Romain PINQUIE

MSc Computer Aided Engineering

Computational Fluid Dynamics

Assignment Report

09/01/2012

Cranfield University – School of Engineering

I. INTRODUCTION 3

II. FORMULATION OF THE FLOW PROBLEM 3

1. What is the objective? 3

2. What is known? 3

3. What is best analysis approach? 4

III. GENERATION OF THE GRID 6

IV. SPECIFICATION OF THE BOUNDARY AND INITIAL CONDITIONS 9

V. SET UP THE CFD SIMULATION 10

1. Physical model 10

2. Operating conditions 12

3. Numerical algorithm inputs 15

VI. EXPECTATIONS 16

1. Pressure 17

2. Velocity 17

3. Stream pressure 18

4. Stream velocity 18

VII. PERFORMING AND MONITORING OF THE CFD SIMULATION 19

1. Fine grid 19
   A. 4 l/s 19
   B. 8 l/s 21
   C. 12 l/s 22

2. Coarse grid 23
   A. 8L/s 23

VIII. EXAMINATION AND PROCESSING OF THE CFD RESULTS 24

1. Fine grid 25
   A. 4L/s 25
   B. 8L/s 29
   C. 12L/s 33

2. Coarse grid 37
A. Pressure evolution 37
B. Velocity magnitude evolution 37
C. Relative total pressure 38
D. Velocity magnitude 39
E. Velocity stream 40
F. Observations & Comments 41

IX. FURTHER ANALYSIS 42

1. RNS 42
A. Relative total pressure evolution 43
B. Velocity evolution 44
C. Relative total pressure 44
D. Velocity Magnitude 45
E. Velocity stream 45
F. Observations and comments 45

X. CONCLUSION 46

I. Introduction

The aim of this report is to display the results of the simulation carried out and naturally discuss them. Moreover, the main goal of the simulation was to investigate the fluid behaviour, which in this case corresponds to a water flow through a variable area venture meter. More precisely, the pressure difference exerted on a bluff body object and the flow field from the inlet to the outlet have been studied. The report contains the following steps: simulation technology, results, discussion and a conclusion. It is necessary to underline that all the simulation have been performed by using the CFD package ANSYS FLUENT.

II. Formulation of the flow problem

1. What is the objective?

Concretely, the objective of this CFD simulation is to compute different physical properties (pressure and velocity). Two meshed grids provided with the assignment have been used in order to get the necessary data to assess the simulation. The first one is not elaborate, it is a coarse grid of 13, 580 cells, whereas the second is more accurate since there are 54,118 cells. Using both the pressure difference 25mm upstream and downstream of the top plate for each of the three inlet flow rates have been studied. In addition, the pressure difference 25mm upstream and downstream of the top plate using the coarse grid has also been investigated to compare the results with the fine one. More details of the environment are available in the next parts which contains sketch and initial values of the problem.

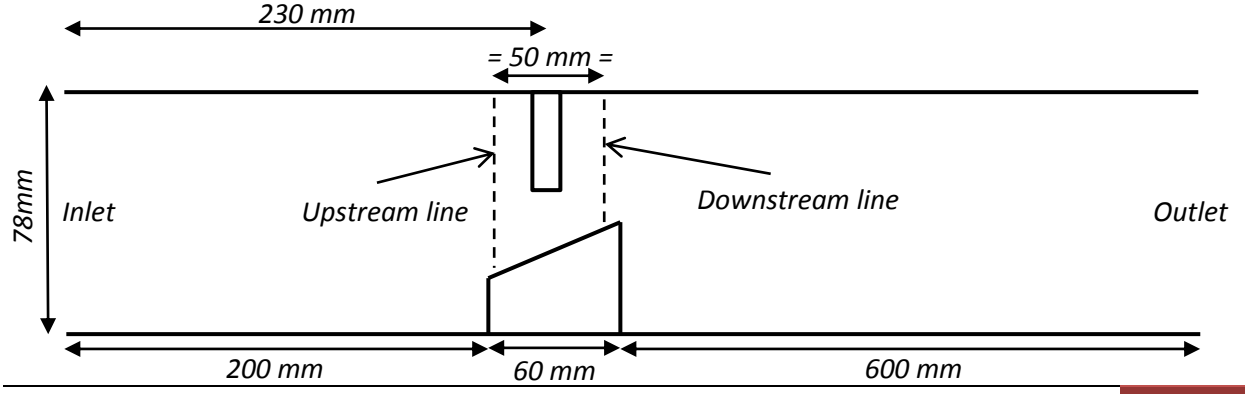
2. What is known?

The freestream conditions provided correspond to the inlet flow rates, which are 4l/s, 8l/s and 12l/s, the conversion to international system of units, i.e. m/s is exposed in the next step.

Water properties:

- *Density (ρ) = 998.2 kg/m^3; Dynamic viscosity (μ) = 0.001003 kg/ms*

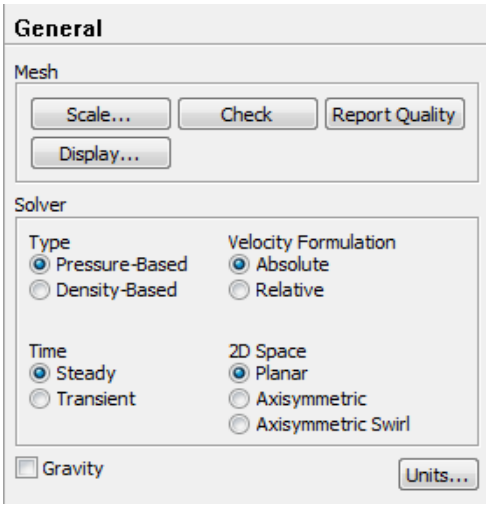
Layout of the environment:



3. What is best analysis approach?

To set up the simulation, the fluid flow had to be characterized by calculating several data including inlet velocity, Reynolds numbers and turbulence intensities. All these values allow the classification of the fluid flow and obviously to anticipate its behaviour through the venture meter.

Water flow is considered as **incompressible**, which means that its density remains constant throughout the flow. Furthermore the stream flows through a venture meter, corresponds to an **internal flow**. In this case, the flow does not change with time, it is called **steady flow**. Therefore the governing equations are the following:



Problem:

Fluid: Water
Velocity 1 (u1) = 4 l/s
Velocity 2 (u2) = 8 l/s
Velocity 3 (u3) = 12 l/s
Pipe Height (D) = 0.078 m
Dynamic viscosity (μ) = 0.001003 kg/ms
Density (ρ) = 998.2 kg/m^3

For the following calculations the use of abbreviations will be usefull (μ=mu and ρ=rho).

```
u1:=4; u2:=8; u3:=12; D:=0.078; mu:=0.001003; rho:=998.2;
 4

 8

 12

 0.078

 0.001003

 998.2
```

Conversion of velocities from *l/s* to *m/s*:

```
v1:=(u1*10^-3)/(PI*(D/2)^2); float (v1)
```

$$\frac{2.629848784}{\pi}$$

0.837106867

```
v2:=(u2*10^-3)/(PI*(D/2)^2); float (v2)
```

$$\frac{5.259697567}{\pi}$$

1.674213734

```
v3:=(u3*10^-3)/(PI*(D/2)^2); float (v3)
```

$$\frac{7.889546351}{\pi}$$

2.511320601

=> Velocity 1 = 0.8371 m/s; Velovity 2 = 1.6742 m/s; Velocity 3 = 2.5113 m/s

$$v_i = \frac{u_i.10^{-3}}{\pi\left(\frac{D}{2}\right)^2}$$

The Reynolds numbers have been calculated using the following definition:

$$Re = \frac{\rho v D}{\mu}$$

Reynolds numbers calculation:

```
Re1:=(rho*v1*D)/mu; float(Re1)
```

$$\frac{204146.5348}{\pi}$$

64981.86024

```
Re2:=(rho*v2*D)/mu; float(Re2)
```

$$\frac{408293.0695}{\pi}$$

129963.7205

```
Re3:=(rho*v3*D)/mu; float(Re3)
```

$$\frac{612439.6043}{\pi}$$

194945.5807

=> Reynolds number for velocity 1 = 64981; Reynolds number for velovity 2 = 129963; Reynolds number for velocity 3 = 194945

Reynolds number is a ratio between inertial and viscous forces, in our cases the Reynolds numbers are quite high, which indicate that the inertial forces are more significant than the viscous force. Therefore, the flow has been assumed to be an **inviscid flow**, thus the viscosity can be neglected compare to inertial terms. Moreover, a Reynolds number bigger than 3000 involves turbulent flows that are evaluated in the next part by using the following formula:

$$T = 0.16 * \left(\frac{\rho u D}{\mu}\right)^{-\frac{1}{7}}$$

Turbulence intensities calculation:

```
T1:=0.16*Re1^(-1/7); float(T1)
```

$$\frac{0.16}{\sqrt[7]{\frac{204146.5348}{\pi}}}$$

0.03285324318

```
T2:=0.16*Re2^(-1/7); float(T2)
```

$$\frac{0.16}{\sqrt[7]{\frac{408293.0695}{\pi}}}$$

0.0297559598

```
T3:=0.16*Re3^(-1/7); float(T3)
```

$$\frac{0.16}{\sqrt[7]{\frac{612439.6043}{\pi}}}$$

0.0280813556

=> **Turbulence intensity for velocity 1 = 3,29%; Reynolds number for velovity 2 = 2,98%; Reynolds number for velocity 3 = 2,81%**

To analyse the flow's physical properties it is necessary to solve the Navier-Stokes by approximation, obviously. The most common model to compute all information required is to use the K-Ɛ Model (2 equations) and to have a second point of view that would be interesting to undertake another simulation using the Reynolds Stress Model (5 equations) and compare the results.

III. Generation of the grid

No grid generation is required for this assignment since it has been provided for this project, but to make sure that it is the good one a checklist is recommended. The use of the following tools apply to the *fine grid* model allow displaying its properties.

Firstly, the existence of errors has been sought.

```
Mesh Check

 Domain Extents:
   x-coordinate: min (m) = 0.000000e+00, max (m) = 8.600000e-01
   y-coordinate: min (m) = -3.900000e-02, max (m) = 3.900000e-02
 Volume statistics:
   minimum volume (m3): 9.013846e-09
   maximum volume (m3): 4.322714e-06
     total volume (m3): 6.468500e-02
 Face area statistics:
   minimum face area (m2): 1.242559e-04
   maximum face area (m2): 4.431793e-03
 Checking mesh.....................
Done.
```

The output window (see above) indicates that there are no errors. Therefore it is almost certain that the mesh file has been loaded correctly.

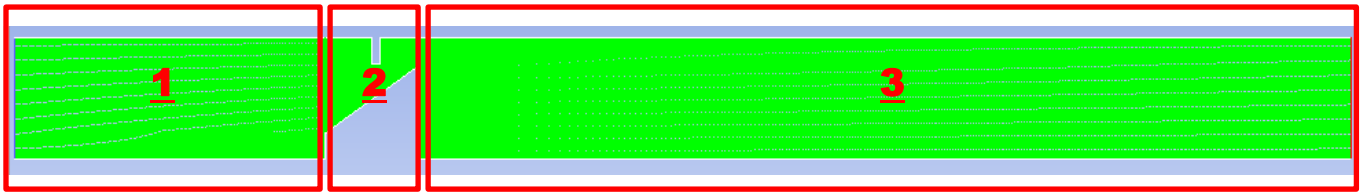
Secondly, the grid size has been checked using the *size* tool:

```
Mesh Size

Level    Cells    Faces    Nodes   Partitions
    0    54118   101121    47004            1

 1 cell zone, 5 face zones.
```

The number of cells given in the paper sheet (54,118) is the same as ANSYS FLUENT (see above), then it is sure that the mesh corresponds to the fine grid.

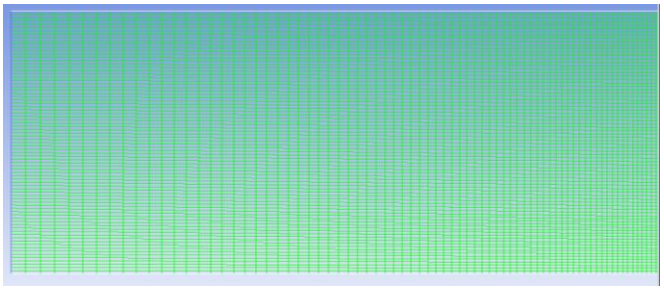
Now that the grid has been correctly loaded and defined, the simulation can be continued by displaying it.

The grid displayed is the good one, now it is possible to analyse more precisely its design. By focusing on the different areas (1, 2 and 3) it is noticeable to see that several methods of meshing have been used.



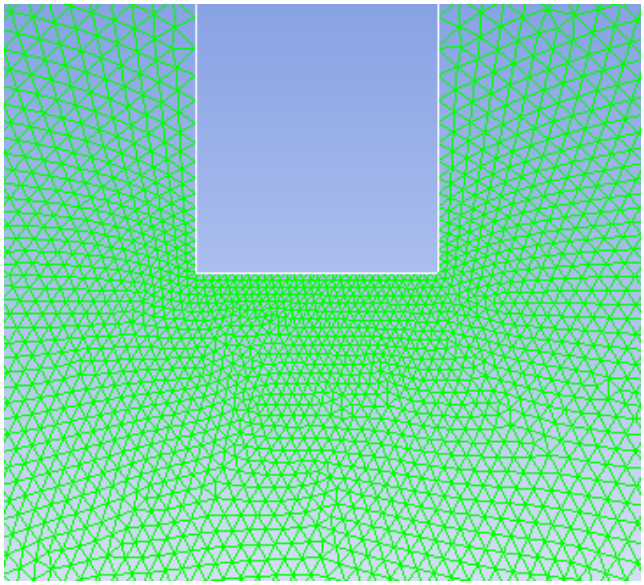
- **Area 1:**

Going into details of the zone 1 it is easy to notice that the kind of mesh is single-side grading towards the constricted area since the length of mesh elements decrease as one goes along which provide a best accuracy of the results in the more interesting domain, the variable cross-section.



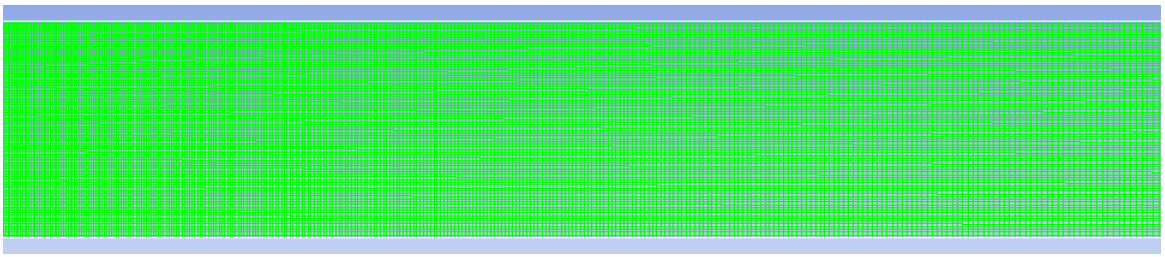
- **Area 2:**

As visible on the following picture, an unstructured mesh has been chosen at the zone 2, which means that the cells as well as points have no particular ordering. Moreover, a 2D triangular method has been preferred to design the meshed grid, which allows improving the accuracy of the results by having a good space discretization of complex shapes like right angle.



- **Area 3:**

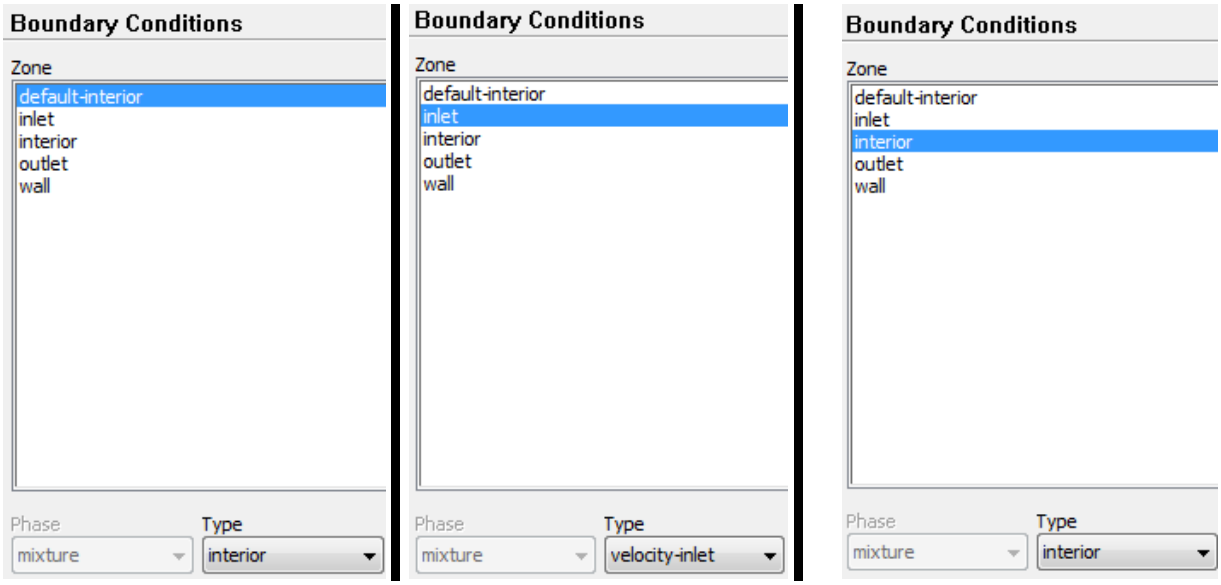
As for the first area the grid has been designed with the single-sided grading method but with the opposite direction always to get the best results at the shrunk area.

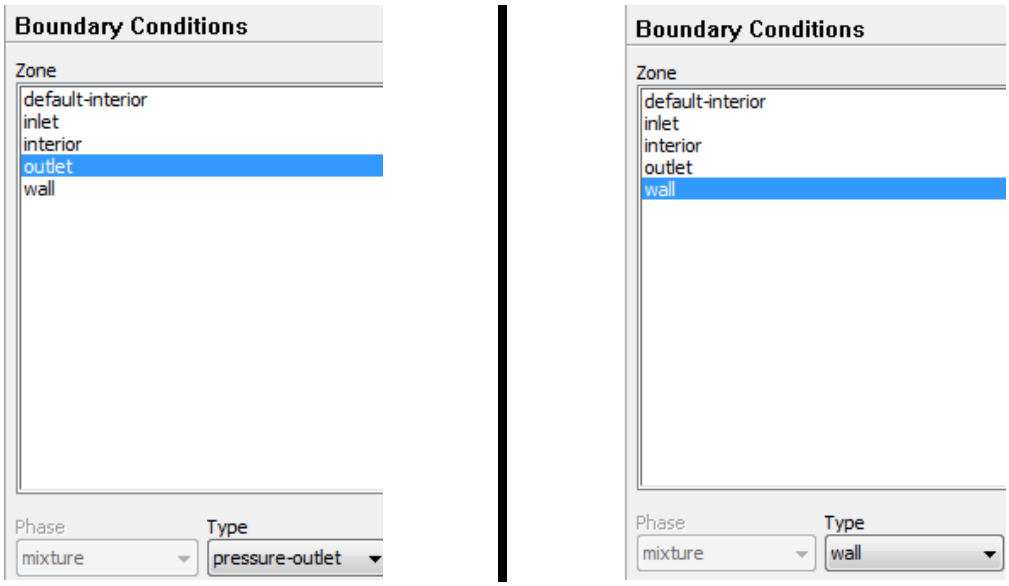


To sum up, the meshed grids have been designed so that the finest area match with the turbulence zone, the elements have been chosen to trace the real shape and with the main goal to be enough accurate in the critical area (2).

IV. Specification of the boundary and initial conditions

In order to check whether the boundary conditions correspond to the settings previously defined in IcemCFD or not, the use of the *Boundary conditions* menu may be useful:



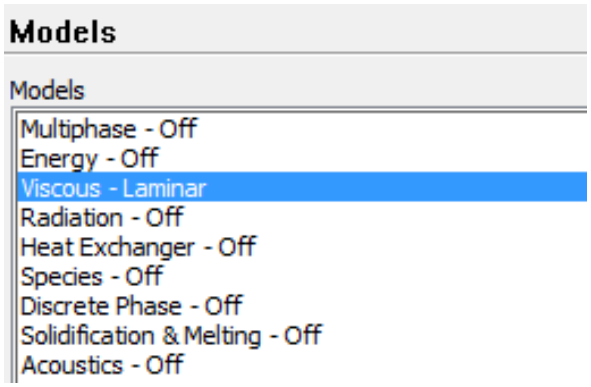


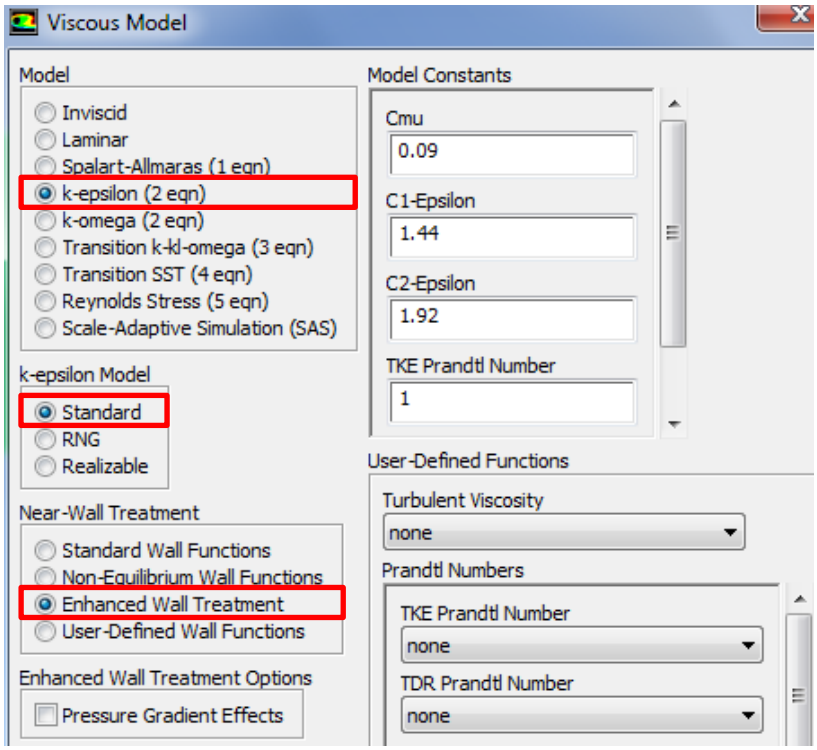
All the boundary conditions contain in the screenshots printed above match those defined previously in IcemCFD, e.g. the type of the *inlet* zone is a *velocity-inlet* and the *outlet* is a *pressure-outlet* as required.

V. Set up the CFD simulation

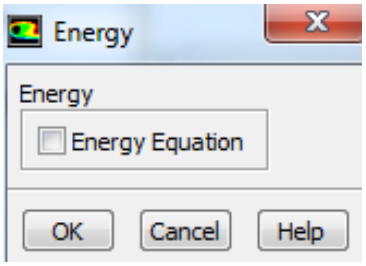
1. Physical model

The K-Ɛ model has been chosen as turbulence model because it is the most widely used model for industrial application. Moreover, the K-Ɛ turbulence model was applied with wall functions, for this case the *Near-Wall Treatment* function has been selected so that the results got are as accurate as possible close to the boundaries defined as walls. Scientifically, the viscous layer is not resolved, but approximations are introduced to account for the flow behaviour across it. With this model velocity and pressure are uncoupled and the equations are solved one after the other, computing the magnitude of the turbulent viscosity from the turbulence kinetic energy (K) and the dissipation rate (Ɛ).

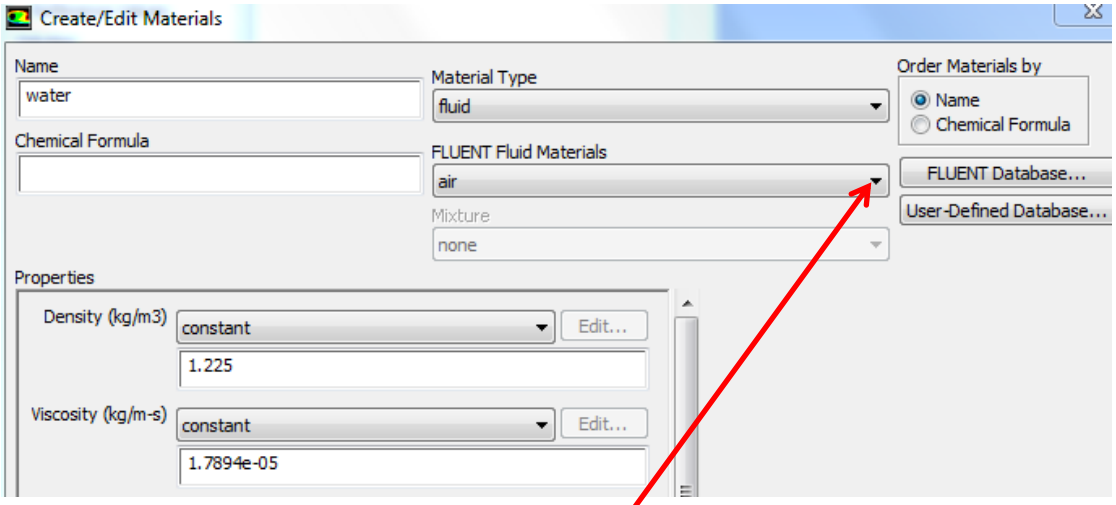


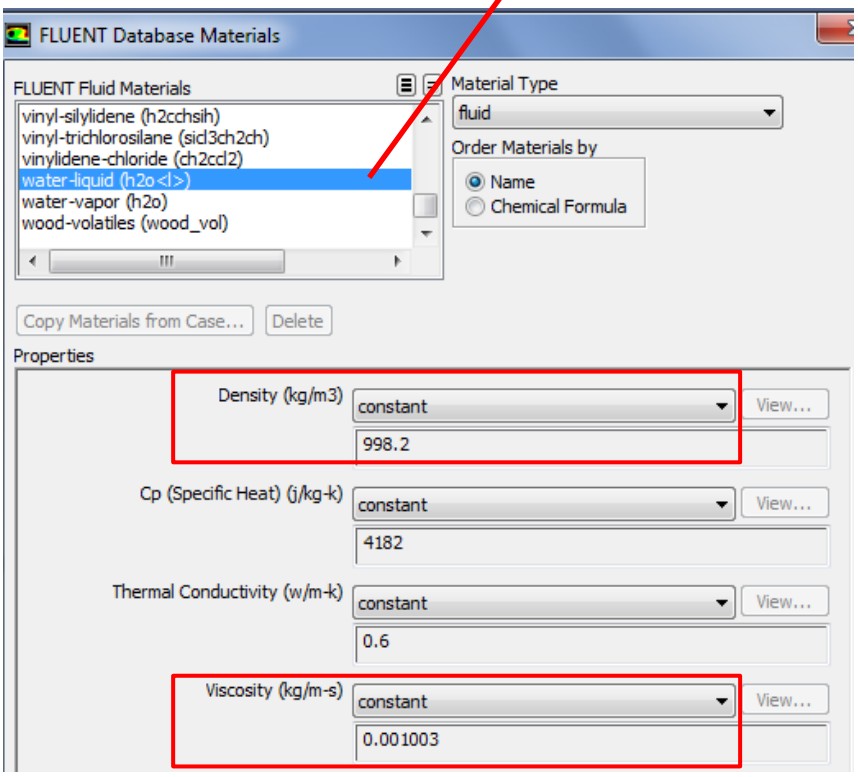


The energy equation has been turned off since it is an incompressible flow and the temperature properties do not matter us in this simulation.

