INGENIØRHOJSKØLEN I ÅRHUS

AIR FLOW STUDY IN A TWO BRANCHES MANIFOLD

MECHANICAL DESIGN FINAL PROJECT

PABLO ANDRÉS YAGÜE
RICARD ESTEVE MONTES
JAUME LLOPART VALLS

SUPERVISOR: JENS BRUSGAARD VESTERGAARD

June 3rd 2010

ACKNOWLEDGEMENTS

We owe a great thanks to many people who helped and supported us during the development of this project.

Our deepest thanks to our supervisors, Jens Brusgaard Vestergaard and Søren Gundtoft, who suggested us the project theme and have always been there to guide and help us when we have needed it. Without the help of their valuable suggestions, guidance and encouragement, this project would not have been possible.

We also express our thanks to the IHA, for giving us the opportunity of coursing here the international program Mechanical Design and offering us everything we have needed: teachers, equipment and even accommodation.

Our deep sense of gratitude to our home universities, especially to the coordinators, who have followed the project development in order to check if it was compatible with our home studies and have been available to answer our questions.

We could not finish the acknowledgements without express our thanks to Gerner, the carpenter who prepared all the wood pieces we demanded in order to build the models; Berner, who lent us all the equipment we needed in our tests; and Aage, who sent us the CFDesign data.

Sincerely yours,

Spring semester 2010, Project group Omega

PABLO ANDRÉS YAGÜE   RICARD ESTEVE MONTES   JAUME LLOPART VALLS

TABLE OF CONTENTS

1 INTRODUCTION

This report is the result of a semester group project and it supposes for all of us our previous step to the working world, which encouraged us to do our best and focus our effort on it. When our coordinators exposed us the idea to develop a cooling system for a wind turbine generator we did not hesitate, as we regarded the subject as very interesting and a possibility to show all the knowledge acquired in the Aerodynamic course and in the Multidisciplinary project we did last semester.

In this report, we tried to display what we have learned in the last four months and a half and to show our understanding of the subject matter.

1.1 PROJECT DESCRIPTION

This project consists on a cooling system for a wind turbine generator. The main concept used to cool the generator is based on a manifold, which consists in an arrangement of pipes used to redistribute the flow of a fluid or gas, typically from a single inlet to a number of outlets, or vice versa.

The first idea was to design a manifold with an inlet and four outlets using an equation solver program called EES, SolidWorks Flow Simulation as an application of computational fluid dynamics (CFD) analysis and a wooden model built and tested, and try to get the total control of the system.

A first step has been to carry out all the calculations for a two branch model. This has given us a general idea of how the flow works and how it is distributed between the branches, and this two branches model (instead of four) simplifies the calculations of the model at first. This model, which its main branch is 30 mm wide and the outlet branches widths are 19.5 and 21.5 mm, has been introduced to an EES program, designed in SolidWorks and built in wood to obtain the pressures, velocities and flows for the circulating fluid along the conduits. Once the results for the three methods have been obtained, these has been analyzed and compared between them to draw some conclusions about the similarities or differences of the data from the three methods.

After the first analysis we have decided to build again a two branches model but with narrower conduits, of 5 mm both the main branch and the outlet branches. This second model has been designed to see if the flow behaves the same way when it circulates through narrow conduits. Here we have had to design a new method to measure the parameters inside the thin conduits. This method, which we called “Sphere method”, consists of a small lead ball which moves more or less depending on the pressure applied on its surface. Once compared to the other two computational methods, it has been tested in the wide branches model and compared to the results obtained there measuring with a Pitot tube, and has resulted to be more accurate than expected, so we have decided to enhance it by working with more lengths to calculate the velocity profile curve and improve the calibration of the threads so the results were even more precise.

In other matters we have studied the pressure loss in a tubular pipe which is narrower in the middle. This study has been performed using three computer programs; the already known EES and SolidWorks Flow Simulation Studio, and CFDesign. This last comparison has been expected to show the differences between three methods based on theoretical calculations.

2 WIDE BRANCHES MODEL

2.1 TEST REPORT

Test Date: March 15th, 2010

Test Engineers: Pablo Andrés, Ricard Esteve, Jaume Llopart

2.1.1 Model Description

The model consists of three wood blocks separated a certain distance so the air can flow between them. These three blocks are covered on the top at some distance by another piece of wood, so we get a main branch that is split in two. This assembly wants to represent a kind of simple manifold. The drawing below shows the dimensions and an entire view of the model.

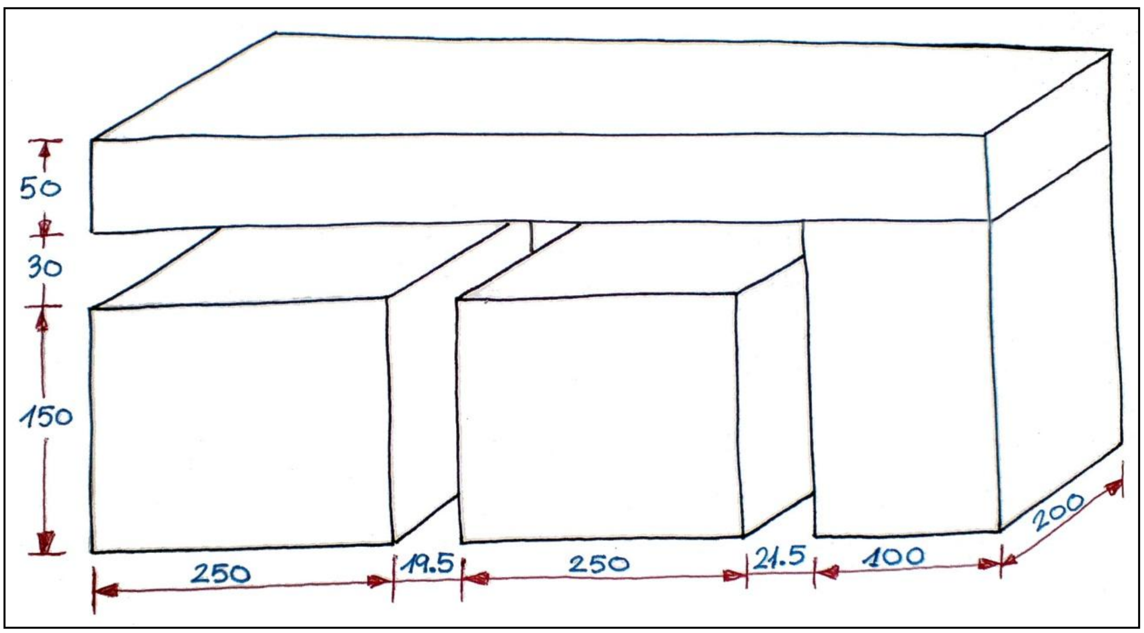


Figure 1. Model overview.

On both sides the system is closed with two transparent holey plastic pieces which at the same time work as union between all the parts. The holes in the plastic covers are distributed so that there are six control points for the main branch and three for each of the secondary ones.

Due to the geometry of the Pitot tube, the holes have to be drilled at some distance from the control point. In the following picture is shown the exact position of the control points.

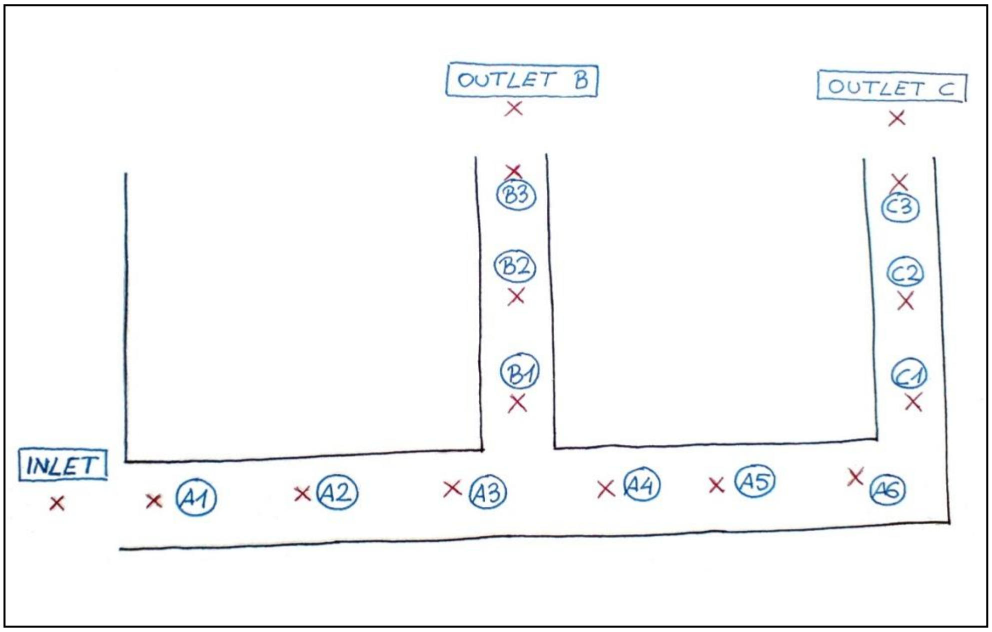


Figure 2. Holes situation.

The air flows through the empty spaces in the model. This air comes from a homemade fan, built with a 1.5 kW three-phase electric motor which gives a rotational speed of up to 2870rpm at its shaft. The motor is held to some blades which produce, thanks to the movement, the draft needed. This fan provides the required speed and flow at its outlet, a flexible tube connected directly to the model. Before entering the model, the flow is forced to circulate through a diffuser so it becomes better even distributed. This diffuser is made by a holey plastic plate with fifty (eight millimetres diameter) holes uniformly distributed in eleven rows, six of them with five holes and the remaining five with four holes each, as shown in Figure 3. This holey plate is placed inside a closed wood box where the air flow is introduced by one side and let out to the model by the opposite side.

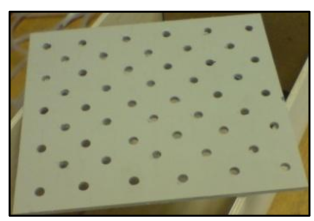


Figure 3. Diffusion plate.

2.1.2 Test Description

The pressure values for each of the fifteen control points are taken with a 4-millimeter diameter Pitot tube. The tube is introduced into a hole and placed completely parallel to the flow. Although the Pitot is being held with the hand during the entire test, it is important to keep it as still as possible to get the most reliable results. The Pitot tube is moved from hole to hole to take all the measurements while the other holes are being covered. In each control point where the

Pitot is located, several measures are taken. The pressure is measured in the middle of the duct and next to the two walls, thus we can take an average value for the pressure in the area under study. Moreover, it is also measured at seven different depths, getting a more accurate representation of the pressure profile. That means twenty-one data values for each control point.

The pressure values are read in three water column scales. These scales are filled with tinted alcohol, and can measure up to twenty millimetres (196 Pa). The first scale is measuring the pressure at the fan outlet, so we can notice if there is any variation in the flow speed. The other two scales are measuring the static and dynamic pressure given by the Pitot tube. These two last scales should be filled with more alcohol than the zero level, due to the depression values measured in some control points throughout the model.

2.1.3 Measurements

The following results were taken in the laboratory, at an ambient temperature of 16,5 ºC (289,65K). Results are an average of the 3 different control points in each depth. Units are in Pascals [Pa].

Depth	1/8			1/4			3/8			1/2		
	Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Fan	0,0	63,7	63,7	0,0	63,7	63,7	0,0	63,7	63,7	0,0	63,7	63,7
I1	84,3	72,6	156,9	91,2	72,6	163,8	98,1	76,5	174,6	111,3	73,1	184,4
A1	54,9	10,1	65,1	58,8	8,8	67,7	65,4	8,8	74,2	72,6	12,3	84,8
A2	65,4	11,1	76,5	69,3	11,1	80,4	70,6	11,4	82,0	80,3	15,0	95,3
A3	60,8	42,8	103,6	63,7	44,1	107,9	64,4	45,1	109,5	70,6	50,3	120,9
A4	61,8	43,1	104,9	63,7	43,8	107,5	65,1	43,5	108,5	71,4	46,6	118,0
A5	61,5	45,1	106,6	62,4	44,8	107,2	64,4	45,1	109,5	71,6	49,7	121,3
A6	60,1	45,1	105,3	61,8	44,1	105,9	62,4	45,1	107,5	70,0	49,0	119,0
B1	20,6	-24,5	-3,9	3,9	-24,5	-20,6	10,5	-21,6	-11,1	24,4	-18,8	5,6
B2	22,9	0,3	23,2	19,3	0,0	19,3	2,9	18,0	20,9	29,1	1,5	30,4
B3	24,8	0,0	24,8	17,7	0,0	17,7	13,7	0,0	13,7	18,5	-0,2	18,3
O1	18,6	0,0	18,6	17,7	0,0	17,7	17,7	0,0	17,7	21,1	0,0	21,1
C1	33,0	-3,9	29,1	32,0	-7,2	24,8	37,3	-6,9	30,4	36,3	-4,6	31,7
C2	46,1	1,3	47,4	38,6	2,3	40,9	38,6	3,6	42,2	41,0	5,4	46,4
C3	47,4	-0,3	47,1	34,7	-1,0	33,7	36,9	-0,3	36,6	35,5	0,2	35,6
O2	37,3	0,0	37,3	33,3	0,0	33,3	33,3	0,0	33,3	32,4	0,0	32,4

Table 1. Test results I.

Depth	5/8			3/4			7/8		
	Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Fan	0,0	63,7	63,7	0,0	63,7	63,7	0,0	63,7	63,7
I1	117,7	68,6	186,3	115,7	78,5	194,2	115,7	78,5	194,2
A1	70,9	11,8	82,7	73,2	8,5	81,7	66,7	6,2	72,9
A2	87,0	16,3	103,3	87,3	16,7	104,0	90,9	16,0	106,9
A3	76,5	52,6	129,1	77,1	49,0	126,2	73,9	45,1	119,0
A4	75,5	50,3	125,9	71,3	49,0	120,3	60,8	48,4	109,2
A5	77,5	53,9	131,4	74,2	53,9	128,1	66,0	53,3	119,3
A6	74,9	52,6	127,5	71,3	50,7	121,9	66,4	50,0	116,4
B1	38,9	-18,3	20,6	21,6	-18,0	3,6	26,2	-19,6	6,5
B2	25,2	3,3	28,4	23,5	0,7	24,2	19,9	-0,7	19,3
B3	21,6	0,0	21,6	18,6	0,0	18,6	17,7	-0,3	17,3
O1	21,6	0,0	21,6	16,7	0,0	16,7	15,7	0,0	15,7
C1	39,9	-4,2	35,6	31,4	-3,6	27,8	31,4	-1,0	30,4
C2	28,8	5,2	34,0	39,2	5,2	44,5	45,8	3,3	49,0
C3	37,3	0,0	37,3	36,9	0,3	37,3	42,2	0,0	42,2
O2	31,4	0,0	31,4	31,4	0,0	31,4	40,2	0,0	40,2

Table 2. Test results II.

