

UNIVERSITY OF SHEFFIELD

MEC320 COMPUTATIONAL FLUID DYNAMICS

FLOW OVER A BACKWARD FACING STEP ASSIGNMENT 2: NUMERICAL SIMULATION RESULTS

Spring 2020

1 ABSTRACT

The purpose of this report is to follow on from the mathematical model report and discuss numerical simulation results of a CFD analysis. A strategy for generating a good quality computational mesh for the problem under investigation has been discussed, including a method for checking mesh-independence. The typical solution approach used by Fluent is described and finally some data previously obtained by an unexperienced CFD engineer is analysed.

TABLE OF CONTENTS

1. Abstract.....	1
2. Computational Domain and Discretization Process	2
3. Solution Process.....	3
4. Validation of Results	4
5. References.....	7

2 COMPUTATIONAL DOMAIN AND DISCRETIZATION PROCESS

In order to generate a good quality computational mesh, the geometry of the problem domain must first be defined. As the experiment is being replicated from the one conducted by Driver and Seegmiller, the dimensions of the domain will be the same and therefore taken from their paper. The geometry can be set as 2D instead of 3D, as the assumption of 2D flow was justified in assignment 1. In the paper it says the flow was fully turbulent before the step, therefore in order to simulate this correctly in ANSYS the upstream channel length will be extended to 40.1cm as calculated using equation 2.1.

$$x = \frac{\delta_b}{0.16(R_{ex})^{-\frac{1}{7}}}$$

Equation 2.1

Once the geometry has been successfully created, the mesh can be generated. It is important to generate a good quality mesh as it significantly affects rate of convergence, solution accuracy and CPU time required [1]. An important factor of the mesh is topology. As the geometry of the domain is very simple, only consisting of 4 straight lines, a structured grid with quadrilateral elements can be justified as structured grids have better convergence and higher resolution than unstructured grids [2]. A quad mesh can also be justified as it can be aligned with the flow direction/gradient resulting in a reduced truncation error.

Truncation error is an error which occurs due to discretization and depends on the distance between grid points, therefore, can be reduced through mesh refinement. The process of discretization is used in this problem to replace the governing partial differential equations with discrete counterparts which still represent the same physical properties but are in a form that can be solved through an alternative process. Introducing a grid in the domain allows the Navier Stokes equations to be manipulated and transformed into a set of solvable algebraic equations which are solved at the discrete points only, as opposed to continuously throughout the domain.

The discretization method used in ANSYS Fluent is the finite volume method (FVM). This method deals directly with the integral form of the Navier Stokes equations and therefore does not require mathematical continuity, meaning the flow variables (i.e. velocity and pressure) can go through sharp changes within the control volume, making it an ideal approach to solve this flow problem.

Increasing the number of grid points and therefore the number of discrete points to be solved is known as mesh refinement. It is a method used to increase the accuracy of a solution and therefore commonly used within certain sensitive or turbulent regions in order to more accurately capture the flow. In this flow problem, the areas of more importance are those close to the walls, the region vertically inline with the step, the region just after the step and the reattachment zone. A non-uniform mesh can be used in order to create these areas of finer mesh by using the 'Bias' function.

After the mesh has been generated it is very important to check the quality of it. This can be done by checking the aspect ratio and element quality. The element quality is a value that ranges from 0 to 1, with higher values meaning higher quality elements. Therefore, to improve the mesh, actions would be taken to raise the minimum value. The aspect ratio is the ratio of the longest dimension on an individual element to its shortest dimension. Generally,

the aspect ratio in a good mesh should be smaller than 5 [3], therefore if it is greater than this value, the mesh should be altered in order to reduce it.

Finally, it is vital to check that the solution is mesh independent. This means results do not vary significantly even when the mesh is refined further. In order to check for this, a mesh independence study can be performed. This involves refining the mesh throughout the domain each time a new simulation is ran, while recording results of a chosen variable. By creating a graph of number of elements on the x-axis and the chosen variable on the y-axis, it can be seen at which point the increase of elements has an insignificant effect of the variable, and therefore can be described as independent. As it is known that the skin friction coefficient at the reattachment zone is zero, the reattachment length that can be used as the variable to be measured here.