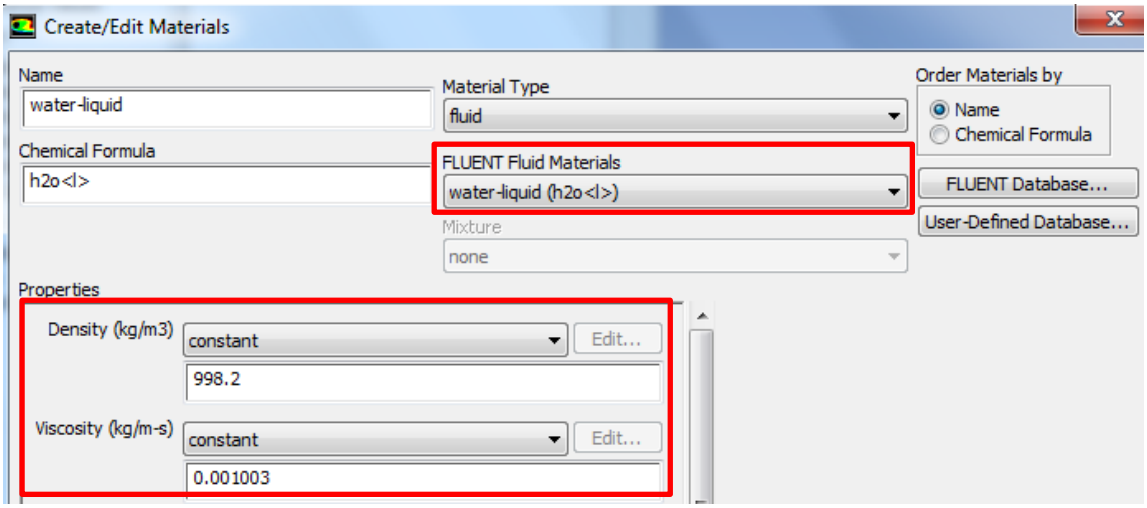


Then the material required, which is the water has been defined with its physical properties using the *Fluent Fluid Materials* library.



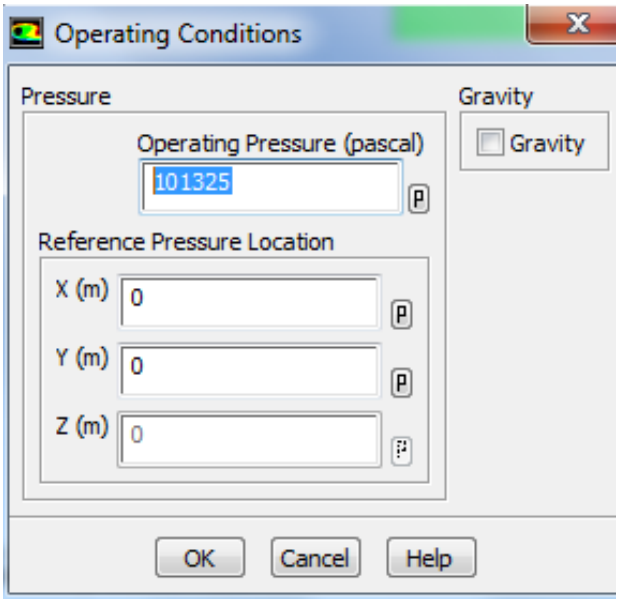


Therefore the final windows must correspond to the following:

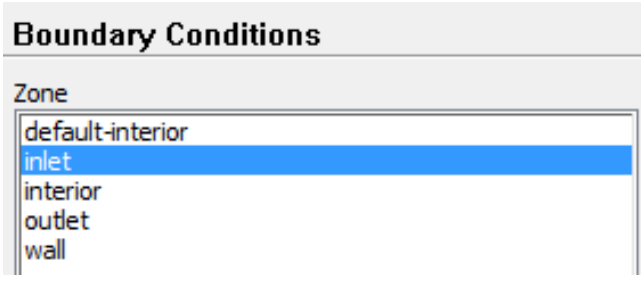


2. Operating conditions

Any time an absolute pressure (operating pressure + gauge pressure) is needed that is why the operating pressure has been defined as 1 tam (101325 Pa).

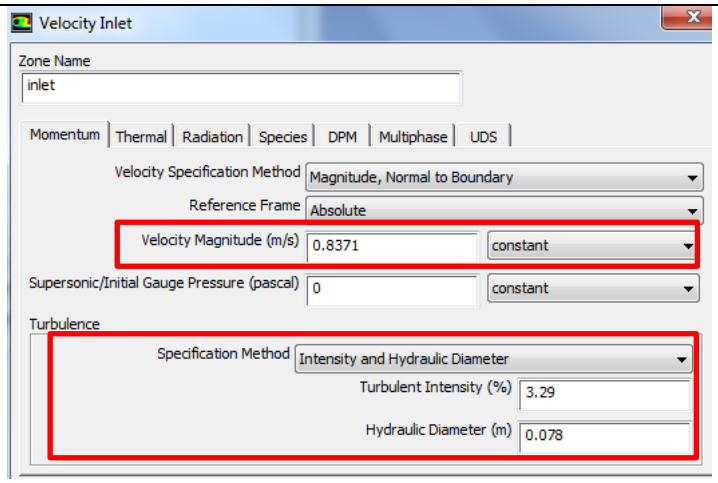


Moreover, for each flaw rates (v1, v3 and v3) the velocity magnitudes and the turbulent intensities calculated at the beginning had to be filling in and the hydraulic diameter stays constant to 0.078 m for all the simulations.

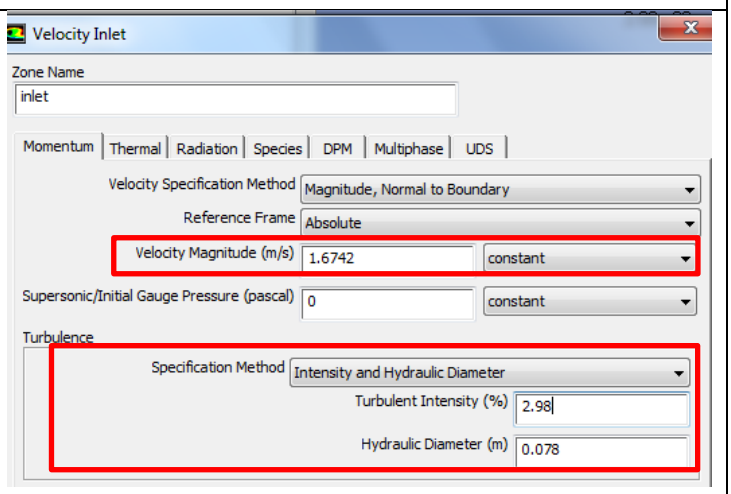


Choosing Intensity and Hydraulic Diameter as *Specification Method* of *Turbulence* the values may be fill in. The following pictures show the inlet specifications and the outlet, which is common to all cases.

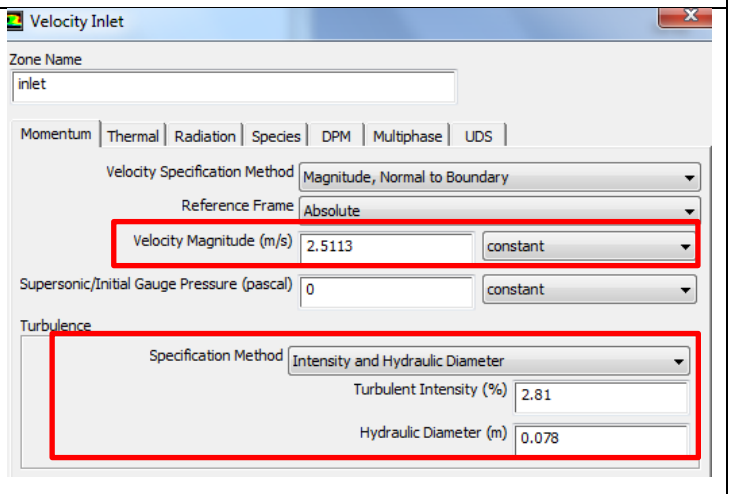
v1= 4l/s

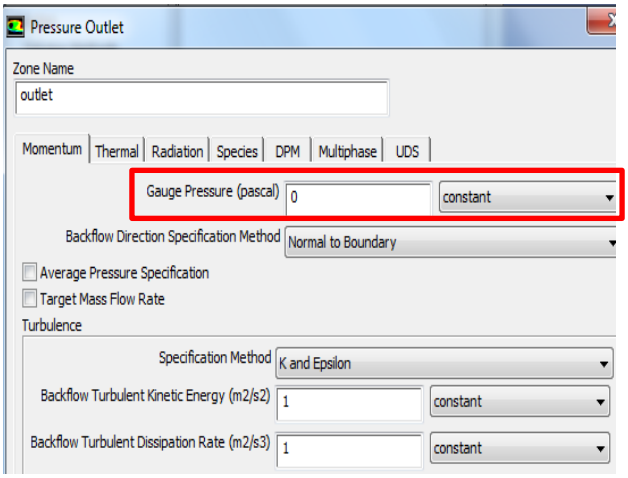


v2= 8l/s



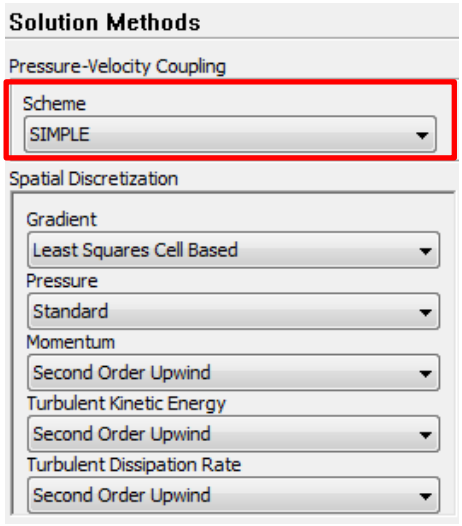
v3= 12l/s



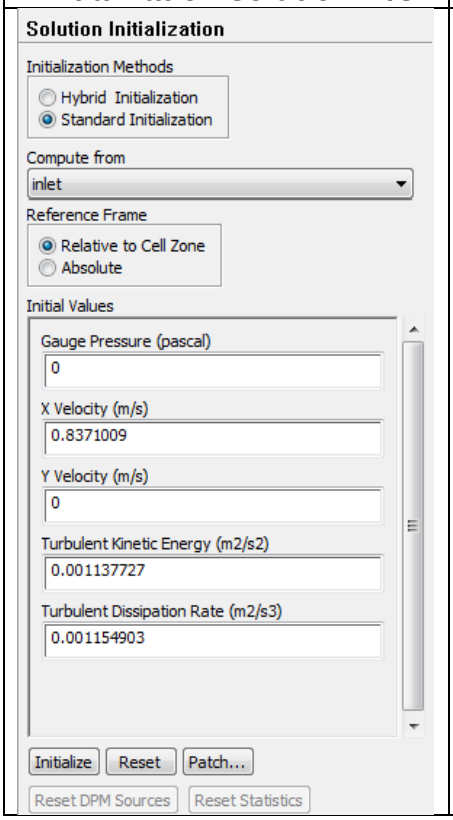
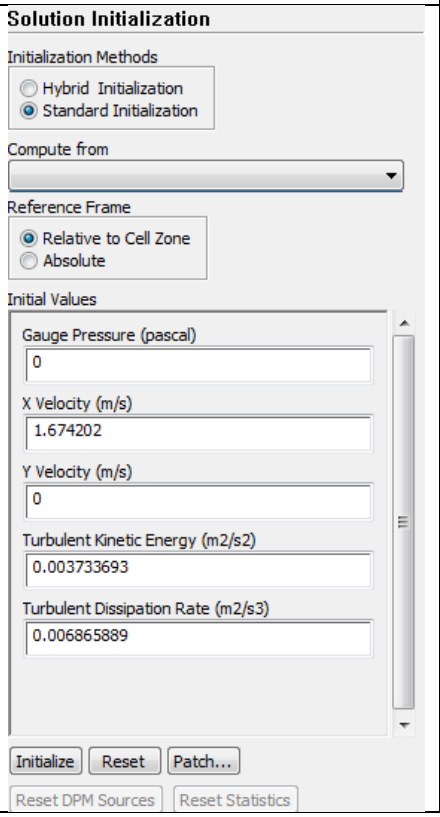
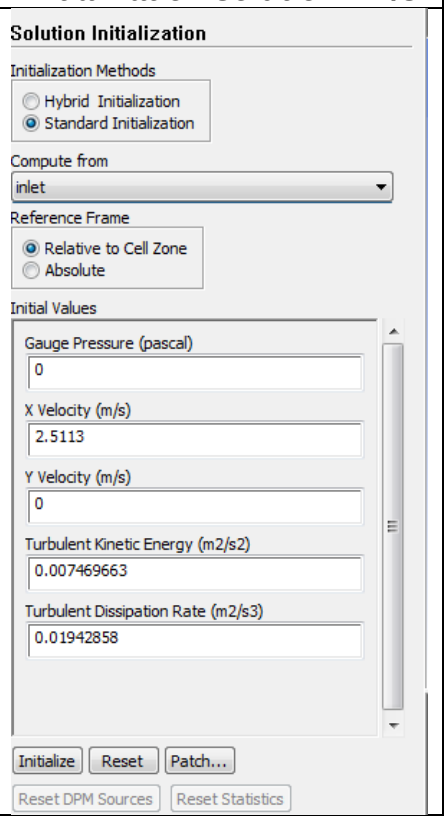


Make sure that the *Gauge Pressure* is equal to 0 Pa since the *absolute pressure* at the outlet is 1 atm.

The *SIMPLE (Semi-Implicit Method for Pressure-Linked Equations)* algorithm has been chosen to solve the Navier-Stokes equations. It has been preferred to the other, because it requires an under-relaxation which increases stability (smoothing).

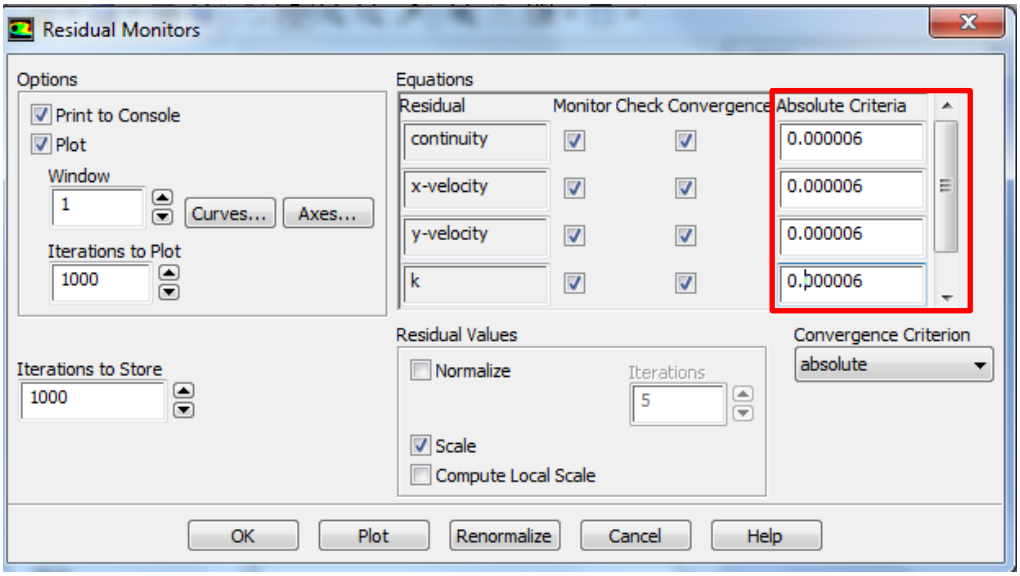


Then, the next operation has been to initialise the solution setting the *initial value*s from the *inlet* therefore the values of the *X Velocity* and *Turbulent Kinetic Energy* must be the same as specified in the previous step.

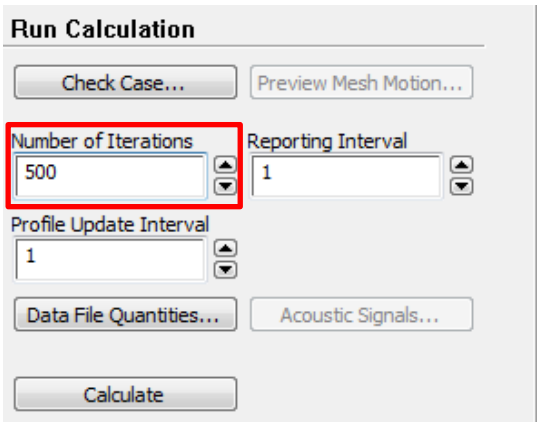
Initialization solution 4l/s	*Initialization solution 8l/s*	*Initialization solution 12l/s*
		

3. Numerical algorithm inputs

CFD software is able to solve the Navier-Stokes equations using discrete form of each governing equations, thus a residual is calculate at each iteration in order to analyse the convergence of the solution. In this case, the *absolute criterion of convergence* has been fixed to **6x10^{-6}**. Residal smaller than 1*e*-06 is a sufficient but not a necessary condition in order to claim that the solution converges. In fact, a residual equals to 1*e*-04 can also proof the convergence of the solution. In general, convergence means that the residual of conservation equations should approach zero as the number of iterations increases or that the results change very little from the previous refinement.

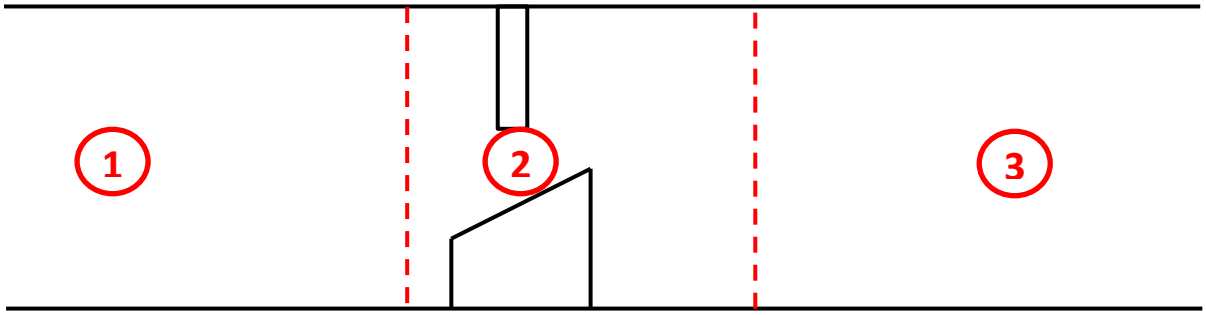


the number of iterations needed to solve the problem is determined in function of the powerfull of the computer, the cost and the time allocated for the simulation and other confines. In this survey the number of iterations has been estimated at 500 to get the necessary information. If the residual is not smaller than 1*e*-06, then the operator decides to start more iterations or to stop it when he considers that the accuracy is sufficient.



VI. Expectations

To make expectations of the results the domain has been spited into 3 areas as following:



Calculations:

The water being incompressible, the flow rate can be writing as following:

- **Q_v = V.S**
  - *Q_v = flow rate (m^3/s)*
  - *V = velocity (m/s)*
  - *S = cross-sectional area (m^2)*

In these cases **q_v** stays **constant** through the venture, consequently by writing **V=Q_v/S** we remark that the **velocity increases when the cross-sectional decrease and *vice versa***.

Thus, $Q_{v1} = V_1.S_1 = Q_{v2} = V_2.S_2$

⇨ $V_2 = \frac{(V1.S1)}{S2} > V_1$

⇨ **$V_2 > V_1$**

In addition, Bernoulli's principle by ignoring difference of gravity (z1 = z2) enables to write:

$$\frac{1}{2}\rho {V_2}^2 + P_2 = \frac{1}{2}\rho {V_1}^2 + P_1$$

With:

- *ρ = density = 998.2kg/m^2*
- *P = static pressure (Pa)*

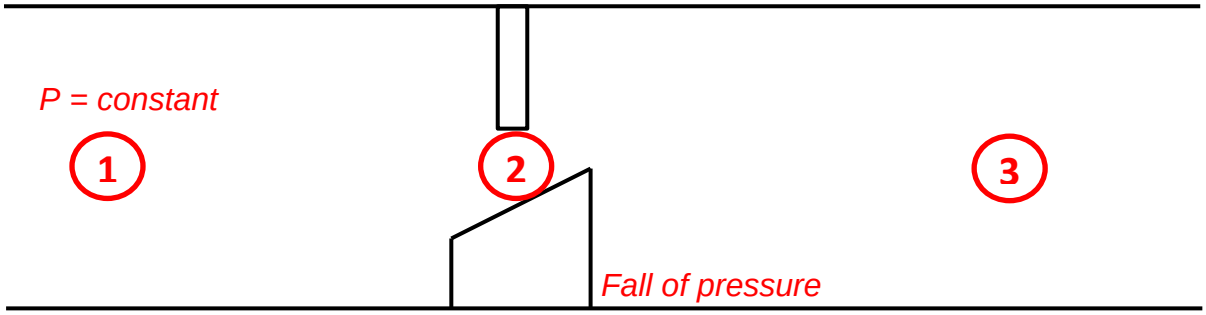
Rearranging the equation:

$$P_2 - P_1 = \frac{1}{2}\rho\left({V_2}^2 - {V_1}^2\right) > 0$$

Therefore **$P_1 > P_2$.**

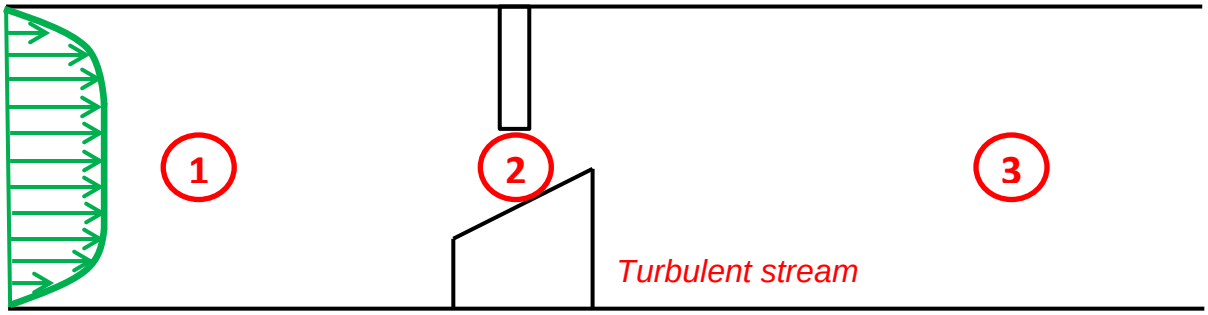
The best way to make sure that the results are coherent is to make predictions in order to imagine the behaviour of the flow. According to Bernoulli's principle an increase in the speed of the fluid occurs simultaneously with a decrease in pressure (see calculations above). The following part elaborates some ideas and drawing showing how the fluid and its properties vary through the venture meter.

1. Pressure



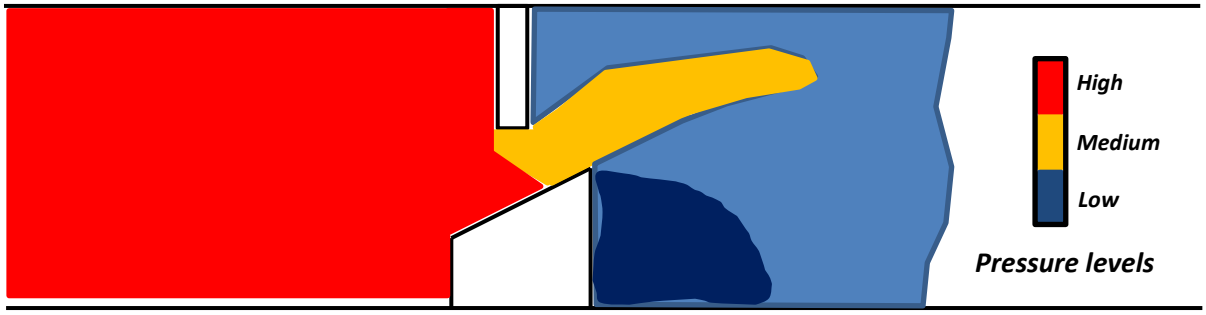
In the area 1 the pressure must be constant and the most important since the cross-sectional area at 1 is greater than at 2. In the area 3 the pressure should be a little bit important at the top right-hand corner of the variable area with an important fall of pressure in the bottom left-hand corner of the area 3 due to the bluff body.

2. Velocity



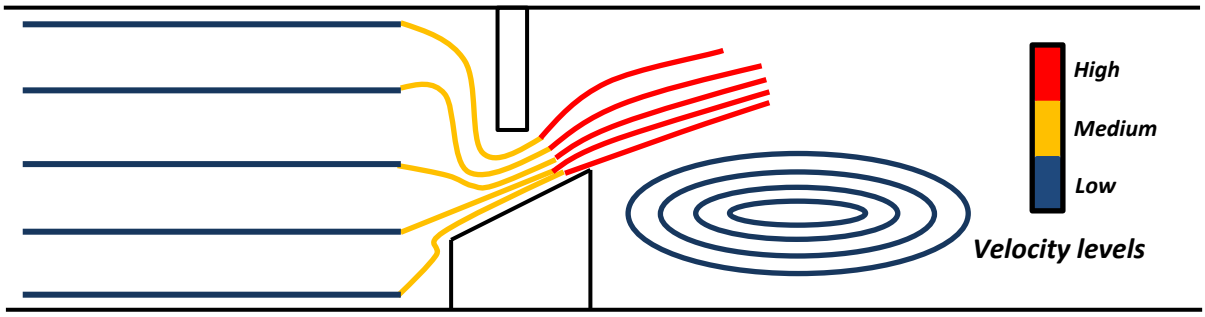
Since *Flow rate = Velocity*cross-sectional*. If the area is smaller with a constant flow rate, then the velocity increases logically as proved before. Therefore the velocity in the area 2 should be higher than the velocity in the area 1. The case which has been studied is a turbulent flow therefore a fairly flat velocity distribution should be appears like on the velocity curve.

