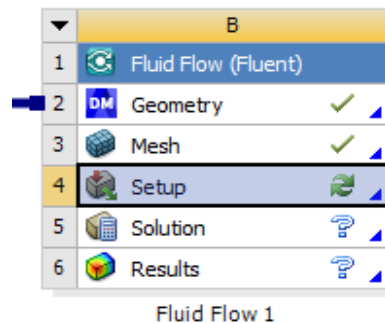


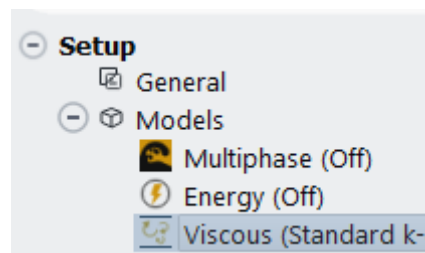
# ENGSCI 344 – Tutorial 4

This tutorial covers some of the basic concepts of convergence in CFD simulations. The basic criterion for the selection of turbulence models will also be addressed.

1. Download the necessary files from Canvas and open the Workbench project file in ANSYS Workbench.
2. Open Fluent (Setup tab from the **Fluid Flow 1 FLUENT template** (the module on the right)). Visualise the mesh. What type of cells are dominant in the mesh? Is it a structured or an unstructured mesh?



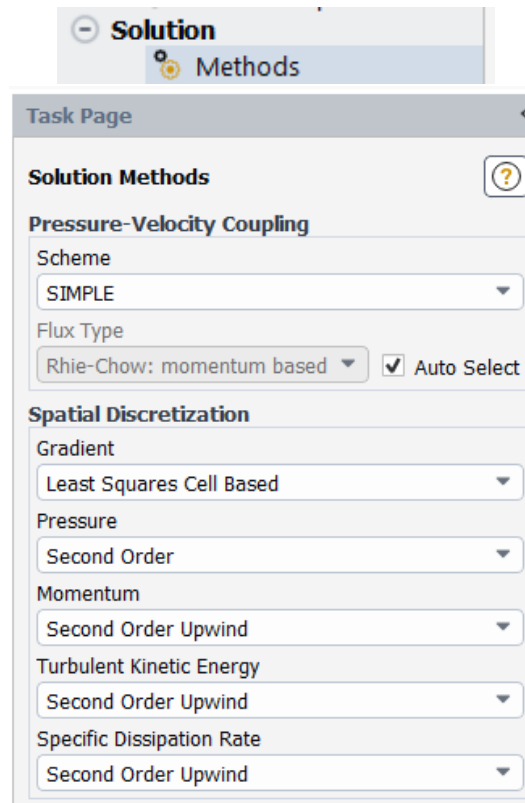
3. Set the turbulence model to the standard k-epsilon model (Viscous Tab in Model). Leave the model constants and the Wall Treatment to their default values. A brief introduction to the k-epsilon model can be found here: <https://www.simscale.com/docs/simulation-setup/global-settings/k-epsilon/>



4. The fluid flows from left to right. Ensure that the boundary conditions are set up properly for the inlet and outlet. A uniform velocity inlet of 2 m/s and a pressure outlet condition must be set. Set the turbulence intensity to 5% and set the turbulent length scale to 0.035m for the inlet boundary condition. Leave the backflow turbulence quantities to their default values for the pressure outlet boundary. We do not expect any backflow in the current simulation.

Turbulence intensity, length scale, dissipation and turbulent viscosity are problem specific. Refer to the above link to get a brief idea on how to calculate these values for simple geometries. Note that the turbulence intensity and length scale are usually low for external flows.

5. In the Methods tab, set the solver to SIMPLE, pressure to second order and all other discretisation schemes to second order upwinding. Use the Auto Select option for flux-type.



