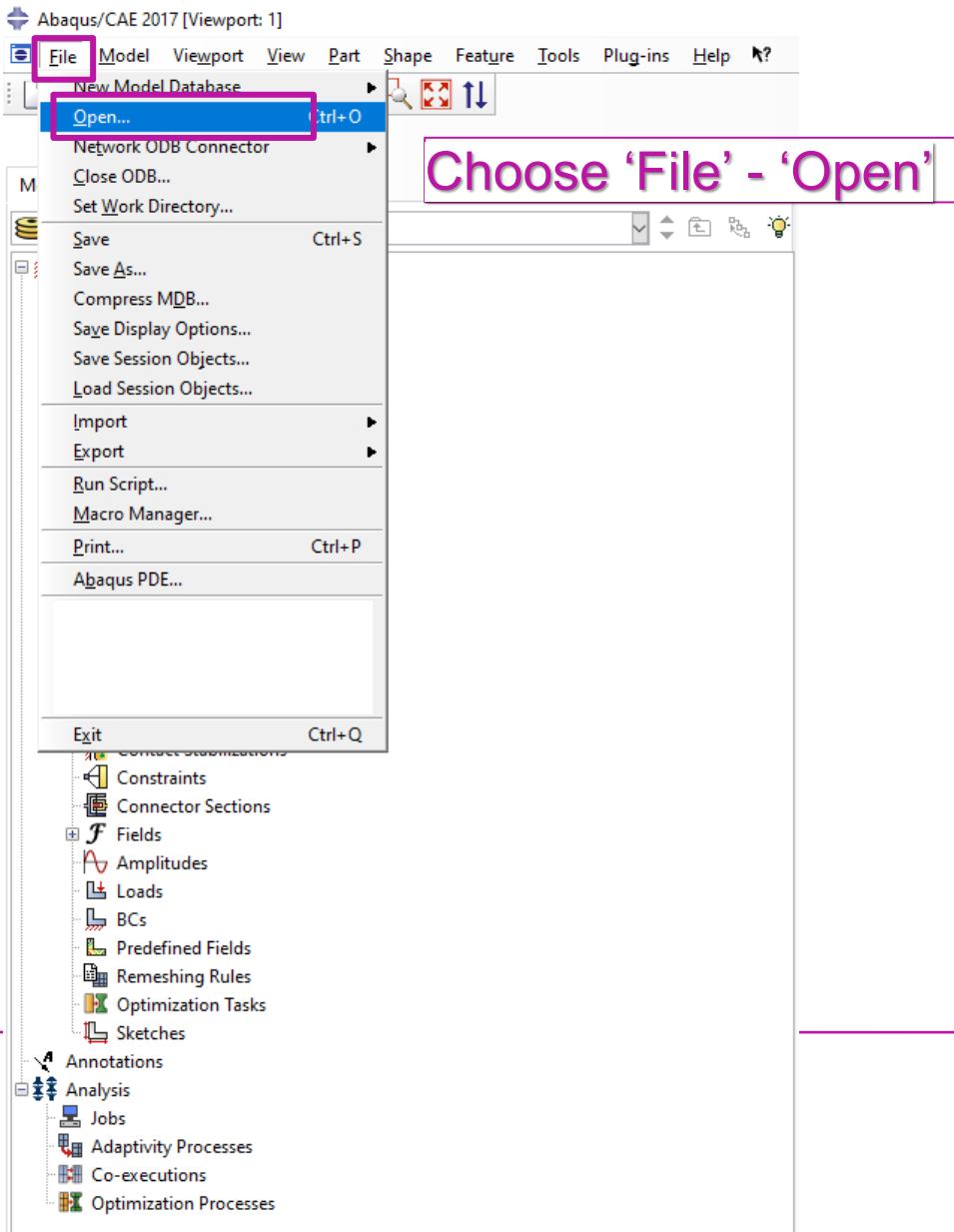
/MSEcourse

Note: In file selection dialog boxes, you can click the work directory icon to jump to the current work directory.

