



## Chapter two

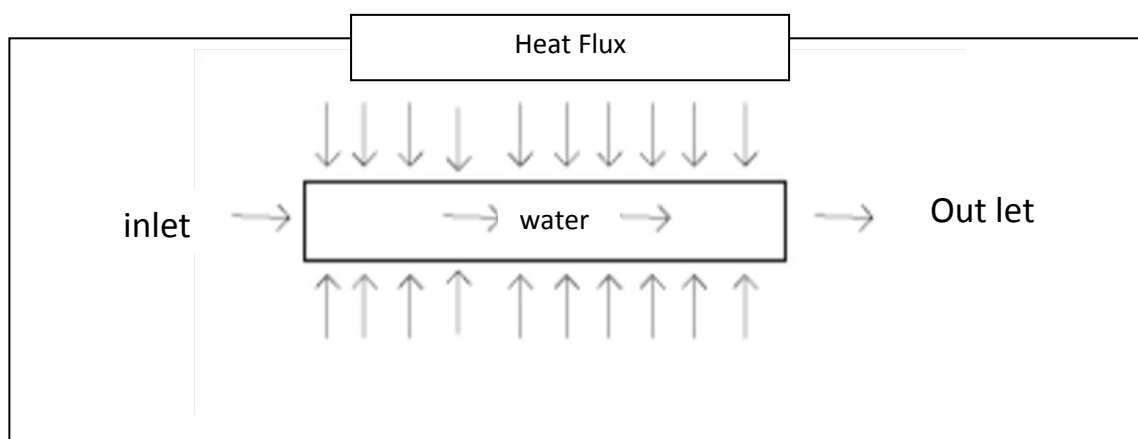
**1- example 1** :This tutorial illustrates how to use CFD-Post to visualize a two dimensional laminar fluid flow.

This tutorial demonstrates how to do the following:

- 1.1 Create a Working Directory
- 1.2 Launch CFD-Post
- 1.3 Display the Solution in CFD-Post
- 1.4 Save Your Work
- 1.5 Generated Files

### **Problem Description**

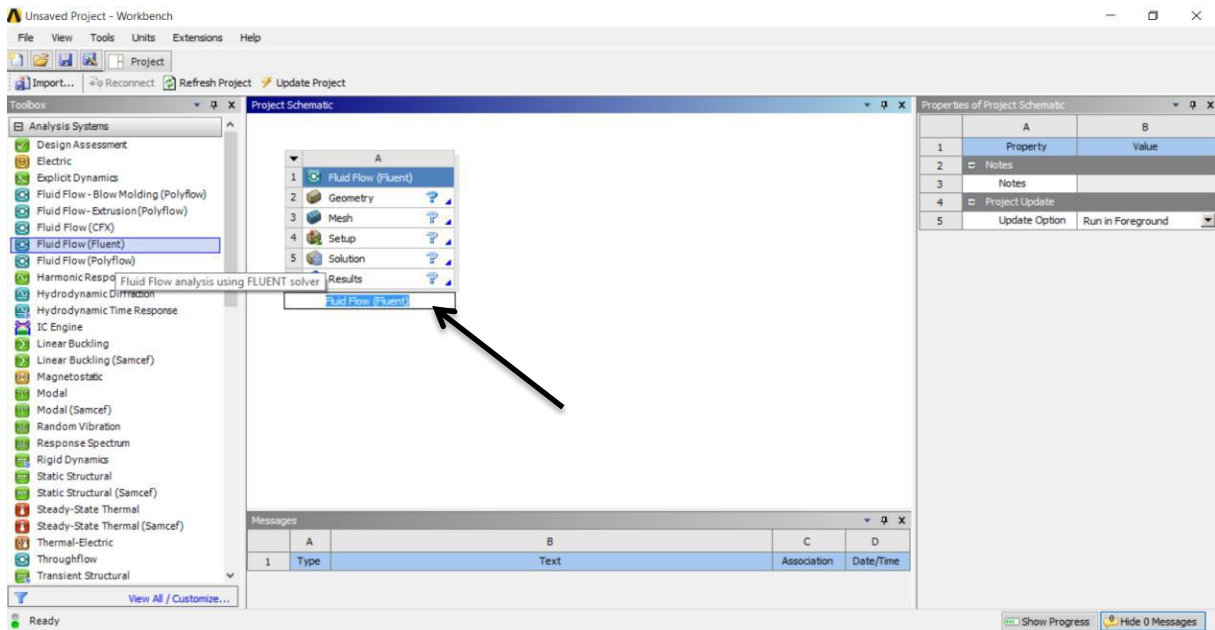
The problem to be considered is shown schematically in Figure below: A cold fluid at  $25^{\circ}\text{C}$  flows into the pipe through a large inlet. The pipe dimensions are in mm (200, 30 mm), but the fluid properties and boundary conditions are given in metric units. The Reynolds number for the flow at the larger inlet is 2,000, so the flow has been modeled as being laminar. The walls are exposed to a  $300\text{ W /m}^2$  heat flux.



