

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

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

FLUID FLOW MODELING

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

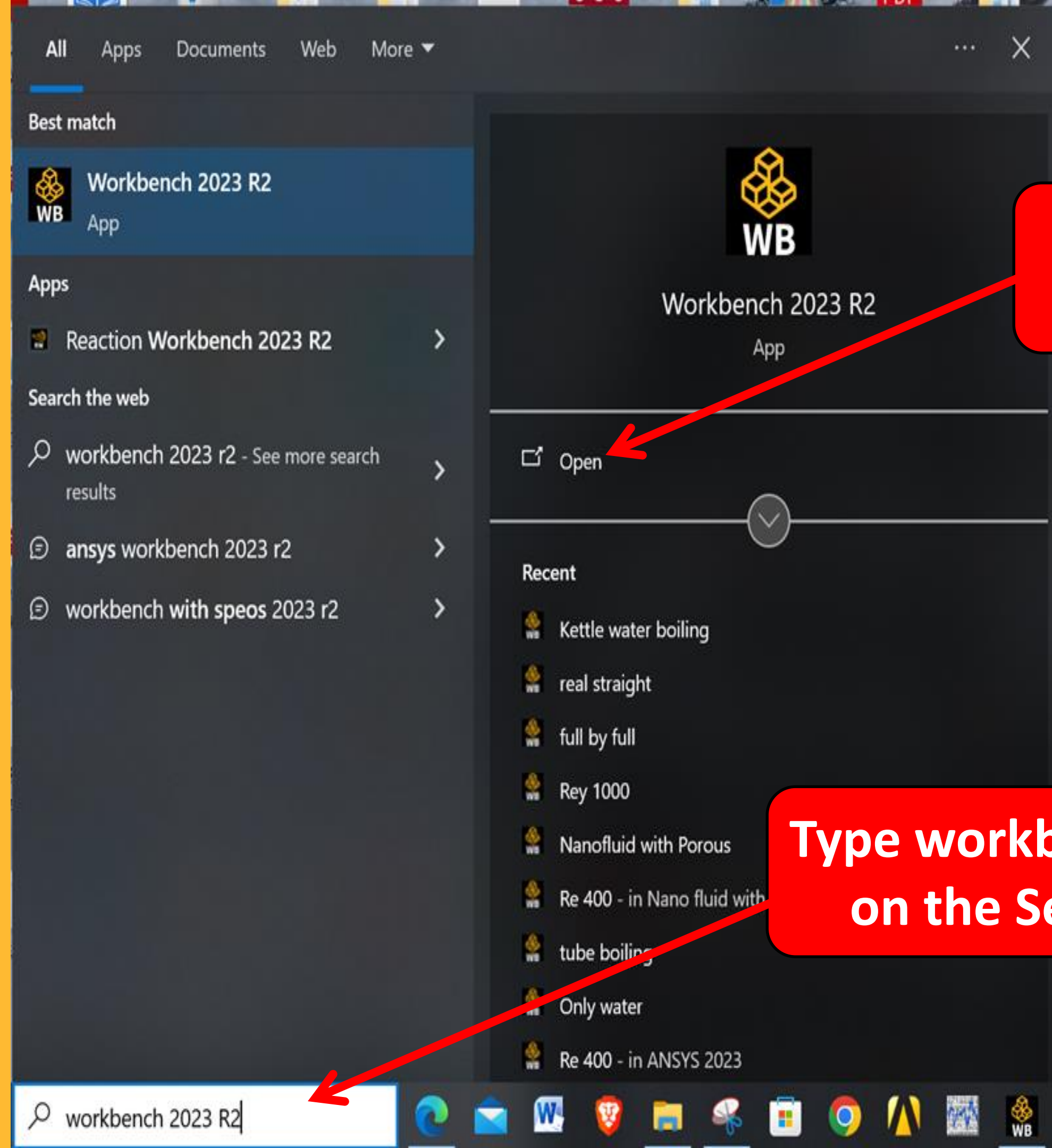
- 01 Start up your ANSYS Workbench**
- 02 Create the geometry of a Circular Tube**

