

ENGSCI 344 Tutorial 3: Flow over a bump

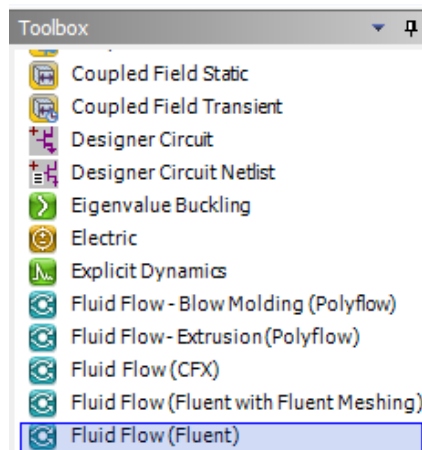
In this tutorial, you will create a mesh and set boundary conditions, solver settings and fluid properties.

You are provided with a Design Modeler file, which needs to be imported into Workbench to begin the tutorial.

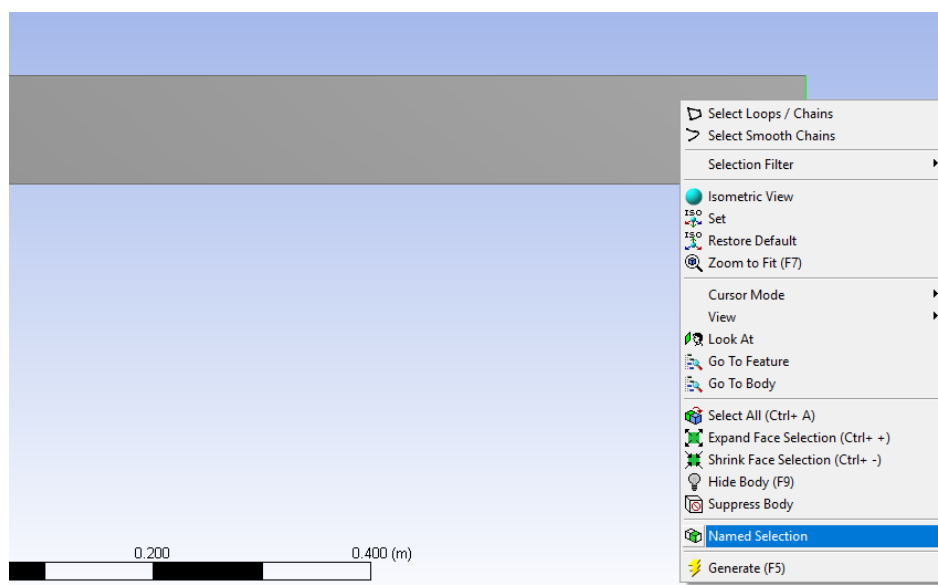
PART A

Geometry and Meshing

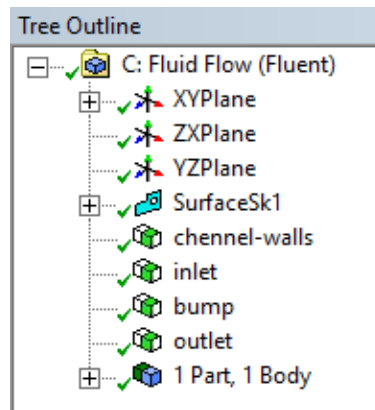
1. Open Workbench and create a new standalone FLUENT template. Import the geometry (Right-click on Geometry tab and use the import option).



