



Chapter three

Example 3 :

Parallel Flow Heat Exchanger **(fluid-solid conjugate heat transfer)**

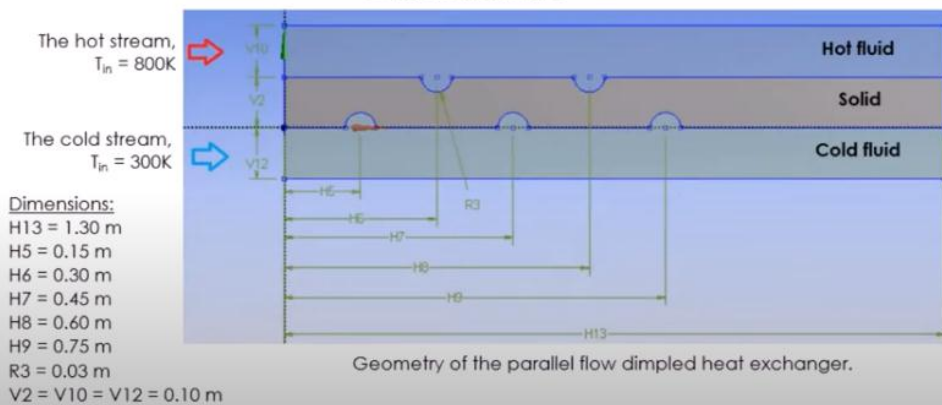
Outcome:

1. Learn 'Face Split' in DesignModeler
2. Learn on energy equation activation (for heat transfer)
3. Learn on more advanced meshing procedure
4. Learn on setting up the heat transfer involving both solid and fluid.

Purpose of study:

To determine the outlet temperatures of both cold and hot streams in the heat exchanger.

DESIGNMODELER Dimensions



SETUP

Fluid properties:

$\rho = 1000 \text{ kg/m}^3$
 $\mu = 0.15 \text{ Ns/m}^2$
 $k = 10 \text{ W/mK}$
 $C_p = 25 \text{ J/kaK}$

Solid property:

$\rho = 8000 \text{ kg/m}^3$
 $k = 50 \text{ W/mK}$
 $C_p = 500 \text{ J/kaK}$

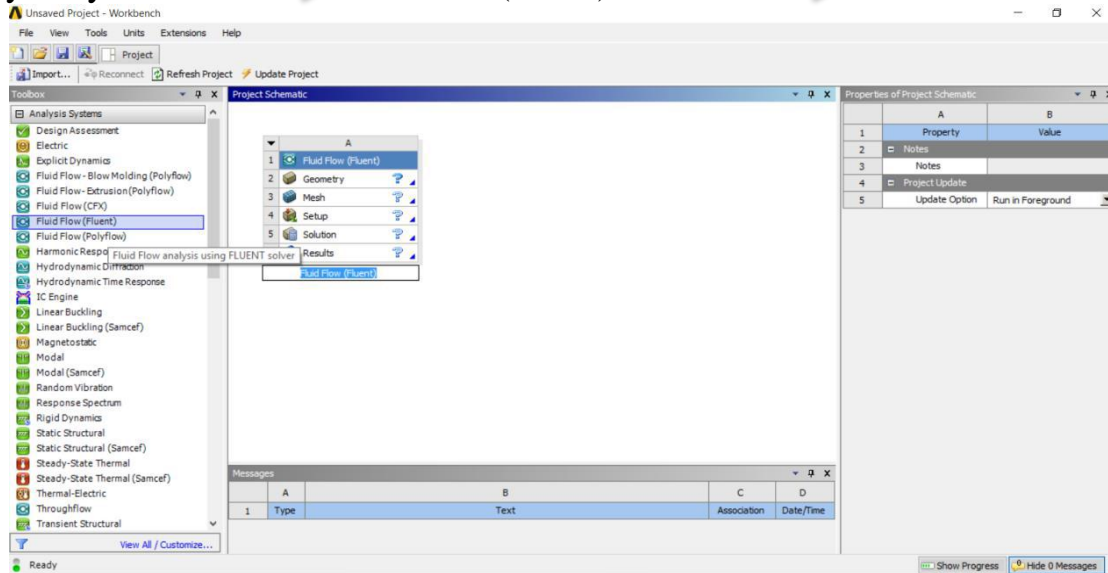
Boundary condition:

Inlet velocities for both hot and cold flow streams: 0.1 m/s
 Treat the flow as laminar.
 $T_{hot} = 800 \text{ K}$
 $T_{cold} = 300 \text{ K}$