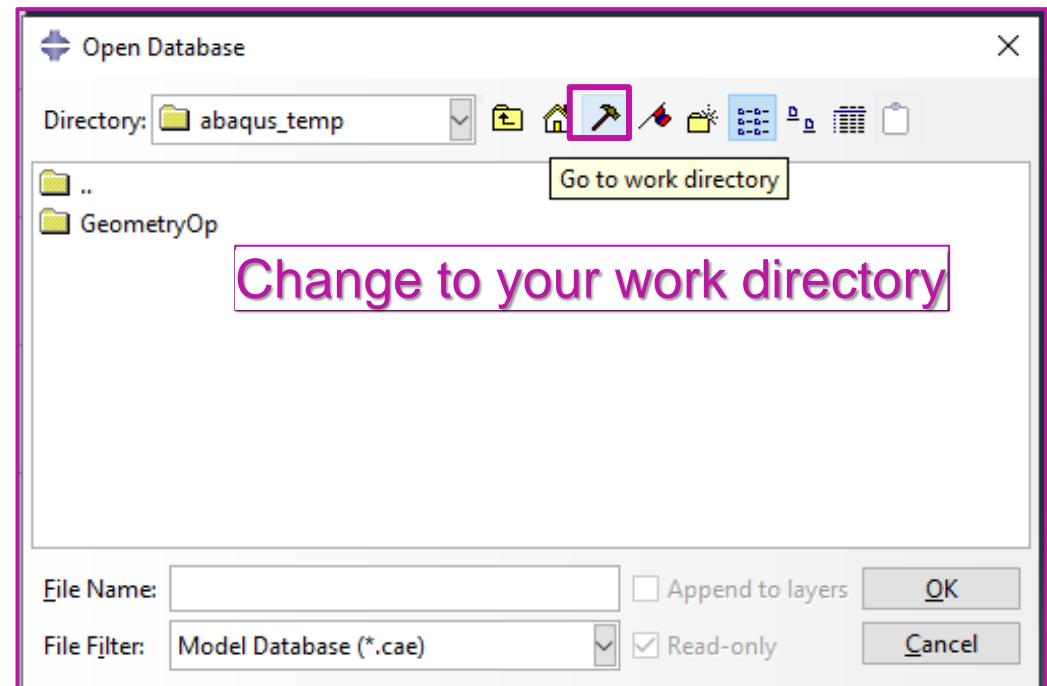
Open files



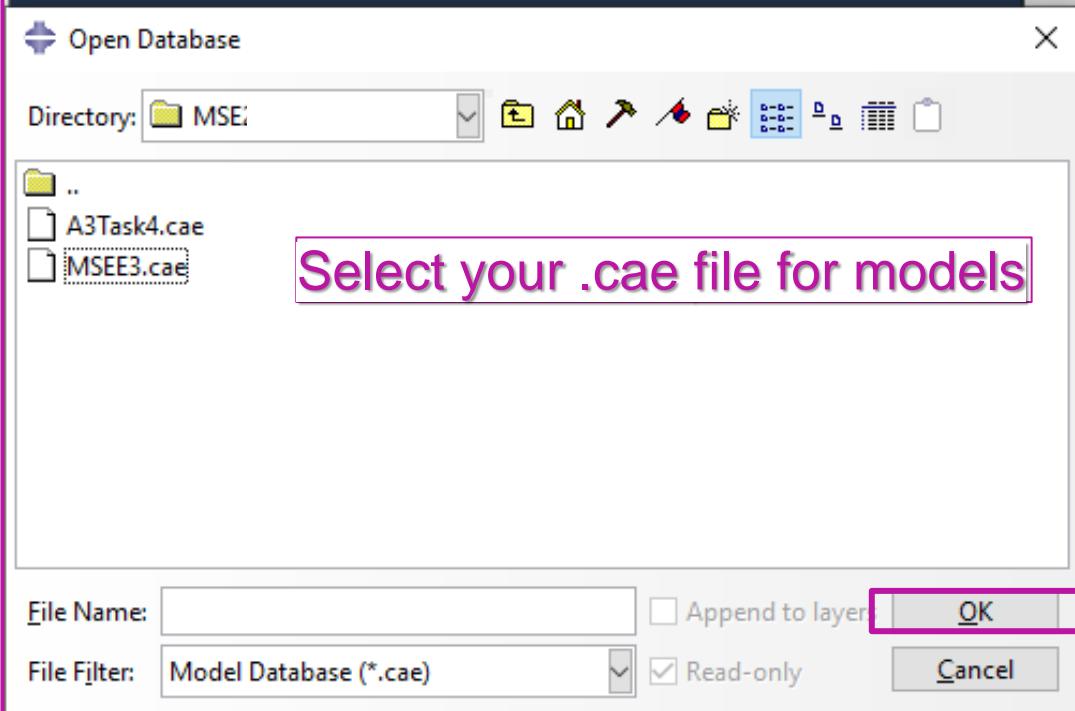
Or



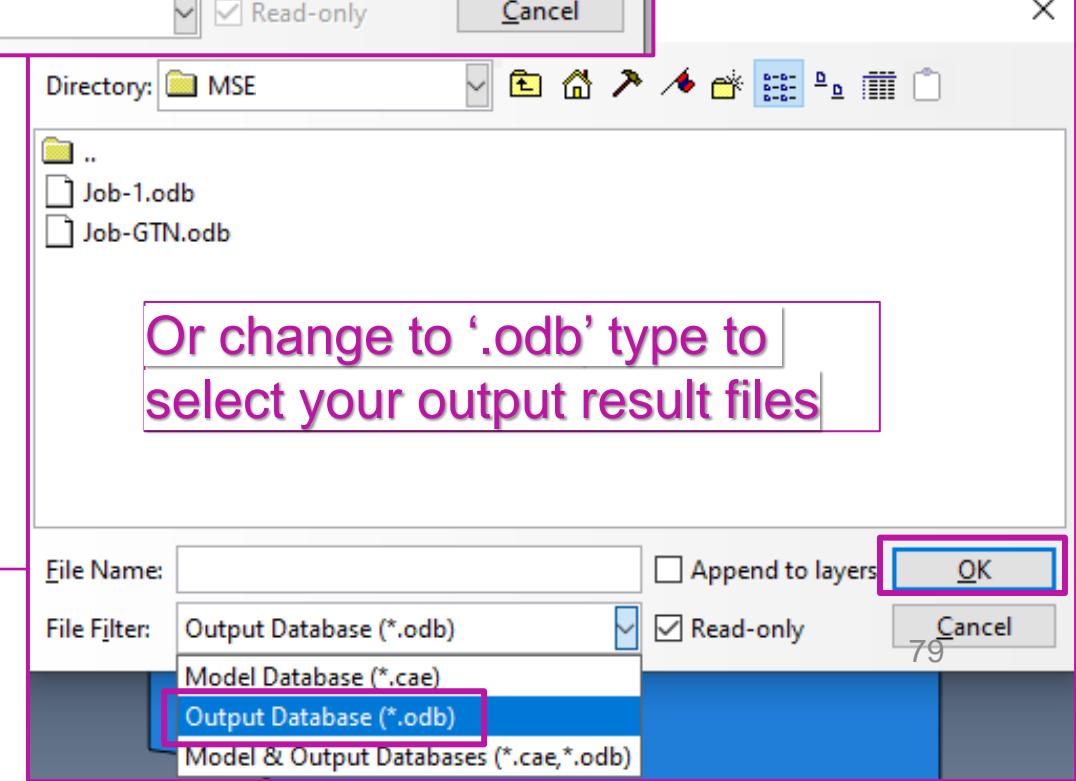
Open files



Change to your work directory



Select your .cae file for models



Or change to '.odb' type to