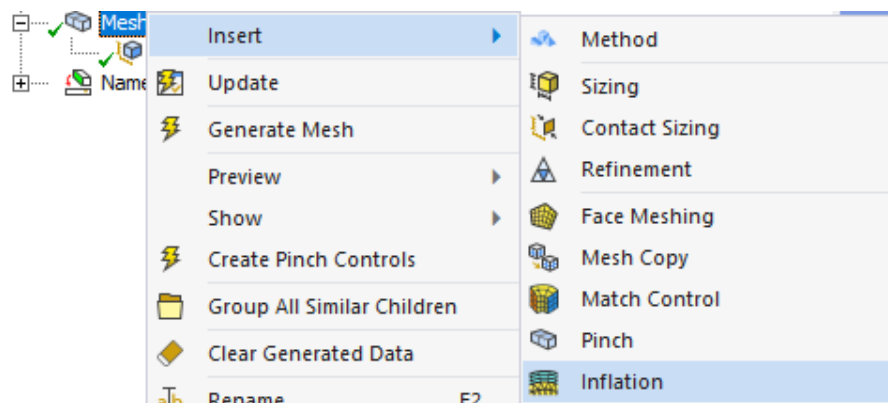
2. Open Design Modeler and generate the part. In this geometry, the fluid flows from right to left. You are to create boundary conditions mimicking this behavior.
3. Select the right boundary, right-click and create a named selection "inlet." Update the geometry.



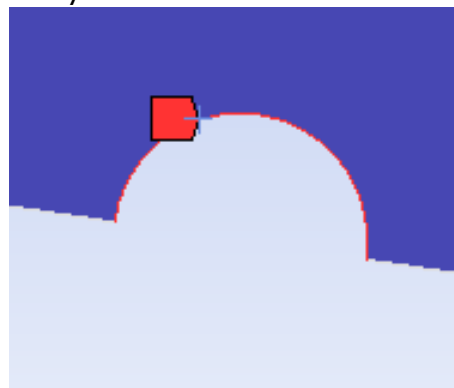
4. Similarly, select the bump and the left boundary, and create named selections as “bump” and “outlet,” respectively.
5. The rest of the boundaries are already assigned a name. If not, create a named selection highlighting the remaining boundaries. Having named selections in the geometry file makes it easier to assign boundary conditions in FLUENT.
6. At the end, the Tree Outline on the left should look like the picture below:



7. Close Design Modeler and open the meshing module. Using the strategies from the earlier tutorials, mesh the entire body with local refinement at the bump.
8. Create inflation layers on the bump. Right-click on Mesh > Insert > Inflation. Why do we need inflation layers?



9. Select the entire body for the geometry and select the edges representing the bump for the boundary.



- Set the maximum number of layers to 10 and click on generate mesh.

Details of "Inflation" - Inflation	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	1 Edge
Inflation Option	Smooth Transition
<input type="checkbox"/> Transition Ratio	Default (0.272)
<input checked="" type="checkbox"/> Maximum Layers	10
<input type="checkbox"/> Growth Rate	1.2
Inflation Algorithm	Pre

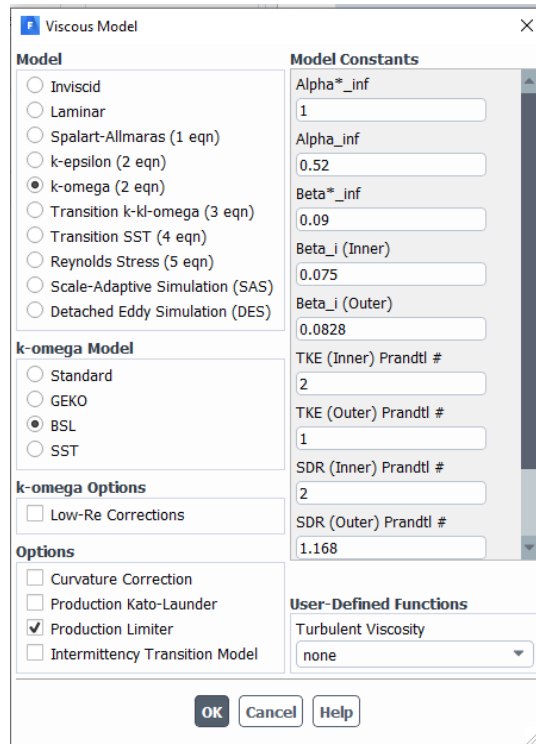
Setup

- Once you've meshed the domain, close the meshing module and open Setup from the template. Select "Double Precision" and run it as a parallel job using four cores (the number of cores available depends on the CPU).
- Once the FLUENT window opens, expand the models option on the left. Similar to Tutorial 1, all the operations can be performed using the tree on the left. You are free to use the ribbon on the top.
- Open the General section to ensure that you're solving a steady-state problem with a Pressure-based solver. Think whether including gravity makes any reasonable difference to the fluid flow.