STARTING UP ANSYS WORKBENCH

Starting Up

ANSYS

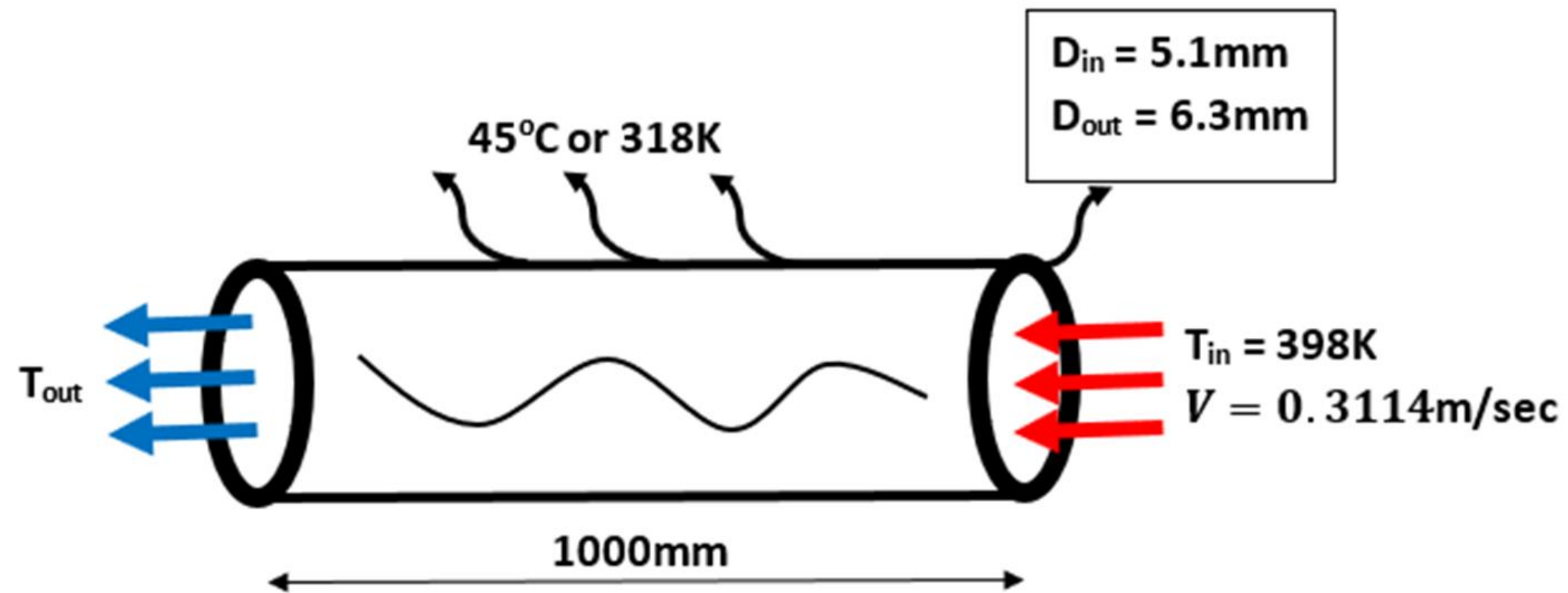
Workbench



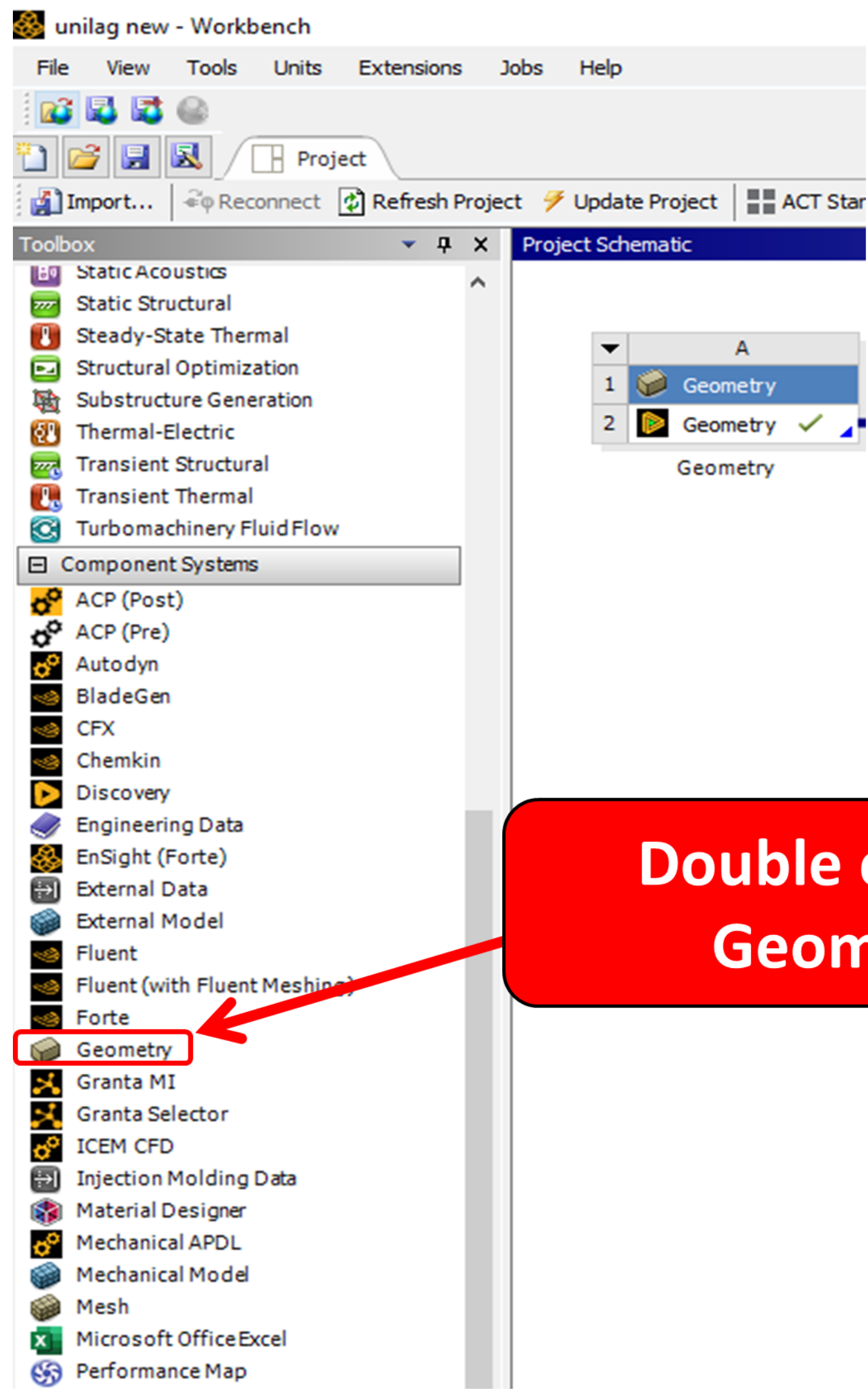
Click Open