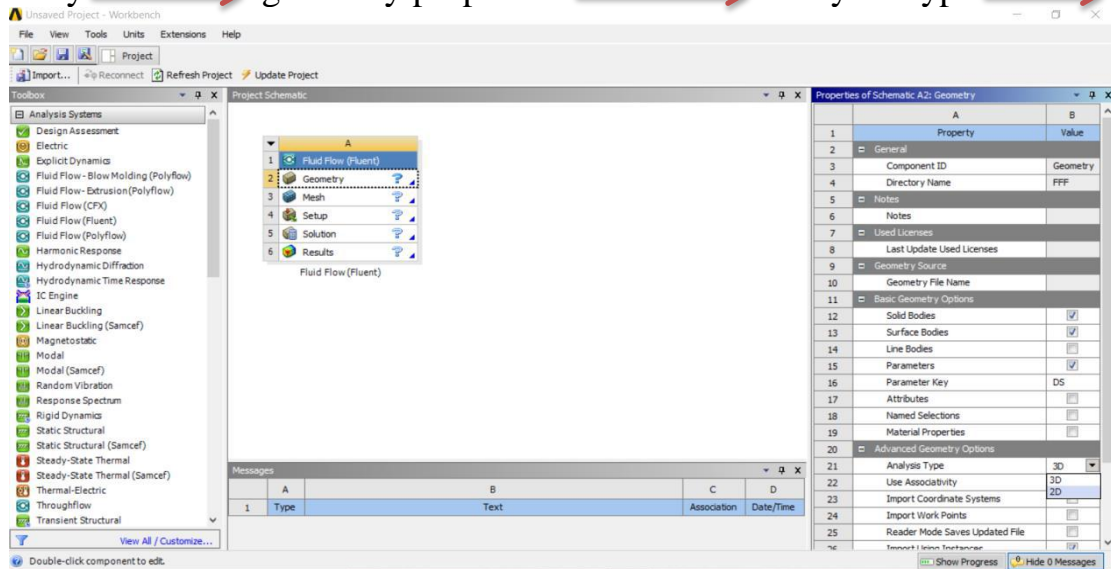


Geometry

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

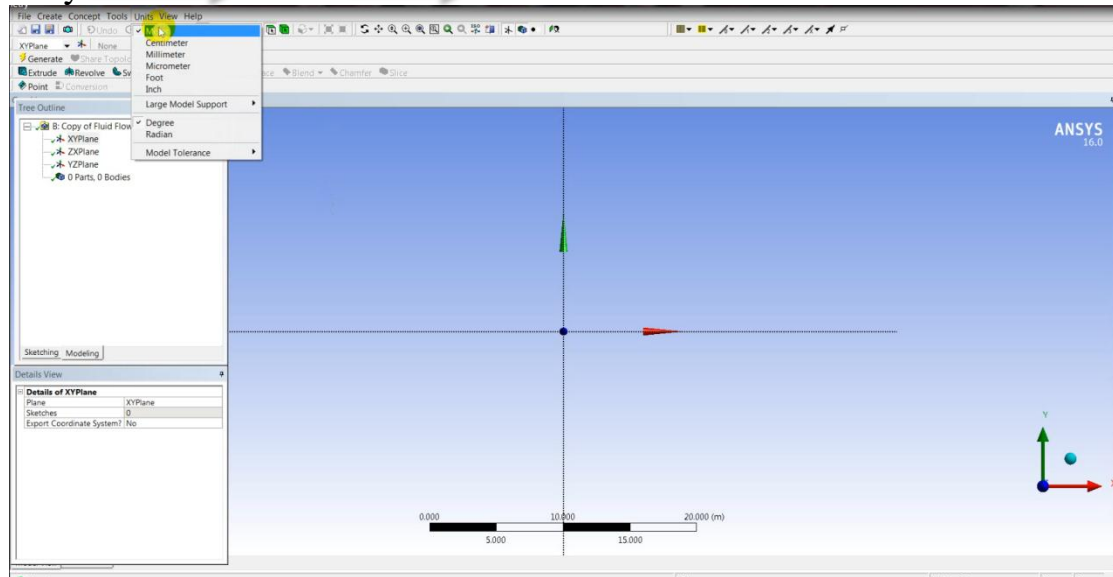


2- Geometry → geometry properties → Analysis Type → 2D

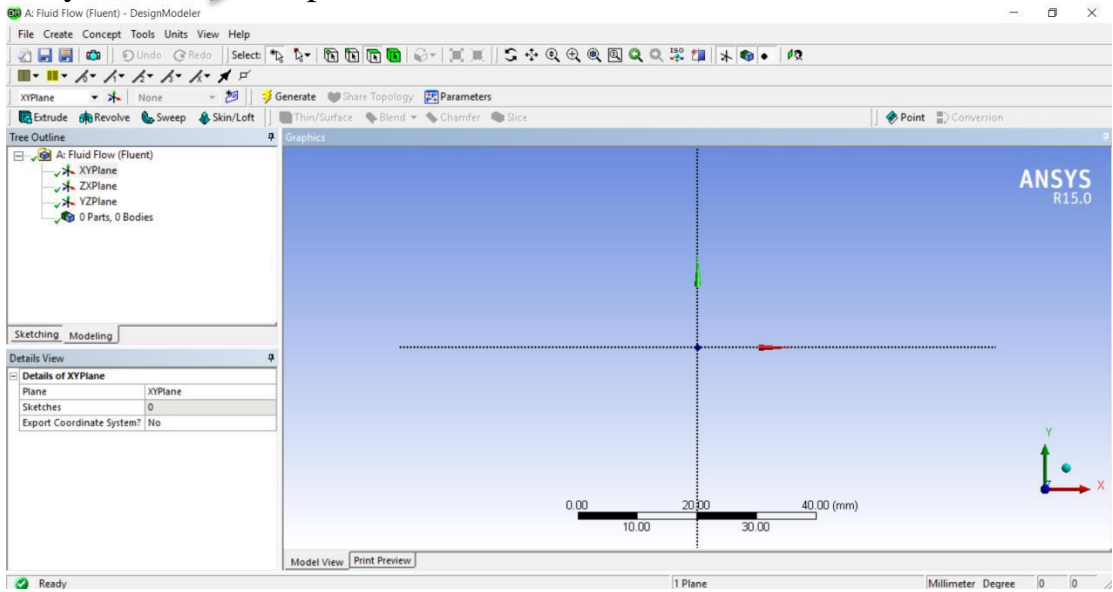




3- Geometry \longrightarrow unites \longrightarrow meters

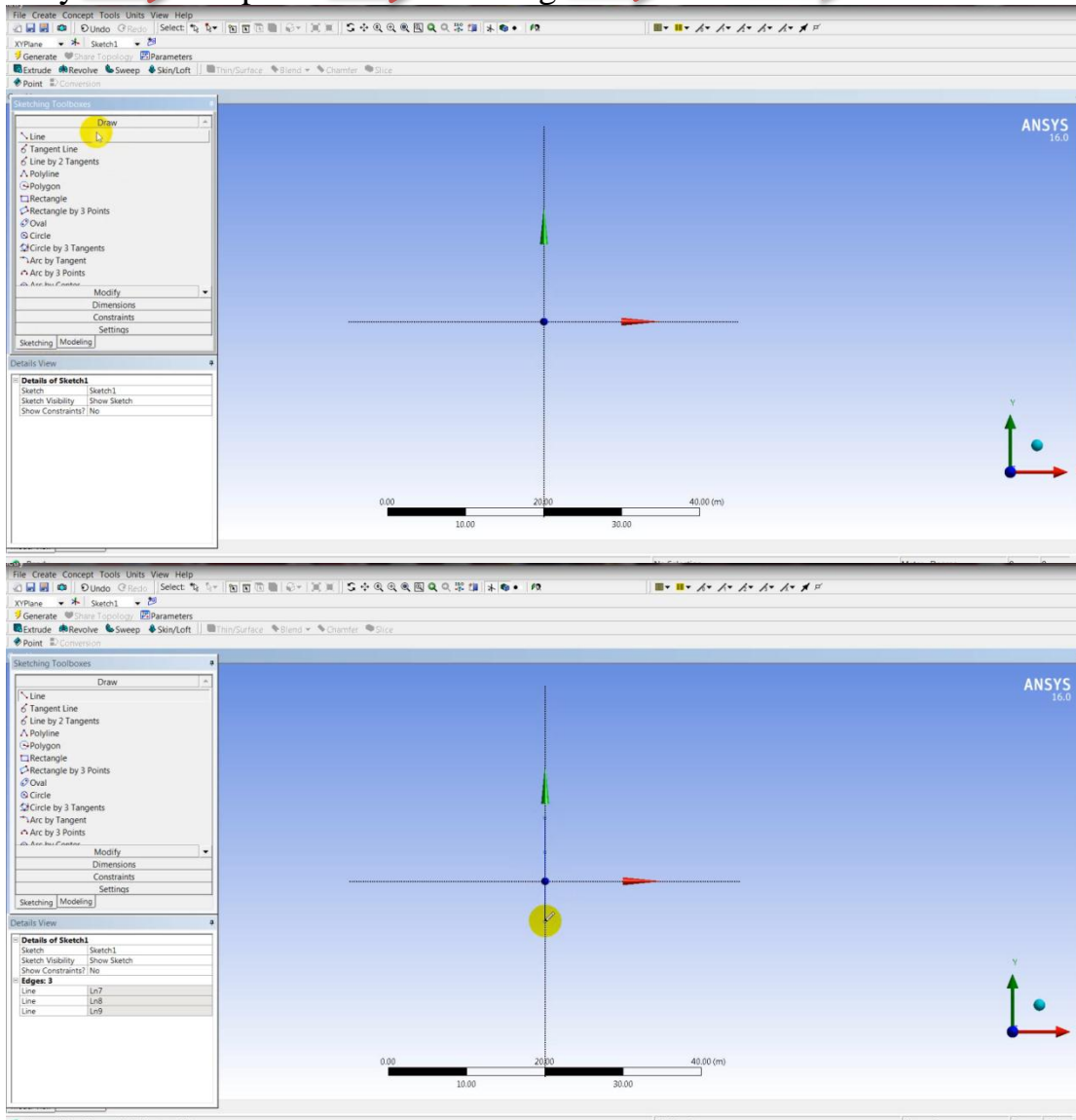


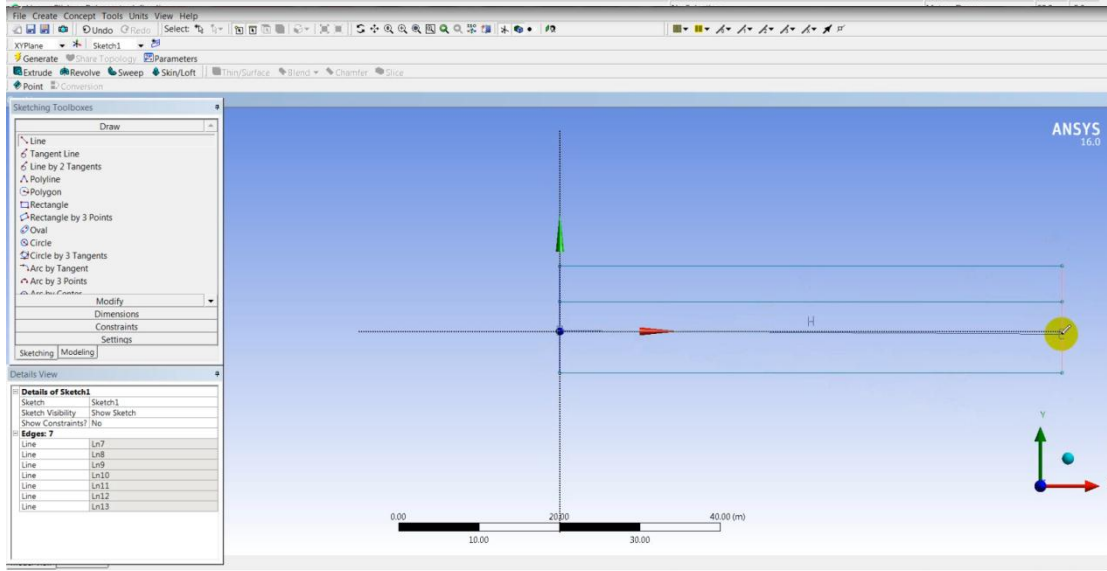
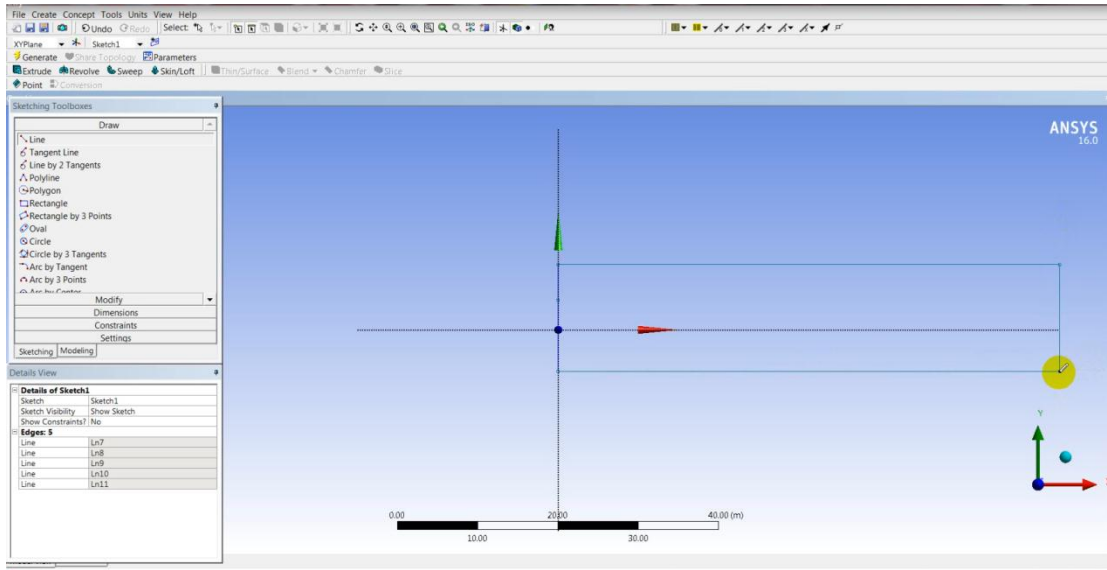
4- Geometry \longrightarrow XY-plane