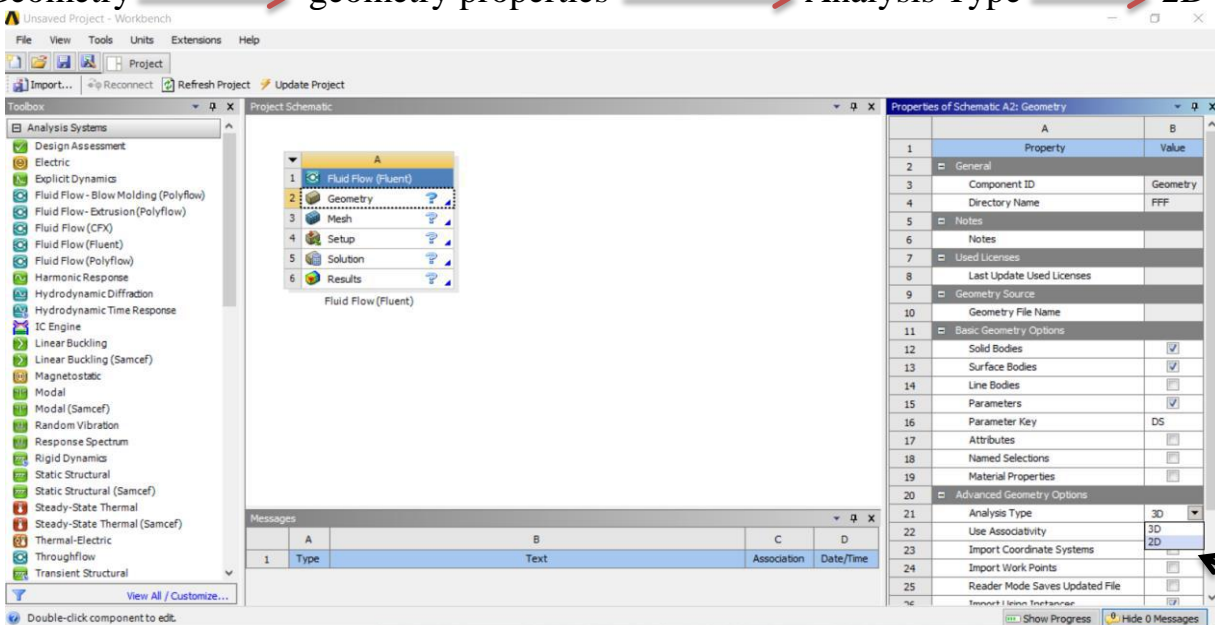


## SOLUTION:

1- Analysis systems → fluid flow (fluent) → create name

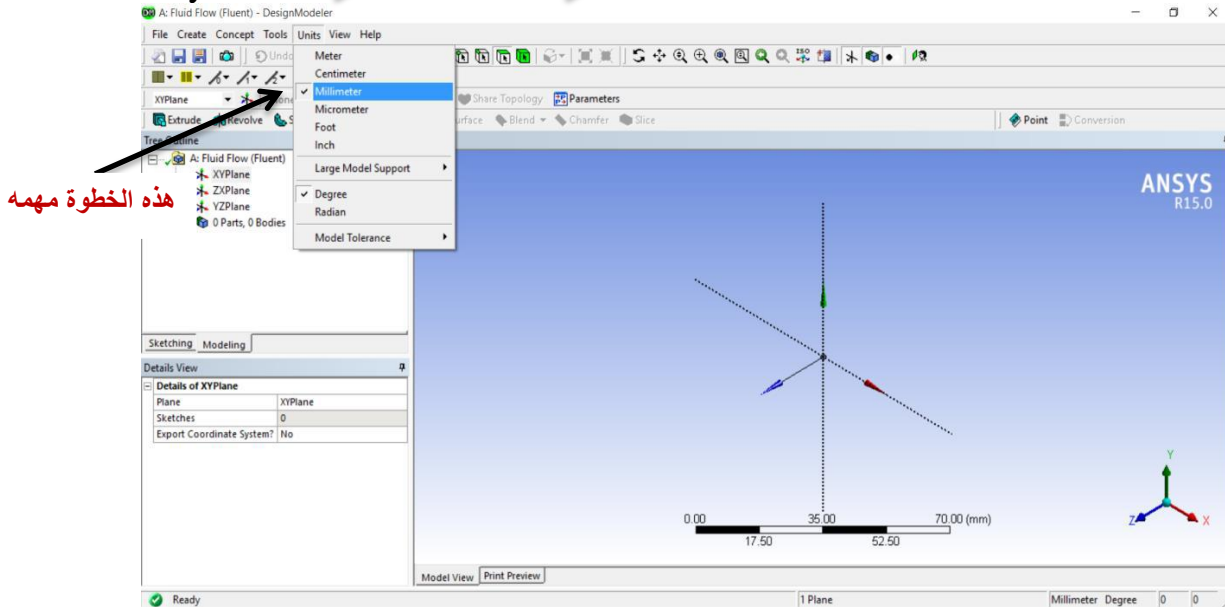


