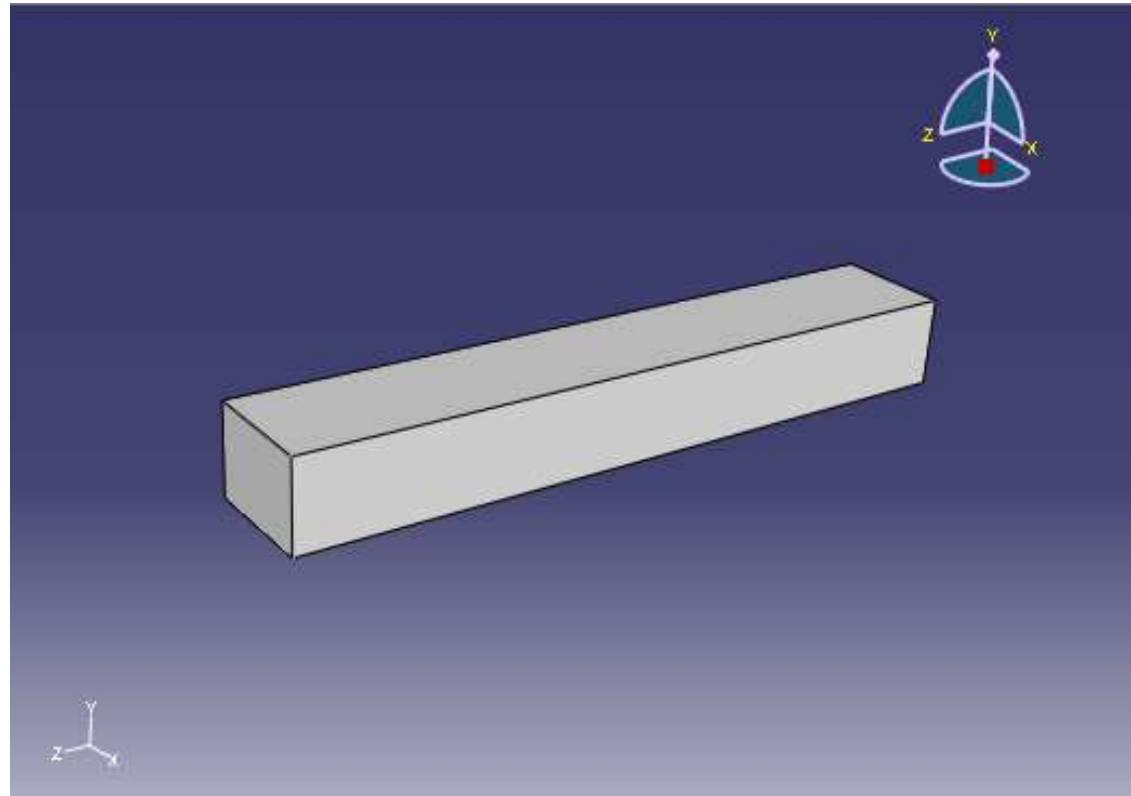




# Exercise 1 - introduction

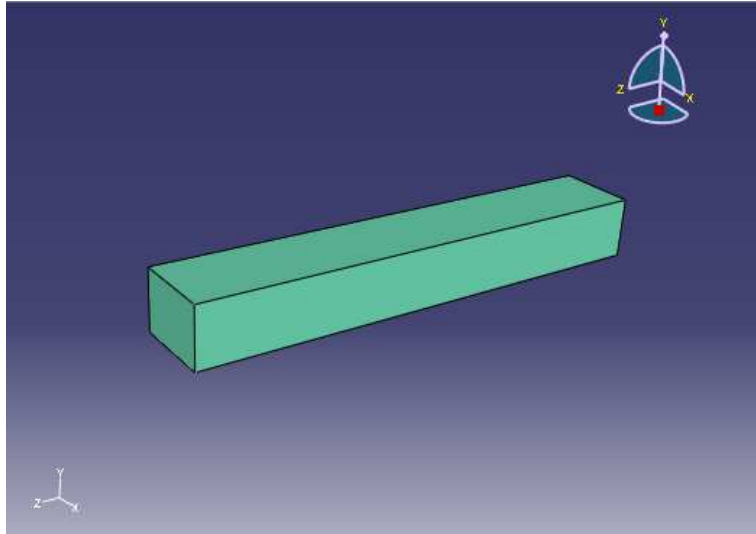
## Elastic problem

Beam

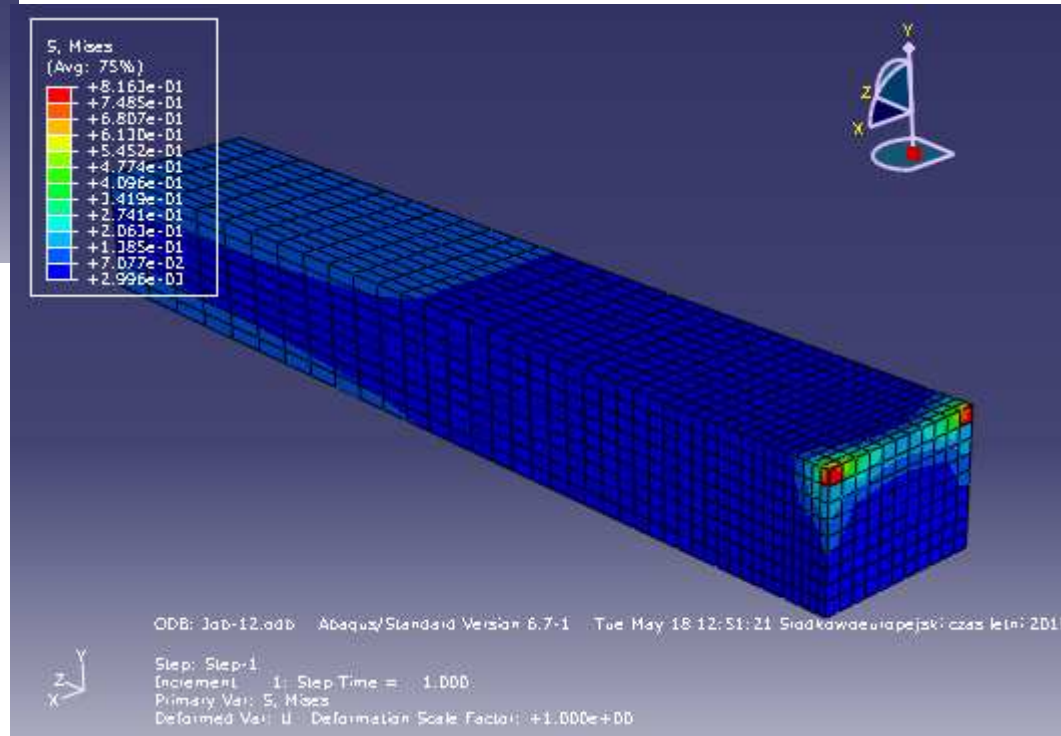




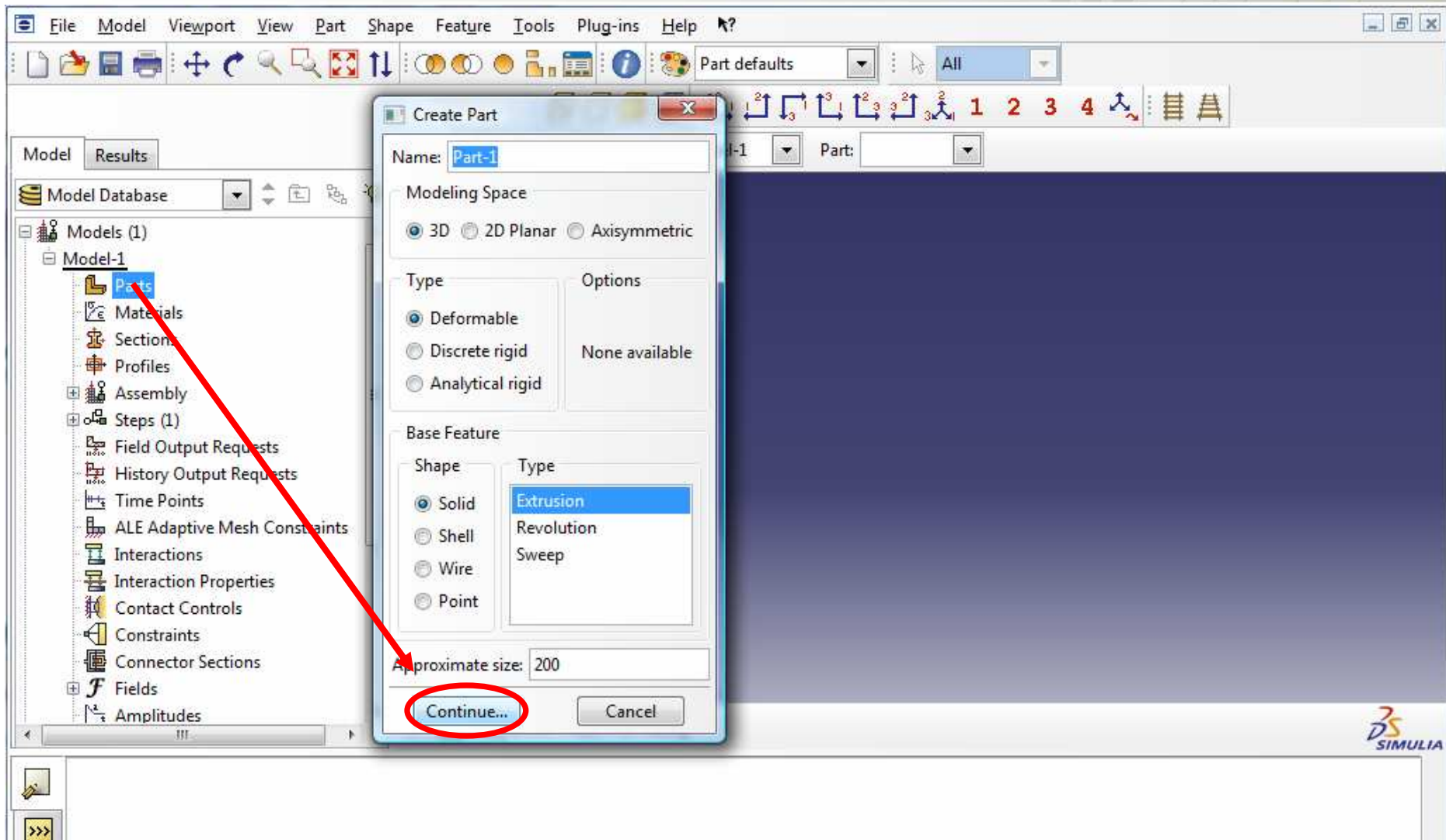
# Exercise 1 – elastic analysis



Flow stress??