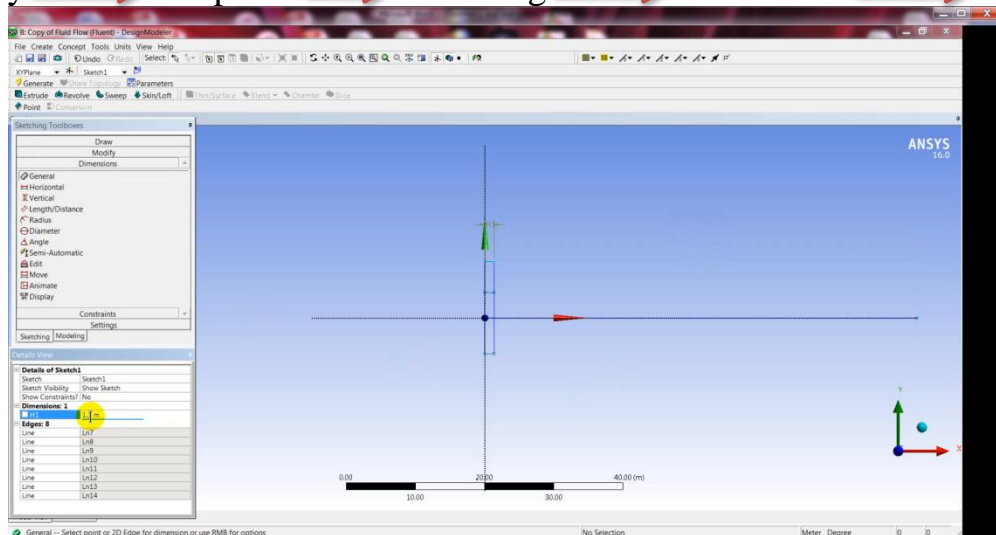


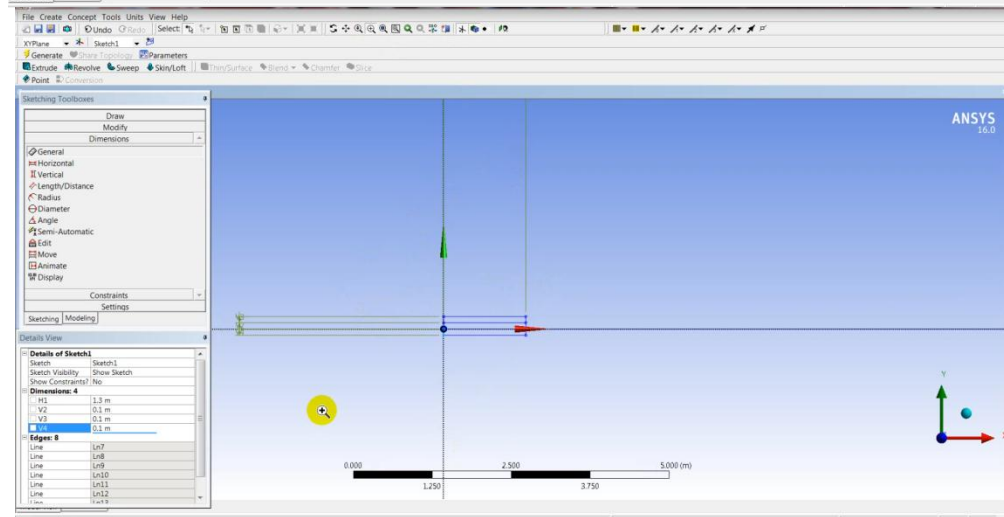
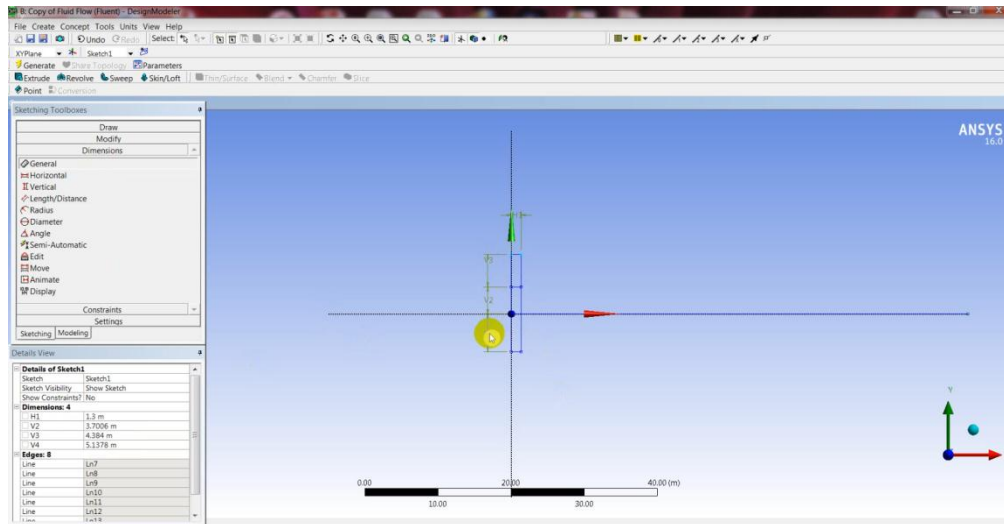
5- Geometry → XY-plane → Sketching → Draw → Line



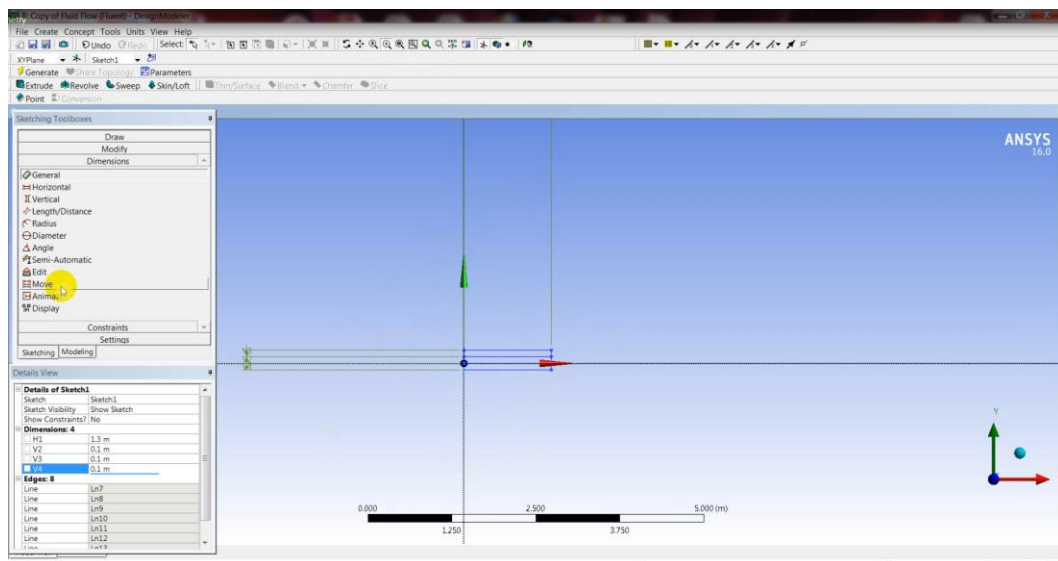


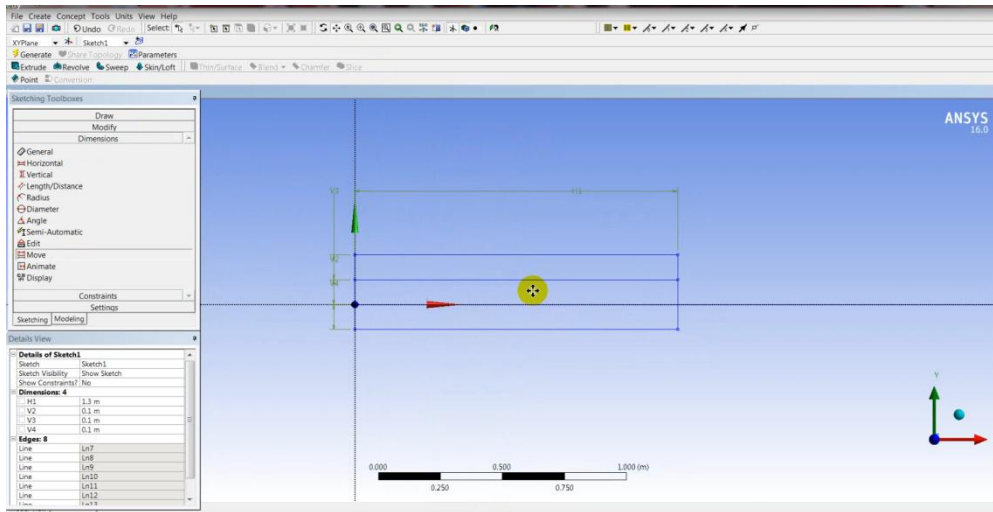
6- Geometry \longrightarrow XY-plane \longrightarrow Sketching \longrightarrow Dimensions \longrightarrow General