2- Geometry → geometry properties → Analysis Type → 2D



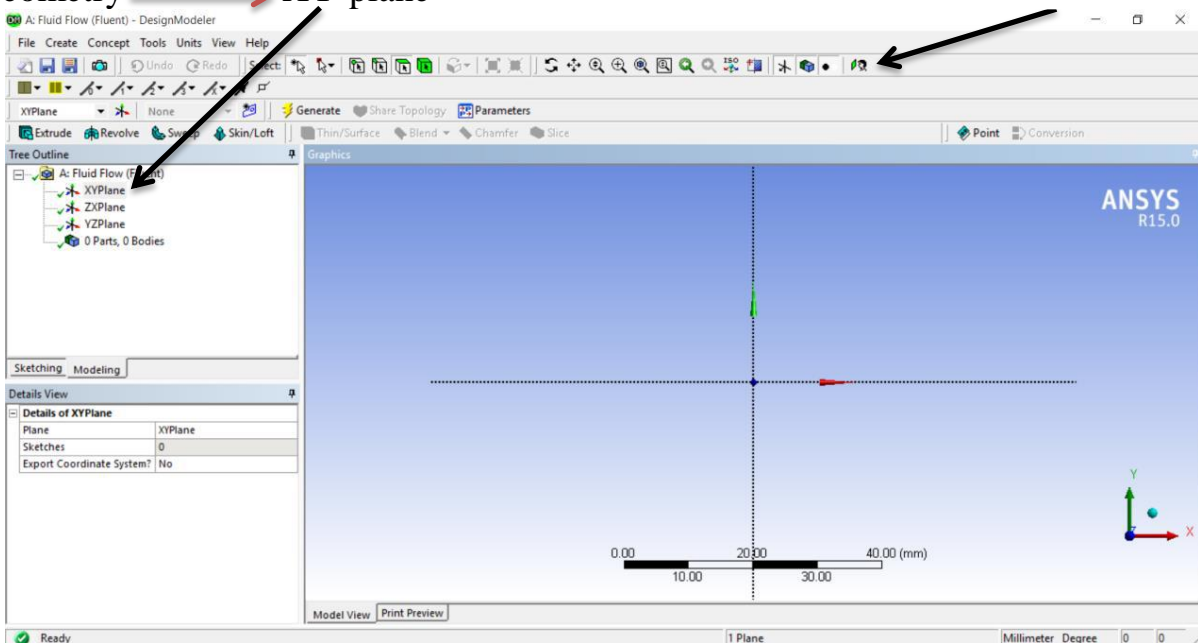


3- Geometry  $\longrightarrow$  unites  $\longrightarrow$  millimeters



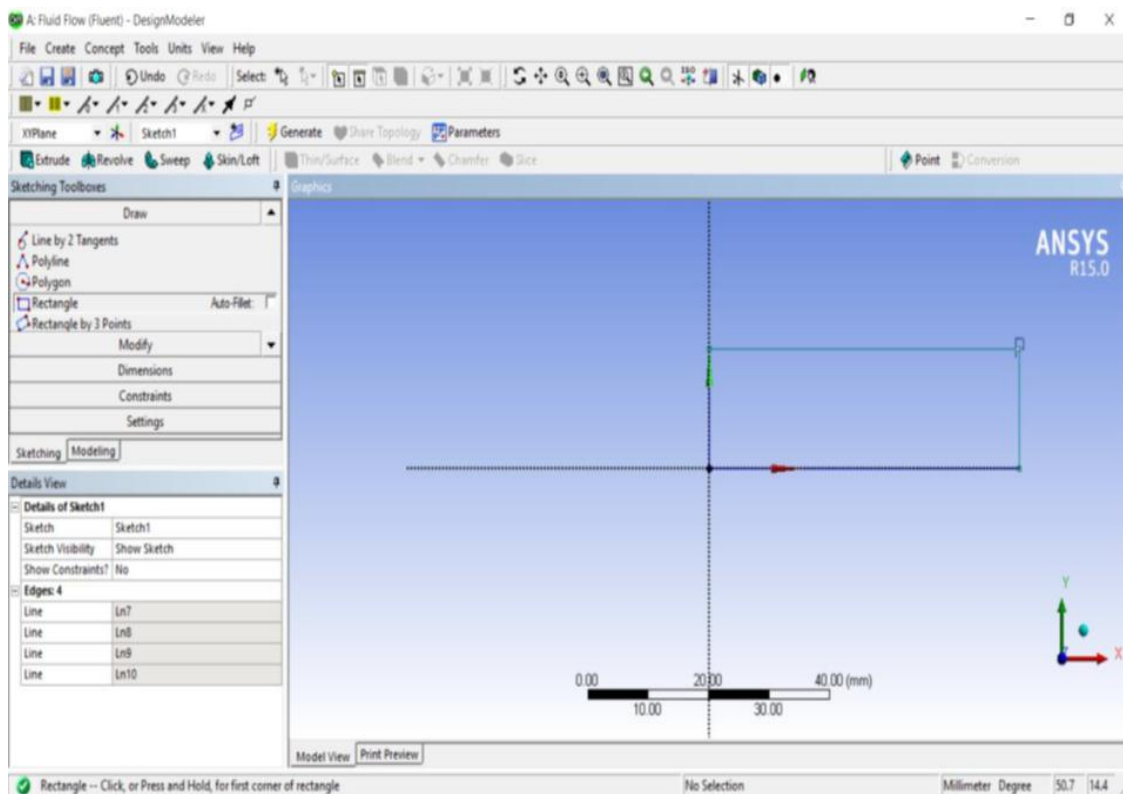
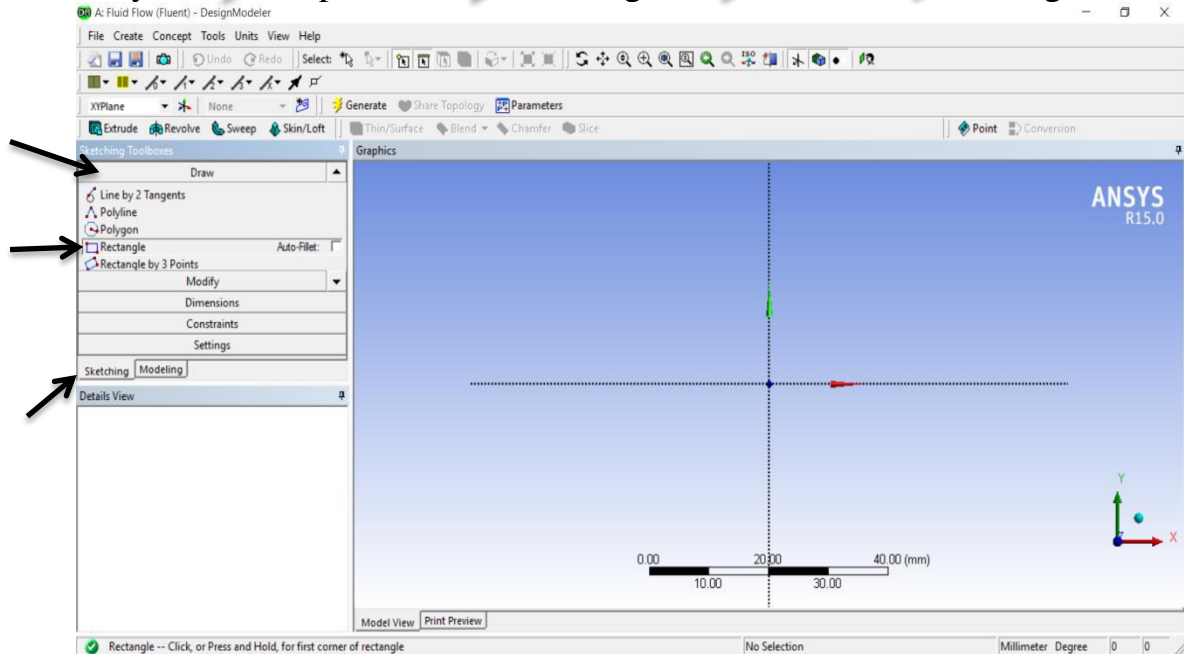
4- Geometry  $\longrightarrow$  XY-plane

