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

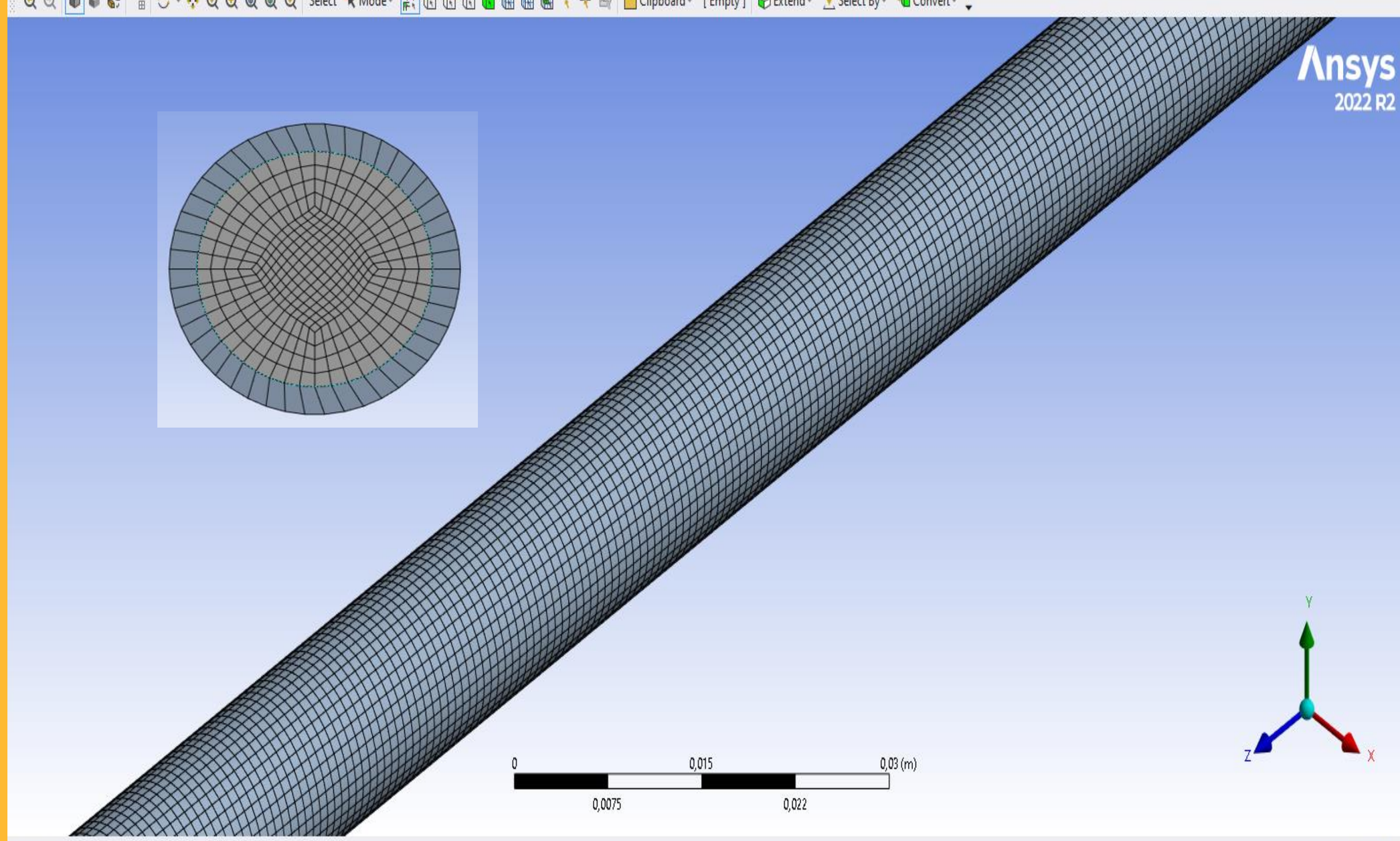
01 Create named selections on the boundaries of the geometry