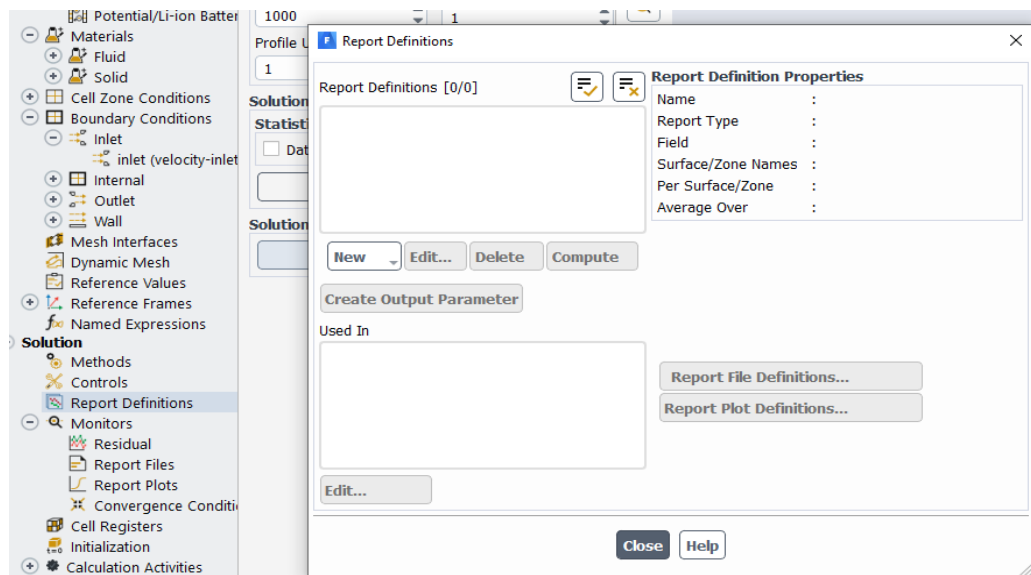
Here, we are using second-order accurate schemes to discretise the Navier-Stokes equations. Note that the first order upwinding schemes are too dissipative and can lead to erroneous solutions due to numerical diffusion, and the results from first order schemes should be interpreted with caution. Some of the differences between the first order and the second order schemes are highlighted here:

<https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node779.htm>

Ideally, while solving CFD problems, first, run the simulation with first order upwinding schemes to ensure that there is reasonable convergence and that the problem can actually be solved. Once you are confident with the simulation setup, solve the CFD problem with second-order accurate schemes.

6. Open the Residuals tab and set a value of  $10^{-4}$  for all residuals. In a steady state simulation such as the current tutorial, the convergence can be monitored by ensuring that the solution satisfies a couple of conditions.
  - a. The residual RMS Error values have reached the accepted value.
  - b. Our values of interest have reached a steady solution.
    - i. This ensures that we would get the same solution even if we run the simulation for a greater number of iterations.
  - c. Mass conservation is satisfied. Why?

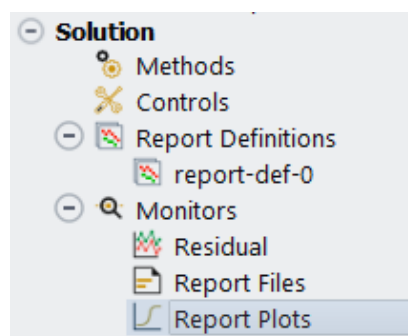
## 7. Create a new Force Report Definition.



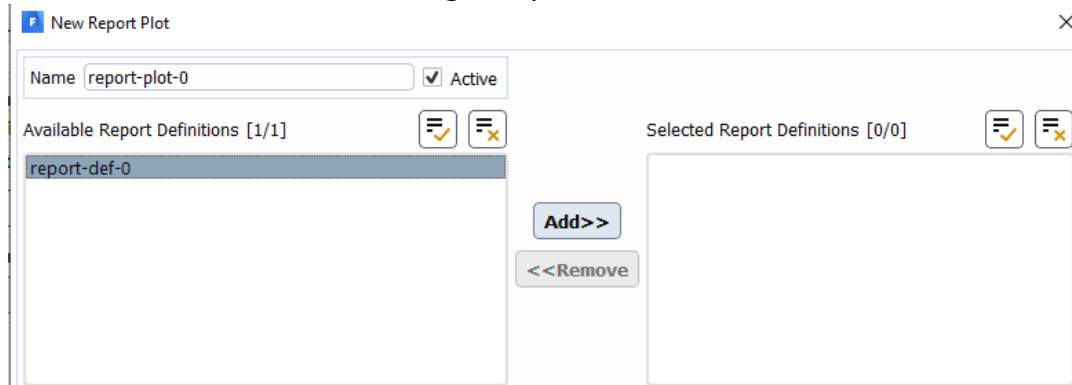
## 8. Set the Zones to walls. This will calculate the forces on the walls.



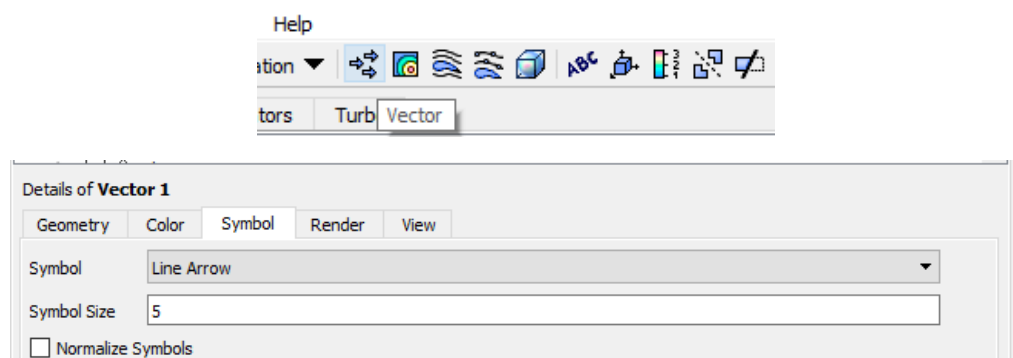
## 9. Create a new Report plot if one is not already created. If the Report Plots already has a variable, skip to step 11.