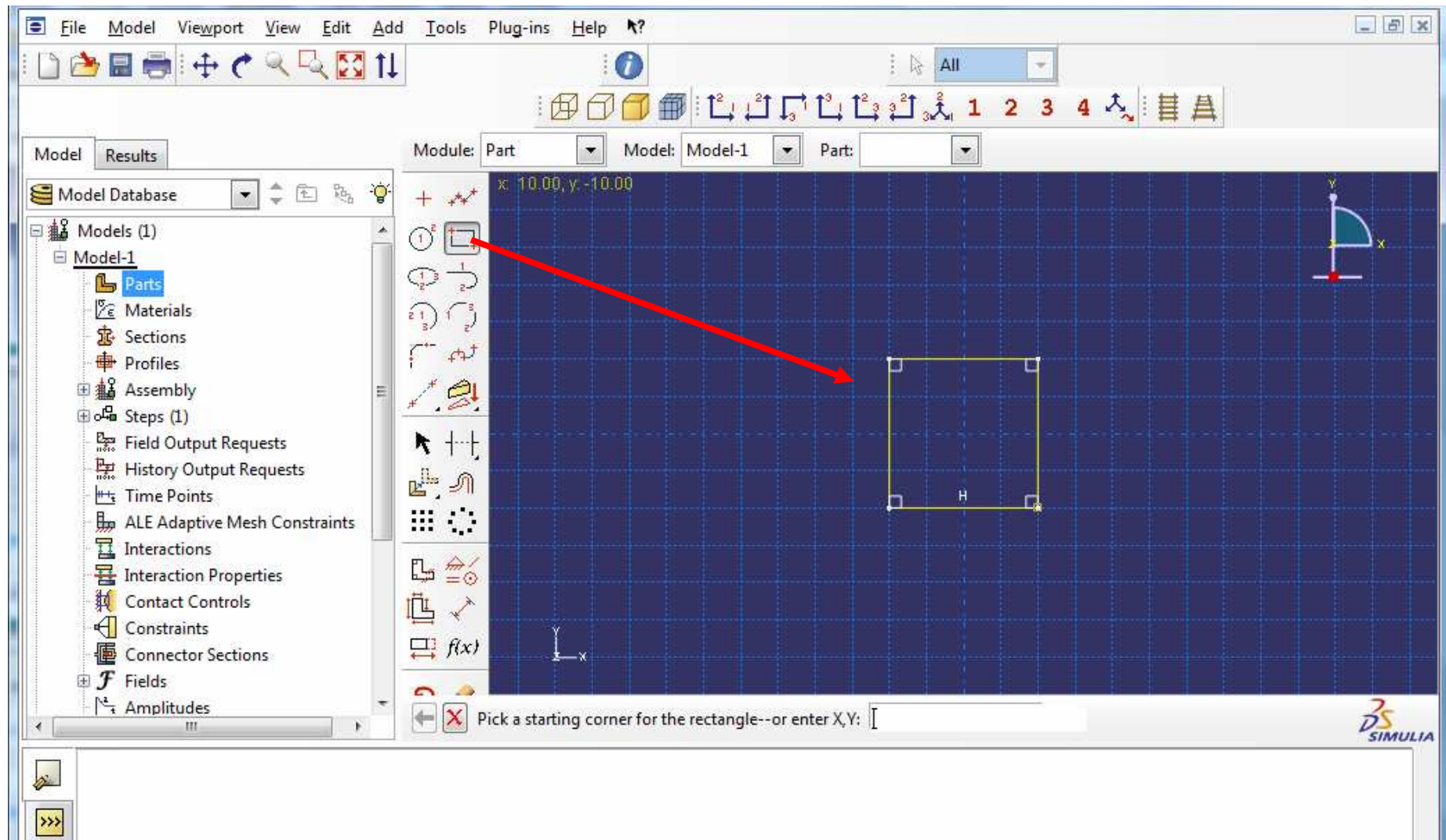


# Exercise 1 – create part



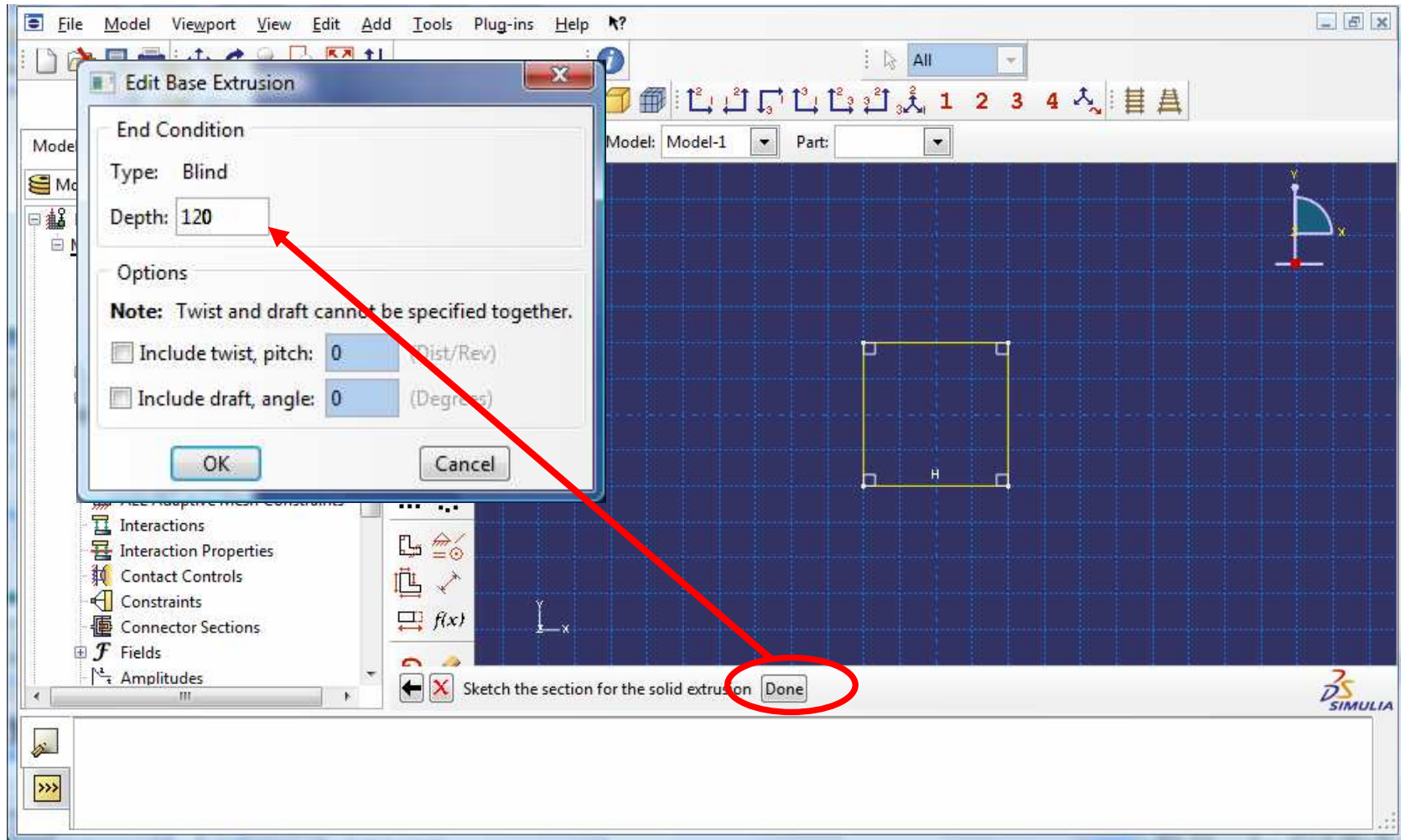


# Exercise 1 – create part



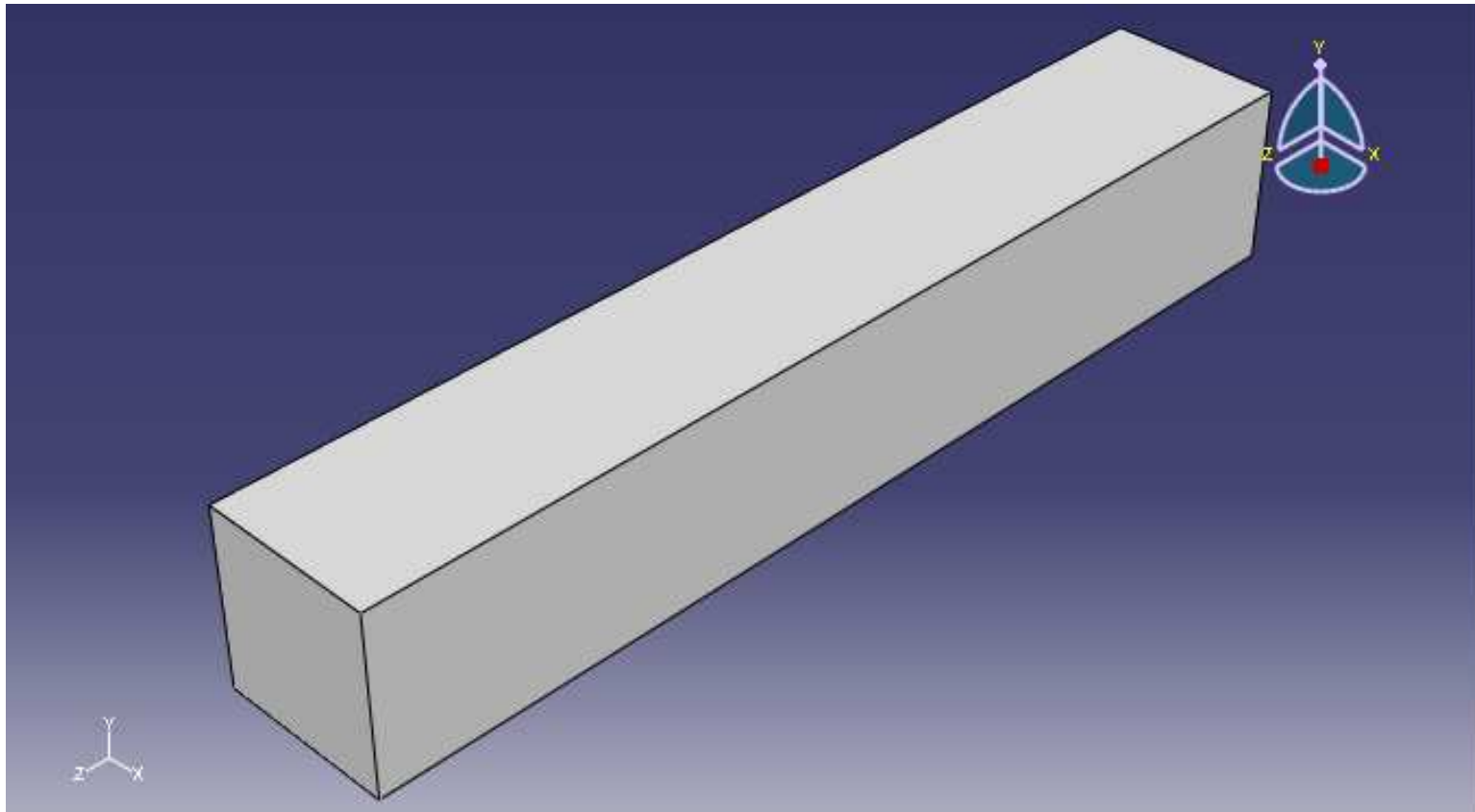


# Exercise 1 – sketch extrude

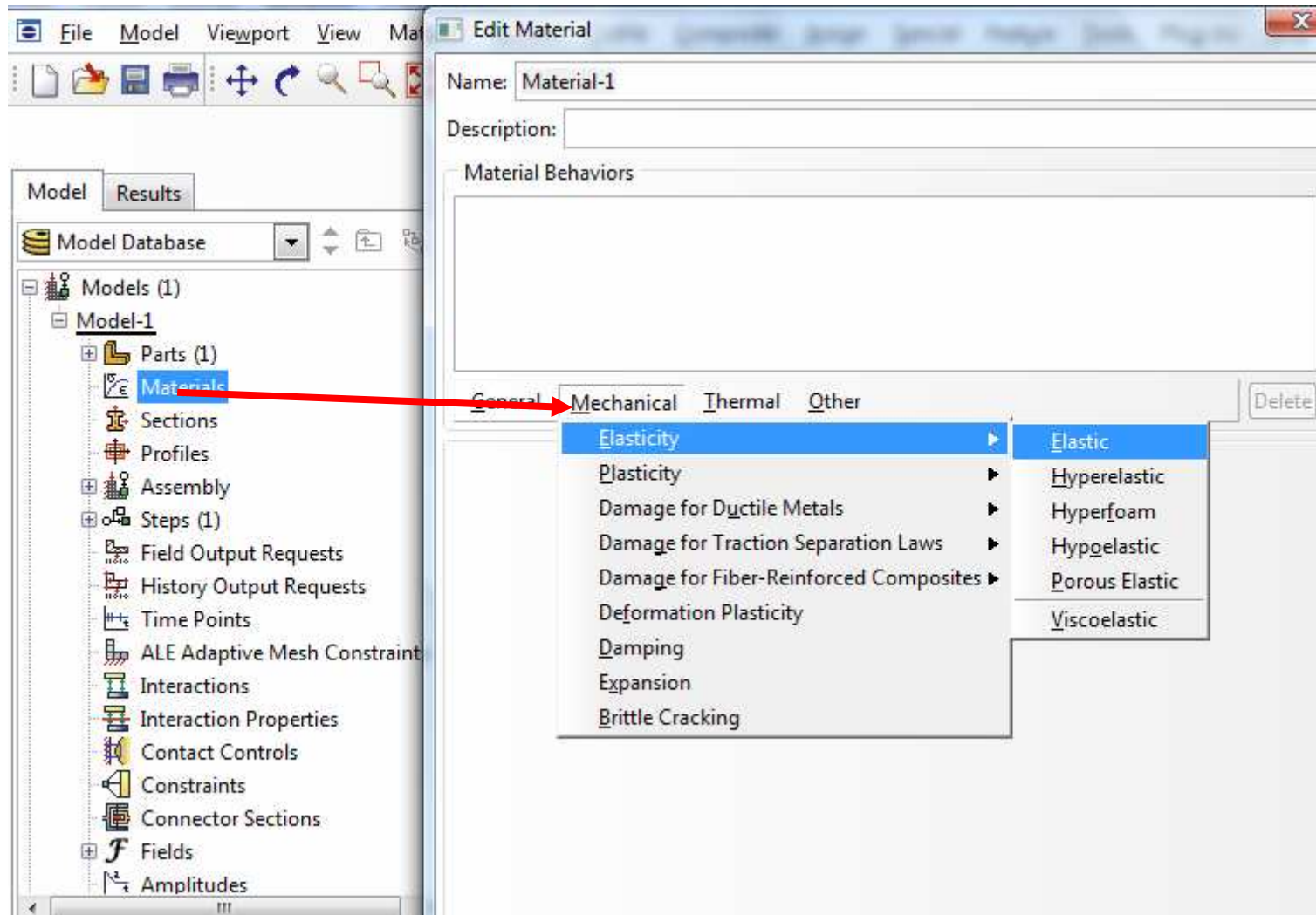




# Exercise 1 – 3D model

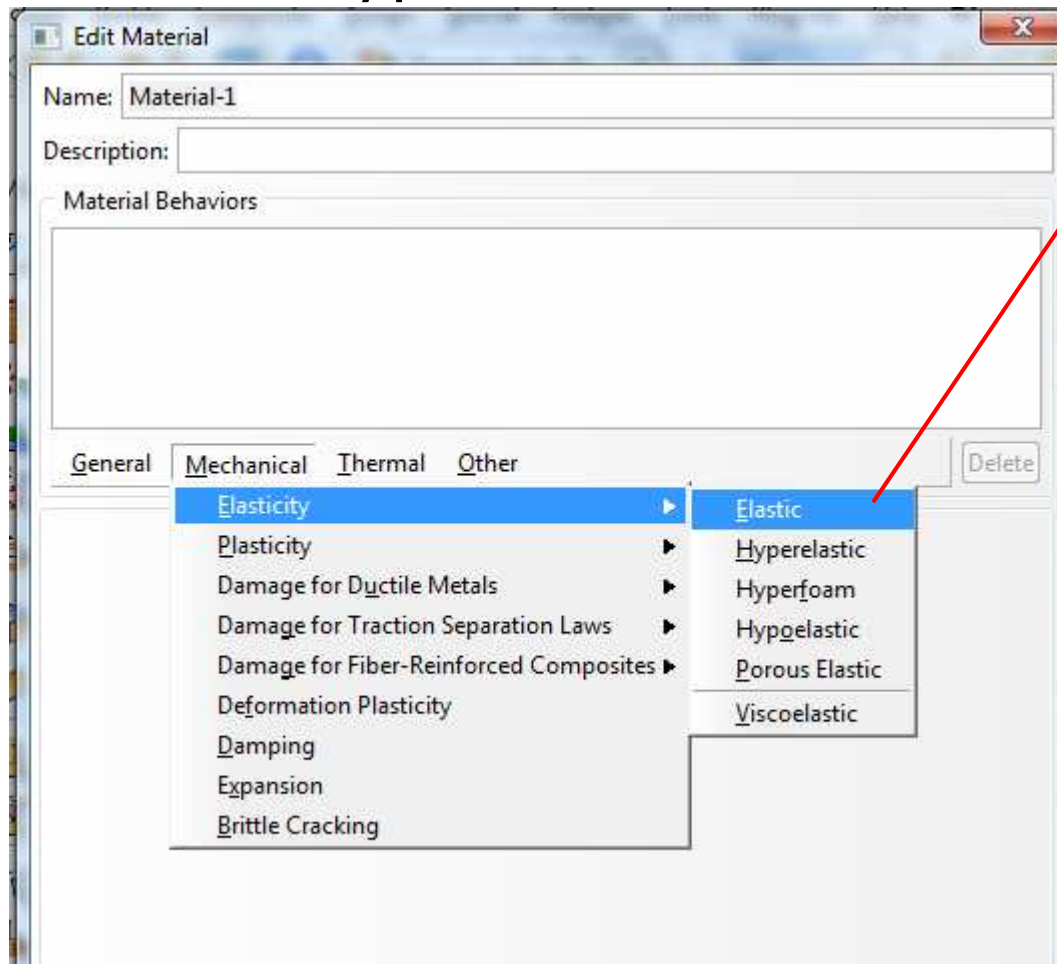


# Exercise 1 – materials data



## Exercise 1 – elastic data

- Material types

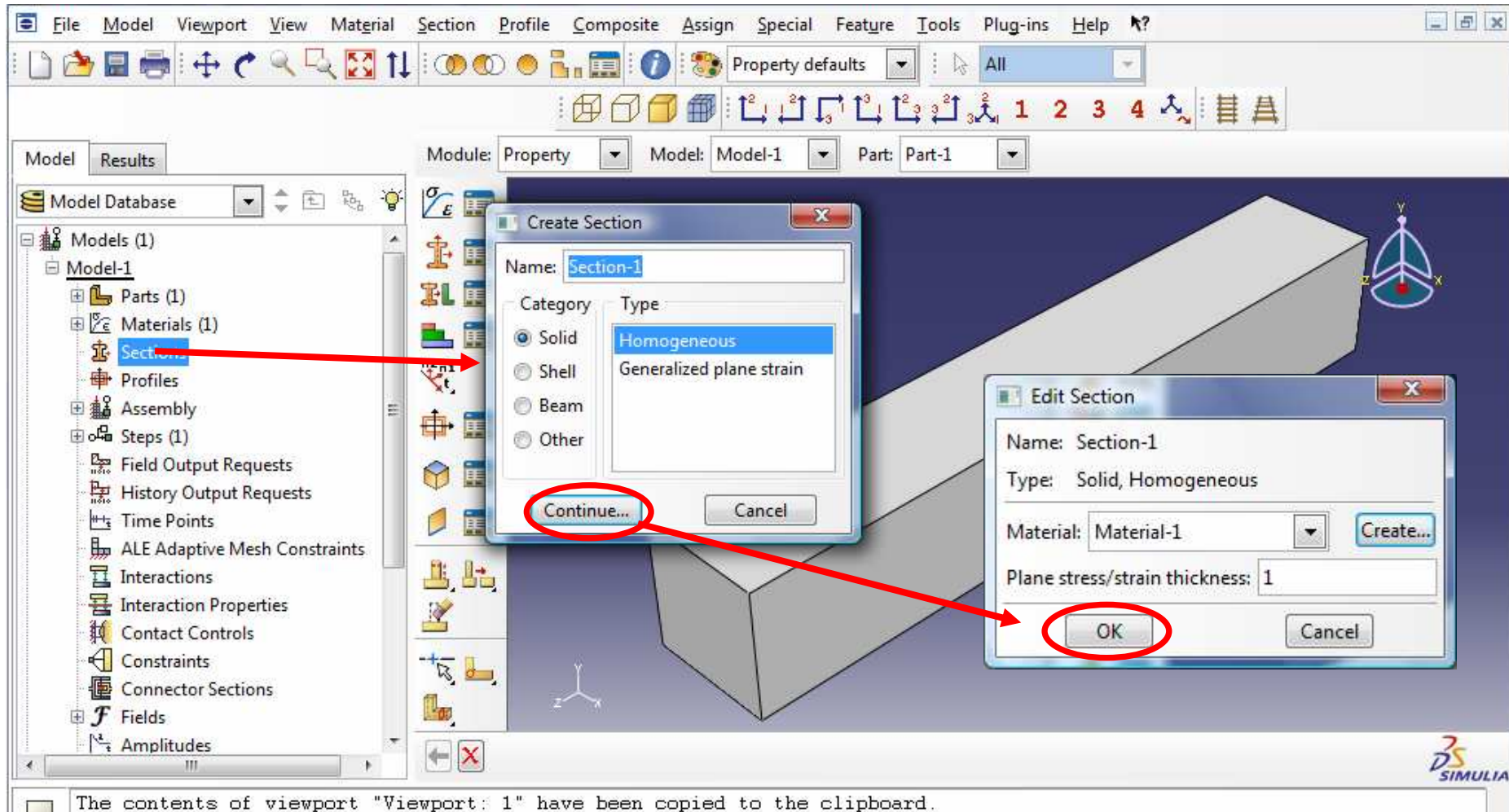


	Young's Modulus	Poisson's Ratio
1		

- Young Modulus – 200GPa
- Poisson's ratio – 0.35
- Create section
- Assign section



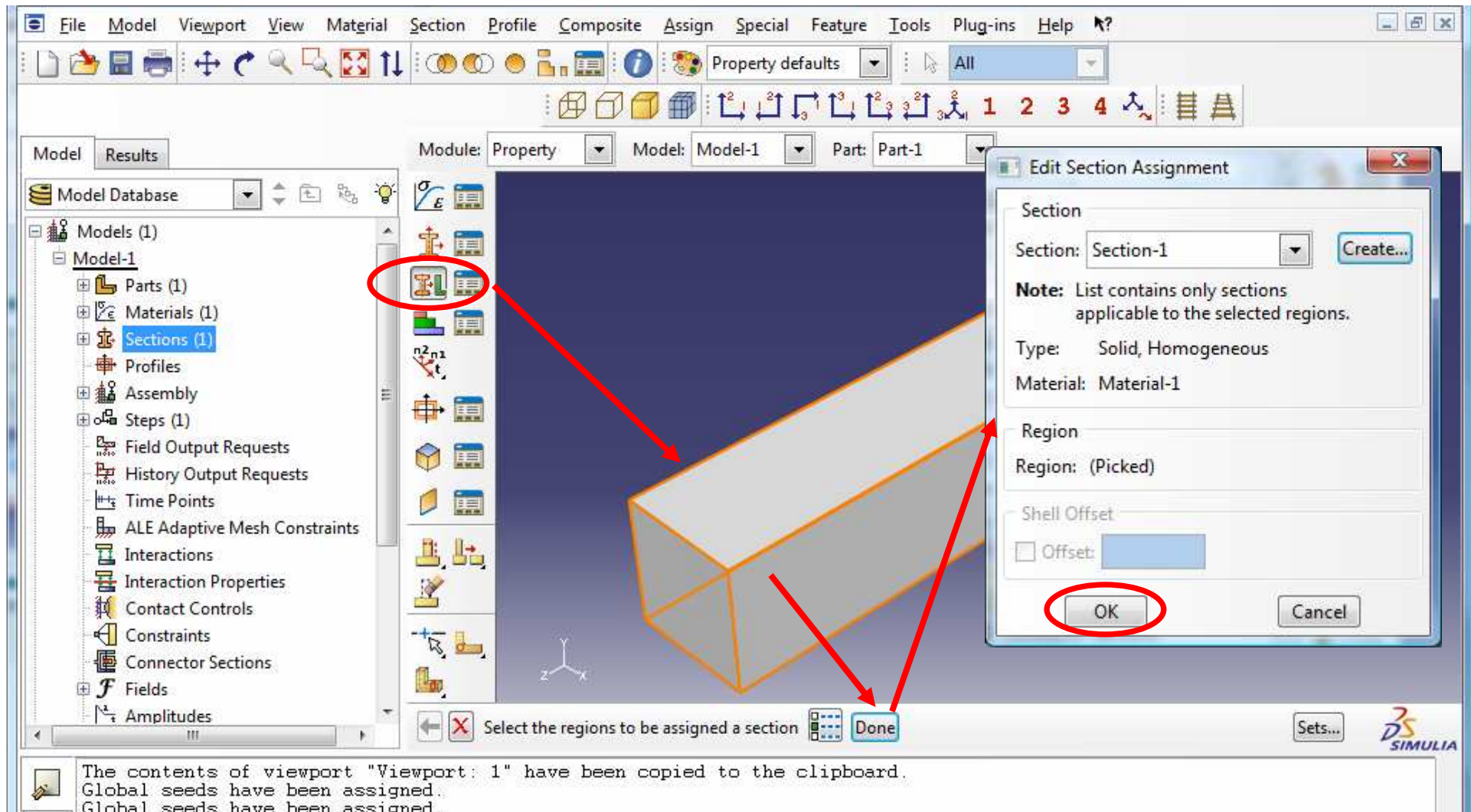
# Exercise 1 – material section



The screenshot displays the Abaqus software interface with the following elements:

- Menu Bar:** File, Model, Viewport, View, Material, Section, Profile, Composite, Assign, Special, Feature, Tools, Plug-ins, Help.
- Toolbar:** Standard Abaqus toolbars including navigation and analysis tools.
- Model Database:** A tree view on the left showing the model structure: Model-1 > Parts (1) > Materials (1) > Section-1 (highlighted).
- Create Section Dialog:** A dialog box titled 'Create Section' with the following fields:
  - Name: Section-1
  - Category: Solid (selected)
  - Type: Homogeneous (selected)
  - Buttons: Continue... (circled in red), Cancel
- Edit Section Dialog:** A dialog box titled 'Edit Section' with the following fields:
  - Name: Section-1
  - Type: Solid, Homogeneous
  - Material: Material-1
  - Plane stress/strain thickness: 1
  - Buttons: OK (circled in red), Cancel, Create...
- Status Bar:** A message at the bottom reads: "The contents of viewport 'Viewport: 1' have been copied to the clipboard."

# Exercise 1 – assign section



The screenshot displays the Abaqus software interface during the process of assigning a section to a model. The main viewport shows a 3D model of a rectangular block with a section assignment dialog box open. The dialog box, titled "Edit Section Assignment", has the following settings:

- Section:** Section-1 (with a "Create..." button)
- Note:** List contains only sections applicable to the selected regions.
- Type:** Solid, Homogeneous
- Material:** Material-1
- Region:** (Picked)
- Shell Offset:**  Offset: [text box]
- Buttons:** OK (circled in red), Cancel

The "Done" button at the bottom of the viewport is also circled in red. The "Model Database" on the left shows "Sections (1)" selected. The status bar at the bottom indicates: "The contents of viewport 'Viewport: 1' have been copied to the clipboard. Global seeds have been assigned. Global seeds have been assigned."