select your output result files

Feedback - Assignment

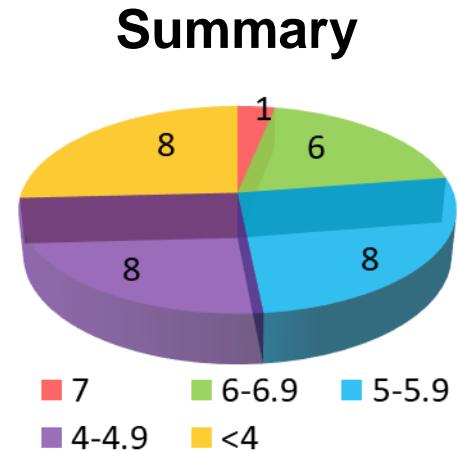
Assignment 02 - Summary

Submission: 31 in total.

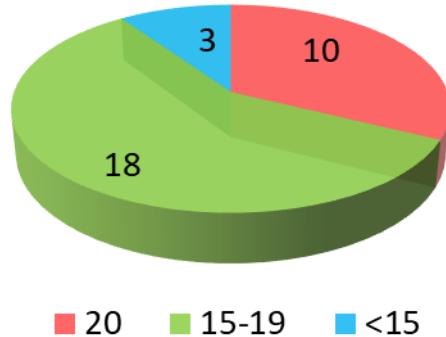
Full points: 100, is calculated as 7 points in the final grade system.