نحول الاحداثيات الى احداثي ثنائي



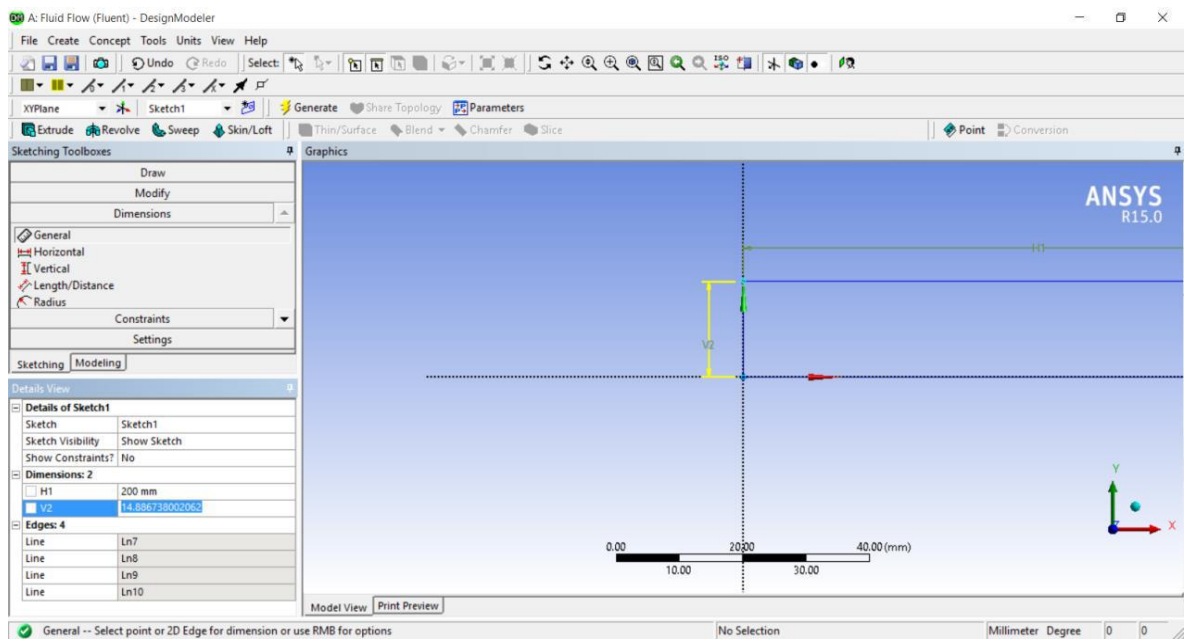
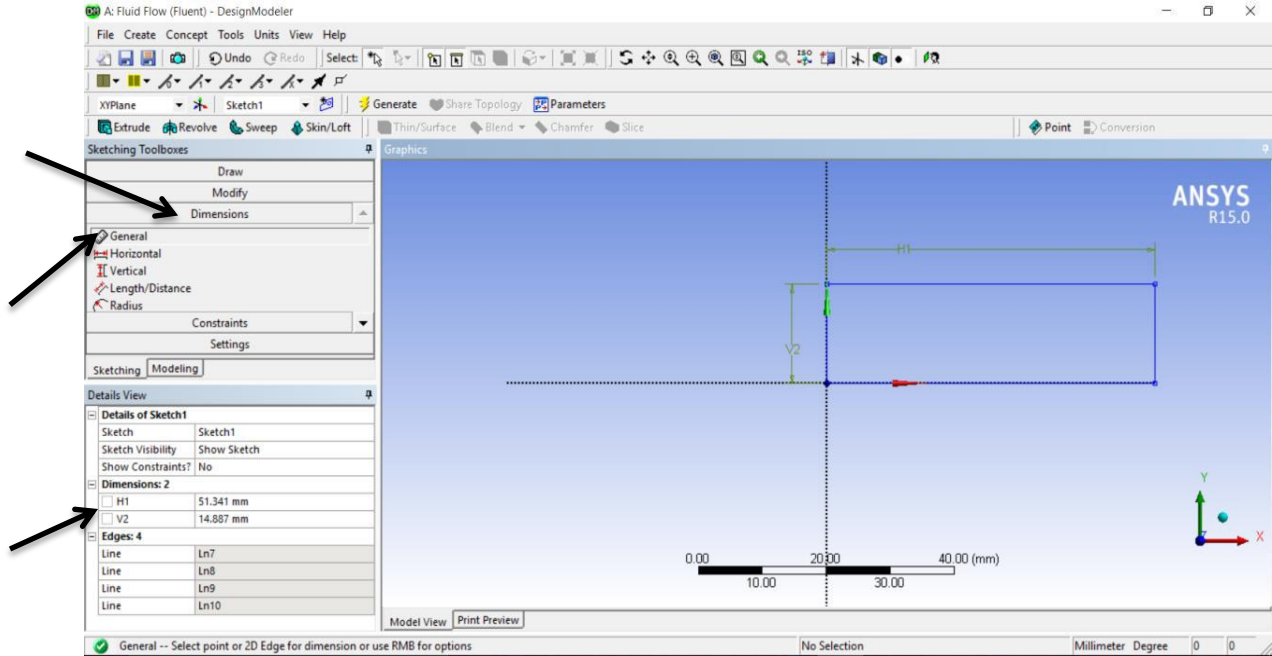