**Type workbench name
on the Search bar**

CREATING THE GEOMETRY

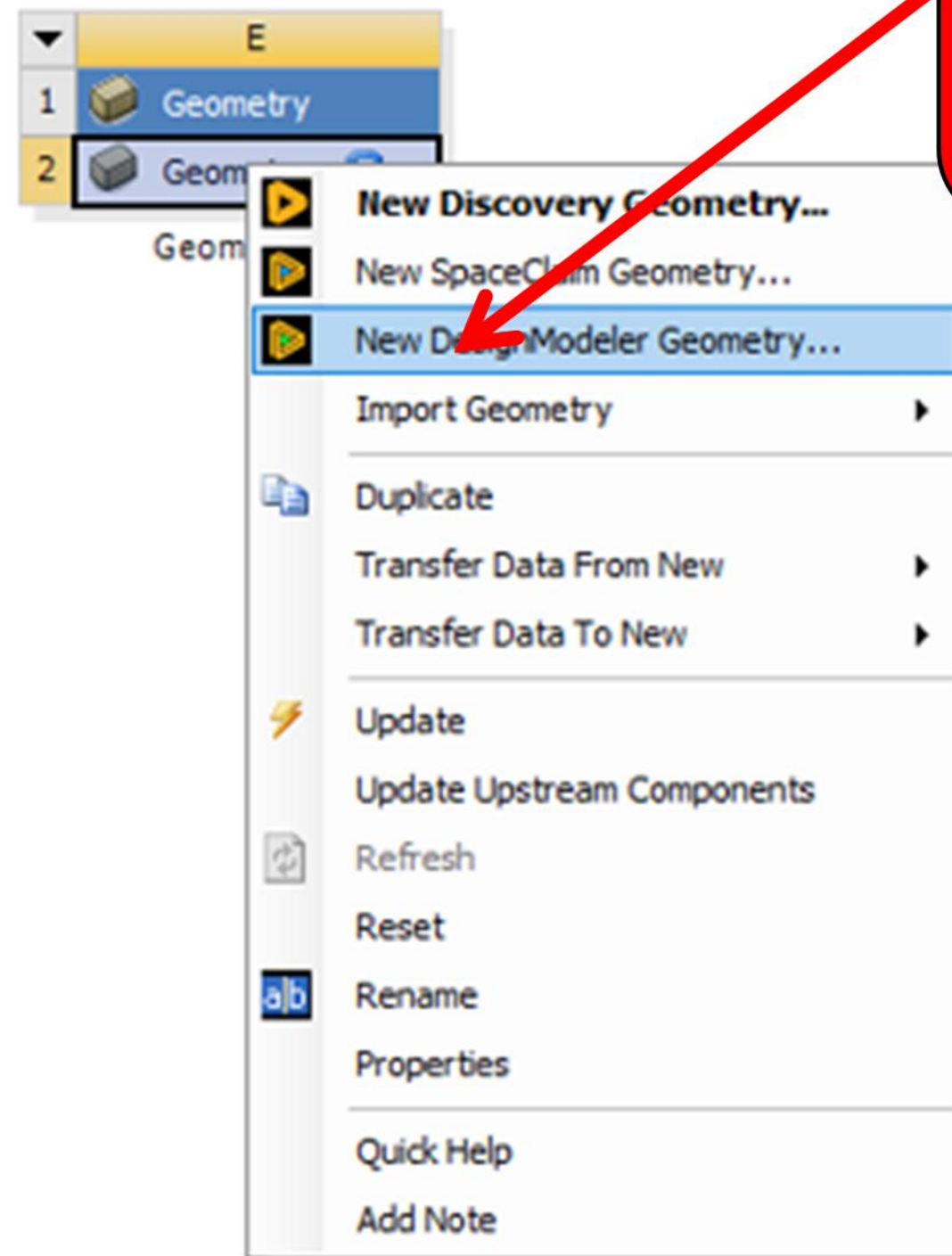


Creating Geometry



**Double click on
Geometry**