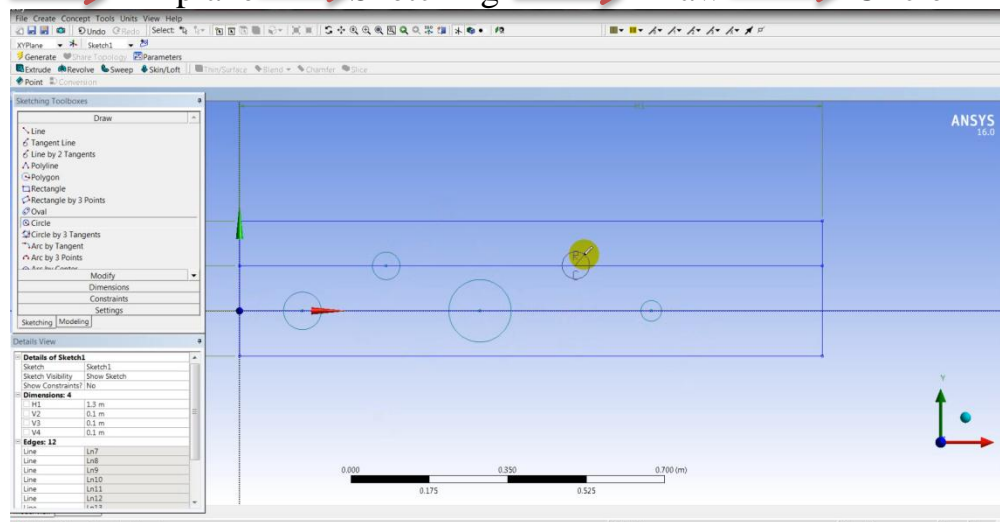


7- Dimensions → Move

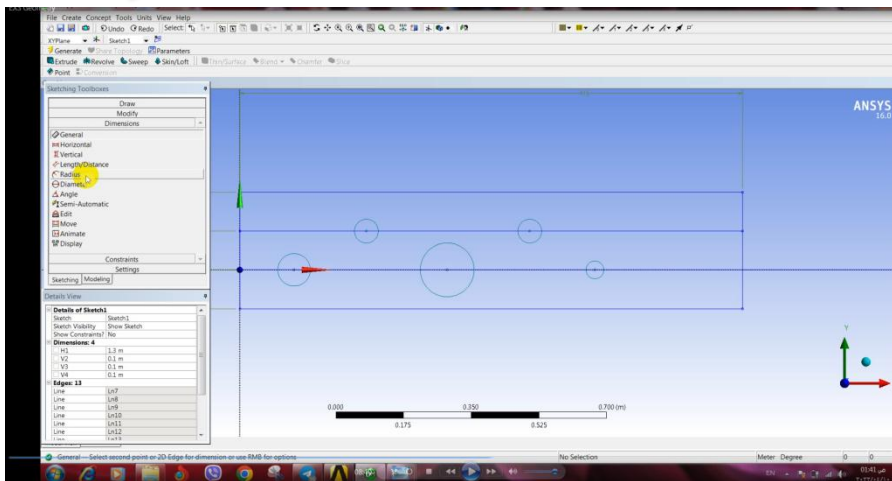


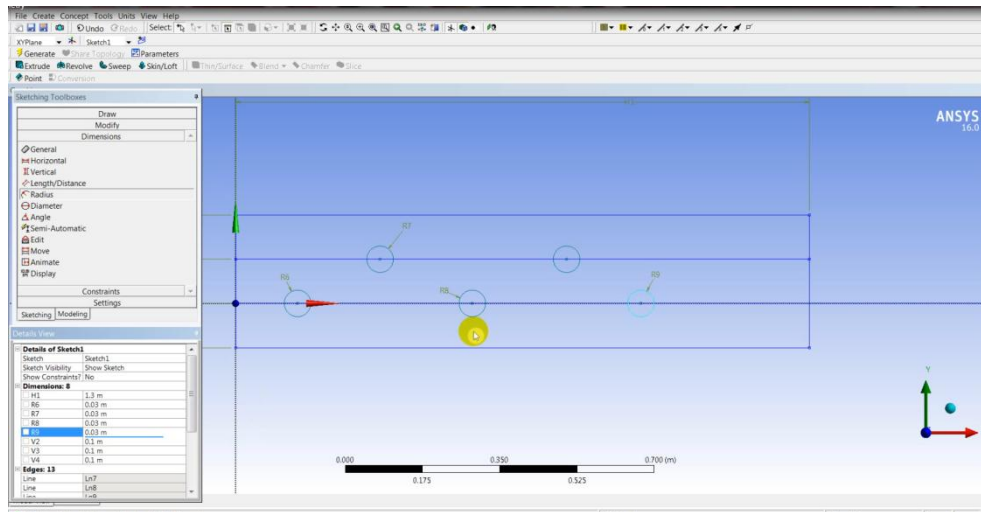


8- Geometry \longrightarrow XY-plane \longrightarrow Sketching \longrightarrow Draw \longrightarrow Circle

