CHPR4407 Assignment 2
Flow over a backwards-facing step

Bennett Lovelady
20519235

September 2015
Introduction & Literature Review

The purpose of this project is to develop two- and three-dimensional CFD models of the backward-facing step flow using ANSYS Fluent, and validate the results by referring to prior research. The geometry of the flow scenario, along with the general nomenclature, is shown in Figure 1. The inlet boundary layer separates from the bottom wall at the top of the step and reattaches a distance $x_R$ downstream. Below the separated boundary layer is a recirculation region containing spanwise vortices of varying sizes as depicted in Figure 2 – it is the size of this region and the length of $x_R$ which have been of most engineering interest.

![Figure 1: General flow scenario](image)

Many experimental investigations have been carried out over the years. There has been a general focus on the laminar and transitional flow regimes as these have been found to give rise to complex three-dimensional vortical structures in the flow, and to be more sensitive to boundary conditions.

![Figure 2: Flow past the step [1]](image)
This project only regards flow in the turbulent regime so only experiments involving high Re will be considered.

The reattachment length $x_R$ is strongly dependent on the Reynolds number in the laminar and transitional regimes ($Re_H < 3400$) but above this value the reattachment length settles to $x/H \approx 7$\(^3\). The reattachment has also been found to be transient in nature and actually oscillates in the range $x/H \approx 7 \pm 1$\(^4\), but in this project I will only model the steady-state flow.

In addition to the Reynolds number, the aspect ratio $AR = W/h_1$ of the duct has been found to affect $x_R$. If the aspect ratio is too low, three-dimensional effects come into play and the reattachment becomes more sensitive to the Reynolds number\(^5\).

The expansion ratio $ER = \frac{h_1 + H}{h_1}$ also has an effect on the reattachment length. At very low expansion ratios, the value of $x_R$ tends towards the accepted value of 7\(^6\) but Makiola et al recorded a value of $x_R/H = 8.2$ with an expansion ratio of 2\(^7\).

As BFS flow is one of the simplest examples of turbulent boundary layer reattachment, it has been used extensively in validation of numerical models. Good results have come from DNS methods\(^8\) but these are prohibitively expensive. The $k-\epsilon$ model is notoriously bad at predicting $x_R$, often underestimating by a factor of 2\(^3\). The $k-\epsilon$ RNG model has been found to give much better results than standard $k-\epsilon$ in 2D but still is not ideal, especially not in 3D\(^9\). Reynolds Stress Models have shown promising results in both 2D and 3D but can be difficult to converge\(^1,10\).
Model Formulation

The model was based on the 1985 experiment by Driver & Seegmiller\cite{11}, which used a wind tunnel at $Re_H = 36,800$. This model uses a moderately high Reynolds number and has been used as a test case in several numerical experiments. Table 1 shows the relevant data to set up the model. The air conditions were not specified in their experimental setup so Fluent’s default values of $\rho$ and $\mu$ were used.

Table 1: Model data

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h_1$</td>
<td>101 mm</td>
</tr>
<tr>
<td>$H$</td>
<td>12.7 mm</td>
</tr>
<tr>
<td>$W$</td>
<td>151 mm</td>
</tr>
<tr>
<td>$U_1$</td>
<td>44.2 m/s</td>
</tr>
<tr>
<td>$\rho$</td>
<td>1.225 kg/m$^3$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>$1.789 \times 10^{-5}$ kg/m·s</td>
</tr>
</tbody>
</table>

In order to ensure the flow upstream of the step was fully-developed, I first set up a (2D) model of the upstream section – a rectangular duct of height $h_1$ and length $L = 20H$, with a velocity inlet ($U = 44.2$ m/s) and a pressure outlet. By plotting the flow x-velocity along the duct centreline, it can be seen that after about $x/H = 3$ the air accelerates uniformly as expected; the inlet should then be placed at least 3 step heights upstream of the step.

2D Geometry

As a rule of thumb the turbulent effects of an obstacle of height $H$ are insignificant after $x/H = 10$, so the outlet should be placed at least this far downstream of the step.

For both the 2D and 3D models I used an upstream length $L_1 = 3H$ and a downstream length $L_2 = 20H$. This should ensure that the flow around the step is not affected by the open boundaries.

