ME/AE 5212

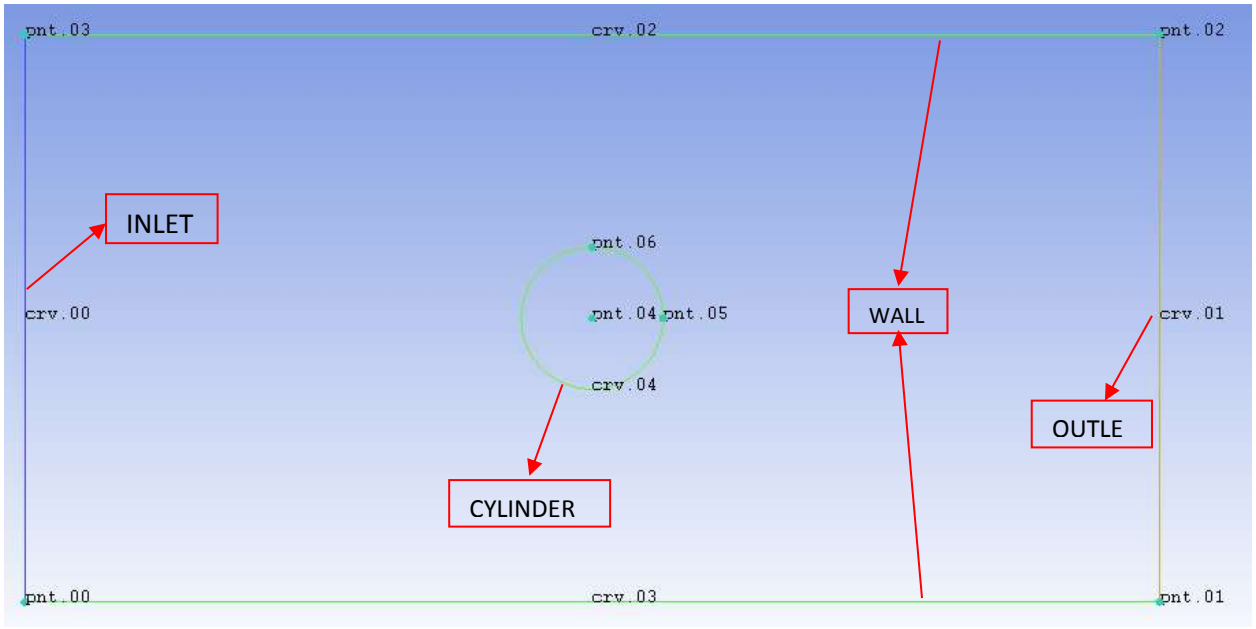
Computer Project Using ANSYS

Hint for Fluid Problem:

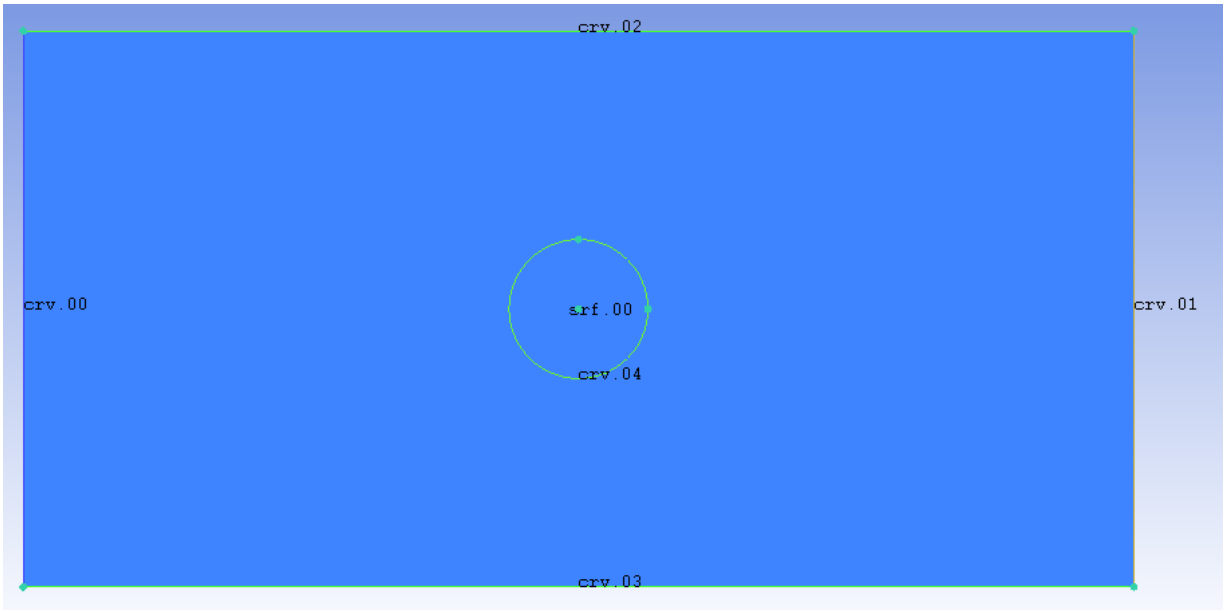
Apply the following settings before you start the problem.

- Settings⟶Selection⟶Auto pick mode should be turned OFF.
- Settings⟶Geometry Options⟶Name new geometry must be turned ON.
- Settings⟶Geometry Options⟶Replace same name item must be turned ON.
- Settings⟶Geometry Options⟶Inherit Part Name⟶Create new must be toggled ON.
- Settings⟶Geometry Options⟶Create surface topology must be turned OFF.

1. Create 7 points and then 5 curves as shown below. Curve lengths and dimensions are given in the problem.



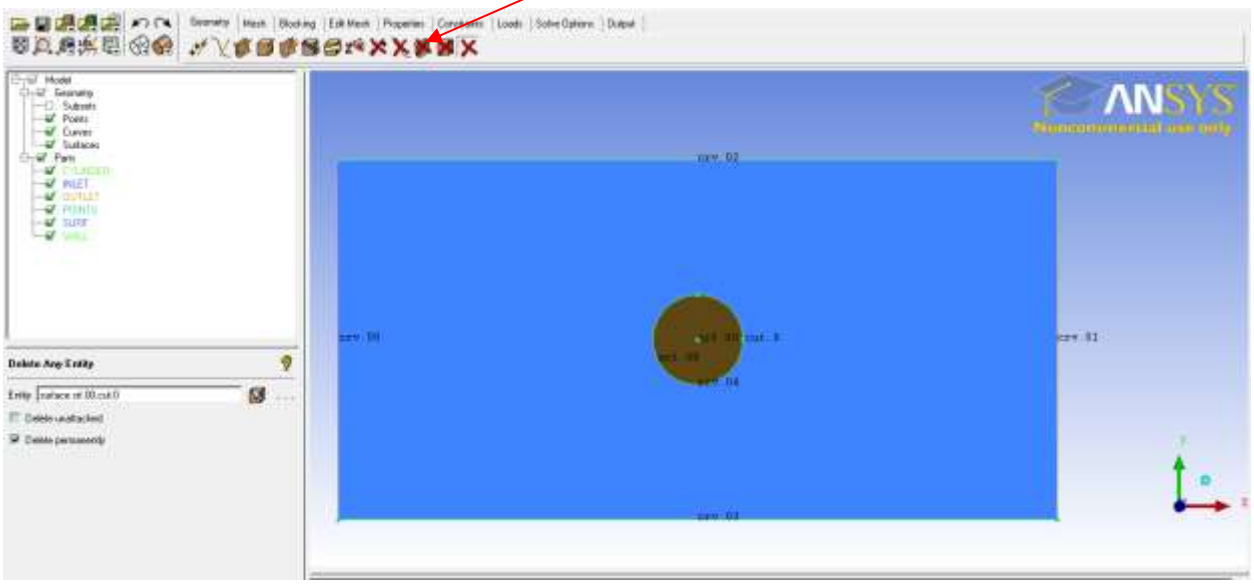
2. Create a surface using curves crv.00, crv.01, crv.02 and crv.03 as shown in the below figure



