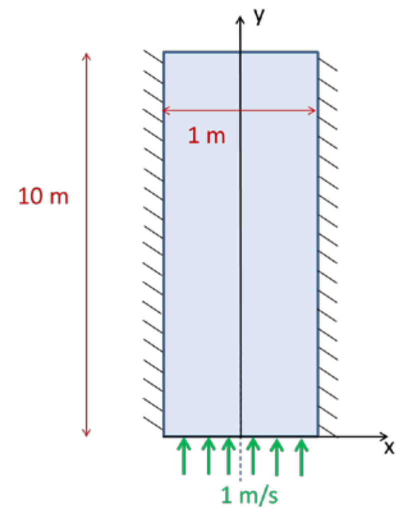
Vertical Channel Flow

Learning Goal

Practice applying the CFD process to a simple flow problem and systematically verifying results. Use the same procedure in more complex CFD problems.

Problem Specification

Consider 2D, incompressible, steady flow in a vertical channel at a Reynolds number of 100.



Channel length *l* and width *w* are 10 m and 1 m, respectively, as shown in the figure. Assume unit thickness in the z-direction. The velocity is constant at the inlet and equal to 1 m/s. The Reynolds number is defined as:

$$Re = \frac{\rho\, \bar{v}\; w}{\mu}$$

$\bar{v}$ is the average velocity at any cross-section. Take $\rho = 1\ kg/m^3$ and adjust μ to get the desired Reynolds number.

You need to obtain the flowfield using ANSYS FLUENT and report the results outlined below. Model only half the 2D domain using symmetry.

1. Pre-Analysis

a. What is the mathematical model (governing equations and boundary conditions) that needs to be solved using ANSYS FLUENT?

b. Perform hand calculations to predict or derive the following:
    - Velocity profile in the fully-developed region
    - Skin friction coefficient (i.e. non-dimensionalized wall shear) in the fully-developed region

    What terms in the Navier-Stokes equations are set to zero in the full-developed region in the above analysis? Note that ANSYS FLUENT will keep these terms everywhere but they will turn out to be negligible in the full-developed region.

2. Numerical Results from FLUENT

Obtain the numerical solution using FLUENT following a process similar to that in the laminar pipe flow tutorial at https://confluence.cornell.edu/x/6YQaBQ. In the mesh, use 100 divisions along the channel (i.e. in the *y*-direction) and 10 divisions across the half-channel.

Present the following results:

a. **Filled contours of the velocity variation in the channel**. Black-and-white printout is fine for all plots. Discuss briefly the velocity variation trends along the channel and normal to the channel. How long does it take for the flow to become fully developed? What is the range of the velocity values?
b. **Filled contours of the pressure variation in the channel**. Discuss briefly the pressure variation trends along the channel and normal to the channel. What is the range of the gauge and absolute pressure values?
c. **A plot of the velocity profile at the outlet**. Add the corresponding profile from your fully-developed hand calculation to this plot. Distinguish between the two curves using a legend. How well does the FLUENT solution in the fully-developed region agree with the analytical solution?
d. **A plot the skin friction coefficient along the wall**. Compare the FLUENT value in the fully-developed region with the corresponding hand calculation. What is the percent error in the FLUENT value?

3. Verification of FLUENT Results

You have already looked at the comparison to the analytical results in the fully-developed region. Carry out the following additional verification studies.

a. Derive an integral equation for the y-momentum balance in the domain. Use this equation to check whether the FLUENT solution satisfies y-momentum balance in the domain. What is the percentage y-momentum imbalance relative to the drag?
b. Check the dependence of the results on the mesh by repeating the calculation on a finer mesh. Double the number of divisions in both directions. Add the finer mesh result to the previous plots of the outlet velocity profile and skin friction coefficient. Are these results reasonably independent of the mesh or do you have to do additional levels of refinement? (You don't have to perform any additional mesh refinement).
Re-calculate the percent error in the skin friction coefficient in the fully-developed region. Has the percent error dropped on refining the mesh?