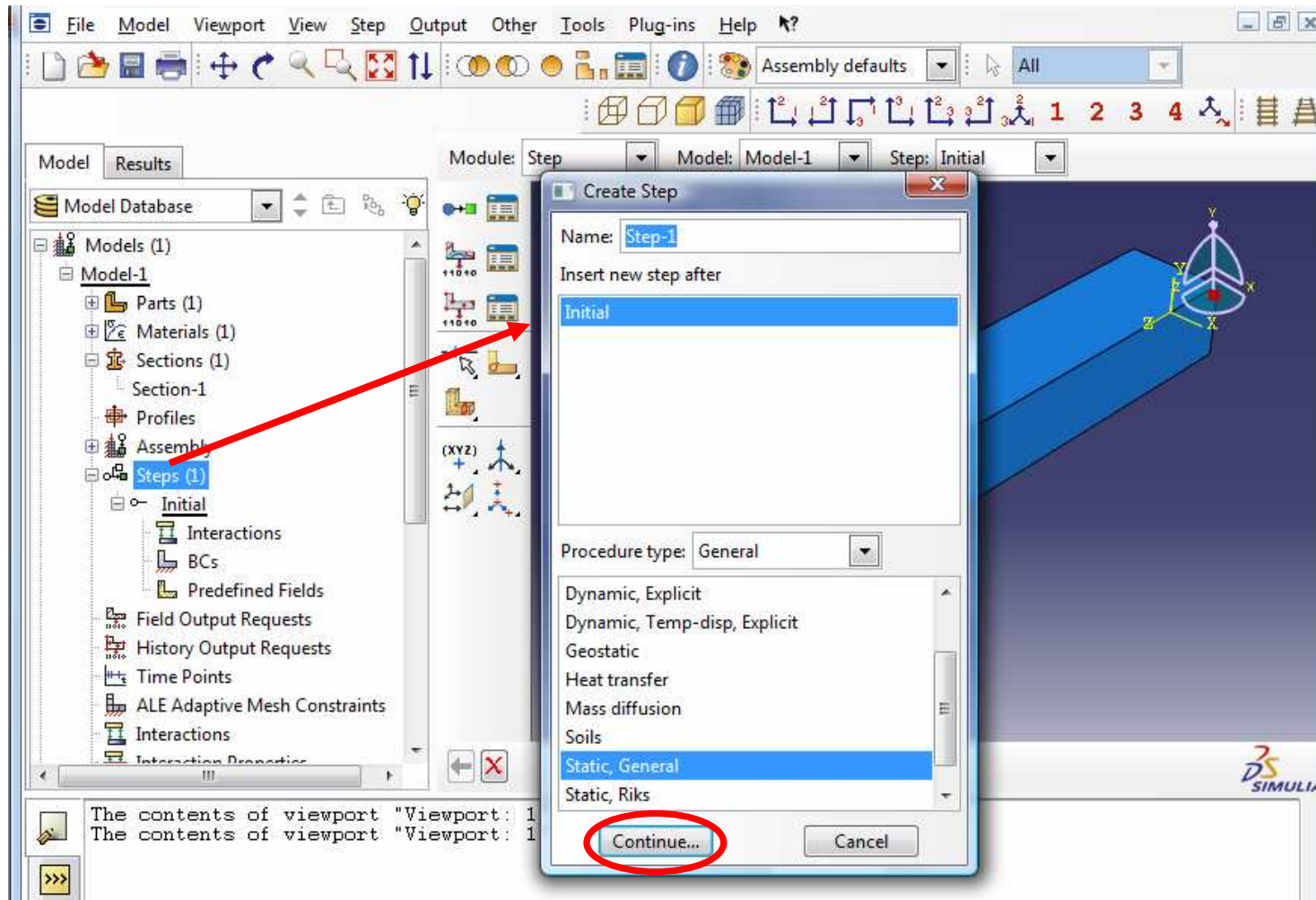


# Exercise 1 – assign section

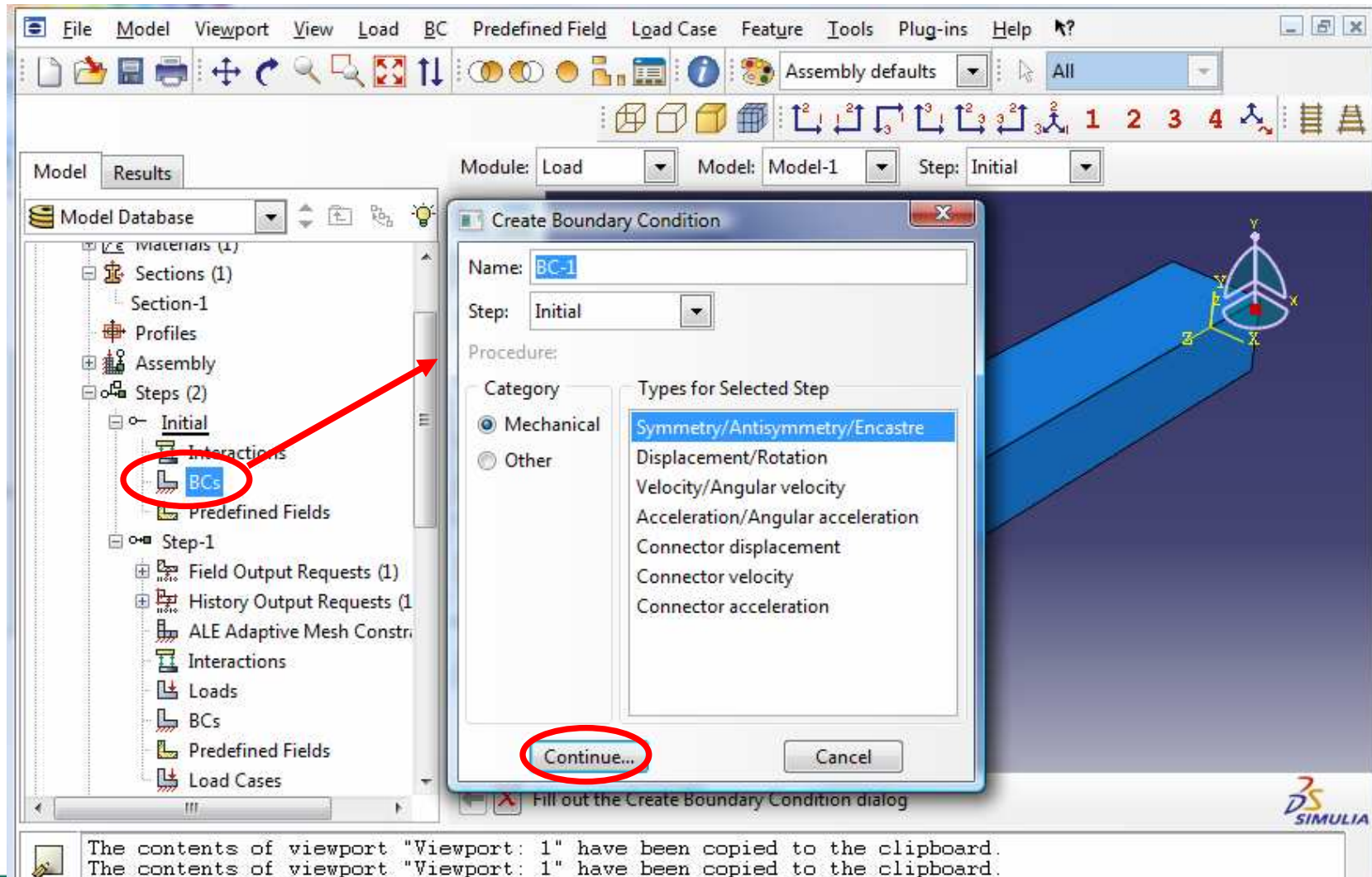
The screenshot displays the Abaqus software interface. The main window shows a 3D model of a rectangular block in a teal color, oriented along the X, Y, and Z axes. The interface includes a menu bar at the top with options like File, Model, Viewport, View, Material, Section, Profile, Composite, Assign, Special, Feature, Tools, Plug-ins, and Help. Below the menu bar is a toolbar with various icons for file operations, navigation, and modeling. The left side of the interface features a Model Database tree with a hierarchy: Models (1) > Model-1 > Parts (1) > Materials (1) > Sections (1) > Profiles > Assembly > Steps (1) > Field Output Requests > History Output Requests > Time Points > ALE Adaptive Mesh Constraints > Interactions > Interaction Properties > Contact Controls > Constraints > Connector Sections > Fields > Amplitudes. The 'Sections (1)' folder is highlighted. The central viewport shows the 3D model with a coordinate system. The status bar at the bottom contains a message: "The contents of viewport 'Viewport: 1' have been copied to the clipboard. Global seeds have been assigned. Global seeds have been assigned. The contents of viewport 'Viewport: 1' have been copied to the clipboard."



# Exercise 1 – analysis steps

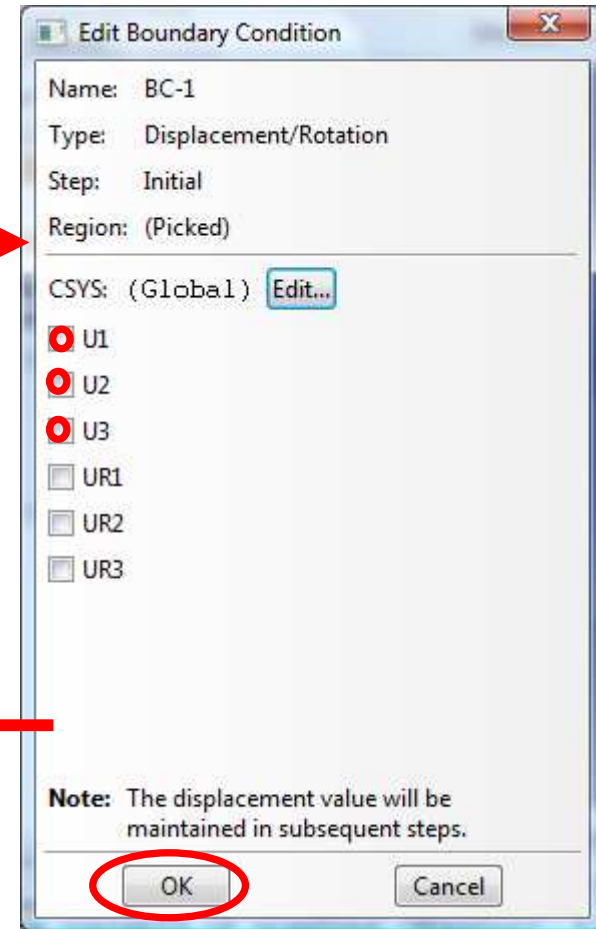
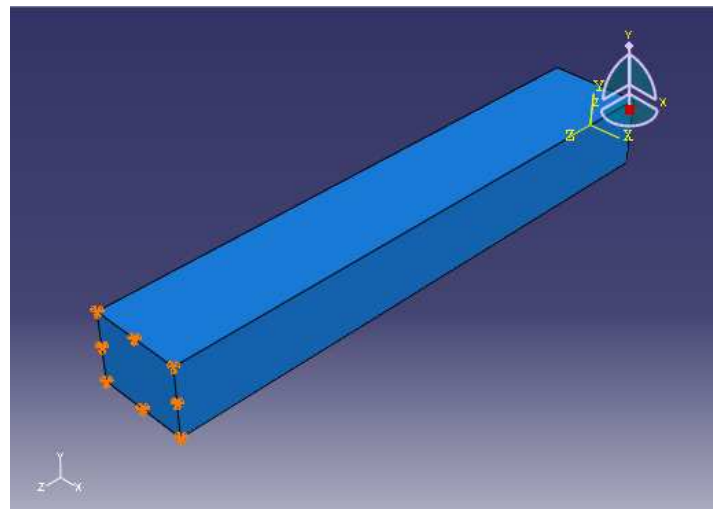
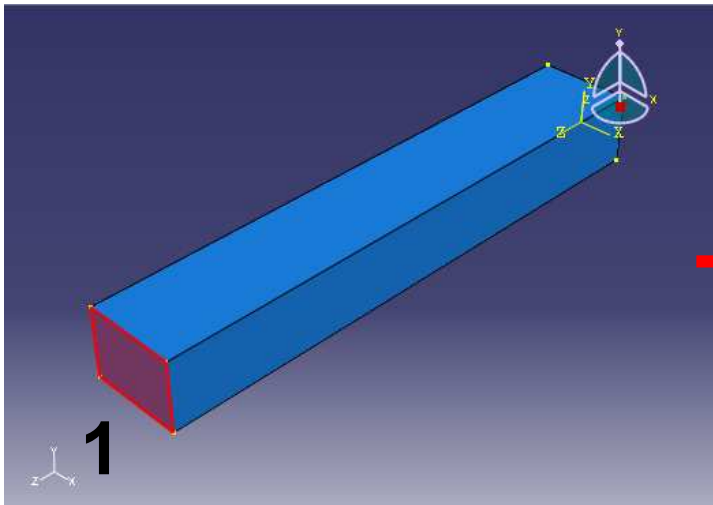


# Exercise 1 – boundary condition



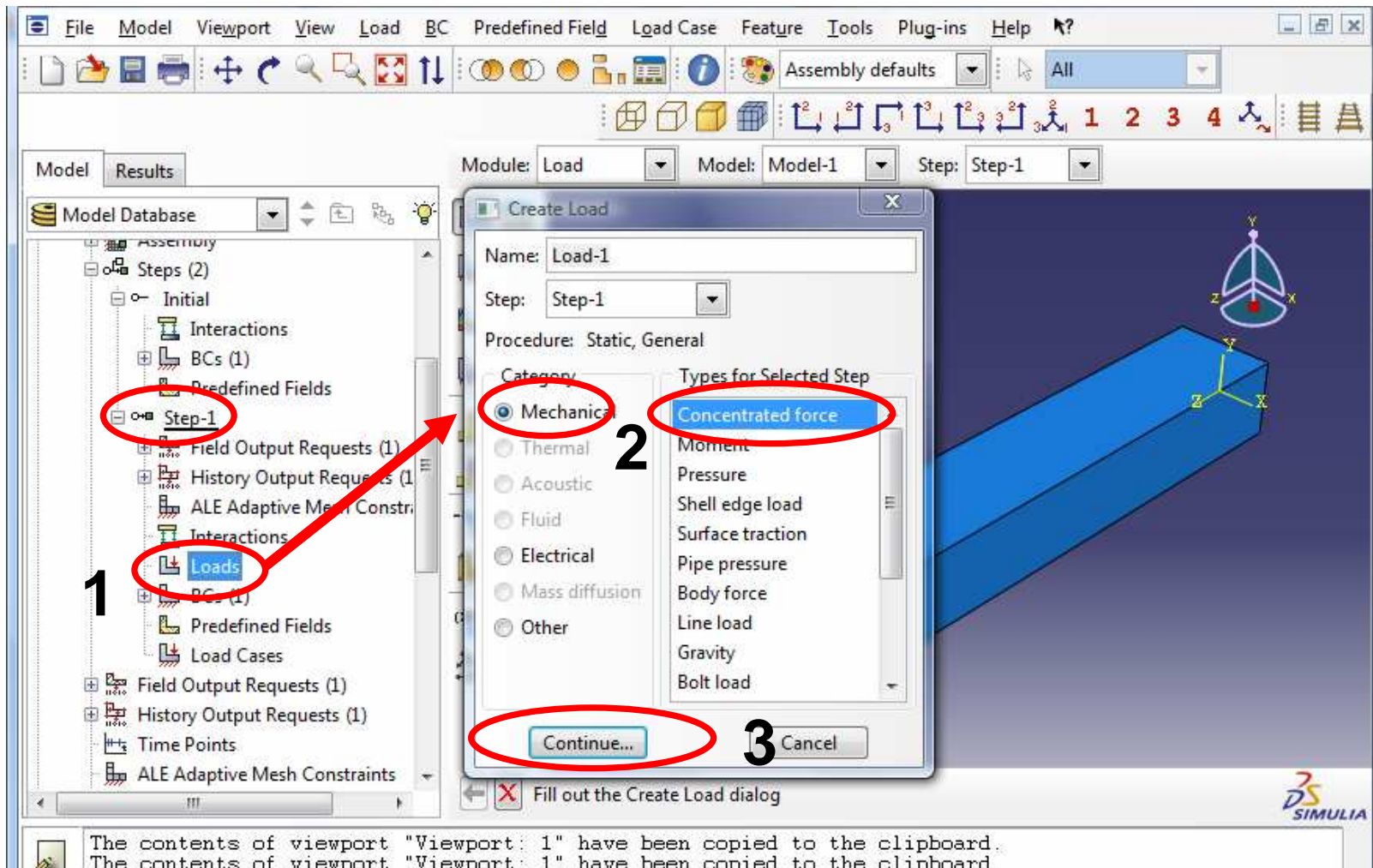
The contents of viewport "Viewport: 1" have been copied to the clipboard.  
The contents of viewport "Viewport: 1" have been copied to the clipboard.