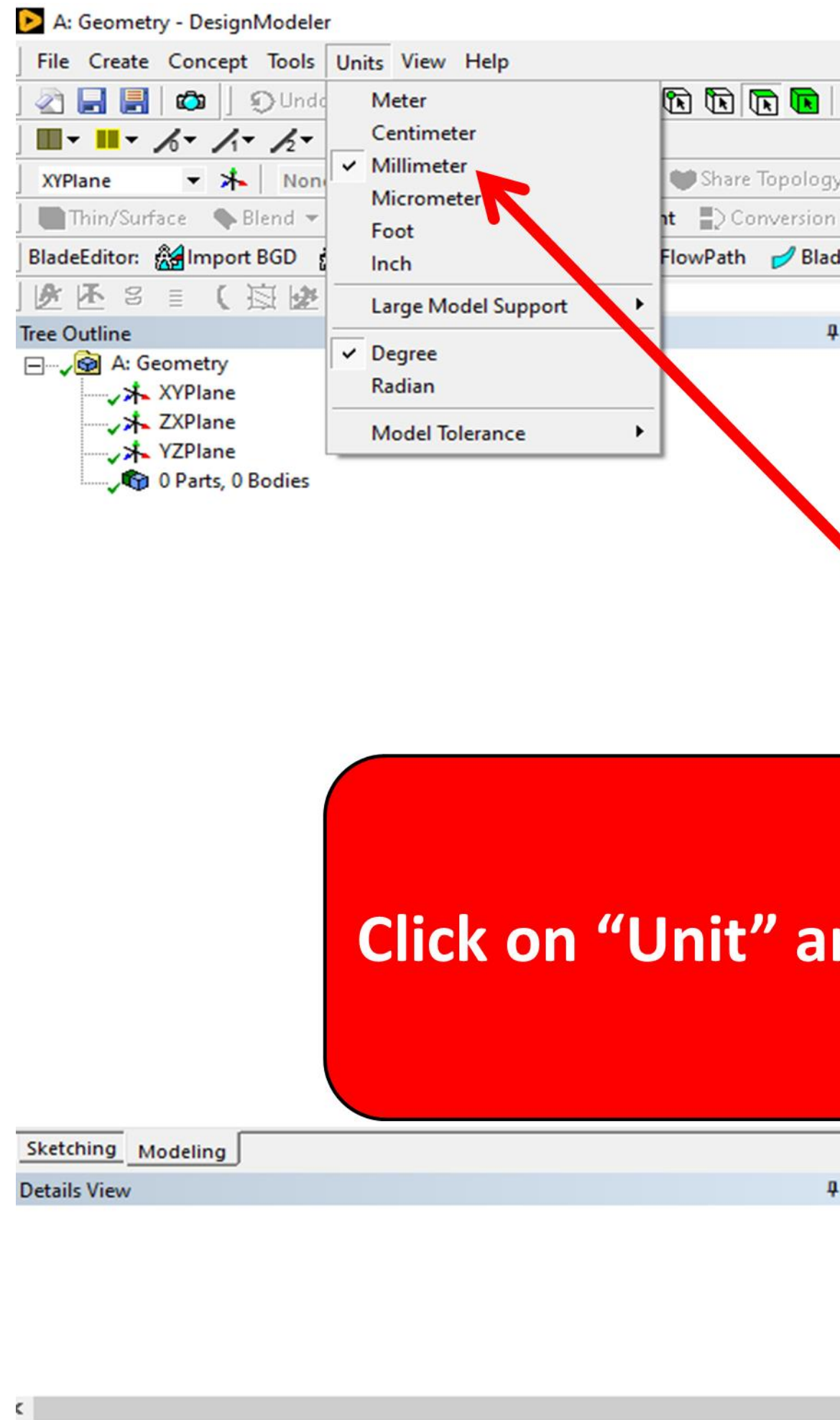
Creating Geometry



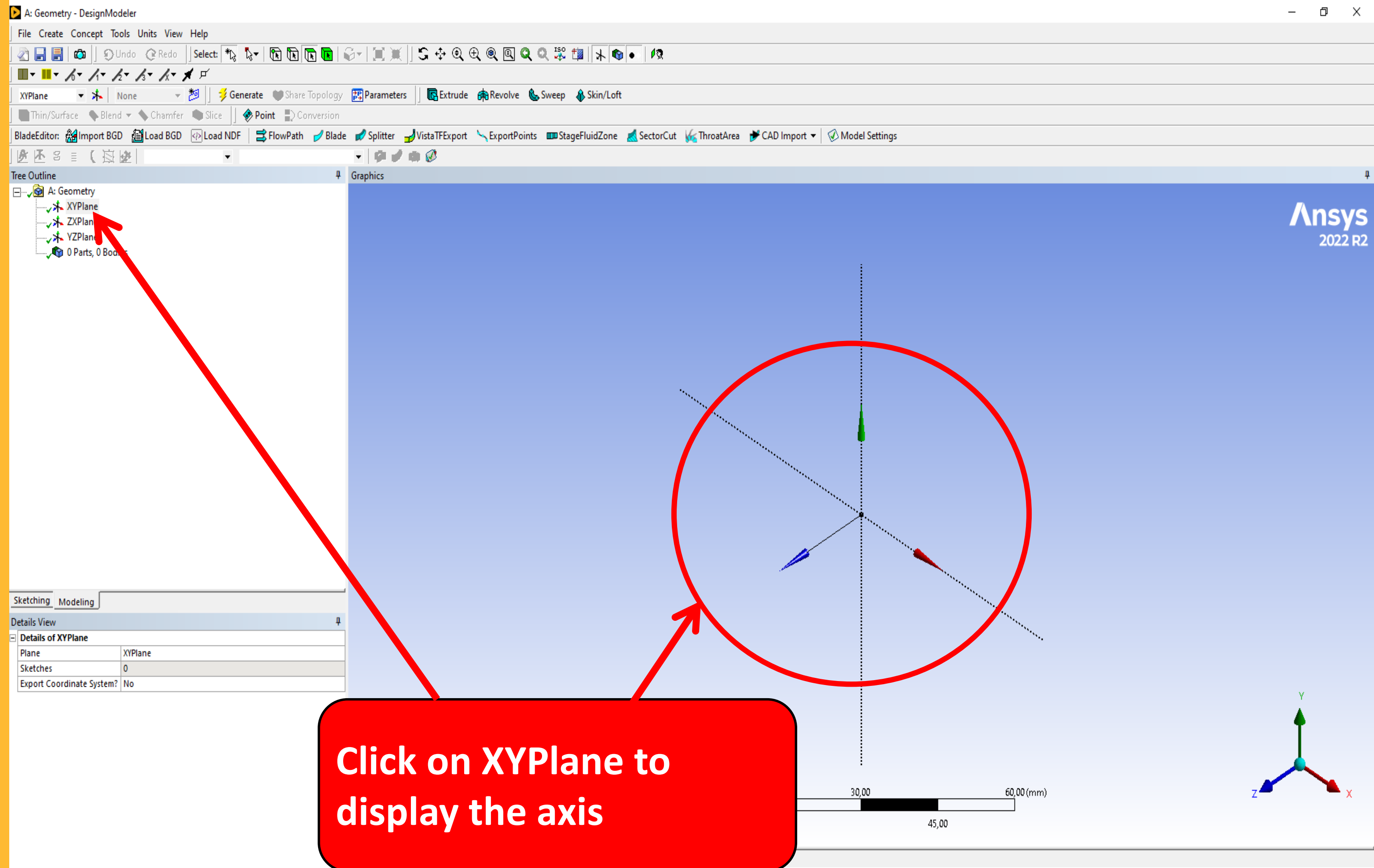
1. Right Click on “Geometry”
2. Click on “New Design Modeler”

