

# Mid-term: 2d-axis and 3d laminar flow computation in Ansys Fluent

Mingxi Chen 999019482

May 12, 2024

## 1 Computational domain

### 1.1 2d

$$R = 0.2m, L = 2.4m$$

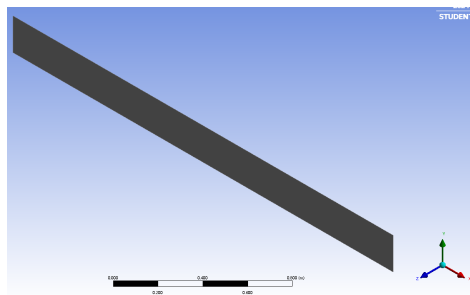


Figure 1: 2D domain

### 1.2 3d

$$R = 0.2m, L = 1.2m$$

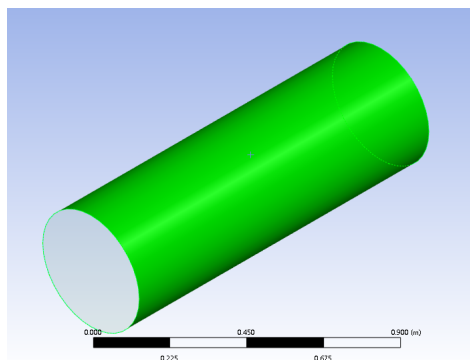


Figure 2: 3D domain

## 2 Mesh and its parameters

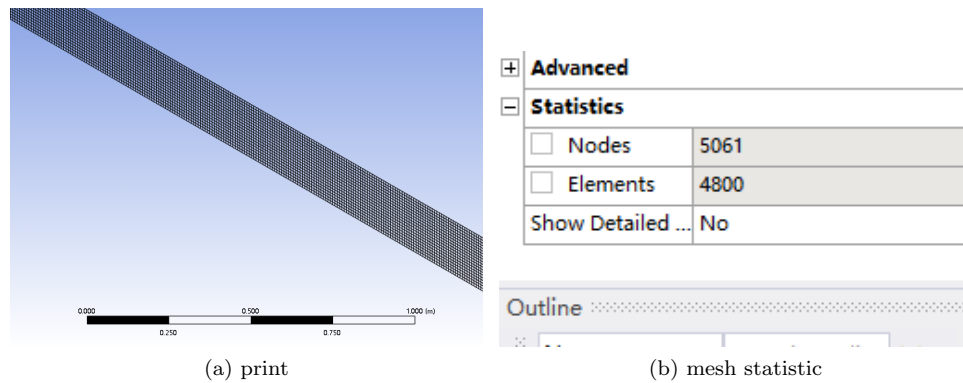


Figure 3: 2D mesh

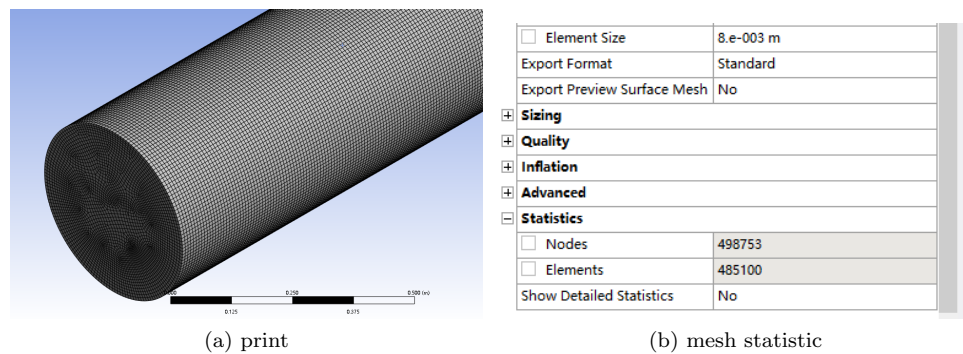


Figure 4: 3D mesh

The different mesh num done in this assignment:

2D mesh	3D mesh
192	3792
560	<b>*6150</b>
763	17015
1200	53558
<b>*2080</b>	177860
5874	282414
7500	485100
9946	726350
19200	-

Table 1: mesh setting

## 3 Model setup and boundary conditions

Since no turbulent flow will be took into consideration in this task, Laminar model was chosen, the comparison of different models(SST,  $k - \epsilon$ ) will be presented later. Boundary conditions for two Reynolds' number were used to do simulation.