# Exercise 1 – boundary condition



3

# Exercise 1 – loads



The screenshot shows the Abaqus software interface. The 'Model Database' on the left has 'Step-1' and 'Loads' circled in red. A red arrow points from 'Loads' to the 'Create Load' dialog box. In the dialog box, 'Mechanics' is selected under 'Category', and 'Concentrated force' is selected under 'Types for Selected Step'. The 'Continue...' button is also circled in red. The background shows a 3D model of a blue rectangular block with a coordinate system.

1

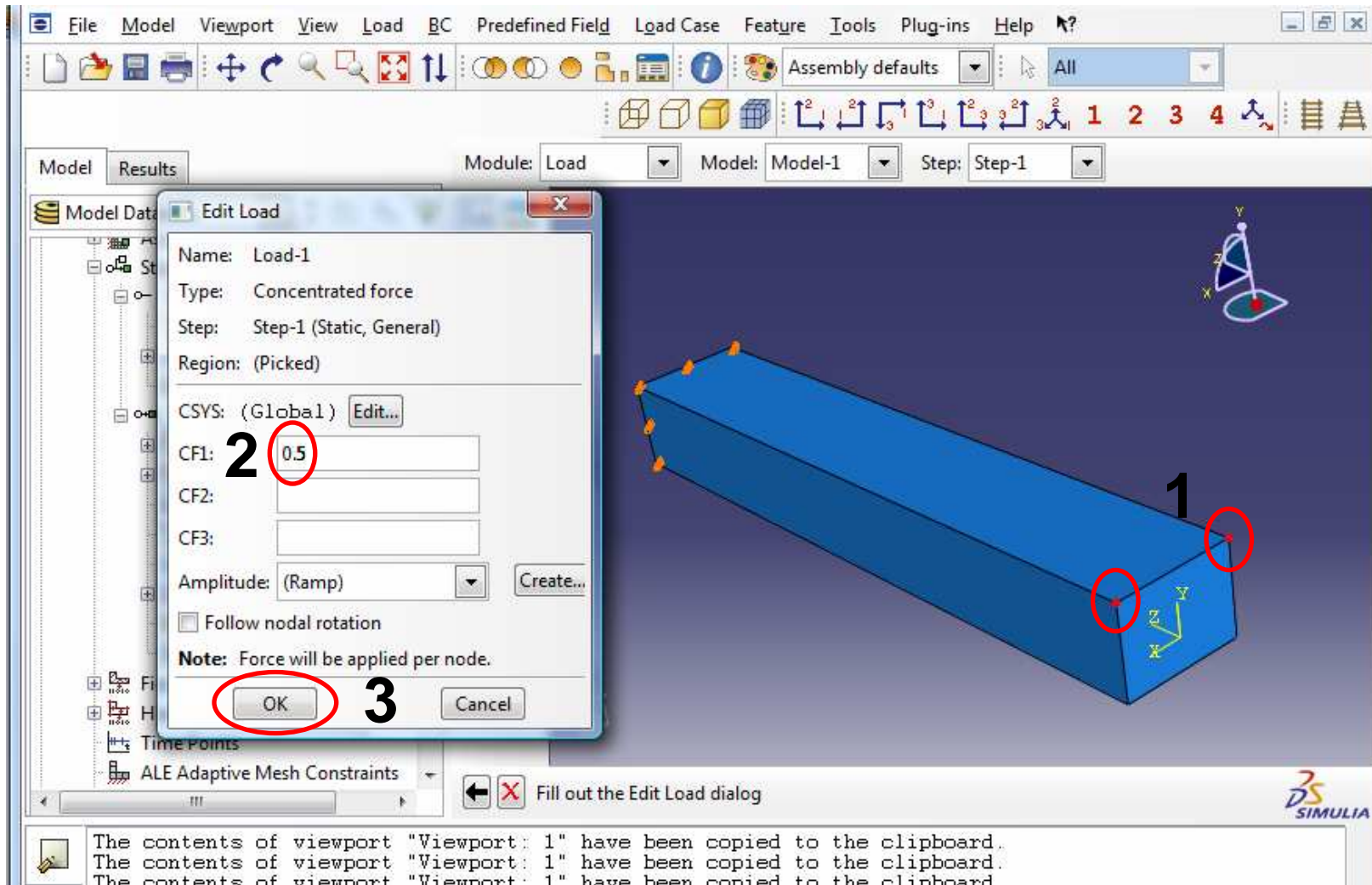
2

3

The contents of viewport "Viewport: 1" have been copied to the clipboard.  
The contents of viewport "Viewport: 1" have been copied to the clipboard.



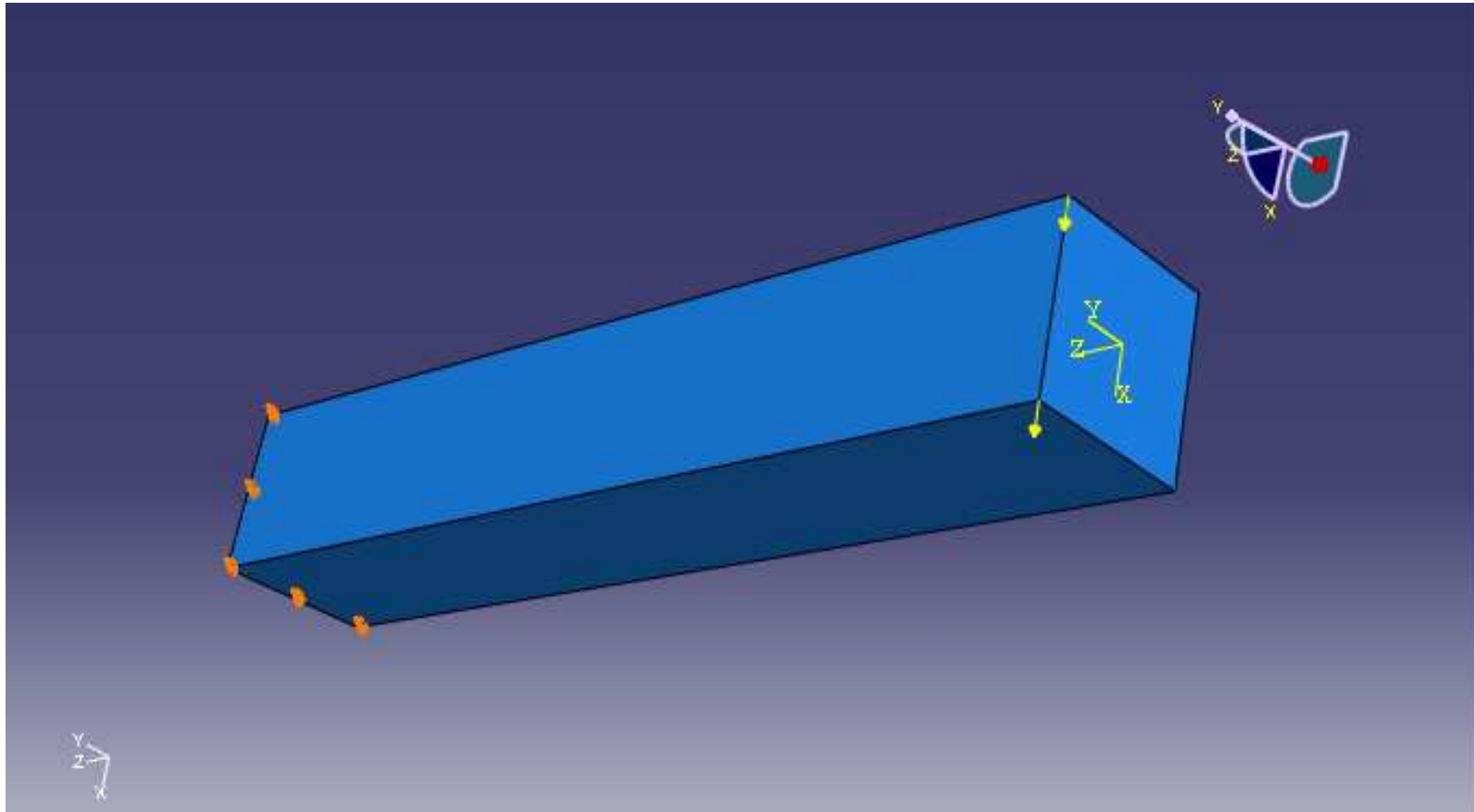
# Exercise 1 – loads



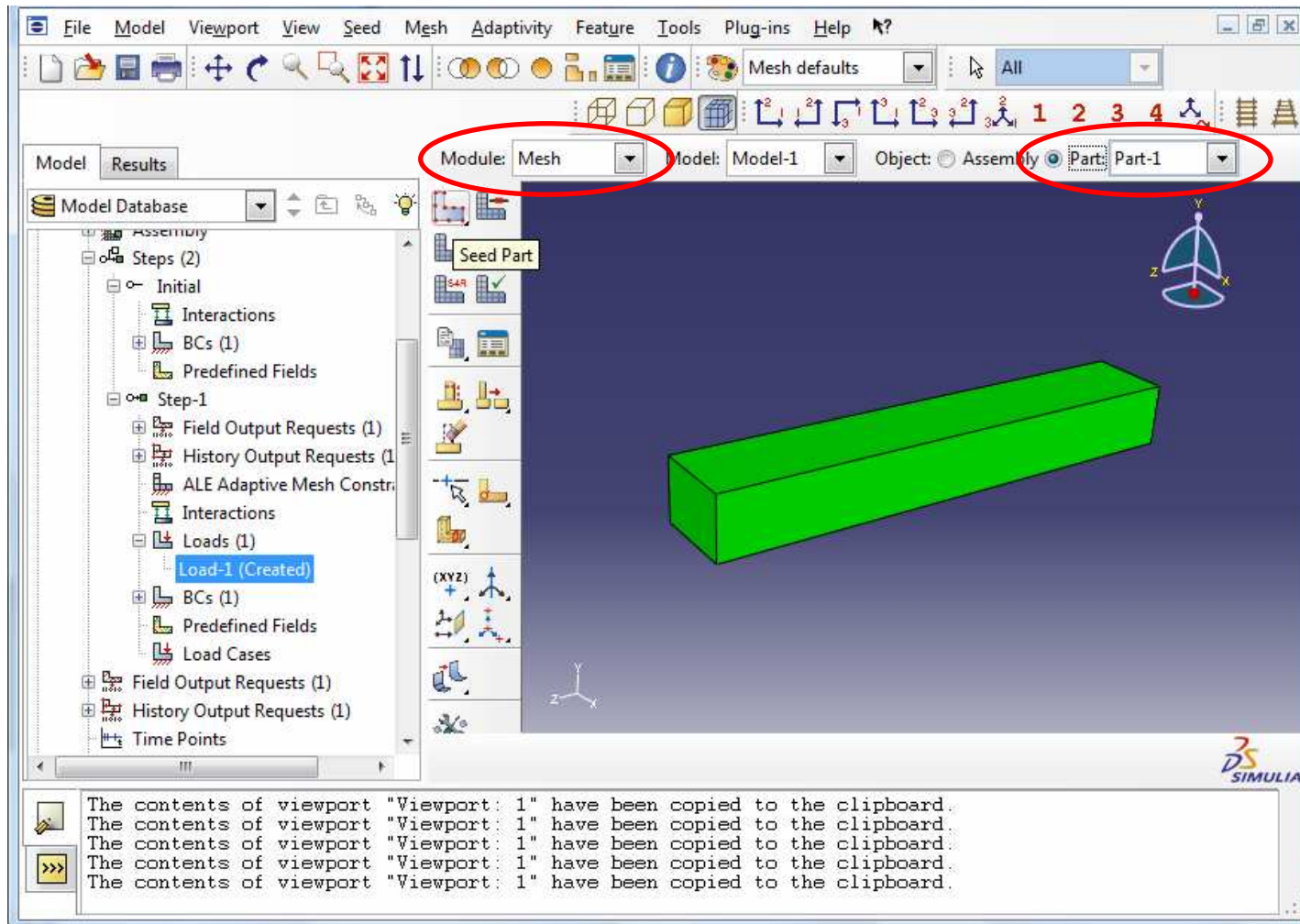
The contents of viewport "Viewport: 1" have been copied to the clipboard.  
The contents of viewport "Viewport: 1" have been copied to the clipboard.  
The contents of viewport "Viewport: 1" have been copied to the clipboard.