Creating

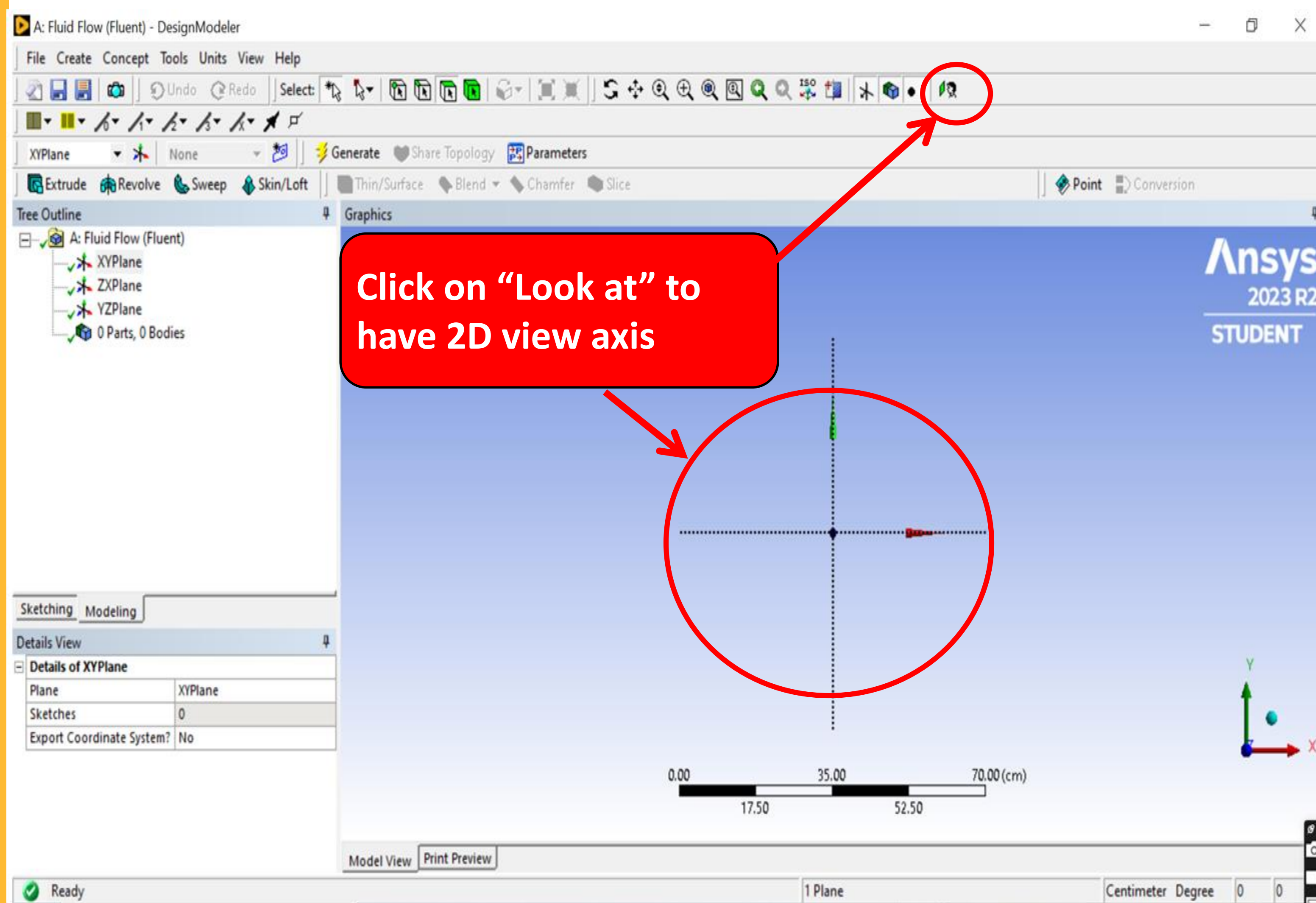
Geometry



Creating Geometry

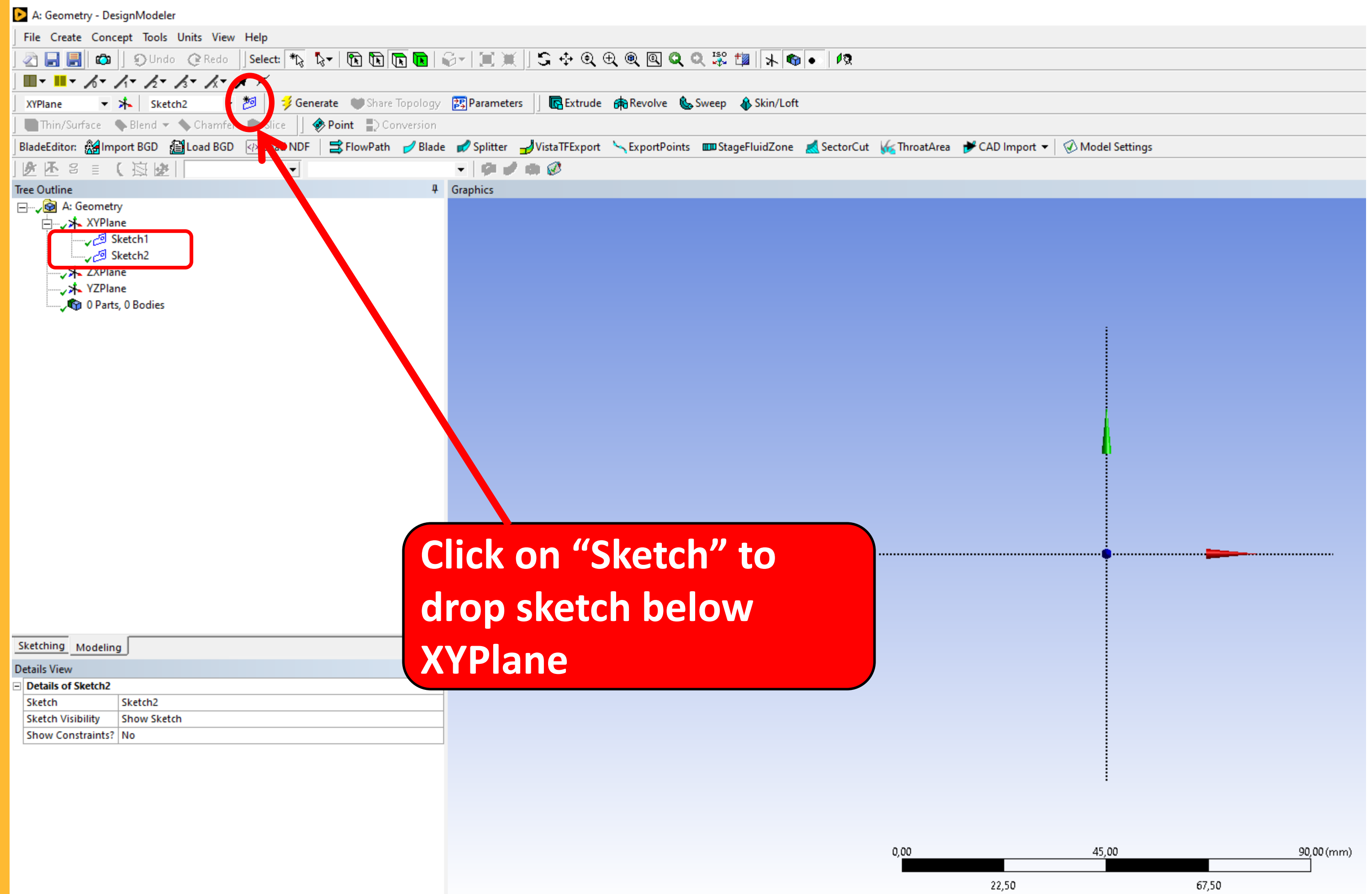


Creating Geometry

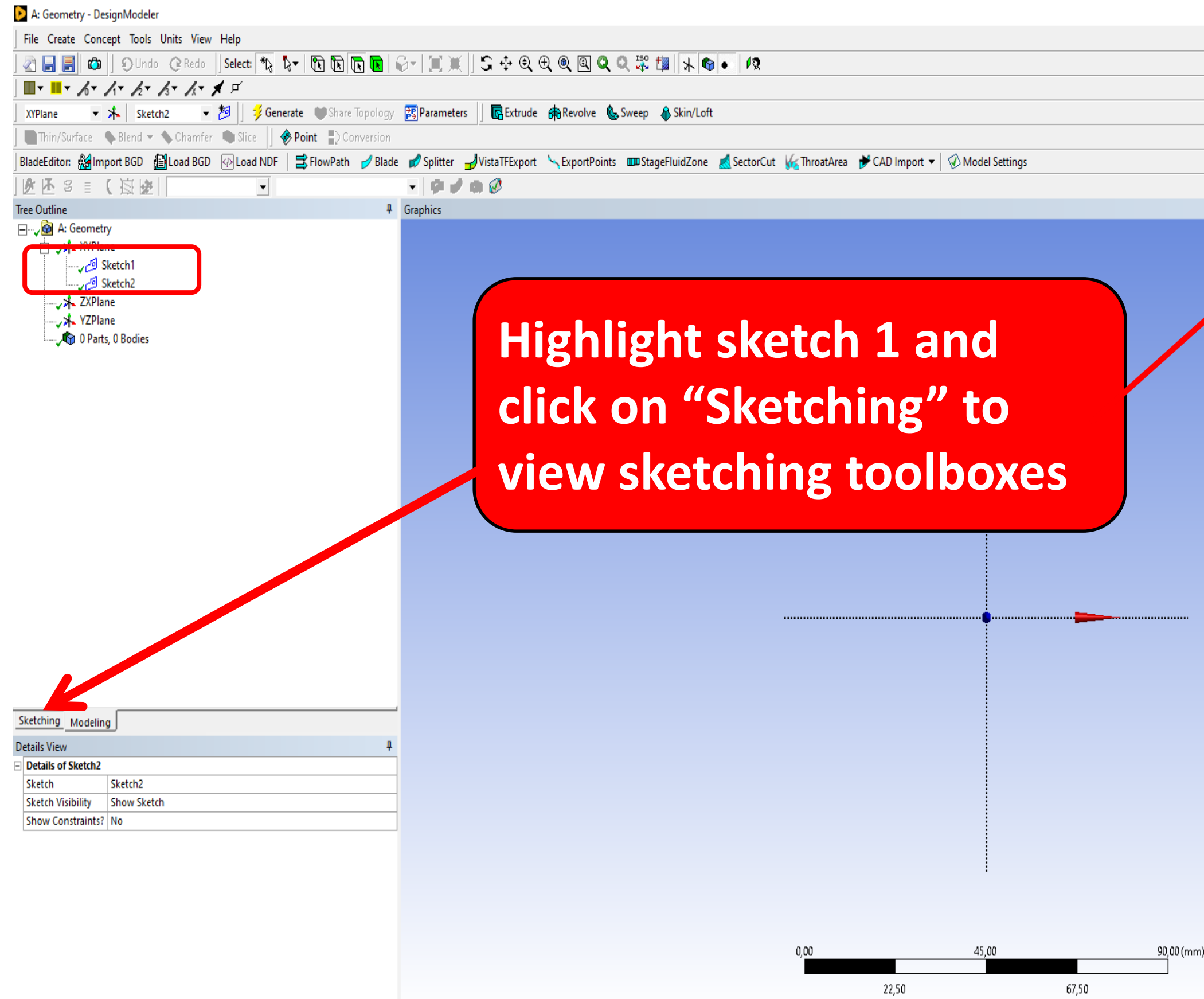


Creating

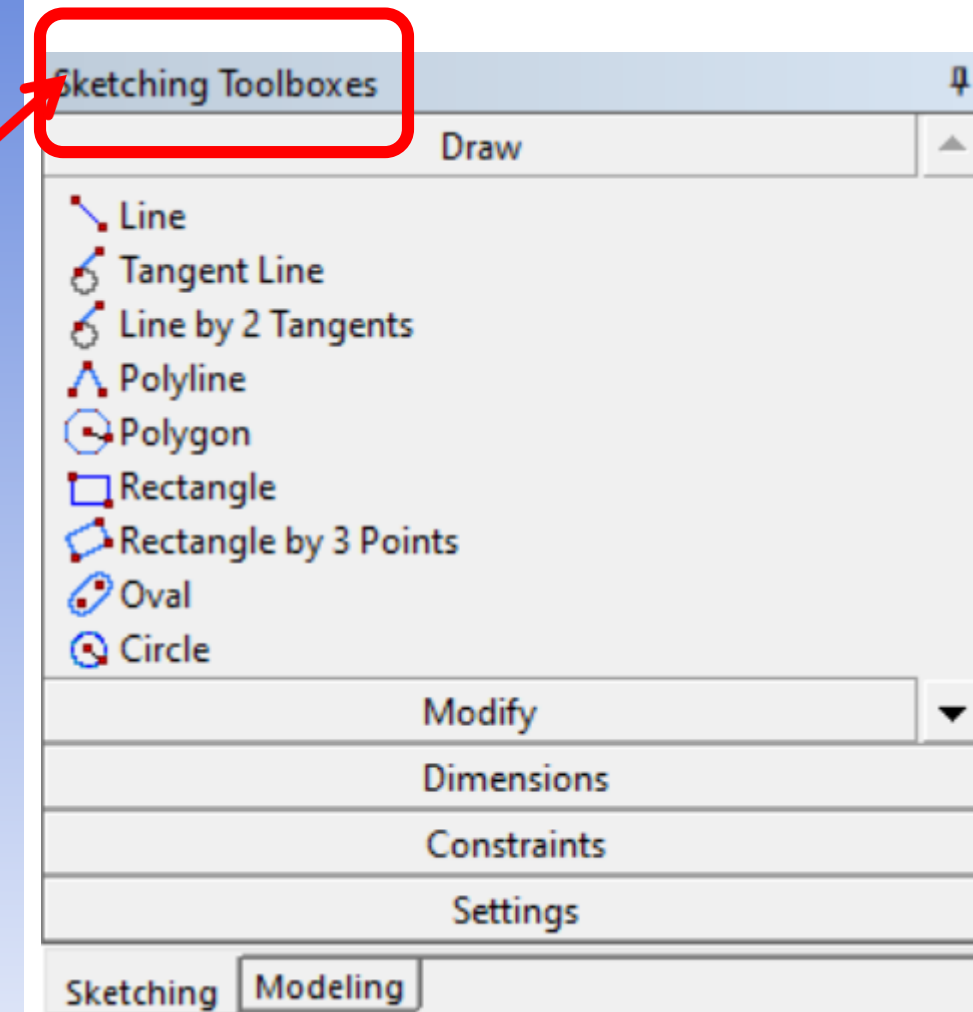
Geometry



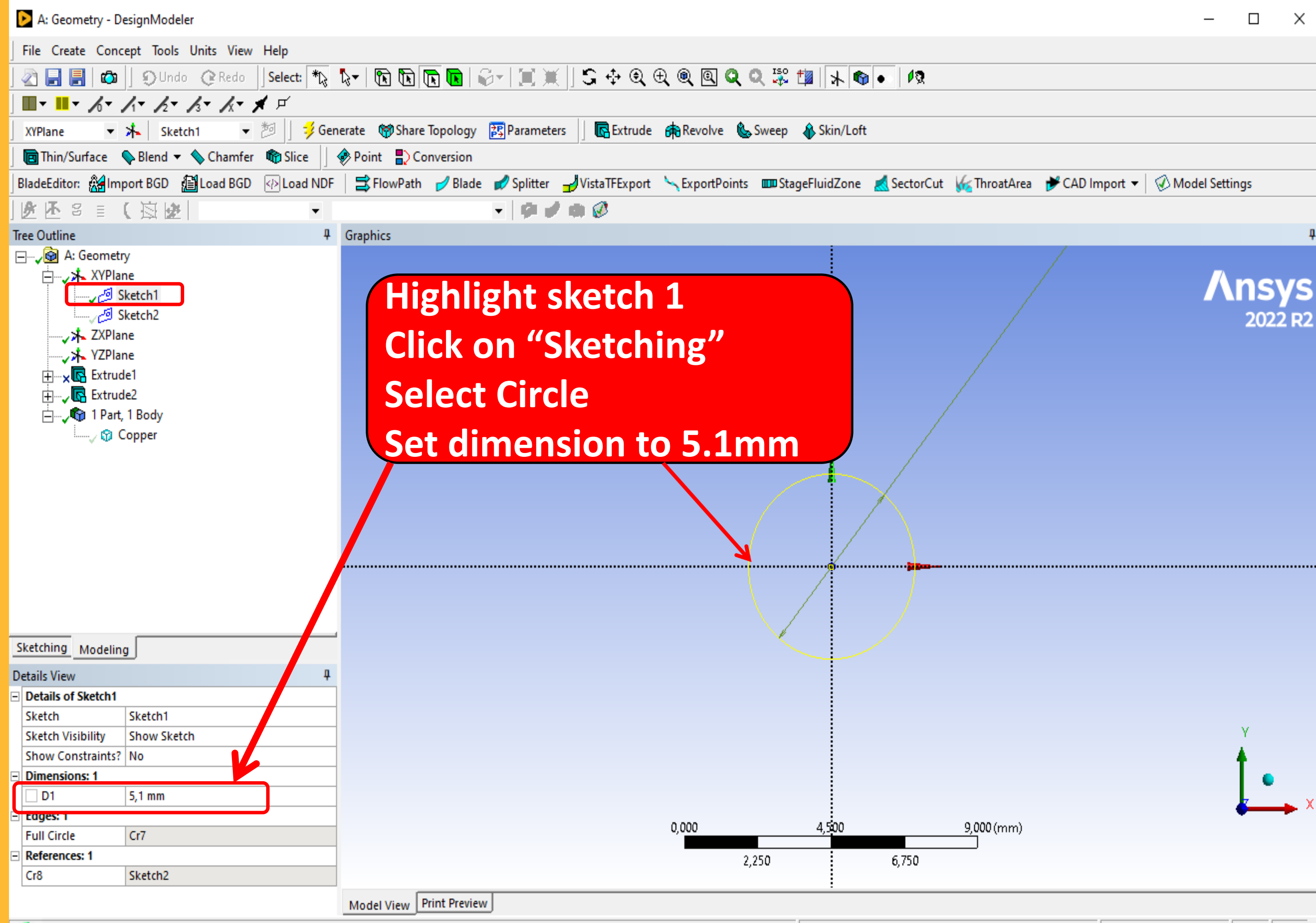
Creating Geometry



Highlight sketch 1 and
click on “Sketching” to
view sketching toolboxes

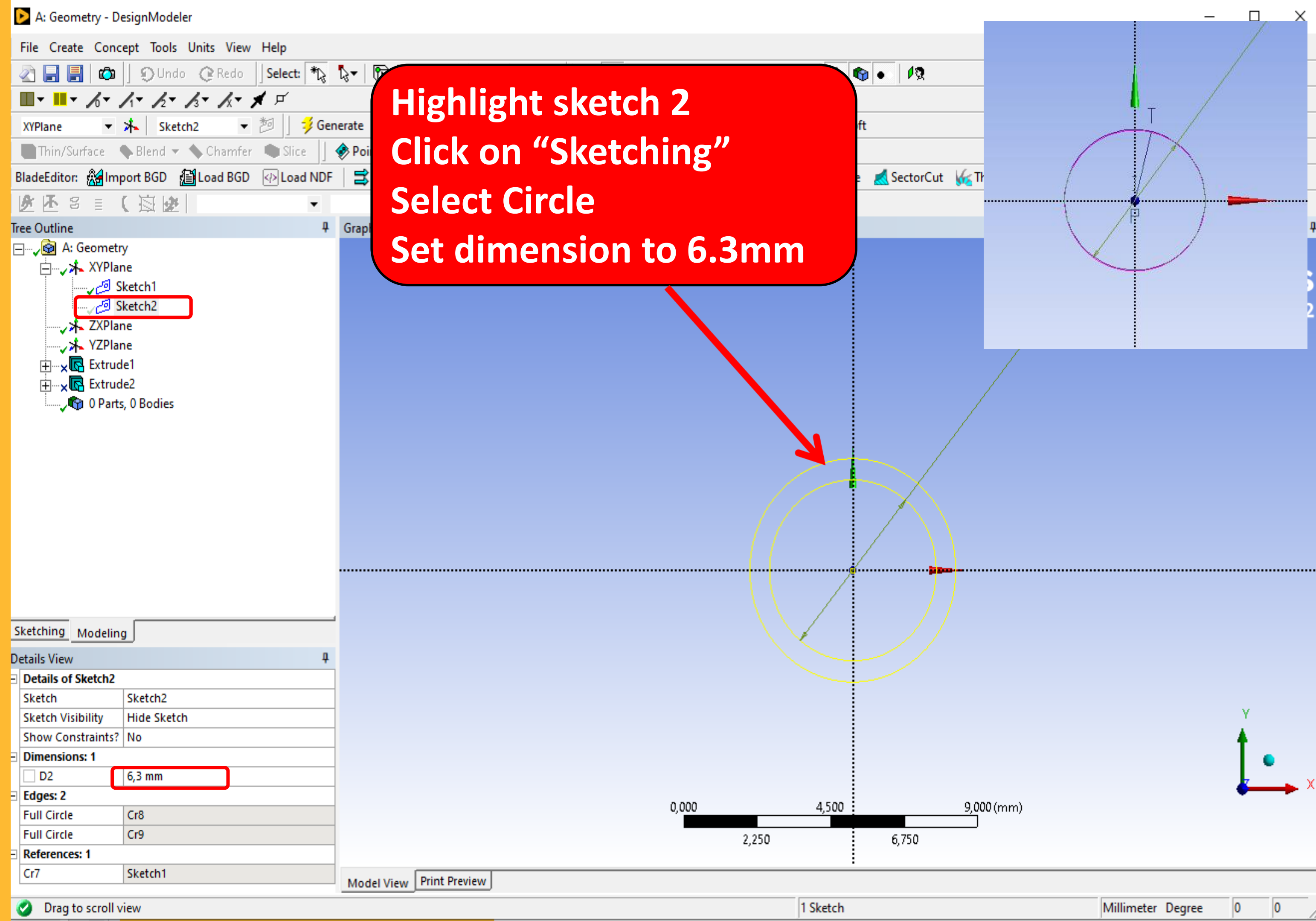


Creating Geometry

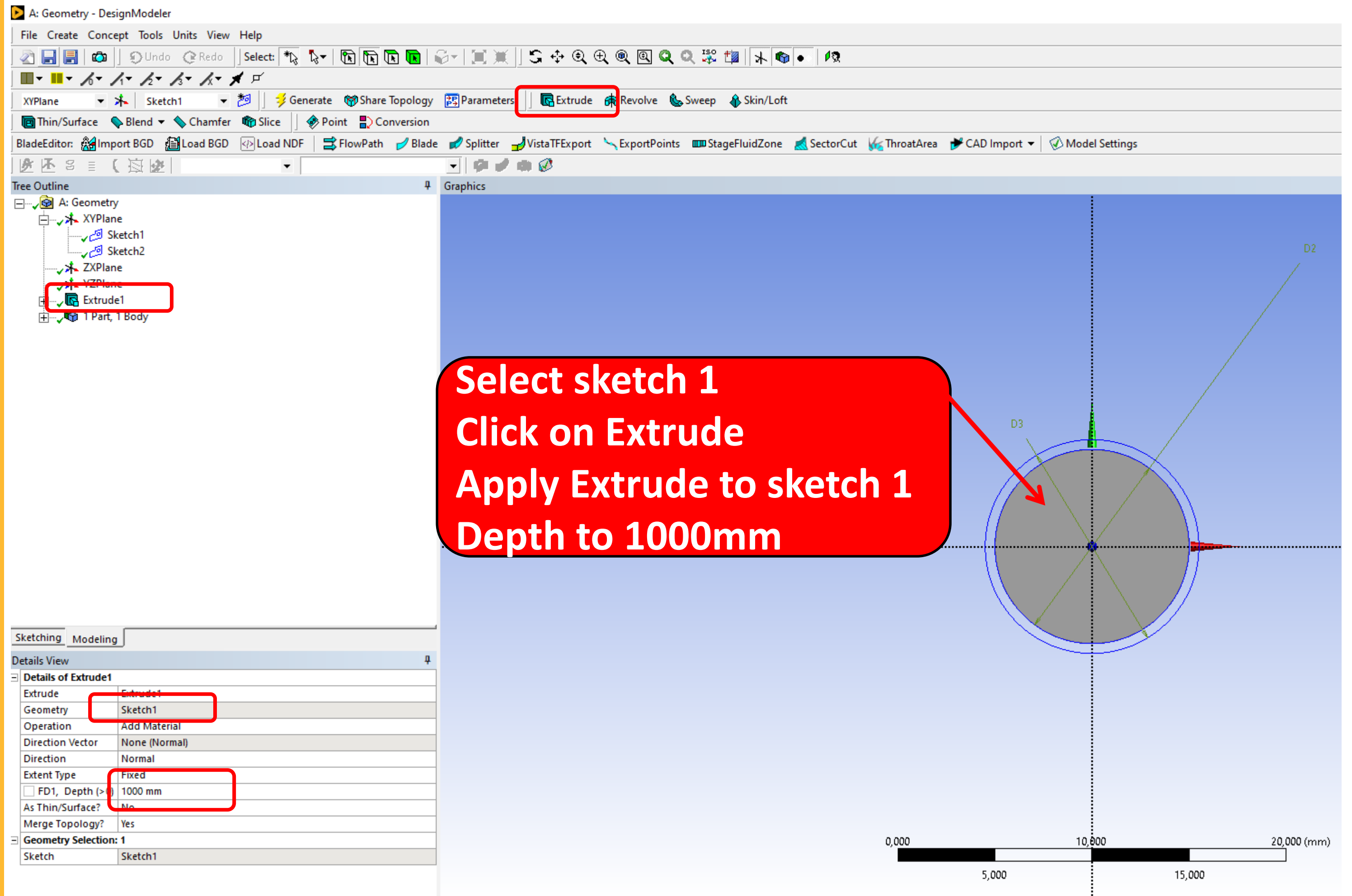


Creating