In Table 1 and Table 2: *Dyn* = Dynamic pressure. *Stat* = Static pressure. Both measured with the Pitot tube. *Total* = Total Pressure, being the sum of both Dynamic and Static pressures.

Pitot tube readings give an uncertain error of ±0.1 mm.w.c. in each measurement.

2.1.4 Analysis of results

2.1.4.1 <u>Pressures</u>

These are the resulting pressure values from the average calculations in each control point. The average is done taking the values in every single depth.

	Averages in control points [Pa]		
	Dynamic p.	Static p.	Total p.
Fan	-	63.7	63.7
I1	105.7	74.2	179.8
A1	66.9	9.8	76.7
A2	78.9	14.1	93.0
A3	69.7	47.4	117.1
A4	67.6	46.4	114.0
A5	68.6	49.4	118.1
A6	67.1	48.2	115.3
B1	21.3	-20.5	0.8
B2	21.5	3.1	24.5
B3	18.9	-0.1	18.8
O1	18.8	0.0	18.8
C1	34.7	-4.5	30.2
C2	39.9	4.0	43.8
C3	38.3	-0.1	38.2
O2	34.0	0.0	34.0

Table 3. Control points averages.

The first conclusion drawn from the table is the uncertainty from taking averages. Although the table is only for an overall analysis, it must be noticed that in certain points the pressure profile has a characteristic curve which is neglected in the table.

Concerning the main branch, it can be appreciated no big pressure loss between points A3 and A4, where first branch is located. Consequently, pressure values, both dynamic and static, are almost constant all along the main branch.

It is also specifically in the point between A3 and A4, in which there is the 90^{o} branch division, where fluid trajectory profile has a special curve.

This characteristic curve explains the negative static pressure values in the beginning of the branches. This is due to the fact that the trajectory of the fluid right after a 90º bend tends to go close to the furthest wall, so that the bend is not so closed.

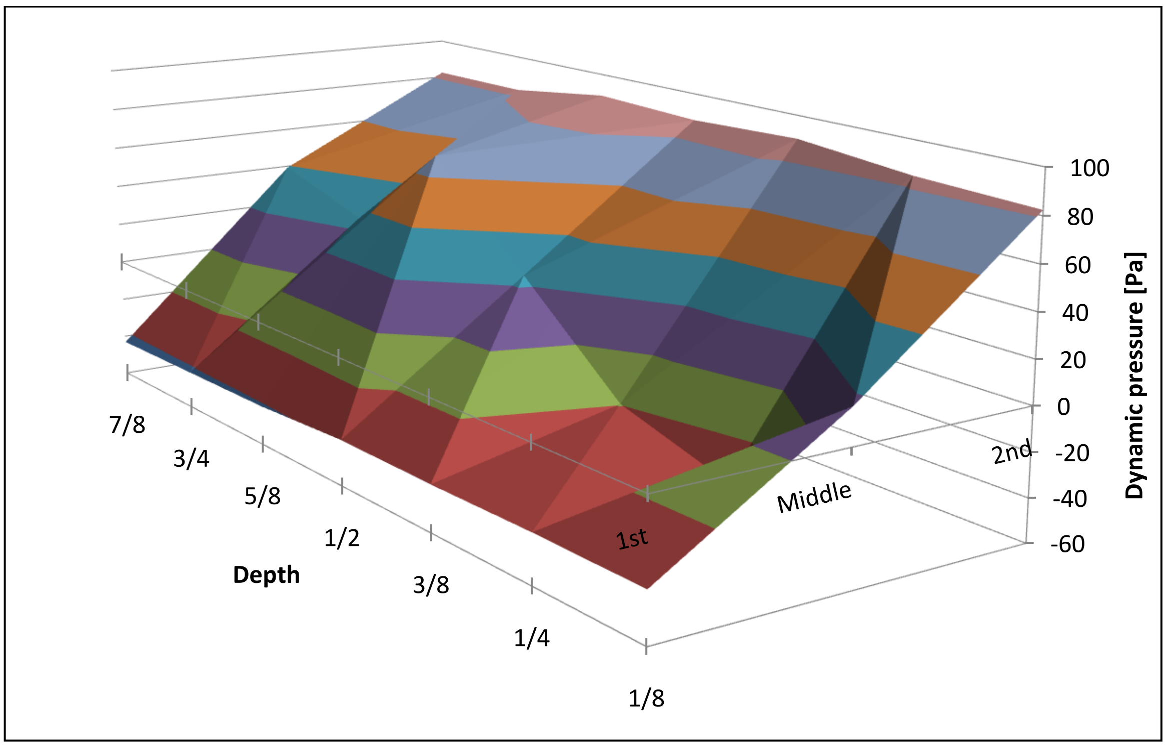
Therefore, the most complicated point to analyse is probably B1, the first control point in the branch. Its pressure profile, caused by the fluid trajectory along the division bend, made the point test more difficult than the others, as the minimum movement in the Pitot tube causes a big variation in the water column scale. Owing to this fact, a more accurate analysis in B1 is considered appropriate.

In the following table, there can be seen the complete measurements in point B1, in Pascals [Pa].

B1	1st wall			Middle			2nd wall		
	Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
1/8	-37,3	-22,6	-59,8	16,7	-26,5	-9,8	82,4	-24,5	57,9
1/4	-37,3	-24,5	-61,8	-35,3	-26,5	-61,8	84,3	-22,6	61,8
3/8	-38,2	-20,6	-58,8	-19,6	-24,5	-44,1	89,2	-19,6	69,6
1/2	-38,2	-16,7	-54,9	24,0	-22,6	1,5	87,3	-17,2	70,1
5/8	-41,2	-13,7	-54,9	68,6	-20,6	48,1	89,2	-20,6	68,6
3/4	-42,2	-17,7	-59,8	24,5	-16,7	7,8	82,4	-19,6	62,8
7/8	-43,1	-19,6	-62,8	39,2	-19,6	19,6	82,4	-19,6	62,8

Table 4. B1 values.

From the table it can be noticed, because of the fluid trajectory along the bend, the increasing of the dynamic pressure in the 2nd wall. In the following graphic it is represented the profile of the dynamic pressure along the width and the depth of the branch.



Graphic 1. B1 pressure profile.

This fluid trajectory is important to understand the test results and its values, especially when averages are done. The particular fluid behaviour in this point causes very different values from one wall to another, besides that can also cause inaccurate measurements in the Pitot tube.

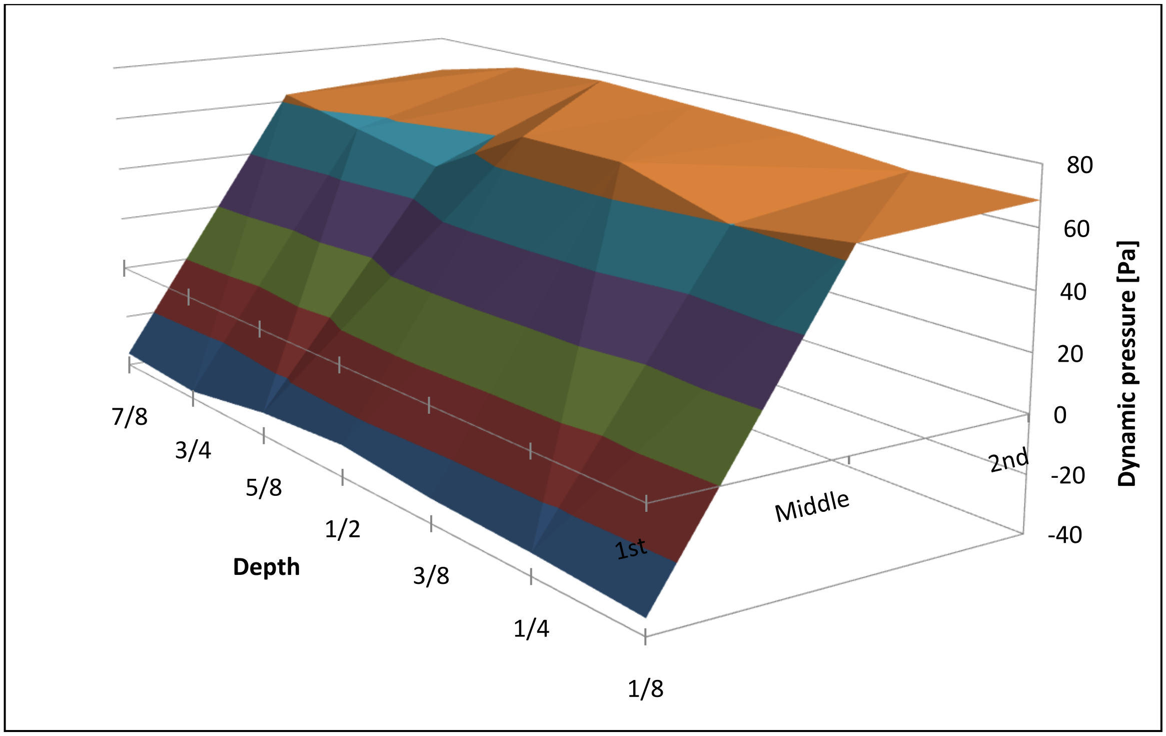
Moreover, when the average is calculated combining the values from the three widths, all the pressure difference between one width and the other becomes neglected, making the average value inconsistent in comparison with the others along the branch.

Another special case, also involving a bend, occurs in C1 just after the last 90º bend which the flow takes along the manifold. It can be seen from the results table that there is a substantial pressure loss between points A6 and C1.

Furthermore, the fluid trajectory after the bend behaves similar to the trajectory in B1, tending the fluid to go closer to the second wall. In the following table and graphic there can be seen the complete measurements in C1 and the dynamic pressure profile.

C1	1st wall			Middle			2nd wall		
	Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
1/8	-34,3	-3,9	-38,2	64,7	-3,9	60,8	68,6	-3,9	64,7
1/4	-32,4	-4,9	-37,3	59,8	-9,8	50,0	68,6	-6,9	61,8
3/8	-31,4	-4,9	-36,3	70,6	-9,8	60,8	72,6	-5,9	66,7
1/2	-28,4	-4,4	-32,9	70,6	-9,8	60,8	74,5	-4,9	69,6
5/8	-31,4	-2,0	-33,3	51,0	-3,9	47,1	76,5	-3,9	72,6
3/4	-37,3	-1,0	-38,2	56,9	-5,9	51,0	74,5	-3,9	70,6
7/8	-35,3	2,9	-32,4	62,8	-2,9	59,8	66,7	-2,9	63,7

Table 5. C1 values.



Graphic 2. C1 pressure profile.

2.1.4.2 <u>Speeds</u>

Speeds are very useful to imagine and analyse how the fluid is behaving as it is circulating in the branches.

In order to calculate the speeds in our system, the formula below was used:

$$c = \sqrt{\frac{\rho_w \cdot 2 \cdot g \cdot \Delta h}{\rho_{air} \cdot 1000}}$$

Where:

- c = speed, in [m/s]
- ρ_w = density of water, being 1000 kg/m^3
- g = gravity, being 9,80665 m/s^2
- Δh = dynamic pressure in each point, in [mm.w.c.]
- ρ_{air} = density of air, being 1,225 kg/m^3

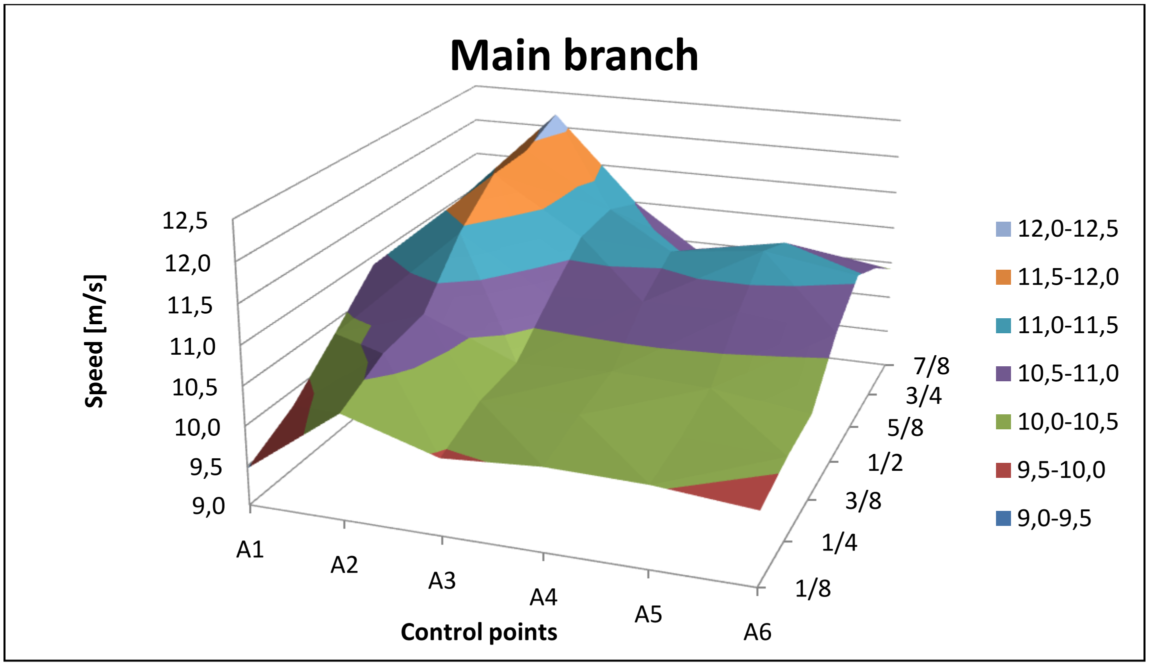
The formula was applied to every dynamic pressure value taken from the tests. In the following table there can be seen all the results:

	Speeds [m/s]							
	1/8	1/4	3/8	1/2	5/8	3/4	7/8	Average
I1	11,73	12,20	12,65	13,48	13,86	13,75	13,75	13,06
A1	9,47	9,80	10,33	10,88	10,76	10,93	10,43	10,37
A2	10,33	10,64	10,74	11,45	11,91	11,94	12,18	11,31
A3	9,96	10,20	10,25	10,74	11,18	11,22	10,98	10,65
A4	10,04	10,20	10,31	10,80	11,10	10,79	9,96	10,46
A5	10,02	10,10	10,25	10,81	11,25	11,01	10,38	10,54
A6	9,91	10,04	10,10	10,69	11,06	10,79	10,41	10,43
B1	5,80	2,53	4,13	6,31	7,97	5,93	6,53	5,60
B2	6,11	5,61	2,19	6,89	6,41	6,20	5,71	5,59
B3	6,37	5,37	4,73	5,49	5,93	5,52	5,37	5,54
O1	5,52	5,37	5,37	5,87	5,93	5,22	5,06	5,48
C1	7,34	7,23	7,80	7,70	8,07	7,16	7,16	7,49
C2	8,67	7,94	7,94	8,18	6,85	8,00	8,64	8,03
C3	8,80	7,52	7,77	7,61	7,80	7,77	8,30	7,94
O2	7,80	7,38	7,38	7,27	7,16	7,16	8,10	7,46

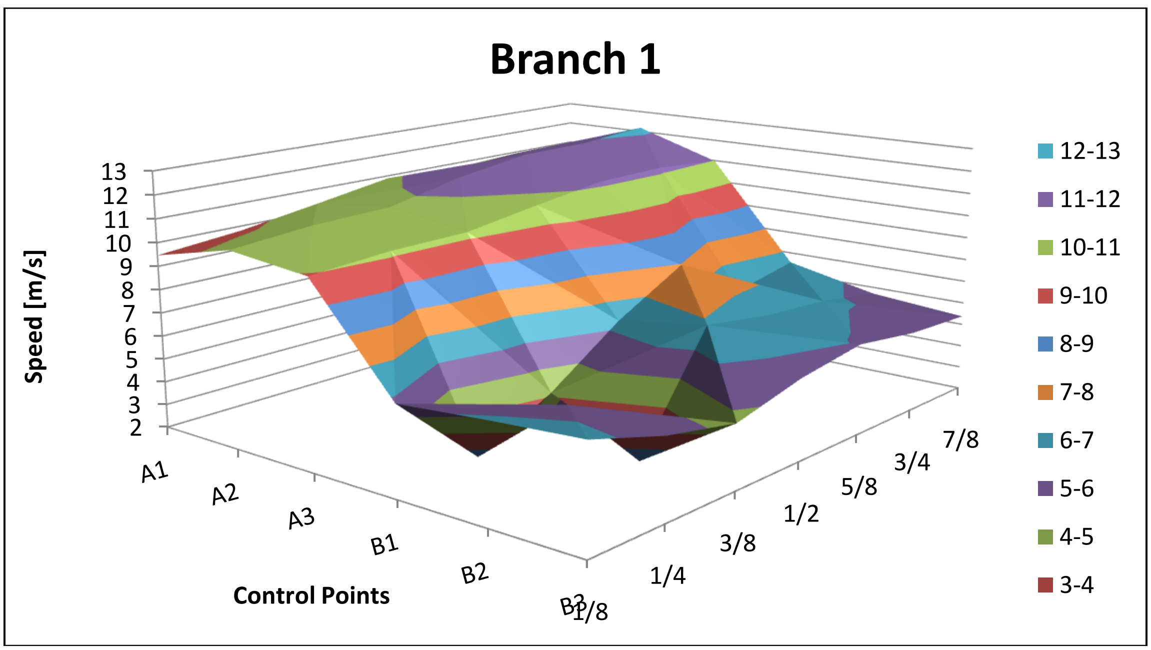
Table 6. Speed values.

Taking all the results from the calculations, some graphics were done in order to check the speed profiles in each point, being able to see if the profiles were symmetrical as the theory shows.

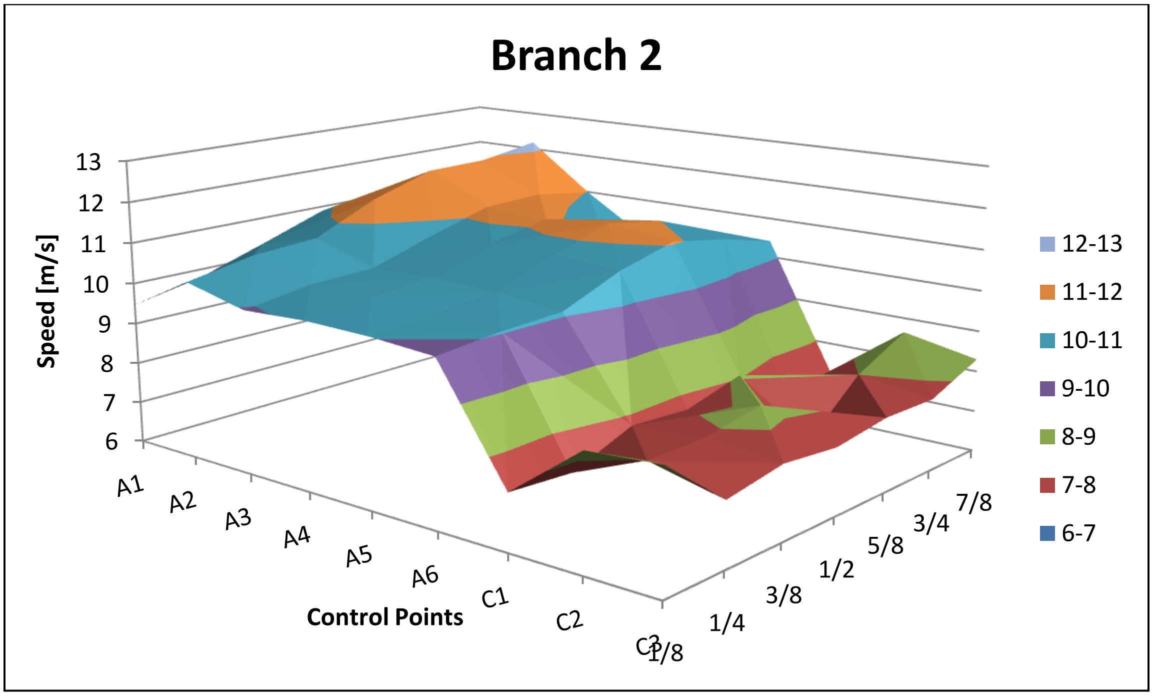
The graphics were done following the fluid trajectory along the branches: one following the main branch, and the two others following the different branches as well as the previous trajectory through the main branch. This previous path was considered in order to see the speed drop in the bends.



Graphic 3. Main branch speed profile.



Graphic 4. Branch 1 speed profile.



Graphic 5. Branch 2 speed profile.

Analysing the results and the graphics, it can be seen that the speed profiles are not as symmetrical as they should be following the theory. It can also be noticed the speed drops just after the branch bend, in points B1 and C1.

Besides, the flow behaviour taking the bends can explain the speed irregularities all along the branches 1 and 2. This case is especially apparent in the branch 1 graphic, where the profile takes some ups and downs across the depths.

2.1.4.3 Flows

Once the speeds were calculated, the last things to analyse were the flows. The flows allow checking the amount of air running through the manifold, and enable to prove if there is any air loss along the way.

The flows are calculated using the average speed in each control point and the branch section area in that point. For these calculations, the following data was used:

Branch	Width [m]	Depth [m]	Area [m^2]
Main	0,03	0,2	0,006
First	0,0195	0,2	0,0039
Second	0,0215	0,2	0,0043

Table 7. Area data.

And in the following table there can be seen the calculated values:

	Flows [m^3/s]
A1	0,0622
A2	0,0679
A3	0,0639
A4	0,0627
A5	0,0633
A6	0,0626
B1	0,0218
B2	0,0218
B3	0,0216
C1	0,0322
C2	0,0345
C3	0,0341

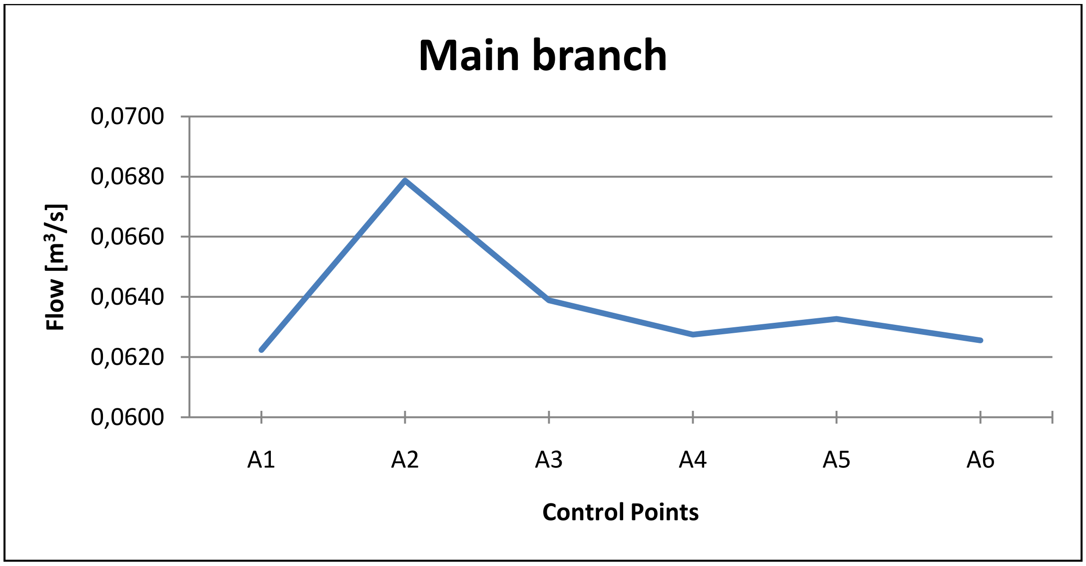
Table 8. Flow values.

In Table 8, flow values are the result from the multiplication of the speed in each point and the branch area in those points.

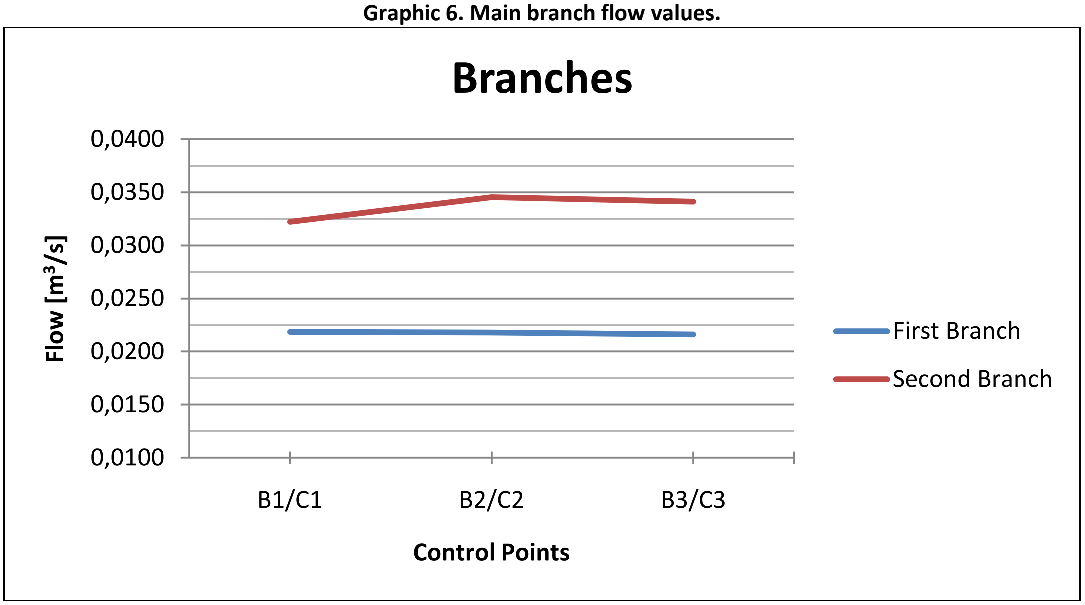
For a better comprehension of how the flows behave along the manifold, the following figure and graphic were done:

A1	A2	A3	A4	A5	A6
0,0622	0,0679	0,0639	0,0627	0,0633	0,0626
	B1	0,0218		C1	0,0322
	B2	0,0218		C2	0,0345
	B3	0,0216		C3	0,0341

Table 9. Flow values.



Graphic 6. Main branch flow values.



Graphic 7. Flow values in the branches.

From the results table it can be noticed some incongruent values, probably caused in part by measurements imprecision. First, a flow raise occurs in point A2 and then it drops in A3, staying constant then all along the main branch. Next, between points A3 and A4, where the first branch division is placed, there is no substantial flow decrease, when the first branch flow is almost one third of the main branch flow. Finally, although the value in A6 is 0,626 m^3/s, the next point in the way, C1, has only 0,0322 m^3/s, almost the half when there is no flow division in this point.

It is also important to mention that the inlet flow is 0,0622 m^3/s, while the both combined outlet flows are 0,0557 m^3/s, which means there is 10,5% flow loss all along the manifold system. From this result it is noticed that as the outlet flow is reasonably correct compared to the inlet flow, the most incongruent values inside the manifold are from A4 to A6, where they stay equal to the inlet flow and should be lower to compensate the lost flow in the first branch.

2.2 EES REPORT

2.2.1 Model Description

The system studied with EES consists of a main conduct divided in two branches. The inlet is placed in the main conduct and the outlets in the branches. The dimensions of the model are the same as the device tested in the laboratory, so that the results can be compared. Furthermore, a roughness value of 0.0001 m is assumed.

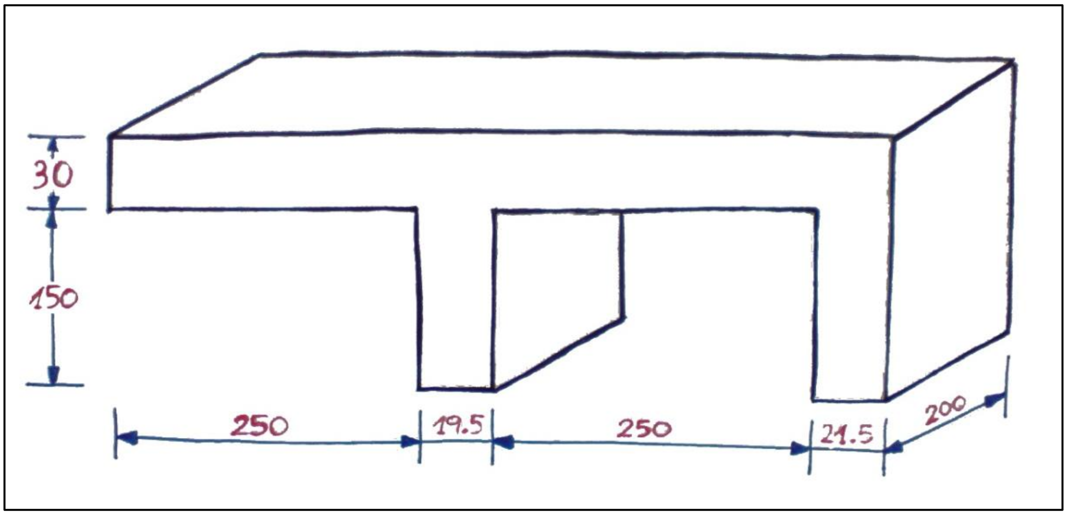


Figure 4. EES model dimensions.