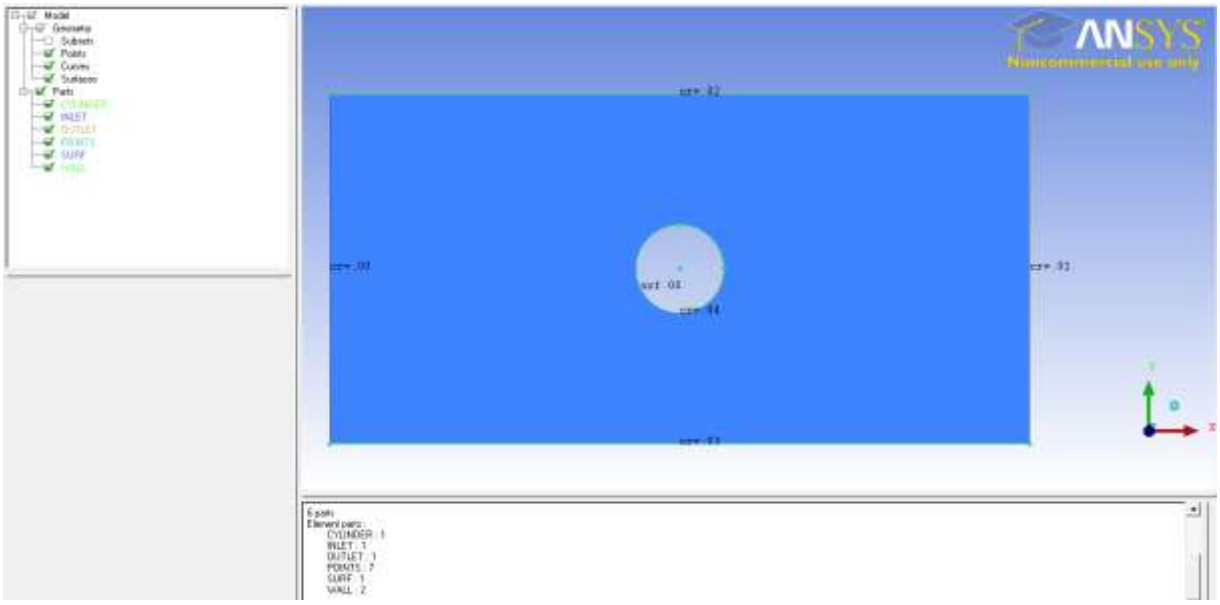
3. Segment the surface using the "Segment/Trim surface by curve" option. The surface corresponding to the cylinder must be cut out from the previously created surface **SURF** (**srf.00**). The segment surface option is shown below

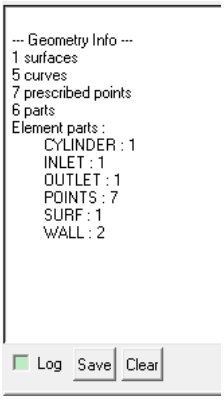
Select surface **srf.00** and the curve **crv.04** which is the outline of the cylinder, click OK.