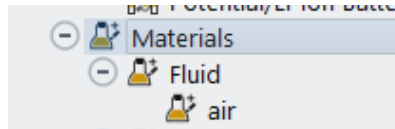
Solver	
Type	Velocity Formulation
<input checked="" type="radio"/> Pressure-Based	<input checked="" type="radio"/> Absolute
<input type="radio"/> Density-Based	<input type="radio"/> Relative
Time	2D Space
<input checked="" type="radio"/> Steady	<input checked="" type="radio"/> Planar
<input type="radio"/> Transient	<input type="radio"/> Axisymmetric
	<input type="radio"/> Axisymmetric Swirl

- Double-click on "Viscous" in Models tab (Outline view on the left) and select the K-omega 2 eqn model (BSL in k-omega model). Doing this will set the flow to be turbulent and assign the corresponding turbulence model.

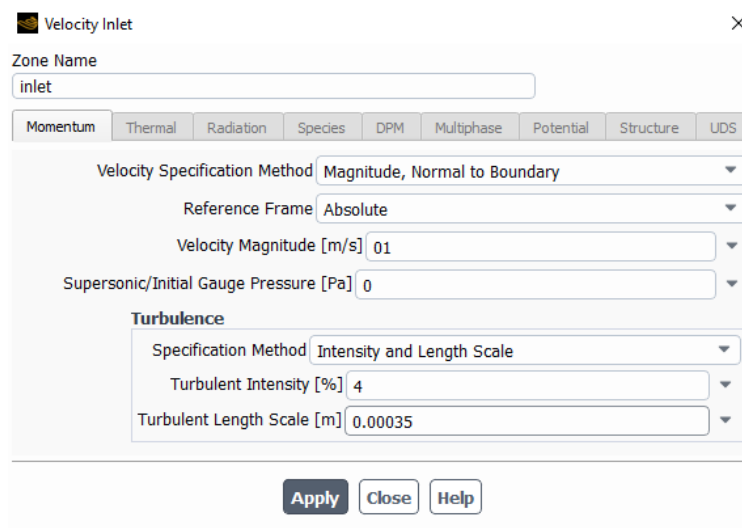
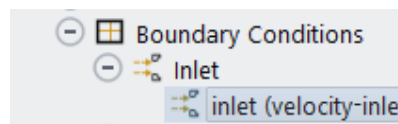
Remember, modelling turbulence is still a developing field of research and there cannot be a single answer/model for all the problems. Selecting a turbulence model largely depends on the problem at hand. The ANSYS FLUENT Theory Guide provides a very brief yet useful explanation concerning the features and use cases of turbulence models. This documentation can be accessed using Fluent help.



15. Open the materials section and change the properties of air to the following values:
 - a. Density: 1.1 kg/m^3
 - b. Viscosity: $1.77\text{e-}5 \text{ kg/ms}$



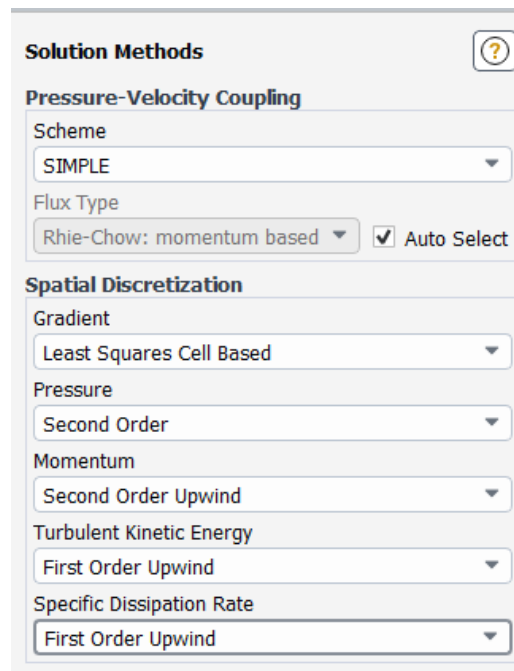
16. Open the boundary conditions and set the inlet velocity to 1 m/s . Set the turbulence intensity to 4% and the length scale to 0.00035 (turbulence intensity and length scale are problem specific).