Due date: 18:00, 14.11.2021 (5 delayed submission)

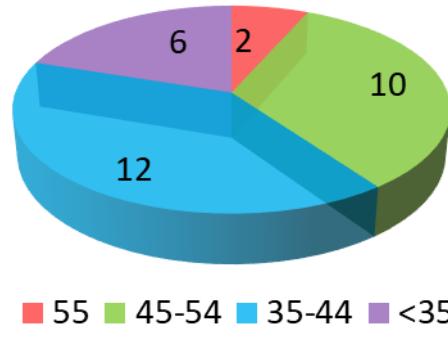
Please pay attention to all comments to you!



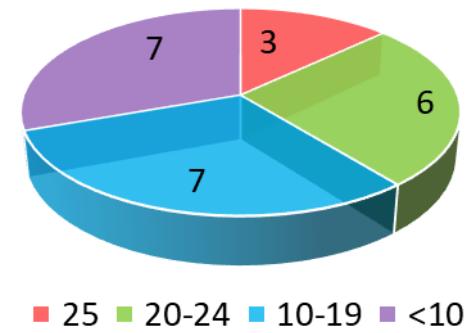
Task 1: 31, 20 points



Task 2: 30, 55 points

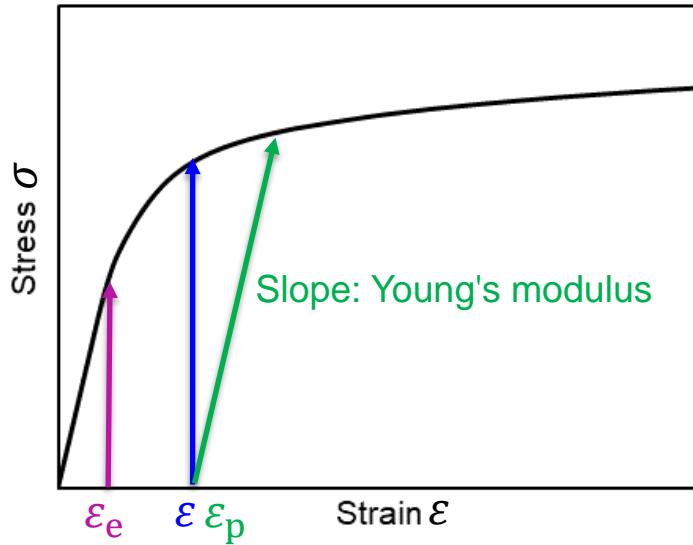


