CFX Simulation of a Free Surface Water Channel
Flow over a Step

Colin Hartloper

December 6, 2010
Abstract

This report covers a brief study into the two-phase free surface water channel flow over a step profile using Ansys CFX. Three meshes of progressively finer elements were created, all with automatic refinement around the water surface and two with inflated meshes along the bottom of the channel. The vertical wall effects of the channel were neglected by creating a symmetry boundary on the walls of the channel to create an infinitely wide channel.

It was found that the inflated mesh along the bottom of the channel modeled the boundary layer development much more realistically than when an inflated mesh was not present. In addition to this, swirling effects were only noticed when an inflated mesh was present. The sub-critical nature of the flow through the channel was investigated and found to be consistent to its theoretical nature.

A velocity profile was plotted for all three meshes on a plane 50° from the horizontal 225mm downstream of the step, and convergence was demonstrated as the mesh was refined. It was noticed that the base of the plane was located upstream of the reattachment point of the flow – thus the swirling phenomenon was acting on the bottom portion of the plane. This would be detrimental if one wished to place a turbine on this plane, as the spinning turbine blades would have a very non-uniform loading.
## Contents

1 Problem Background 4

2 Theory 5
   2.1 Continuity and Momentum Equations 5
   2.2 Turbulence Model 5
   2.3 Critical Flow 6

3 CFD Set-up 6
   3.1 Geometry 6
   3.2 Meshing 7
   3.3 Boundary Conditions
      3.3.1 Symmetry 8
      3.3.2 Free Surface Condition 8

4 Results 10
   4.1 Boundary Layer Development 10
   4.2 Velocity Profile 11
   4.3 Comparison to Analytical Results 13

5 Conclusion 15

6 References 15
1 Problem Background

A VLHT (Very Low Head Turbine) is a hydro-power turbine which operates with a very low amount of water head relative to other hydro-power turbines. The particular turbine that is being investigated in this report was developed by a French company called MJ2 Technologies. A picture of a working prototype of the VLHT can be seen in Figure 1 [3]. Coastal Hydropower, along with Canadian Projects Ltd (CPL), has bought the North American rights to install the VLHT, however they want to conduct further research into some of the aspects of the turbine before installing any units.

One of the areas that was deemed important to look into is the effect of various upstream geometries on the performance of the VLHT. This is important because several locations that the VLHT could be installed have some sort of geometry in the channel, and if these geometries have a very negative effect on the performance of the turbine then it would either require a modification to the geometry or it would make the location no longer suitable for installation. This paper presents a computational analysis, using the CFX software package, on the effect of the Step Profile (see Section 3 for more information) on the downstream flow conditions in an infinitely wide channel. It especially focuses on the velocity profile at a plane 50° from the horizontal 225mm downstream of the step. This plane is important because the velocity profile could be indicative of the potential power generated by a VLHT placed at this location in the flow.
2 Theory

2.1 Continuity and Momentum Equations

The governing equations that are used in CFX are the conservation of mass and momentum equations. For this particular case, I have two fluids (air and water) and it was a three dimensional problem, thus there were five equations (two mass and three momentum) that were being solved simultaneously at each node.

The two main assumptions that are inherent in the solution is that each fluid is incompressible (which is valid for air at Mach numbers lower than 0.3) and that the simulation has reached a steady state. Under these assumptions the conservation of mass, or continuity equations have the form:

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]  

where:
- \( u \) is the fluid velocity in the \( x \) direction (m/s)
- \( v \) is the fluid velocity in the \( y \) direction (m/s)
- \( w \) is the fluid velocity in the \( z \) direction (m/s)

The conservation of momentum equation in the \( x \) direction can be written as:

\[ \rho \left( u \frac{\partial u}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial u} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + \rho g_x \]

where:
- \( \rho \) is the fluid density (kg/m\(^3\))
- \( \mu \) is the fluid viscosity (kg/(s-m))
- \( g_x \) is the gravitational acceleration in the \( x \) direction (m/s\(^2\))
- \( \frac{\partial p}{\partial u} \) is the pressure gradient in the \( x \) direction (Pa/m)

Note that the conservation in \( y \) and \( z \) direction have a similar form to the equation above.

