FLUENT STEP-by-STEP Instructions for CFD Project (MEMS1071)

1. Start ANSYS, double click the Fluent package and create a new project of Fluent. You can change the project name if you need.

2. Define the geometry.
   a. Select 'Units' on the top, make sure 'Meter' is used.
   b. Select 'Z' axis to change your view to 'XY' plane.
   c. Set the coordinate. Click 'Sketching' an 'Settings'. Click 'Grid', then check 'Show in 2D' and 'Snap'. It will help snap the geometry to the grid. Click 'Major Grid Spacing', and input '5'. Click 'Minor-Steps per Major', and input '5'. It will set the major grid spacing as 5 meters, and minors as 1 meter.
   d. Draw the geometry. Click 'Sketching' and 'Draw'. Select 'Rectangle'. Draw a rectangle from the origin point with any size.
   e. Change the dimension of your geometry. Select 'Dimension' then 'General'. Click the edge of the rectangle you want to change then drag it out. You will be able to input the value in the window of 'Details View', 'Dimensions'.
   f. Generate the Calculation Domain. Select 'Modeling'. Expand the 'XY Plane' in 'Tree Outline' window. Click 'Concept' on the top then select 'Surfaces From Sketches'. Click 'Sketch 1' which you just expanded. Then click 'Apply' in the 'Details View' window. Click 'Generate' on the top. Now the color of your surface becomes gray which means surface generation is successful.
   g. Close the window. There will be a green check just behind 'Geometry' which means this step is successful.

3. Create the mesh.
   a. Double click 'mesh'. Fluent will upload the geometry automatically. Click 'Units' on the top, make sure metric unit is used as 'm, kg, N, s, V, A'.
   b. Click 'Z' axis to make it to 'XY' plane. Click 'Generate Mesh' icon on the top to make a very course mesh. You can check your mesh by clicking the 'Mesh' in the 'Outline' window.
   c. Refine the Mesh. Right click 'Mesh' in the 'Outline' window and insert the 'Sizing'. Select your surface by the 'Face' icon on the top. If this does not work, you can use the icon of 'Select Mode' then 'Box Select' to choose the whole geometry. Then click the 'No Selection' which has a yellow color. Then Click 'Apply'. The 'face sizing' function will be added. Click the 'Default' behind 'Element Size' in the window of 'Details of 'Face Sizing'' and make it to the size which you want. The size will be the edge length of your element. Notice about the unit. Then click 'Update' or 'Generate Mesh' on the top. Fine mesh will be generated. You can check the mesh by clicking 'Mesh' in the 'Outline' window.
   d. Refine the mesh on the boundary—two walls. Right click the 'Mesh' then insert 'Inflation'. Select the whole geometry first, as did in the previous step. Then click the 'Apply' just behind 'Geometry'. Select the upper and lower edges by the 'Edge' icon on the top, or use the 'Select Mode' and 'Single Select'. Press 'Ctrl' if necessary, it will help you select more than one object. Then click 'No selection' behind the 'Boundary' in the 'Details of 'Inflation''

window, then 'Apply'. The 'Inflation' will be added to the edge you just selected. Change the number of 'Maximum Layers' and 'Growth Rate' as you want. Then click 'Update' or 'Generate Mesh'.

e. Name the edges. Select a single edge of your geometry as you did in previous step and right click. Use 'Create Named Selection', name your edges as 'wall 1', 'wall 2', 'inlet' and 'outlet' respectively. You can check them in the 'Named Selections' of 'Outline' window. After checking your mesh, close the window. Go to the project window, if there is no green check after the 'Mesh' of you project, right click the 'Mesh', then 'Update'.

4. Calculation. Double slick 'Setup' in the project and use 'Double Precision'. Program will upload geometry and mesh automatically.

a. Add another fluid. The project will be steady state laminar flow and pressure-based. Therefore, go to the 'Materials' directly. Click 'Fluid' and 'Create/Edit'. Go to the 'Fluent Database' and choose 'water-liquid' then 'Copy'. Water will be added into 'Fluid' category.

b. Click 'Cell Zone Conditions'. Then 'Edit' the 'surface_body'. Choose 'water-liquid' for the 'Material Name' then 'OK'. Now, the program will calculate the fluid for your geometry as 'water' instead of the defaulted 'air'. If you want, you can go to the previous 'Materials' to delete the 'air' under 'Fluid' category.

c. Define boundary conditions. Click the 'Boundary conditions' and 'inlet' for example. Choose the 'Type' as 'velocity-inlet'. Then click 'Edit', then input the velocity Magnitude based on your calculation from known Reynolds Number. The positive means this boundary will be an inlet—fluid will come into your geometry from this edge. Vice versa for the negative value. The 'Gauge Pressure' will be '0' if your simulation is under atmosphere. Then click 'OK'. Define the rest of your boundaries in the same way. The 'outlet' will be 'pressure-outlet' under 1 atm and both of 'wall 1' and 'wall 2' should be the 'wall' type.

d. Define the convergence criteria. Click 'Monitors' and select 'Residuals' then 'Edit'. Input the 'Absolute Criteria' as you want. Then 'OK'.

e. Initialization. Click 'Solution Initialization' then use 'Standard Initialization'. Choose the 'compute from' 'inlet'. The inlet will be initialized as the velocity you defined before. Positive value means in the positive 'X' direction and vice versa. Then click 'Initialize'. Except the inlet, other area will be initialized as '0' automatically.

f. Calculate. Click 'Run Calculation' then input the 'Number of Iterations'. The calculation will be stopped either reach the 'absolute criteria' you defined before or reach the 'Number of Iterations' here. Then click 'Calculate'. Watch the three curves in the new window. When the calculation is completed, all of these curves should be stable, otherwise, the solution may not be converged.

5. Check your simulation results.

a. Check the pathline. Click 'Graphics and Animations' then 'Pathlines'. Click 'Set up'. Let the pathlines 'Release from Surfaces' 'inlet'. Then 'Display' if only part of the pathlines is shown, just add the 'Steps'. You can control the number of pathlines by using 'Path Skip'.

b. Check the 'U' velocity profile. Click the 'Set up' of 'Graphics'. Then 'New Surface' and 'Line/Rake'. Input the coordinate of two points and the cross setion will be generated based on this straight line. In your project, if you want to show the profile normal to the walls, 'x'

coordinate of the two points should be the same for one cross section. Input the 'New Surface Name' and click 'Create'. One cross section will be generated and added to the category of 'Release from Surfaces'. Create other cross sections you want to look into. More surfaces should be added to the region near the inlet because of the fluid developing. It could help you observe the velocity profile change. After all the cross sections are made, close the window.

c. Make the velocity profile. Click 'Plots' under the 'Results', then use 'XY Plot' and 'Set up'. Change the 'Plot Direction' into 'X 0' and 'Y 1', because you want to see how the velocity change in 'Y' direction on a cross section. Choose the 'Y Axis Function' as 'Velocity'. If you want to observe 'U' velocity, choose 'X Velocity' underneath. Select to 'Surfaces' you want to observe then 'Plot'. 'U velocity' profiles will be shown. If you want to save the data, check the 'Write to File' then 'Write'. The data file could be open by notepad or other softwares.

d. Fine the average pressure at each cross section. Click 'Reports' then 'Surface Integrals' and 'Set Up'. Choose the 'Report type' as 'Area-Weighted Average'. Select the 'Field Variable' as 'Pressure' and then 'Static Pressure'. Choose the 'Surfaces' you want to see the pressure and 'Compute'. Results will be reported. This data could also be saved by 'Write'.