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**

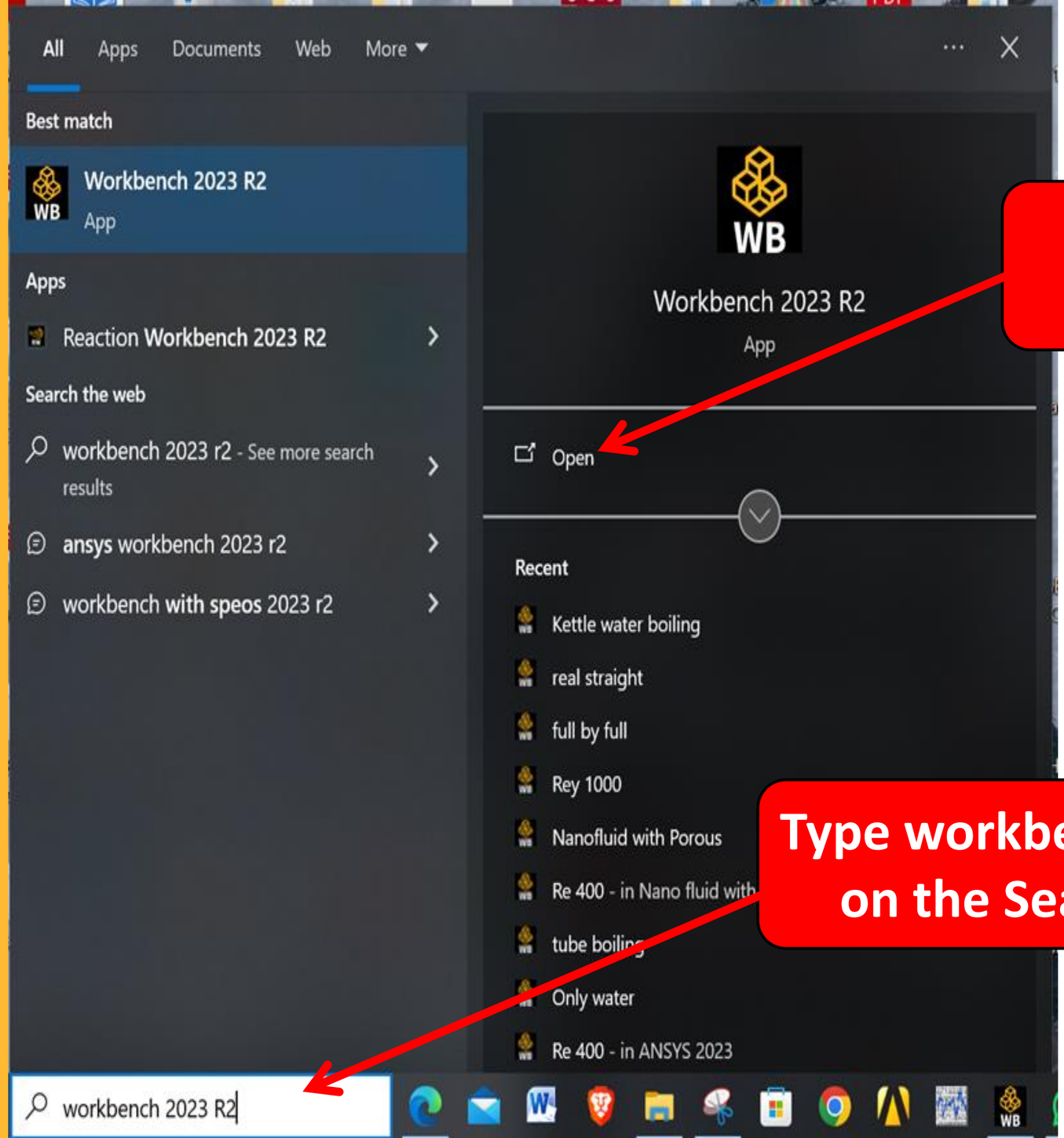
02 **Create the geometry of a Circular Tube**

