

MODELING & ANALYSIS OF FLUID FLOW IN CIRCULAR TUBE USING ANSYS FLUENT

UNILAG ANSYS HANDS-ON TUTORIAL 1B (MEG 222)

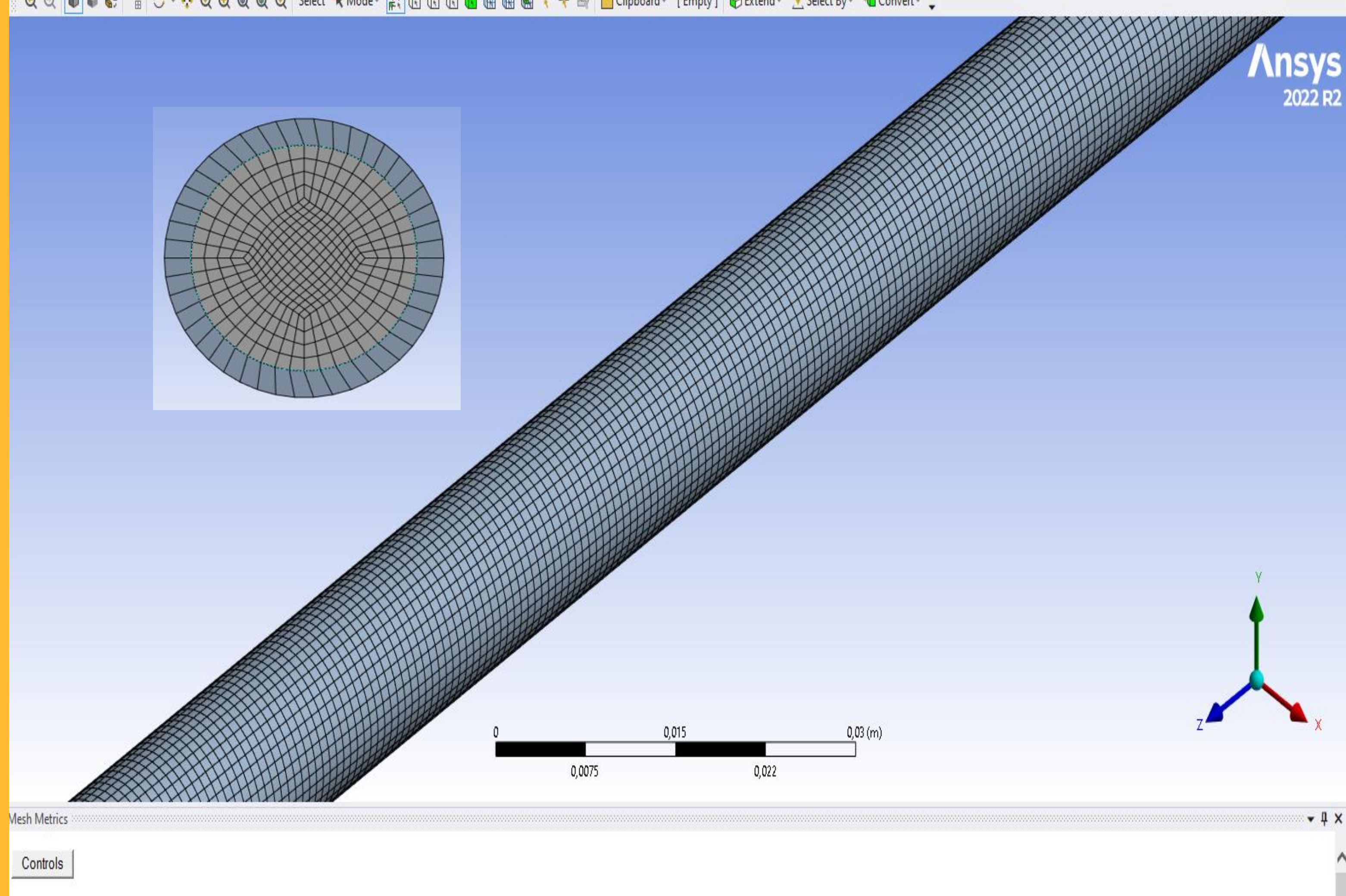
FLUID FLOW MODELING