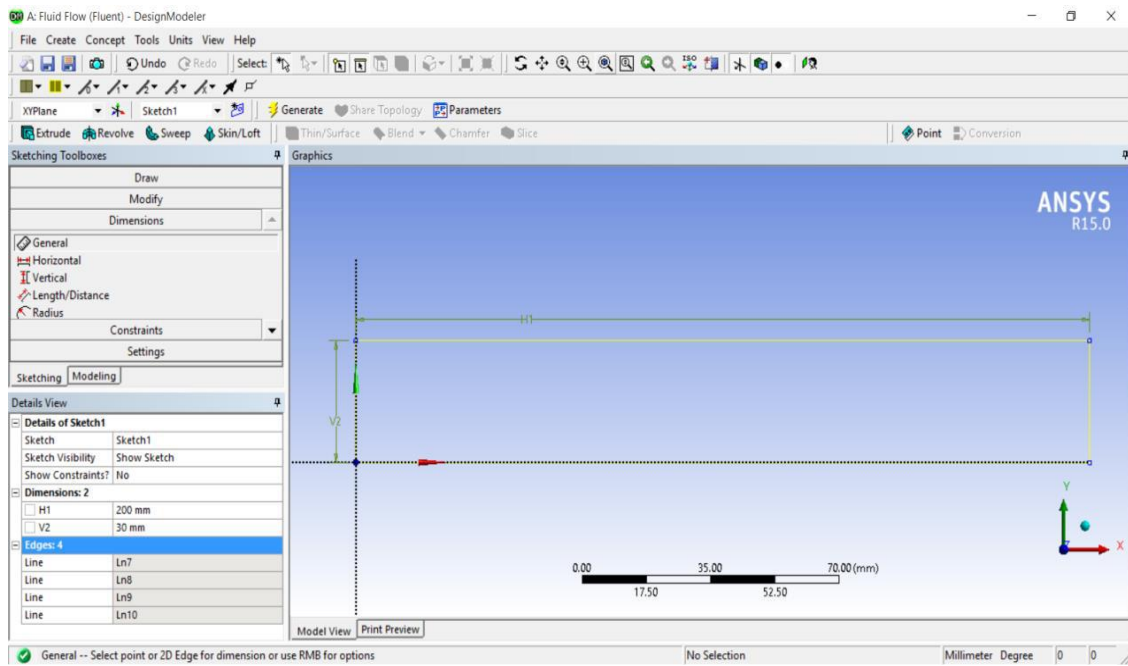
5- Geometry → XY-plane → Sketching → Draw → Rectangular