STARTING UP ANSYS WORKBENCH

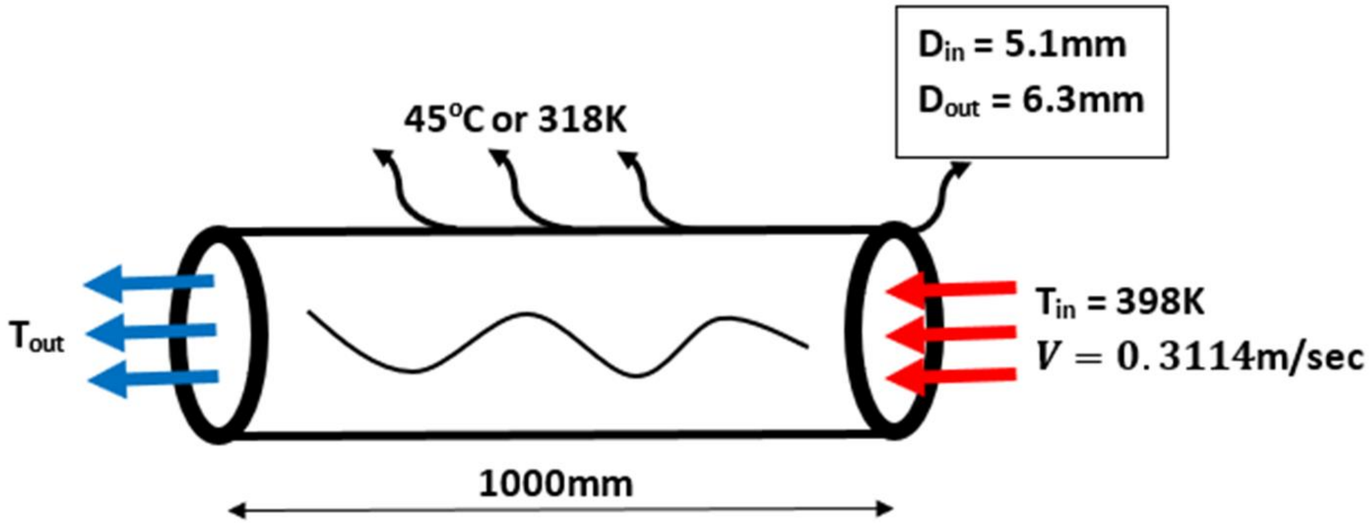
Starting Up

ANSYS

Workbench

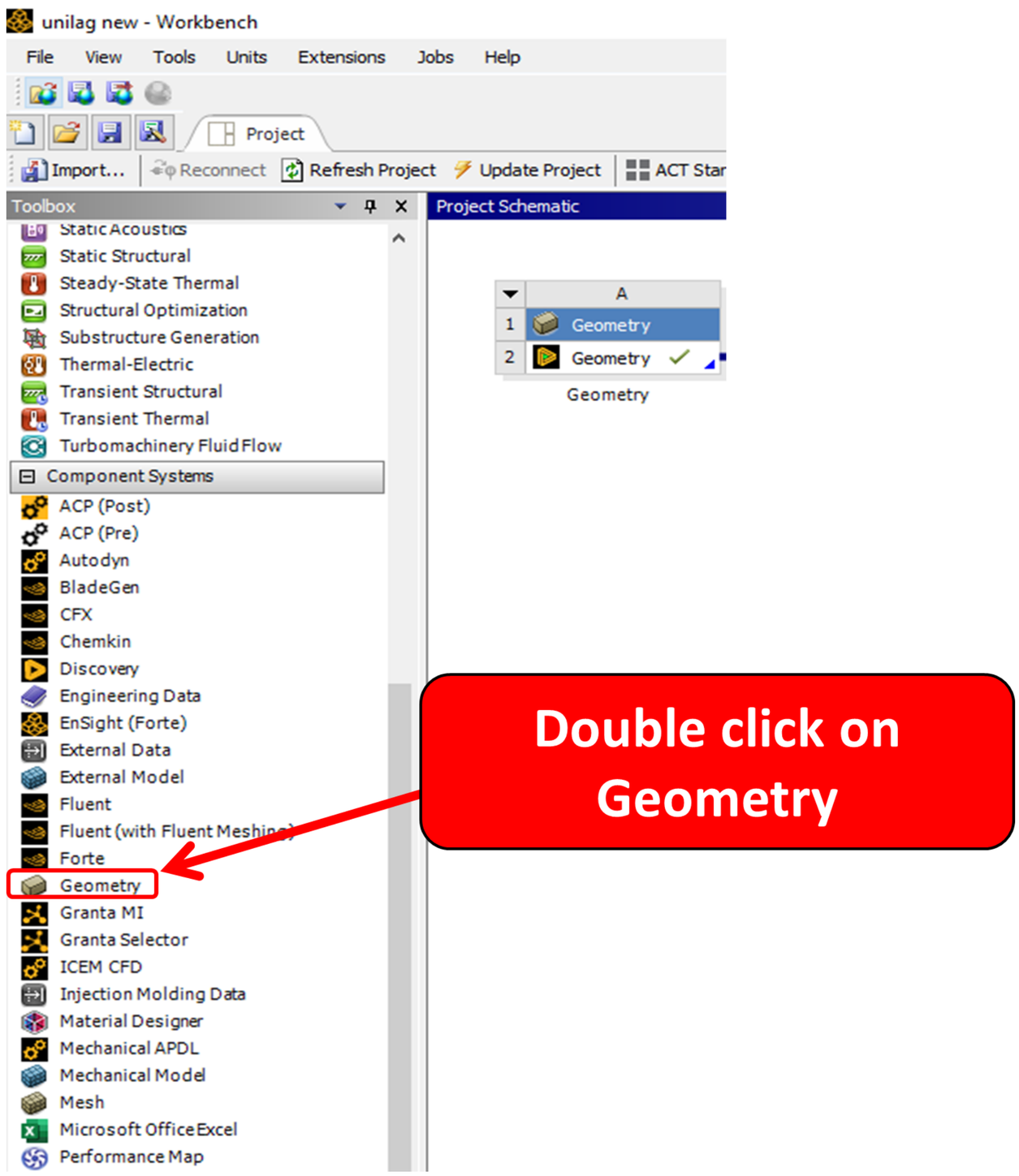