10. Add the newly created Report Definition to the Report Plot. Click on Ok. You do not have to plot it now as it will automatically plot it while solving the problem. Using a Report Plot will allow us to monitor the mean force on all the walls while solving the problem.

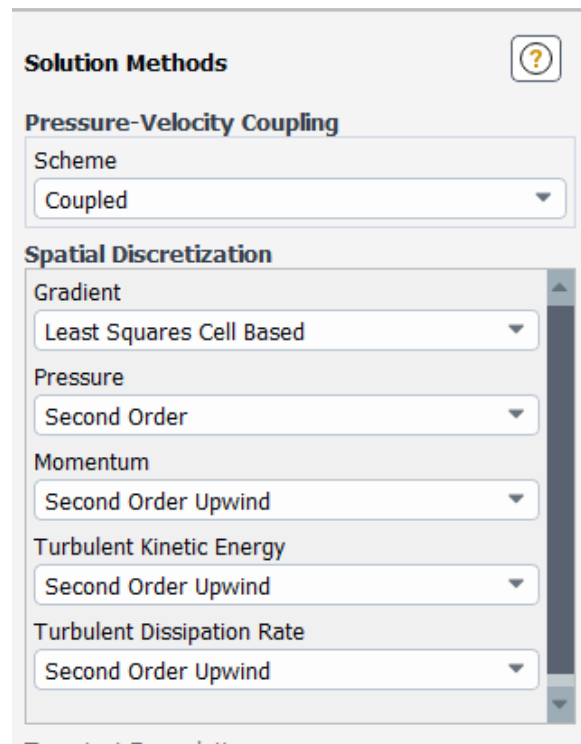


11. Initialise the problem using Hybrid initialisation. Set the number of iterations to 100 and solve it.
12. Did you achieve convergence? If not, increase the number of iterations to 5000 and solve it again. Check whether the residuals are below the tolerances and whether the mean force has converted to a steady value.
13. If the problem is well-defined, increasing the number of iterations should allow the residuals to converge. There is no set value for the number of iterations to be used for a steady state simulation and it largely depends on the problem at hand. Varying the number of iterations should be one of the first steps to follow while checking for convergence. Solving it using a coarse mesh might aid in convergence but will not necessarily produce the correct solution.
14. Close Fluent and open CFD Post to visualize the flow. Plot the velocity contours and the streamlines. Plot the velocity vectors on one of the Symmetry planes to find the recirculation region. You can change the size of the vectors in the symbol tab.



15. Close CFD Post and return to Workbench.
16. Clear the data from the solution tab in the Fluid Flow 1 template (Right-click on Solution and select 'Clear Generated Data').

17. Open Fluent (from the same Fluid Flow 1 template) and change the solution method from SIMPLE to Coupled (Found in the Methods tab in the tree on the left). SIMPLE solves the pressure and velocity equations using a segregated approach while the Coupled solver solves the pressure and velocity equations in a coupled manner using a fully implicit method. In most situations, the coupled algorithm offers superior convergence than segregated solvers such as SIMPLE.



18. Solve the problem. Save a screenshot of the Force vs Iterations plot after reaching convergence. Compare the convergence rate of the SIMPLE and the Coupled solver.
19. View the results in CFD Post. Is a secondary recirculation region present along with the primary recirculation region?