Re	$v_{in}$
95.589	0.0001 $m/s$
440	0.000463 $m/s$

Table 2: inlet velocity

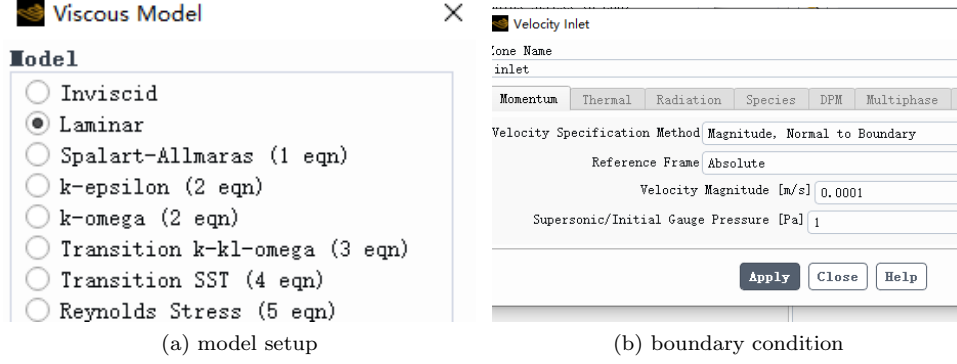


Figure 5: 2D setting

## 4 Post analysis

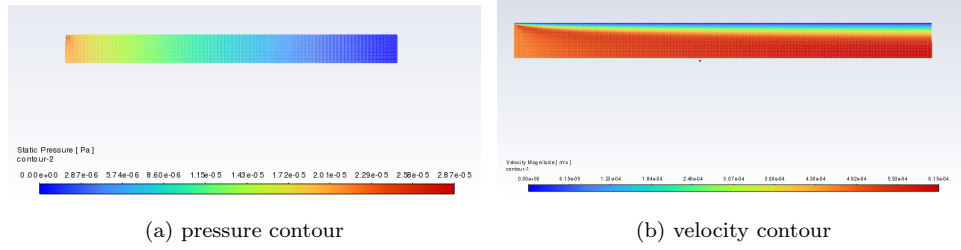


Figure 6: 2D results in Fluent

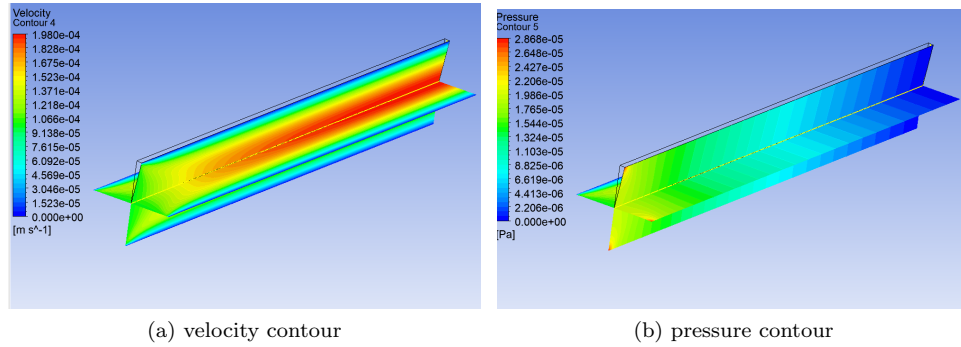


Figure 7: 2D results in CFD-post

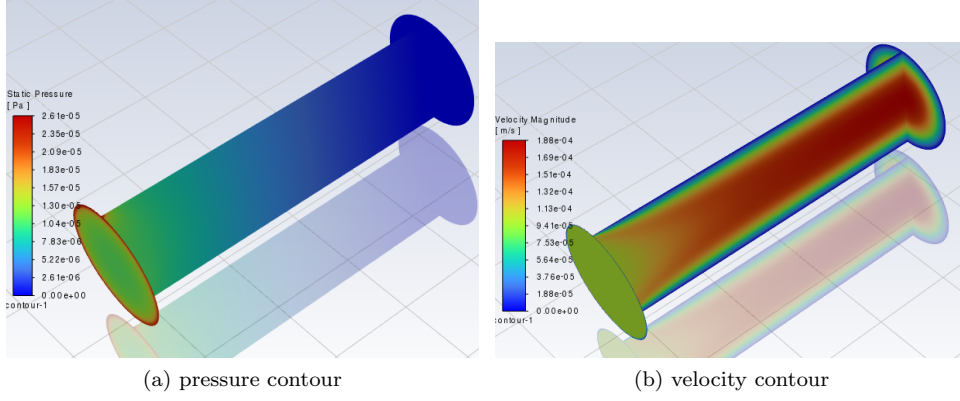


Figure 8: 3D results in Fluent