CREATING THE GEOMETRY



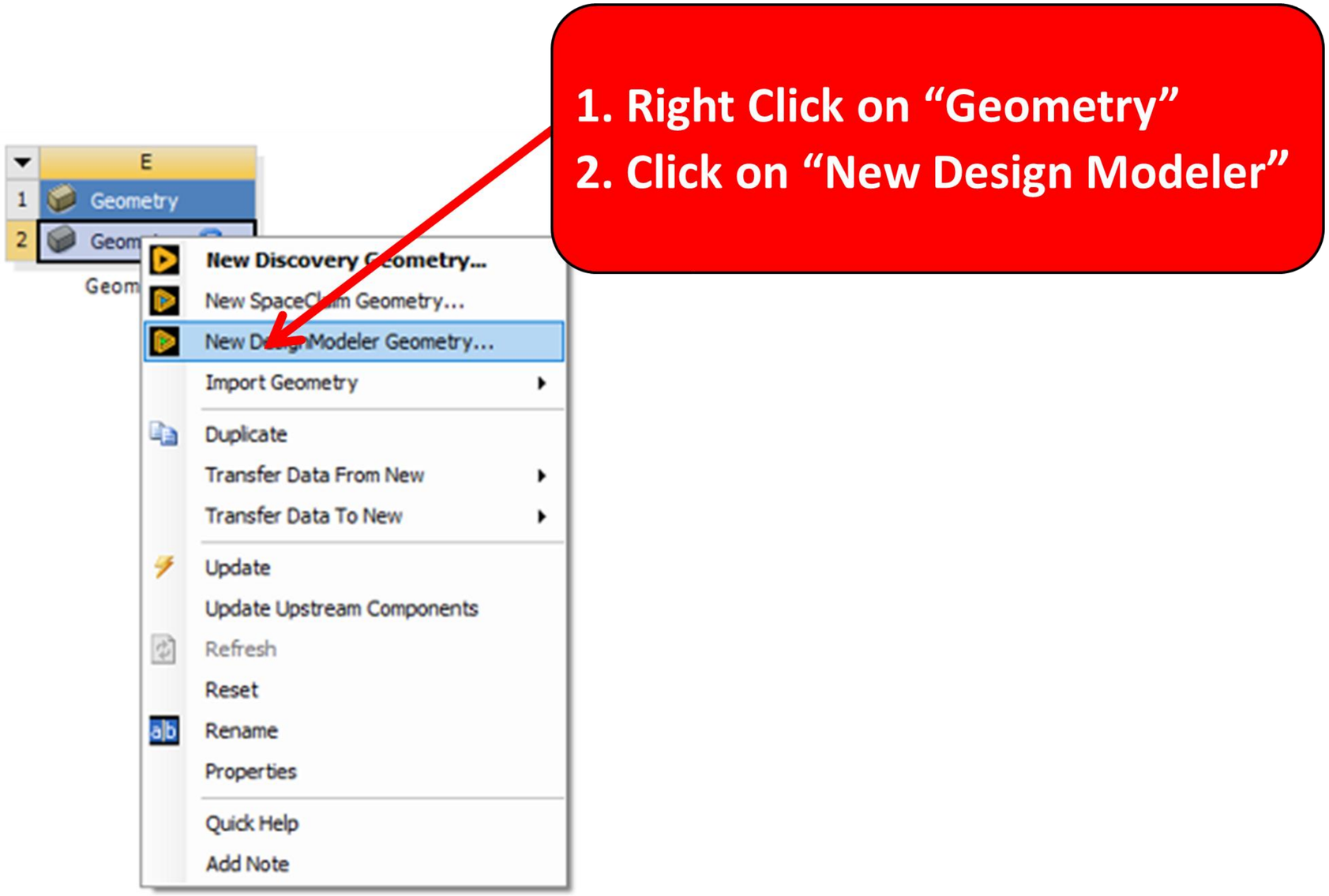
Creating

Geometry



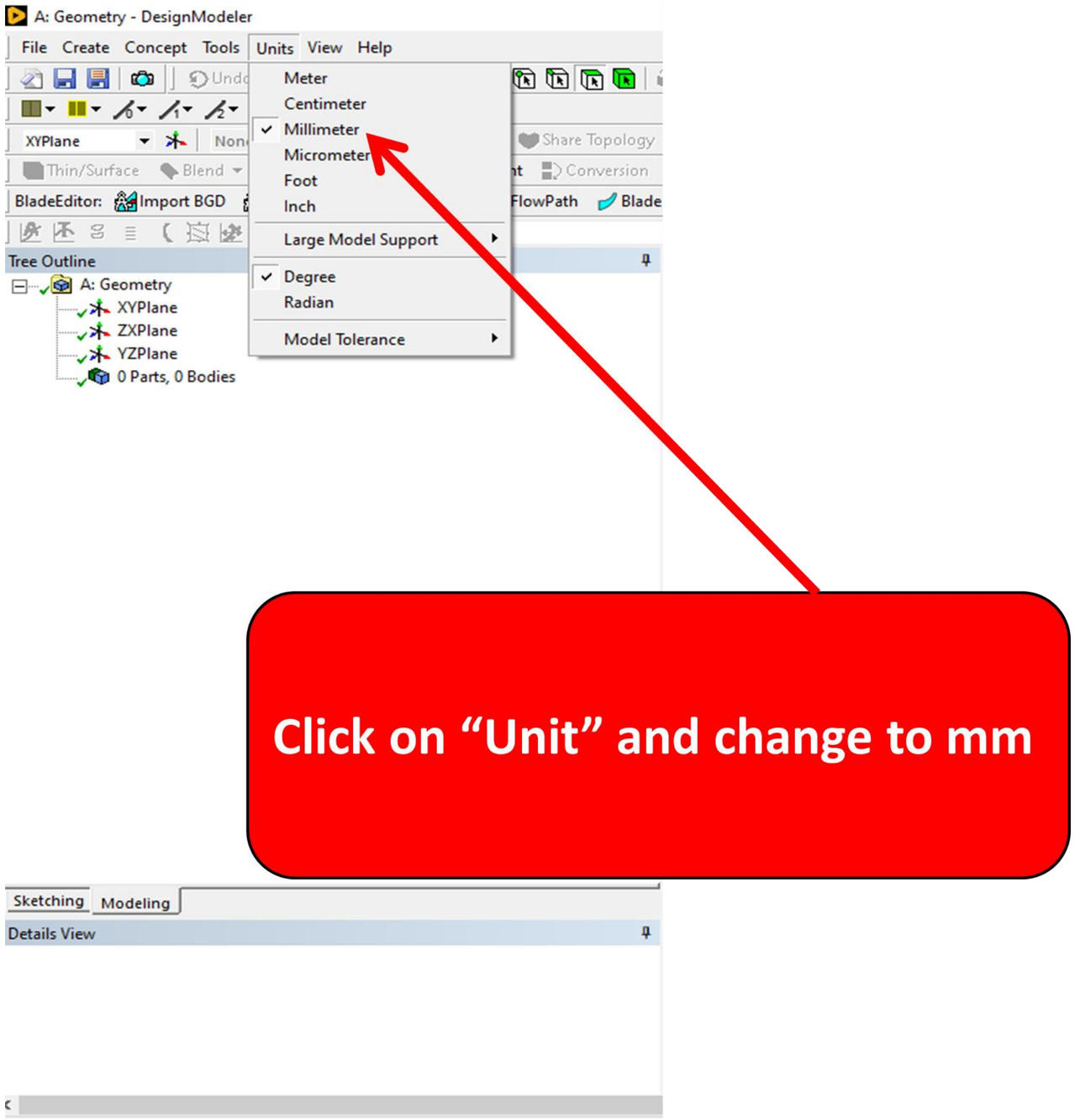
Creating

Geometry