02 Mesh the circular tube geometry

MESHING THE GEOMETRY

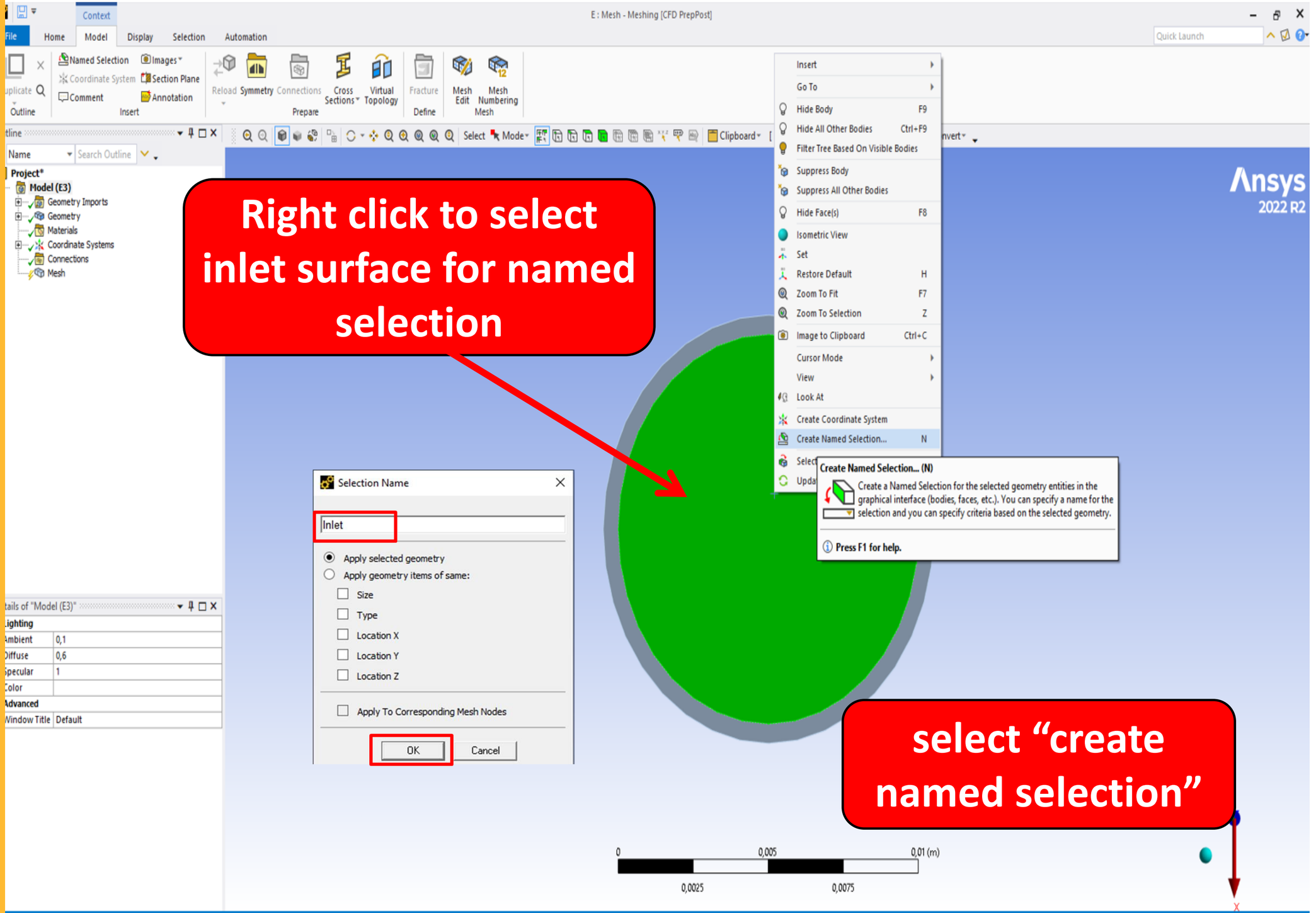
CREATING

MESH