17. Check whether the outlet boundary condition is specified properly. You can leave the backflow values at their defaults as, in this problem, we do not expect any backflow to develop.
18. Also, check whether the bump and other walls are set as no-slip walls.
19. FLUENT automatically sets few boundary conditions depending on the names given. Using meaningful names (in named selection) while creating the geometry/mesh will save you time while assigning the boundary conditions.

Solution

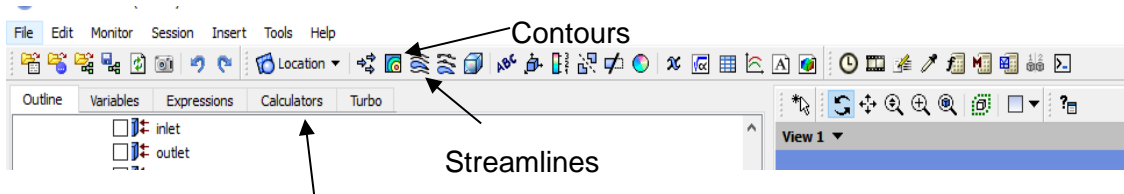
20. Expand the Solution module in the tree and open the methods section. Set the Scheme to SIMPLE and set second-order Spatial Discretization for pressure and momentum. Use the default first-order methods in other options.



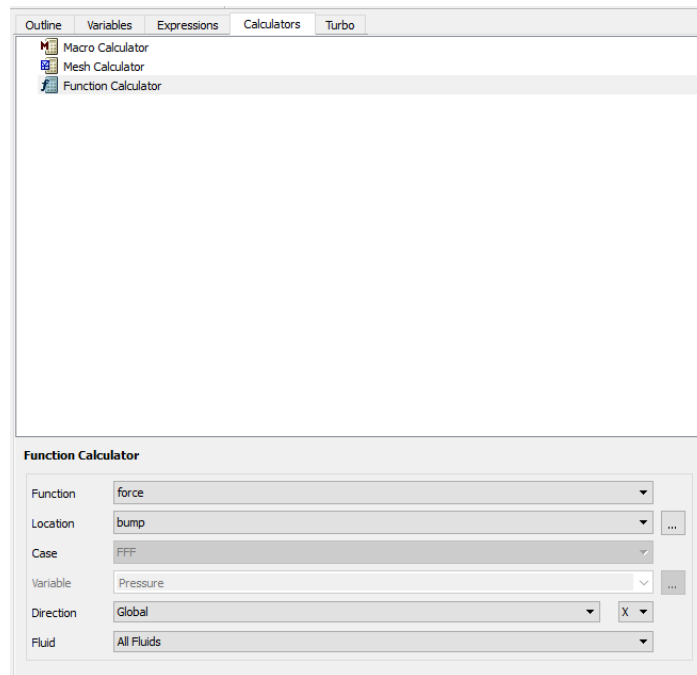
21. Expand the Monitors section and double-click on Residual. Set all the values in the Absolute criteria section to 0.0001.
22. Open the Initialization section and initialize the problem using Hybrid Initialization. Why?
23. Open Run Calculation section, set the number of iterations to 1000 and click on "Calculate."
24. The residuals plot that is shown gives a rate of convergence of the quantities of interest. Here, along with the pressure, velocity and continuity terms, you will also notice convergence metrics for the turbulent quantities. This graph is very helpful for refining your fluid model. Non-convergence of quantities could mean multiple things but at the very basic level, one could relate it to problems with the model. Check (in the order of preference):
 - a. That the boundary conditions are valid. Ensure that there is an outlet along with an inlet.
 - b. Wall boundary conditions.
 - c. Material properties.
 - d. If your mesh is too coarse.
 - e. The turbulence model.
 - f. Solver settings.