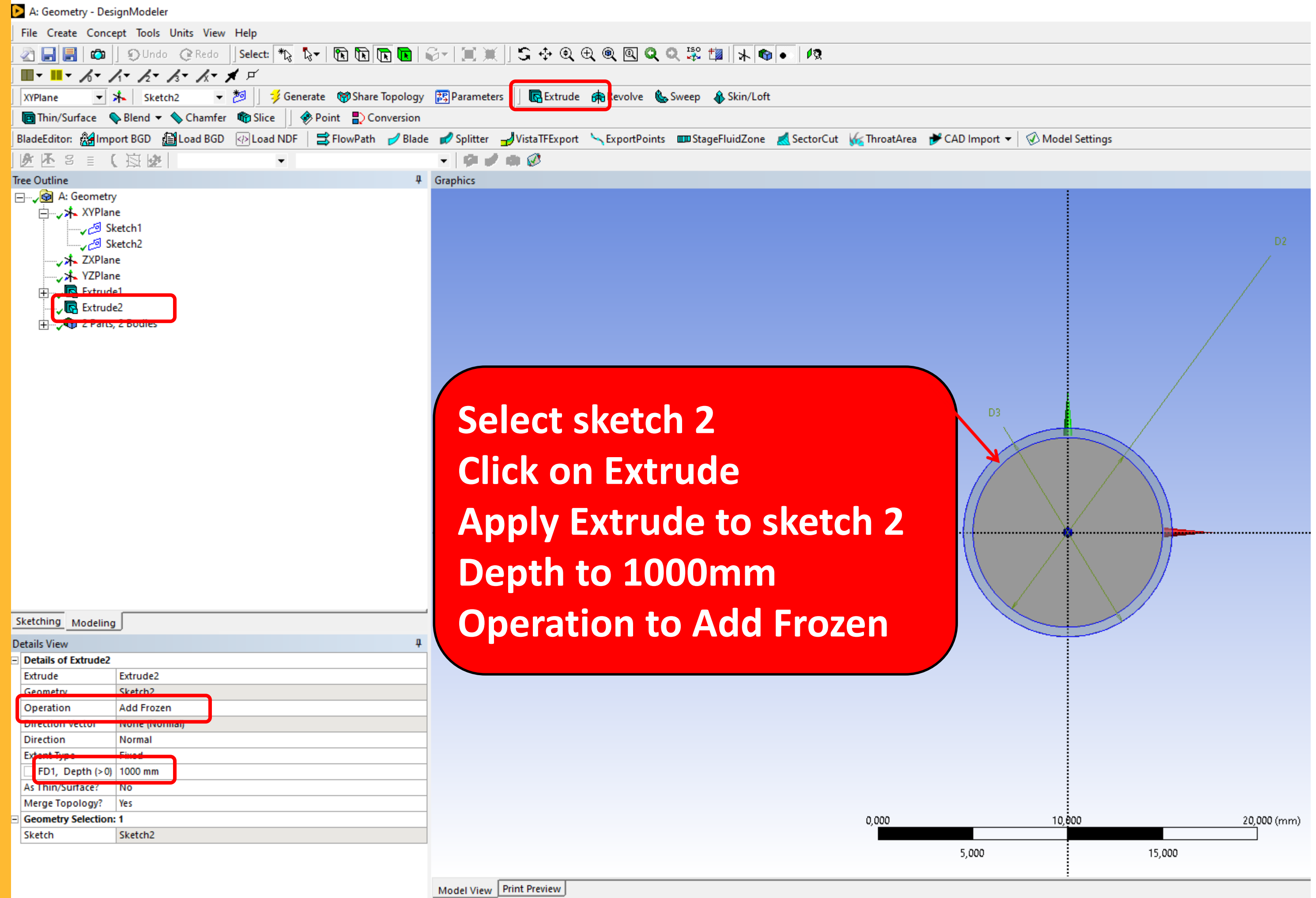
Geometry



Creating Geometry



Creating Geometry



Creating Geometry

The screenshot displays the Ansys 2022 R2 DesignModeler interface. The main workspace shows a 3D model of a cylindrical part. The Tree Outline on the left lists the model's structure: A: Geometry, XYPlane, Sketch1, Sketch2, ZXPlane, YZPlane, Extrude1, Extrude2, 1 Part, 2 Bodies, Part, Heat Transfer Fluid, and Aluminum. A red box highlights the 'Part' entry. A context menu is open over the 'Heat Transfer Fluid' entry, with 'Form New Part' selected. A red callout box points to this menu with the text: 'Select both body Right click Select Form New Part'. The Details View on the bottom left shows the properties of the selected body. A red box highlights the 'Body' row, and another red box highlights the 'Fluid/Solid' row, which is set to 'Fluid'. A red callout box points to these rows with the text: 'Change body from Solid to Fluid'. The bottom status bar shows '1 Body: Volume = 78540 mm³'.

**Select both body
Right click
Select Form New Part**

**Click to view
3D**

**Change body from Solid to
Fluid**

Details View	
Details of Body	
Body	Heat Transfer Fluid
Volume	78540 mm ³
Surface Area	31573 mm ²
Faces	3
Edges	2
Vertices	0
Fluid/Solid	Fluid
Shared Topology Method	Default
Geometry Type	DesignModeler

1 Body: Volume = 78540 mm³

SUMMARY

In this first tutorial, you were able to

- 01** Start up your ANSYS Workbench
- 02** Create the geometry the circular tube