At the end of this second tutorial, you will be able to

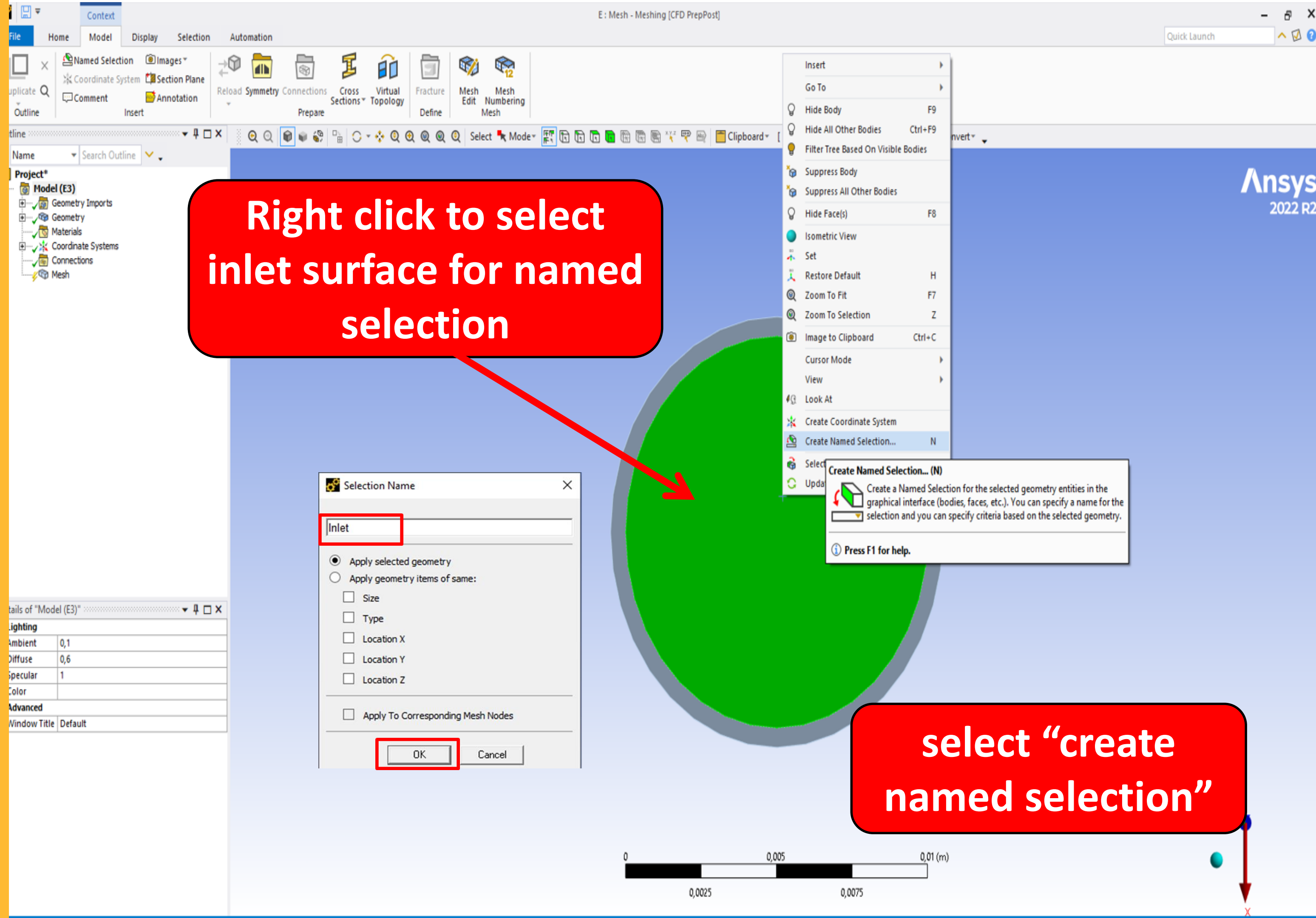
- 01** Create named selections on the boundaries of the geometry
- 02** Mesh the circular tube geometry

