

Using Boundary Layers in Fluent.

Introduction.

This note explains how to add a boundary layer profile to a *fluent* flow simulation. The process is straightforward but non-obvious hence this guide. Further details can be found in the fluent manual in Section 6.26 ¹. This guide uses a boundary layer approaching a cylinder as an example.

The Example.

The example consists of a circular cylinder and an end wall. It is shown in Figure 1, only half the problem domain is modelled as the geometry and flow are symmetrical. You should be able to obtain the mesh file from the same place that you got this document.

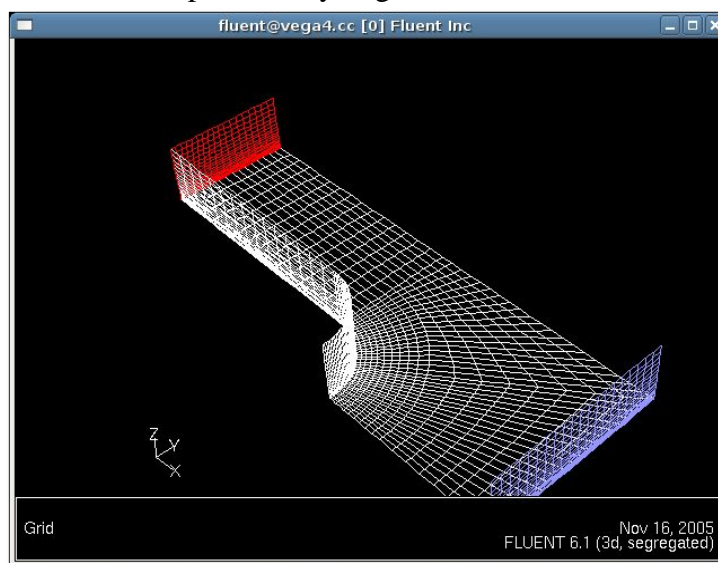


Figure 1: The Meshed Domain.

An example problem domain is meshed and there is a need to add a boundary layer profile to the inlet.

The Boundary Layer

There are a number of ways to generate a boundary layer profile – one could use a $1/7^{\text{th}}$ power law profile or experimental data. What we need for Fluent is a set of points that cover the area of interest for our boundary layer. In our example our cylinder is 0.3m high and we wish to specify a $1/7^{\text{th}}$ power law boundary layer to a height of 0.1m. An example spreadsheet with a boundary layer profile should be in the same place that you found this file and the actual profile to be used is shown in Figure 2.

Note that fluent performs a zero order interpolation (i.e. it doesn't interpolate but picks the nearest value!) so you need to ensure any boundary layer profile you produce has a large enough number of

¹ If you are using an ITS Linux machine this can be found at <file:///usr/local/fluent/current/Fluent.Inc/fluent6.1/help/html/ug/node262.htm#sec-bc-prof>

points to give a smooth profile.