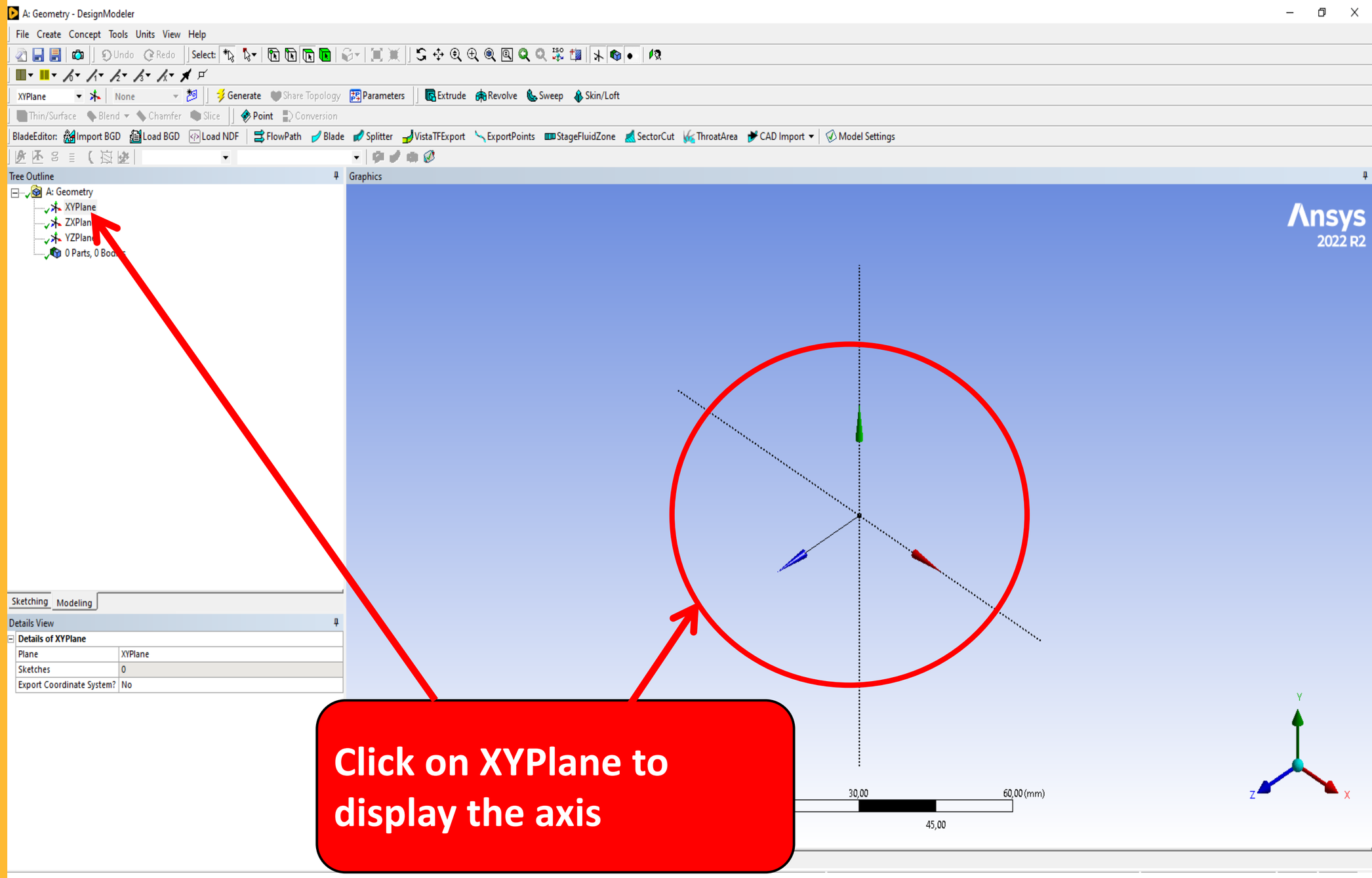
Creating

Geometry



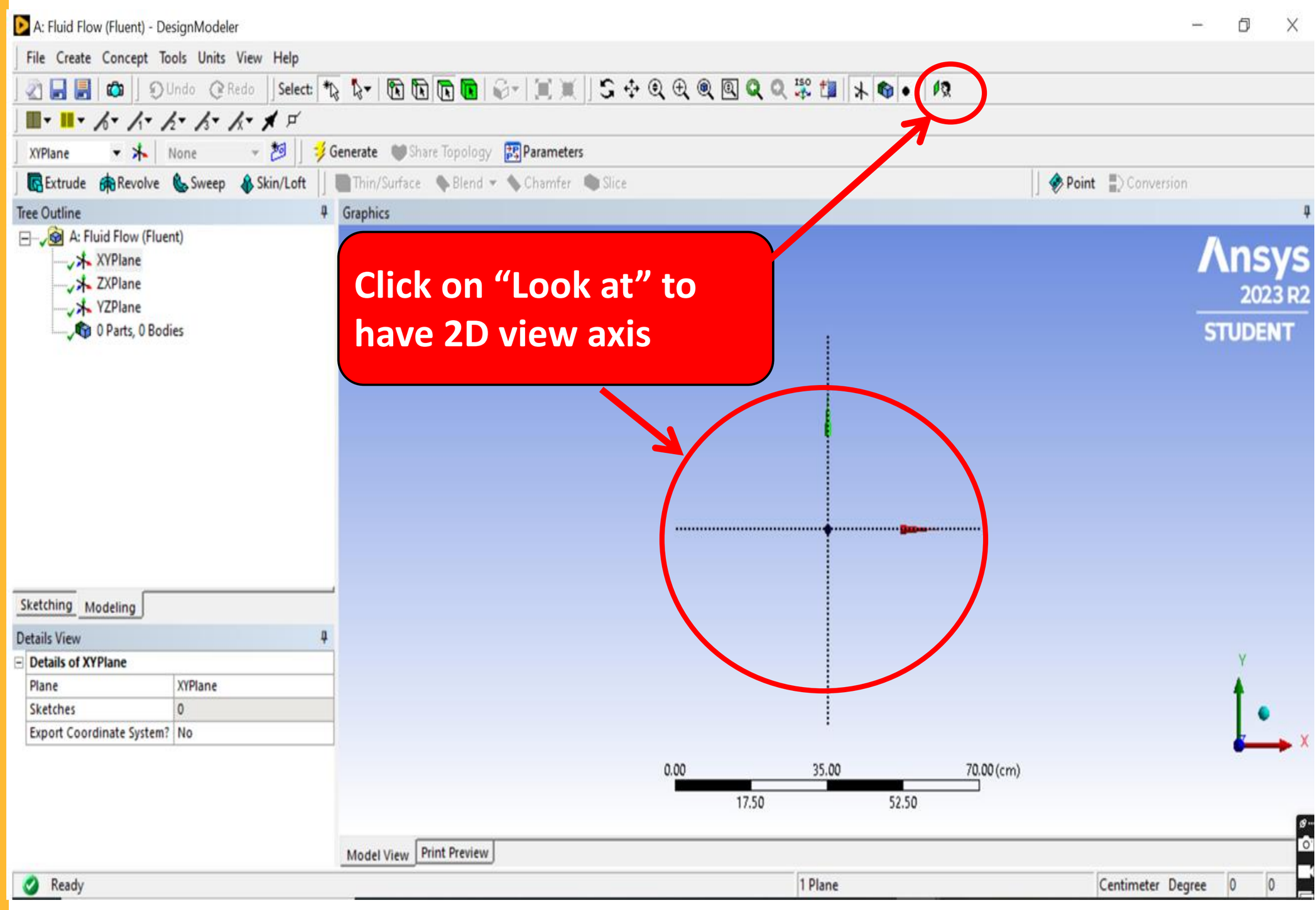
Creating

Geometry



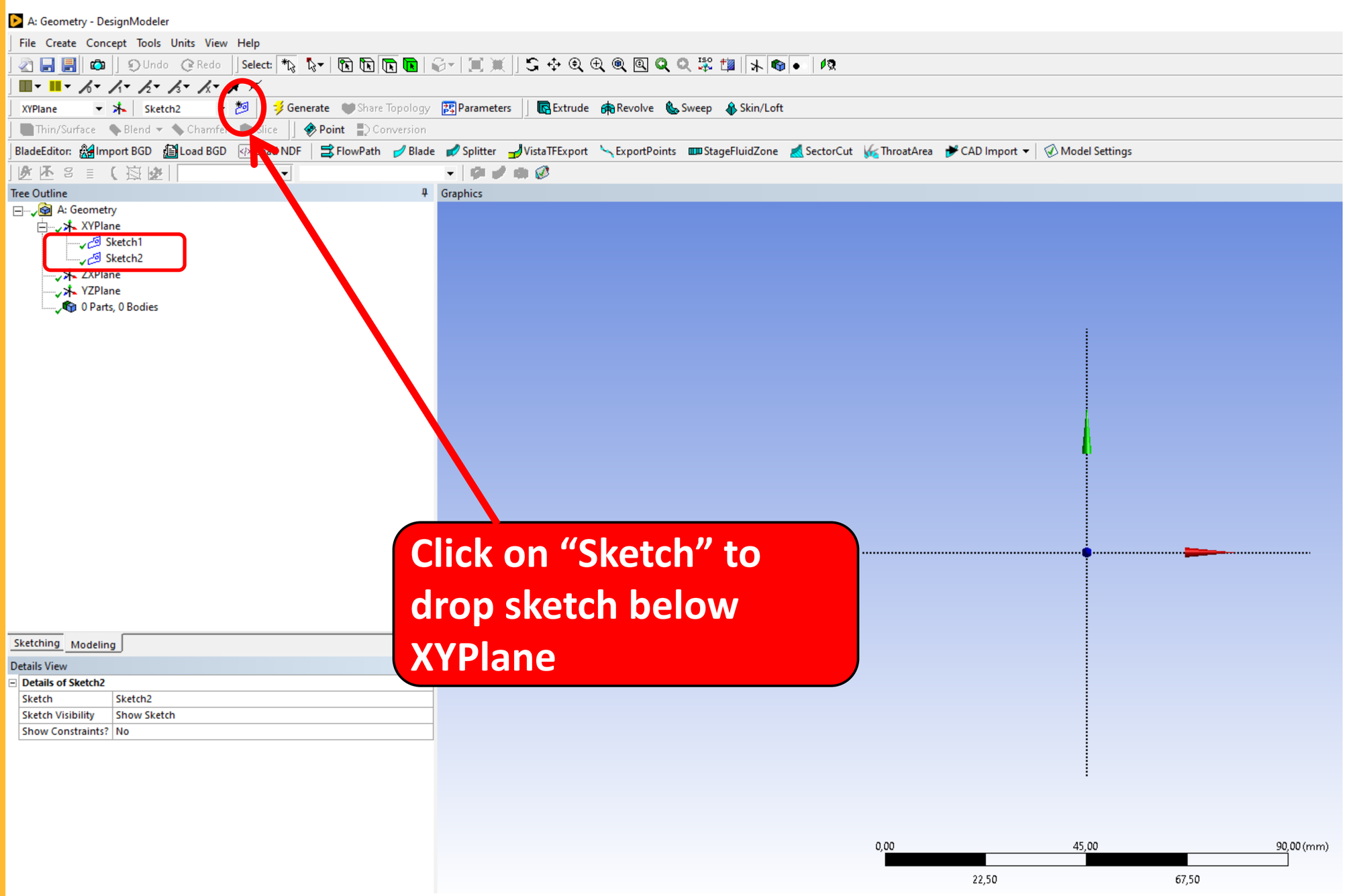