2.2.2 System Description

11 states have been defined in the program as control points (see the figure below):

- State 1: 10 cm. before the inlet.
- State 2: Inlet.
- State 3: Main branch, right before the first branch.
- State 4: First branch, right after the division.
- State 5: Main branch, right after the division.
- State 6: Main branch, before the 90^0 bend.
- State 7: Second branch, after the 90^0 bend.
- State 8 and 9: Outlets.
- State 10 and 11: 10 cm. after the outlets.

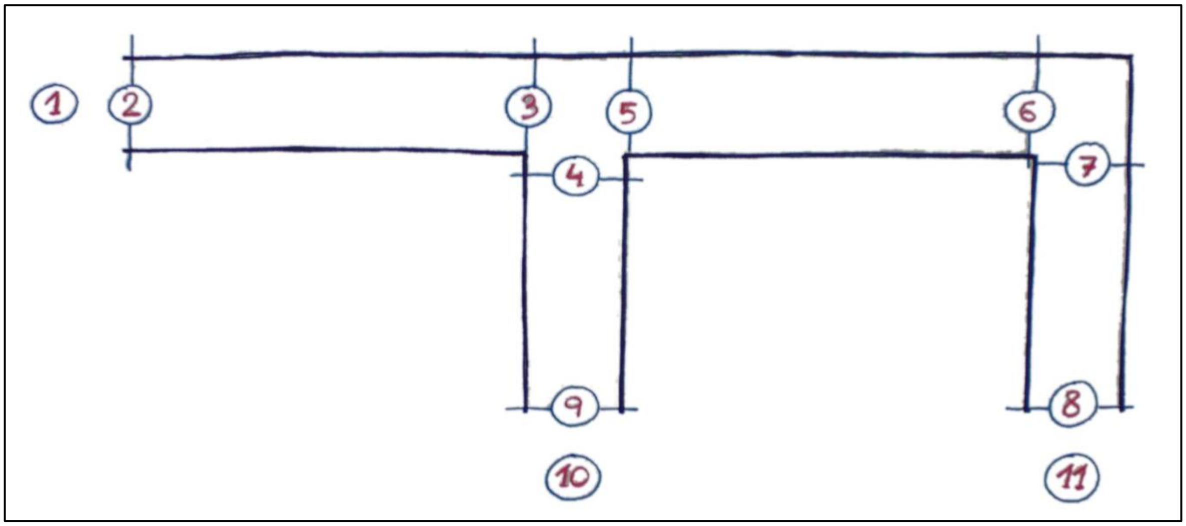


Figure 5. States defined in EES.

However, these states are only employed in EES. To avoid confusion, the equivalent states employed in the laboratory test are specified in the tables where the EES solutions are shown.

The goal of realizing this program is to obtain the values of static, dynamic and total pressure, pressure loss, zeta coefficients and speed in all the control points, as well as the outlets flow, so that we can understand the behaviour of the air inside the device and compare these parameters with the results in CFD and the laboratory.

2.2.3 Program Construction

The whole program is defined in the Equations Window (see Appendix 2) where all the equations and input data are introduced. The input data in this case have been the properties of the air (density and kinematic viscosity), the geometric values (dimensions and roughness) and the inlet flow.

The eleven states have been defined by using the formulas for pressure (pressure loss, dynamic pressure and total pressure), Reynolds number and flow:

- $Q = v \cdot A$
- $Re = u \cdot \frac{d}{v}$
- $P_t = P + P_d$
- $P_d = \frac{1}{2} \cdot \rho \cdot u^2$
- $dP_{1,2} = \zeta_{1,2} \cdot P_{d_{1,2}}$
- $P_{t_2} = P_{t_1} - dP_{1,2}$

The zeta coefficient to calculate the pressure loss between two control points in straight stretches are calculated with a function dependent on the Reynolds number and the roughness called "FUNCTION zeta_fric (Re,ksd)" (Friction factor for flow in tubes, including laminar and transient area).

To calculate the pressure loss in the intersection of the main branch with the first branch a double interpolation has been employed. The introduced data in the interpolation are the flow ratio and the area ratio, while the output data are "zeta_through" and "zeta_branch".

There is a critic point in the configuration process, state 4. In this point the speed has to be guessed because if not, there will be more variables than equations, which makes impossible to find a desirable solution. However, with the guessed speed, the pressure in state 10 (first outlet) is incorrect since it is different from zero. The way to change this is, once the number of equations is the same as the number of variables, pressure 10 has to be set equal to zero. Afterwards, pressing F2, the solution is recalculated and the correct speed in state 4 is found.

Finally, all the input data has been highlighted to facilitate the understanding of the program by someone who has not participate in its configuration.

2.2.4 Solution

All the solution provided by the program in the Solution Window has been resumed in the table below. The units are shown in SI units. The whole Solution Window can be seen in Appendix B2.

		Solution Window				
State (EES)	**State (Test)**	**Speed**	**Dynamic P.**	**Static P.**	**Total P.**	**Flow**
1	I1	1.50	1.38	134.80	136.20	0.0622
2	A1	10.37	65.83	37.42	103.20	undefined
3	A3	10.37	65.83	35.85	101.70	"
4	B1	6.83	28.54	0.76	29.30	"
5	A4	5.93	21.53	76.52	98.05	"
6	A6	5.93	21.53	75.97	97.50	"
7	C1	8.27	41.93	1.07	43.00	"
8	C3	8.27	41.93	0	41.93	"
9	B3	6.83	28.54	0	28.54	"
10	O1	0	0	0	0	0.0266
11	O2	0	0	0	0	0.0356

Table 10. Solution Windows resume.

The flow is undefined in the Equations Window from state 2 to sate 9 because it is unnecessary to calculate it in these states. The only flows which are defined in the Equations Window are: "Q_total" (state 1), "Q_1st_branch" (state 10) and "Q_2nd_branch" (state 11).

All the input and output data, as well as a sketch of the system are shown in the Diagram Window (see the figure below).

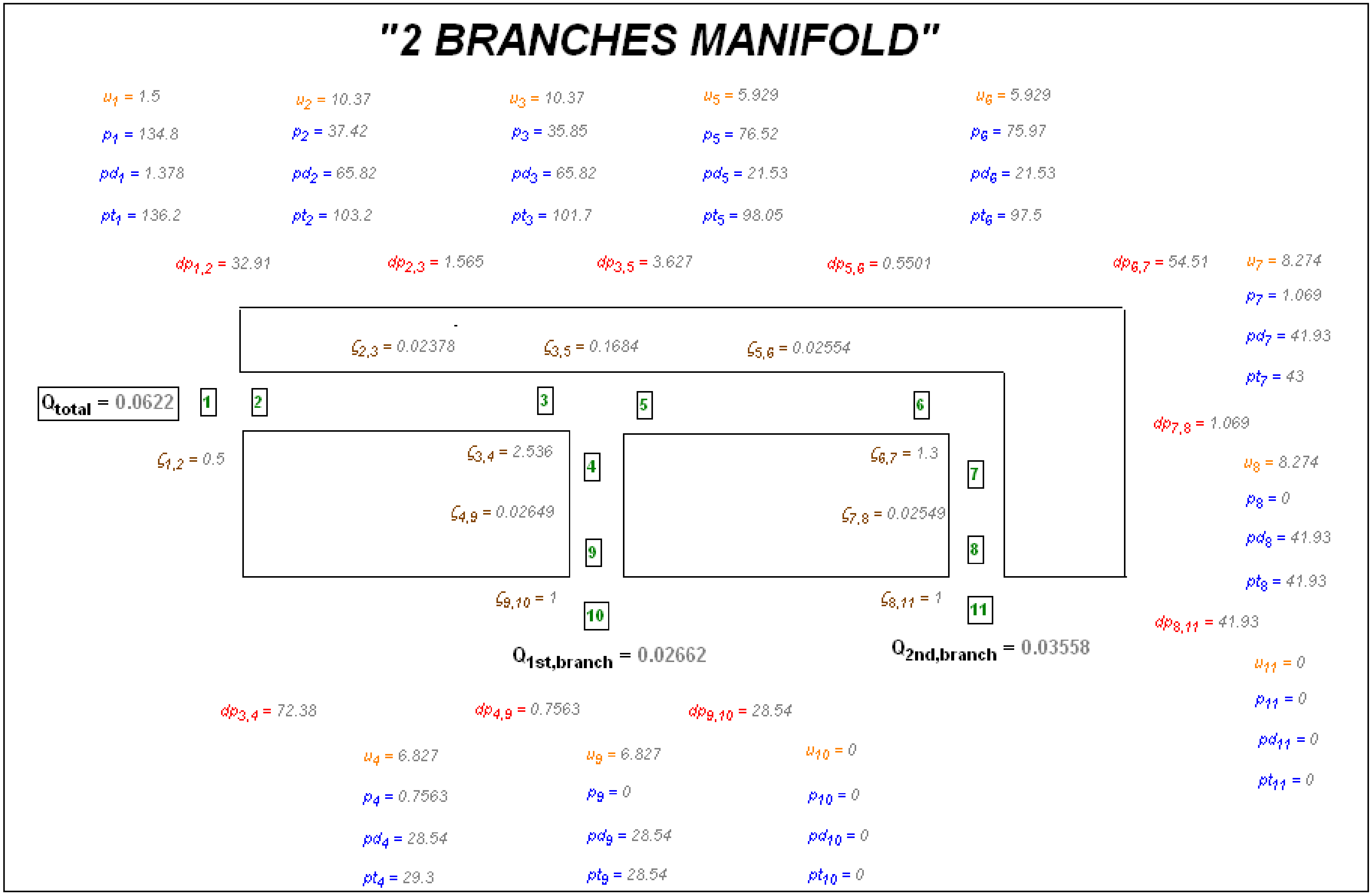
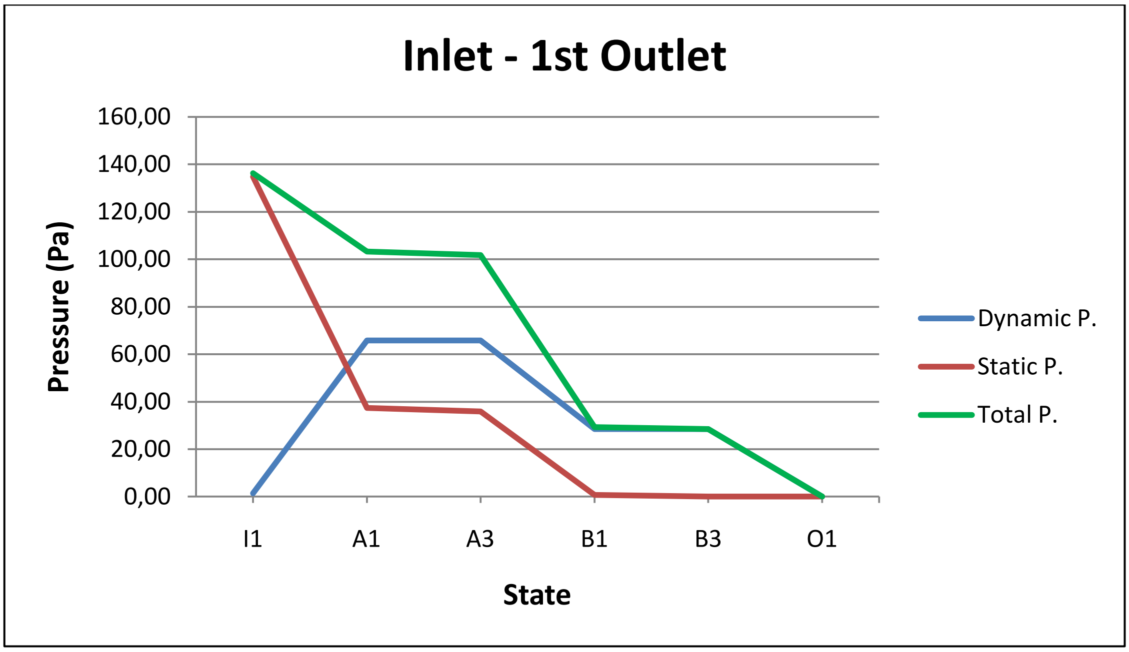


Figure 6. Diagram Window.

2.2.5 Analysis of results

2.2.5.1 Pressure



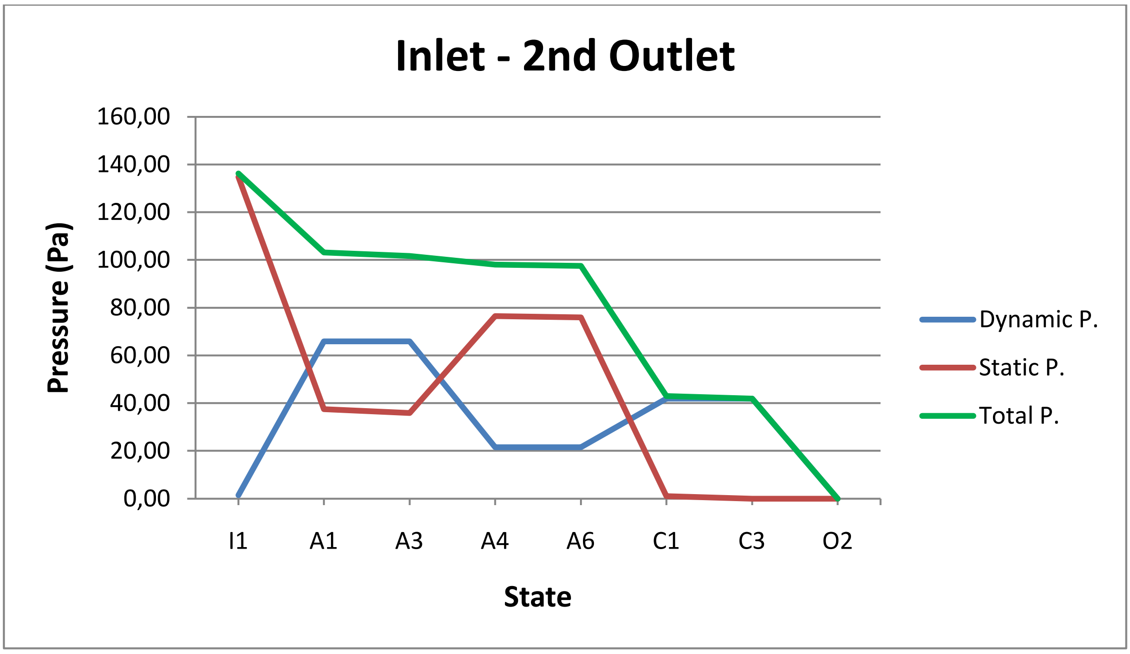
Graphic 8. Pressure values from the inlet to the 1st outlet.