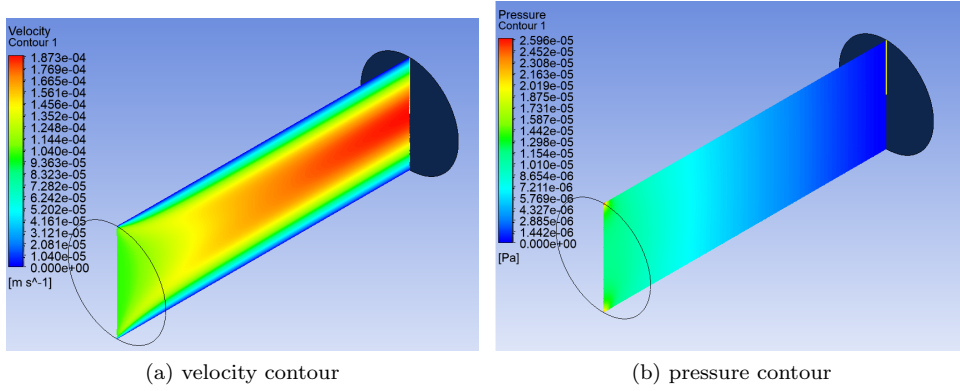


Figure 9: 3D results in CFD-post

## 5 Error analysis

### 5.1 2D case:

To perform error analysis, we collect data from CFD-post to compare with analytical solution of fully developed laminar pipe flow. Import  $u(r)$  in CFD-post to .csv file and use excel to calculate the  $err(r) = \frac{|u(r) - u'(r)|}{u(r)}$ . The num of sample points in r is 100 in each csv file.