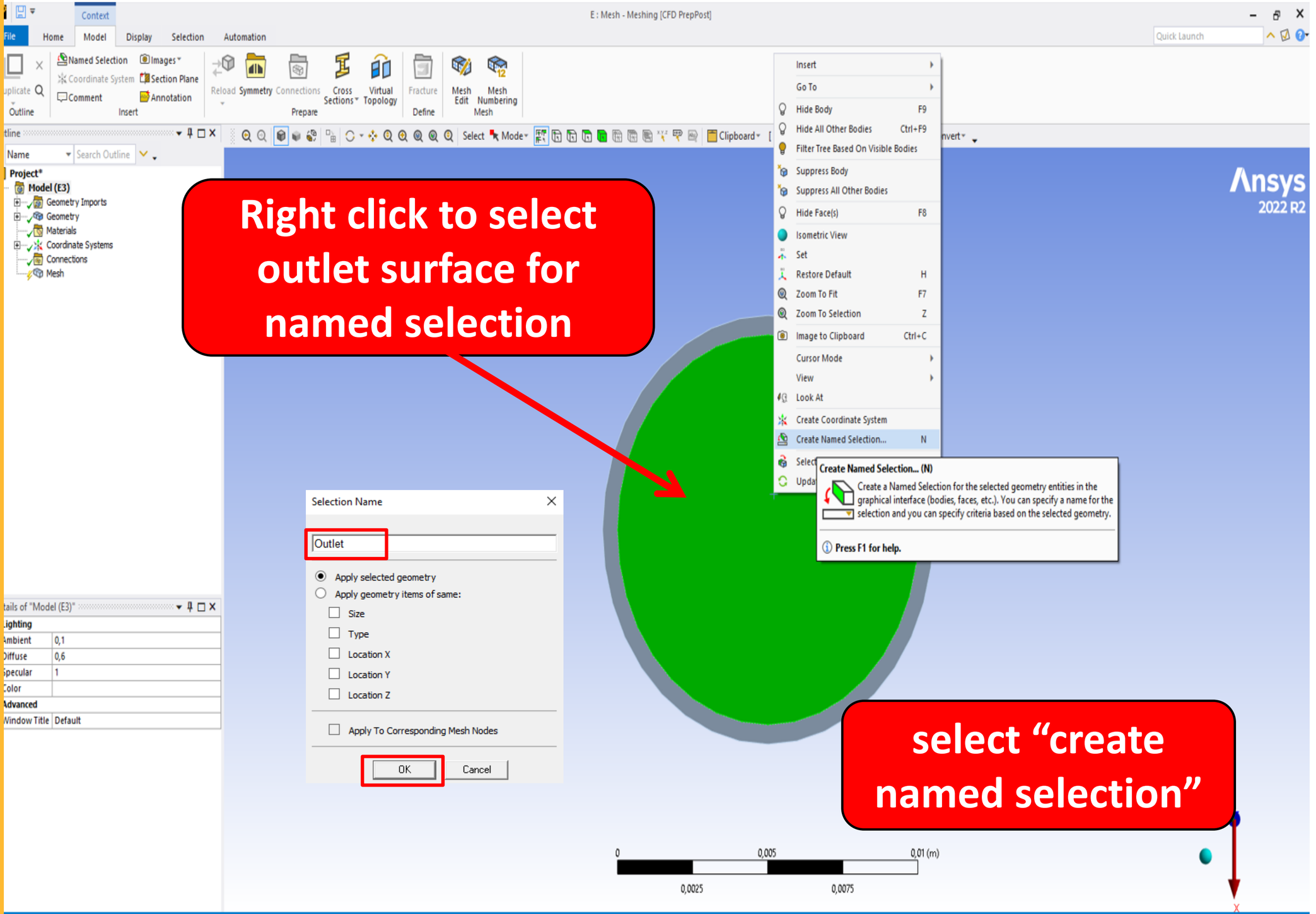
CREATING

NAMED SELECTION



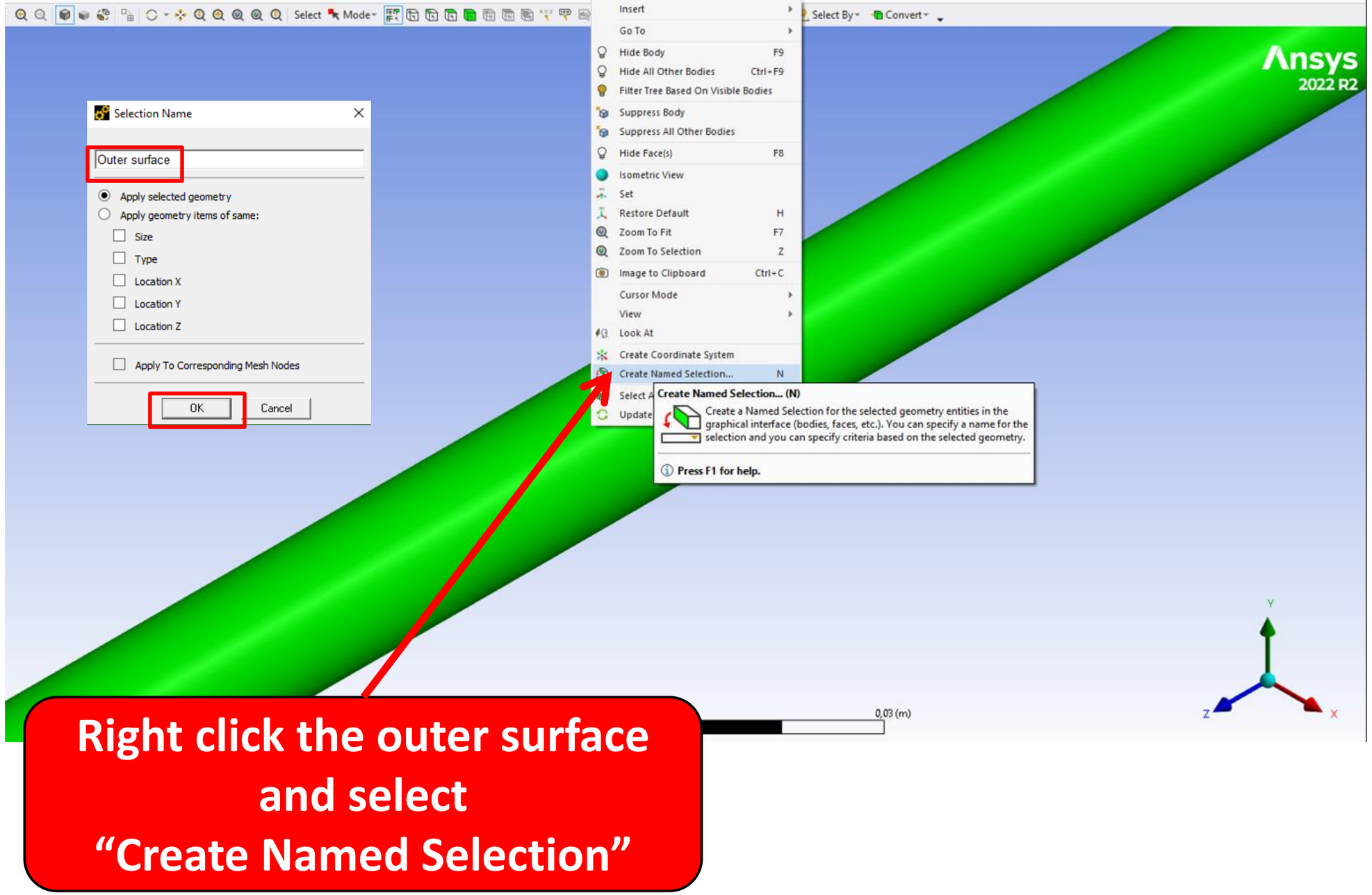
CREATE

NAMED SELECTION



CREATE