2D Mesh

The flow region was broken up into sections as shown in Figure 3 to allow fine control over the mesh in different parts of the flow area. These regions were intended to capture the boundary layers along the top and bottom walls, as well as the recirculation region behind the step. No edge biasing was used in the 2D mesh, but the grid was selectively refined in those regions of interest.

![Figure 3: Mesh breakup. Borders are drawn at $y = H$, $2H$, $h_1-H$ and $x = L_1 + 10H$](image-url)
In order to reliably use wall functions in the $k$-$\epsilon$ model, the near-wall nodes should have $y^+ > 11.63$ and ideally $30 < y^+ < 500$, where

$$ y^+ = \frac{\rho u_T y}{\mu} $$

Here $u_T$ is the frictional velocity, related to the wall shear stress $\tau_w$ by

$$ u_T = \sqrt{\frac{\tau_w}{\rho}} $$

The wall shear stress can be calculated from the skin friction coefficient $C_f$ by definition:

$$ \tau_w = \frac{1}{2} C_f \rho U_{\infty}^2 $$

Combining these three relations gives an estimate for distance $y$ of the nearest node to the wall:

$$ y = \frac{\sqrt{2} \mu}{\rho \sqrt{C_f U_{\infty}^2}} \cdot y^+ $$

The maximum value of $C_f$ reported by Driver & Seegmiller is $C_f = 0.001^{11}$; using this value of $C_f$ along with the other relevant data from Table 1 above gives $0.4 \text{ mm} < y < 7 \text{ mm}$.

The first 2D mesh was a coarse structured mesh, using edge sizings as shown in Table 2 below. The factor of 2 between coarse and fine regions was maintained for all mesh refinements.

<table>
<thead>
<tr>
<th>Region</th>
<th>Sizing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Near-wall</td>
<td>$dy = 4$ mm</td>
</tr>
<tr>
<td>Duct core</td>
<td>$dy = 8$ mm</td>
</tr>
<tr>
<td>Upstream</td>
<td>$dx = 10$ mm</td>
</tr>
<tr>
<td>Downstream</td>
<td>$dx = 20$ mm</td>
</tr>
</tbody>
</table>

The coarse mesh is shown in Figure 4. This mesh had good orthogonality, and had the aspect ratio aligned with the majority of flow as is ideal.

![Figure 4: Coarse 2D mesh with 559 nodes](image)
2D turbulence models
As stated in the introduction, good results have been found in 2D using $k$-$\varepsilon$ RNG and RSM turbulence closure models. I used $k$-$\varepsilon$ RNG initially and converged the mesh, then ran the RSM closure models on the same meshes.

2D boundary conditions
- Top edge: Non-slip wall boundary
- Bottom edge & Step: Non-slip wall boundary
- Right edge: Pressure outlet
- Left edge: velocity inlet, with velocity normal to boundary set as in Table 1, and turbulence defined by turbulent intensity = 0.5 % and hydraulic diameter = 0.121 m.

The inlet turbulent characteristics were not directly stated by Driver & Seegmiller. The hydraulic diameter was calculated from the geometry of their wind tunnel as $D_h = \frac{2h_1W}{h_1 + W}$ and the turbulent intensity was assumed based on similar papers that did report inlet turbulent intensity\(^3\).

3D geometry
The 3D model used the same upstream/downstream lengths as the 2D model as the inlet/outlet conditions are the same as for the 2D case. The step profile was extruded by $\frac{W}{2} = 75.5$ mm, to take advantage of the symmetry in the flow field; Driver & Seegmiller confirmed two-dimensionality along this plane\(^{11}\).

3D mesh
The 3D mesh was constructed differently to the 2D mesh – a mapped face meshing on the side- and symmetry-walls was swept across the z-axis to ensure mesh structure. The mesh sizing was determined by bias factors: with a bias factor of 4 and an average element size of 8 mm, the near-wall elements have an appropriate size as discussed above and the core elements are larger to reduce computation. Edge biasing was not used along the step’s vertical face, a uniform distribution of cells was selected instead. Figure 5 shows the coarse 3D mesh with 10,220 nodes

3D turbulence models
The standard and realizable $k$-$\varepsilon$ models proved most successful in 3D, with neither $k$-$\varepsilon$ RNG nor RSM models managing to converge.