As it can be appreciated in the graphic, there is an important pressure loss in the inlet because of the obvious change of geometry from the point before the inlet to the first point inside the main branch, so there is a zeta coefficient to be taken into account (ς_{inlet}= 0.5). The dynamic pressure rise is explained by the rise of velocity which involves the reduction of the area.

From A1 to A3 there is only a little pressure drop owing to the friction caused by the roughness of the device and the speed of the fluid, calculated with the function "zeta_fric (Re,ksd)" as explained before.

The point where the first branch is separated from the main branch (A3 to B1) is one of the most complex points in the system because the flow is divided, as well as the area is reduced and the direction changes. It is here that the double interpolation is used to calculate the value of "zeta_branch" and a very high value is obtained (ς= 2.536).

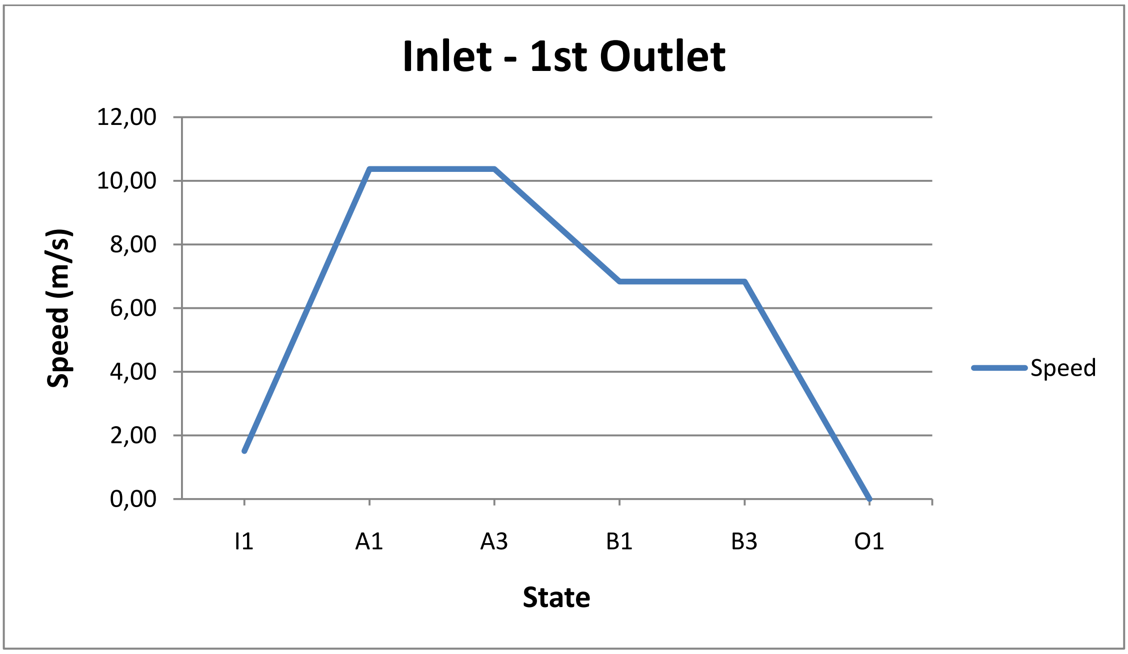
In the first branch the static pressure is very close to zero due to the influence of the environment pressure. The dynamic and total pressure drop in the outlet as expected, what is explained by the reduction of velocity as well as by the zeta coefficient in the outlet (ς_{outlet}= 1).



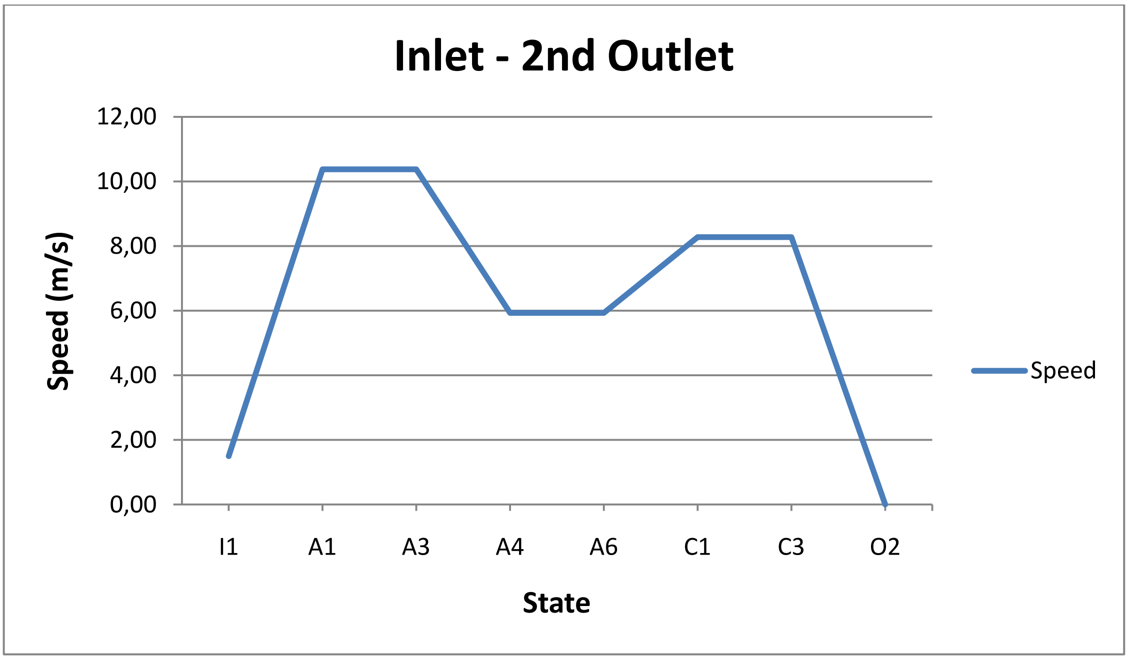
Graphic 9. Pressure values from the inlet to the 2nd outlet.

The only difference shown in this graphic with regard to the previous one is the behaviour between A3 and A4. The double interpolation had to be employed to calculate the value of "zeta_through" (ς=0.1684). As the area after the division is the same as before, the area ratio in this interpolation is equal to one. The change in the flow explains the dynamic pressure drop. The rest of pressure variations are caused by friction, the 90^0 bend in the second branch and the second outlet.

2.2.5.2 Speed



Graphic 10. Speed values form the inlet to the 1st outlet.



Graphic 11. Speed values from the inlet to the 2nd outlet.

All the changes of speed are due to area changes and flow variation. The first situation takes place in the inlet and the outlets. As the area in I1 is unknown, the speed in this point has to be guessed (1.5m/s). The speed increases when the air flow enters the device and decreases to zero in the outlets O1 and O2. As it can also be appreciated, the speed varies in the branches in comparison to the main branch. The width of branches (19.5mm. and 21.5mm.) is less than the width of the main branch (30mm.), so the speed has to increase in order to maintain a constant flow. Needless to say that in B1 (first branch), as the flow is divided; the speed value drops despite the decrease in area.

2.2.5.3 Flow

The inlet flow, which is obtained in the laboratory tests (0.0622m^3/s), is introduced as an input value in the Equation Window. After reaching the control point A3 the flow is divided and a 42.8% (0.02662m^3/s) goes through the first branch while a 57.2% (0.03558m^3/s) goes through the second one.

There is a really interesting tool in EES, which enable the user to know the relative influence of certain variables on another variable. This tool, called Uncertainty Propagation, can be found in the menu Calculate or when pressing F6.

In the case of our study it has been very useful in order to know the factors which determine the flow division. To do that, the flow in both branches has been selected as the variables to be calculated and the dimensions of the branches, as well as the total flow have been set as measured variables. In the figure 7 the absolute and relative uncertainties entered in the program are shown.

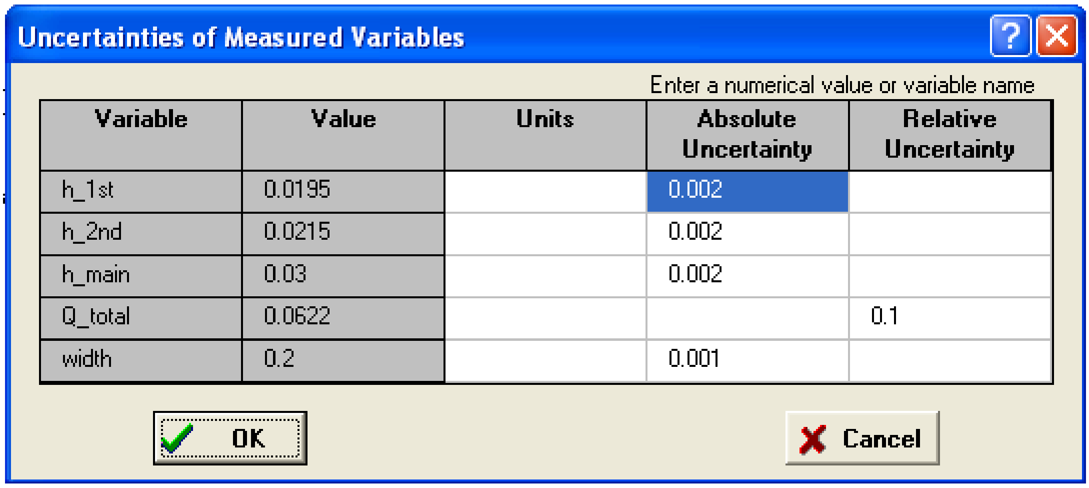


Figure 7. Uncertainty definition for the flows.

As it can be seen in the table below, the flow in each one of the branches is very dependent on the inlet flow. However, there is a point which could be surprising, the second branch dimensions; because no one would expect that the flow in the first branch is more dependent on the width of the second branch than on the width of the first one. Finally, another fact that has to be remarked is the little influence of the width of the main branch on the branches flows.

Variable	Variable ± uncertainty	% of uncertainty
Q 1st branch	0.02662 ± 0.003557	
1st branch width	0.0195 ± 0.002	5.31%
2nd branch width	0.0215 ± 0.002	32.14%
Main branch width	0.03 ± 0.002	6.58%
Q total	0.0622 ± 0.00622	55.97%
Device width	0.2 ± 0.001	0.00%
Q 2nd branch	0.03558 ± 0.00427	
1st branch width	0.0195 ± 0.002	3.68%
2nd branch width	0.0215 ± 0.002	22.30%
Main branch width	0.03 ± 0.002	4.57%
Q total	0.0622 ± 0.00622	69.45%
Device width	0.2 ± 0.001	0.00%

Table 11. Uncertainty results.

2.3 CFD REPORT

2.3.1 Model Description

The model consists of two cubic solid blocks (1) (2) separated a certain distance, an L-shaped solid (3) covering both and two plastic flat pieces (5) (6) on both sides of the blocks to create a controlled volume where the flow will go through. Furthermore, due to the requirement of the SolidWorks' Flow Simulation Studio of a closed volume to be controlled, there are three rectangular lids (7) (8) (9), each one placed in an inlet or outlets to get a closed conduit.

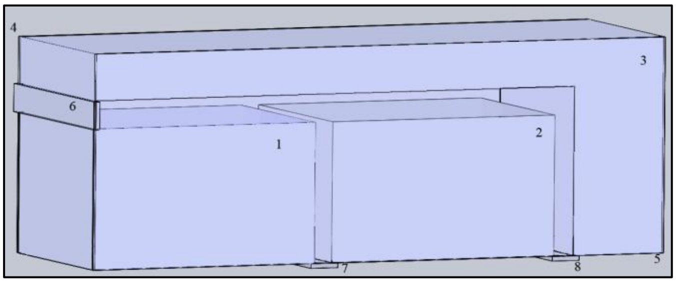


Figure 8. SolidWorks model overview.

Figure 8 shows how the model recreation looks like in SolidWorks. This model is the one the program will work with. The gap between blocks 1 and 2 is 19,5 mm and 21,5 mm for the distance between 2 and the lower part of 3. The main branch, which is the hole between the upper part of 3 and the two cubic blocks 1 and 2, is 30 mm thick.

As SolidWorks' Flow Simulation Studio works with a conduit and its characteristics, it is not very relevant the size of the blocks but the conduit dimensions must be the ones described before. The main branch is 200 mm wide and 541 mm long. At 250 mm from the inlet it is derived the first branch, which is also 200 mm wide and 150 mm long until the first outlet. At the end of the main branch begins the second branch, which is, like the first one, 200 mm wide and 150 mm long.

It is important that the pieces covering the inlet and outlets are big enough to cover the whole area of the branches so the program can determine a closed volume.

2.3.2 System description

The objective of the CFD test is to get enough results that can be compared with the ones taken in the lab. To do so it is necessary to set several control points along the conduit. As shown in figure 9, the points to be controlled are at the inlet and outlets and just before and after the deviations. From all these control points will be taken the

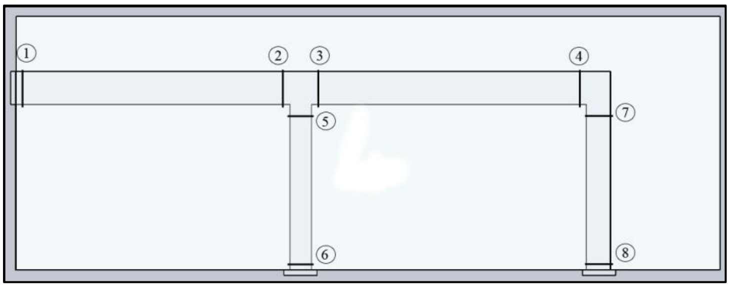


Figure 9. Control points.

pressures and flow velocities, as well as other data such as the fluid temperature, density and mass flow rate.