Creating

Geometry



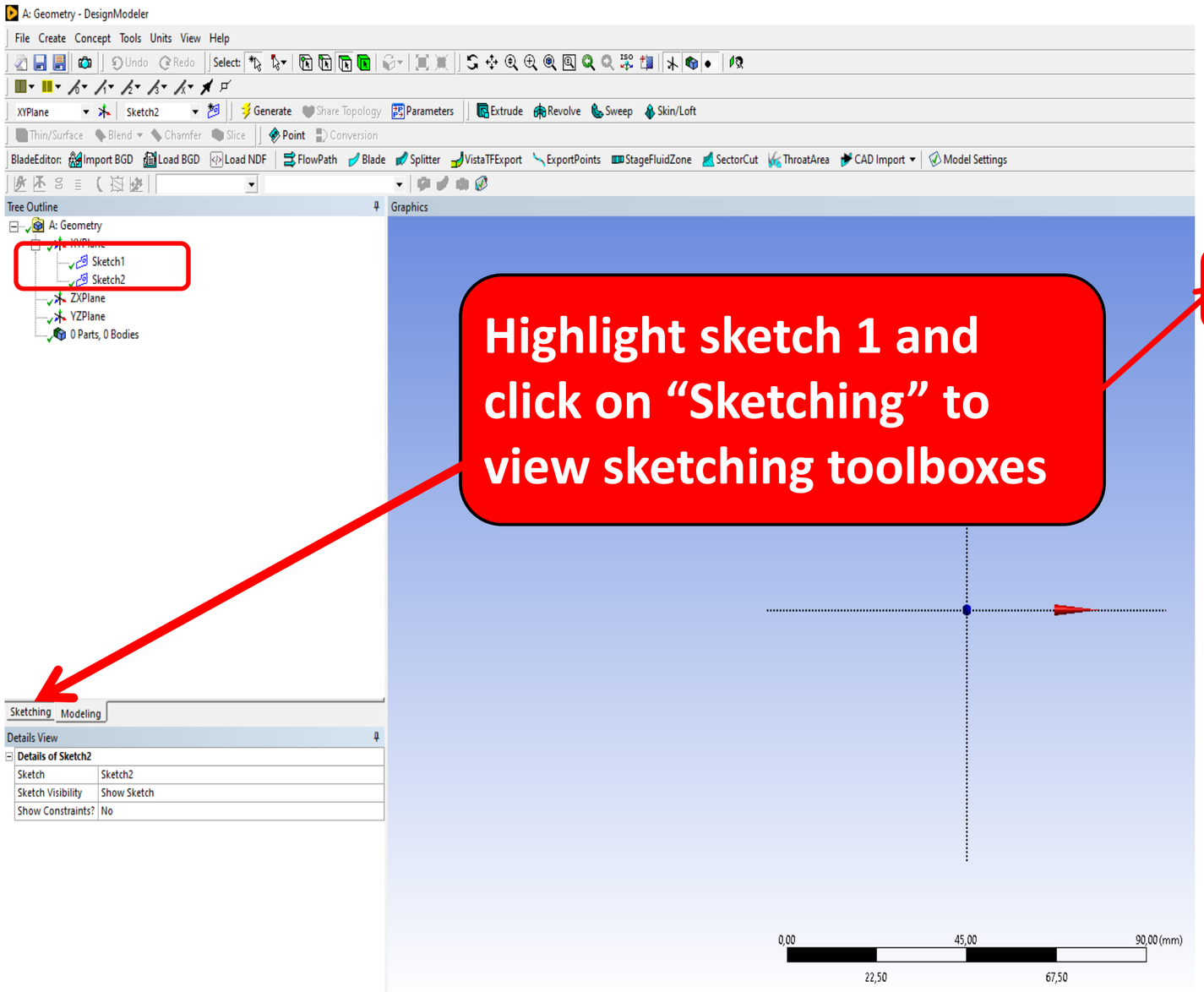
Creating

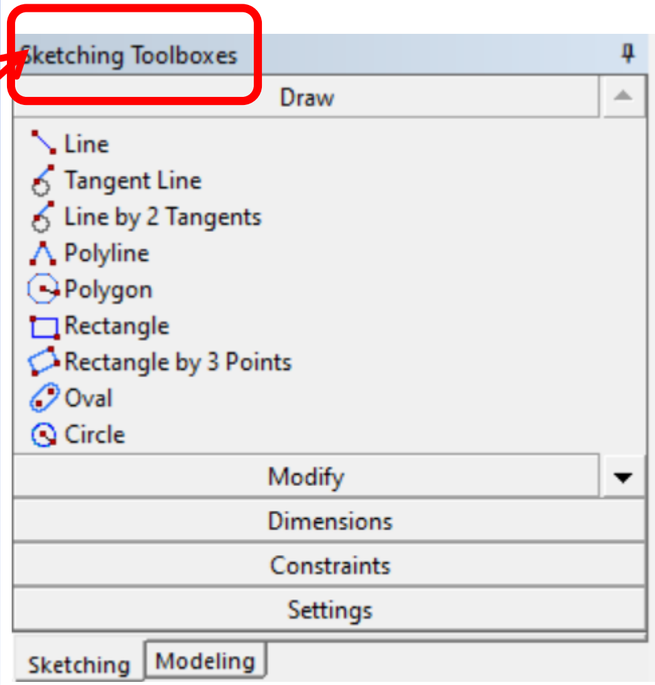
Geometry



Creating