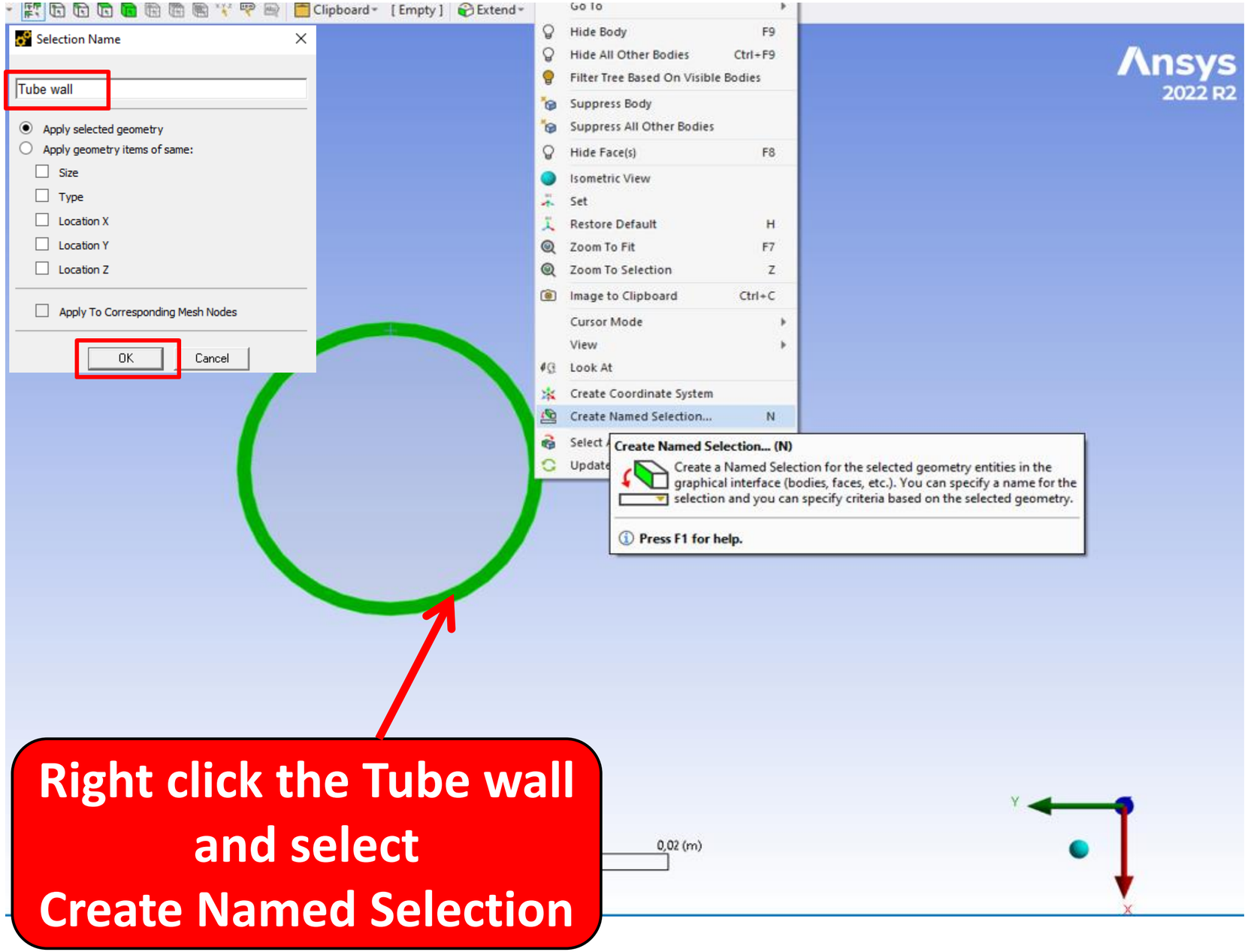
NAMED SELECTION



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

CREATE