Figure 10:  $v_{in} = 0.0001 m/s$

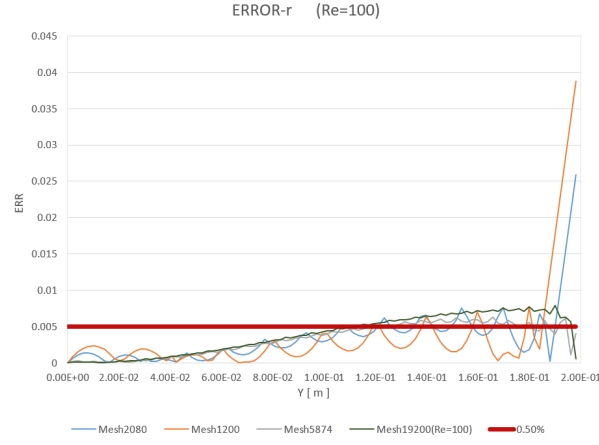


Figure 11: Plot error as function of r

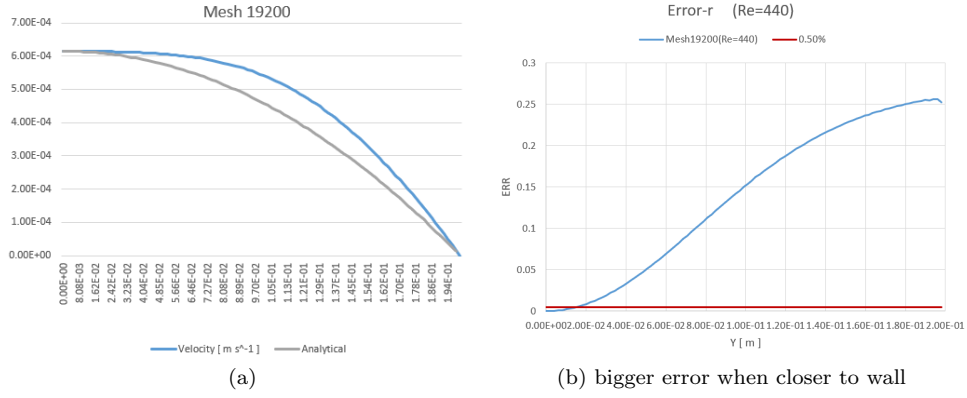


Figure 12:  $v_{in} = 0.0004603m/s$

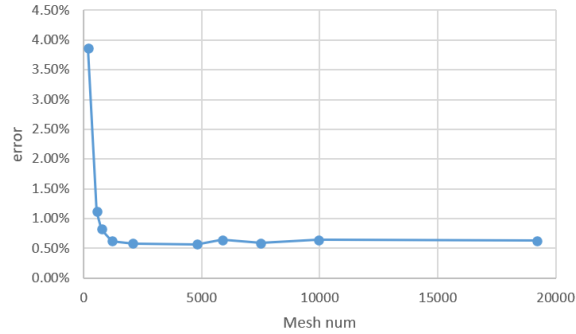


Figure 13: err in different quantities of mesh elements

Conclusion: Errors of the numerical simulation are bigger when Re is high(Re=440 in this case), especially the area where fluid is closer to the wall. Small errors compared to analytical solution when Re is relatively small(Re=95.589 in this case), and minimal number of mesh elements to get solution error less than 0.5% is around **2080**.

## 5.2 3D case:

Same methodology was applied to 3D case:

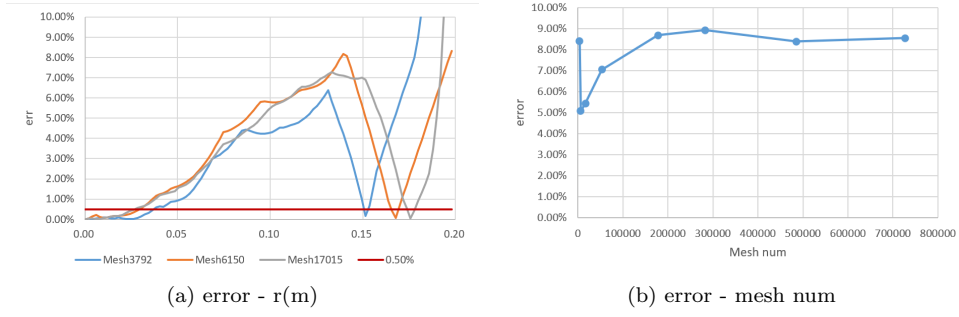


Figure 14:  $v_{in} = 0.0001 m/s$

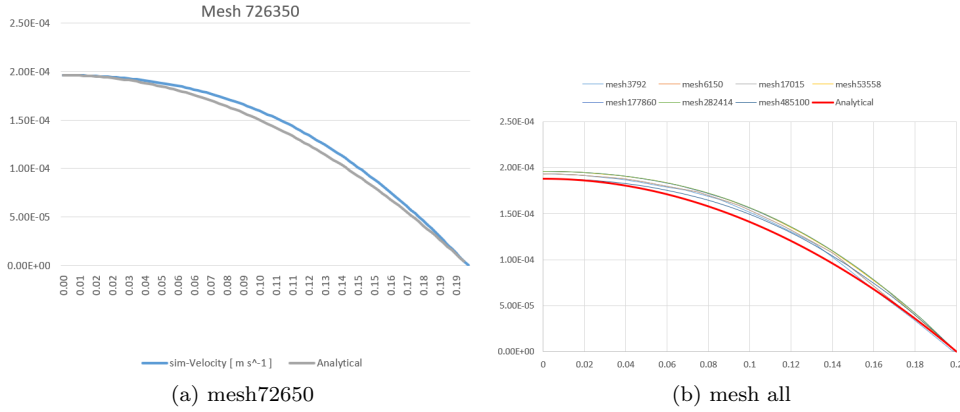


Figure 15: velocity distribution

Conclusion: Minimal number of mesh elements to get the least solution error is around **6150**. The overall error is significantly bigger than 2D case, presumably because a) the pipe length is not long enough to get fully developed flow b) the  $y^+$  dimensionless parameter is not set properly c) defects of mesh.

Due to the limited time I have, further improvement can't be done in this assignment. However, the main source of error is the fluid near the surface, which can be observed in both 2D and 3D case, consistent with conjecture.