3. Stream pressure



This picture is divided into 3 areas outlining the evolution of pressure through the venture. According to the laws of the fluid mechanics it is possible to roughly predict the behaviour of the fluid. For example, it is almost sure that the pressure will be lower at the constricted area than at the inlet and it should be the lowest just after the bluff body at the corner of the third zone *(night blue area).*

4. Stream velocity



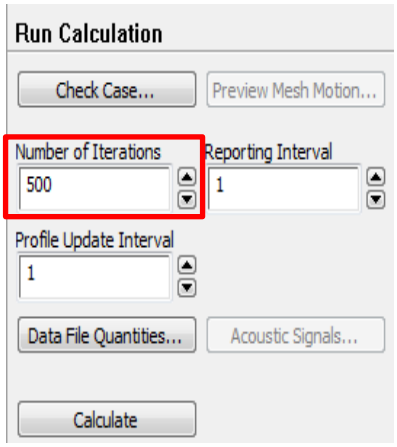
Through this drawing it is easily comprehensible that the velocity of the fluid increases when the duct shrinks. Moreover, a turbulent flow should appear after the obstacle.

These hypotheses will allow checking whether the simulation is reliable or not. The results of the simulation must be nothing else but a confirmation of the predictions accompanied by numerical results.

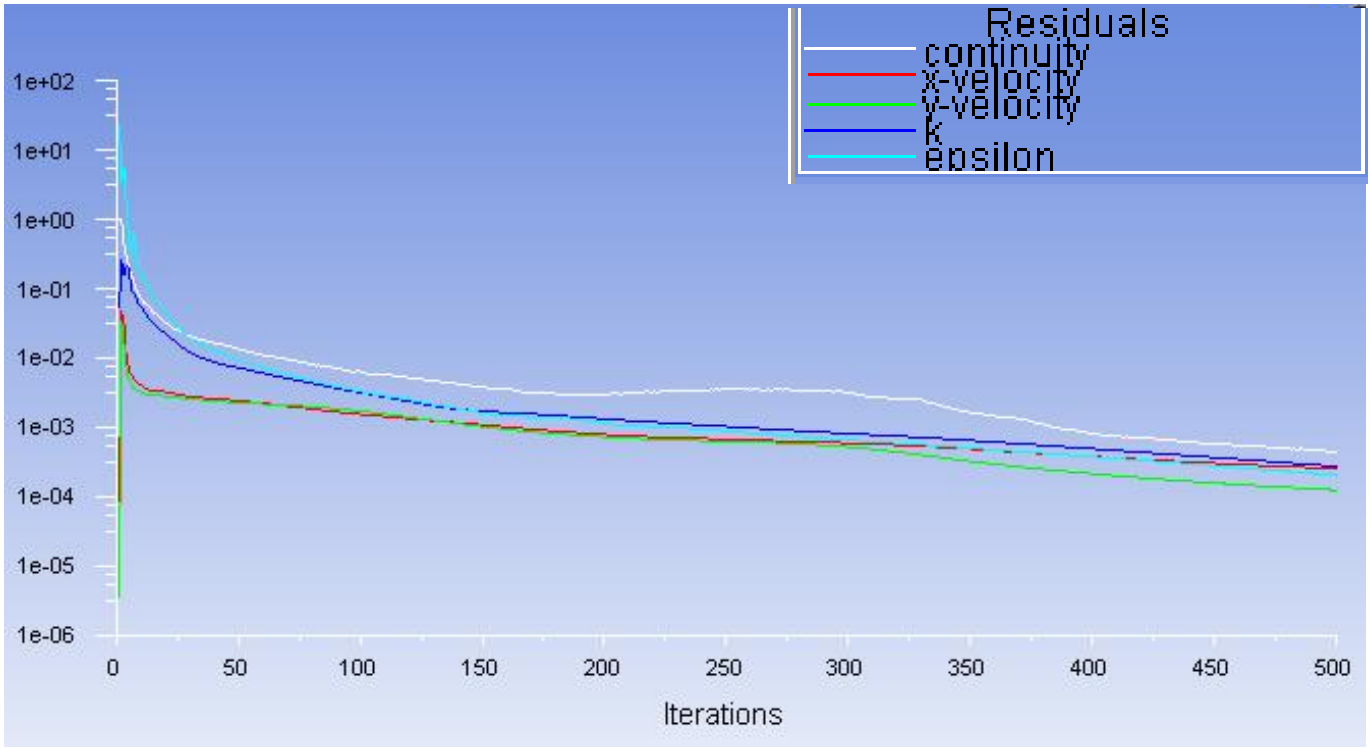
VII. Performing and monitoring of the CFD simulation

1. Fine grid

A. 4 l/s

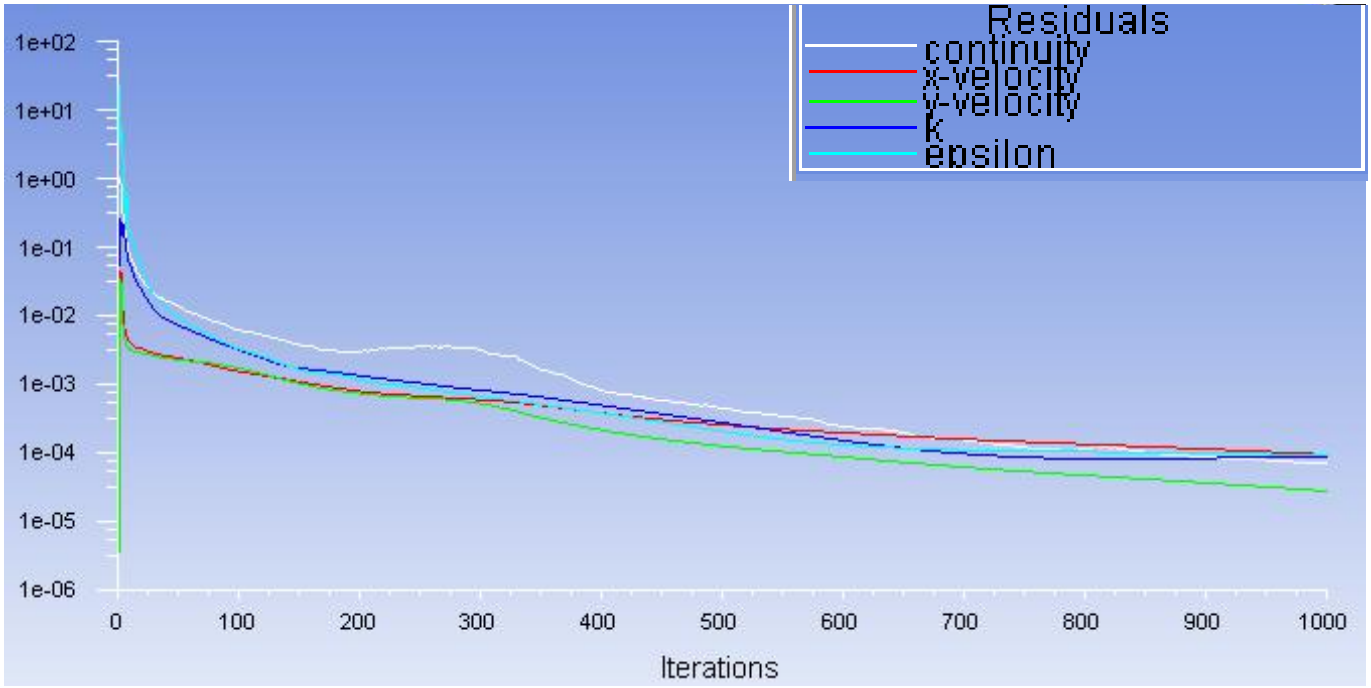


To compute the solution for the flow equals to 4l/s five hundred iterations have been run and the following graphics have been obtained.



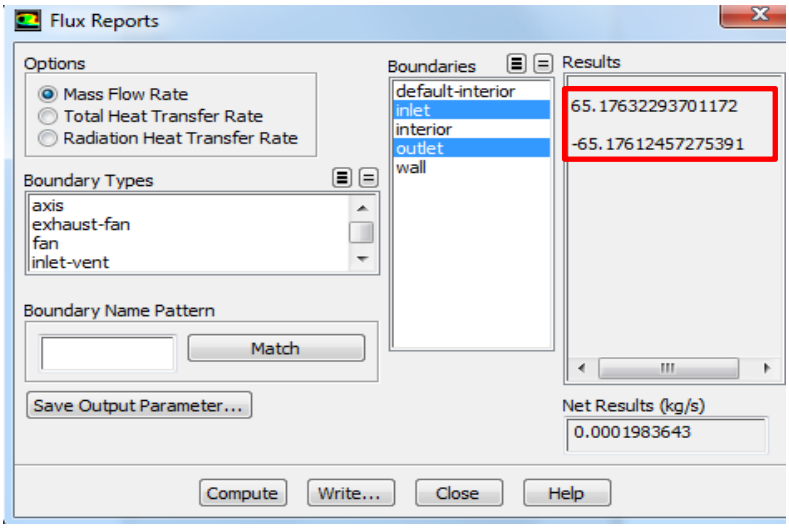
iter	continuity	x-velocity	y-velocity	k	epsilon	time/iter	
496	4.5654e-04	2.6008e-04	1.2782e-04	2.8254e-04	2.1152e-04	0:00:01	4
497	4.5804e-04	2.5922e-04	1.2728e-04	2.8085e-04	2.1022e-04	0:00:00	3
498	4.4810e-04	2.5837e-04	1.2675e-04	2.7917e-04	2.0893e-04	0:00:00	2
499	4.4829e-04	2.5753e-04	1.2626e-04	2.7750e-04	2.0767e-04	0:00:00	1
500	4.4288e-04	2.5672e-04	1.2571e-04	2.7583e-04	2.0642e-04	0:00:00	0

Reading the last five iterations the residual seems to be convergent but in order to make sure the simulation has been carried on for five hundred iterations more.



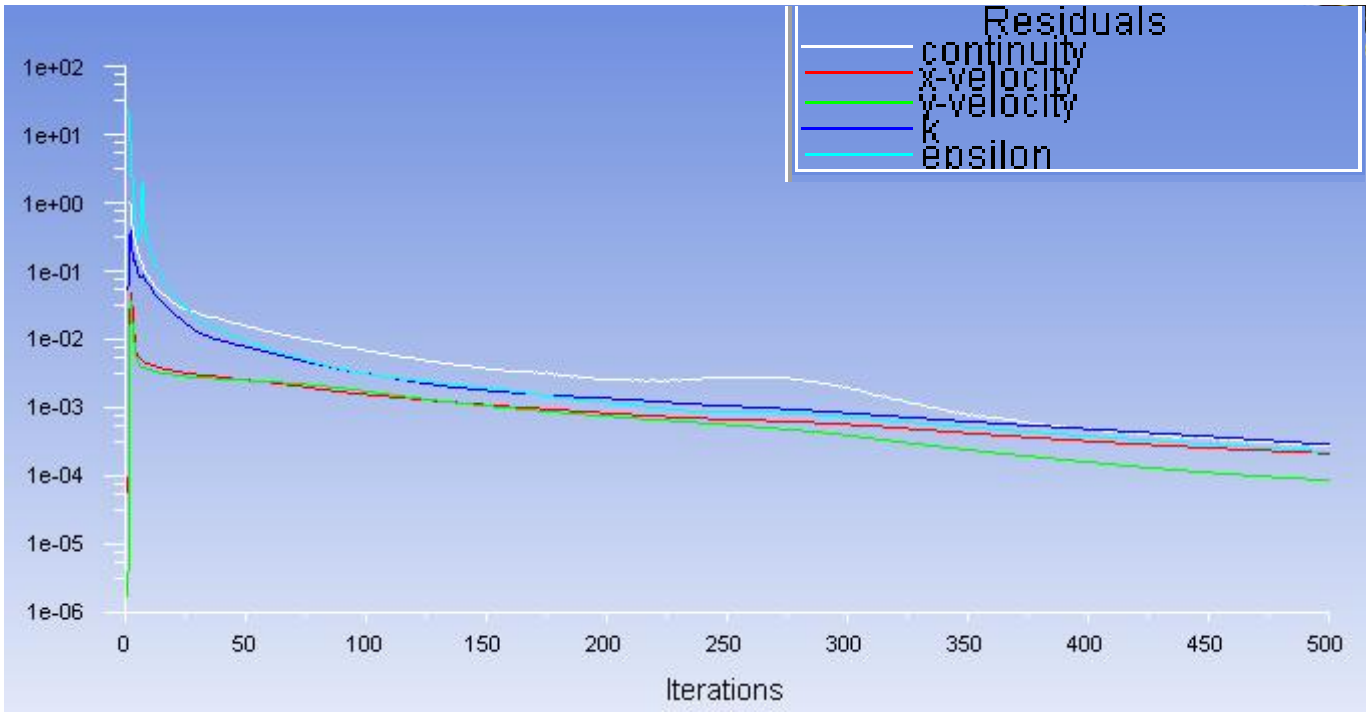
iter	continuity	x-velocity	y-velocity	k	epsilon	time/iter	
995	7.2275e-05	9.5661e-05	2.8156e-05	8.9925e-05	9.3707e-05	0:00:00	5
996	7.2226e-05	9.5470e-05	2.8064e-05	8.9989e-05	9.3636e-05	0:00:01	4
997	7.1595e-05	9.5281e-05	2.7985e-05	9.0054e-05	9.3564e-05	0:00:01	3
998	7.1263e-05	9.5095e-05	2.7900e-05	9.0120e-05	9.3490e-05	0:00:00	2
999	7.1536e-05	9.4903e-05	2.7827e-05	9.0186e-05	9.3417e-05	0:00:00	1
1000	7.1262e-05	9.4711e-05	2.7745e-05	9.0250e-05	9.3346e-05	0:00:00	0

Even if a residual lower than 1*e*-06 has not been reached the solution is convergent. Indeed, a residual equals to 1*e*-05 is absolutely acceptable to prove the convergence of the solution.



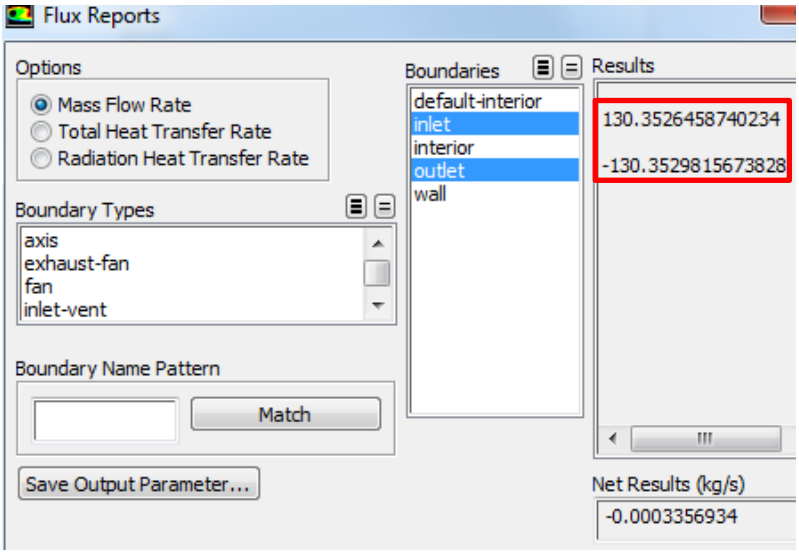
Thanks to the *Flux Reports* tool the mass flux conservation has been checked selecting the *inlet* and *outlet* boundaries. Results show approximately 65.176 kg/s for the incoming flux and 65.176 kg/s for the mass flux leaving the system.

B. 8 l/s



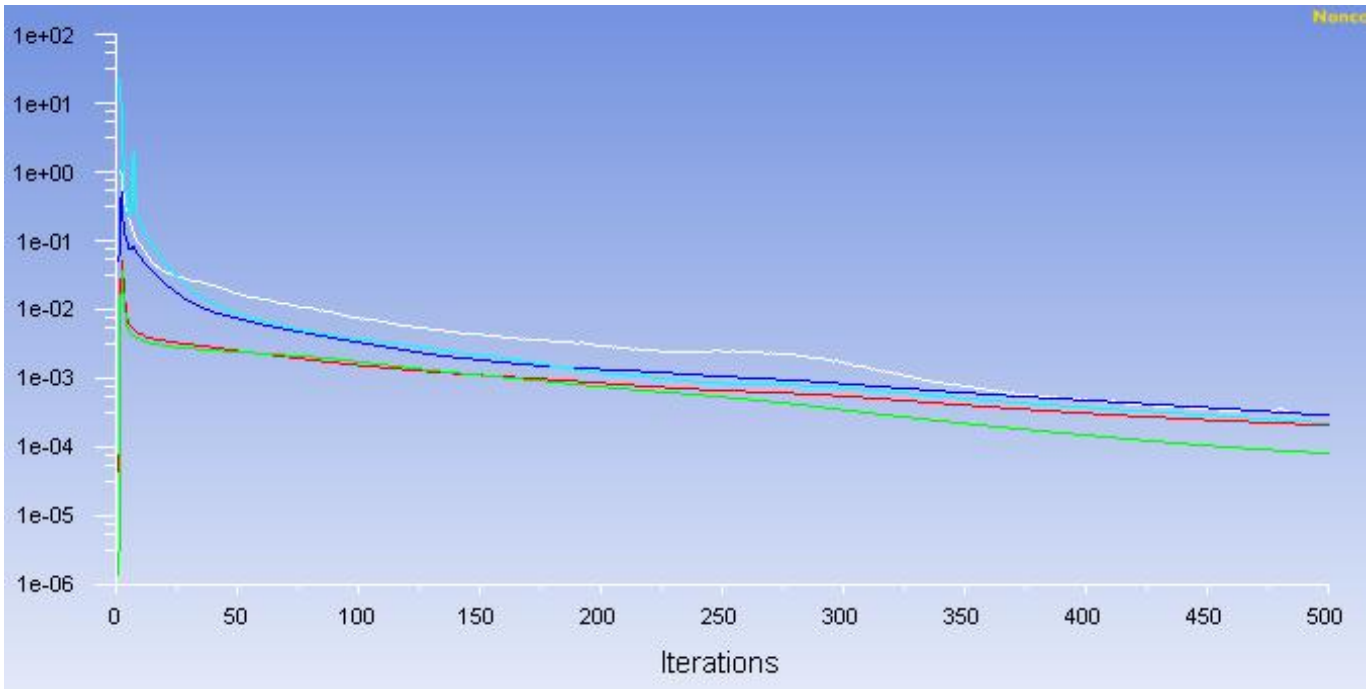
iter	continuity	x-velocity	y-velocity	k	epsilon	time/iter	
496	2.7941e-04	2.1819e-04	8.8190e-05	2.9472e-04	2.2985e-04	0:00:01	4
497	2.8099e-04	2.1741e-04	8.7891e-05	2.9305e-04	2.2862e-04	0:00:01	3
498	2.7733e-04	2.1679e-04	8.7491e-05	2.9140e-04	2.2742e-04	0:00:00	2
499	2.7289e-04	2.1614e-04	8.6997e-05	2.8975e-04	2.2622e-04	0:00:00	1
500	2.7835e-04	2.1536e-04	8.6672e-05	2.8812e-04	2.2503e-04	0:00:00	0

As in the previous graphic we have done five hundred iterations and the residual is enough small (≈1*e*-04) to assure that the solution converges.



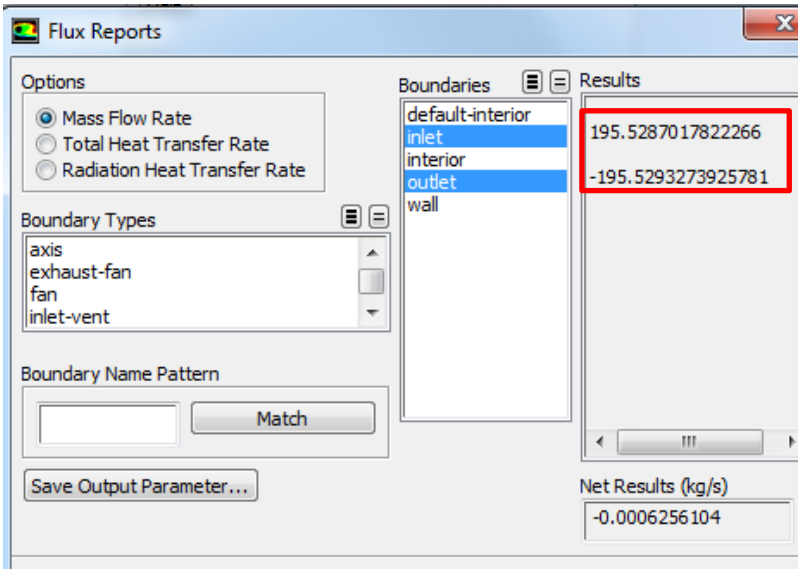
The incoming and outcoming mass flux are the same and equal to 130.35 kg/s. Therefore the mass flow rate through the venture stays constant. It is essential to check the conservation of mass, because it must be neither created nor destroyed.

C. 12 l/s



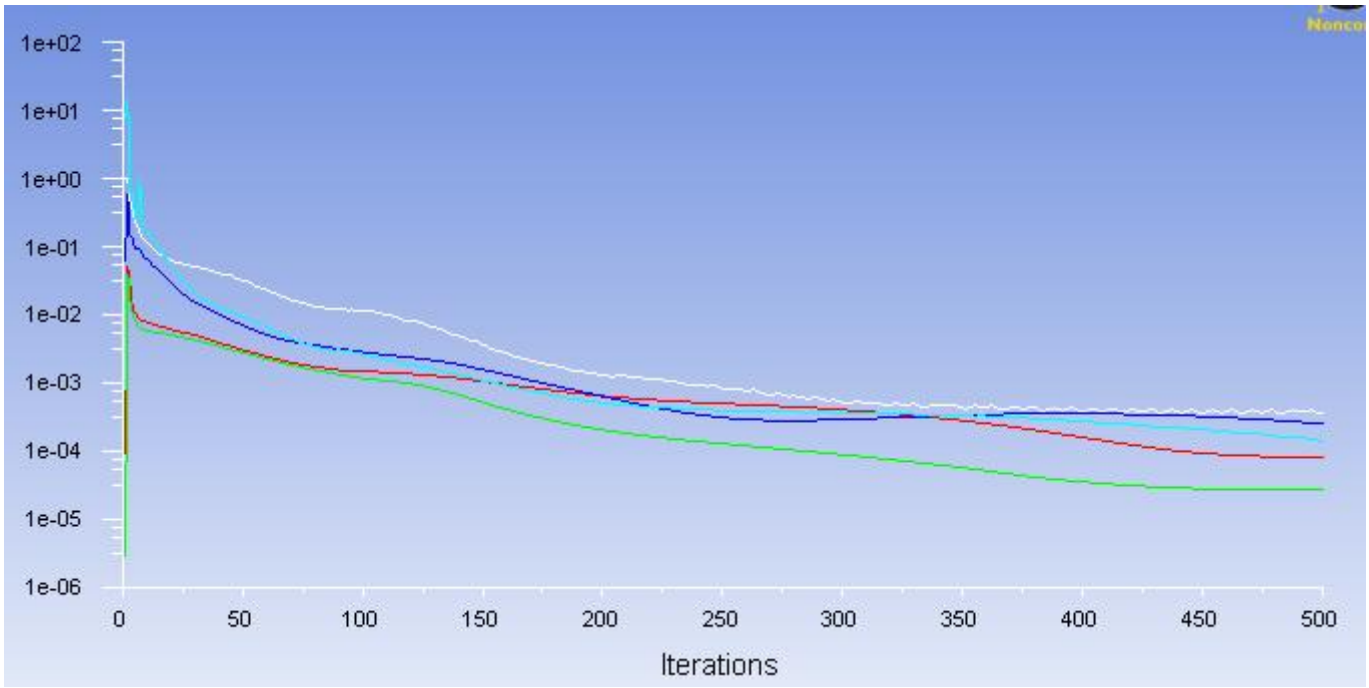
iter	continuity	x-velocity	y-velocity	k	epsilon	time/iter	
496	2.9533e-04	2.1183e-04	8.2684e-05	2.9539e-04	2.3045e-04	0:00:01	4
497	2.9178e-04	2.1123e-04	8.2312e-05	2.9379e-04	2.2924e-04	0:00:00	3
498	3.0035e-04	2.1061e-04	8.1878e-05	2.9220e-04	2.2804e-04	0:00:01	2
499	3.0034e-04	2.0993e-04	8.1632e-05	2.9062e-04	2.2684e-04	0:00:00	1
500	2.9214e-04	2.0933e-04	8.1230e-05	2.8905e-04	2.2566e-04	0:00:00	0

For the flow rate equals to 12l/s, which corresponds to the last one using the fine grid, the accuracy obtained is roughly the same as the others, thus once again the changes in the solution from one iteration to the next are negligeable, hence the convergence has been reached. Finally, the following screenshot enables once more to check the mass flux conservation which is equal to 195.5 kg/s.



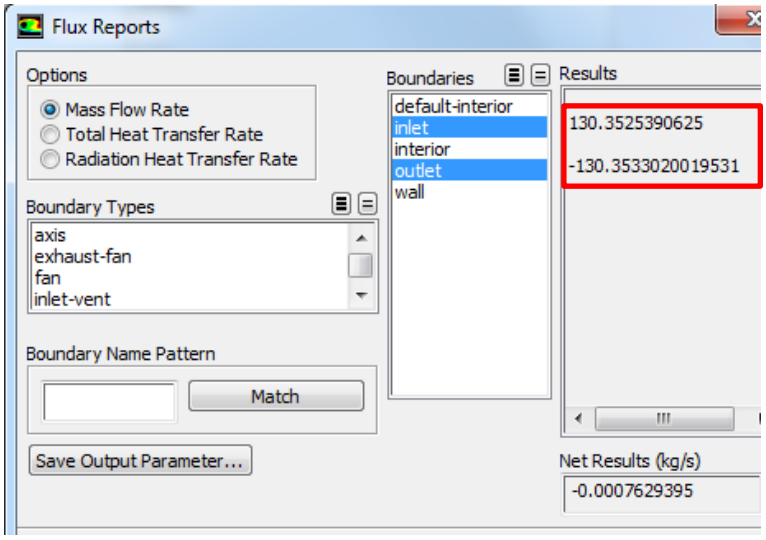
2. Coarse grid

A. 8L/s



iter	continuity	x-velocity	y-velocity	k	epsilon	time/iter	
496	4.0038e-04	8.2524e-05	2.6821e-05	2.6004e-04	1.4752e-04	0:00:00	4
497	3.8476e-04	8.2448e-05	2.6784e-05	2.5847e-04	1.4705e-04	0:00:00	3
498	3.6539e-04	8.2419e-05	2.6788e-05	2.5656e-04	1.4498e-04	0:00:00	2
499	3.5928e-04	8.2399e-05	2.6804e-05	2.5453e-04	1.4316e-04	0:00:00	1
500	3.5863e-04	8.2433e-05	2.6817e-05	2.5290e-04	1.4292e-04	0:00:00	0

This graphics correspond to the computing process to solve the 8l/s flow rate case. As for the cases using the fine grid, the solution converges slowly but enough to demonstrate it.