MESHING THE GEOMETRY

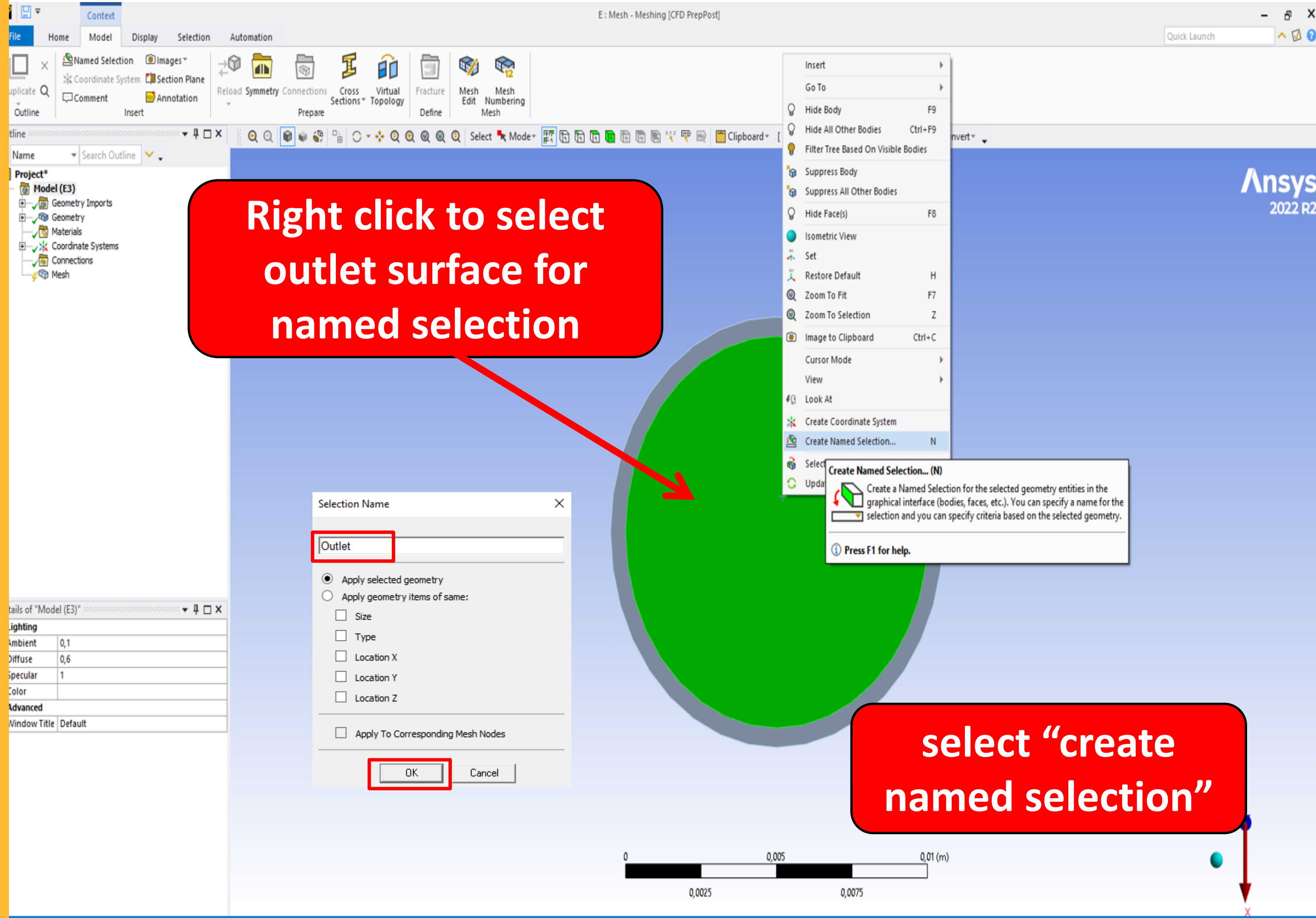
CREATING MESH



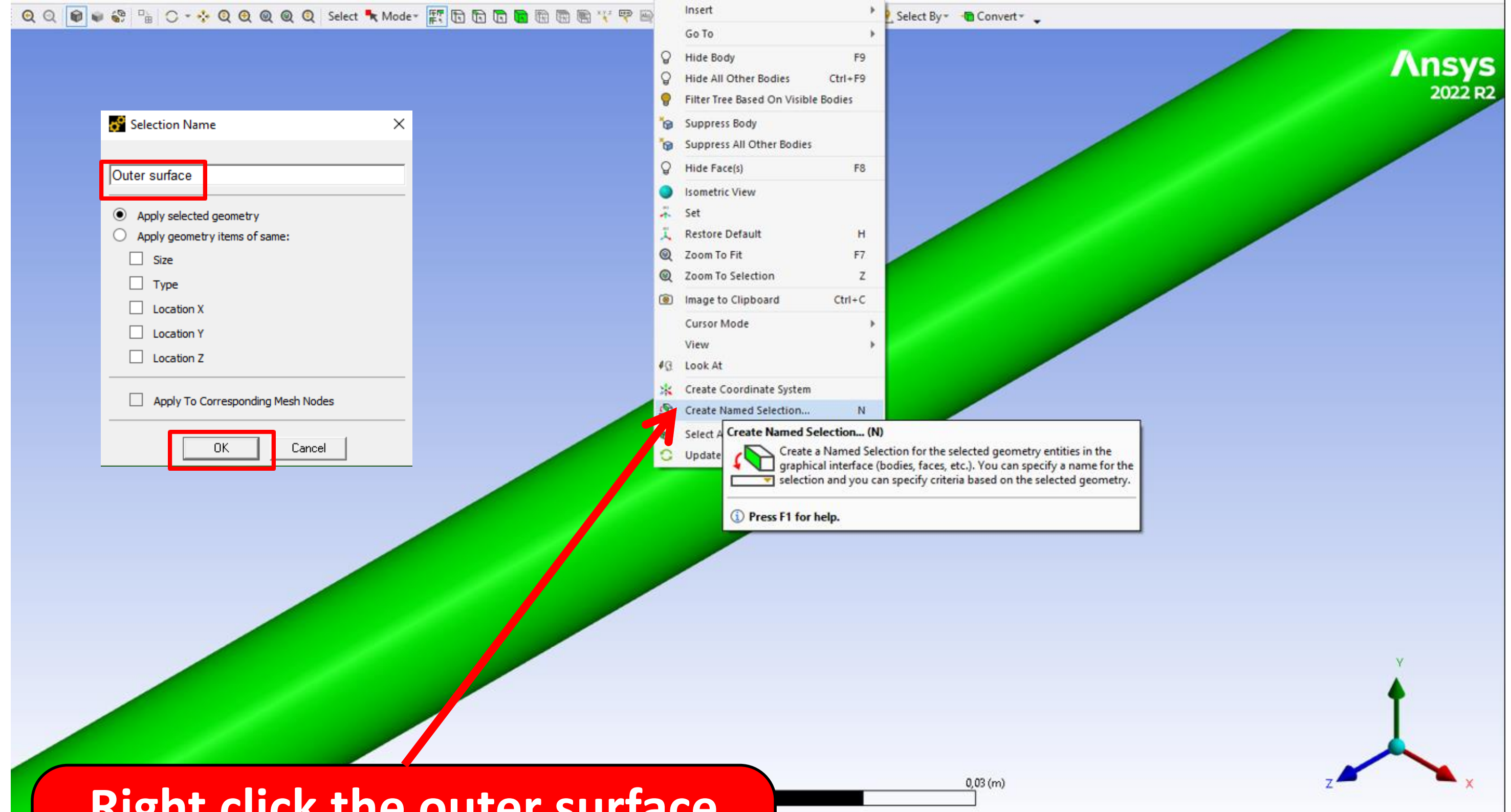
CREATING NAMED SELECTION



CREATE NAMED SELECTION

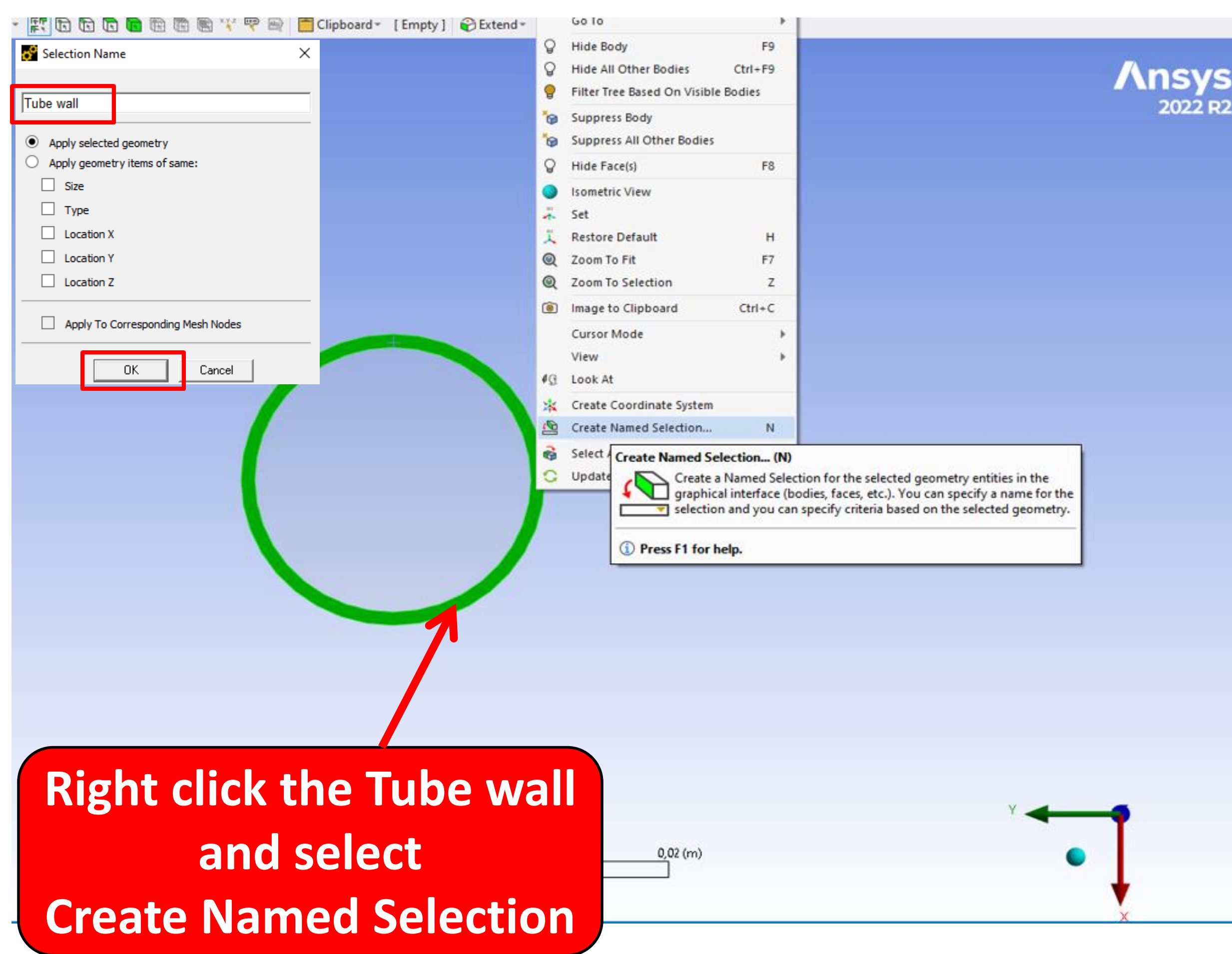


CREATE NAMED SELECTION



Right click the outer surface
and select
“Create Named Selection”

CREATE NAMED SELECTION



CREATE STRUCTURED MESH

Outline

Name Search Outline

Project

- Model (B3)
 - Geometry Imports
 - Geometry
 - Materials
 - Coordinate Systems
 - Connections
 - Mesh**

Insert

- Update
- Generate Mesh
- Preview
- Show
- Create Pinch Controls
- Group All Similar Children
- Clear Generated Data
- Rename F2
- Start Recording

Method

- Sizing**
- Control
- Refinement
- Face
- Mesh
- Match
- Pinch
- Inflation
- Weld
- Mesh Edit
- Mesh Numbering
- Contact Match Group
- Contact Match
- Node Merge Group

Sizing

Control size-related settings such as element size, number of divisions along an edge, use of sphere or body of influence, minimum size, etc.