2.2 Turbulence Model

When the conservation of momentum equations are being solved numerically, you run into a problem when there is turbulence in your system and the magnitudes of the off-axial velocities are changing much more rapidly than the velocity of the axial component. This phenomenon would require a very fine mesh to solve, which is not computationally practical. Therefore, turbulence averaging techniques were developed, which average the velocity components of the conservation of momentum equations and then add on an extra momentum term which accounts for turbulence.
The turbulence model that I chose to use is the k-epsilon model. It introduces two constants into the momentum equations, $k$ and $\epsilon$. $k$ represents the energy carried by the turbulence, while $\epsilon$ represents the length scale of the turbulence. I chose to use this model in my simulations because “the K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients." [2]

The disadvantage to using a turbulence model is that you end up averaging out the actual turbulent effects in the flow and then estimating them by a function of $k$ and $\epsilon$. Because of this your results will probably be less accurate than if you were able to run the simulation without any turbulence model.

2.3 Critical Flow

The Froude Number ($Fr$) for channel flow is a dimensionless number which is the ratio between the water velocity and the gravitational wave velocity [1] (see equation 3). When $Fr < 1$ the flow is subcritical, which is analogous to subsonic flow for a gas. In practice this means that if the flow undergoes a decrease in elevation, it will decrease in velocity and increase depth. When $Fr > 1$, the opposite effect is observed – if the flow undergoes a decrease in elevation the velocity will increase with the depth decreasing. This type of flow is called supercritical, and is unstable and will correct itself downstream by means of a so-called hydraulic jump (analogous to a shock in supersonic gas flow).

$$Fr = \frac{V}{\sqrt{gd}} \quad (3)$$

where:

- $V$ is the average fluid velocity (m/s)
- $g$ is the acceleration due to gravity (m/s$^2$)
- $d$ is the fluid depth (m)

Critical flow theory has an important application in the flow over a step profile, as the flow undergoes a decrease in elevation. See the Results section (pp. 10) for more details.

3 CFD Set-up

3.1 Geometry

The geometry for the simulation was based on specifications set by CPL. The dimensions were scaled down from an ideal full scale situation. A side view of the geometry with dimensions is shown in Figure 2. Note that although the thickness of the geometry was set to be 0.03m, due to the symmetry conditions (as explained in section 3.3) the geometry is actually infinitely thick.
Figure 2: The geometry of the water channel. Note that although it is 1m high the water surface is at an elevation of about 0.34m, everything above that is air.

3.2 Meshing

The CFX meshing program runs a patch conforming iteration to mesh the volume. This means that it starts at the boundaries, as those are where the most significant and small scale changes will happen, and then works its way out away from the boundary with progressively larger elements. Three meshes were ran – a coarse mesh, a coarse mesh with an inflated mesh along the walls, and a fine mesh with an inflated mesh along the walls. The number of elements and nodes of each mesh is listed in Table 1.

<table>
<thead>
<tr>
<th>Simulation #</th>
<th>Mesh Description</th>
<th># of Nodes</th>
<th># of Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Coarse</td>
<td>13812</td>
<td>55960</td>
</tr>
<tr>
<td>2</td>
<td>Coarse with Inflation</td>
<td>22818</td>
<td>74039</td>
</tr>
<tr>
<td>3</td>
<td>Fine with Inflation</td>
<td>27864</td>
<td>96271</td>
</tr>
</tbody>
</table>

Table 1: The number of nodes and elements for each of the three different meshes

The standard element used by the CFX mesh is a tetrahedral element, with four nodes. However, when an inflated mesh is desired, the meshing software places wedge elements on the surfaces that you indicate. These wedge elements help to model the development the boundary layer as they have rectangular horizontal faces, causing them to stack up like bricks rather than randomly oriented triangles (see Figure 3 for a comparison). This causes a layering effect on the velocity profile similar to what you would expect from the boundary layer for a real fluid wall interface. You can view the effect of the inflated boundary mesh in Section 4.1.

The CFX software package encourages that you use auto-mesh refinement whenever you run a simulation with a free surface. This feature will automatically refine the mesh in locations which are experiencing large fluctuations in a given variable. For the case of a free surface flow, we want the mesh to be very fine at the water surface, so the chosen variable will be the volume fraction of water in a given element.