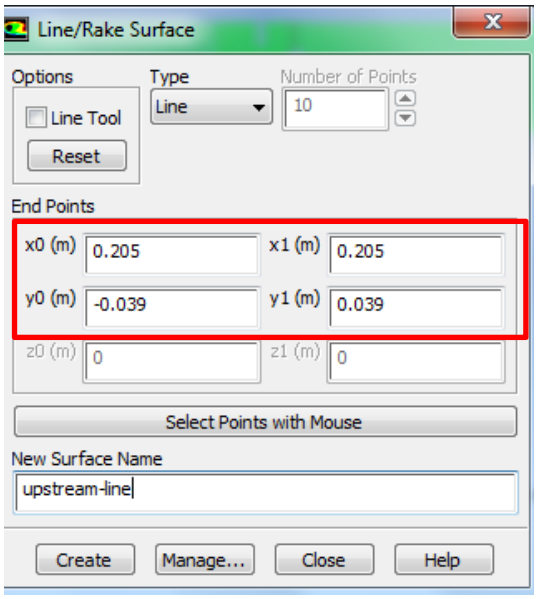


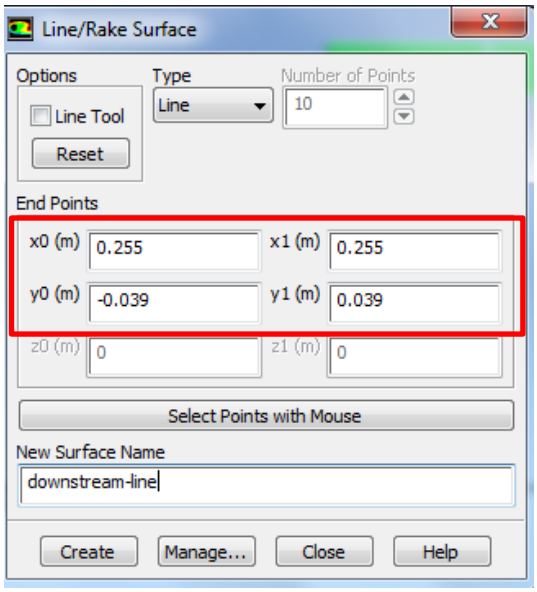
The result of the flux report notifies that there is mass flux conservation from the inlet to the outlet with 130kg/s.

Now that the calculations have been checked the post-processing part can be carried out.

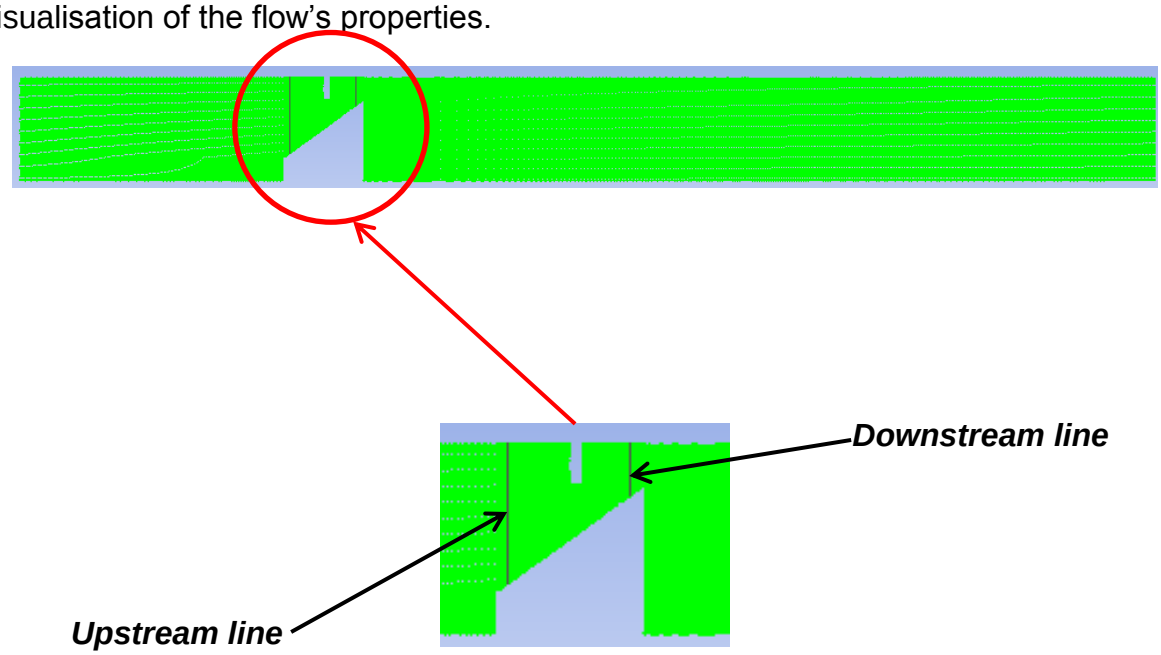
VIII. Examination and processing of the CFD results

Velocity and pressure had to be plotted at the inlet and outlet of the constricted area. Consequently, two lines have been added to get the necessary data. As provided in the subject of the assignment, the first one is located at 205mm from the venture's inlet and the second one at 255 from its outlet (+/- 25 mm from the middle of the bluff body).





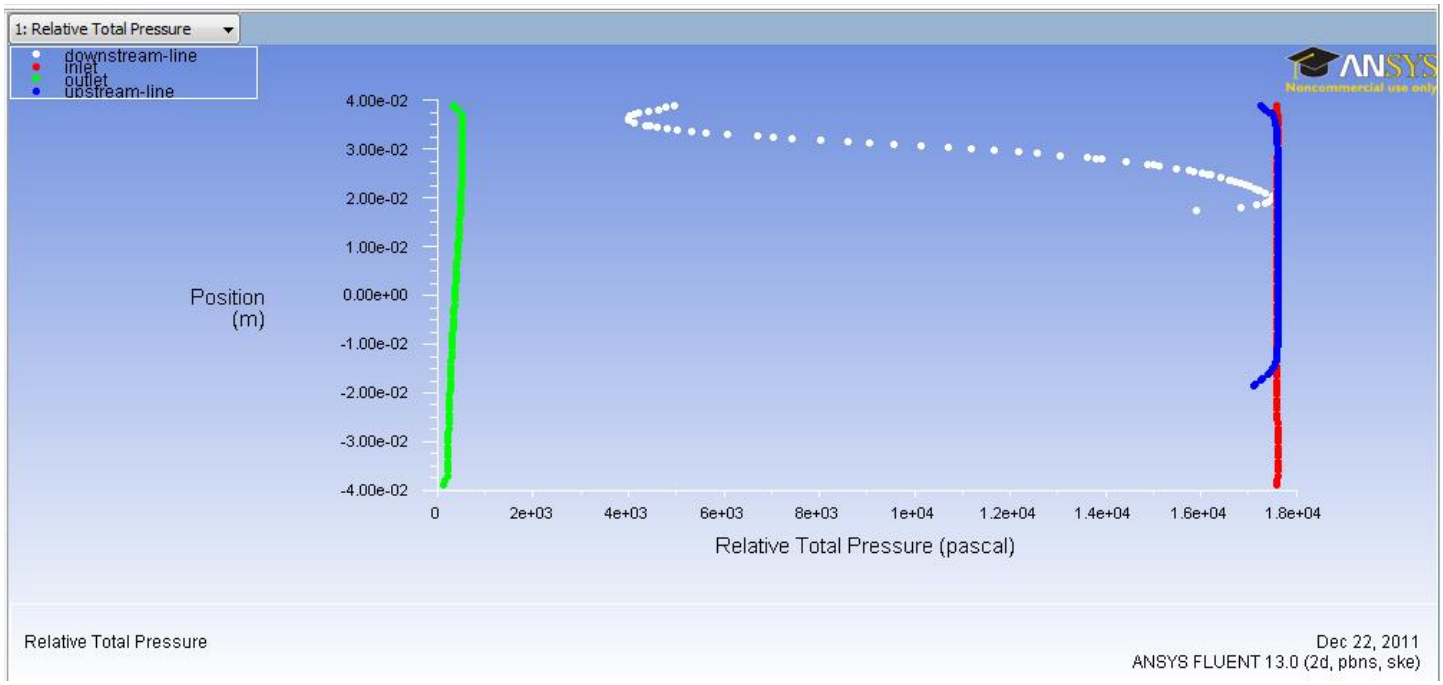
The following pictures show the fine grids including both lines allowing the visualisation of the flow's properties.