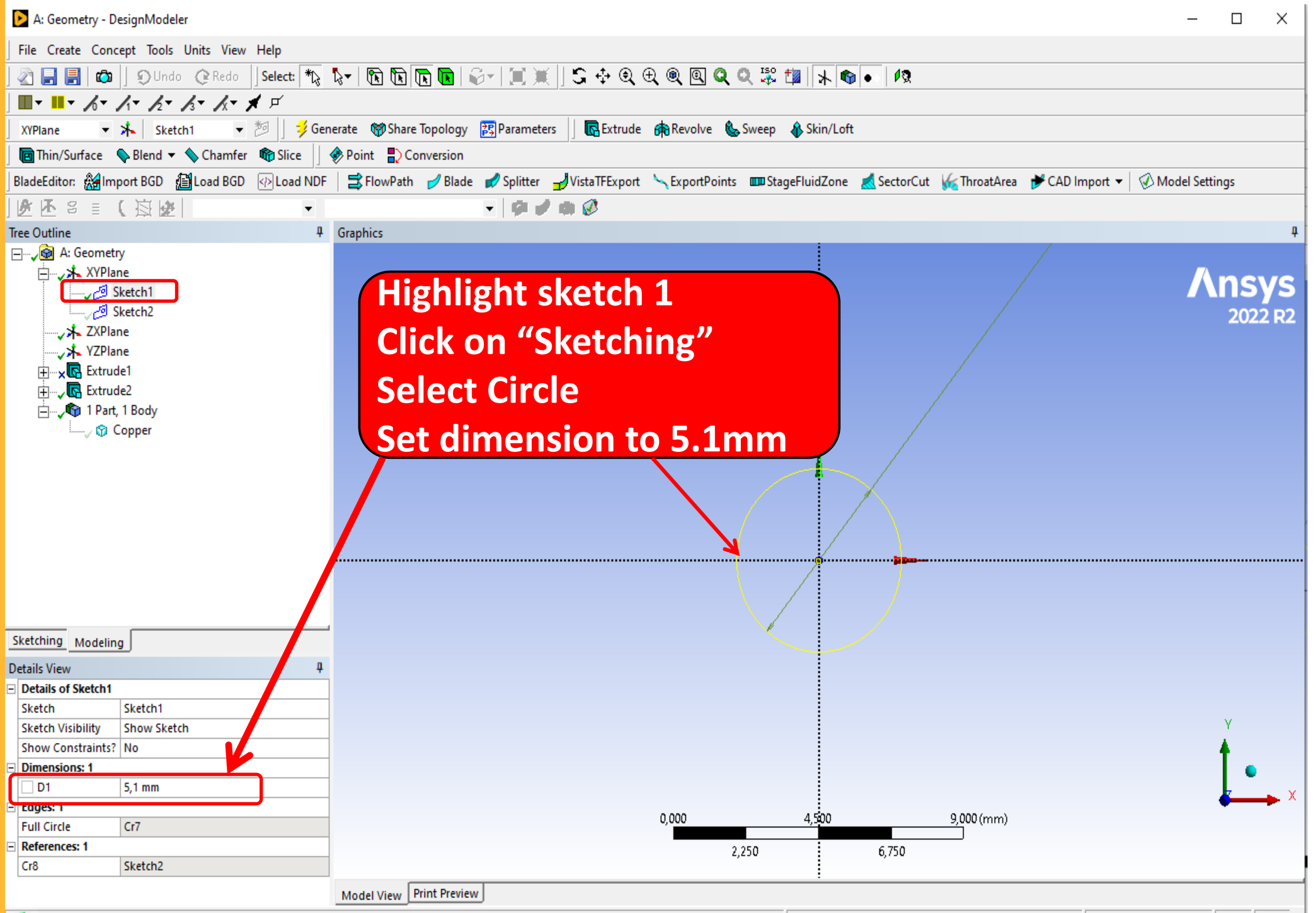
Geometry





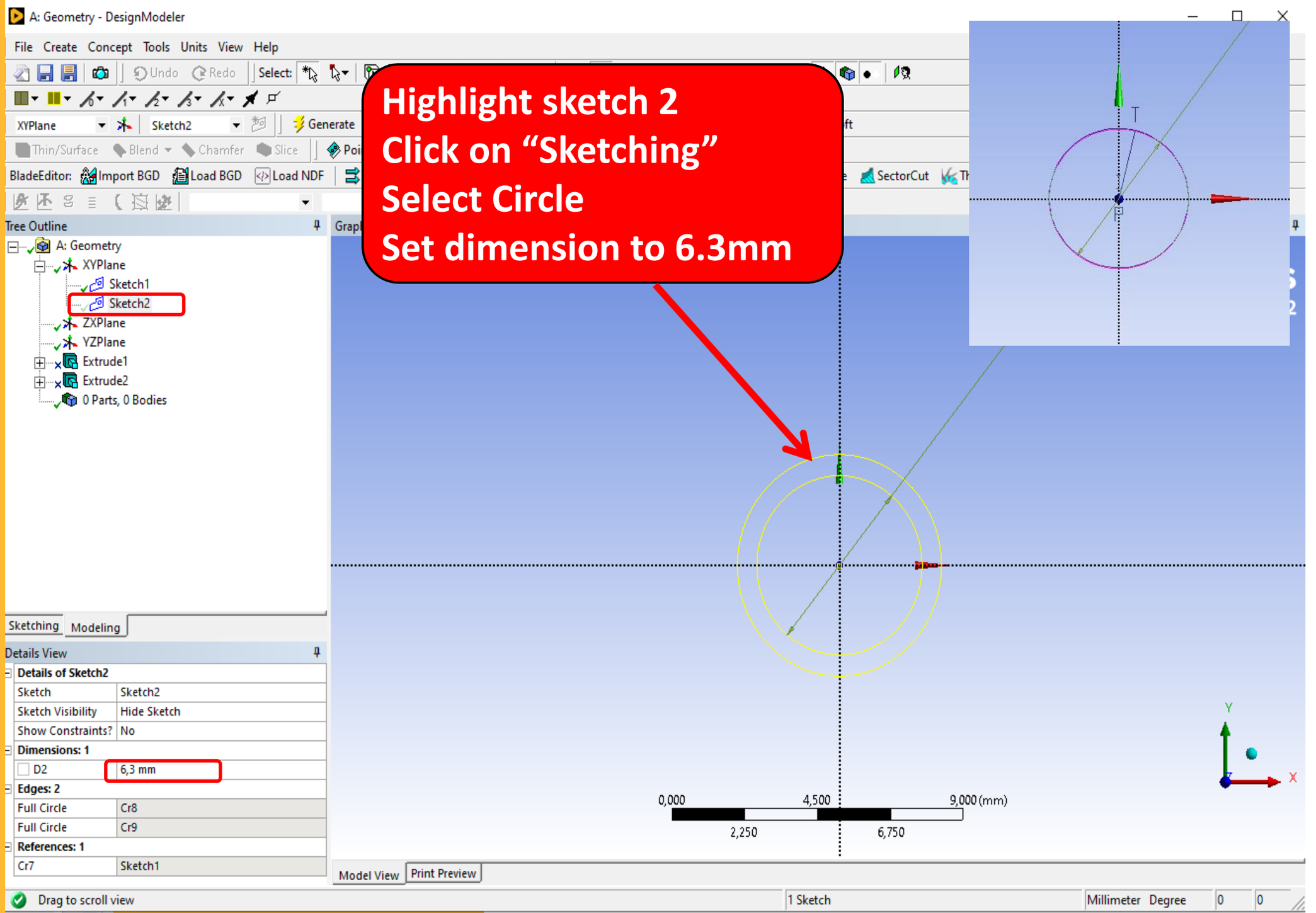
Creating

Geometry



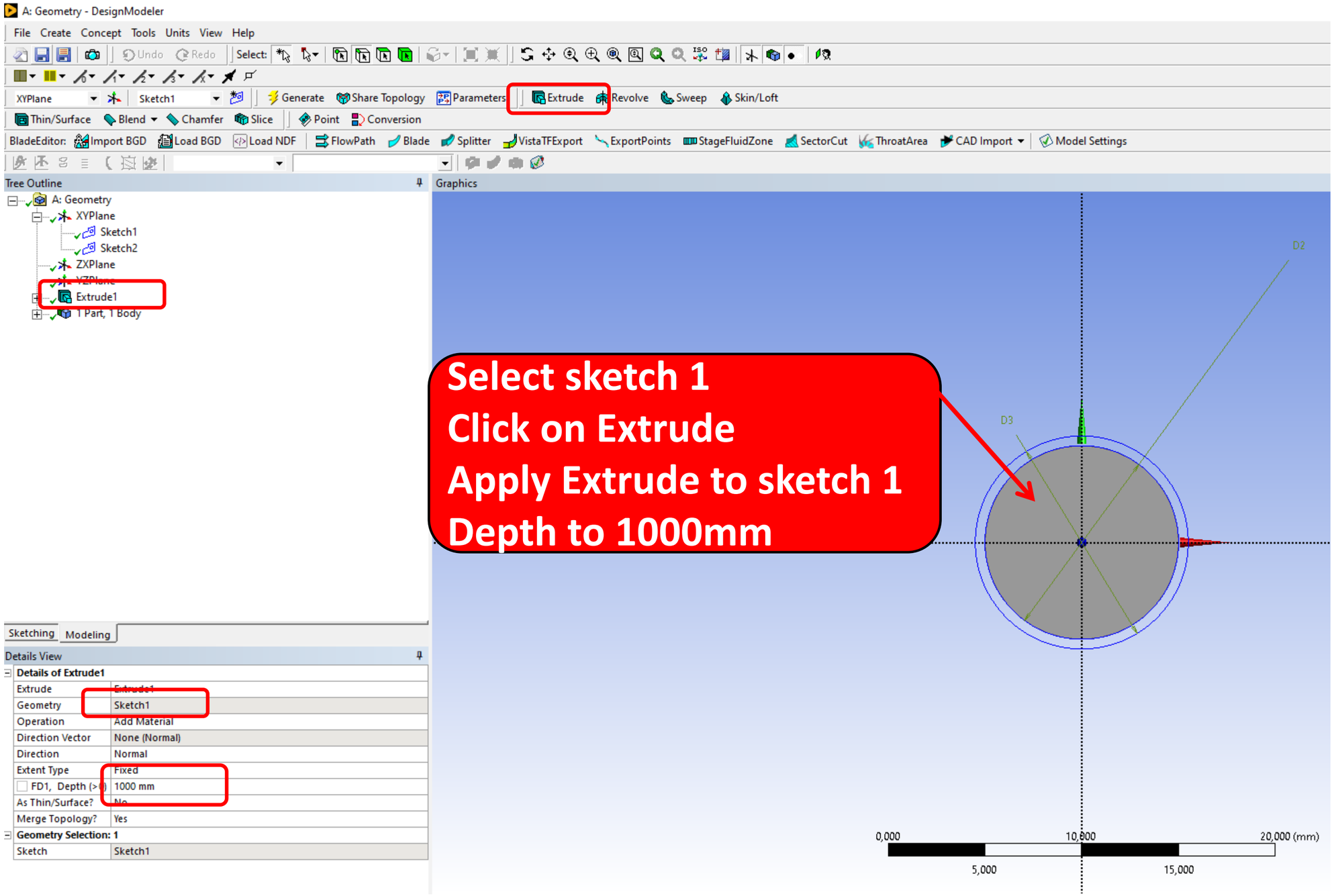