Figures 4 through 6 show the meshes for the three different simulations – coarse,
coarse with inflation, and fine with inflation. Note that the picture is zoomed in on the step, which is the most interesting region of the simulation.

3.3 Boundary Conditions

The boundary conditions for the model are presented in Table 2 below. Note that the velocity and inlet fluid depth values are scaled down from full scale data for the VLHT.

<table>
<thead>
<tr>
<th>Geometry Face</th>
<th>Boundary Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>( u = 0.148 \text{ m/s}, \ h = 337.5 ), hydrostatic pressure distribution</td>
</tr>
<tr>
<td>Outlet</td>
<td>( P_{\text{gauge}} = 0, \ h = 337.5 )</td>
</tr>
<tr>
<td>Front, Back</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Top</td>
<td>Opening, air</td>
</tr>
<tr>
<td>Bottom, Step</td>
<td>Wall, no slip, smooth wall</td>
</tr>
</tbody>
</table>

Table 2: Boundary conditions for the CFX simulation. Please see the symmetry (pp. 8) and free surface condition (pp. 8) sections for more information.

3.3.1 Symmetry

Due to the large increase in elements when auto-mesh refinement is used, a symmetry condition was applied to the two side “walls” of the geometry. This allowed the simulation to solve for the flow conditions without having an excessive amount of nodes and elements. Thus the simulations actually represent an infinitely thick water channel, rather than the 0.3 metre thick water flume that we want to model. However, the analysis may be applicable to the center of the actual flume, where the wall effects (which are essentially neglected in this simulation) are small.

3.3.2 Free Surface Condition

Although the simulation has two materials (air and water), the only region where they interact is at the free surface of the water flow. The term free surface comes

Figure 3: An example of a wedge element (left) and a tetrahedral element (right)
Figure 4: The coarse mesh. Notice the mesh refinement where the free surface occurs.

Figure 5: The coarse mesh with inflation. Notice the horizontally layered elements along the floor of the step profile.

Figure 6: The fine mesh with inflation. Notice the smaller element size.
from the fact that there is zero gauge pressure on the surface of the water (ie the pressure at the surface equals the atmospheric pressure). In order to create the free surface phenomenon, several boundary conditions had to be set across the simulation.

On the inlet boundary, the initial depth of the water was specified, and that region was specified to have a density of 997 kg/m$^3$ (the density of water at room temperature), and was given a hydrostatic pressure distribution. The area above the water was designated to be air (density of 1.3 kg/m$^3$) and had a gauge pressure of zero. The velocity of everything coming from the inlet was set to be 0.148 m/s; although this is not technically true as the air should not be moving, it was much more convenient and was not judged to have a large impact on the results.

On the outlet boundary, the depth of water and the pressure distribution were set equal to the values at the inlet. This could be done because the flow never became super-critical when it went over the step (see Section 4.3), and thus when the flow exited the step region it diffused and retained the original surface height.

On the top boundary, the material was purely air, as it should be.

4 Results

Note that for all of the velocity plots in the results section the velocity being plotted is the superficial water velocity in the x direction, with the x axis being parallel to the length of the channel. The superficial velocity is a volume-fraction weighted velocity – for example the superficial velocity of an element consisting of 100% water is the same as its velocity, however the superficial velocity of an element consisting of 0% water is zero. The superficial velocity is being plotted because the “air” region actually has a very small amount of water in it, and if just the water velocity were to be plotted then the air region would have a very fast velocity, which doesn’t make any physical sense. Since the volume fraction of the water in the air is minimal, when the superficial water velocity is plotted then the velocity in the air region is essentially zero.

4.1 Boundary Layer Development

It is common knowledge that if there is at a fluid solid interface with a no-slip condition the fluid velocity will increase parabolically as you move away from the solid surface. This region that consists of this parabolic flow profile is known as the boundary layer, and once it is fully developed it should have layers of constant velocity for laminar flow. However, if you look at Figure 7, you can see that for the coarse mesh with no inflation, the velocity near to the bottom surface of the step has a rather random distribution. If you compare
this to the uniform horizontal velocity layers observed in the coarse mesh with inflation, you can see that the mesh inflation at the wall is clearly doing its job in modeling the boundary layer much more realistically.