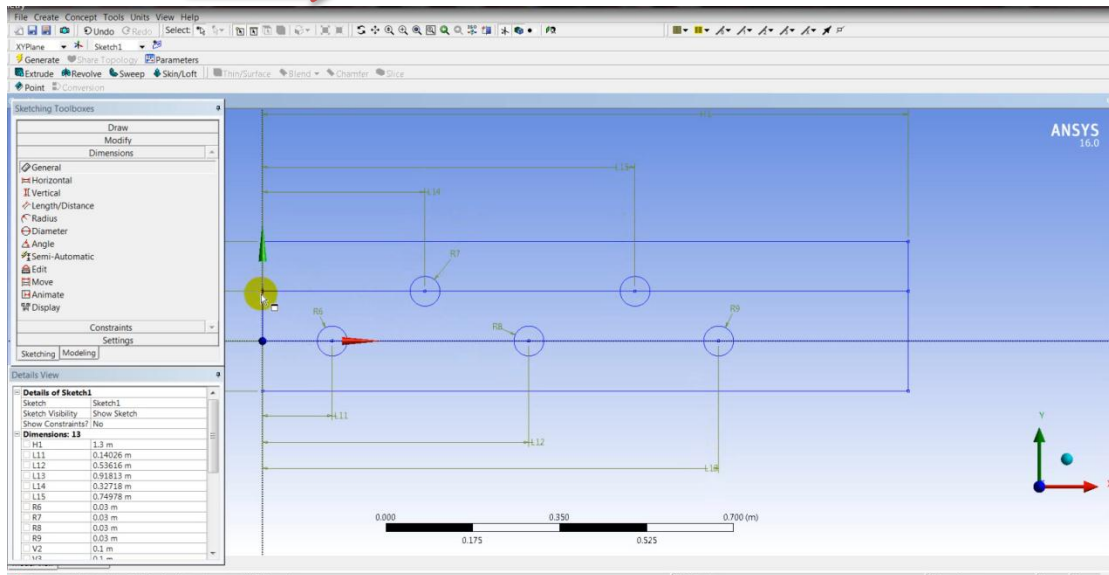


9- Dimensions \longrightarrow Radius

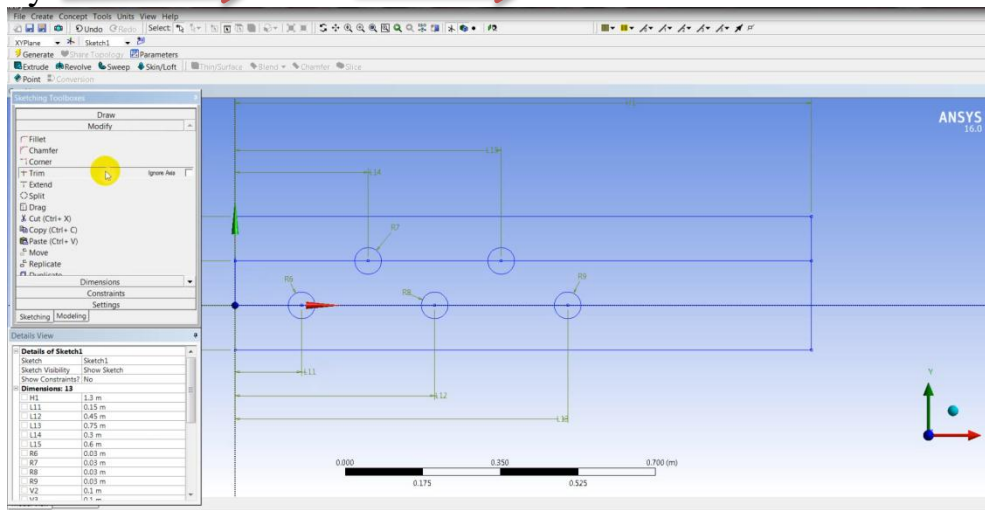


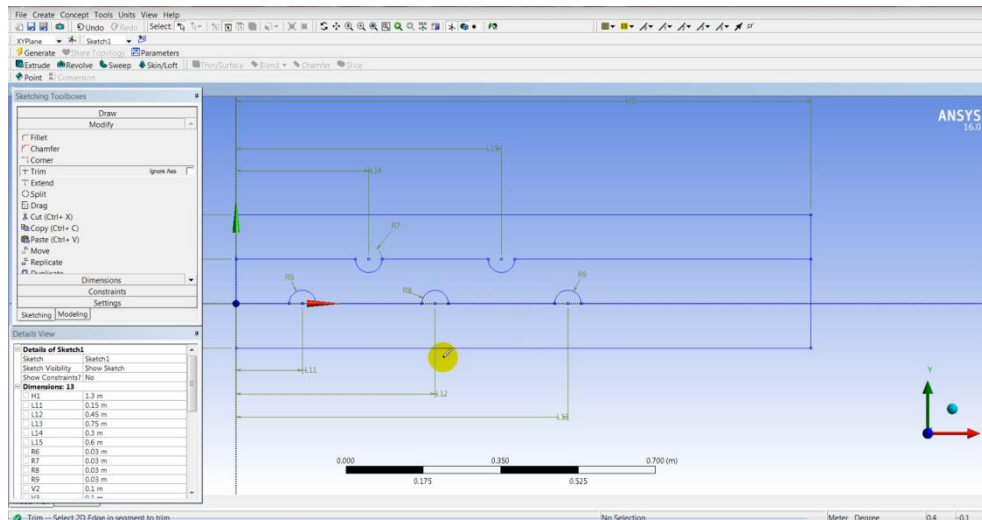


10- Dimension → General

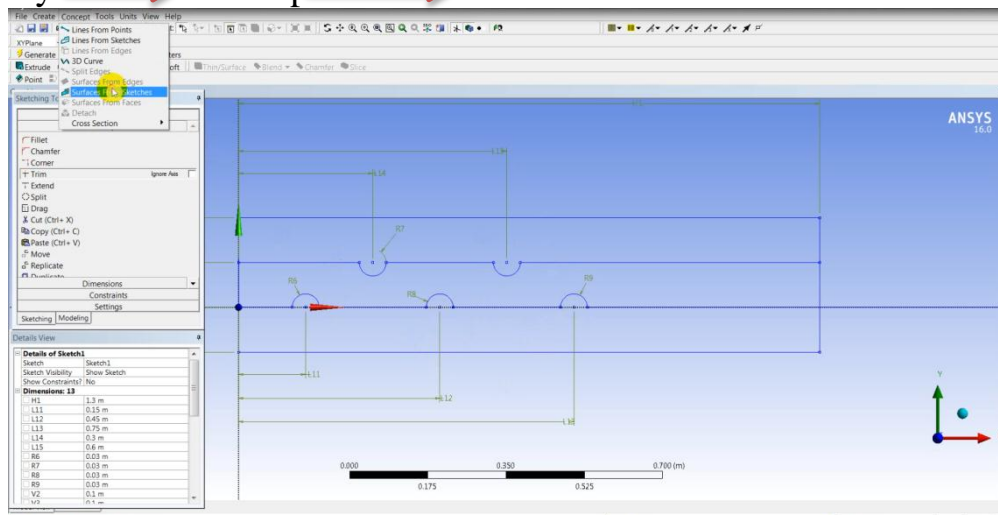


11- Modify → Trim → circle.

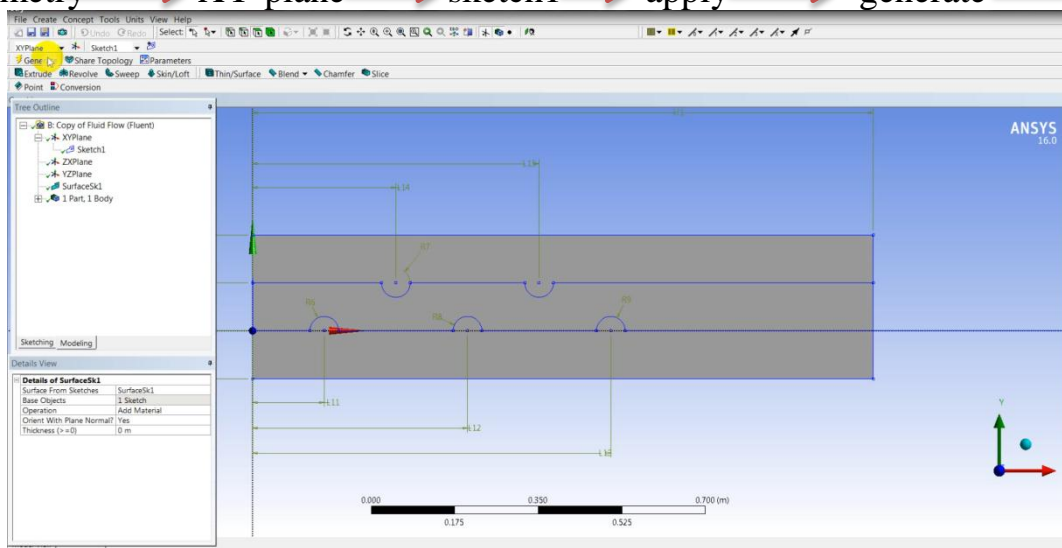




12 - Geometry → concept → surface from sketches

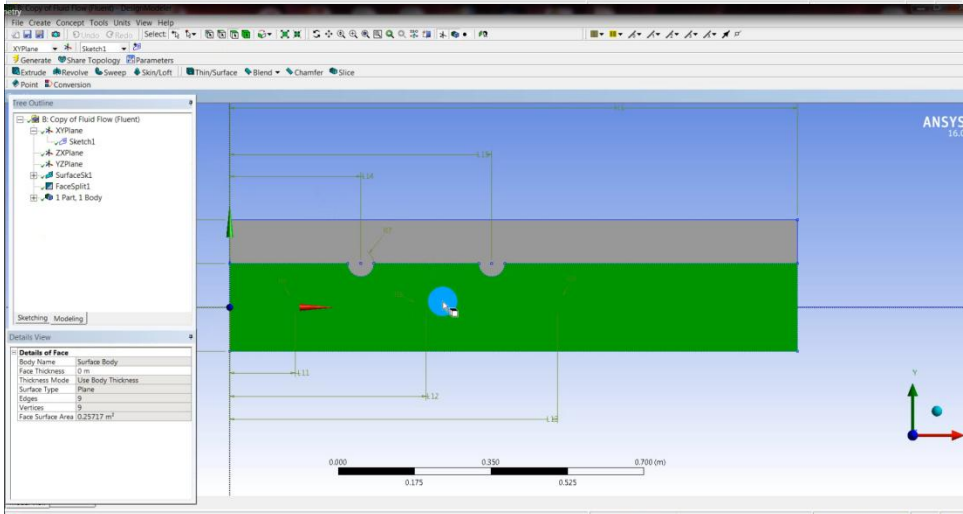
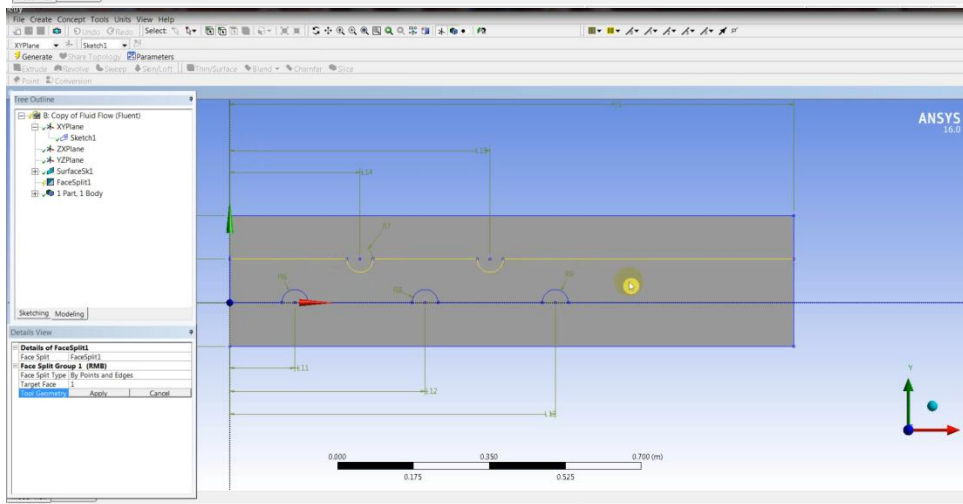
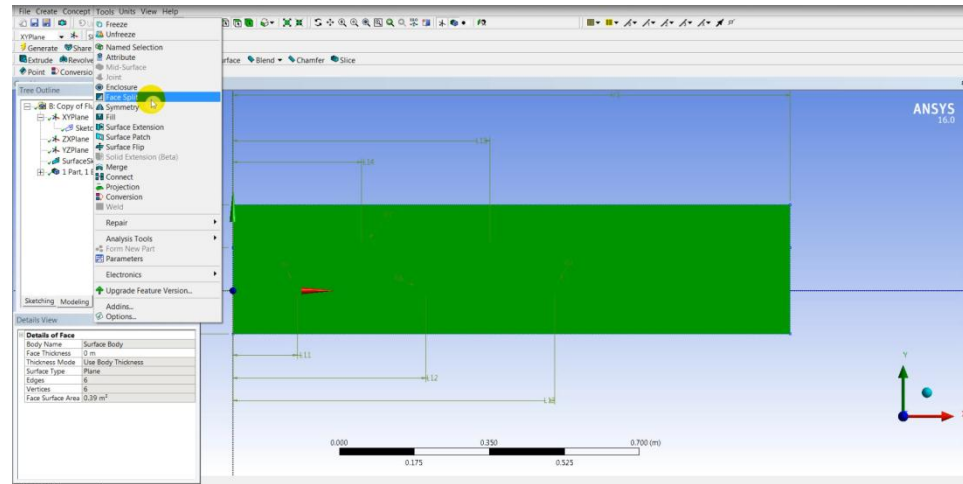