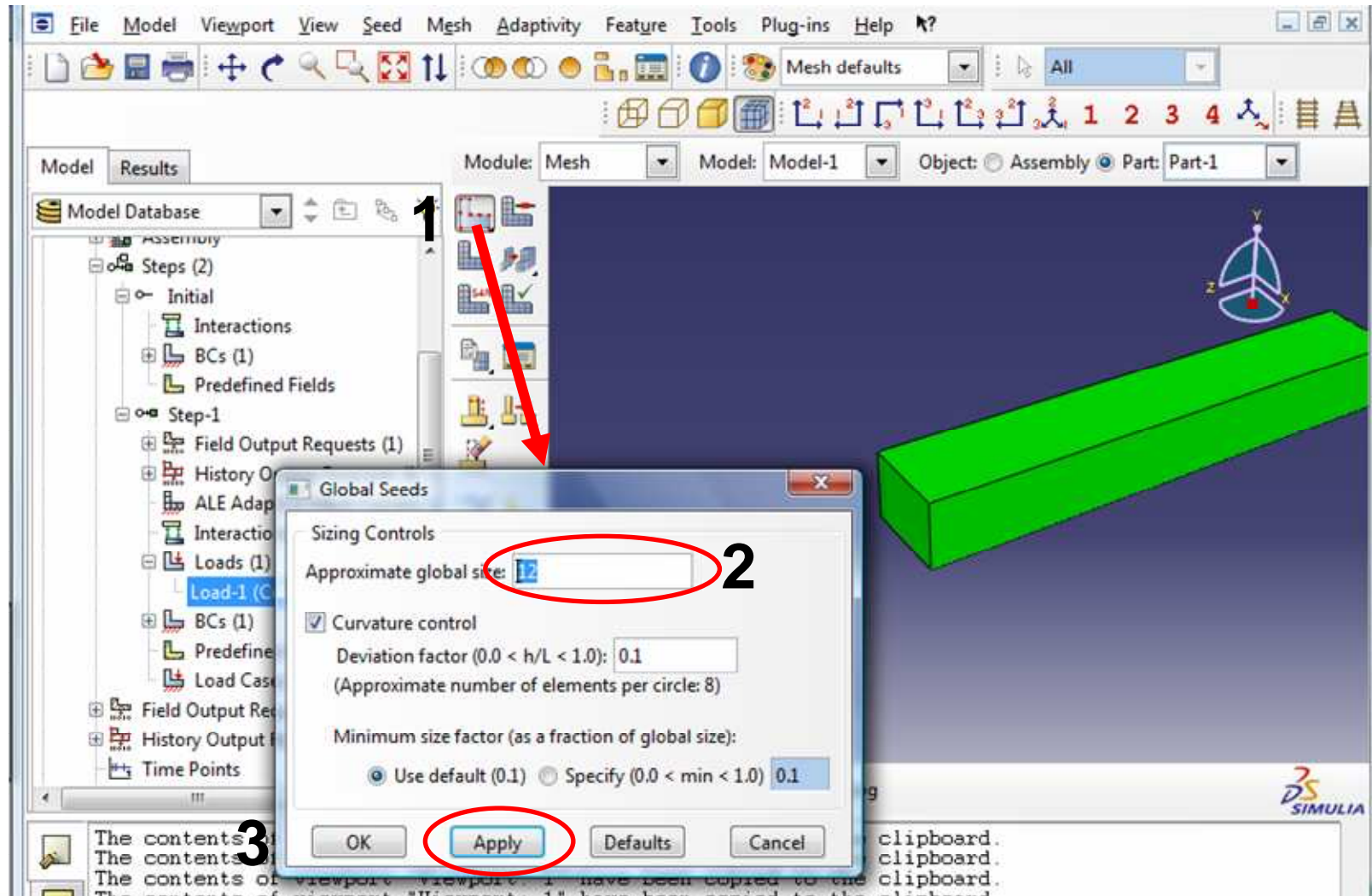
# Exercise 1 – loads



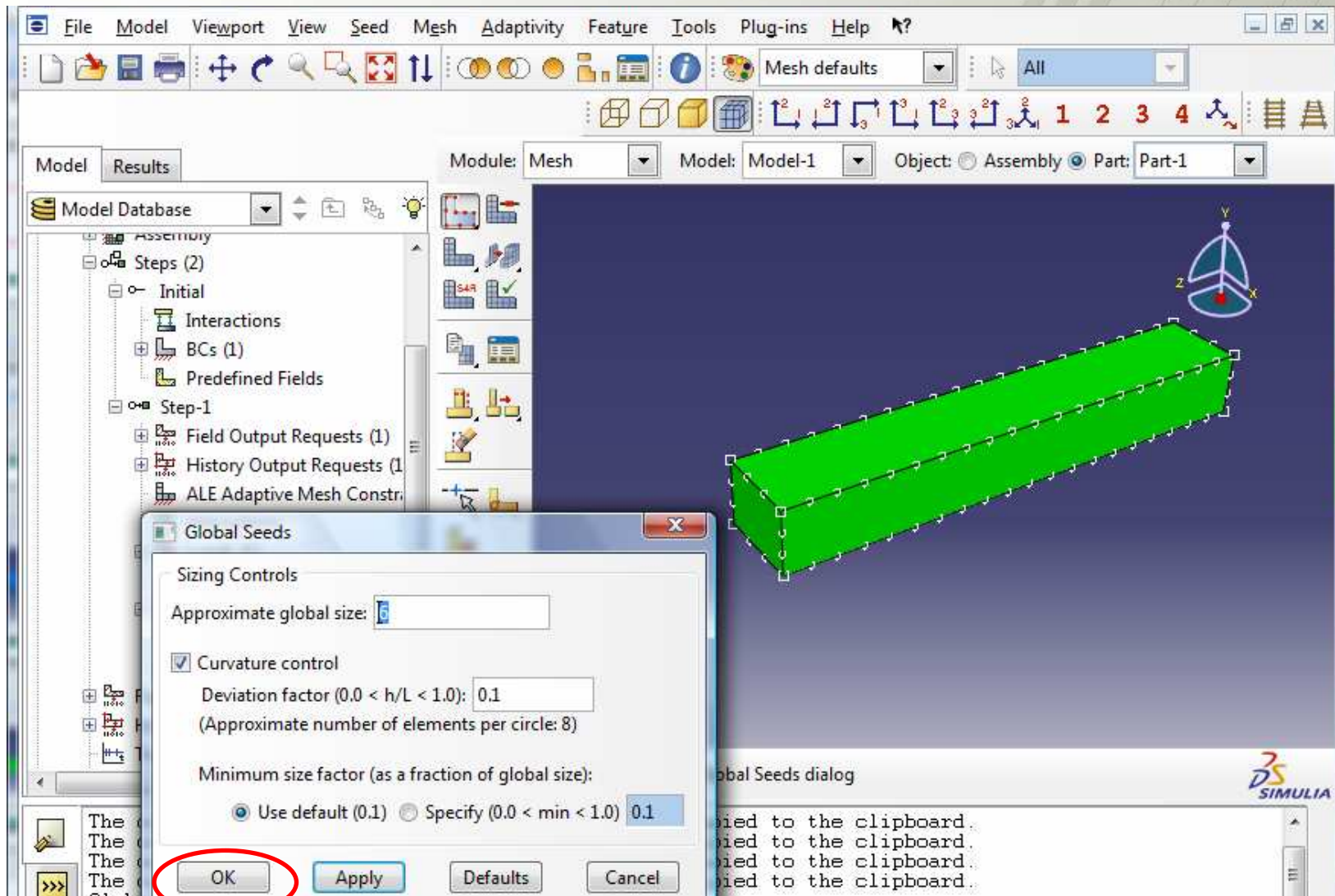
# Exercise 1 – mesh



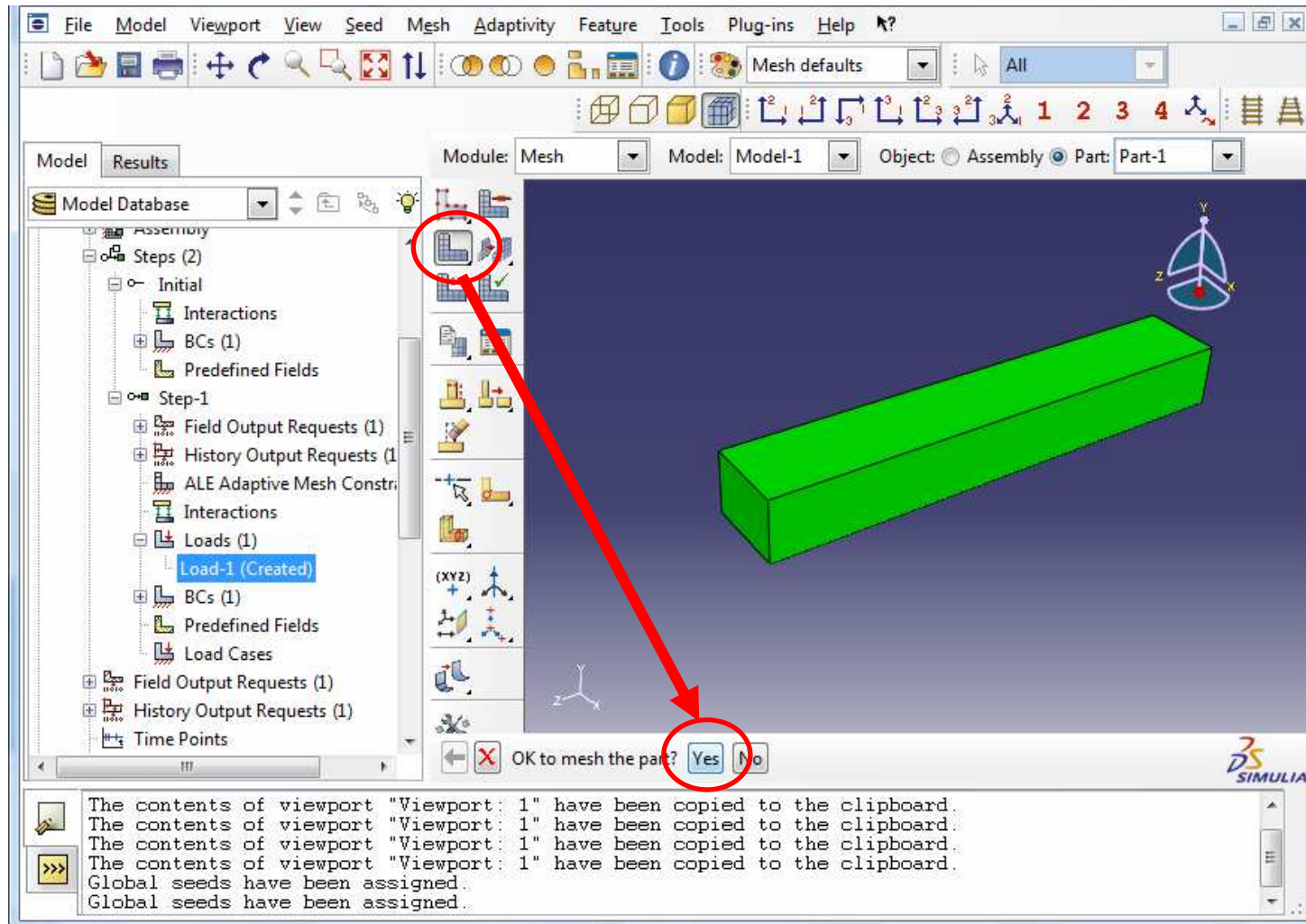
# Exercise 1 – mesh



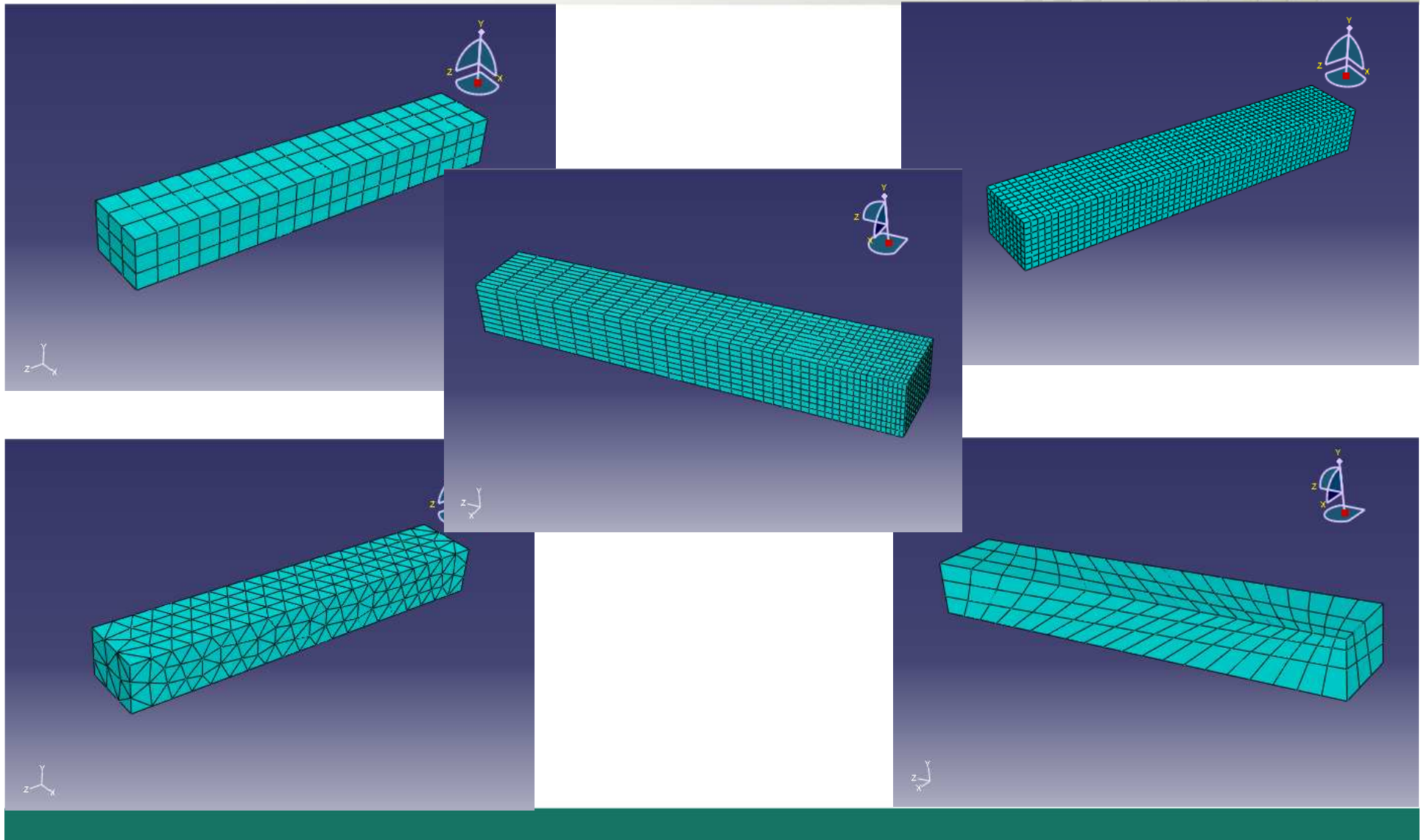
# Exercise 1 – mesh



# Exercise 1 – mesh



# Exercise 1 – mesh





# Exercise 1 – create job

The screenshot displays the Simulia software interface with several dialog boxes open. In the background, the 'Model Database' tree shows 'Model-1' expanded, with 'Jobs' circled in red. A 'Create Job' dialog box is open, showing 'Name: Job-1' and 'Source: Model-1' selected in a list. A red arrow points from the 'Jobs' entry in the tree to the 'Create Job' dialog. In the foreground, an 'Edit Job' dialog box is open, showing 'Name: Job-1' and 'Model: Model-1'. The 'Submission' tab is active, with 'Full analysis' selected under 'Job Type'. The 'Background' radio button is selected under 'Run Mode'. The 'Submit Time' section has 'Immediately' selected. The 'OK' button at the bottom of the 'Edit Job' dialog is circled in red. The status bar at the bottom shows the message: '180 elements have been generated on part: Part-1'.





# Exercise 1 – job submit

The screenshot displays the ABAQUS Job Manager interface. The main window shows a table with the following data:

Name	Model	Type	Status
Job-1	Model-1	Full Analysis	None

Below this table, the 'Submit' button is circled in red. A second Job Manager window is overlaid, showing a table with the following data:

Name	Model	Type	Status
Job-12	Model-1	Full Analysis	Running

In this second window, the 'Status' column for Job-12 is circled in red. A warning dialog box titled 'ABAQUS' is also present, with the following text:

Job files already exist for Job-1.  
OK to overwrite?  
 Show this warning next time

The 'OK' button in the dialog is circled in red. Red arrows point from the 'Submit' button in the first window to the dialog, and from the 'OK' button in the dialog to the 'Status' cell in the second window.