3 SOLUTION PROCESS

Once a good quality mesh has been generated, the systems of linearized algebraic equations, formed due to discretization, can then be solved. There are many different solution techniques that can be adopted. Some fall under direct methods which give direct simultaneous solutions of all equations, while with other techniques, solution is approached gradually through an iterative method. These are known as indirect or iterative methods and have an advantage over the direct methods in that they require simultaneous storage of only non-zero equation coefficients, as opposed to storage of N^2 equations coefficients in core memory. In ANSYS in order to solve the pressure-velocity problem indirect/iterative methods called pressure-correction methods are used, as direct methods of solving simultaneously Navier Stokes and Continuity equation is impractical.

The pressure-velocity problem is to do with the coupling between pressure and velocity in the Navier Stokes equations. The set of discretized equations provided by the momentum equations can only be solved for the velocity field if the pressure gradient is known. However very rarely, including the flow problem of the backward facing step, is the pressure gradient known before the analysis. In 2D, incompressible flow problems, like the flow problem in question, neither pressure nor density appear in the continuity equation, therefore the continuity equation cannot be used to directly obtain pressure and the momentum and continuity equations are said to be uncoupled. This is why iterative guess and correct methods need to be used here.

There are many different iterative pressure correction methods, including SIMPLER, SIMPLEC and PISO, but the typical solution approach used by Fluent is a basic guess-and-correct procedure called SIMPLE. As the pressure field is unknown, an estimated value is used to begin with and assigned to every node in the mesh. With this estimated value, the corresponding velocity field (u, v, w) can be found by solving the momentum equation. These terms will then be put into the continuity equation, but it is very unlikely they will satisfy the continuity equation as they are based on estimations, so the algorithm will calculate the change in pressure required to restore continuity.

Putting this change in pressure value back into the momentum equation, new change in velocity field values ($\Delta u, \Delta v, \Delta w$) are calculated. The change in pressure and velocity field values are then added to their original estimated values in order to update/correct them. These

values are then used to repeat the whole process again. At some point the change in pressure and velocity field values will be so small that they become insignificant, and therefore have met the convergence criterion, meaning the pressure and velocity field values have satisfied both the momentum and continuity equation.

The next stage is to check whether the analysis has converged or not. The most commonly used method of doing so is to monitor the residuals. Residuals measure the local imbalance of a conserved variable in each control volume [4]. A lower residual value indicates a more numerically accurate solution, however there is no absolute criterion on the value, therefore this must be chosen for each flow problem. Often for additional information on the convergence of the solution, the values of certain variables in specific regions of interest in the domain are monitored, these are known as monitoring points.

4 VALIDATION OF RESULTS

As discussed in assignment 1, when modelling the flow through a backward facing step, several assumptions were made in order to simplify the problem mathematical description. This is the same with many flow problems and in some cases several assumptions can be wrong, leading the model further away from the real problem. Therefore, it is important to verify and validate results at the end of a CFD analysis. Verification is done to ensure that the mathematical model has been solved as accurately as possible. This is done by quantifying the error in the CFD implementation, however, is not done in this report as the data used here has been previously obtained and is used for the purpose of validation.

Validation is done to ensure that the CFD results obtained, accurately represents the physics of the problem. Figures 4.1 and 4.2 show the contour plots of static pressure and velocity magnitude respectively, along the domain of the flow problem. In assignment 1 it was discussed and understood that just after the step there is an adverse pressure gradient resulting in a low-pressure region right by the step. This can be seen in figure 4.1 as the region just behind the step is dark blue, representing much lower static pressure compared to rest of the domain.