Creating

Geometry



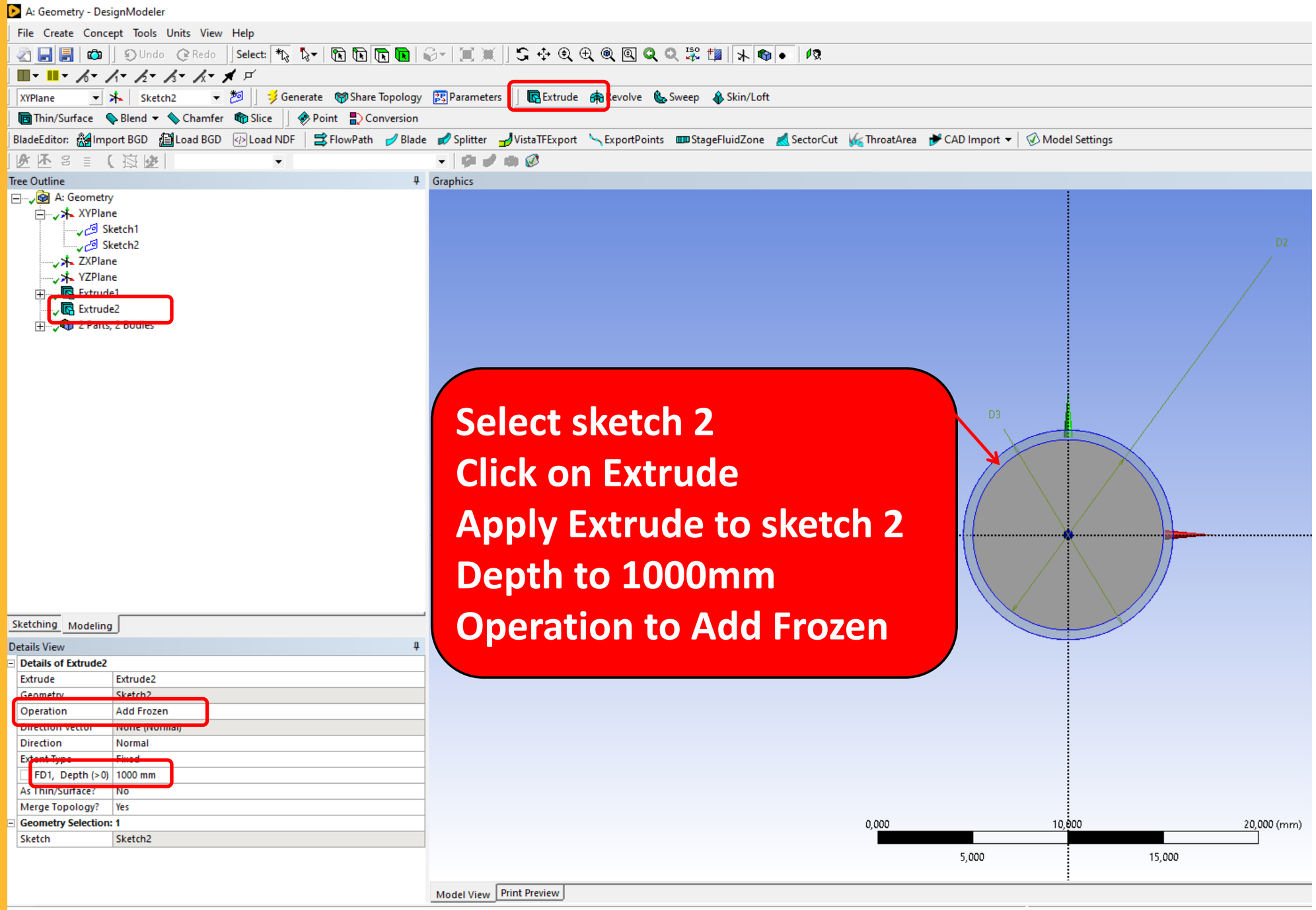
Creating

Geometry



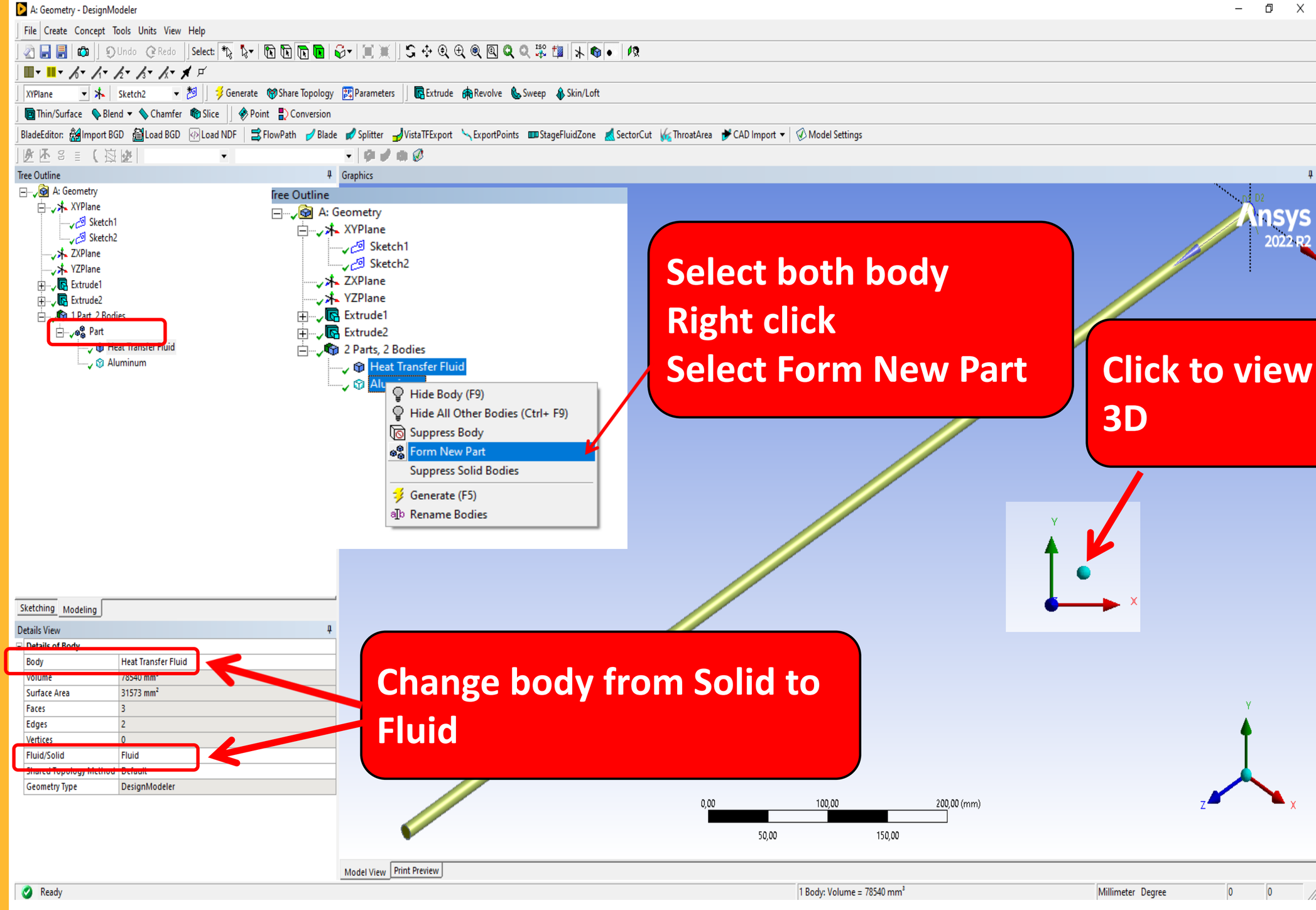
Creating

Geometry



Creating

Geometry



SUMMARY

In this first tutorial, you were able to

01 **Start up your ANSYS Workbench**

02 **Create the geometry the circular tube**