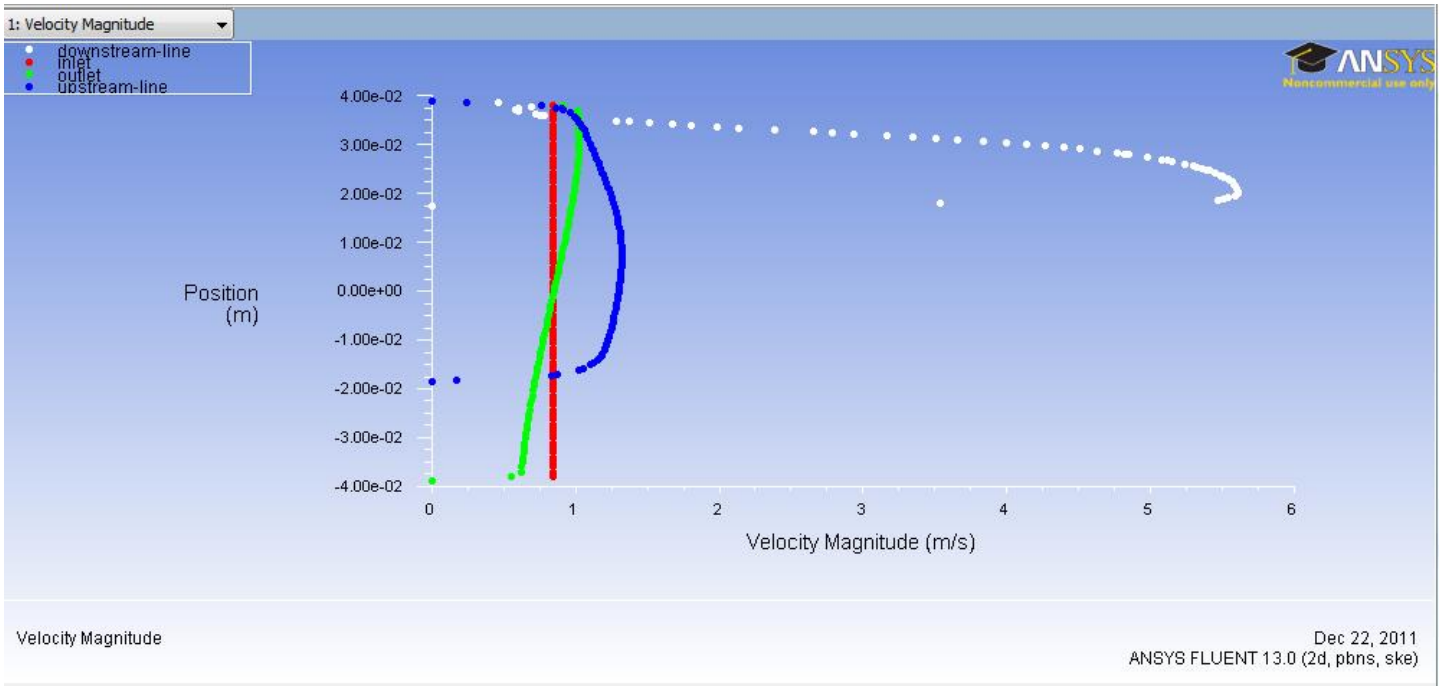
1. Fine grid

A. 4L/s

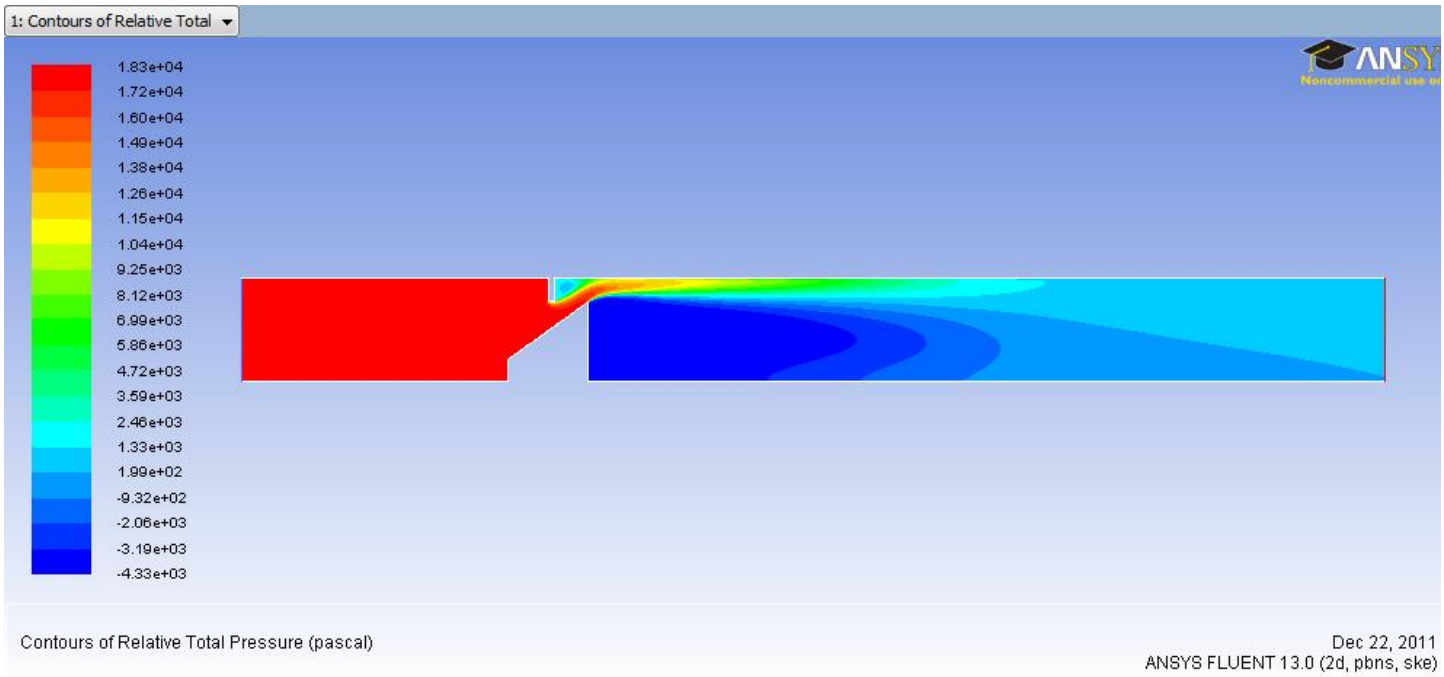
a. Pressure evolution

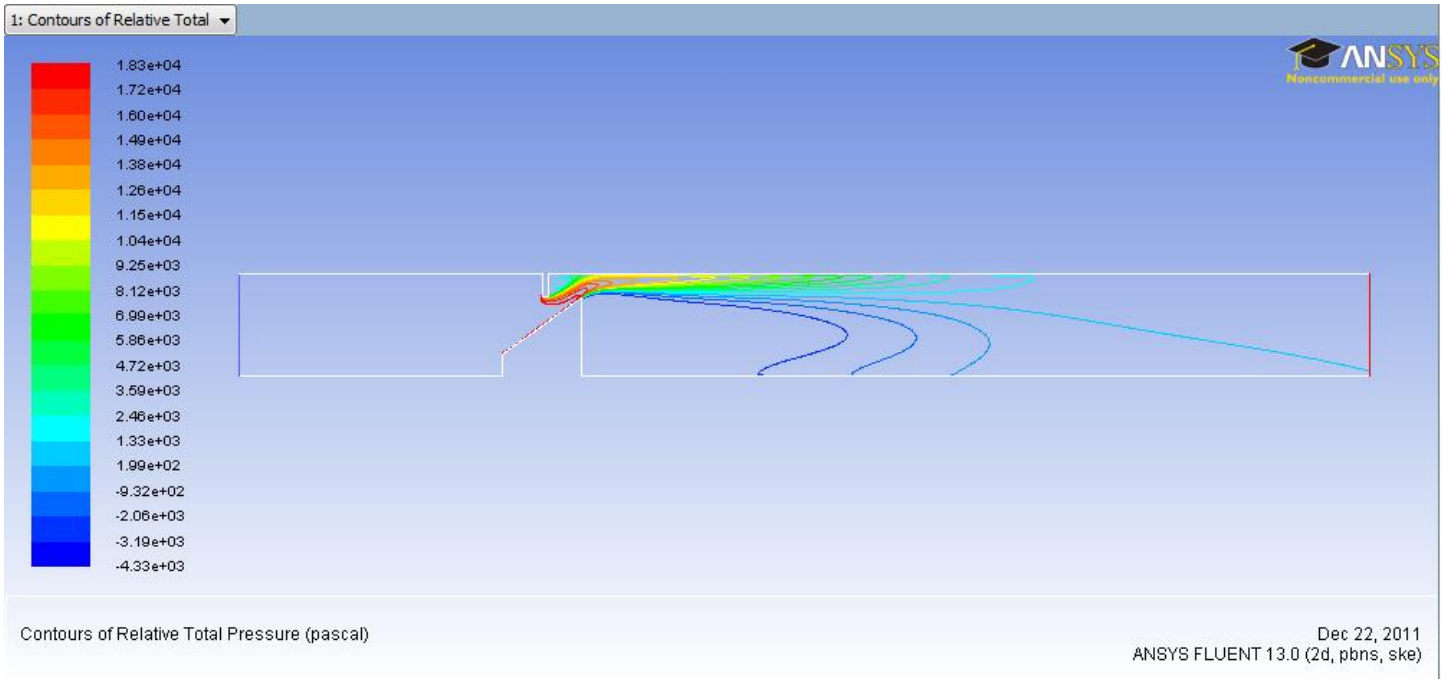


b. Velocity evolution

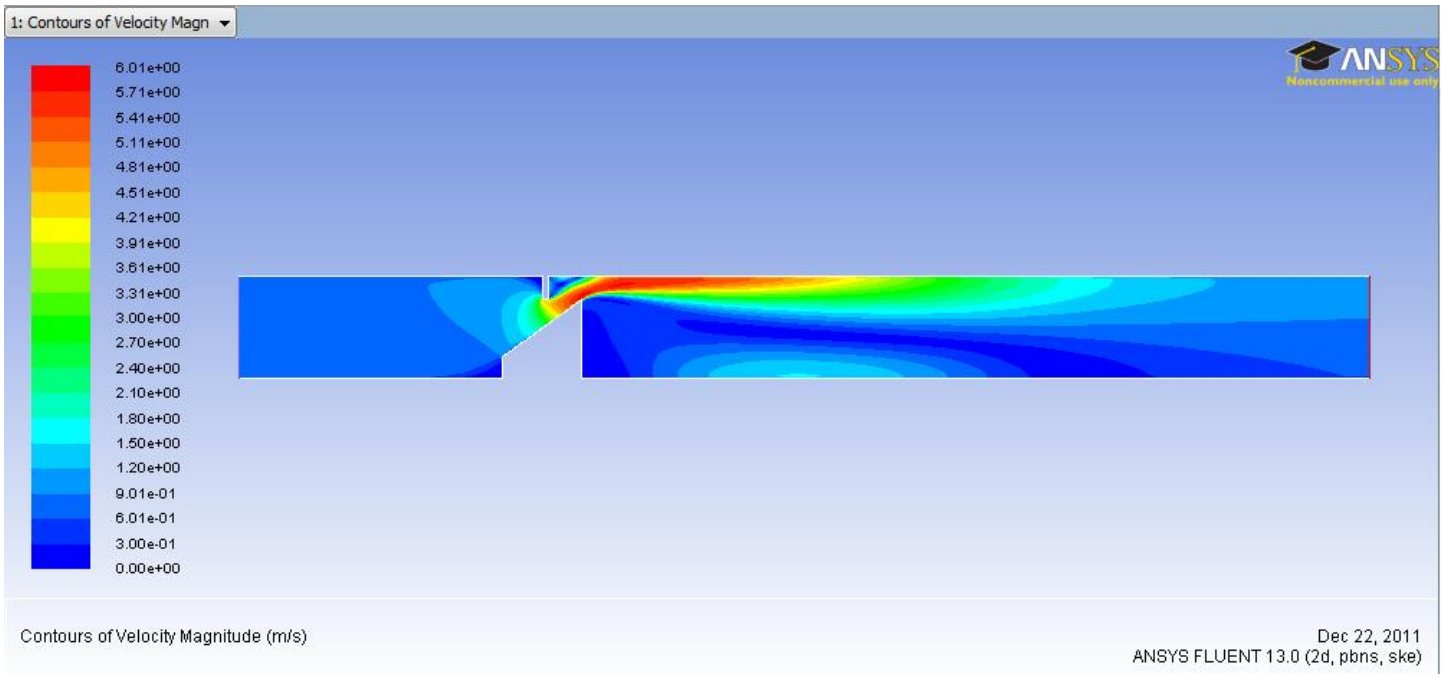


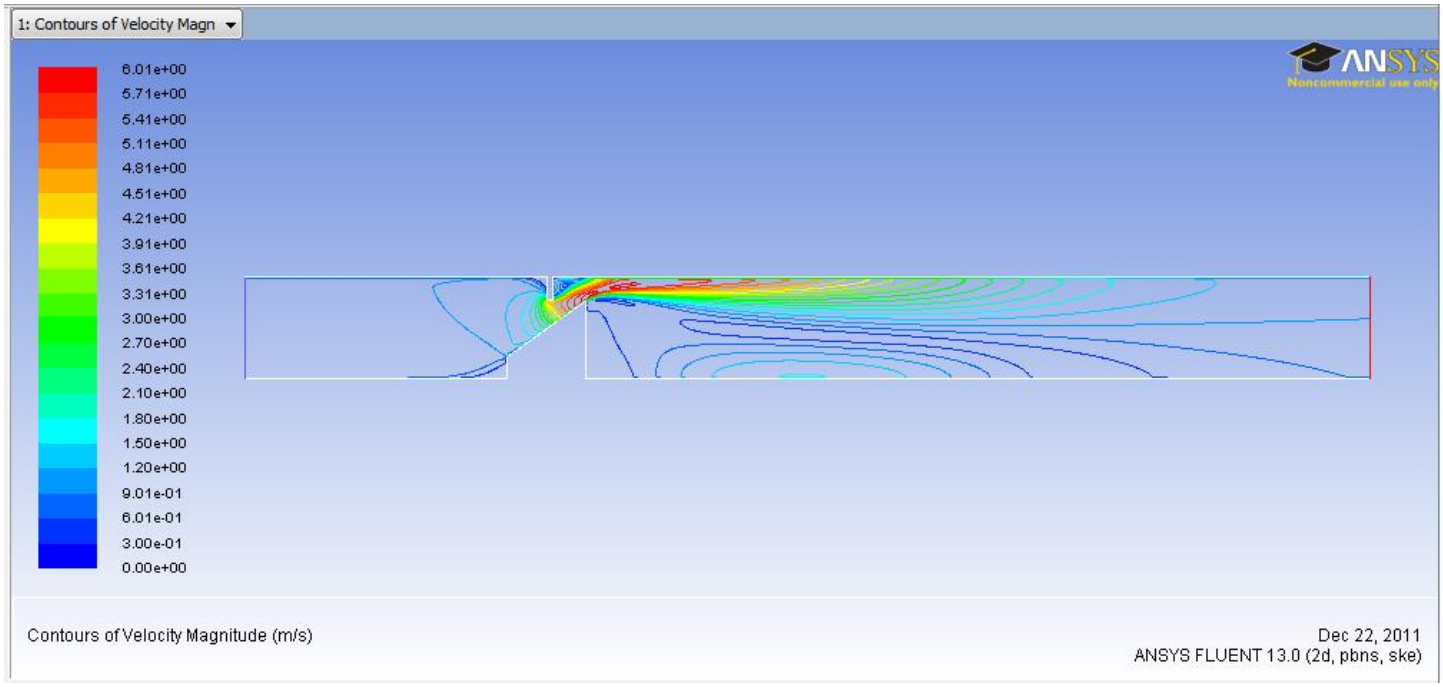
c. Relative total Pressure



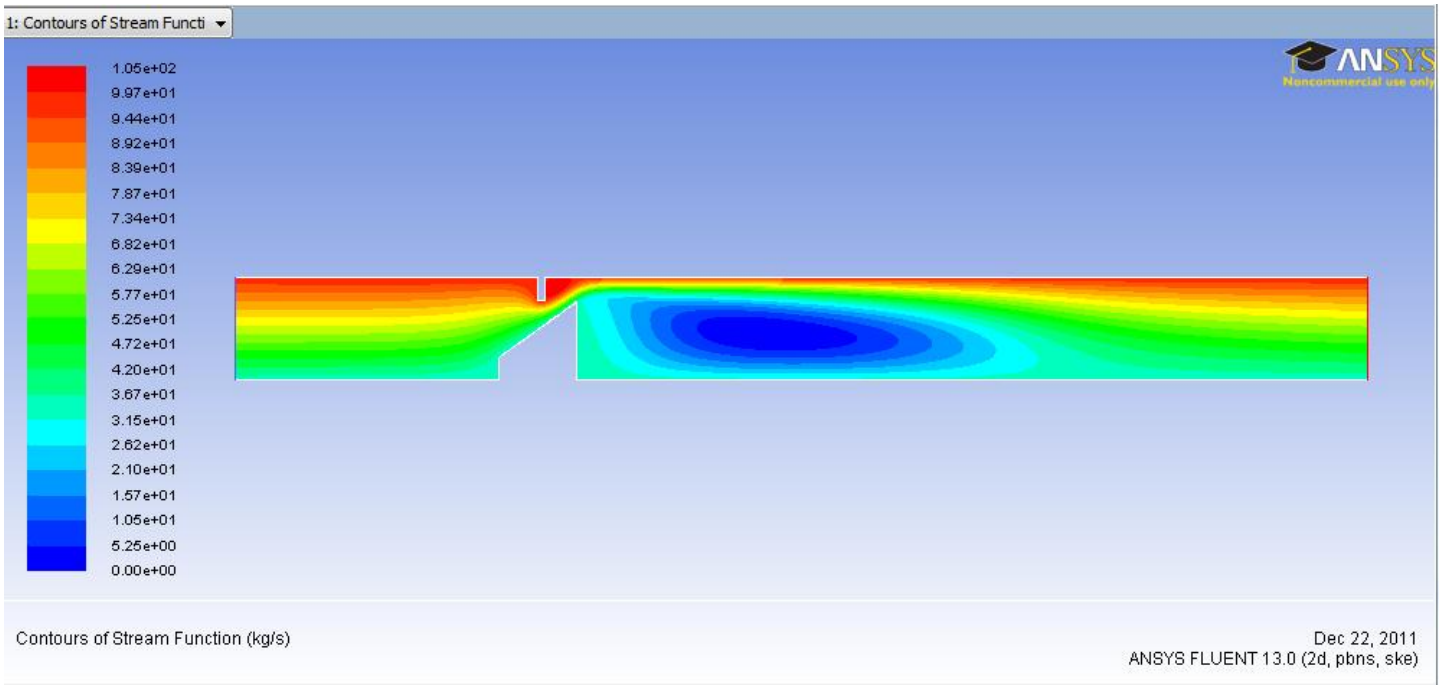


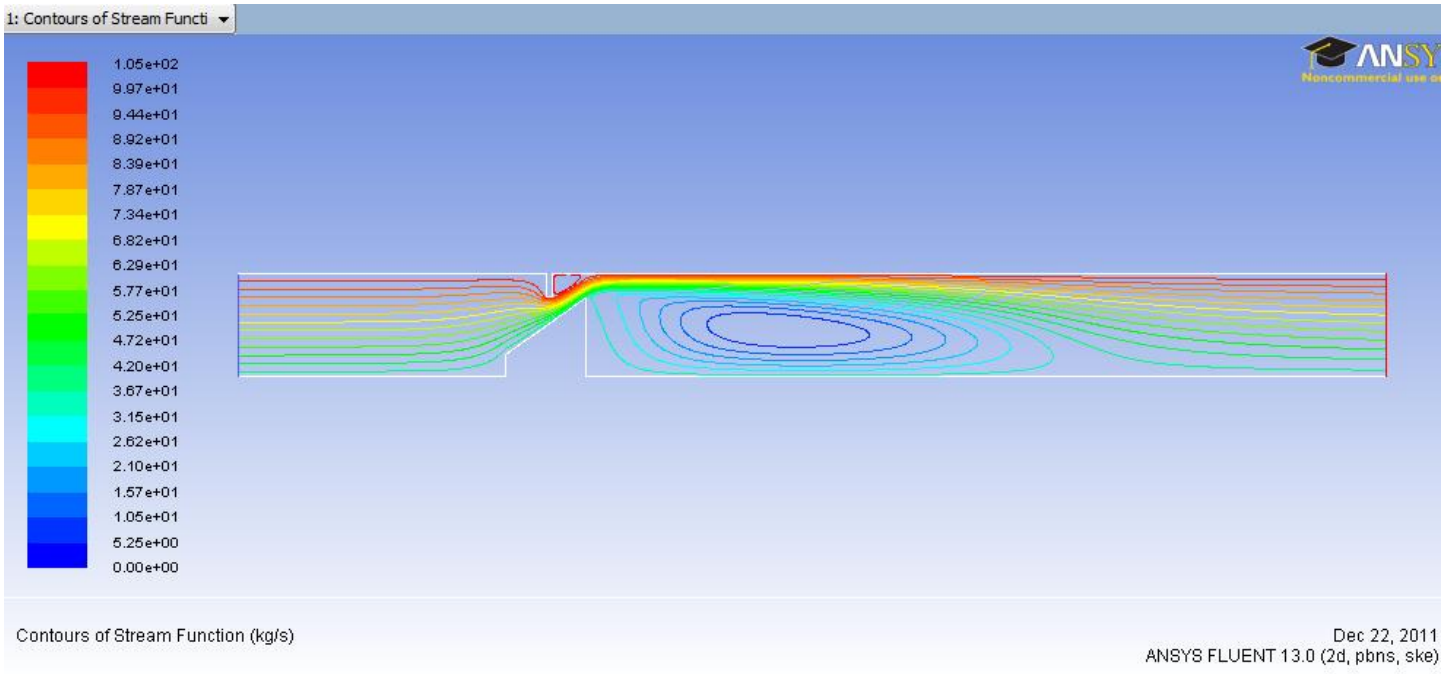
d. Velocity magnitude





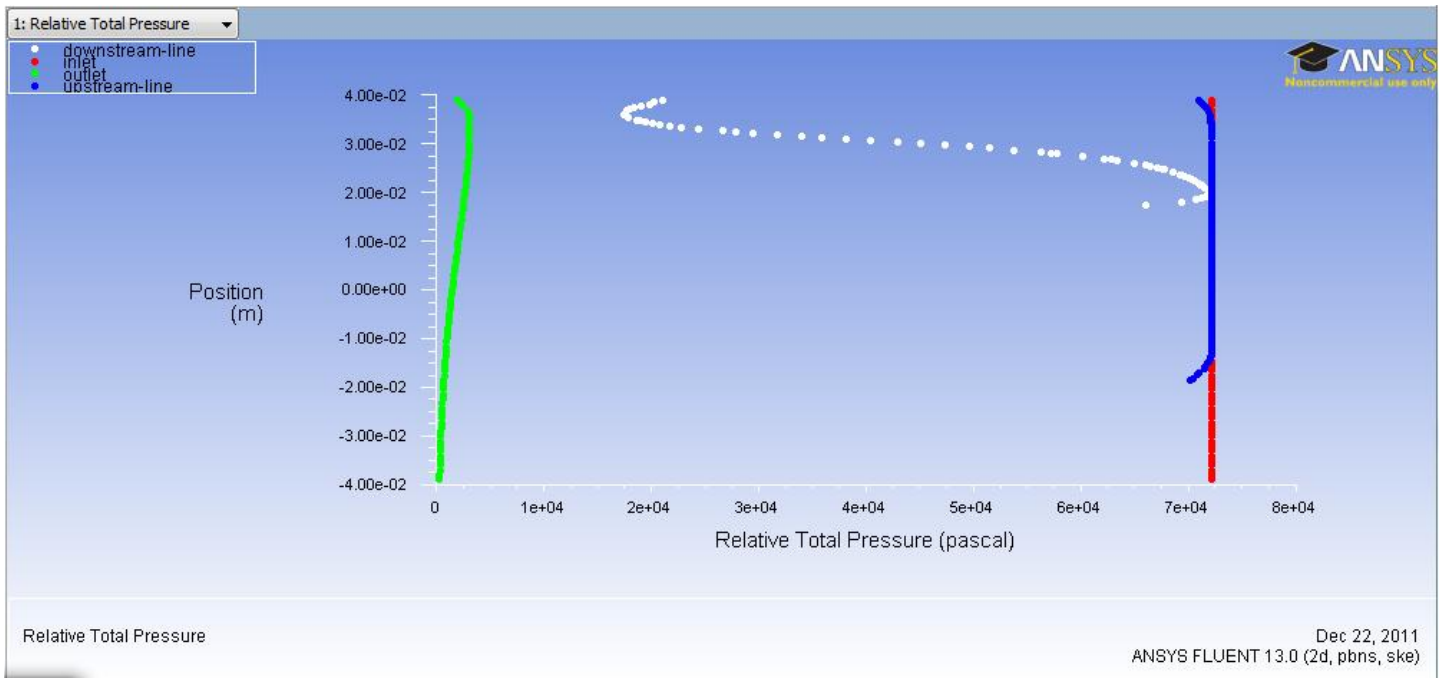
e. Velocity stream function



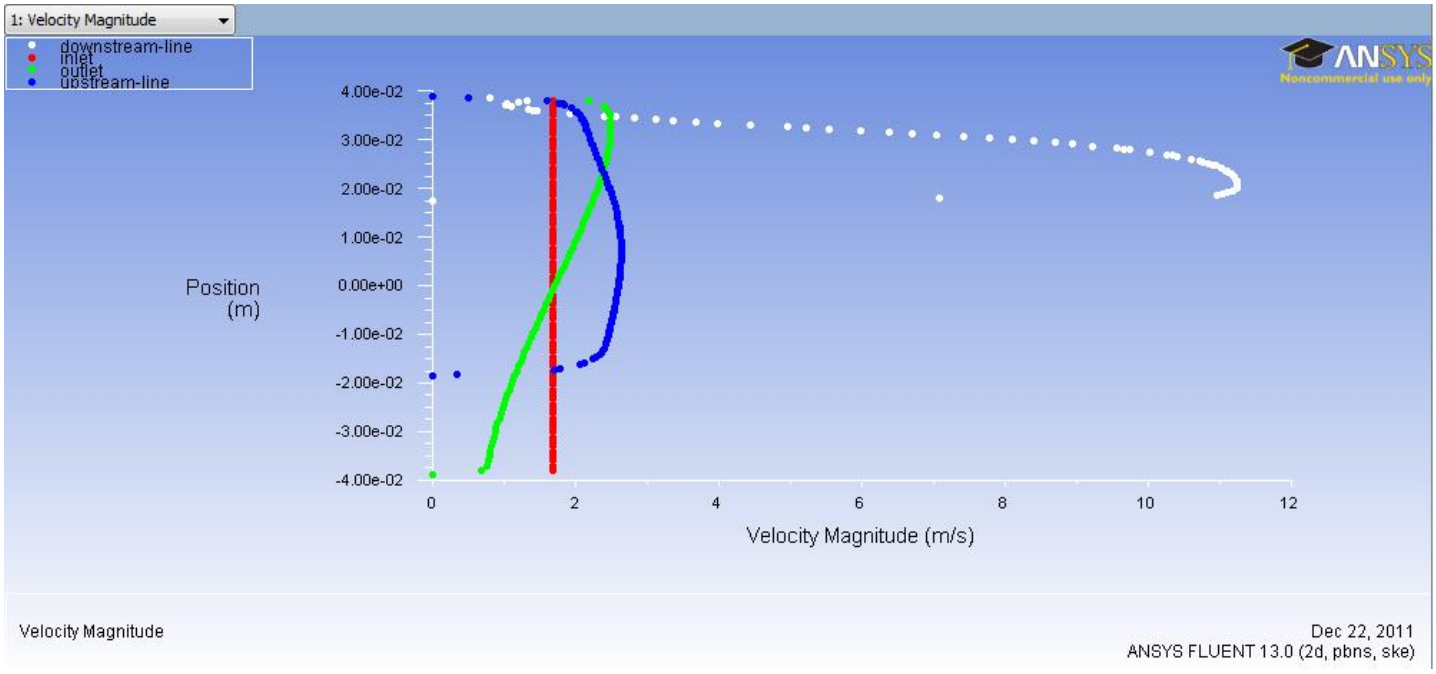


B. 8L/s

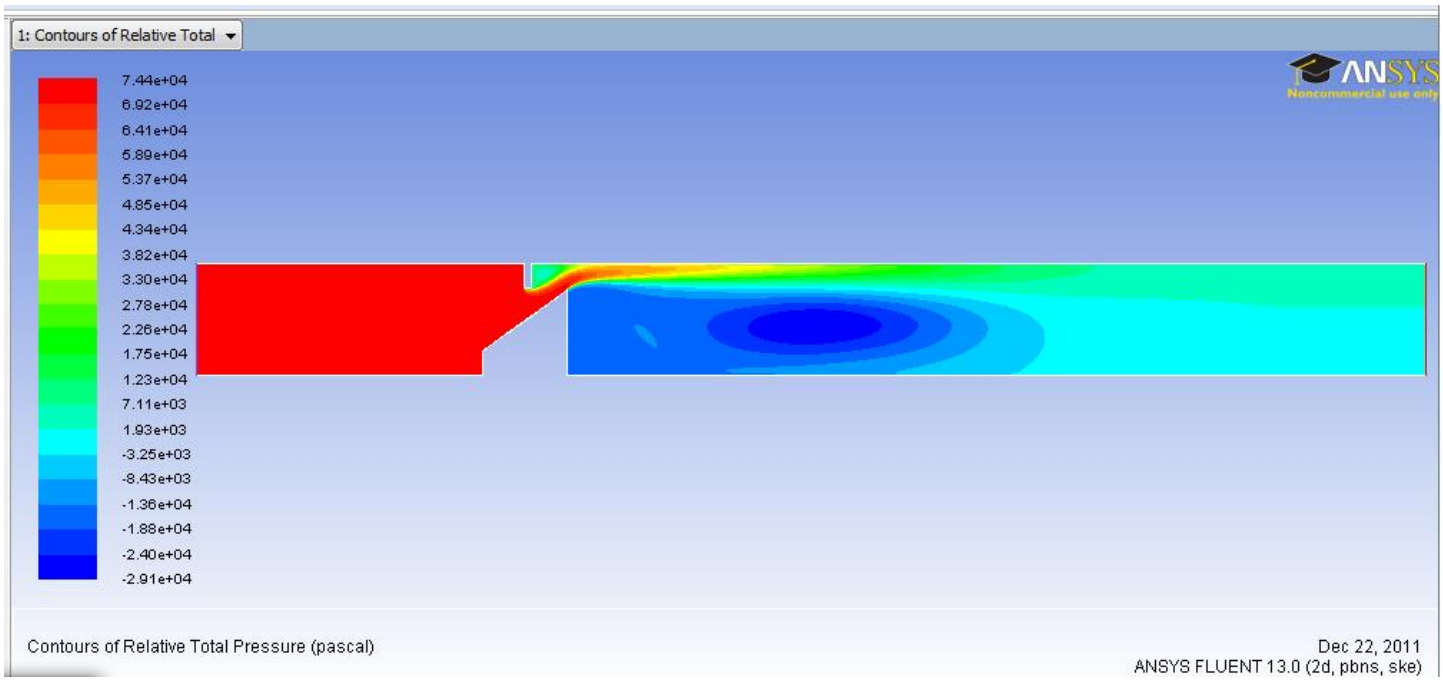
a. Pressure evolution

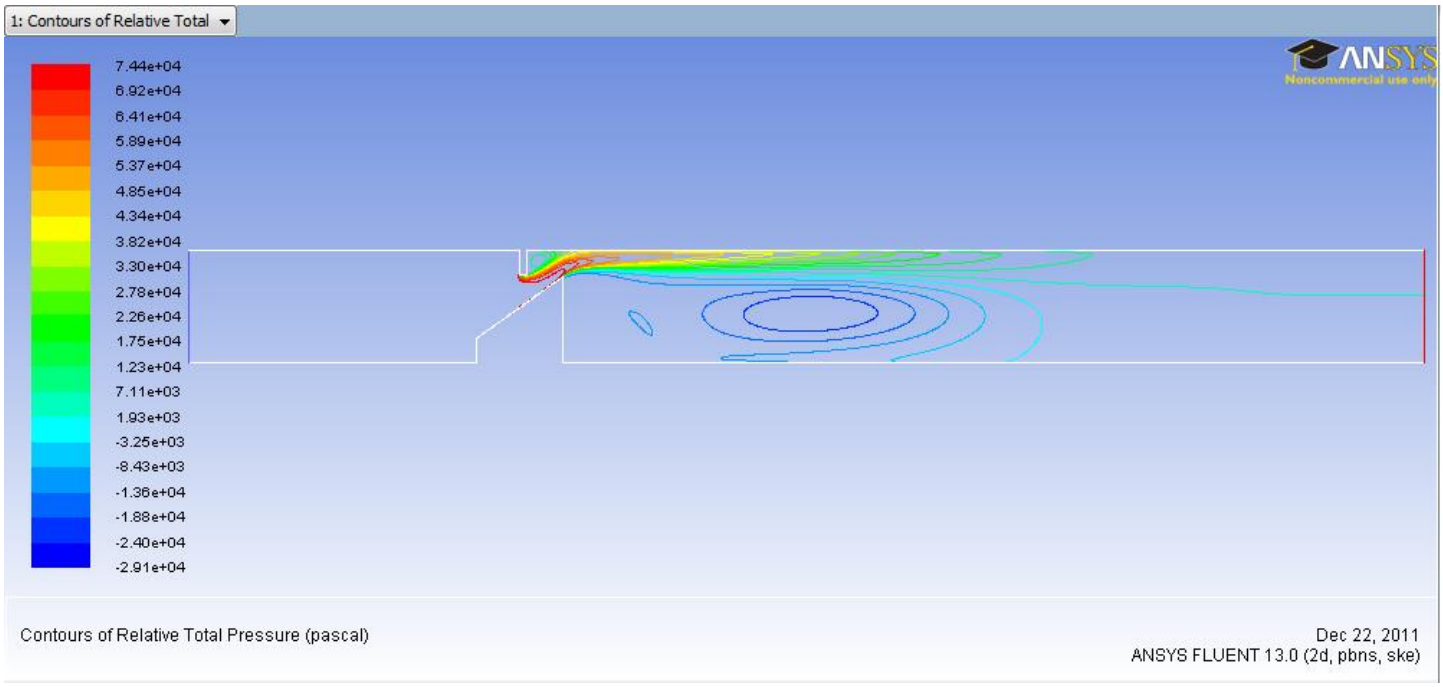


b. Velocity evolution

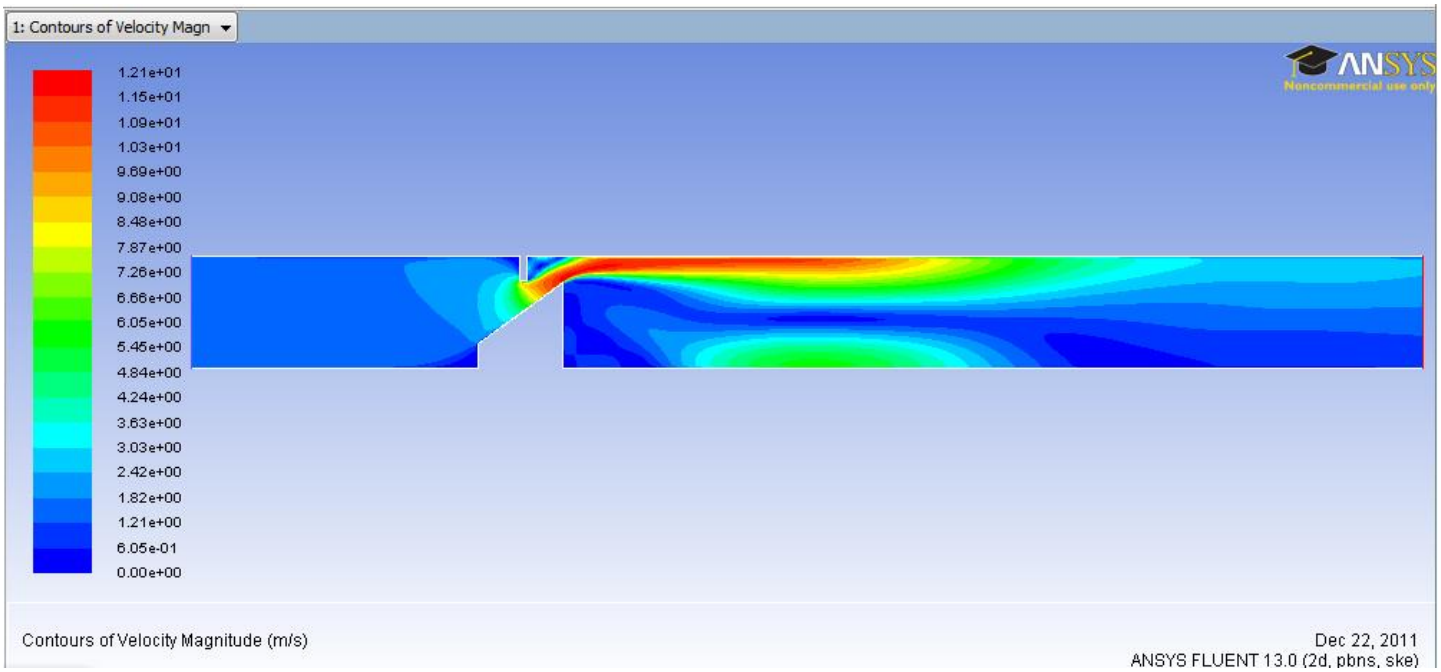


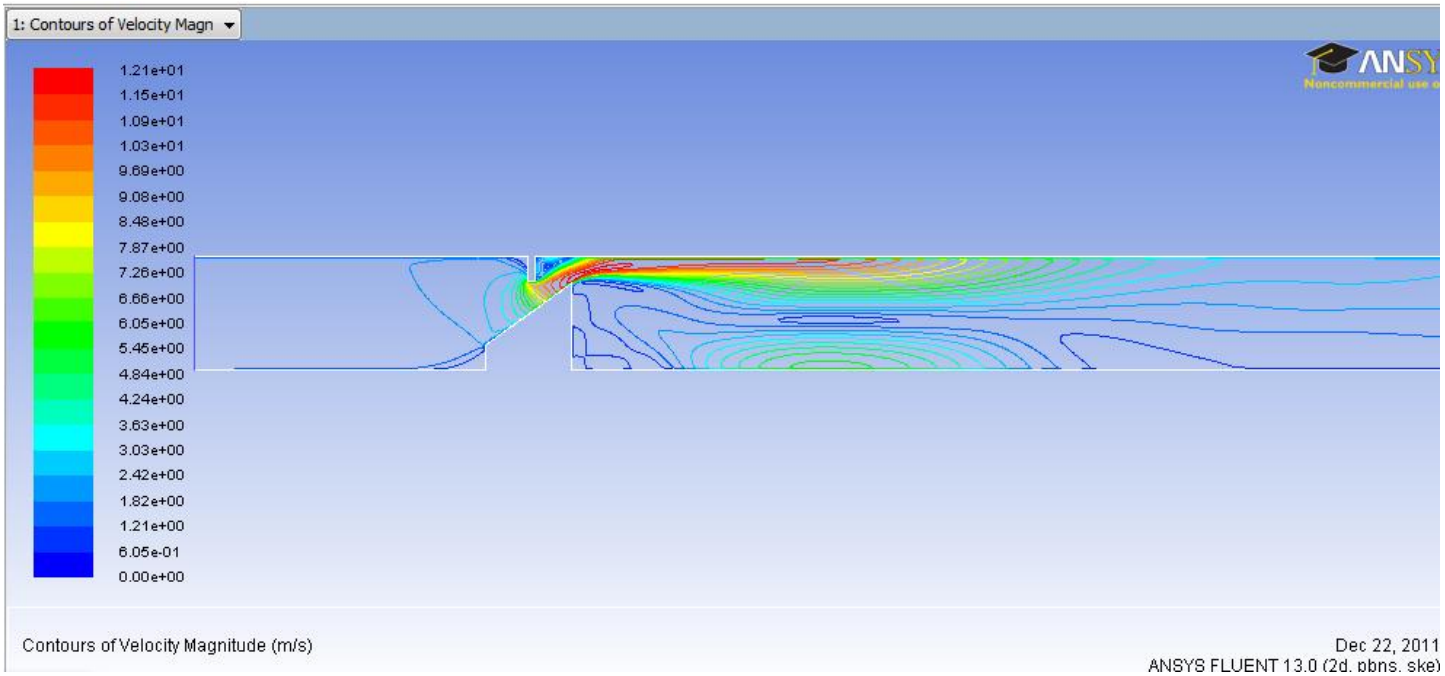
c. Relative total Pressure



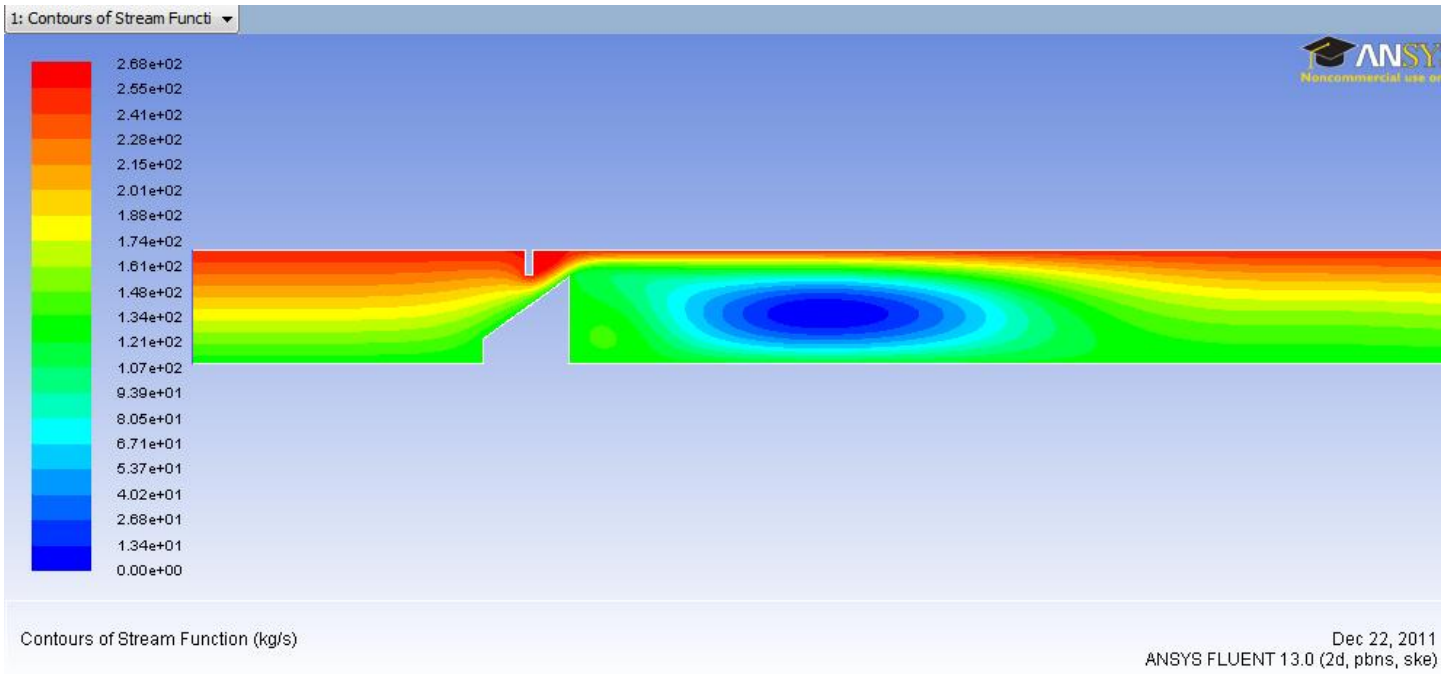


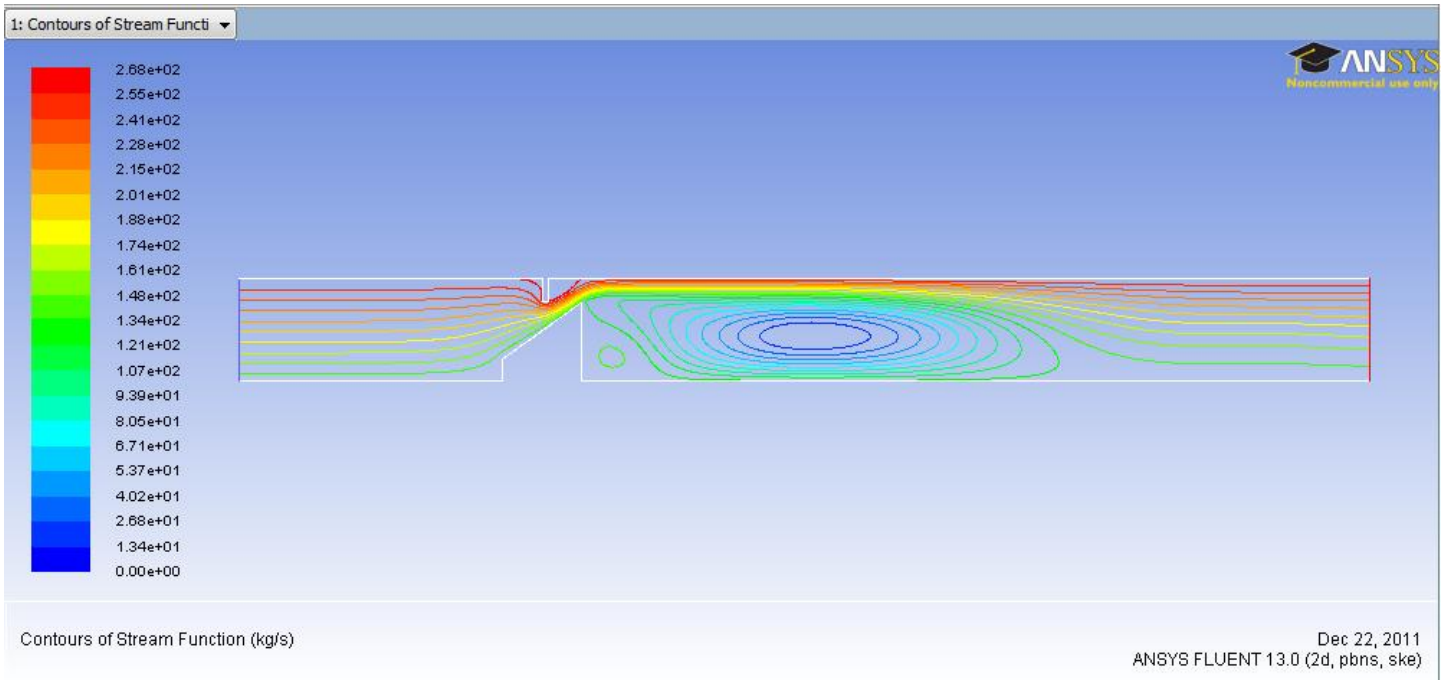
d. Velocity magnitude





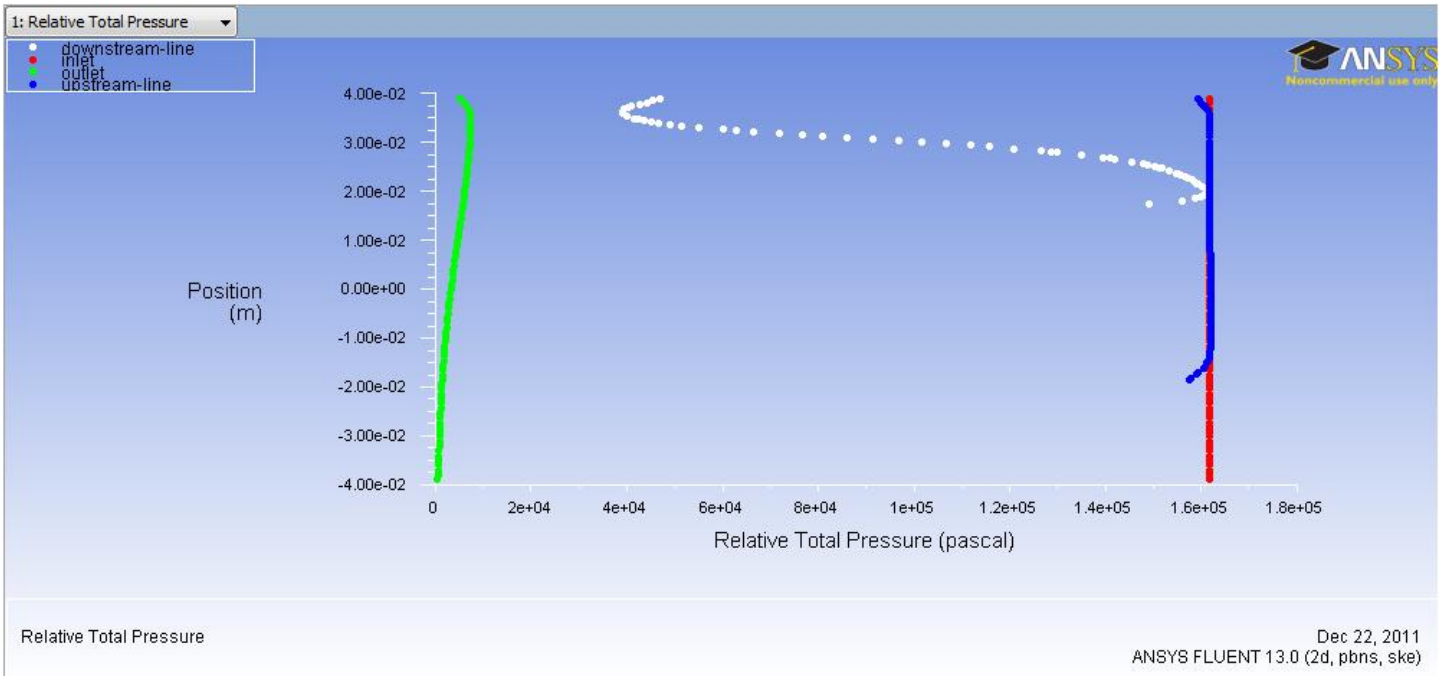
e. Velocity stream function



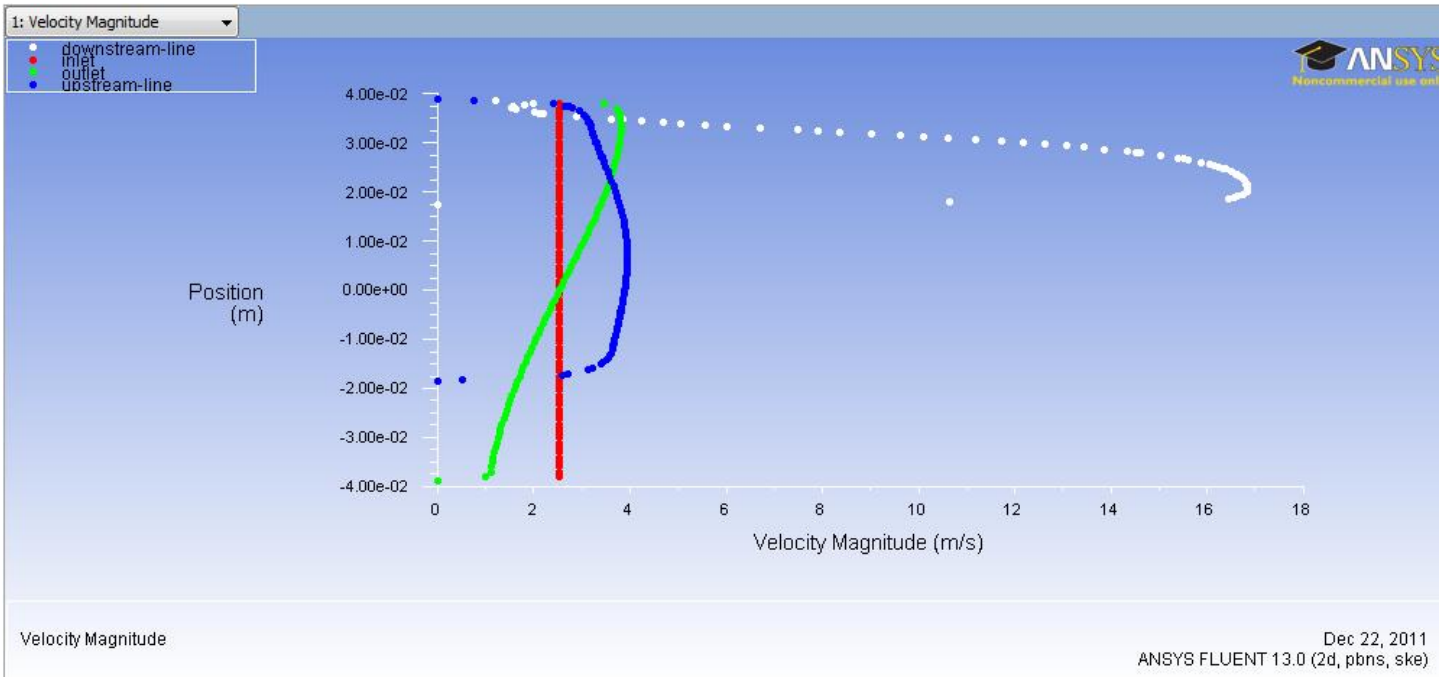


C. 12L/s

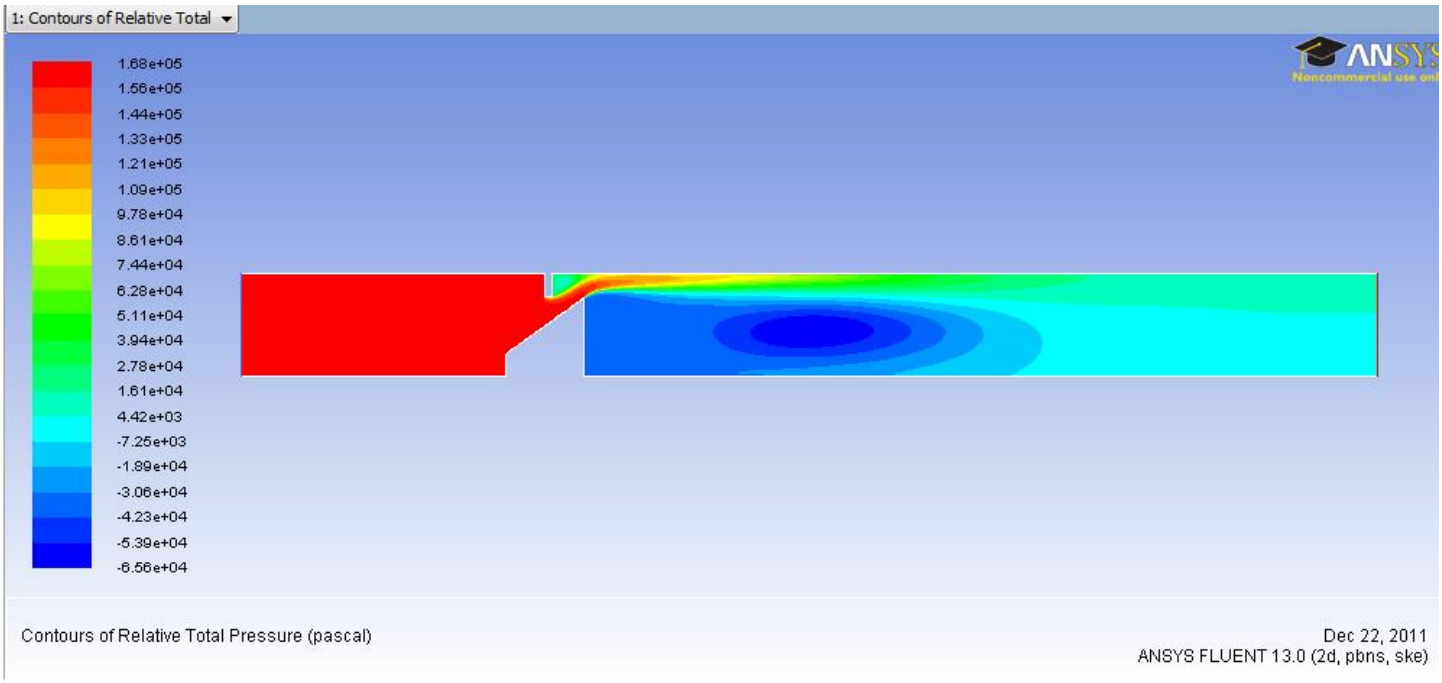
a. Relative total pressure evolution