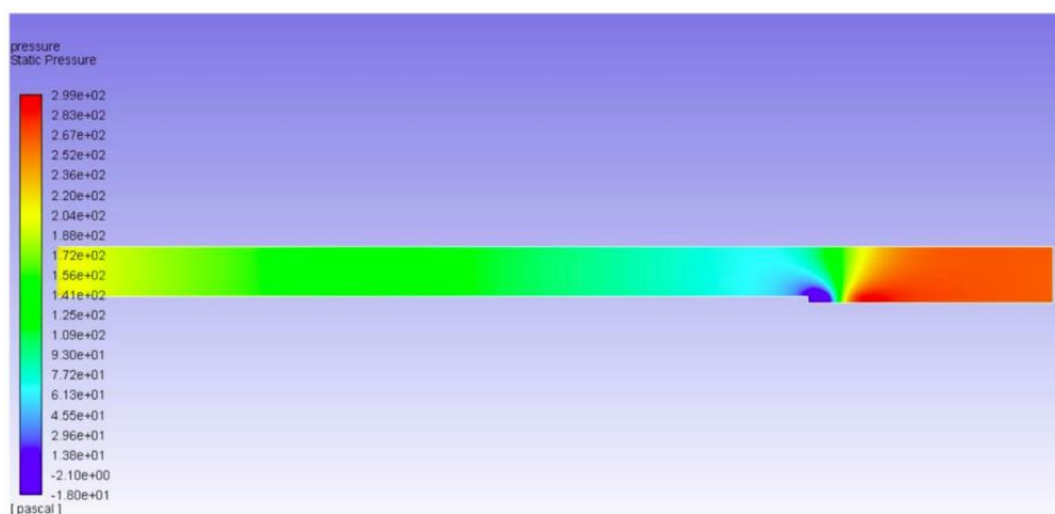


Figure 4.1 Contour plots of static pressure [Pa]

It can also be seen in figure 4.2 that velocity decreases due to the increase in area. This is what was expected to happen due to the conservation of momentum, therefore validates the results to some extent.

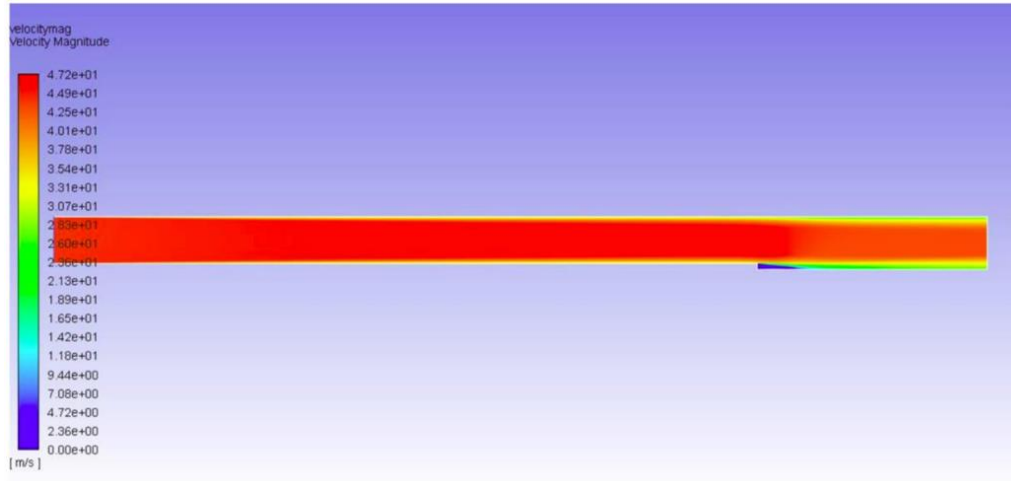


Figure 4.2 Contour plots of velocity magnitude [m/s]

Validation can be done by quantifying input uncertainty, however, is most commonly done by comparing the CFD results with high-quality experimental data, in other words quantifying the physical model uncertainty. Figure 4.3 shows the skin-friction coefficient along the lower wall of the problem domain of the CFD results against experimental data. As it is known that the skin friction at the reattachment point is zero, the reattachment length is found where the data intercepts the x axis (other than at $x/H=0$ which is the step location). The CFD data and experimental data both intercept the x axis at the same location, meaning the reattachment length was the same. Reading off the graph it can be seen that the reattachment length is extremely close to, if not exactly 6.26, which was the length obtained in the driver and Seegmiller experiment. Therefore, further validating these results.