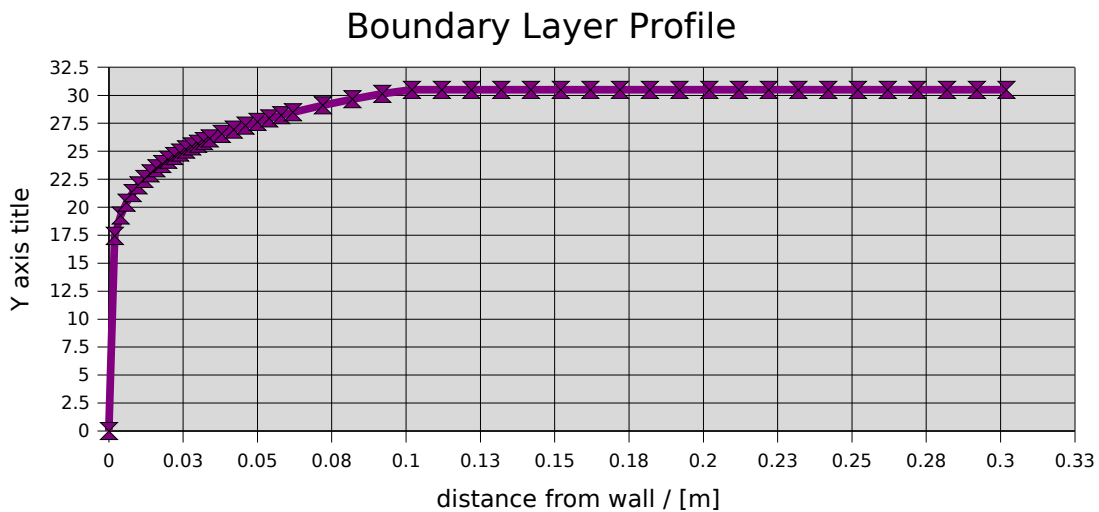


Figure 2: Boundary Layer Profile

The Boundary Layer Profile File.

To tell fluent about the boundary layer we need to specify the velocity variation and its coordinates in a so called “*boundary profile file*”. The boundary layer shown in Figure 2 consists of forty nine points from 0 to 0.3m in the 'z' direction and is uniform across the flow inlet. The inlet plane shown in Figure 1 is located at 1.2m upstream of the cylinder. So we want to specify a boundary layer profile at $x=1.2\text{m}$, $y=0\text{m}$ and $z=0$ to 0.3m. The required boundary profile file is shown in Figure 3

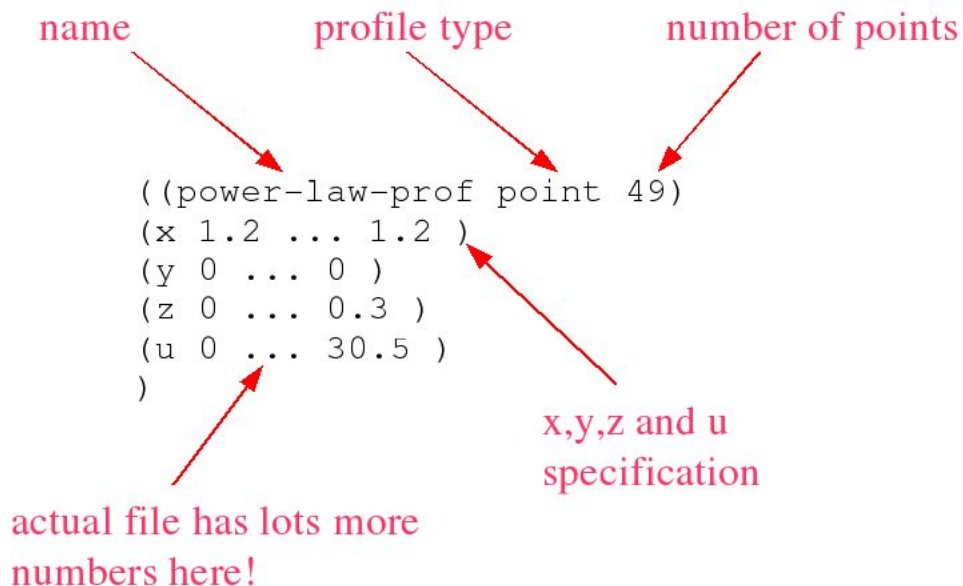


Figure 3: Boundary Profile File Format

The full profile file should be in the same place as this note. **Note that all units must be SI as fluent doesn't scale profile files.**

Setting Boundary Conditions.

File -> Read -> Profile will read the profile into fluent. Check the output in the main fluent window to ensure that the data has been successfully read in.

Define -> Boundary Conditions and set the inlet to be a velocity inlet. You should then make changes to the dialogue box until it looks like Figure 4.