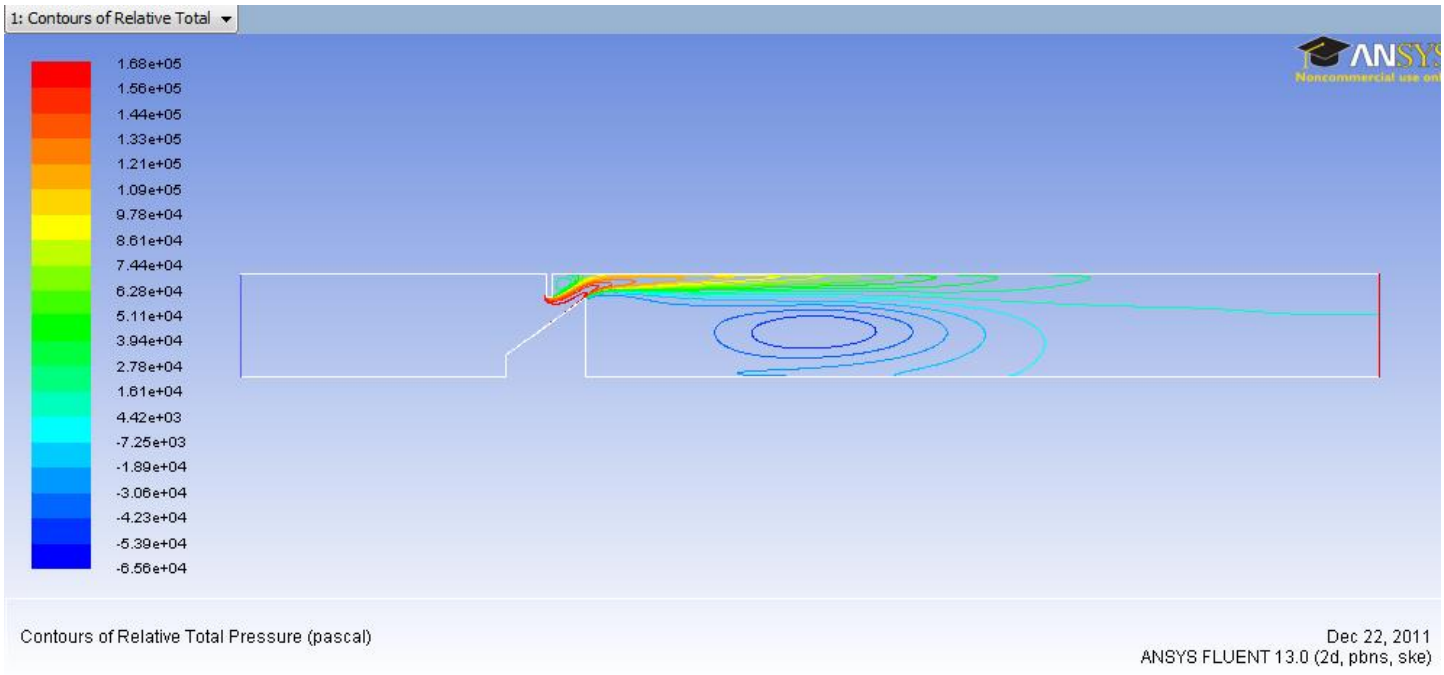


b. Velocity magnitude evolution

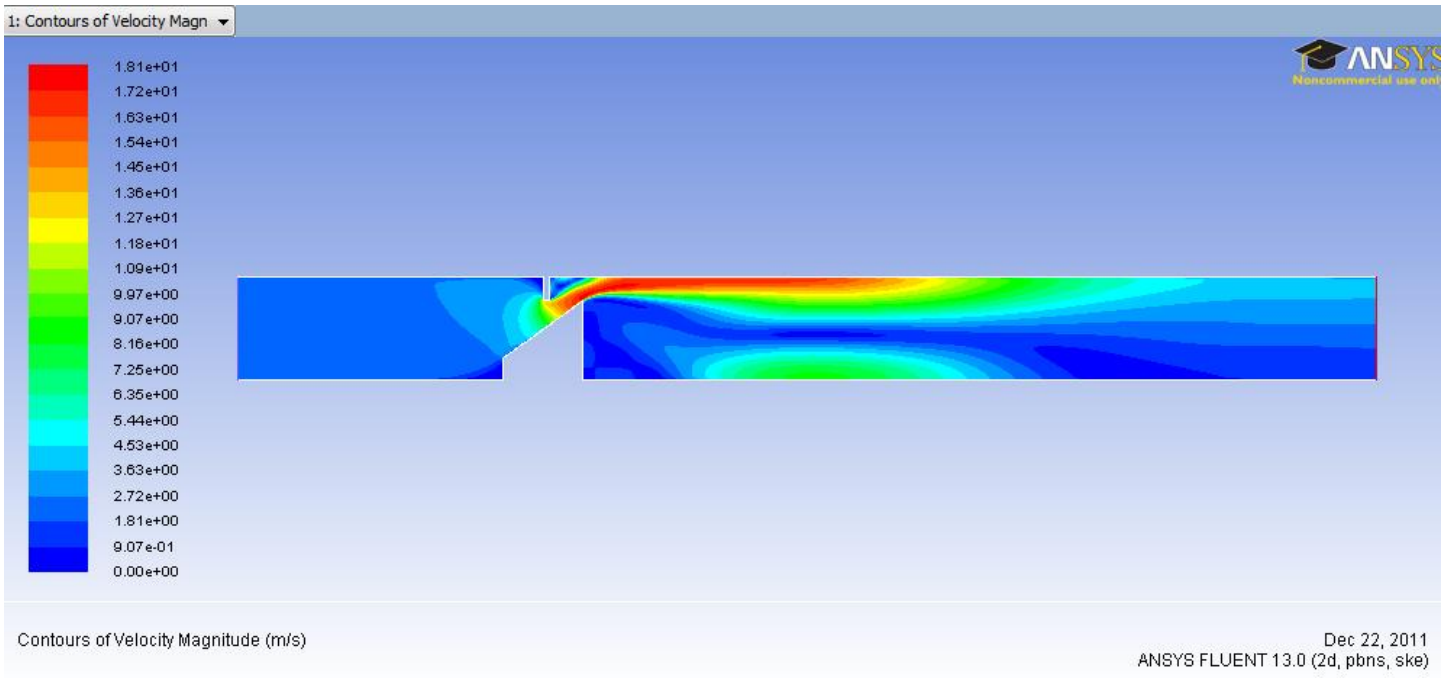


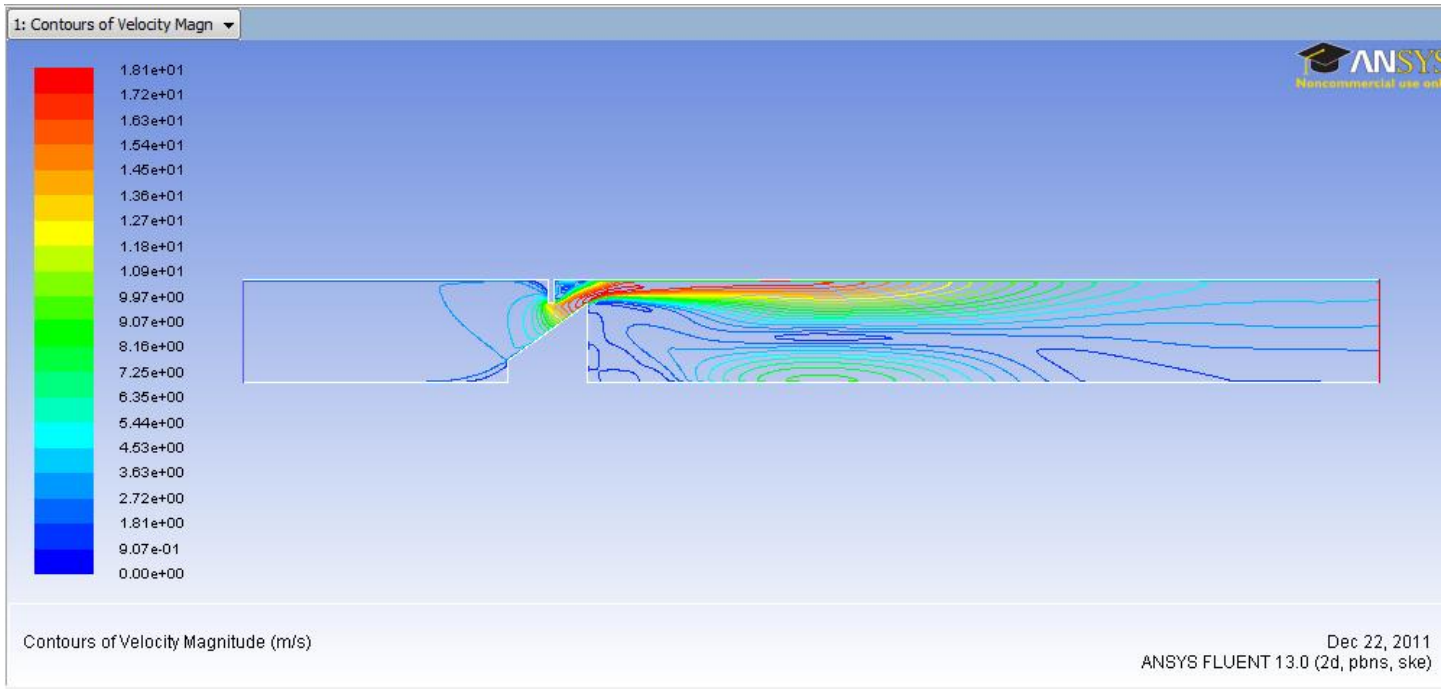
c. Relative total pressure



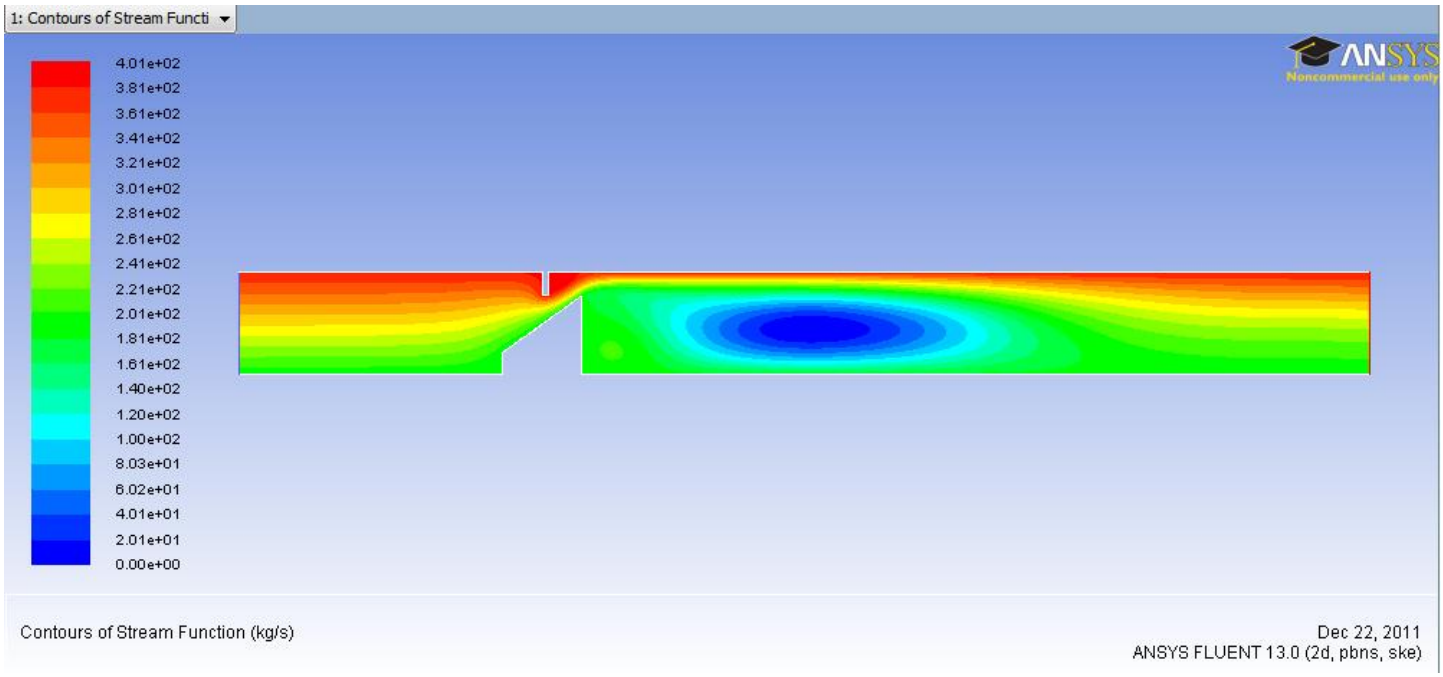


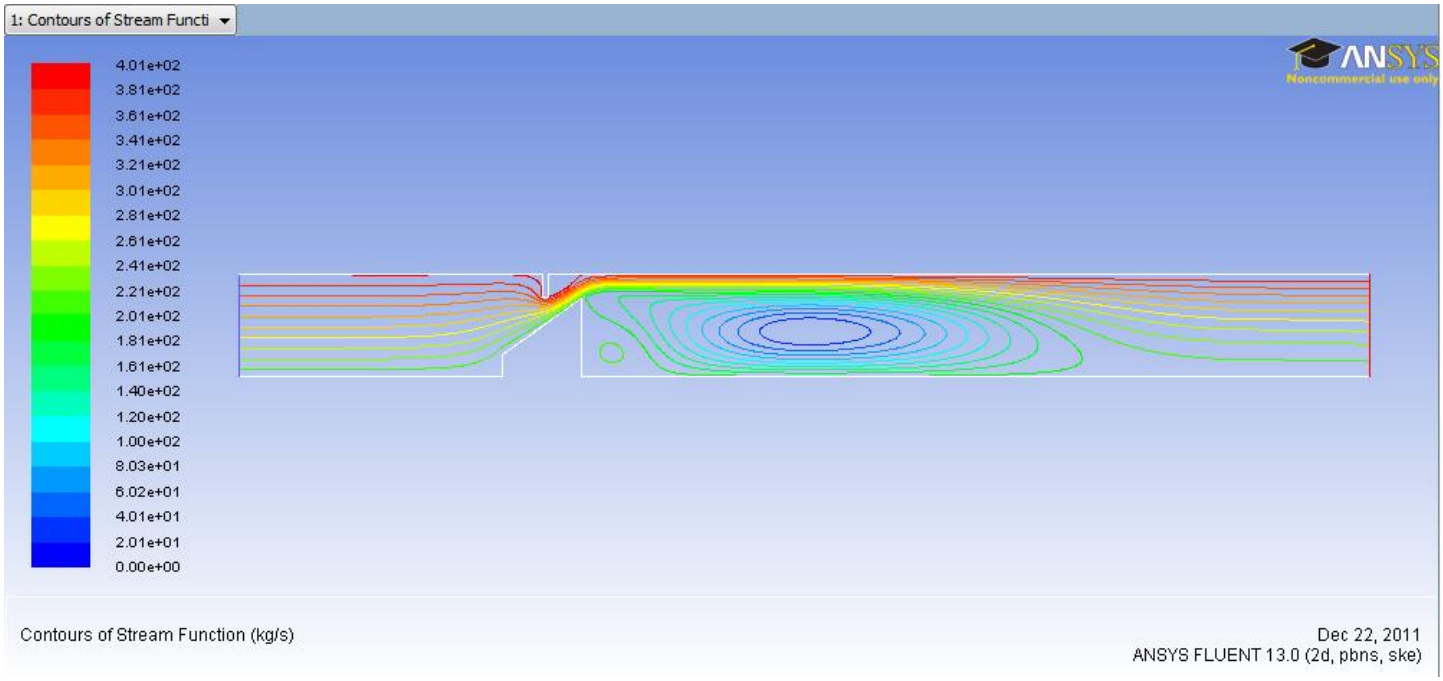
d. Velocity magnitude





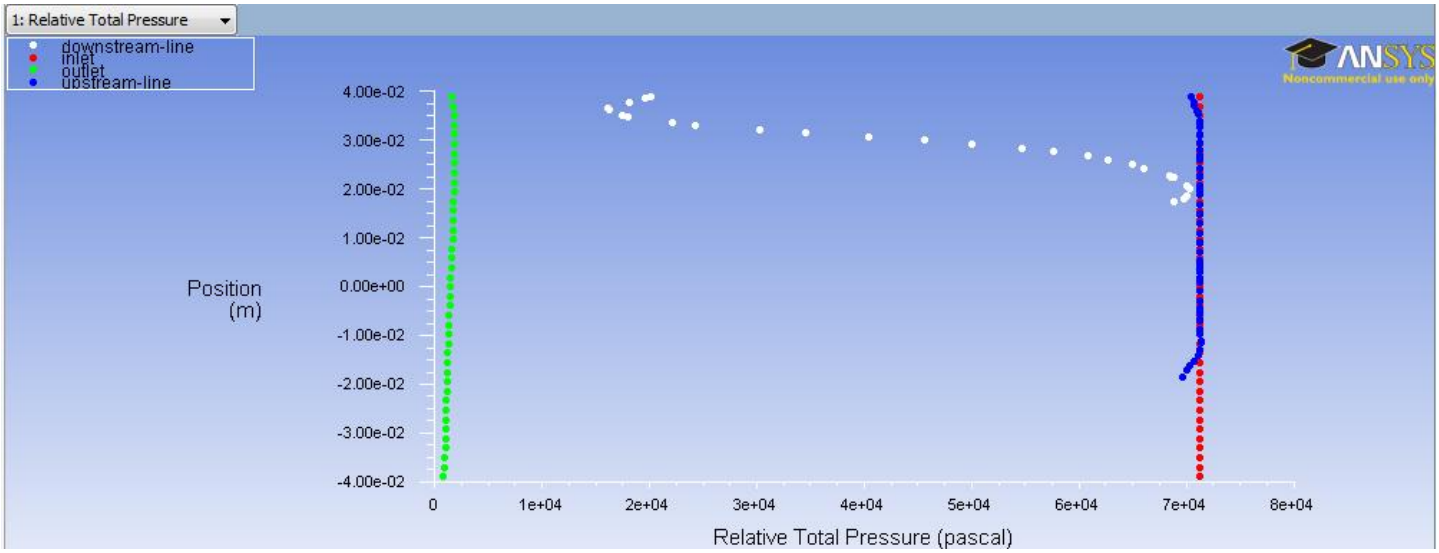
e. Velocity stream



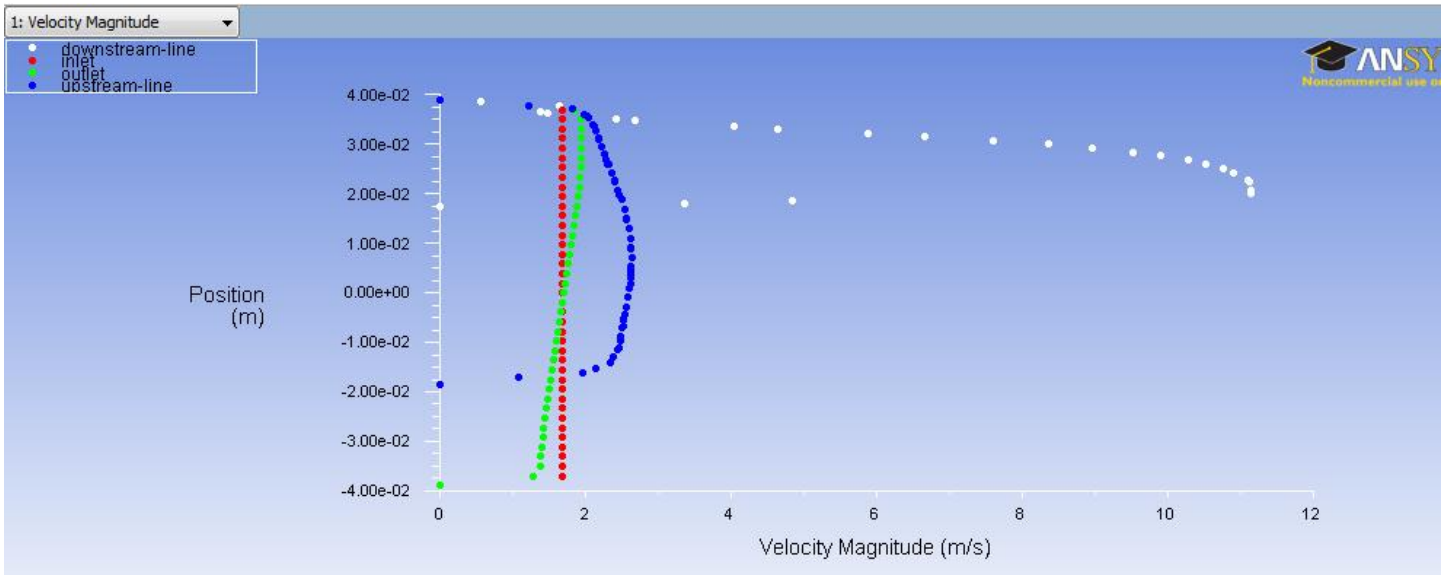


2. Coarse grid

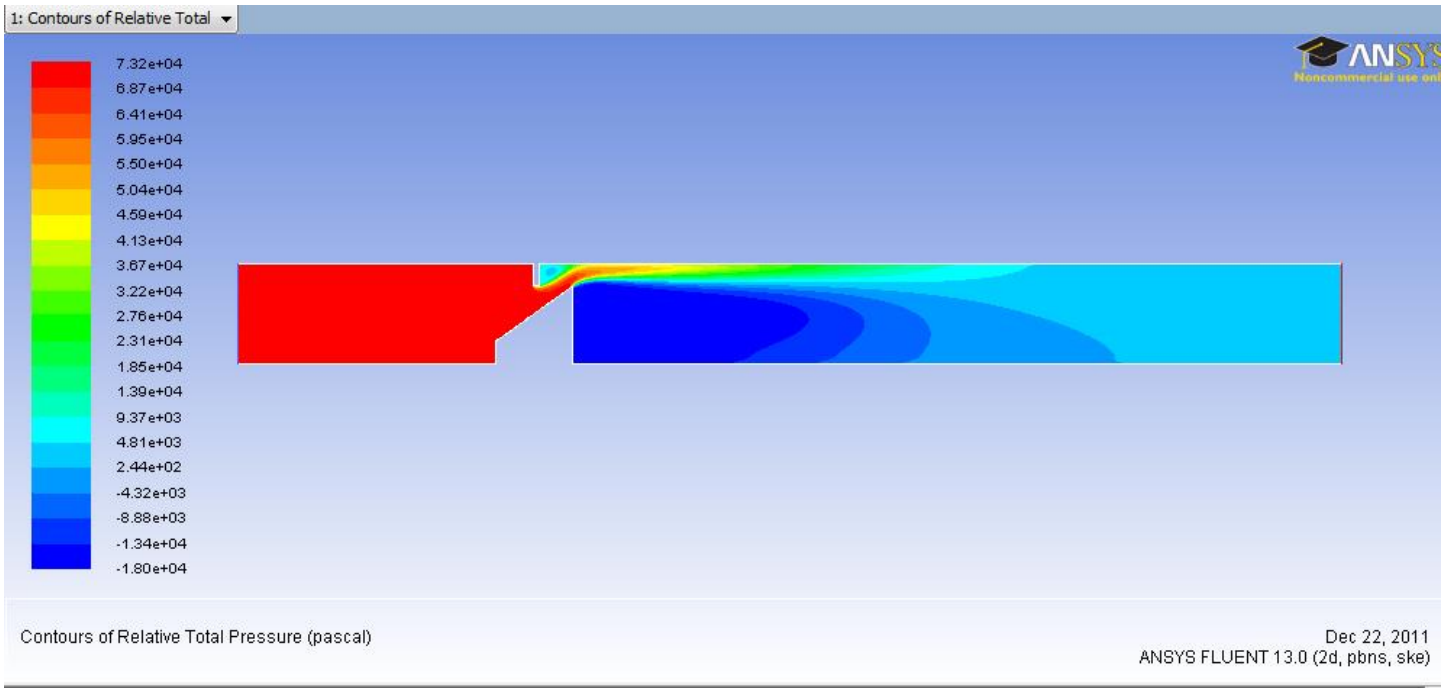
A. Pressure evolution

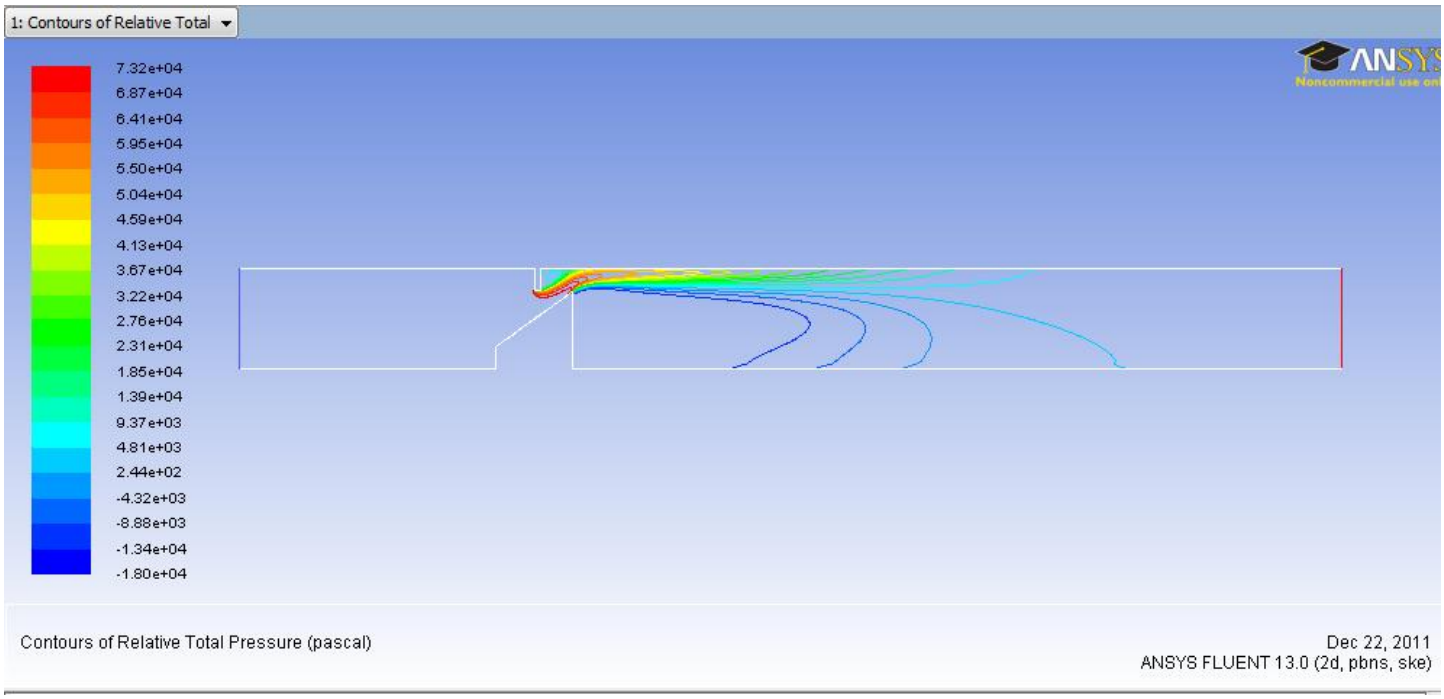


B. Velocity magnitude evolution

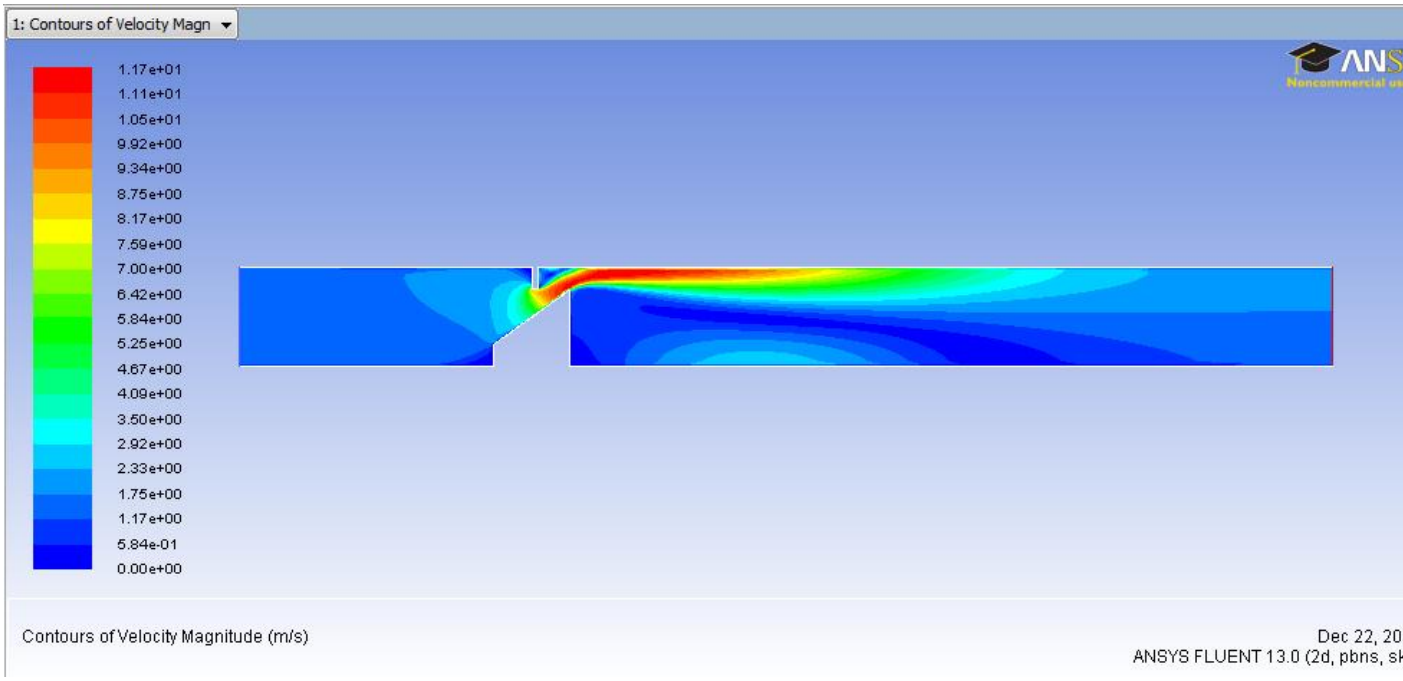


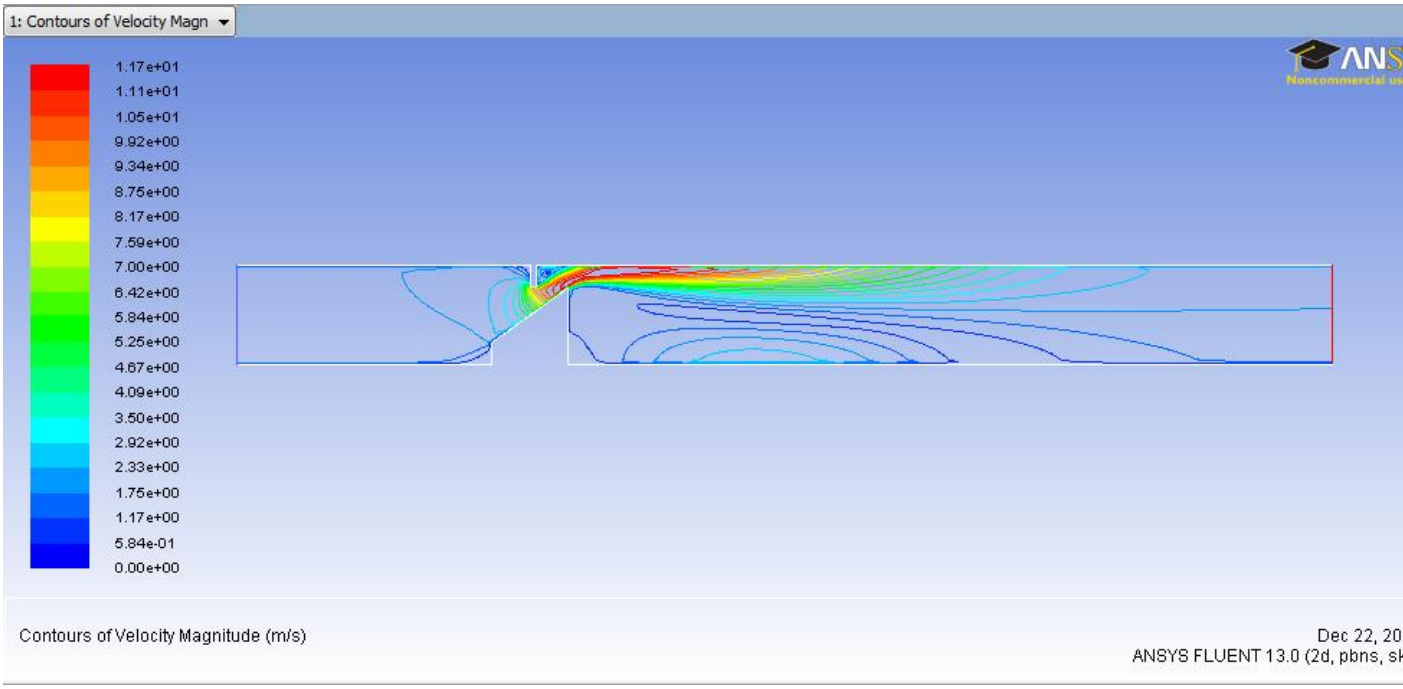
C. Relative total pressure



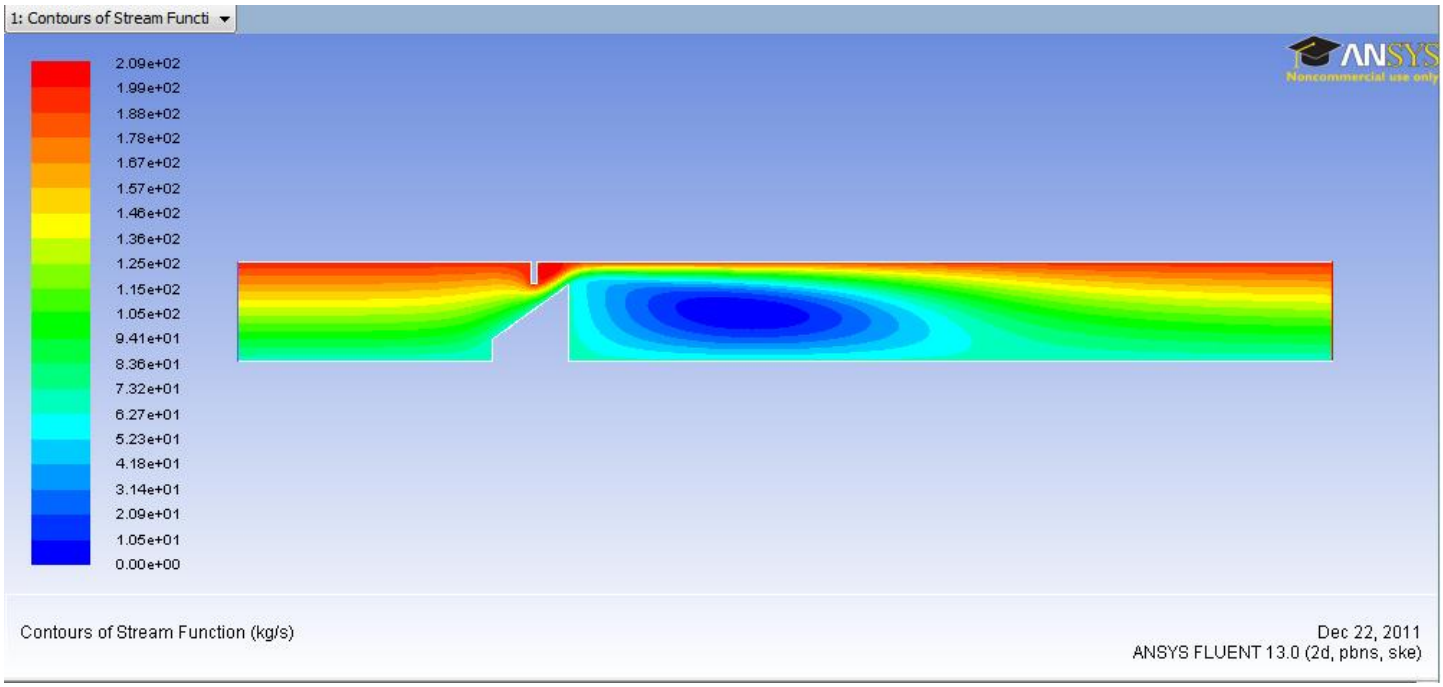


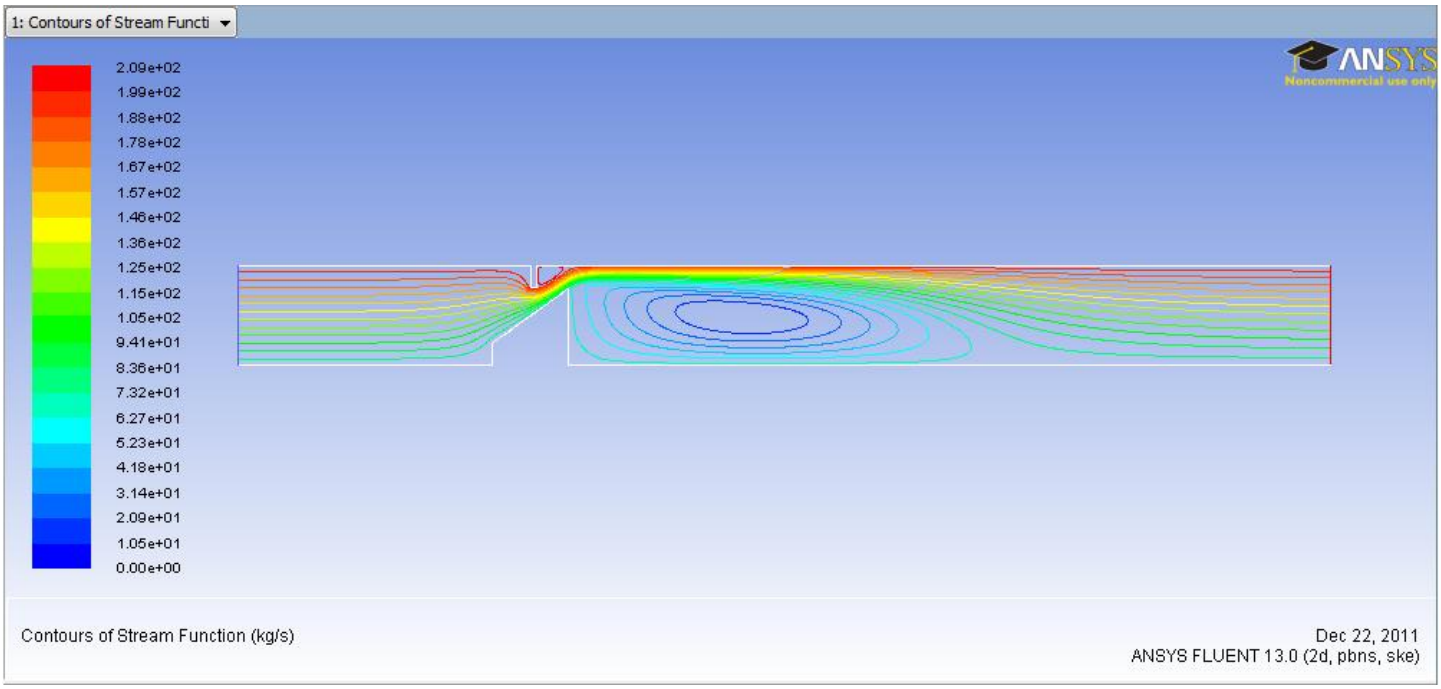
D. Velocity magnitude





E. Velocity stream





F. Observations & Comments

All the pictures outlining either velocity or pressure lead to the same results and observations.

On the one hand, the velocity increases from the outlet to the downstream line and decreases when the flow goes out of the variable area. As predicted in the expectations part, the