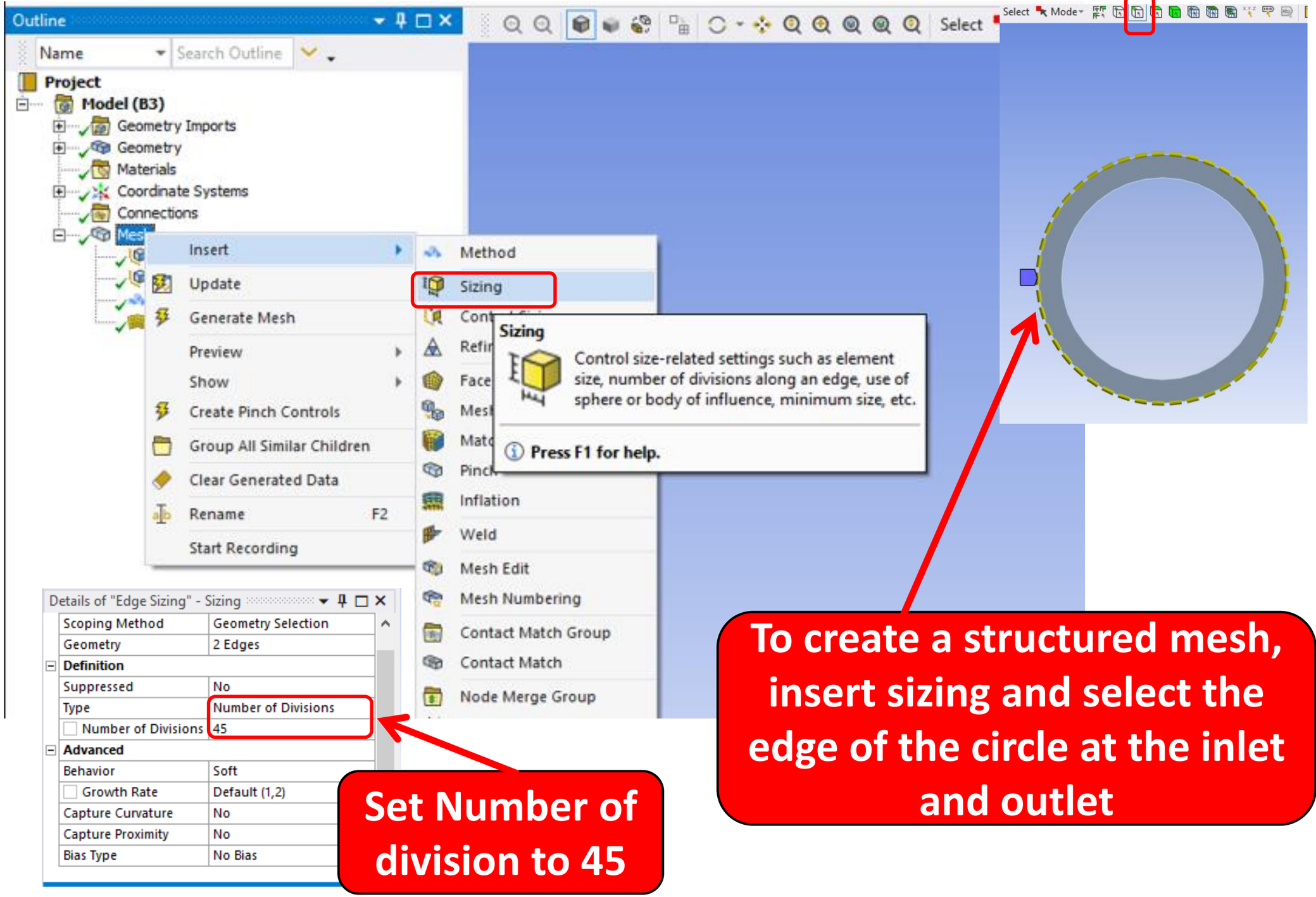
NAMED SELECTION



Right click the Tube wall and select Create Named Selection

CREATE

STRUCTURED MESH