The lids placed at the inlet and outlets will be used as control surfaces, from where we will extract the volume flow rates in each one of the outlets.

2.3.3 Program Construction

Once the manifold assembly is saved and ready to work with, it is necessary to set up a new Flow Simulation project. It is used the Flow Simulation wizard to do so, and there is where the general system characteristics are introduced. The first thing to do when the project name is introduced is to choose the system units. The SI (International system) units have been chosen, so all the data introduced to the program will be according to these units. Then, as the flow studied circulates inside the model, an internal analysis type is selected and the X axis as the reference one. No special physical features are needed to be configured for this project. Now is time to choose the fluid which the program is going to work with, which is air for this test. The default walls thermal conditions are not modified so are left as adiabatic walls, but the roughness must be changed to 0,0001 micrometer. When the initial conditions are confirmed, the result resolution can be modified although in the test being carried out will not be changed.

When this previous set up is done, the Flow Simulation Studio creates a bounding box around the model, this means the interior flow has been found, it is called "Computational Domain". Now is time to enter the inlet flow conditions by selecting the inlet lid face touching the model and configuring its boundary conditions. It has to be set a uniform inlet volume flow, normal to the selected face, of 0,0622 m^3/s and a fluid temperature of 289,65 K (16,5 ºC) so the flow and temperature values are the same than the ones in the laboratory when the test was done. In the other two outlet lids the boundary conditions have to be set as pressure openings at atmospheric pressure at the same temperature than the inlet fluid.

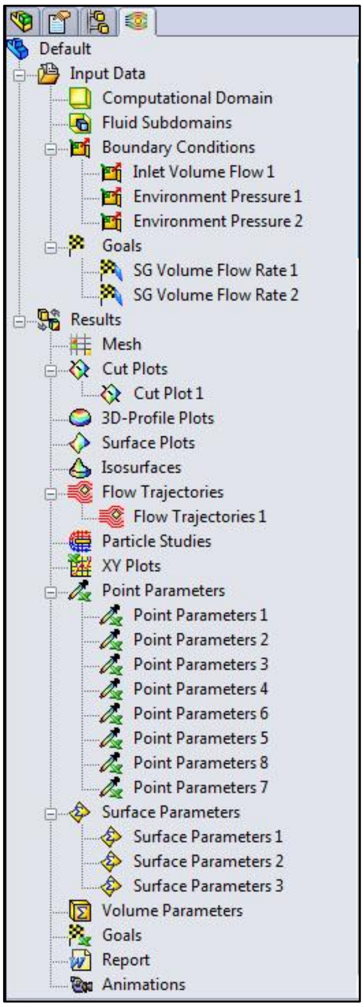


Figure 10. Flow simulation configuration tree.

To continue with the program construction some goals need to be set. In this case is wanted to know the amount of flow going through the branches, concretely the volume flow in each one of the outlets. After choosing the goals the program is run and the first results are obtained.

2.3.4 Analysis of results

When the Simulation has been run the results are already able to work with. Some of the results are given directly by the program, but some other valuable data, which is needed to make a good comparison with the lab test data, has to be previously set to be shown. This is required to know the values in the control points, which are actually control sections if we think in a 3D model. SolidWorks' Flow Simulation doesn't have a tool to control an area inside the controlled volume, so it is necessary to create a point mesh, as shown in figure 11, and later make an average with all the point values obtained.

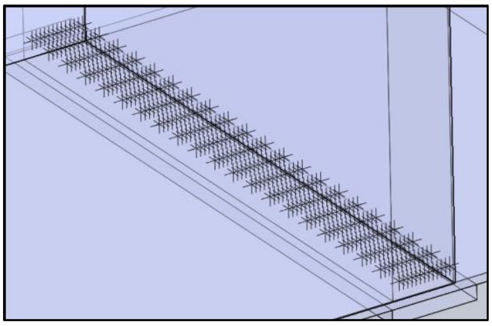


Figure 11. Point mesh.

There are some different ways to see results, and SolidWorks Flow Simulation Studio gives us the opportunity to do so. One of this ways is with the result plots. The cut plot shows a cut view of the flow circulating through the conduit. In figure 12 is represented the flow pressure changes in the fluid in the centre profile of the model.

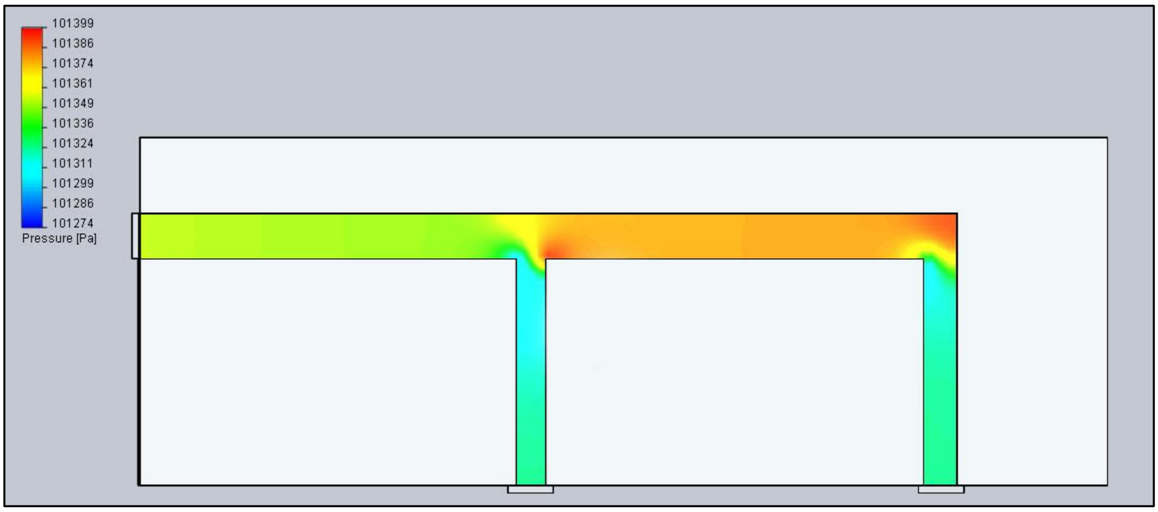


Figure 12. Pressure cut plot.

It can be clearly seen how the pressure changes in every bend. At the inlet the fluid has a pressure higher than the atmospheric (101325 Pa) due to the speed given by the fan. After the first division the fluid in the main branch experiences a pressure increase, and in both outlet branches the pressure decreases considerably influenced by the atmospheric pressure outside the manifold.

The parameter represented on the model can be easily changed, and it allows having an idea of the speed variation throughout the model. In figure 13 can be seen how the speed of the fluid is decreasing after the first division, as some fluid is deviated to the first branch and the area remains constant for the entire main branch. Still in the main, can also be appreciated the lower speed of the fluid circulating close to the walls.

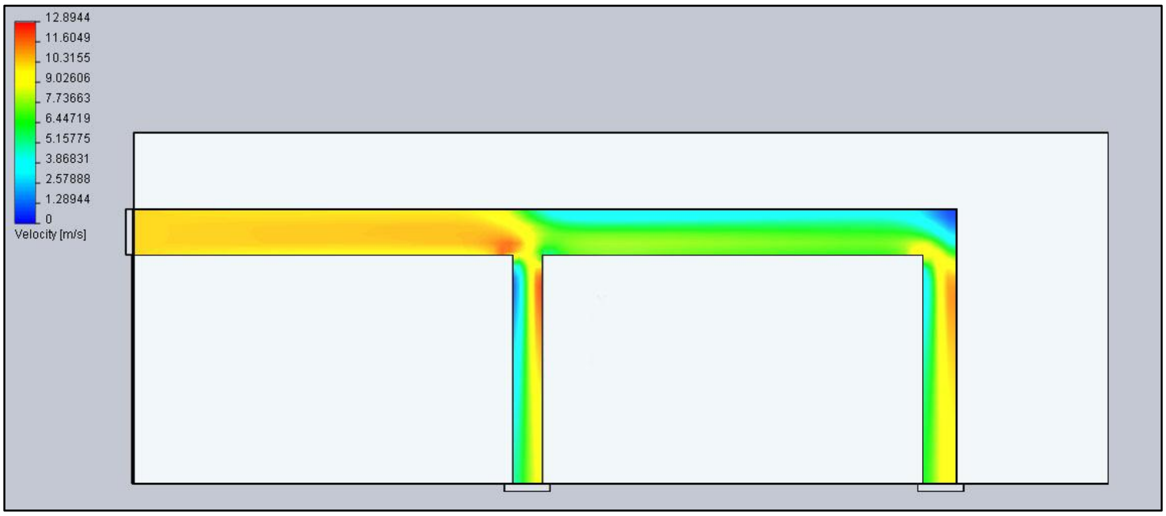


Figure 13. Velocity cut plot.

The speed in the outlet branches is quite higher on the right walls than on the left ones because of the fluid trajectory along the curve, most of it tends to go to the outer side of the channel so a biggest part of fluid has to circulate in the same area, and therefore the speed has to increase.

The results in the following table are going to be analysed and compared to the results taken with EES and the laboratory testing. SolidWorks' Flow Simulation Studio gives some other data as a result of the calculations made for the manifold model studied, such as Temperature variation, density, shear stress in every point, mass flow rate, etc.

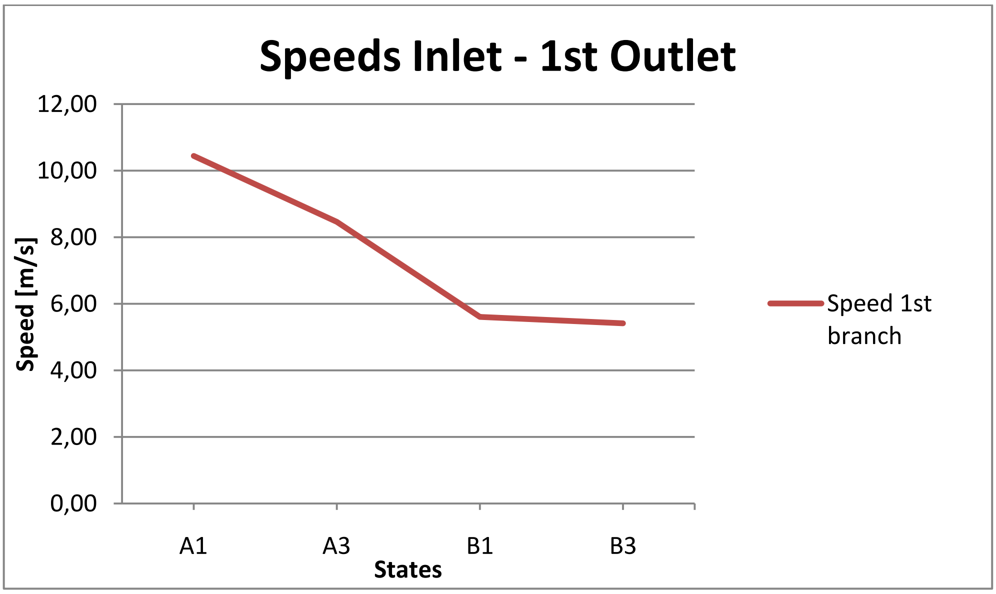
These data comes from the point parameter averages and the surface parameters. Although the average values are not exact at all, due to the limited number of points introduced in the control areas of the model, they can be considered very reliable.

		Solution Window				
Test State	**CFD State**	**Speed**	**Dynamic P.**	**Static P.**	**Total P.**	**Volume Flow [m3/s]**
I1	**I1**	--	--	--	--	0,0622
A1	**1**	10,45	66,82	27,44	94,26	--
A3	**2**	8,46	43,85	20,60	64,45	--
A4	**3**	4,98	15,16	56,15	71,31	--
A6	**4**	4,79	14,06	49,81	63,87	--
B1	**5**	5,61	19,28	5,13	24,41	--
B3	**6**	5,41	17,93	-0,25	17,69	--
O1	**O1**	--	--	--	--	0,0276
C1	**7**	6,88	28,99	8,99	37,98	--
C3	**8**	6,76	27,98	-0,11	27,87	--
O2	**O2**	--	--	--	--	0,0346

Table 12. SolidWorks' Flow Simulation results.

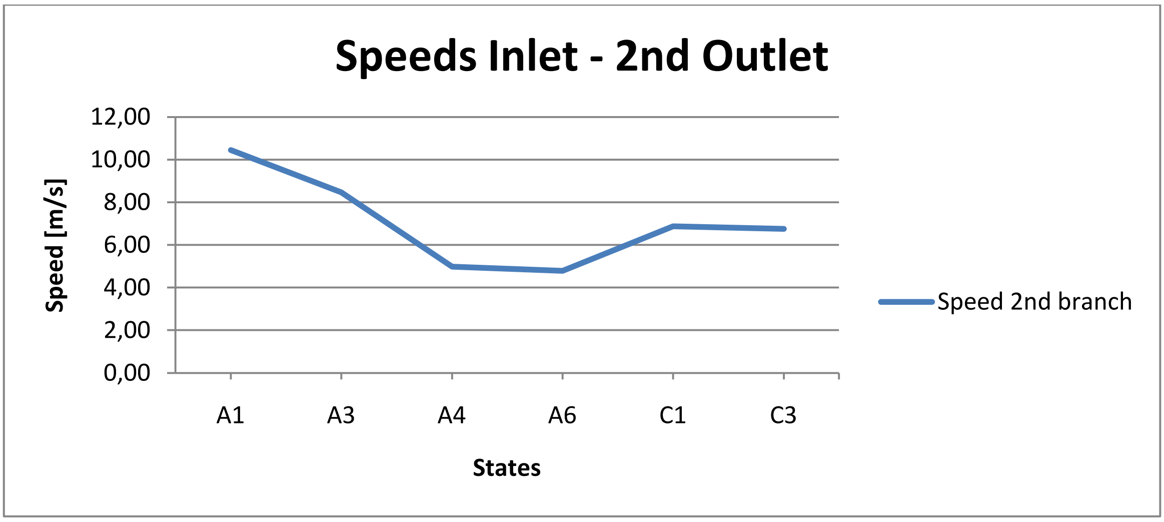
The empty boxes in the table, where the inlet and outlets pressure values should go, are not filled because these control points are out of the computational domain. Anyway it can be guessed that the static pressure would be equal to the atmospheric pressure and the dynamic pressure would be zero. As expected, the volume flow values, given by SolidWorks, show that the flow at the outlets equals the flow at the inlet, meaning there is no pressure loss in the conduit.