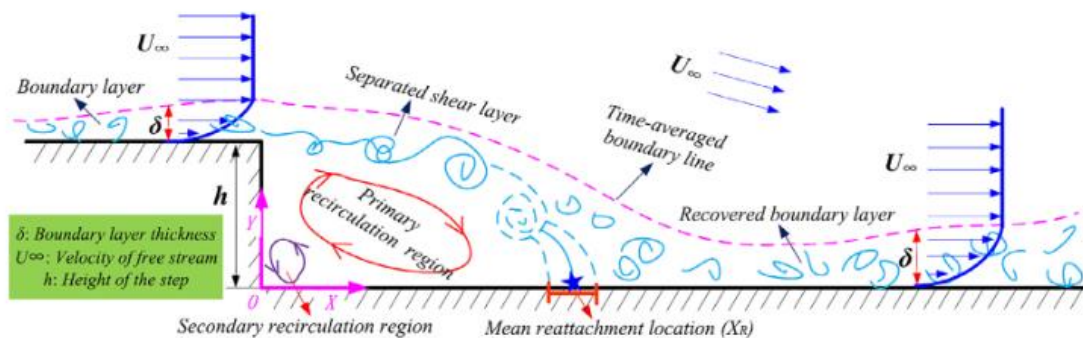


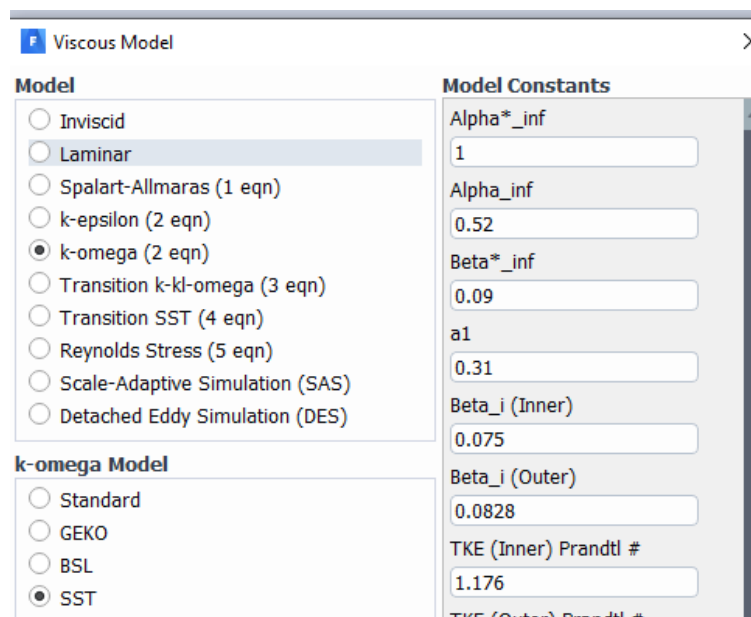
Image Ref: <https://doi.org/10.1016/j.tsep.2018.04.004>

20. The k-epsilon turbulence model is known to be unreliable to model the flow near no-slip walls. The k-omega SST model offers better results when compared to the k-epsilon turbulence model. Have a look at the following for some elementary differences between the two turbulence models.

*Note: Turbulence modelling is an evolving field and there is no single turbulence model to solve all flows of interest.*

<https://www.engineering.com/story/choosing-the-right-turbulence-model-for-your-cfd-simulation>

21. Open FLUENT in the same template and change the turbulence model to k-omega SST instead of k-epsilon.



22. Initialise the solution and solve it using the SIMPLE algorithm for 5000 iterations. Compare the convergence behaviour with the convergence of the k-epsilon turbulence model.
23. Change the solver from SIMPLE to Coupled. Initialise the solution and solve it again. Did the Coupled method offer superior convergence to the SIMPLE algorithm?
24. Open CFD Post and visualise the results. Compare the results of this simulation with the previous simulation.

Note that different flow behaviour can be observed by only changing the turbulence model even when all the other parameters such as inlet velocity, mesh and boundary conditions remain the same! It is important to carefully choose an appropriate turbulence model while modelling turbulence in CFD.

25. Close CFD Post and return to Workbench.
26. Open the Setup in the **Fluid Flow 2 template**.

27. Visualise the mesh and set the boundary conditions used earlier. The current mesh has local mesh refinement and inflation layers along the bottom boundary. Do we need inflation layers at the top boundary?
28. Set the turbulence model to the Standard k-epsilon model.
29. Set the same convergence tolerances as earlier.
30. Initialise the solution and solve it using the coupled algorithm for 5000 iterations (use second-order accurate schemes). Did it converge? If so, save a screenshot of the Force vs Iteration plot. Compare the plot with the coarse mesh.
31. Visualise the results in CFD Post and check for the secondary recirculation region. Save a screenshot of the velocity vectors showing the secondary recirculation region.
32. Change the turbulence model to k-omega SST, initialise the solution and solve it with SIMPLE and Coupled solvers. Save screenshots of the residuals of SIMPLE and Coupled solvers.
33. Visualise the flow and compare the results with each other and with the coarse mesh. Save a screenshot of the velocity vectors showing the secondary recirculation region (for the coupled solver).

The flow field can be different despite achieving convergence. Convergence is important but should not be the only criterion while solving fluid flows. Understanding the fluid behaviour prior to solving the problem is often required as the type of solver to use and the mesh highly depend on the fluid behaviour that we expect to observe.

34. In what other regions can you refine the mesh to improve the accuracy of the solution?