maximum velocity is reached at the downstream line since it is the most constricted zone. It is also possible to visualise two swirls, which are created at each corner of the bluff body end, consequently the flow is well and truly turbulent.

On the other hand, the pressure is decreasing from the inlet to the outlet with a conservation of the high pressure towards the upper part of the venture which corresponds to the exit of the constricted area. On the whole, the most important pressure is at the inlet and it stays constant to the upstream line, which is coherent since the volume is constant and the material is incompressible. Moreover, it has been noticed that an important fall of pressure appears after both obstacles at the top corner and bottom one. The significant difference of pressure brought about a turbulent flow again.

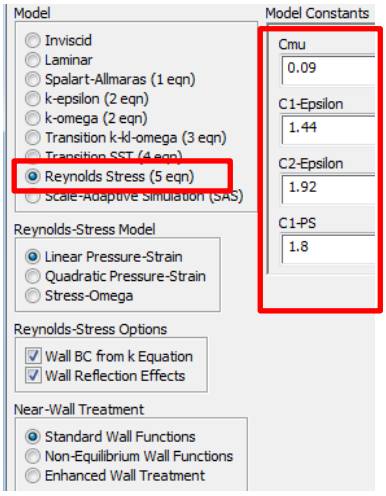
Obviously, the magnitude of the pressure and the velocity vary according to the incoming flow rate. Therefore faster is the incoming flow rate higher is the maximum pressure and velocity since there are linked by a physical law.