The following graphics represent the pressure and speed values for each of the two trajectories followed for the fluid molecules from the inlet to one of the two outlets of the model.



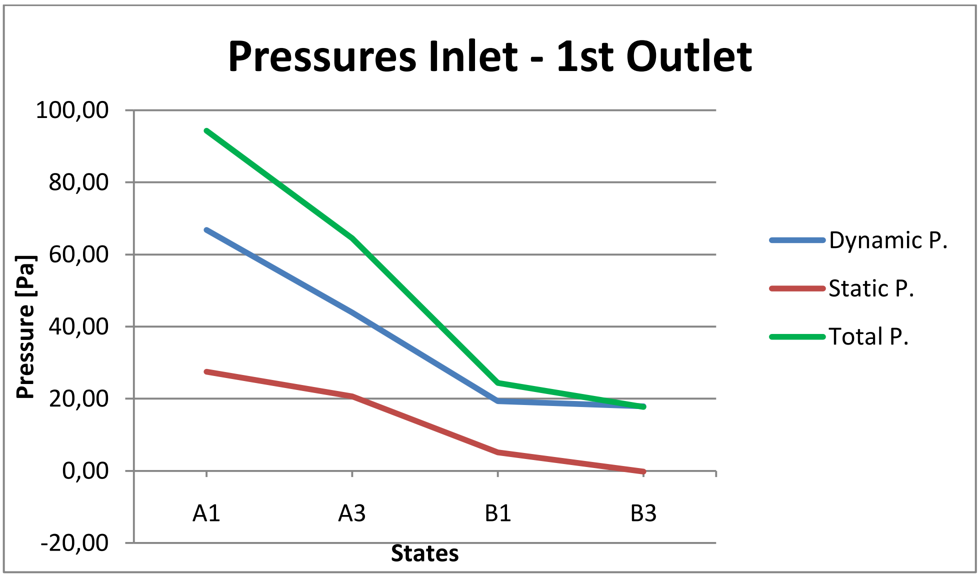
Graphic 12. Fluid speed between the inlet and the first outlet.

In graphic 12, where the first trajectory fluid speeds are represented, the speed decreases considerably when the flow leaves the main branch and reaches the outlet branch, between control points A3 and B1. This is due to the fact that, although the area is reduced a third of its size and it should mean a speed increase, the flow is also reduced around a fifty percent and makes the fluid circulate slower. Moreover, it can also be appreciated a speed decrease between A1 and A3 and between B1 and B3, due to the wall friction and the pressure losses along the branches.



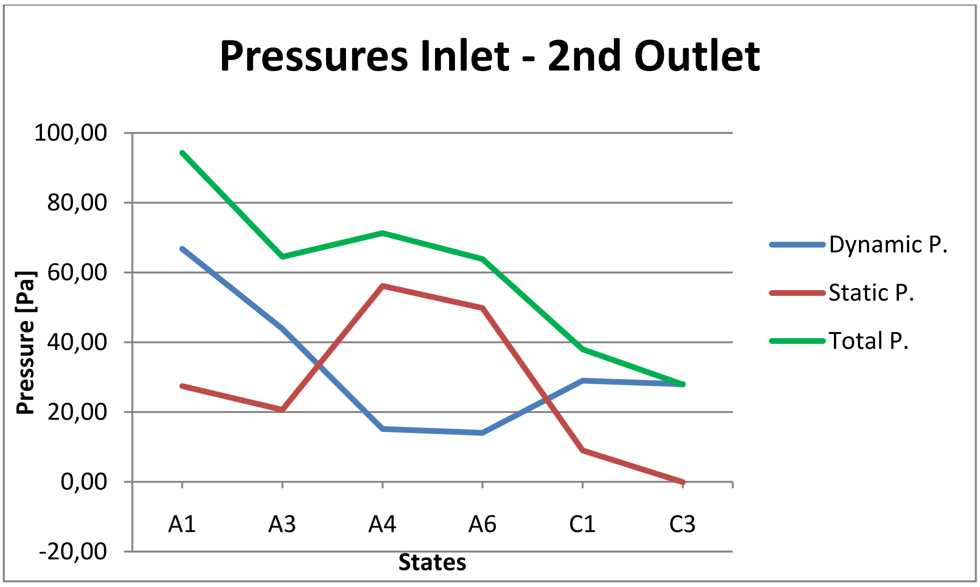
Graphic 13. Fluid speed between the inlet and the second outlet.

Graphic 13 show the speed values along the second fluid trajectory. In this case it can be seen how the speed decreases after the fluid is deviated through the first branch, between points A3 and A4, when the flow is reduced at its half and the area remains constant. Furthermore, when the flow turns at the second branch, the area is reduced and there is no flow decrease, and that is why the speed rises again between A6 and C1.



Graphic 14. Fluid pressure between the inlet and the first outlet.

Regarding to the pressures, graphic 14 shows the variation of the dynamic and static pressures as well as the total pressure. Between the control points A3 and B1 there is a big decrease due to the area reduction after the first deviation, which also means a speed decrease. At the outlet B3, the static pressure tends to zero, as the atmospheric pressure is.



Graphic 15. Fluid pressure between the inlet and the second outlet.

Finally, graphic 15 show the pressure variations from the inlet to the second outlet. After the first division but still in the main branch the dynamic pressure decreases considerably due to the flow decrease in an unchanged section area. After the second branch deviation the pressure increases while the area is reduced and the flow is maintained constant.

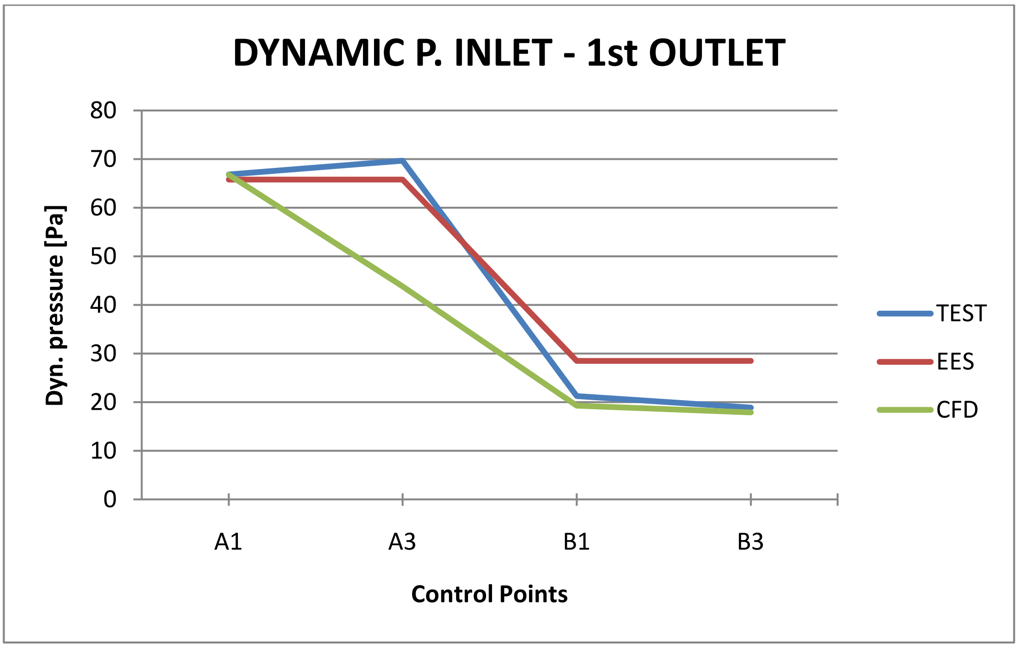
2.4 WIDE BRANCHES MODEL CONCLUSIONS

In the following graphics it is shown a comparison between the results of the three different study methods employed. It is assumed that the test values representation might defer from the other two computational methods, as they both are more accurate in the internal flow analysis. Furthermore, the represented test values are averages from a few measurements taken in the laboratory, and it can lead to some inaccuracies.

2.4.1 Pressures

The high dynamic pressure value in the inlet control point A1 is due to the introduction of an amount of air (0.0622 m^3/s) into a small section area (60 cm^2). At A3 the Test and EES results show that the dynamic pressure keeps almost constant, while CFD shows an unexpected decrease. This fact could be the result of placing the control area too close to the first deviation branch, and then the values would be influenced by the lower flow speed after the branch.

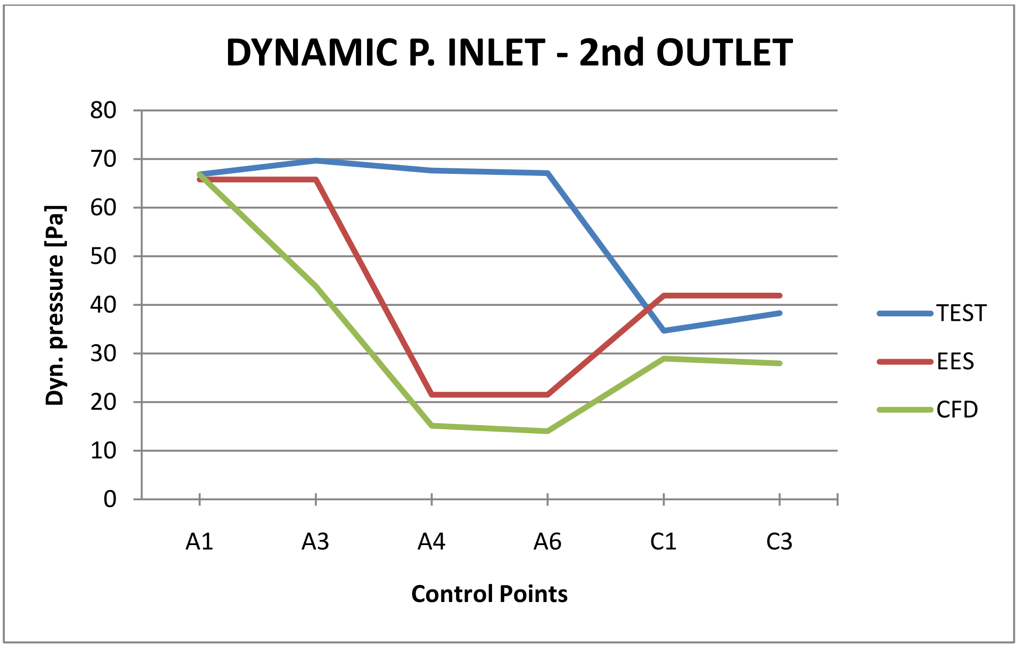
The first control point values in the branch (B1) show the dynamic pressure drop as a result of the flow division. At this point the three results are fairly similar. At the first branch outlet the pressure remains almost constant as the flow and area are invariant.



Graphic 16. Dynamic pressure from the inlet to the first outlet.

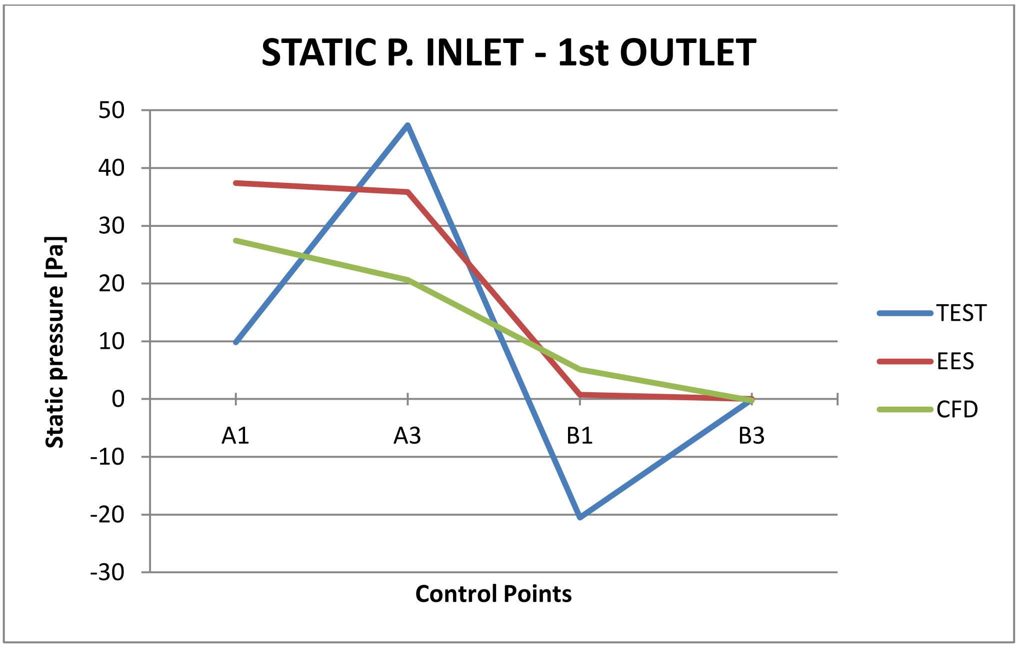
In graphic 17 the A1 and A3 control points are the same described in graphic 16. At point A4 the dynamic pressure experiences a big decrease due to the flow reduction although the area stays invariant. However, the test values do not show this fact because the flow after the first division seems to remain constant. Probably the values taken during the test for the

A4, A5 and A6 control points were mistaken and this makes the averages go wrong. After the second bend, the dynamic pressure rises again (at least in the EES and the CFD lines) while the area has been reduced.