4. Delete the cylinder surface that is created using the **Delete Surface** button on the toolbar.



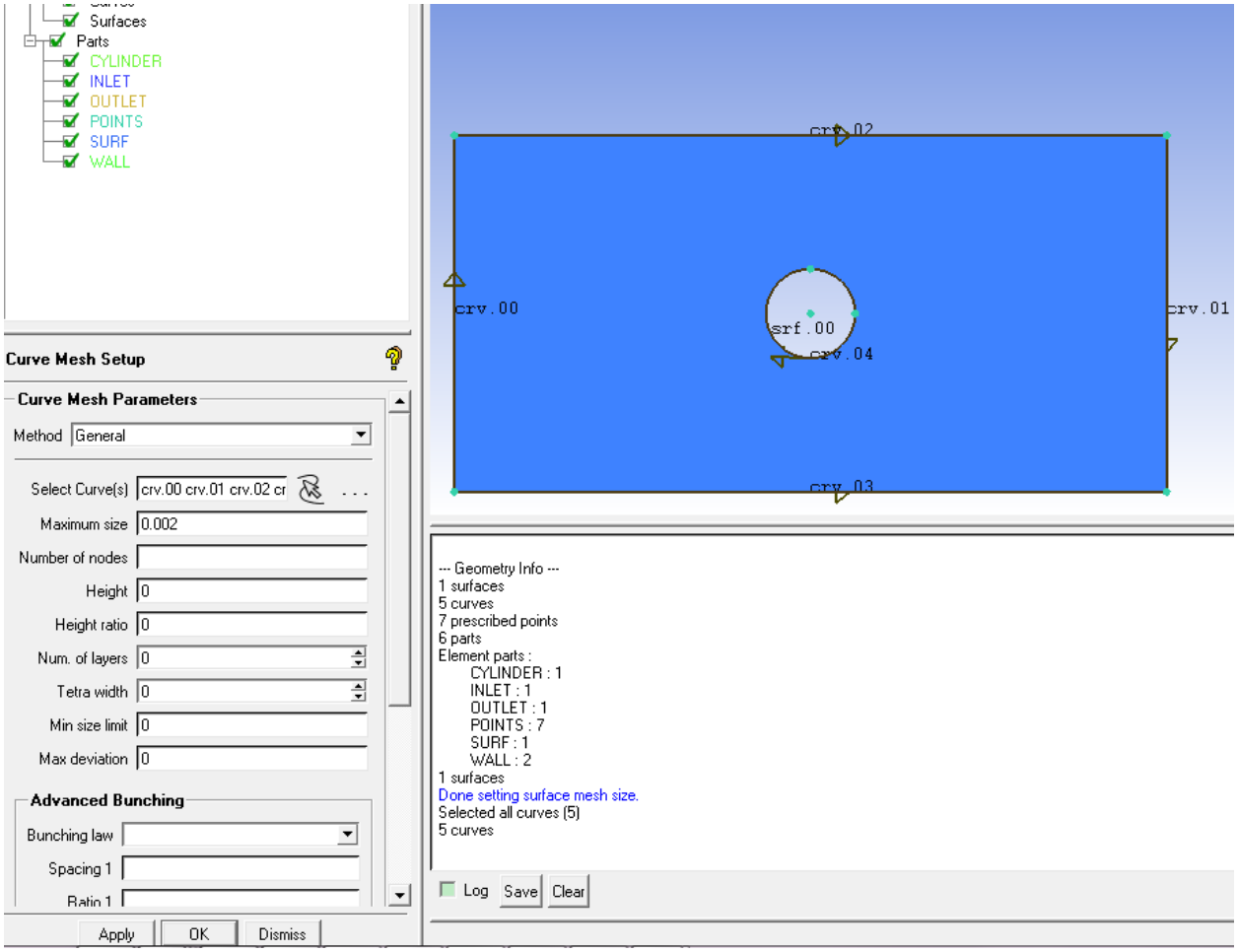
After deletion, the figure is shown as below. You can go to menu Info ⟶ Geometry information, the following geometry information should show up:





5. Use ‘Surface mesh setup’ to set max surface mesh size.

6. Use ‘Curve mesh setup’ to set curve mesh size



7. Mesh the curves using ‘Mesh curve’ tool, and then computer the mesh using triangular element and patch **INDEPENDENT** method, and then output the mesh.

Note: you can use finer mesh around cylinder to study drag phenomenon near wall effects.