Task 3: 21, 25 points



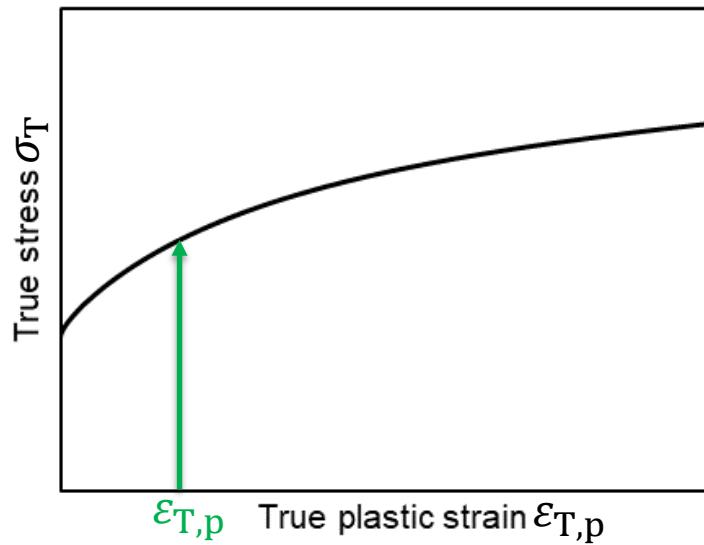
Stress-strain curves

Engineering stress–Engineering strain curve
True stress–True strain curve

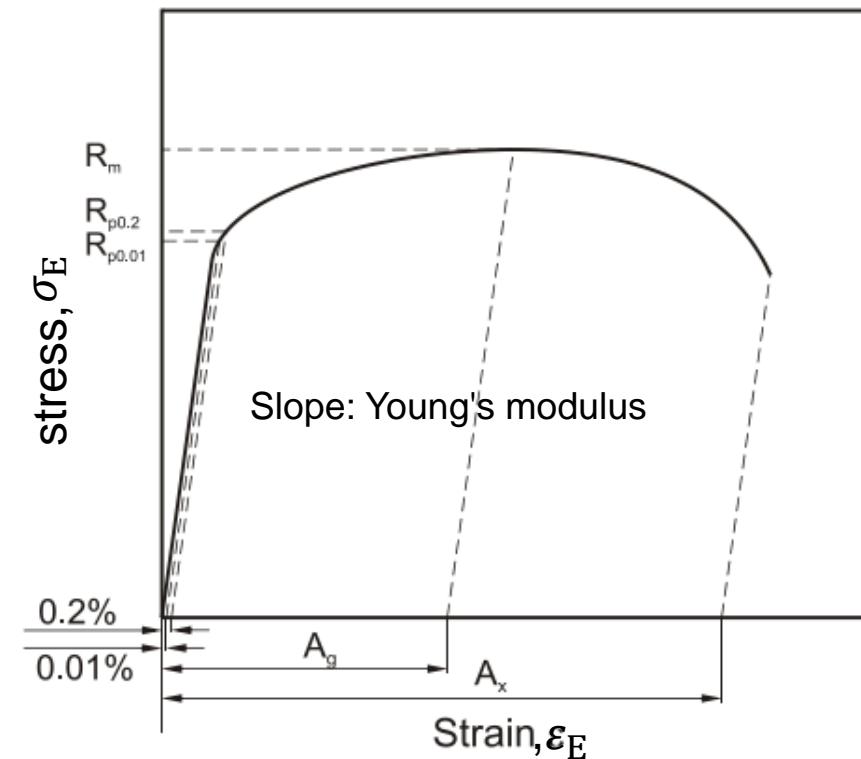


ϵ_E : Engineering strain
 ϵ_T : True strain
 ϵ_e : Elastic strain
 ϵ_p : Plastic strain