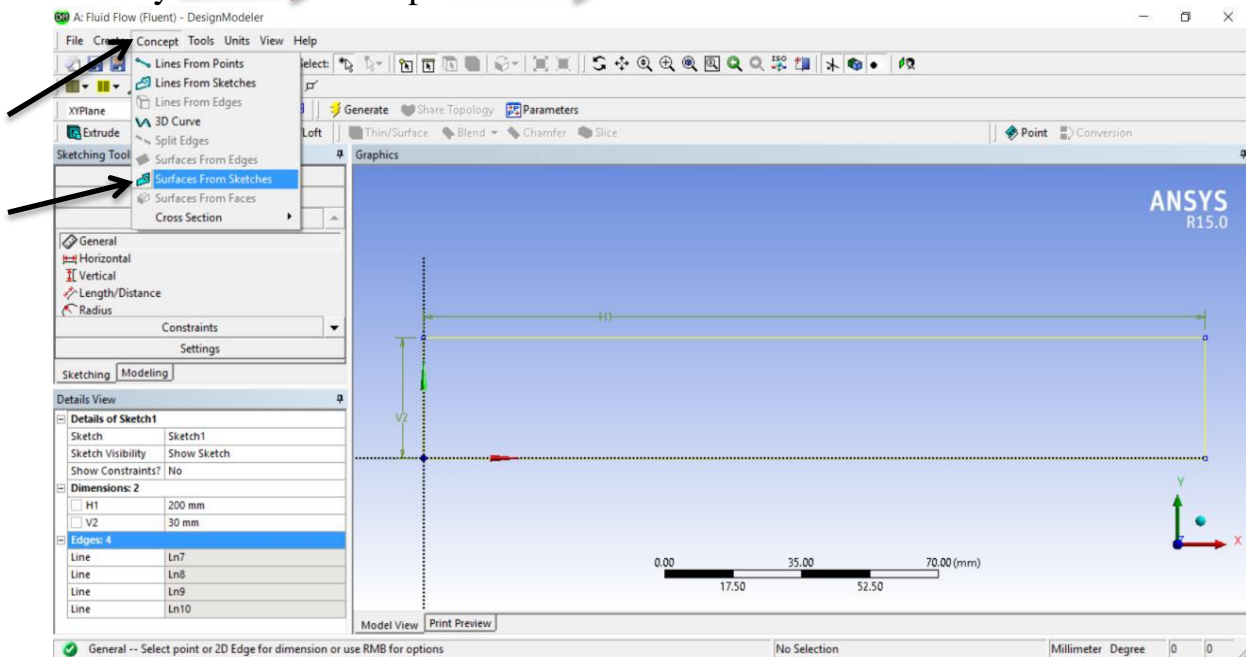


6- Geometry → XY-plane → Sketching → Dimensions → General



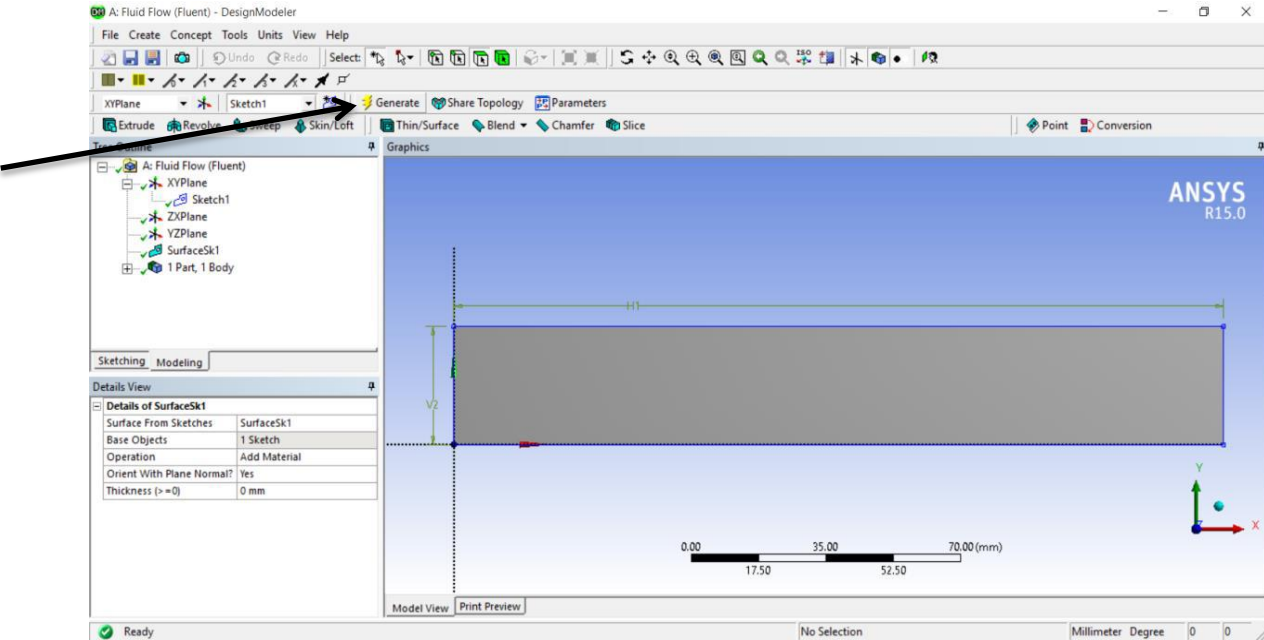
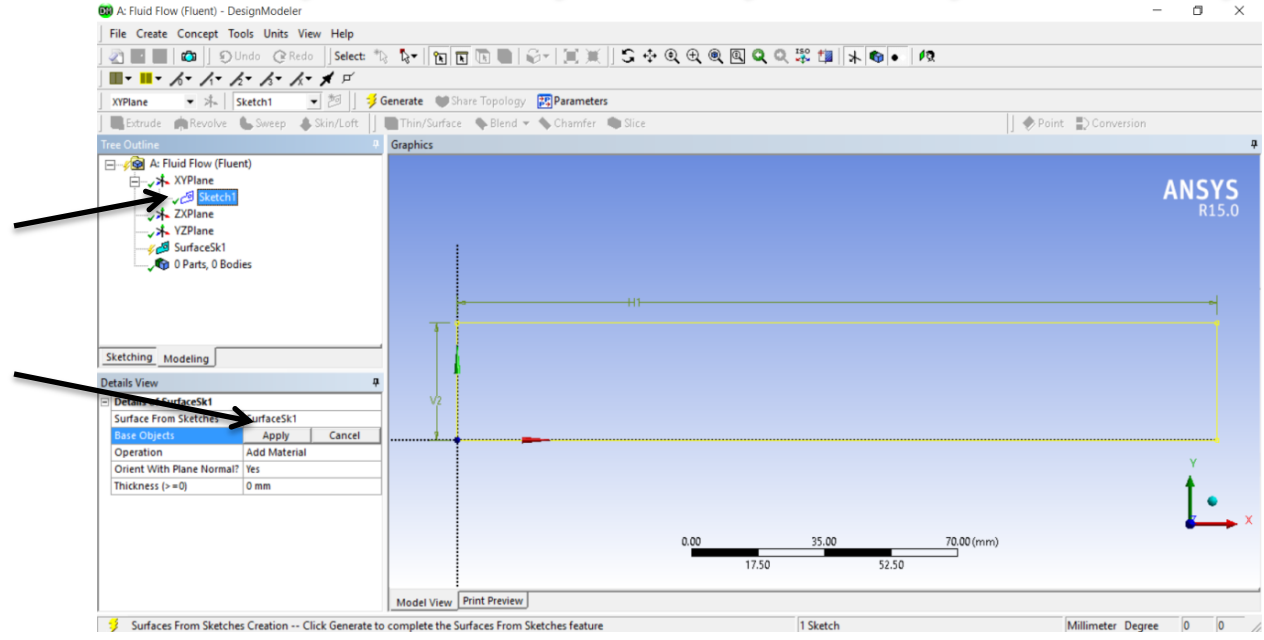


7- Geometry  $\longrightarrow$  concept  $\longrightarrow$  surface from sketches





8- Geometry → XY-plane → sketch1 → apply → generate → close





9- Analysis systems → fluid flow (fluent) → Mesh

The image displays two screenshots from the ANSYS software suite. The top screenshot shows the ANSYS Workbench interface. In the 'Toolbox' on the left, the 'Mesh' component is highlighted with a black arrow. The 'Project Schematic' in the center shows a sequence of components: Fluid Flow (Fluent), Geometry, Mesh, Setup, Solution, and Results. The 'Properties of Schematic A3: Mesh' panel on the right shows a table with the following data:

Property	Value
Component ID	Mesh
Directory Name	FFF
Physics	CFD
Analysis	Any
Solver	FLUENT

The bottom screenshot shows the ANSYS Meshing R15.0 interface. The 'Generate Mesh' button is highlighted with a black arrow. The 'Outline' panel on the left shows the 'Mesh' component selected. The main workspace displays a 3D model of a rectangular block with a coordinate system and a scale bar (0.00 to 100.00 mm).





10- Mesh → generation mesh → Mesh