Post-processing

25. Once convergence is reached, save a screenshot of the residuals plot. Close the FLUENT window and open the Results section in Workbench. This should open CFD-Post (You can also perform post-processing in FLUENT itself). CFD-Post is preferred as it would allow you to create multiple branches in Workbench to simplify your workflow.
26. Plot streamlines, velocity (magnitude) and pressure contours (on one of the symmetry planes). Save screenshots of streamlines and pressure contours.



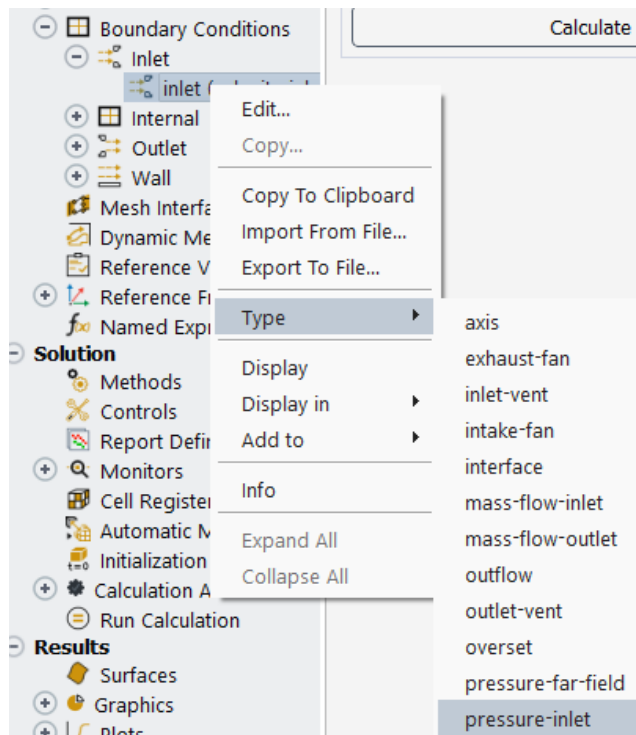
27. Select the Calculators tab shown in the picture above and double-click on the Function Calculator option.
28. Set force as the function, bump as the location and X as the direction. Calculate the force.



29. Similarly, calculate force in the Y direction. Record both values and close CFD-Post.

PART B

1. In Workbench, create a duplicate of the FLUENT template (right-click on the FLUENT template and select duplicate). This will create a new FLUENT template with all the parameters set in the previous section.
2. Open Setup in the newly-created template and change the turbulence model to Standard k-epsilon with scalable wall functions.
3. Initialize and Run Calculation. How does the rate of convergence change?
4. Once the solution is complete, open CFD-Post and plot contours and streamlines. Compare with previous values. Also, calculate the forces and compare with previous values. Are they the same? If not, why? Record the values of forces.
5. Create a duplicate template of the current template (template used for Part B).
6. Change the inlet to pressure inlet and set the Gauge pressure to 2 Pa.



7. Initialize and Run Calculation. How does the rate of convergence change?
8. If the solution converges within 1000 iterations, close FLUENT and open CFD-Post for post-processing. Plot contours and calculate forces. Visualize the velocity contours and pressure contours and observe whether the flow is different in this case? If it is different, why? Save the screenshots of pressure contours, streamlines and forces.
9. If the solution does not converge, refine the mesh and check for convergence.
10. Is it straightforward to set the inlet velocity using the pressure inlet boundary condition? If not, why?
11. If it does not converge, what changes could you make to the model to improve convergence?
12. What other boundary conditions can you use to set the inlet velocity to 1 m/s?