Flow curve:
True stress–True plastic strain curve



Engineering stress–Engineering strain curve



* Wolfgang Bleck, Materials Science of Steel, Aachen, 2016

Basic rules for Miller/Miller-Bravais indices

Miller indices - Cube

Miller-Bravais indices - HCP

Crystal plane: (hkl) $(hkil)$

Crystal plane family: $\{hkl\}$ $\{hkil\}$

Crystal direction: $[uvw]$ $[uvtw]$

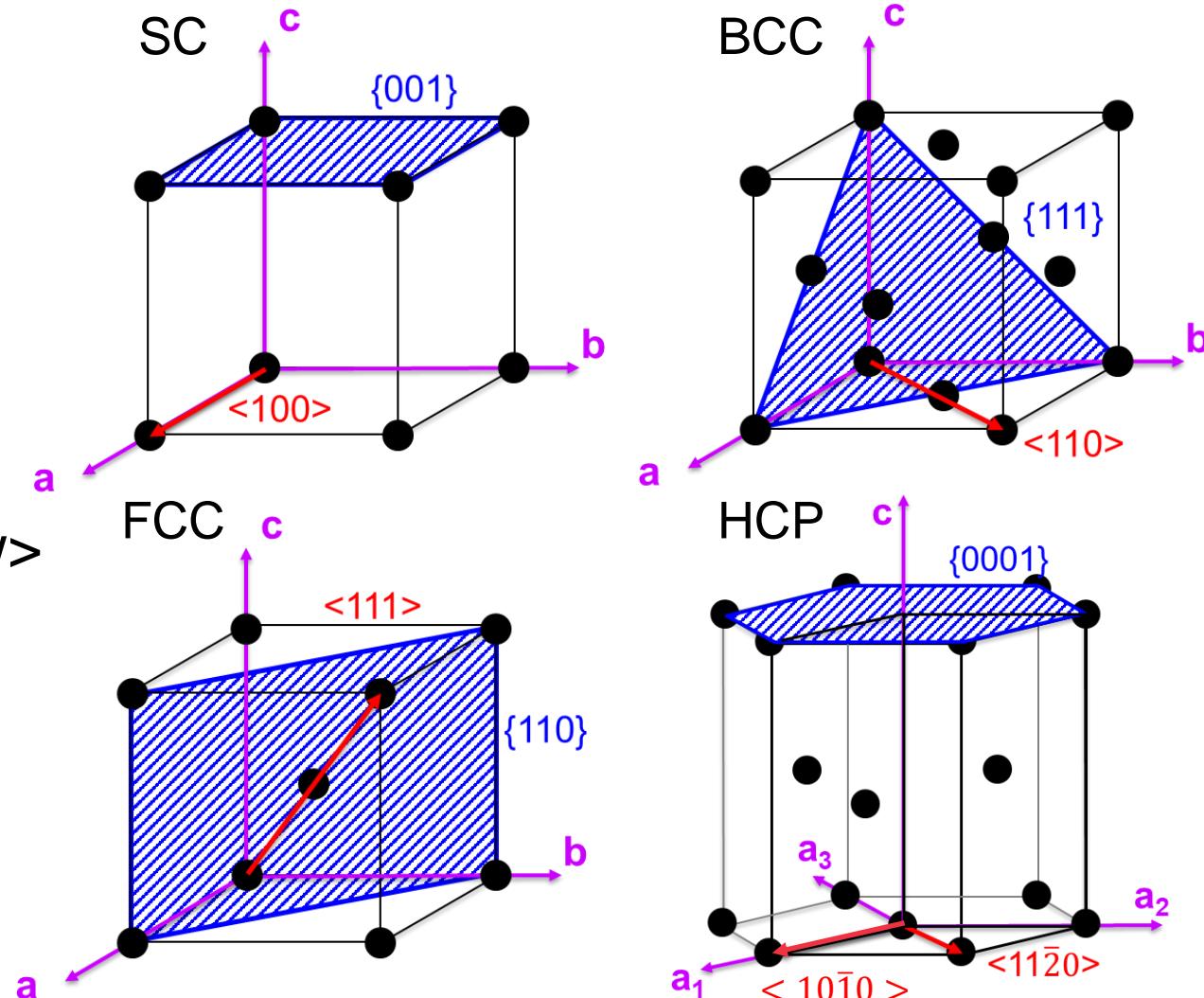
Crystal direction family: $\langle uvw \rangle$ $\langle uvtw \rangle$

minus value: $\bar{1}$ instead of -1

Note:

e.g. for HCP most closed packet planes:

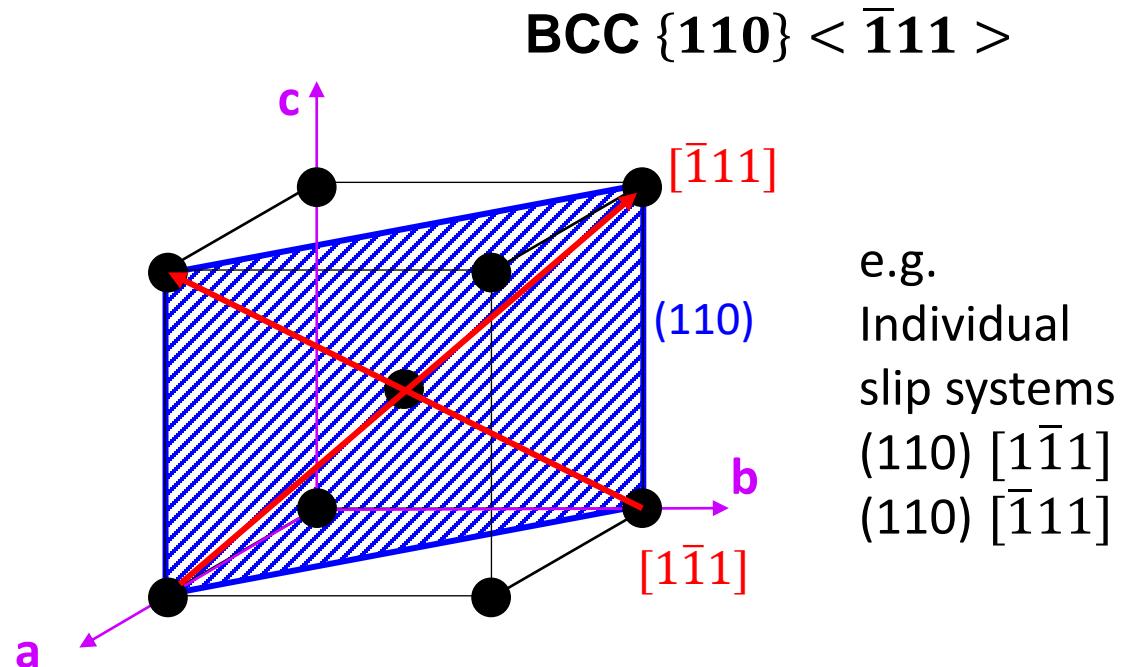
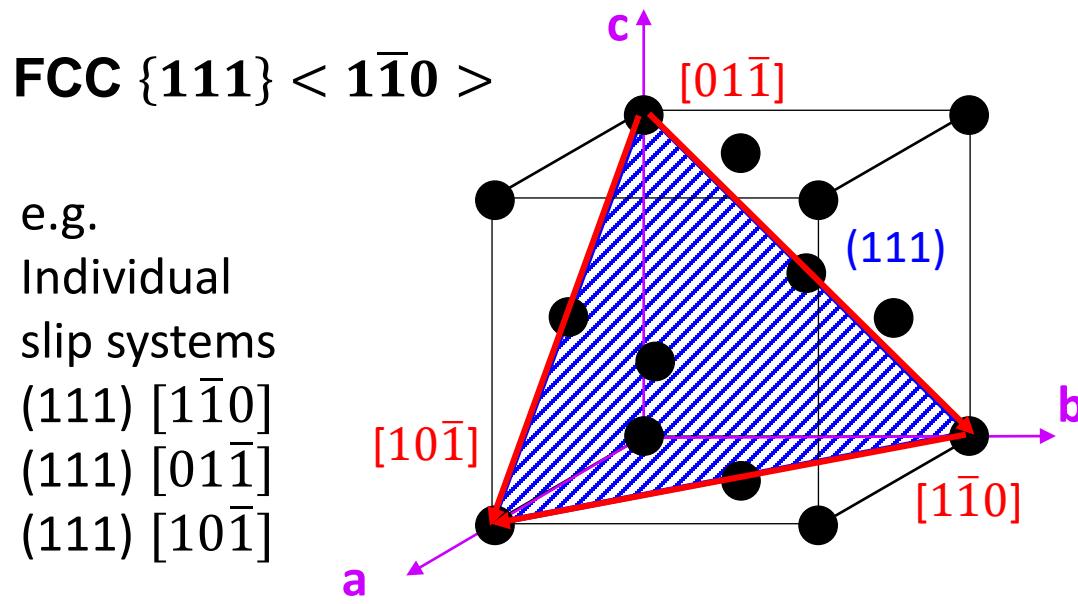
$(001) = (00\bar{1})$ $\langle 11\bar{2}0 \rangle$ ✓
 $[001] = [00\bar{1}]$ $\langle 11-20 \rangle$ ✗
 $\langle 110 \rangle$ ✗



-2 points if the wrong symbol was used in A2!

Slip systems for FCC & BCC

- Slip systems family $\{hkl\} <uvw>$
- Individual slip systems $(hkl)[uvw]$
- uvw direction shall locate on hkl plane



For individual slip systems, see A2 solution.

Schmid's law

Task 2.2 (c-d) The axis of a tensile specimen of an **fcc** Fe single crystal is parallel to the $\bar{[1}25]$ -direction, based on the possible slip systems of fcc structure, calculate the Schmid factor for every slip system. Among all these 12 slip systems, which one will be activated first? Why? If the critical resolved shear stress is 2.7 MPa, for plastic deformation, which tensile stress has to be applied (in MPa)?