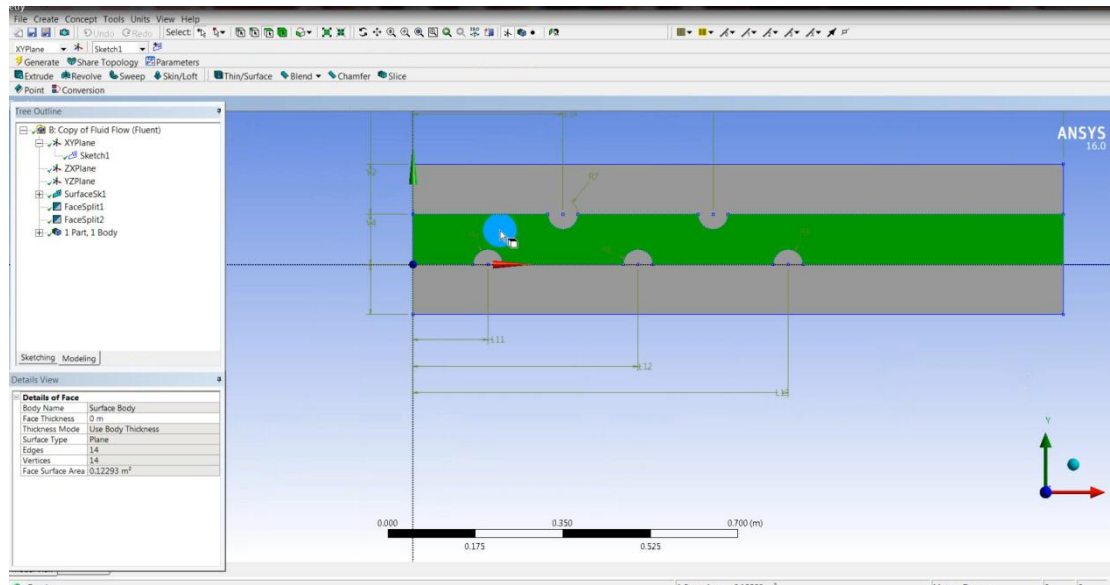
13 - Geometry → XY-plane → sketch1 → apply → generate → close





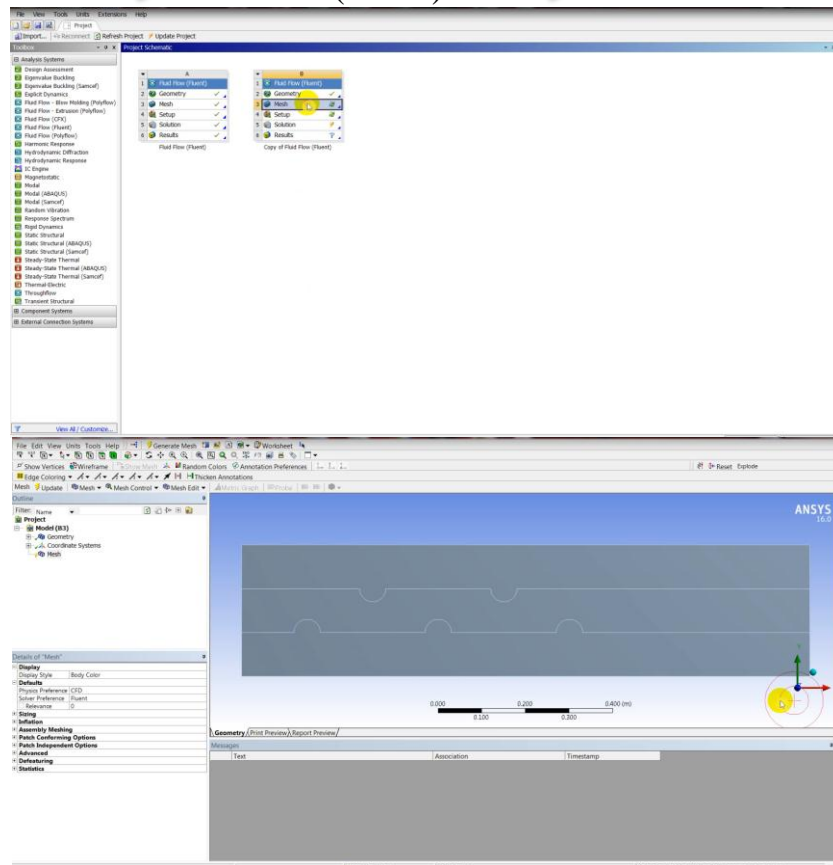
14- Geometry → Tools → Face Split → Select geometry
Apply → Select upper line → Apply → Generate. And return
same procedure for lower line