Press F1 for help.

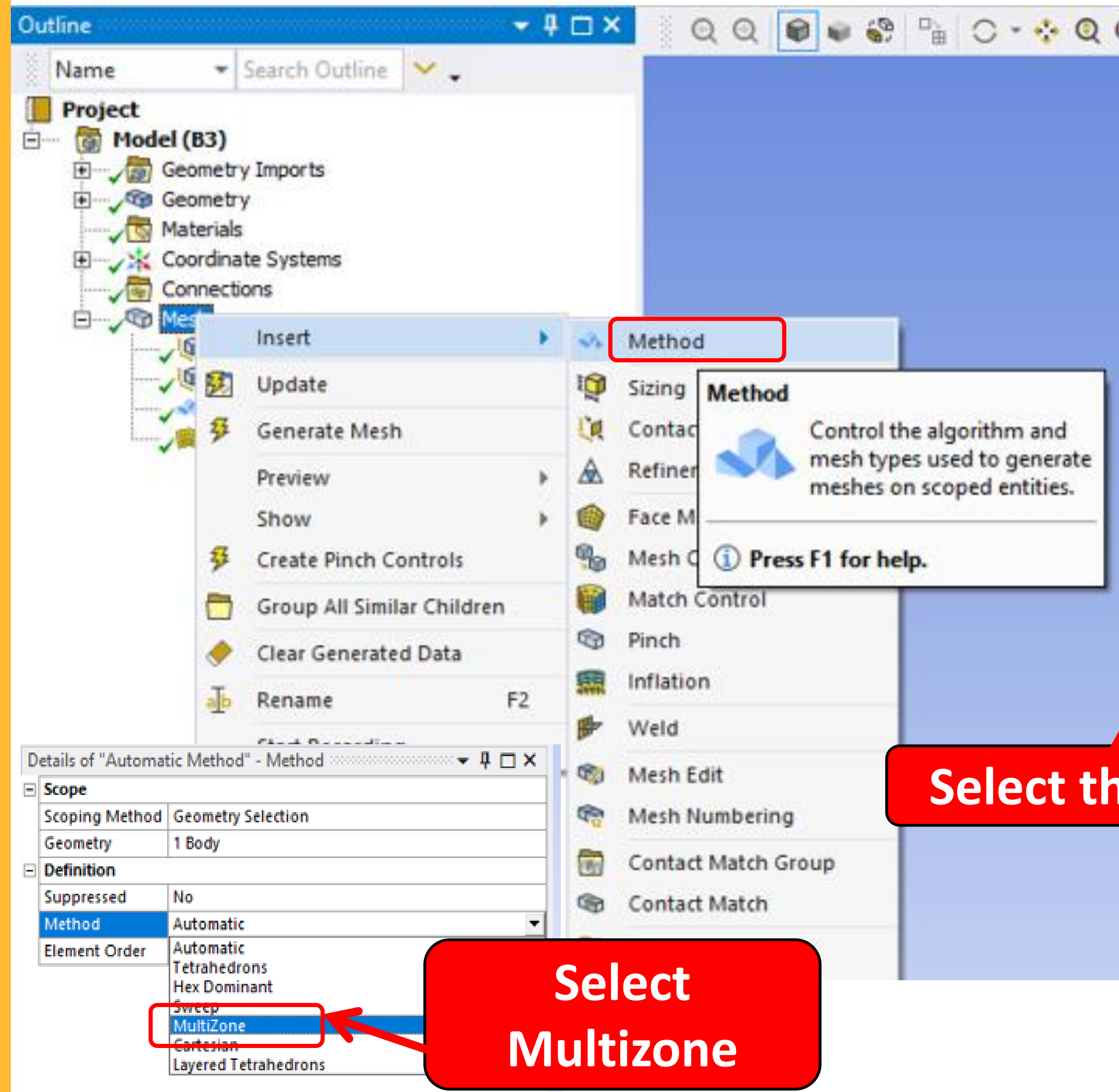
Details of "Edge Sizing" - Sizing

Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
<input type="checkbox"/> Number of Divisions	45
Advanced	
Behavior	Soft
<input type="checkbox"/> Growth Rate	Default (1,2)
Capture Curvature	No
Capture Proximity	No
Bias Type	No Bias