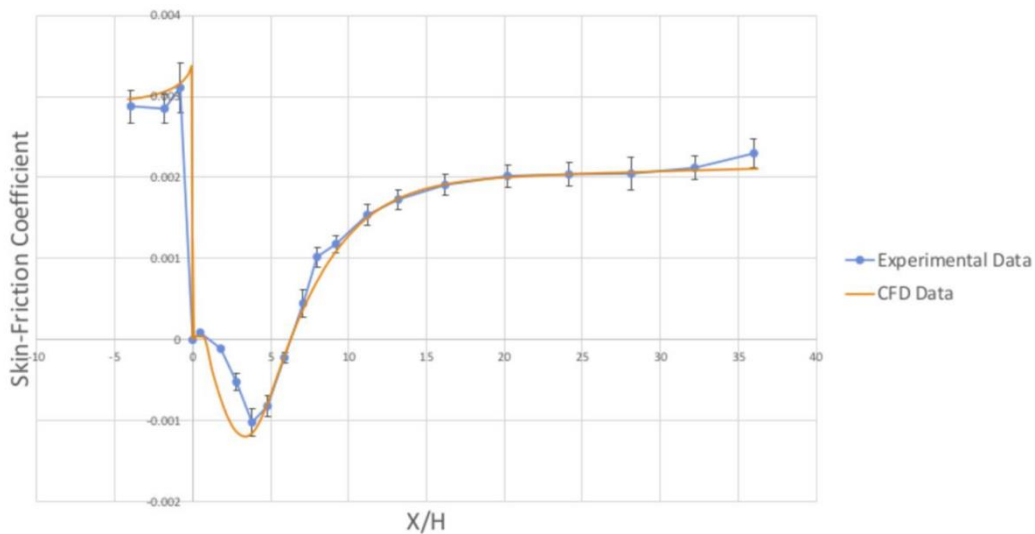


Figure 4.3 Comparison between experimental and numerical data for the skin-friction (step located at $x/H=0$, H is the height of the step)..

It can be seen that on some parts of the graph the data plots aren't at the same points, this suggests that there is some error in the CFD data. The largest errors can be seen just before and just after the step. As mentioned in an early section of this report, mesh refinement is a method often used to increase the accuracy of a solution. Figure 4.3 shows the mesh used for this CFD analysis. As you can see mesh refinement has been used close to the walls, in line with the step vertically and horizontally. However, the area of very dense mesh isn't very thick, therefore increasing this area may lead to greater accuracy. Also adding refined mesh along the whole section behind the step should achieve a better agreement between the numerical and experimental data.

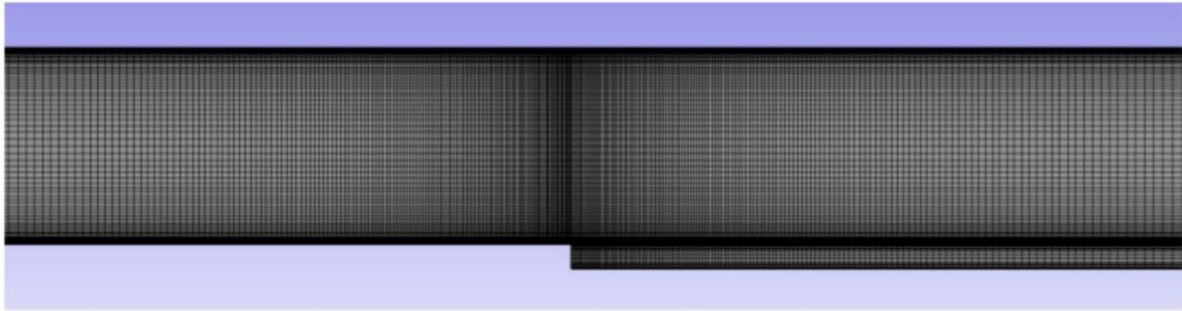


Figure 4.3 *Computational mesh used in the CFD simulation.*

Further agreement could be achieved by using a more suitable turbulence model/wall function in ANSYS. As discussed, and justified in assignment 1, the SST K- ω turbulent model seemed like an appropriate model to use for this particular flow problem. Using this model instead of the very simple turbulence model may have achieved more accurate results.

5 REFERENCES

[1] Dr.A. Marzo, *MEC320 Computational Fluid Dynamics*, Lecture 14 (2020), slide 5

[2] Wikipedia, *Types of mesh*, Available at:
https://en.wikipedia.org/wiki/Types_of_mesh#Quadrilateral [Reviewed May 2020]

[3] University of Sheffield, *MEC320 Laminar Developing Flow in a Circular Pipe*, Lab sheet, page 21.

[4] M.Kuron, *3 Criteria for Assessing CFD Convergence*, CAE Associates (2015), Available at: <https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/9296/3-Criteria-for-Assessing-CFD-Convergence.aspx>