# Exercise 1 – job submit

The screenshot shows the ABAQUS Job Manager interface. The main window displays a table of jobs. A secondary window shows a warning dialog box.

Name	Model	Type	Status
Job-1	Model-1	Full Analysis	None

Name	Model	Type	Status
Job-12	Model-1	Full Analysis	Running

**ABAQUS Warning Dialog:**

Job files already exist for Job-1.  
OK to overwrite?

Show this warning next time

Buttons: OK, Cancel

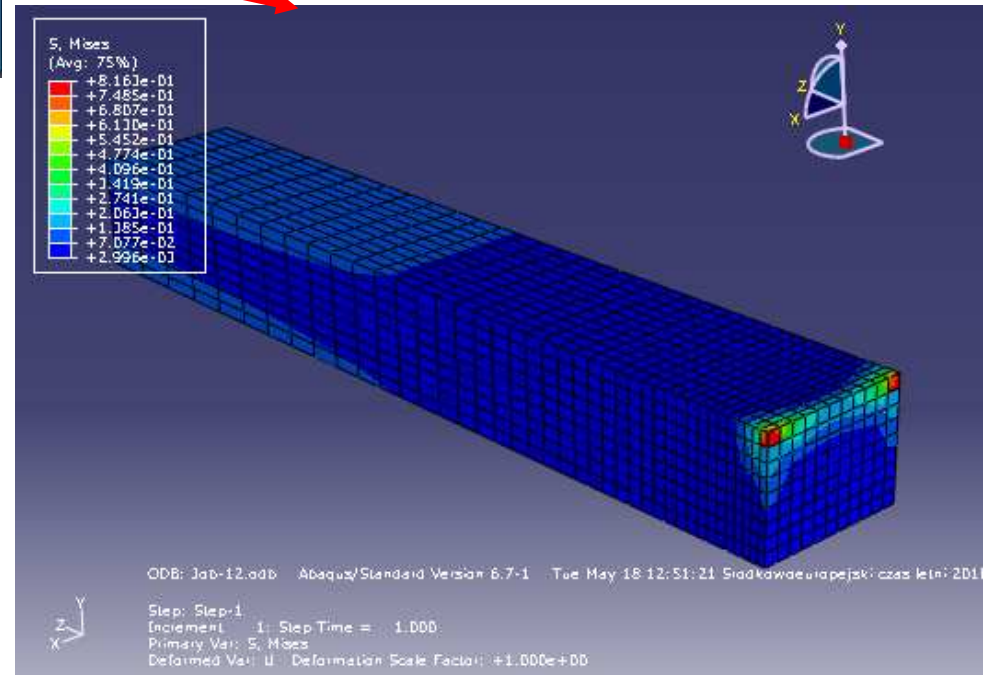
Buttons in Job Manager: Write Input, Data Check, Submit, Continue, Monitor..., Results, Kill, Create..., Edit..., Copy..., Rename..., Delete..., Dismiss



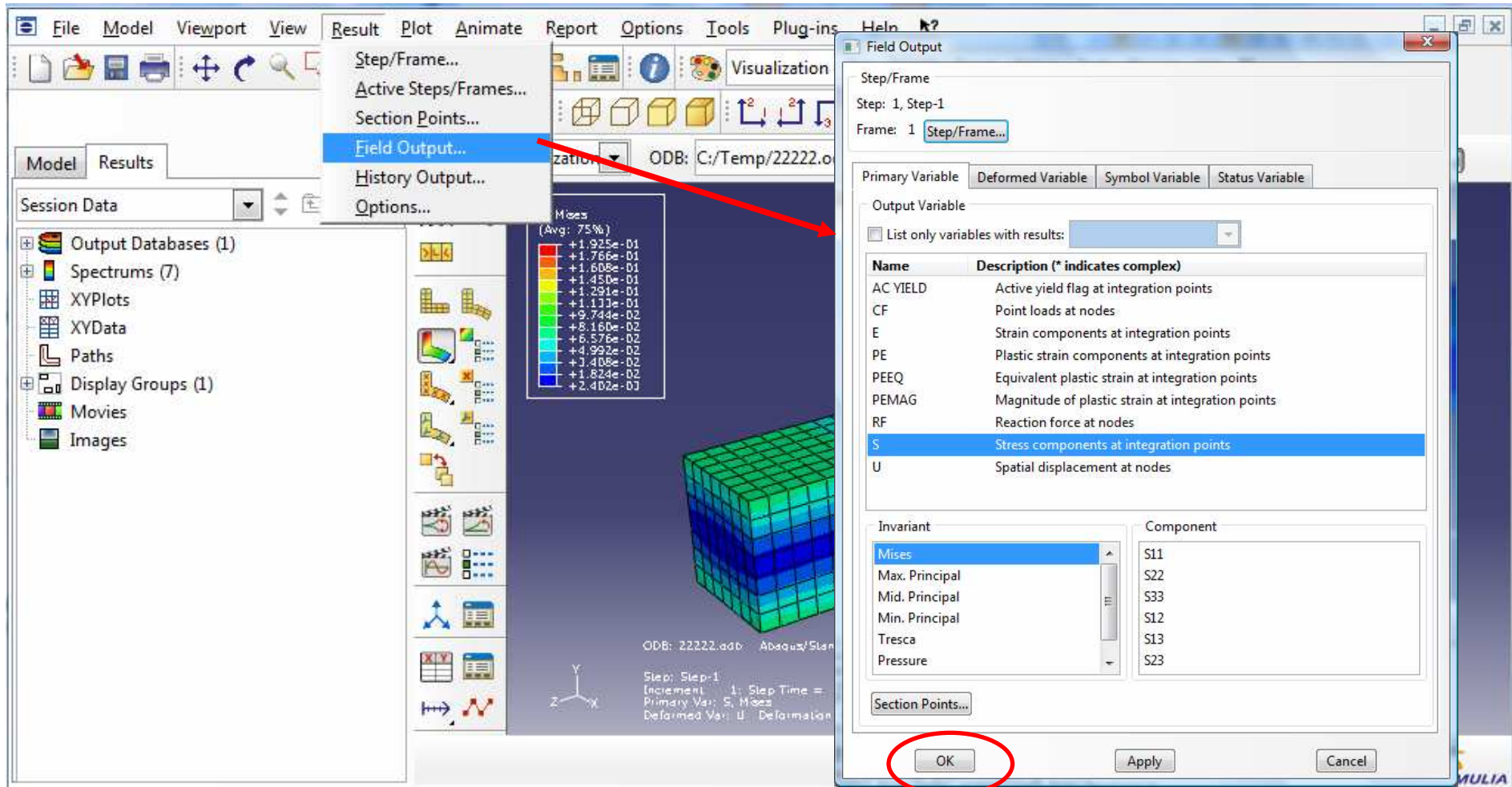
# Exercise 1 – results – misses stress

Name	Model	Type	Status
Job-12	Model-1	Full Analysis	Completed

Buttons: Write Input, Data Check, Submit, Continue, Monitor..., Results, Kill, Dismiss, Create..., Edit..., Copy..., Rename..., Delete...



# Exercise 1 – results reading



The screenshot shows the Abaqus software interface with the 'Field Output' dialog box open. The dialog is configured for 'Step: 1, Step-1' and 'Frame: 1'. The 'Output Variable' list includes:

Name	Description (* indicates complex)
AC YIELD	Active yield flag at integration points
CF	Point loads at nodes
E	Strain components at integration points
PE	Plastic strain components at integration points
PEEQ	Equivalent plastic strain at integration points
PEMAG	Magnitude of plastic strain at integration points
RF	Reaction force at nodes
<b>S</b>	<b>Stress components at integration points</b>
U	Spatial displacement at nodes

The 'Invariant' list includes:

<b>Mises</b>
Max. Principal
Mid. Principal
Min. Principal
Tresca
Pressure

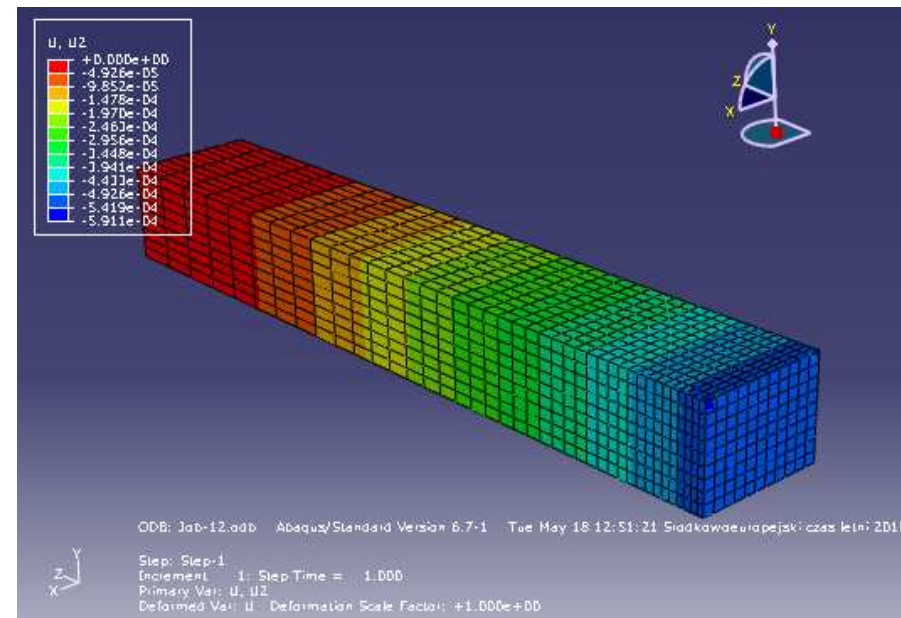
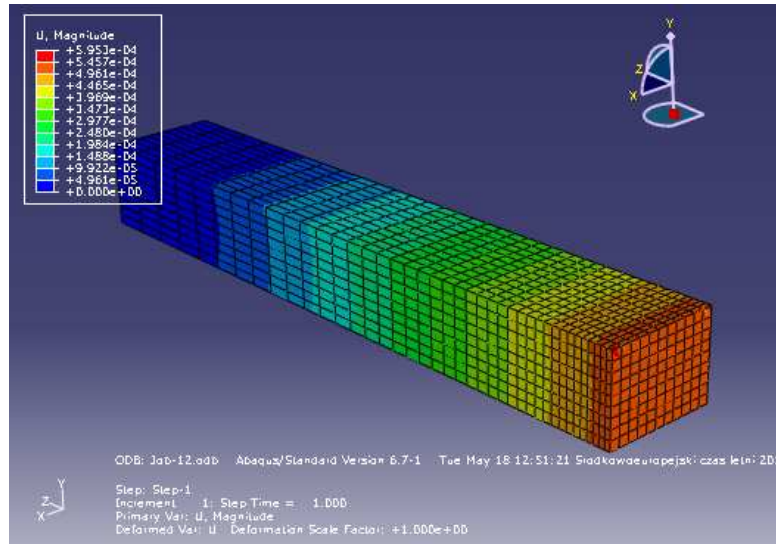
The 'Component' list includes:

S11
S22
S33
S12
S13
S23

The 'OK' button is circled in red.