The $(111)[10\bar{1}]$ slip system will be active first, as it has the **maximum absolute** Schmid factor.

$$\sigma = \frac{\tau}{|\cos \lambda \cdot \cos \kappa|} = \frac{2.7 \text{ MPa}}{0.4899} = 5.5114 \text{ MPa}$$

FCC	Schmid factor for 2.2(c)
$(111)[1\bar{1}0]$	-0.2449
$(111)[10\bar{1}]$	-0.4899
$(111)[01\bar{1}]$	-0.2449
$(\bar{1}11)[110]$	0.1089
$(\bar{1}11)[101]$	0.4355
$(\bar{1}11)[01\bar{1}]$	-0.3266
$(1\bar{1}1)[110]$	0.0272
$(1\bar{1}1)[10\bar{1}]$	-0.1633
$(1\bar{1}1)[011]$	0.1905
$(11\bar{1})[1\bar{1}0]$	0.1633
$(11\bar{1})[101]$	-0.2177
$(11\bar{1})[011]$	-0.3810

Assignment - General rules

Q&A

- Please avoid emails and use the “General discussion” on MyCourses!
- Please check the existing topics in “General discussion” before proposing a new one.
- Reply: Mondays, Tuesdays, Thursdays.
- Face-to-face Q&A time: every Tuesday at 16:30-18:00 (via Zoom: <https://aalto.zoom.us/j/62428835336>).

Timelines

- Open on MyCourses: every Monday before 18:00.
- Deadline: every Sunday at 18:00 to MyCourses.
- Cut-off deadline: every Tuesday at 16:00 to MyCourses.
- Solution open on MyCourses: every Tuesday at 16:30.
- For the last assignment (A6), no extension and later submission allowed, solution will be given on MyCourses before 18:30, 12.12.2021.

Assignment - General rules

Submission rules

- Only PDF type file is accepted for submission, please summarize all your answers/solutions in **one PDF file for every assignment**.
- Please name your assignment files with the assignment number and your first name and surname, and link them with short underlines: '**ANr_Firstname_Surname.pdf**', e.g. for the first Assignment 'A1_Wenqi_Liu.pdf'.
- It is appreciated to **sort the PDF pages in the TaskNr order**, which is helpful to speed up the evaluation process.
- Learning Group work is encouraged for this course. You could form a group with **max one additional peer** to review the lecture/exercise content and discuss the tasks in the assignment. After discussion, please **finish your assignment independently** and submit your individual report. Please note the **duplicate report is not accepted!** If you have a learning group, please **indicate who your group member is** in the submitted report. In addition, clearly state the individual contributions of each group member.

Assignment - General rules

- When required, always show the step-by-step derivation or calculation processes, without which hinting the number does not qualify for grades.
- When required, always give a brief and concise explanation or description, without which hinting the right choice or answer does not qualify for grades.
- **Citation is necessary** if you are using any figures/data that are not generated by yourself.
- Handwriting/plotting is acceptable, just **make sure that your handwriting/final photo in the system is clear enough, otherwise it may affect the grading for details/calculation process.**

Grades

- Delayed submission will be subjected to a penalty function in an exponential relation with time.
- Full points: 100 for every assignment, which will be calculated as 5-7 points in the final grade system. The weighted pointes will be indicated in each assignment. In total, 40 points for 6 assignments.
- Tolerant grading, **high points ≠ exactly accurate answers!**
- **Check the assignment solution carefully!** This is the only standard answer (for calculations, equations, derivations, definitions) if there are similar questions in the exam.

Resources

Materials:

- Lian J., Sharaf M., Archie F., Münstermann S., A hybrid approach for modelling of plasticity and failure behaviour of advanced high-strength steel sheets. International Journal of Damage Mechanics, 2013. 22(2): 188218. <https://doi.org/10.1177/1056789512439319>
- Wenqi Liu, Junhe Lian, Sebastian Münstermann, Damage mechanism analysis of a high-strength dual-phase steel sheet with optimized fracture samples for various stress states and loading rates, Engineering Failure Analysis, 2019, 106: 104138. <https://doi.org/10.1016/j.engfailanal.2019.08.004>
- Wenqi Liu, Junhe Lian, Sebastian Münstermann, Chongyang Zeng, Xiangfan Fang, Prediction of crack formation in the progressive folding of square tubes during dynamic axial crushing, International Journal of Mechanical Sciences, 2020, 176: 105534. <https://doi.org/10.1016/j.ijmecsci.2020.105534>
- Wolfgang Bleck, Materials Science of Steel, Aachen, 2016, 4th ed.

Software:

- Abaqus: <https://edu.3ds.com/en/software/abaqus-student-edition>

Questions?

- **Assignment** submission DL is **18:00 on 21.11.2021**.
- Use the Zoom Chat function or raise your hands!
- Please avoid emails and use the “General discussion” on MyCourses!
- Please check the existing topics in “General discussion” before proposing a new one.

Slides will be uploaded after Exercise.