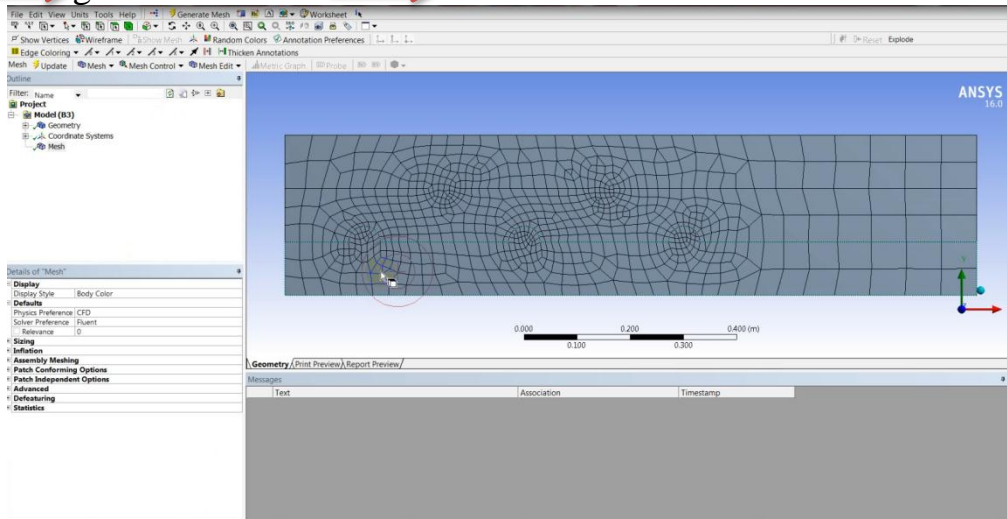
Mesh

1. Analysis systems → fluid flow (fluent) → Mesh

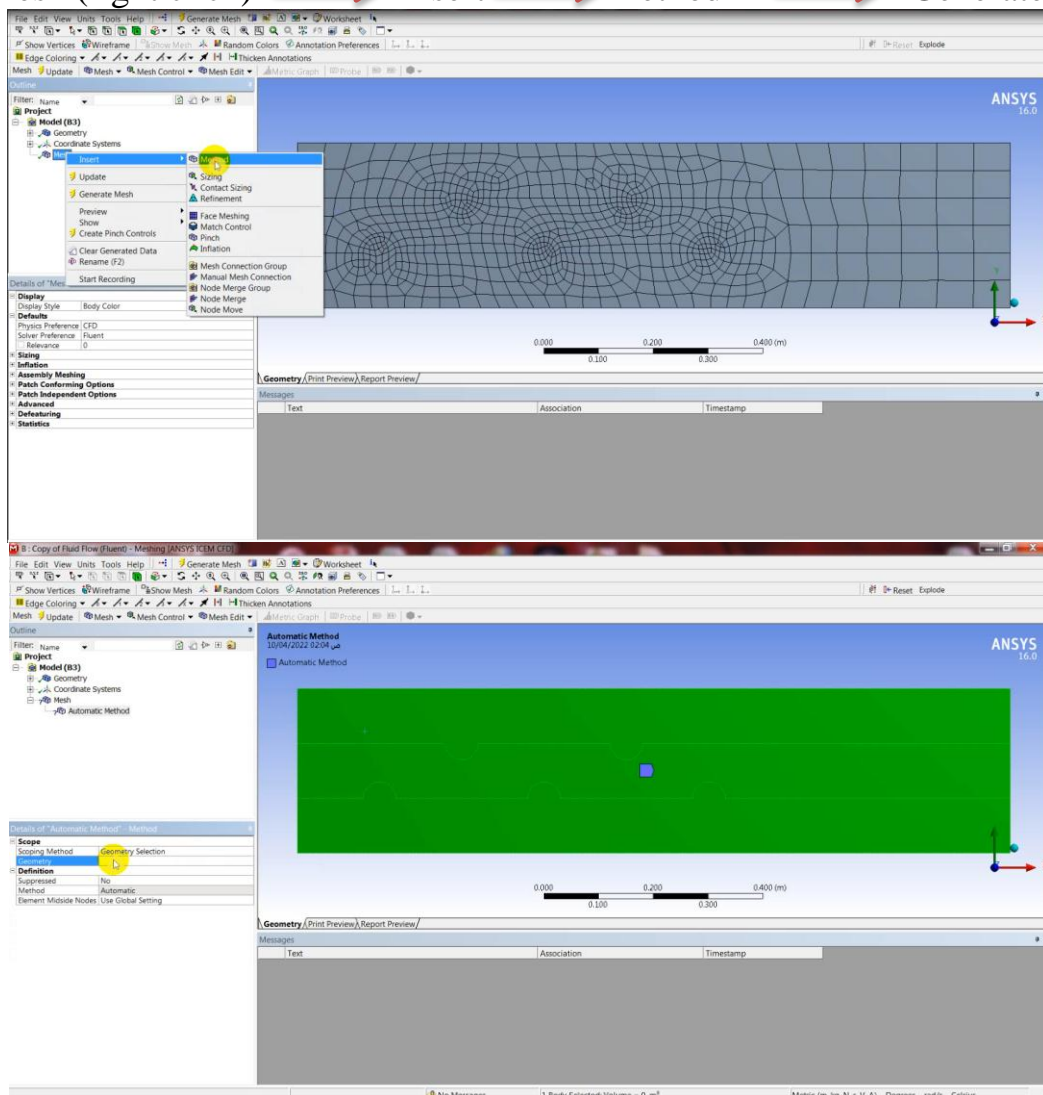




2. Mesh → generation mesh → Mesh

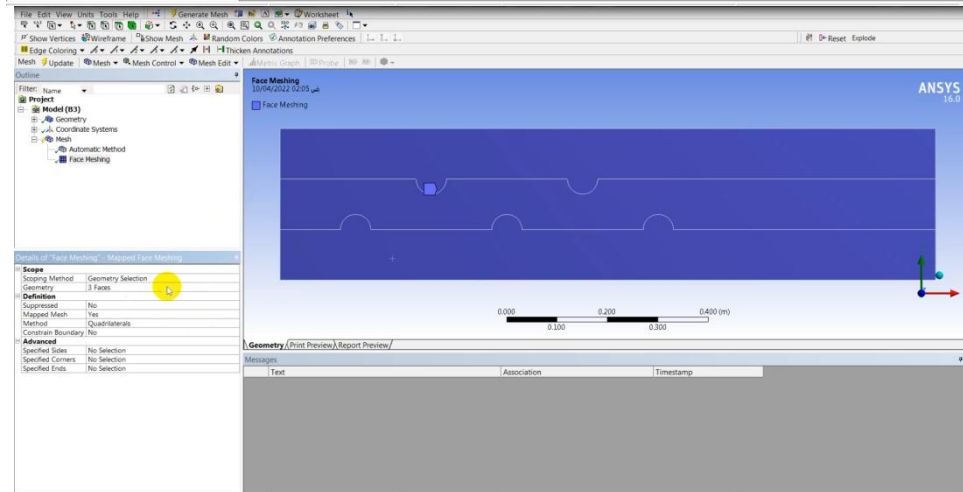
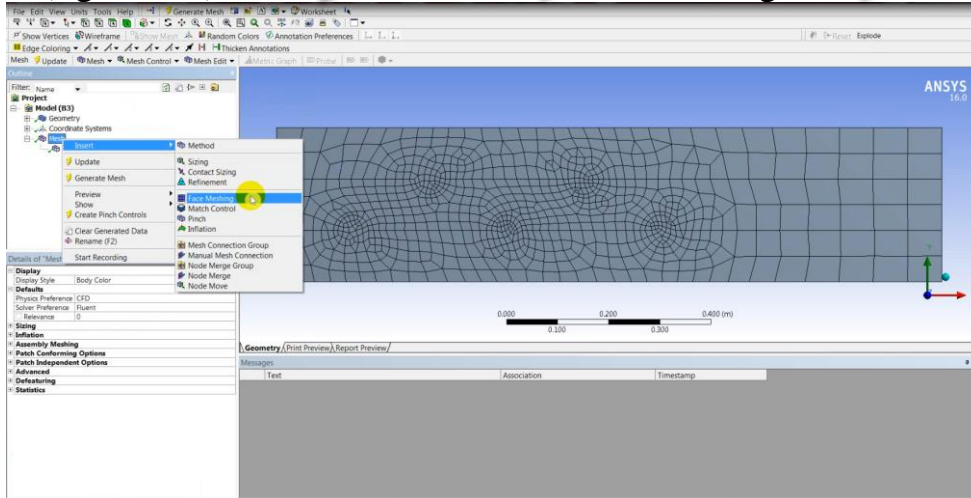