Figure 7: The boundary layer development for the coarse mesh (left) and coarse mesh with inflation (right). The superficial water velocity in the x direction is represented on a blue to white scale with blue being low velocity and white being high velocity.

4.2 Velocity Profile

The superficial water velocity in the x direction is plotted against the vertical elevation along a plane at a 50\(^\circ\) 225mm downstream of the step for the three different meshes is shown in Figure 8. For clarification on where the plane is, see Figure 9. As you can see, as the mesh is refined the velocity profiles seem to be converging. Convergence could not be truly tested however, as any finer meshes caused the solver to crash, possibly due to the maximum element restriction while using a student license.

There are a couple features of Figure 8 worth noting. First off, notice how smoother the parabolic curve is for the finer mesh, due to the smaller element size. Secondly, notice the sharp drop-off of the velocity profile at about 0.32m for the fine mesh, compared to the relatively slow drop-off for the coarse meshes. This drop off is comes about because the superficial water velocity being plotted, and the water surface is located at around 0.33m. Apparently the finer mesh models the free surface as a much sharper transition from air to water, which is more realistic.

The third, and probably most important, item to note is the negative velocity near the floor of the channel. This phenomenon is observed because the boundary layer separates at the end of the step and then reattaches to the floor of the channel somewhere downstream. All of the fluid that is under the line drawn between the detachment and reattaching point is forced into a swirling pattern due to the viscous interactions with the fluid running over it. This swirling pattern is probably easier to visualize graphically, and is shown in Figure 10 for the fine mesh.
Figure 8: Superficial water velocity in x direction vs. height from the bottom of the channel for the three meshes.

Figure 9: The location of the plane on which the velocity profiles were found for the three different meshes.
Figure 10: A vector plot of the superficial water velocity just downstream of the step. Note the backwards flow near the bottom the the plane on which the velocity profile is taken.

This is an important result because, if the simulation indeed represents what happens in the real world, a turbine placed on this plane would not produce an optimum amount of power due to the “dead zone” of fluid directly downstream of the step. In addition to this, the spinning turbine blades would experience a very non-uniform cyclical loading as they rotated, and would be much more likely to become fatigued. Therefore the turbine may have to be placed further downstream, or the geometry may have to be modified.

4.3 Comparison to Analytical Results

Due to the general nastiness of the momentum equations shown in Section 2.1, no closed form solution exists for them unless many simplifications are made. Because of the free surface and non-uniform flow through area of my problem, such simplifications could not be made. However, some sort of analytical verification of the results was necessary, so I decided to find if the results made sense as far as critical flow theory was concerned.

Recall from Section 2.3 that a flow with $Fr < 1$ is deemed sub-critical, while a flow with $Fr > 1$ is deemed to be super-critical. Using equation 3, $Fr$ was calculated for our flow to be 0.24 over the step. This matches the results, as you can see in Figure 11 the water surface is pretty constant throughout the length of the channel.

This results on its own does not prove anything though, as maybe CFX does not know how to handle super-critical flows. A simulation was ran of a faster flow over a small bump in a similar channel to the one used for the step. The $Fr$ over the bump was calculated to be 1.33. Figure 12 shows the water surface for that simulation – notice how the flow accelerates over the bump and how the water depth is significantly reduced downstream of the bump.
Figure 11: The water surface for the flow over the step. With a $Fr = 0.24$, the flow is sub-critical.

Figure 12: The water surface for the flow over a bump. With a $Fr = 1.33$ over the bump, the flow is super-critical.
5 Conclusion

In conclusion, a simulation of a sub-critical flow over a step profile was ran and the resultant downstream velocity profile was plotted. It was found that, as the mesh used in the simulation was refined, the velocity profile seemed to be converging on a set profile, however the convergence was not able to be truly tested as the CFX-solver crashed when a very fine mesh was employed. It was found that the accuracy and realism of the simulation increased when an inflated mesh of brick-like elements was used near the walls rather than the standard tetrahedral element.

In general, the Ansys CFX package does an admirable job of numerically solving a very complex set of equations and presenting the results in an easy-viewing format.

6 References