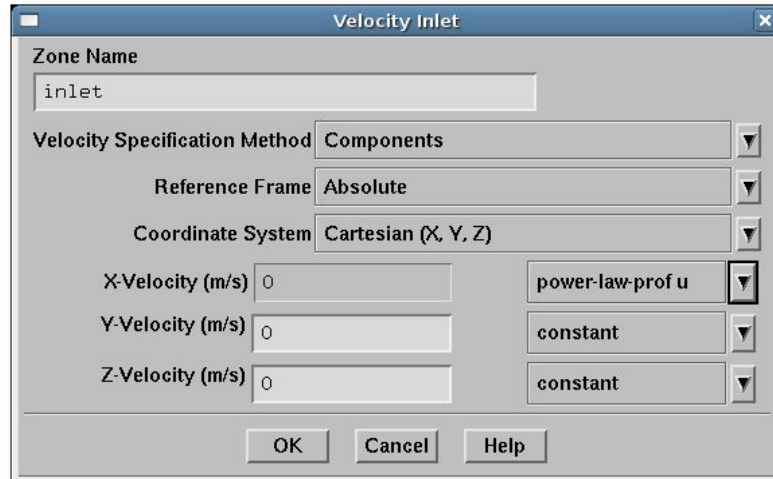


Figure 4: Velocity Inlet Dialogue

You should then be able to run your simulation in the normal way. Figure 5 shows the results of a quick simulation showing the vortices produced by in the inlet boundary layer in the plane of symmetry.

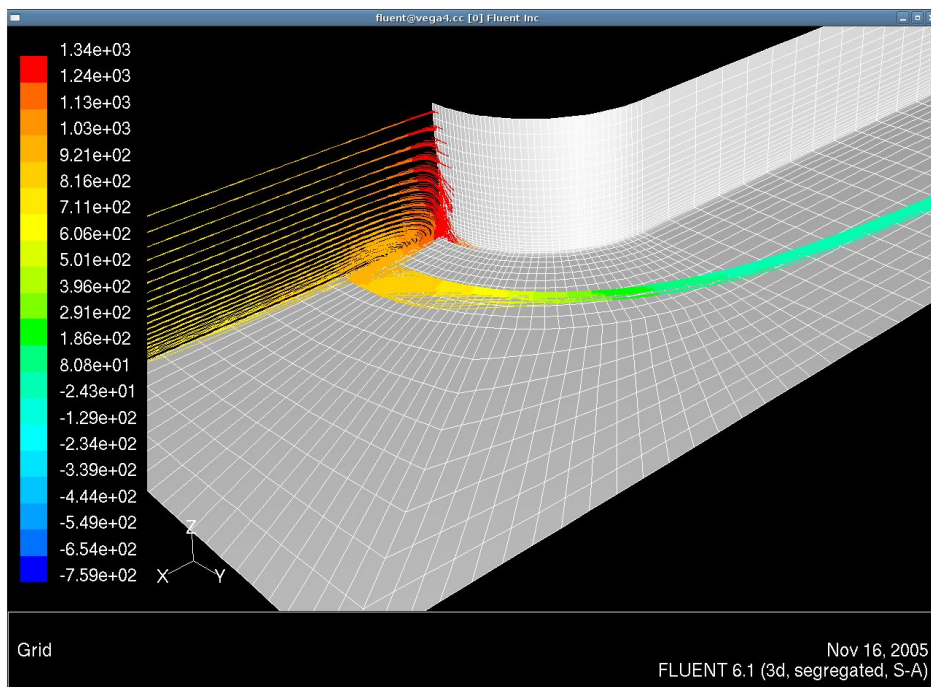


Figure 5: Solution showing boundary layer influence.

Additional files.

In addition to this document you should have the following files:

- 3dmeshnov05.msh - fluent mesh file of the problem
- boundary layer.sxc - spreadsheet (in OpenOffice format) with boundary layer profile in it
- power law profile.prof -boundary profile file

Grant Ingram

16 November 2005