3. Select mesh (right click) → insert → Method → Generate

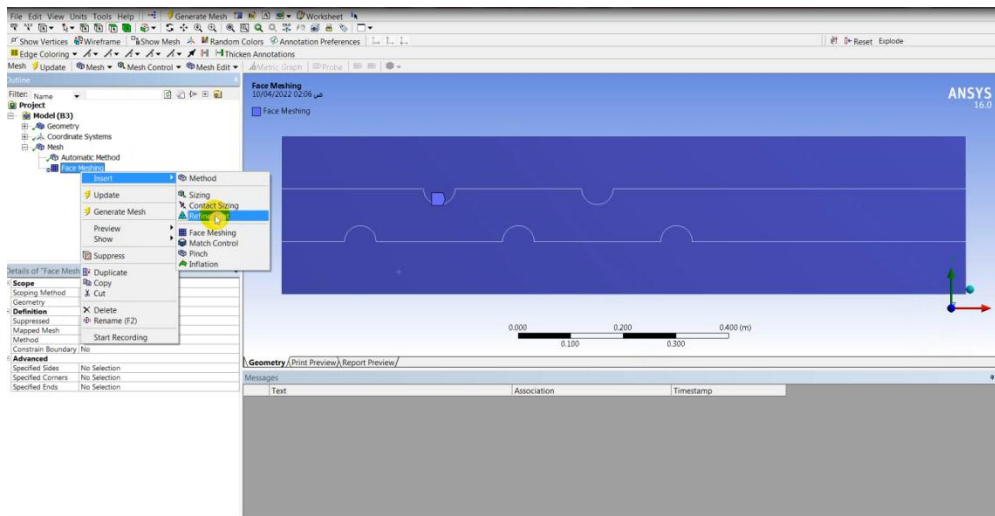


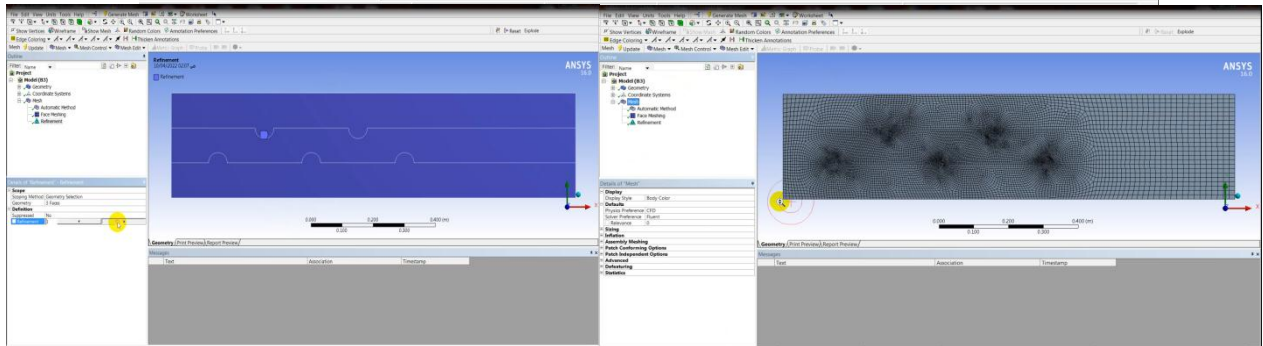
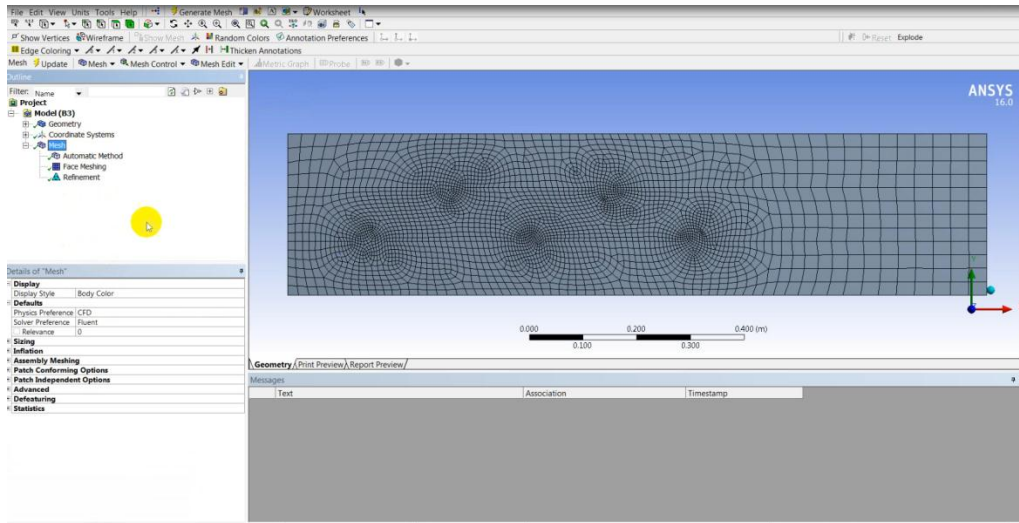


4. Select mesh (right click) → insert → Face meshing → Generate

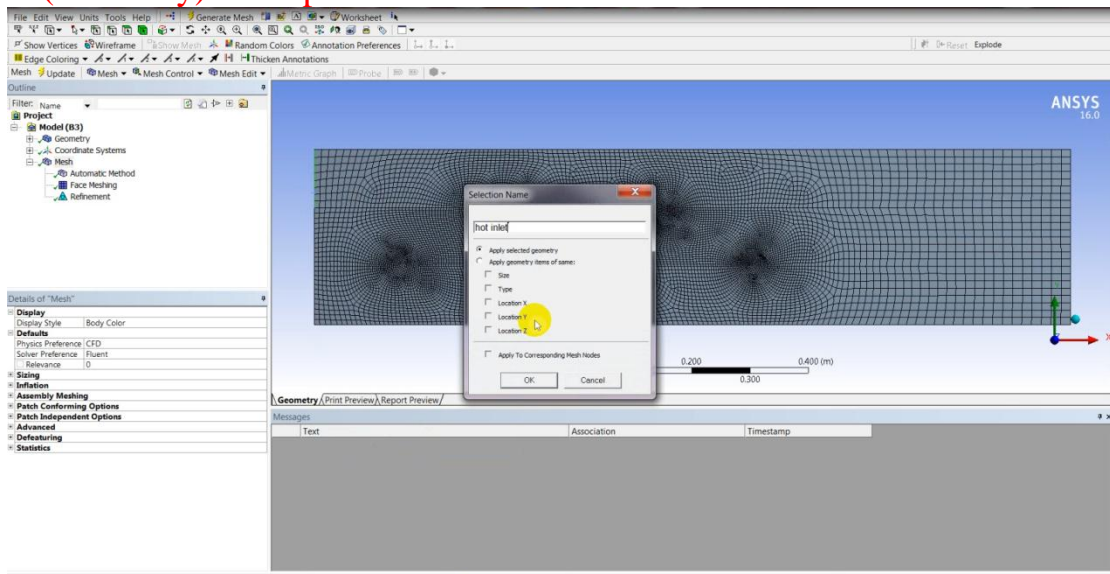


5. Select Face meshing (right click) → insert → Refinement → Generate



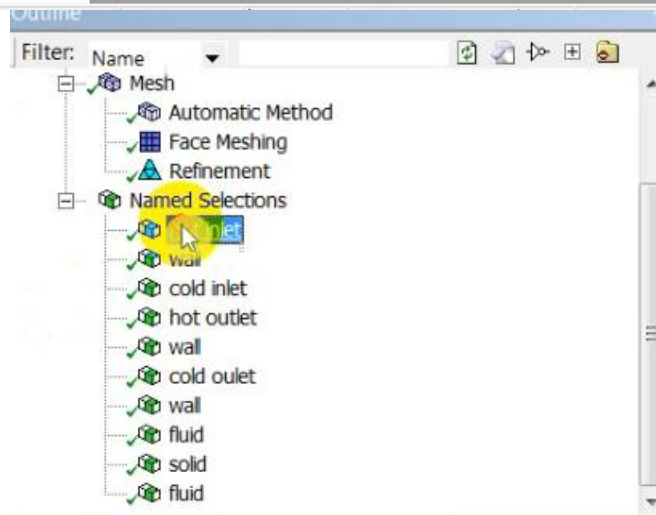
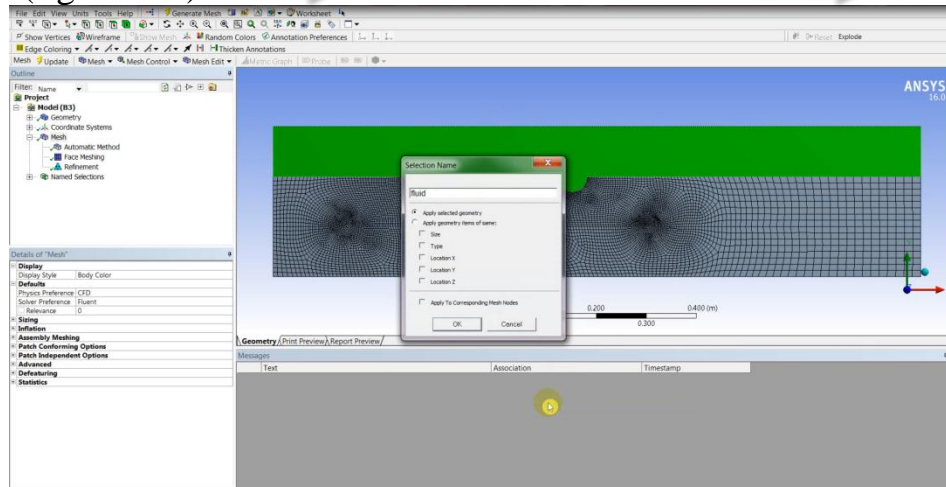


6. Select Boundary (right click) → create named selection → ok
All line (boundary) same procedure

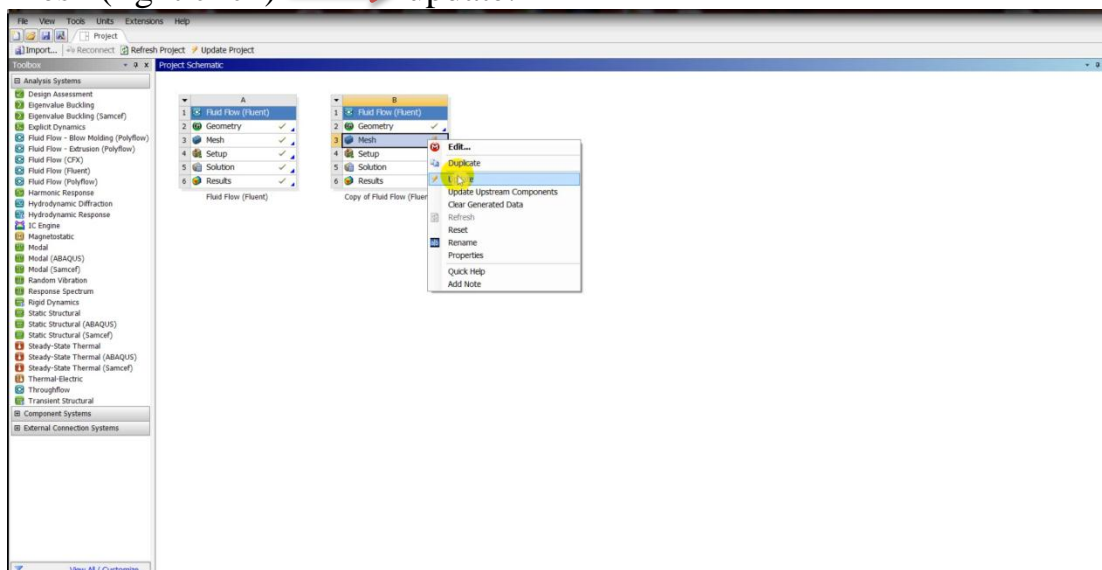




7. Select fluid (right click) → Create named selection → ok



8. Mesh (right click) → update.





Class: third Stage
Lecturer: Wurood yassin Mohsin
E-mail: wurood.yassin@mustaqbal-college.edu.iq