# Exercise 1 – results

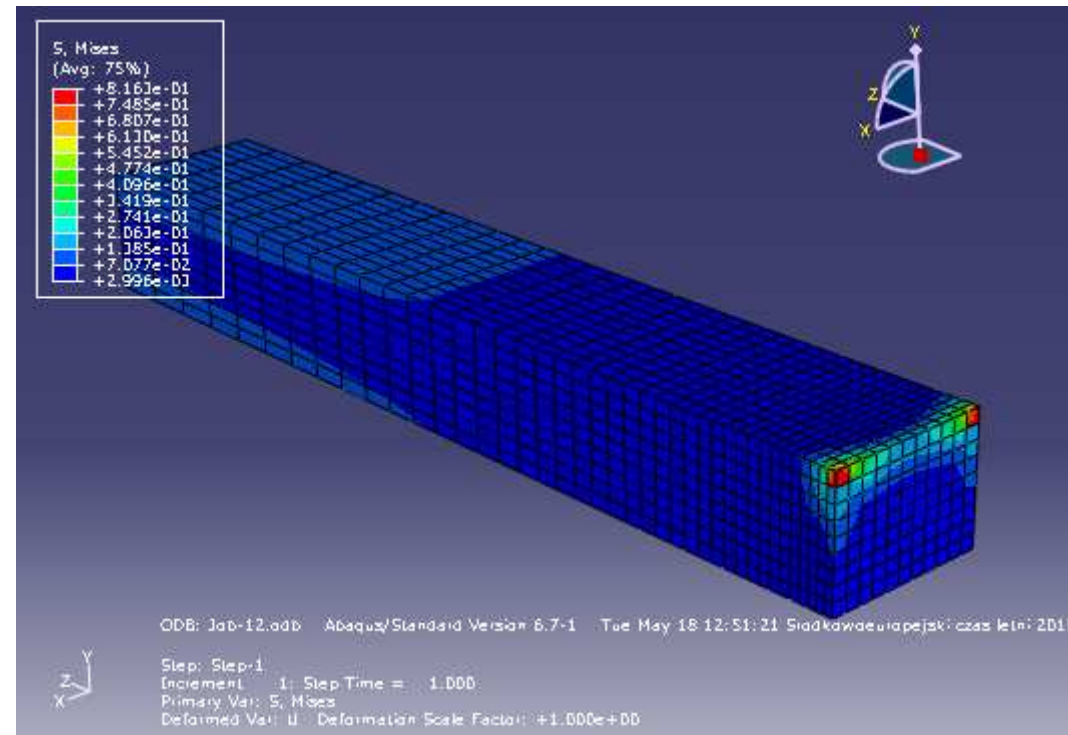
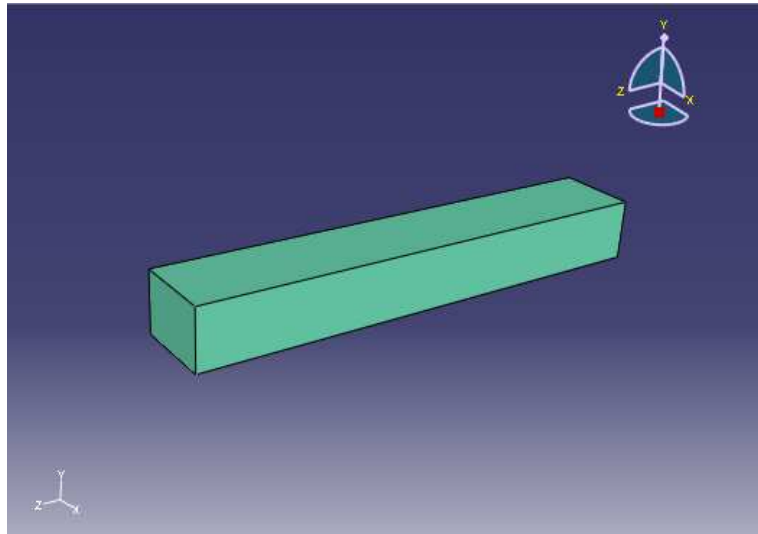
## Displacement - magnitude



## Displacement - u2 direction



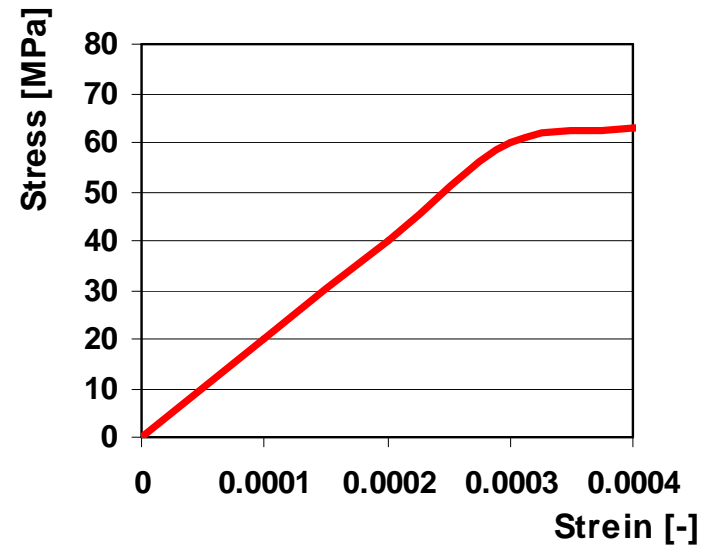
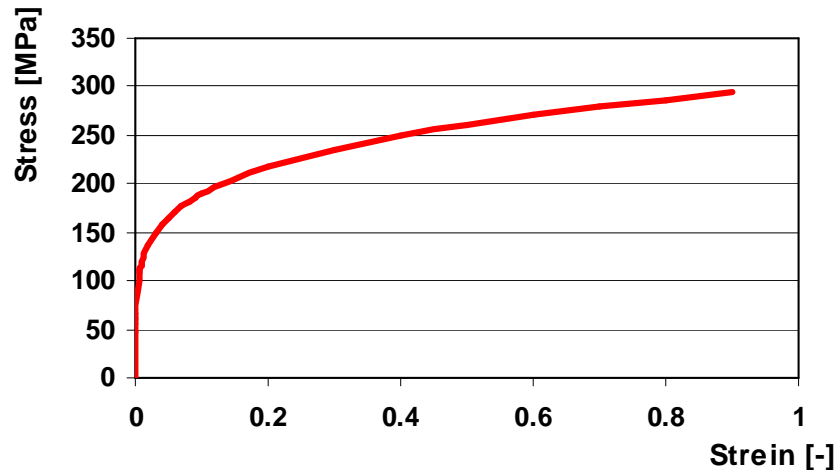
# Exercise 2 – plastic analysis



## Exercise 2 – plastic analysis

### Definition of stress-strain curve:

$$\sigma_p = A \varepsilon_i^n$$



strain	stress
0	60
0.001	75.35
0.002	86.56
0.003	93.87
0.004	99.43
0.005	103.97
0.01	119.43
0.03	148.78
0.05	164.78
0.07	176.28
0.09	185.3
0.11	192.9
0.2	217.43
0.3	235.80
0.4	249.7
0.5	261.16
0.6	270.86
0.7	279.3
0.8	286.90
0.9	293.74
1	300



# Exercise 2 – plastic analysis

Edit Material

Name: Material-1

Description:

Material Behaviors

- Elastic
- Plastic

General Mechanical Thermal Other Delete

Plastic

Hardening: Isotropic Suboptions

Use strain-rate-dependent data

Use temperature-dependent data

Number of field variables: 0

Data

	Yield Stress	Plastic Strain
1		

OK Cancel

Edit Material

Name: Material-1

Description:

Material Behaviors

- Elastic
- Plastic

General Mechanical Thermal Other Delete

Plastic

Hardening: Isotropic Suboptions

Use strain-rate-dependent data

Use temperature-dependent data

Number of field variables: 0

Data

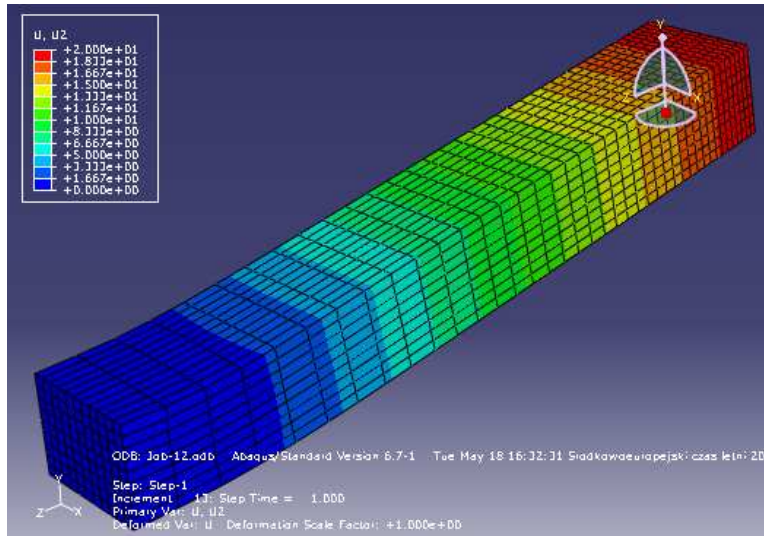
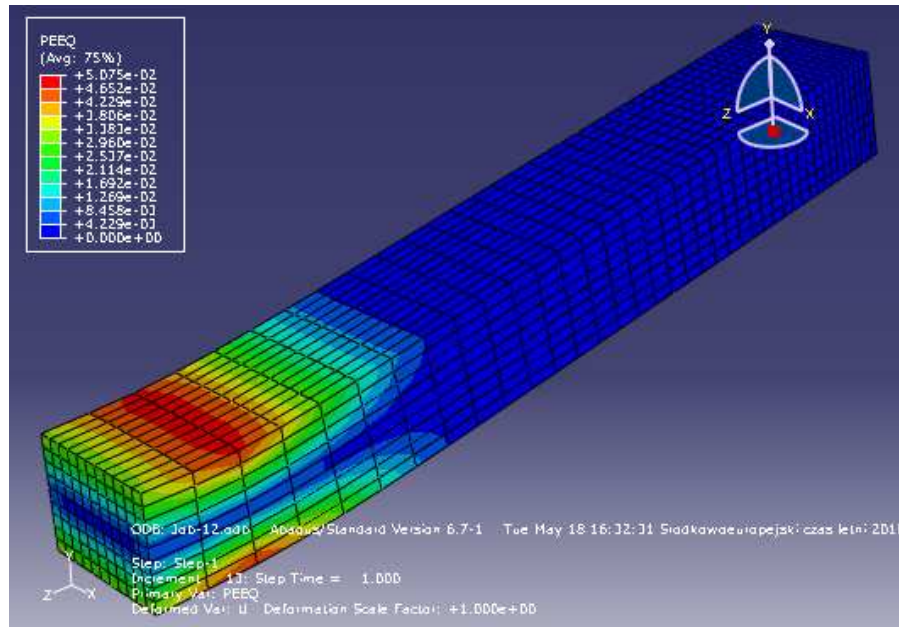
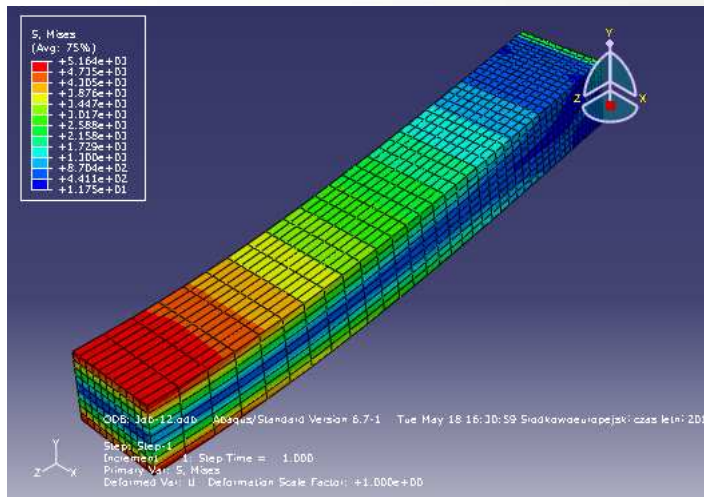
	Yield Stress	Plastic Strain
1	60	0
2	75.35659	0.001
3	86.56199	0.002
4	93.87404	0.003
5	99.43362	0.004
6	103.9717	0.005
7	119.4322	0.01
8	148.7803	0.03
9	161.7811	0.05

OK Cancel





# Exercise 2 – results





# Exercise 2 – results