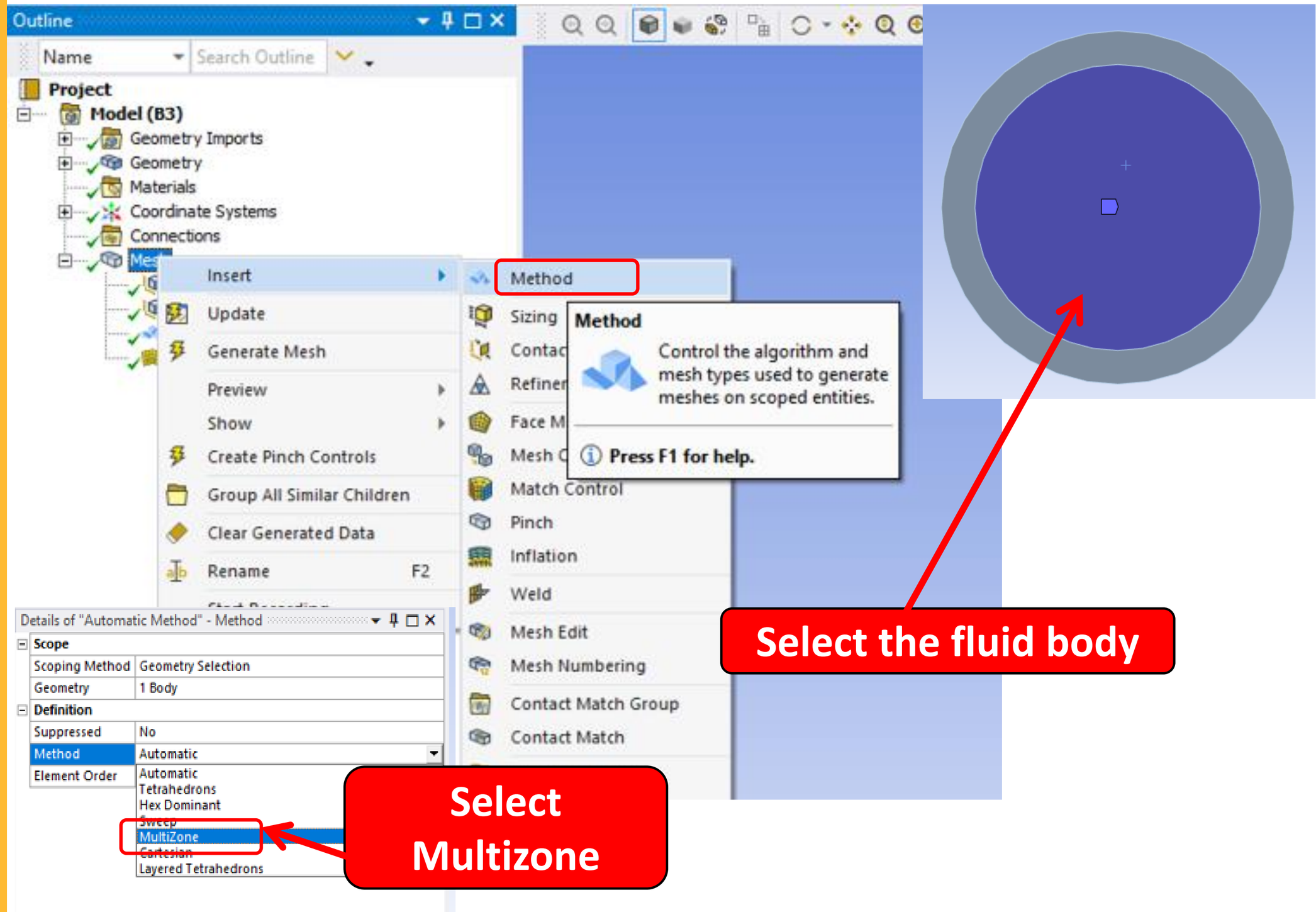


To create a structured mesh, insert sizing and select the edge of the circle at the inlet and outlet