3D boundary conditions
The inlet, outlet, top and bottom walls were defined as in the 2D case. The $z=0$ wall was set as a non-slip wall condition, and the $z=\frac{W}{2}$ wall was set to a symmetry condition.
Determining $x_R$

A reattaching boundary layer is characterised by the flow no longer recirculating. Figure 6 shows an example of boundary layer separation and reattachment in the flow over a stalled airfoil – the flow is visualised using an oil film and the reattachment point is clearly seen as a line where the oil is driven fore & aft. This point can also be characterised as the point at which skin friction (or wall shear) is zero, and I used this definition to find $x_R$ for each model using ANSYS CFD-Post. The value of $x_R$ was used to confirm mesh convergence as well as being the main output of the model.

In the 2D case, the procedure was:

- Define a Line extending from one step-height beyond the step to the end of the domain
- Define an expression equal to abs(WallShear X)
- Define a Point which lies on the Line at the minimum value of abs(WallShear X)
- Define an expression $x_R$ equal to (probe(X)@Point)/H-L_1

The value of $x_R$ then gives the number of step heights beyond the step that the flow reattaches.

The procedure was similar for the 3D case, except the Line was defined on the symmetry plane.
Figure 6: Separating and reattaching boundary layer visualisation [12]
Results & Discussion

After experimenting with various turbulence closure models, discretisation schemes and solution methods I found some that worked and some that did not. The $k-\omega$ SST model gave some promising results at first in the 2D case, but did not converge on any of the finer meshes. I used PISO as the solution method for most simulations – it has more robust convergence characteristics than SIMPLE and can offer faster convergence as well, and I found it worked better in the cases where I compared the two. Very strict under-relaxation (on the order of 0.1) was necessary in order to get the RSM models to converge – this may have been a result of the mesh not being fine enough along the back side of the step in both 2D and 3D cases.

A vector plot of velocities is shown in Figure 7, which clearly shows the recirculation zone behind the step. This plot is taken from the symmetry plane of the 3D case, using the finer mesh and the realizable $k-\varepsilon$ turbulence model.

Figure 8 shows a vector plot from the finest 2D case, using $k-\varepsilon$ RNG and the finest mesh. The secondary recirculation zone is barely visible but has been resolved.
Table 3 shows a summary of results with the most successful methods. All of the methods I used have under-represented the reattachment length $x_R$ by a significant amount, with the closest result being the $k-\varepsilon$ RNG model in 2D. This is as reported in the literature\cite{1,3,10}, where most turbulence closure models do under-represent this value. Lien et al compared 9 different turbulence transport models and found wildly varying results\cite{9}, although their results were more accurate than my own. Again I suspect this is due to my meshing, and a bit more care taken with the mesh downstream of the step would have yielded better results. In the 2D case in particular, my meshes had nodes with very different sizes due to the lack of biased sizing. This may have improved the results as well.

<table>
<thead>
<tr>
<th></th>
<th>2D</th>
<th>3D</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Nodes  $x_R/H$</td>
<td>Nodes  $x_R/H$</td>
</tr>
<tr>
<td>$k-\varepsilon$ RNG</td>
<td>559  4.56</td>
<td>10220  3.29</td>
</tr>
<tr>
<td></td>
<td>997  5.06</td>
<td>18900  3.80</td>
</tr>
<tr>
<td></td>
<td>1912  5.06</td>
<td>37280  3.80</td>
</tr>
<tr>
<td></td>
<td>3640  5.33</td>
<td></td>
</tr>
<tr>
<td></td>
<td>14154  5.33</td>
<td></td>
</tr>
<tr>
<td>RSM</td>
<td>559  4.37</td>
<td>10220  4.30</td>
</tr>
<tr>
<td>Quadratic</td>
<td>997  4.85</td>
<td>18900  4.61</td>
</tr>
<tr>
<td></td>
<td>1912  4.61</td>
<td>37280  4.85</td>
</tr>
<tr>
<td></td>
<td>3640  4.96</td>
<td></td>
</tr>
</tbody>
</table>
Recommendations

For any future attempts at modelling the flow over a backward-facing step, I would recommend taking care with mesh construction and ensuring that the mesh is fine enough in all flow regions. My meshes were fine enough near the horizontal walls, but far too coarse around the vertical step wall, which probably led to the underestimation of $x_R$.

As is recommended throughout the literature, I would also recommend using a nonlinear $k$-$\varepsilon$ model or one of the more recent models which have shown more promise than the default models included in Fluent. These can be implemented through User Defined Functions and manipulation of constants, but I did not get far enough into the investigation to try those.
References