To create a structured mesh, insert sizing and select the edge of the circle at the inlet and outlet

Set Number of division to 45

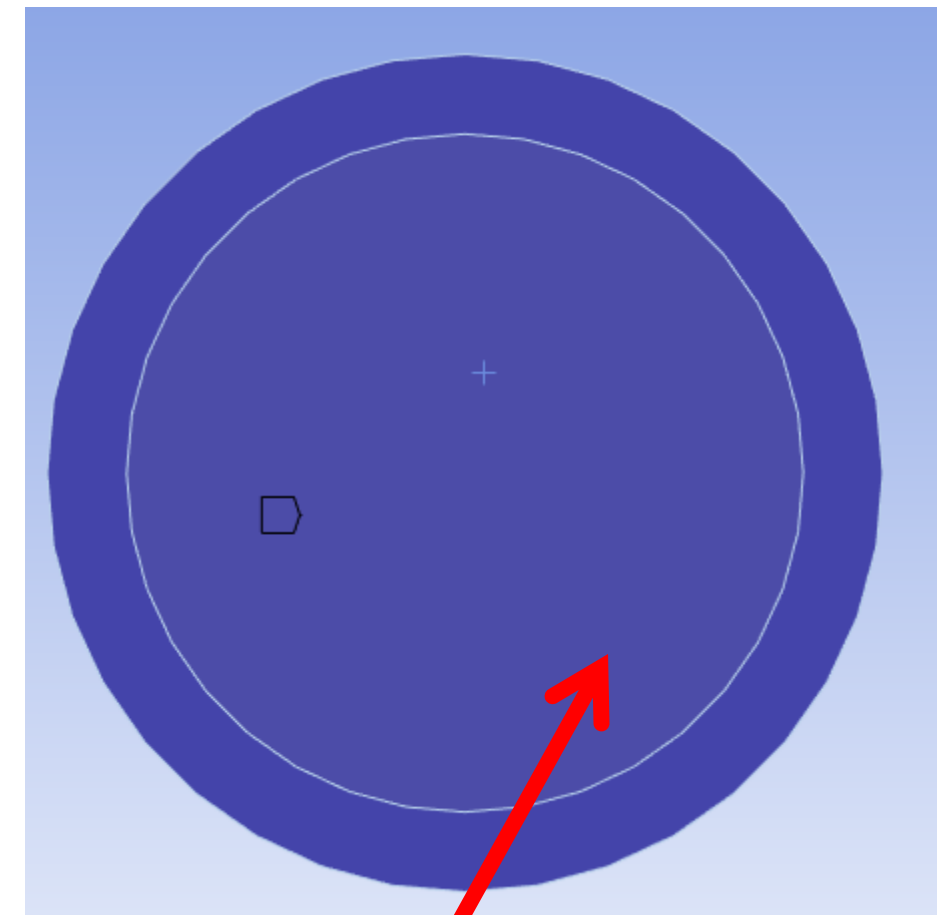
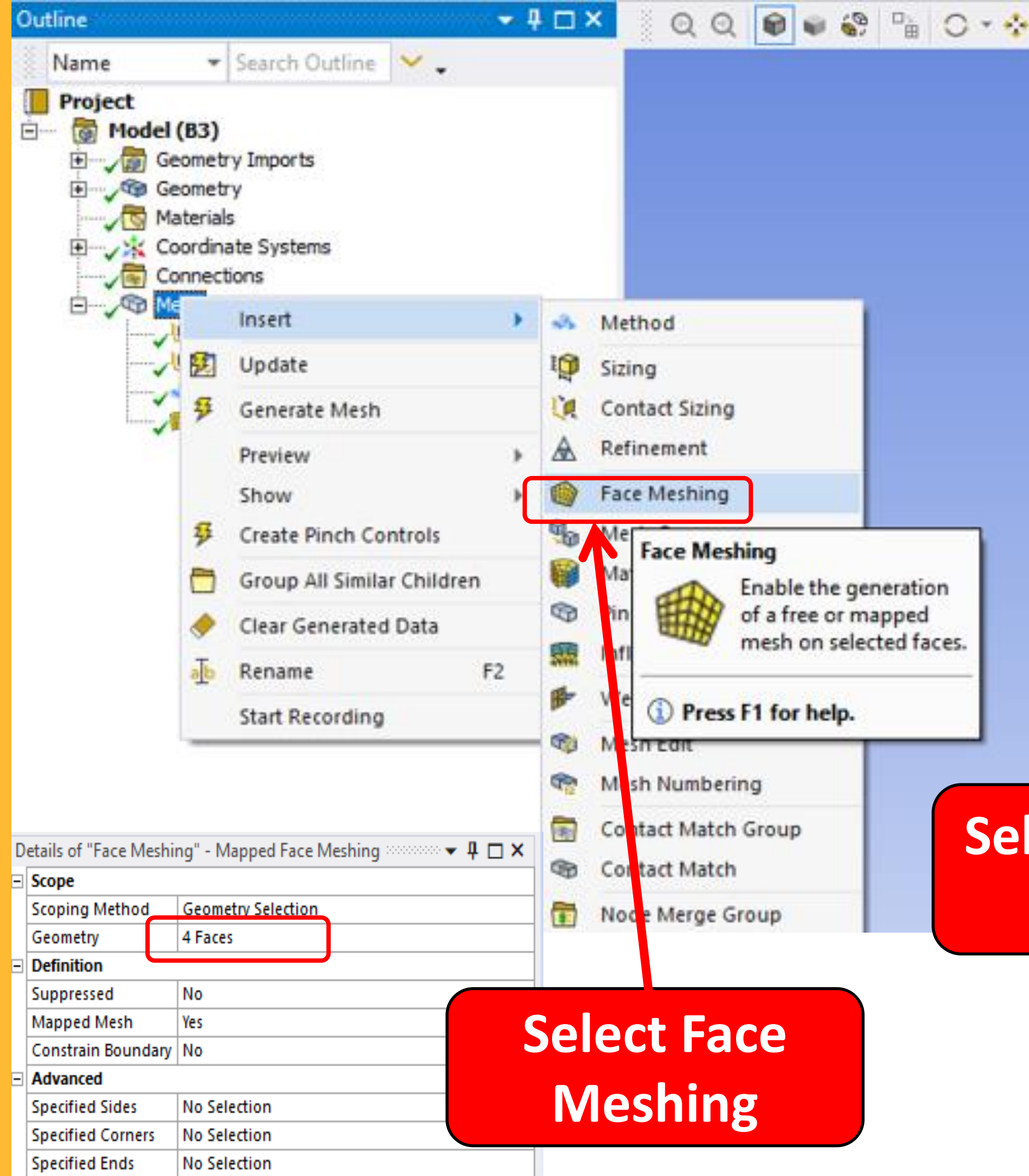
CREATE STRUCTURED MESH



Select the fluid body

Select
Multizone

CREATE STRUCTURED MESH



Select the solid and
fluid surface

Select Face
Meshing

CREATE STRUCTURED MESH

Click "Generate" to have the required Mesh

Select Mesh and do the following

- Physics Preference to CFD
- Solver Preference to Fluent
- ▼ Element Size to 0.0003

The screenshot displays the ANSYS Meshing software interface. The top toolbar shows the 'Generate' button (lightning bolt icon) highlighted. The 'Outline' panel on the left shows the 'Mesh' object selected under 'Model (B3)'. The 'Details of Mesh' panel at the bottom shows the following settings:

Details of "Mesh"	
Display	
Display Style	Use ...
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
<input type="checkbox"/> Element Size	3,e-0...
Export Format	Stan...
Export Preview Surface Mesh	No
Sizing	
Quality	
Inflation	
Advanced	

The main workspace shows a circular domain with a structured mesh. A red arrow points from the 'Generate' button to the mesh. Another red arrow points from the 'Details of Mesh' panel to the list of settings.

SUMMARY

In this second tutorial, you were able to

- 01** Create named selections for the boundaries of the geometry
- 02** Mesh the circular tube geometry