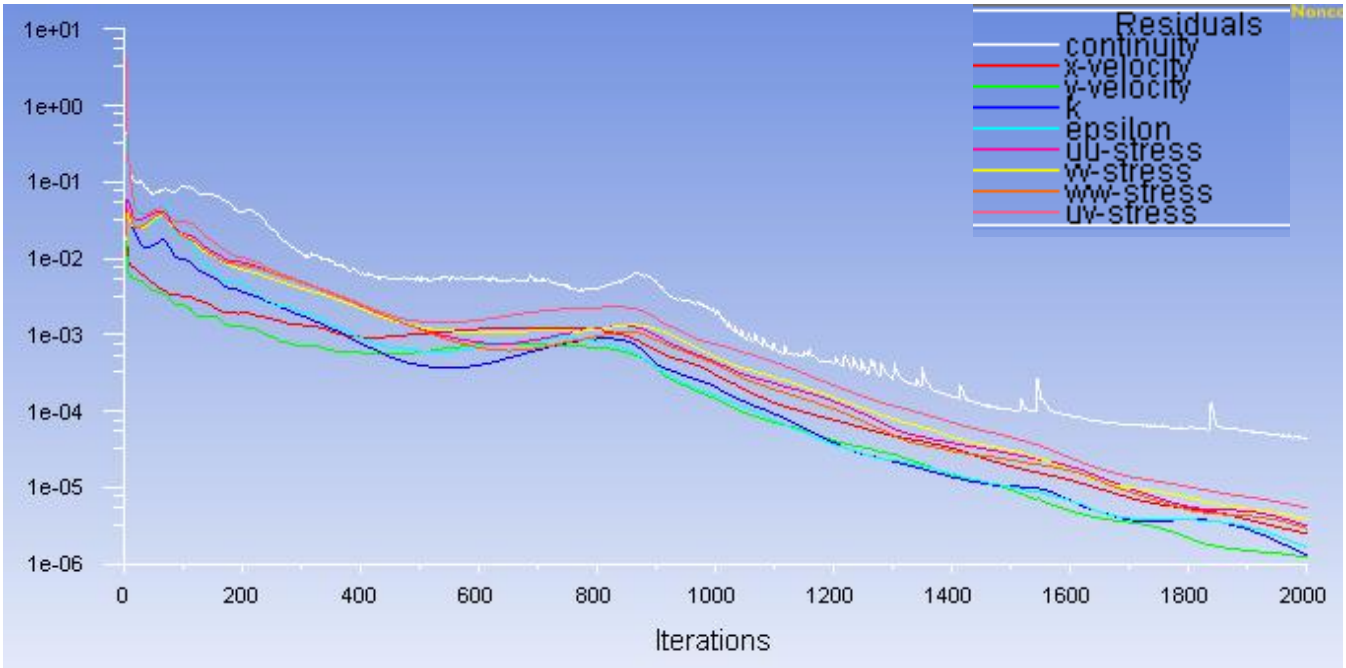
Regarding the differences between the fine and coarse grid for a flow rate equals to 8l/s. The information took down show that the maximum pressure for the fine and coarse grids is respectively equals to $7.44e04$ Pa and $7.32e04$ Pa, i.e. ≈ 1.6% of error. Furthermore, the maximum velocity is equal to 12.1 m/s for the fine grid and 11.7 m/s for the coarse one, i.e. ≈ 3.5% of error. Therefore the error between both grids is definitely negligible since the simulation has been undertaken using important flow rates (0.84, 1.67 and 2.51 m/s), which have not been affected by significant error.

IX. Further analysis

1. RNS

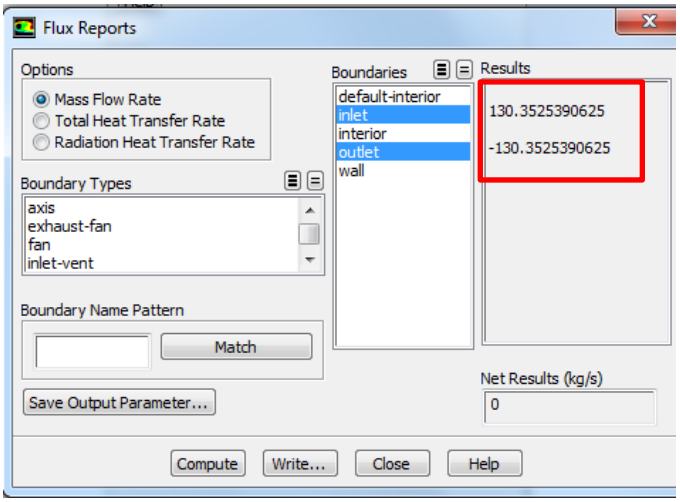
In order to have a second point of view of the fluid behaviour, a new physical model has been run to solve the Navier-Stokes equations using the fine mesh. This model called Reynolds Stress is made up five equations, which is physically most complete model although it requires more CPU time but it is not a priority in that case. However, the simulation has been set up for the flow rate equal to 8l/s to compare it with the two meshes used previously.





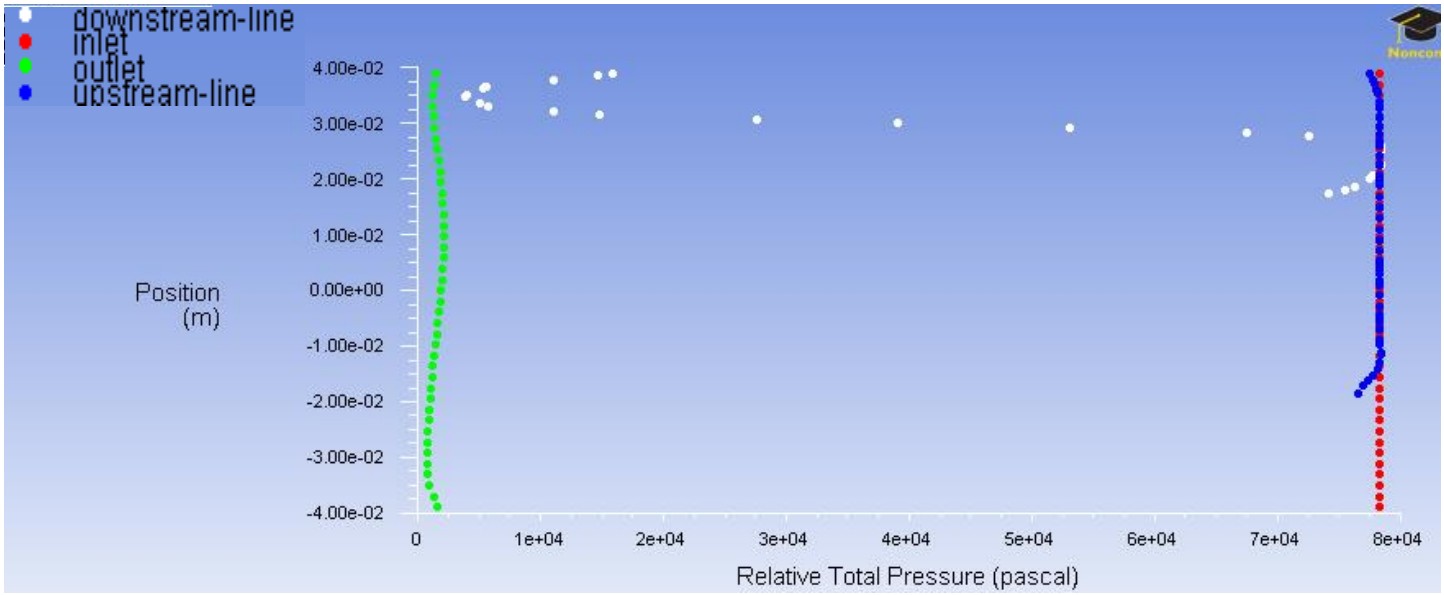
iter	continuity	x-velocity	y-velocity	k	epsilon	uu-stress	vv-stress	ww-stress	uv-stress	time/iter	
1991	4.5816e-05	2.6364e-06	1.2640e-06	1.4468e-06	1.8112e-06	3.3742e-06	4.0882e-06	3.1144e-06	5.6687e-06	0:00:00	9
1992	4.5480e-05	2.6283e-06	1.2616e-06	1.4355e-06	1.7999e-06	3.3523e-06	4.0715e-06	3.0969e-06	5.6452e-06	0:00:00	8
1993	4.5539e-05	2.6165e-06	1.2553e-06	1.4243e-06	1.7871e-06	3.3305e-06	4.0560e-06	3.0793e-06	5.6228e-06	0:00:00	7
1994	4.5858e-05	2.6051e-06	1.2500e-06	1.4155e-06	1.7784e-06	3.3089e-06	4.0397e-06	3.0614e-06	5.6008e-06	0:00:00	6
1995	4.3641e-05	2.5967e-06	1.2477e-06	1.4050e-06	1.7679e-06	3.2877e-06	4.0246e-06	3.0426e-06	5.5781e-06	0:00:00	5
1996	4.4597e-05	2.5857e-06	1.2427e-06	1.3939e-06	1.7549e-06	3.2659e-06	4.0081e-06	3.0248e-06	5.5559e-06	0:00:01	4
1997	4.5013e-05	2.5732e-06	1.2368e-06	1.3847e-06	1.7457e-06	3.2452e-06	3.9919e-06	3.0054e-06	5.5326e-06	0:00:01	3
1998	4.4502e-05	2.5685e-06	1.2373e-06	1.3751e-06	1.7380e-06	3.2235e-06	3.9769e-06	2.9893e-06	5.5103e-06	0:00:00	2
1999	4.4680e-05	2.5578e-06	1.2326e-06	1.3678e-06	1.7244e-06	3.2024e-06	3.9617e-06	2.9704e-06	5.4881e-06	0:00:00	1
2000	4.5191e-05	2.5487e-06	1.2294e-06	1.3596e-06	1.7174e-06	3.1826e-06	3.9465e-06	2.9528e-06	5.4664e-06	0:00:00	0

The list of iterations proves the convergence of the solution since the residuals are roughly equal to 1*e*-06, which corresponds to the absolute criteria of convergence set before.

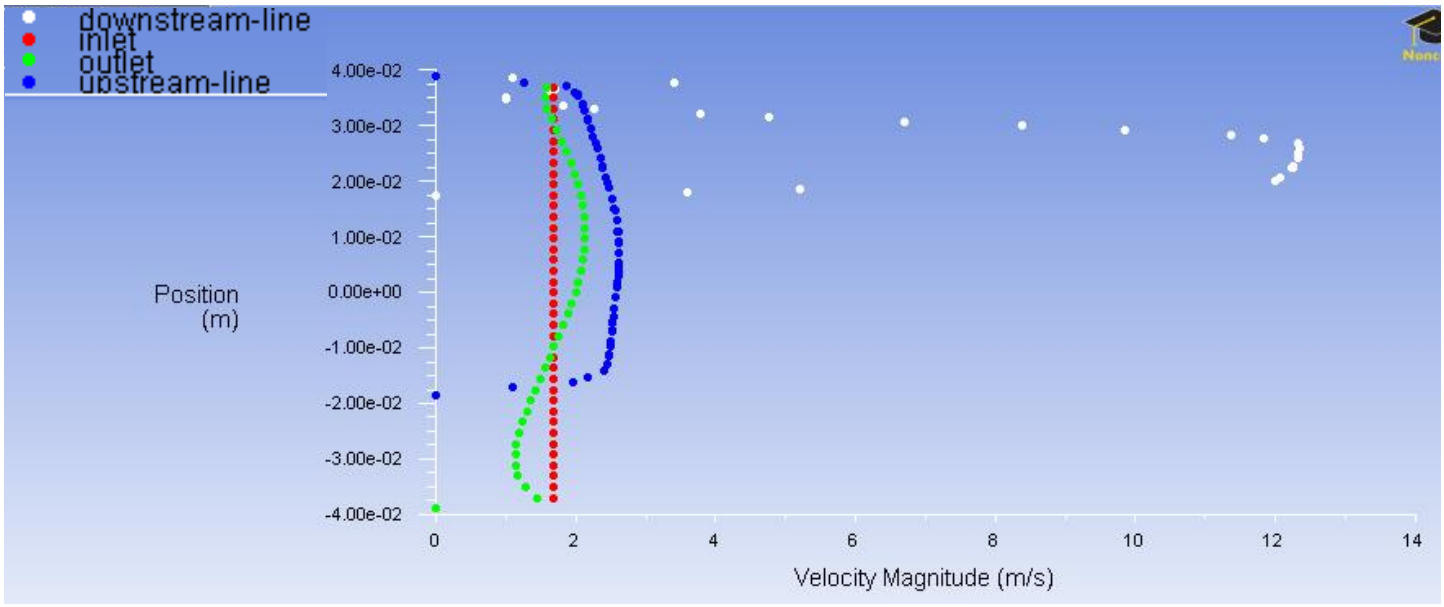


The mass flux is conserved. Simulation, therefore, can be carried on.

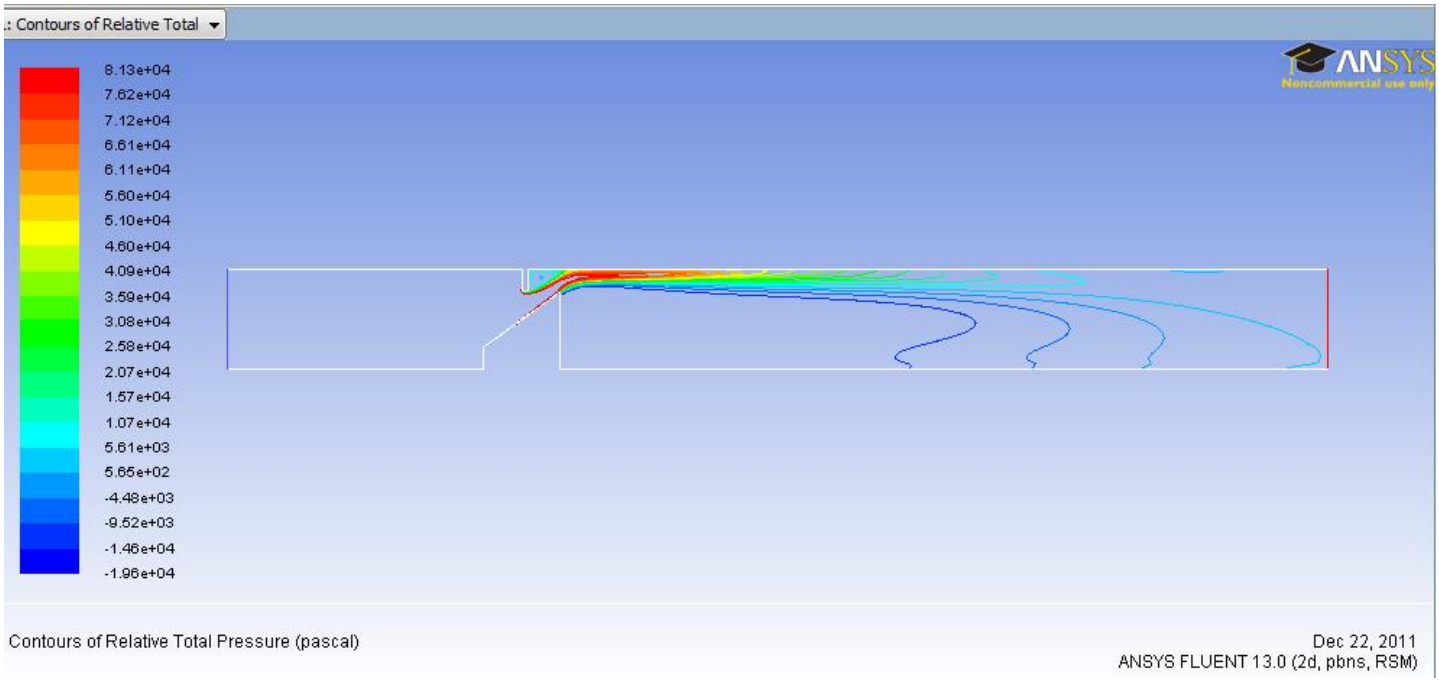
A. Relative total pressure evolution



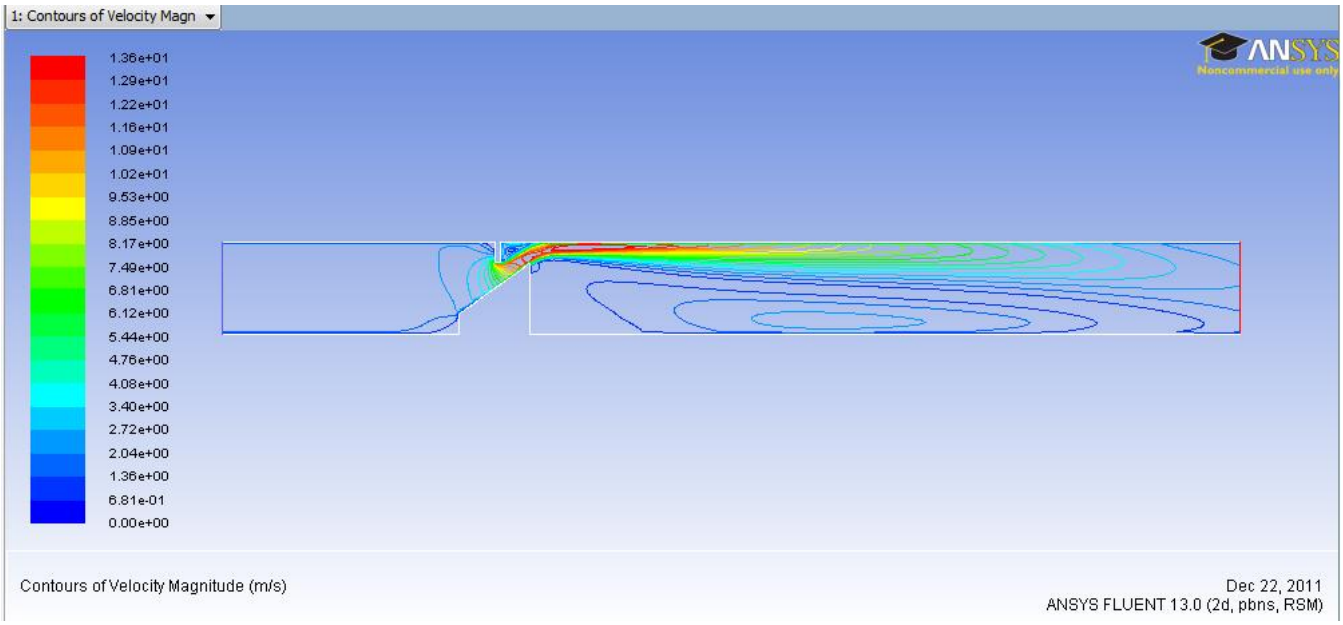
B. Velocity evolution



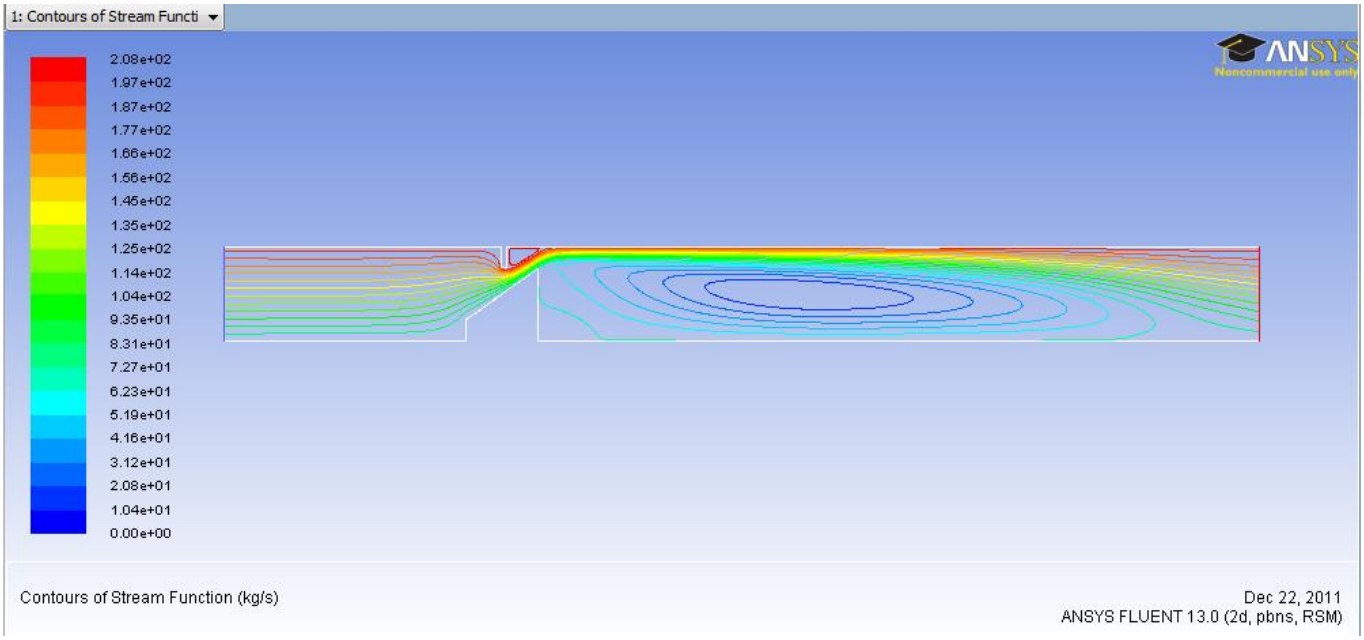
C. Relative total pressure



D. Velocity Magnitude



E. Velocity stream



F. Observations and comments

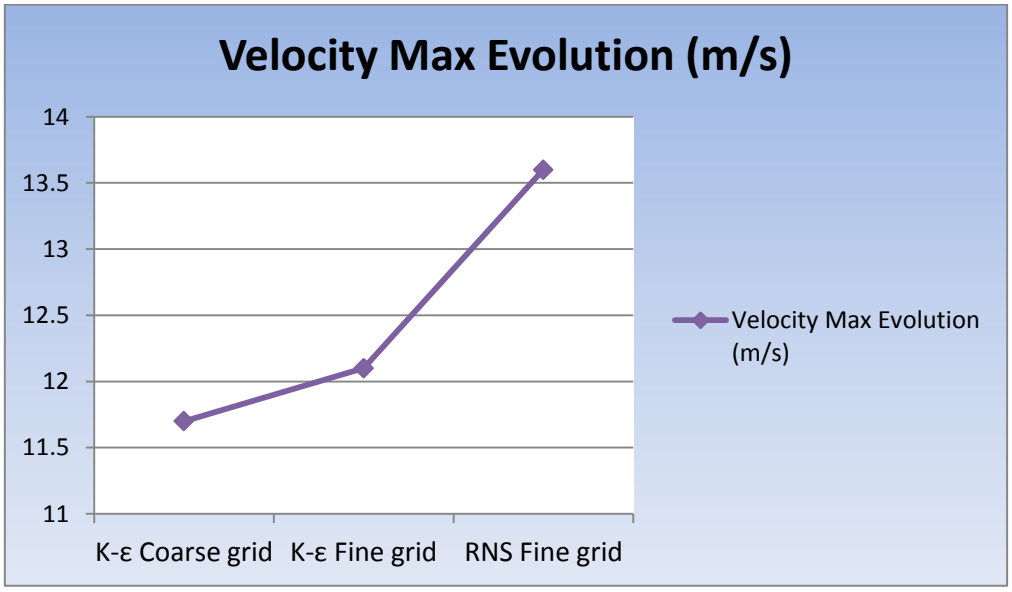
The behaviour of the fluid is quite similar to the simulations carried out using K-ε model. A swirl is also created after the constriced cross-sectional area, which demonstrates a turbulent state. Regarding the pressure and velocity magnitudes there are sizeable differences: respectively 8% and 11% more than the fine grid; 10% and 14% more than the coarse grid. The use of several models is necessary to compare the results and to get disadvantageous.

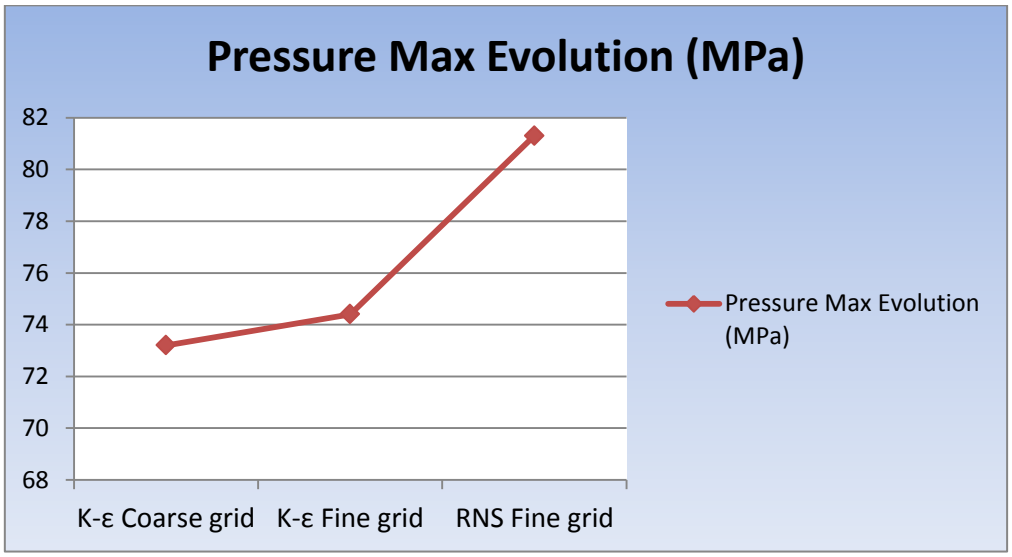
X. Conclusion

The question that must be asked corresponds to the following:

- **Did we get repeatable results with a quantified error?**

The answer is, yes. Even if the results are logically different using different meshes and models, the magnitudes of pressure and velocity do not change much in the cases studied. Therefore, we have got repeatable results with a quantified error, which is roughly equal to 10%.





By plotting the evolution of the maximum pressure and velocity for each models and meshes used for the simulation, the link between both physical properties is proved since the evolution seems to be the same. Moreover, the fine mesh has provided

more accurate results with the same model (K- **ε**), and by using the RNS model, which is the more advanced, the best accuracy has been reached, therefore the simulation can be defined as convergent.

If in doubt, further simulations may be run using different turbulent model and algorithm as SIMPLEC. In addition, the mesh could be improved using different methods and size of elements. But in this case, neither the absurdity of the results nor the accuracy of them has been noticed therefore the research seems to be true, be careful CFD-based predictions are never 100%-reliable!