Set Number of division to 45

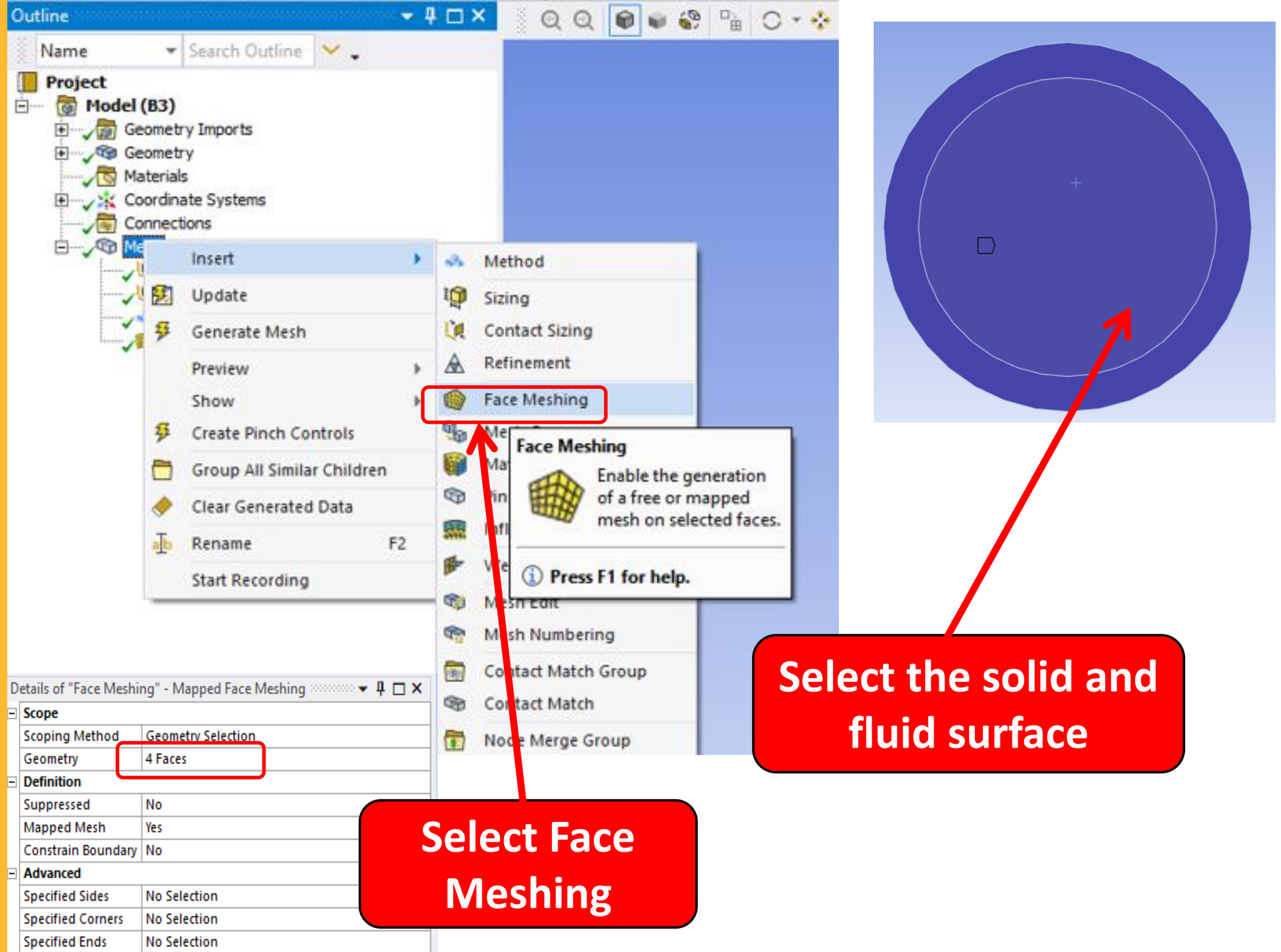
CREATE

STRUCTURED MESH



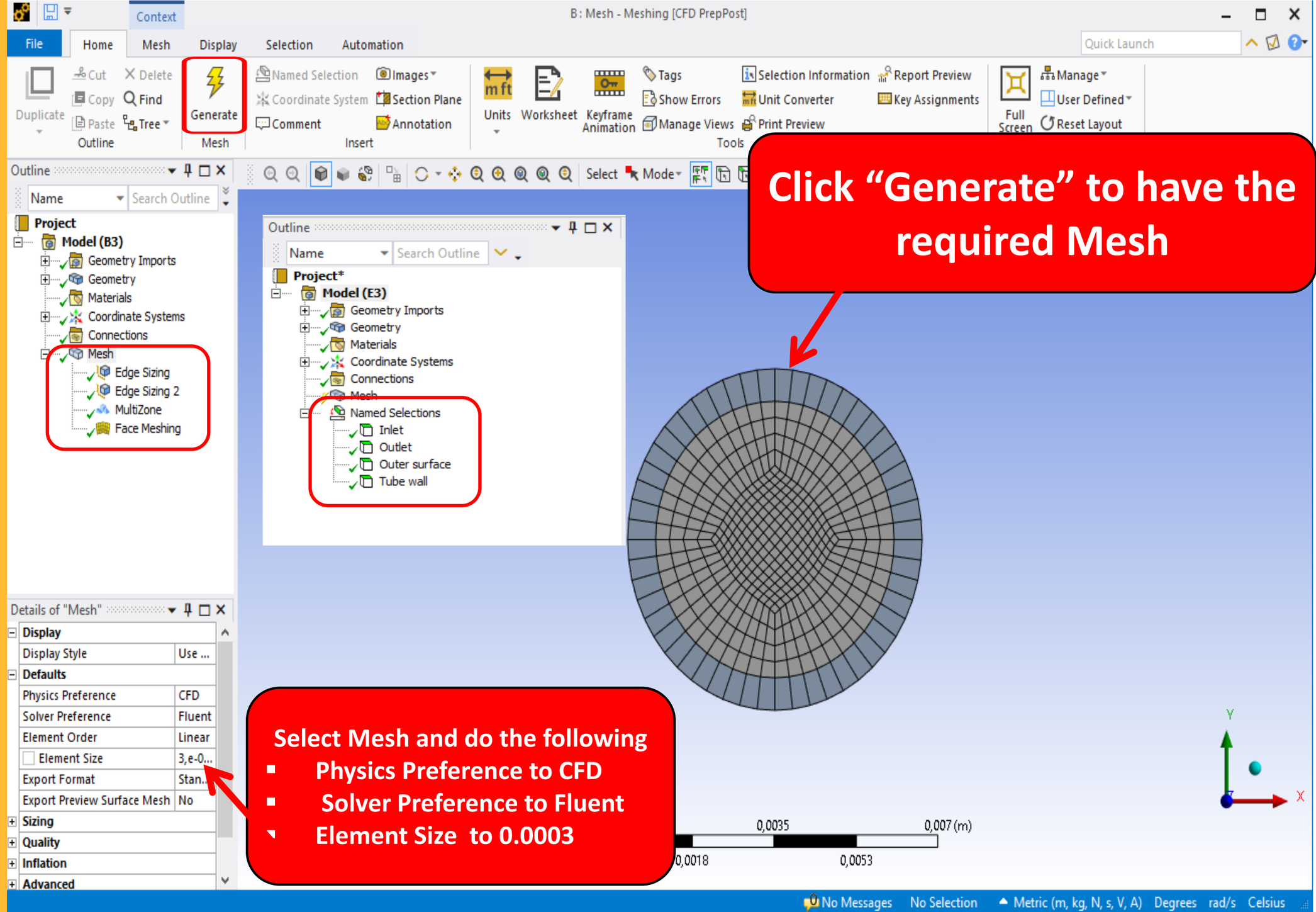
CREATE

STRUCTURED MESH



CREATE

STRUCTURED MESH



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