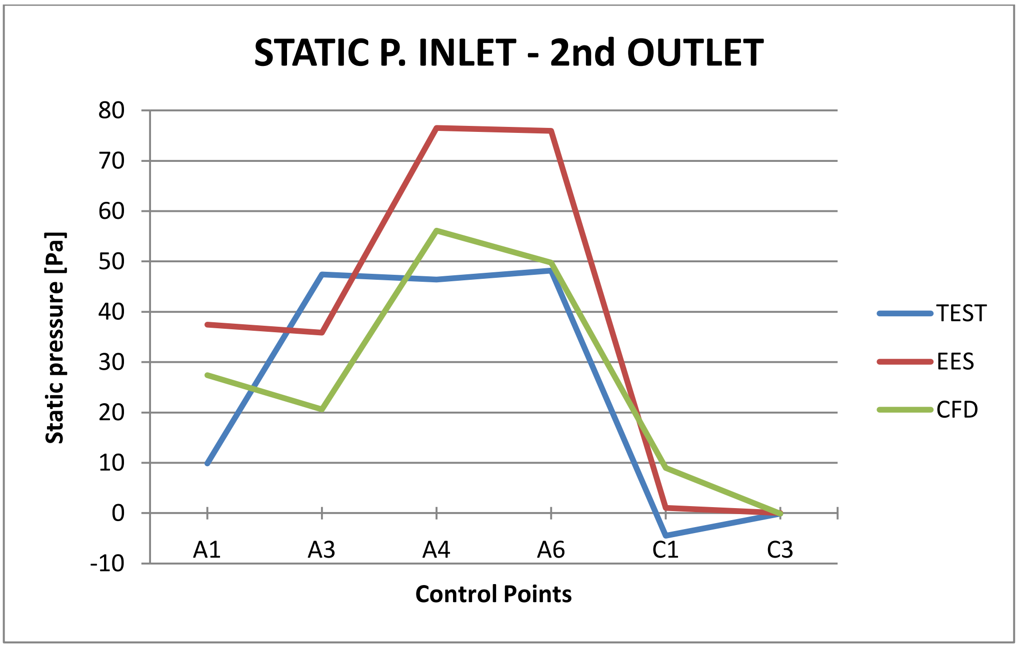
Graphic 17. Dynamic pressure from the inlet to the second outlet.

In graphic 18 it can be seen how the tendency lines are quite similar, although the test line is not as low as the other two are. The static pressure at the main branch is higher due to the injection of air into a certain volume. When the air reaches the first outlet branch the static pressure becomes influenced by the atmospheric pressure and it decreases until zero. The test value at B1 shows a negative static pressure; it can be explained as a lack of data taken in that control point, as the flow trajectory through the outlet branch makes that at the interior wall the static pressure takes negative values. If the values taken in B1 are nearer to the interior wall than to the exterior, the average value is distorted.



Graphic 18. Static pressure from the inlet to the first outlet.

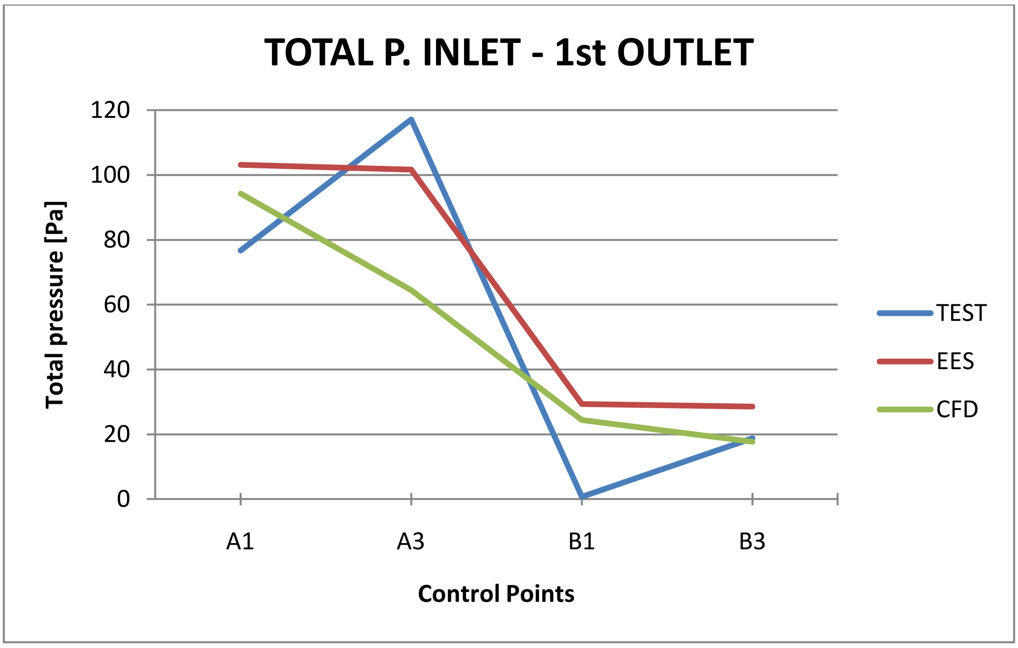
Points A1 and A3 are the same than the ones in graphic 18. At point A4, still in the main branch and after a flow decrease, the static pressure is raised as Bernoulli's equation[1] says when the velocity drops the pressure has to increase. After the bend, the static pressure is affected by the atmospheric pressure at the outlet and it tends to zero again.



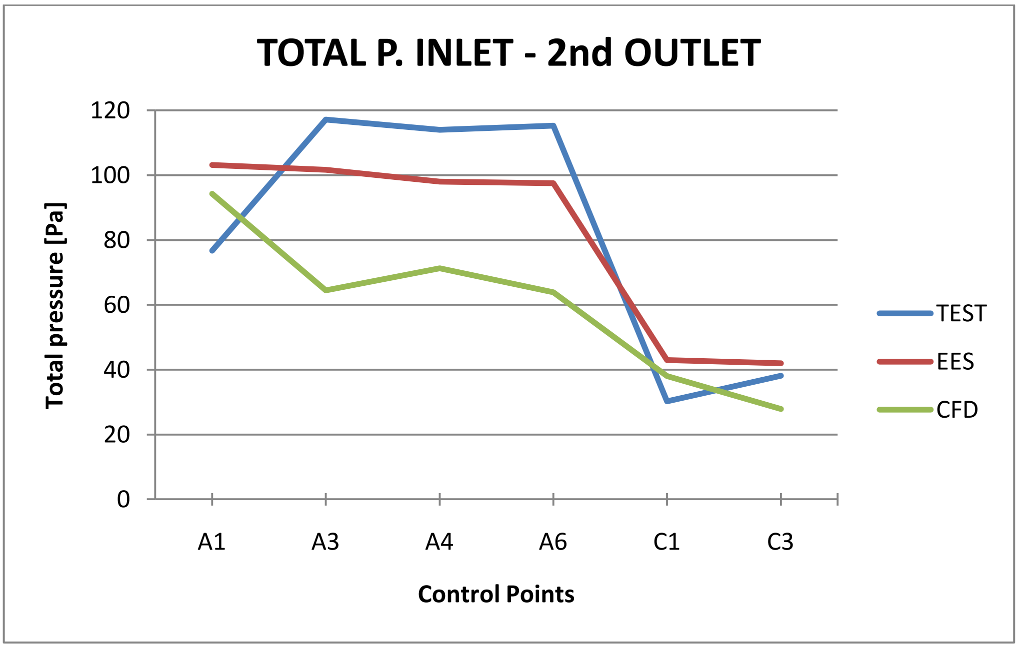
Graphic 19. Static pressure from the inlet to the second outlet.

[1] Bernoulli's equiation: $\frac{1}{2}\rho v^2 + \rho g z + p = \text{constant}$

The following graphics, showing the total pressures for the two trajectories in the manifold, are a sum of the dynamic and static pressures for each of the three methods.



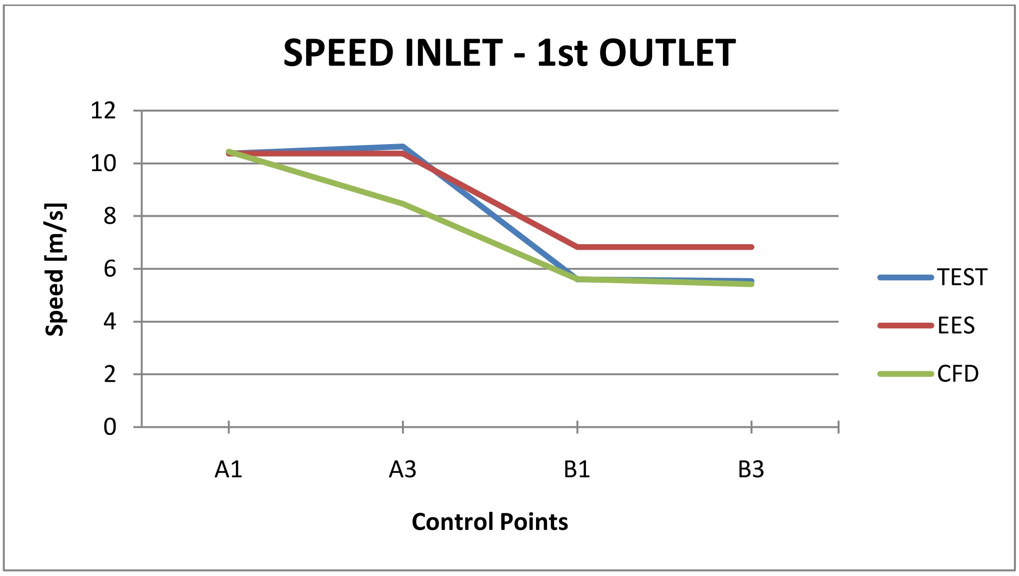
Graphic 20. Total pressure from the inlet to the first outlet.



Graphic 21. Total pressure from the inlet to the second outlet.

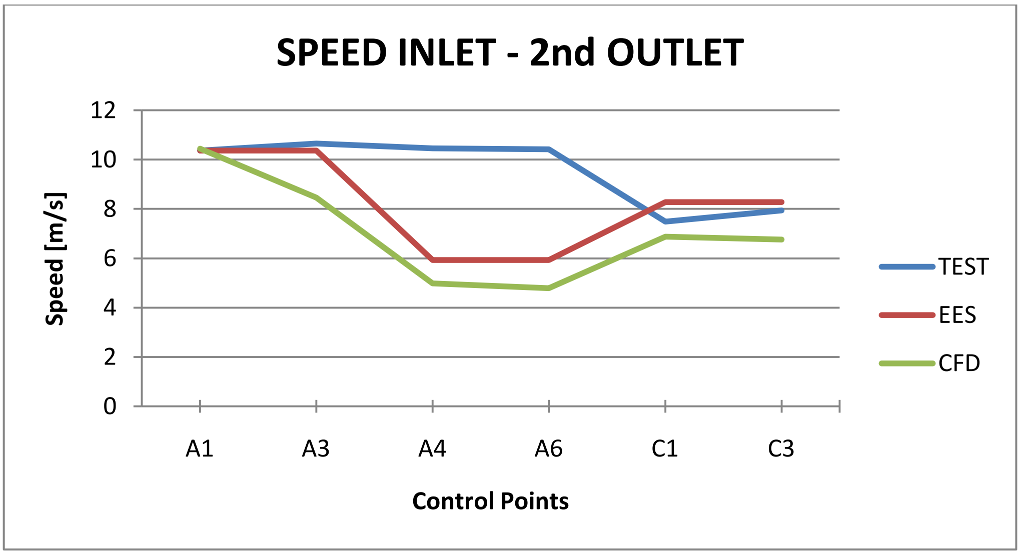
2.4.2 Speeds

As in the graphic 16, where the dynamic pressure is shown, graphic 22 shows that the CFD value for the point A3 is influenced by the speed in the first outlet branch. Nevertheless, the values for the three methods are very alike in all the other control points and the tendencies are very close. The speed at the inlet is the one given by the fan and it does not decrease along the main branch. Once the flow reaches the outlet branch the speed is decreased due to the flow reduction despite the area has been decreased.



Graphic 22. Flow speed from the inlet to the first outlet.

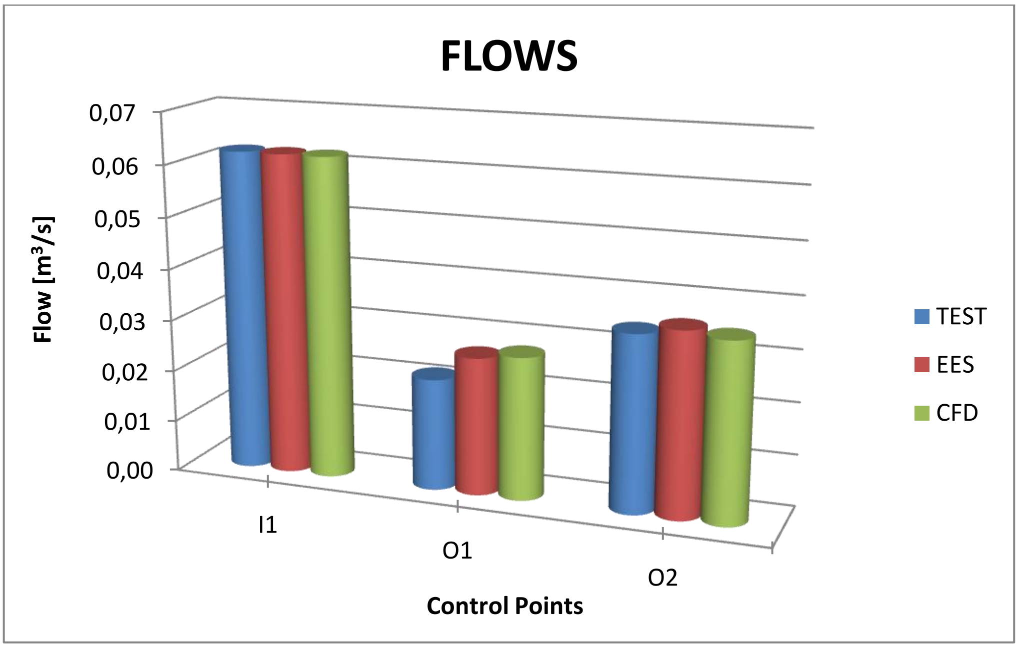
The next graphic shows how the speed is reduced when almost the half of the flow is deviated through the first outlet branch. At the second branch, the speed rises again due to the area reduction. As the speed values are calculated from the dynamic pressure values, the tendency line for the test does not decrease between A4 and A6.



Graphic 23. Flow speed from the inlet to the second outlet.

2.4.3 Flows

The inlet flow value is obtained by testing in the laboratory and it is introduced as an input to the EES and CFD software. The volume flow values for the first outlet are slightly lower than at the second one. This could be a result of the difference between the outlet branches widths. Both EES and CFD sum of the outlet flows equal the inlet flow. Regarding the test, there seem to be a flow loss around the 10%. These are quite acceptable losses considering the possible leaks and the measurement instruments reading error.



Graphic 24. Volume flow rates in the inlet and the outlets.

3 NARROW BRANCHES MODEL

3.1 TEST REPORT

Test Date: May 4th, 2010

Test Engineers: Pablo Andrés, Ricard Esteve, Jaume Llopart

3.1.1 Model Description

This test was made using a similar model than the previous one, changing the branches widths to 5 mm. The model was built using the same materials, wood blocks and plastic pieces on the sides making the union of all the blocks. The following figure shows the new geometry, in mm (figure 14).

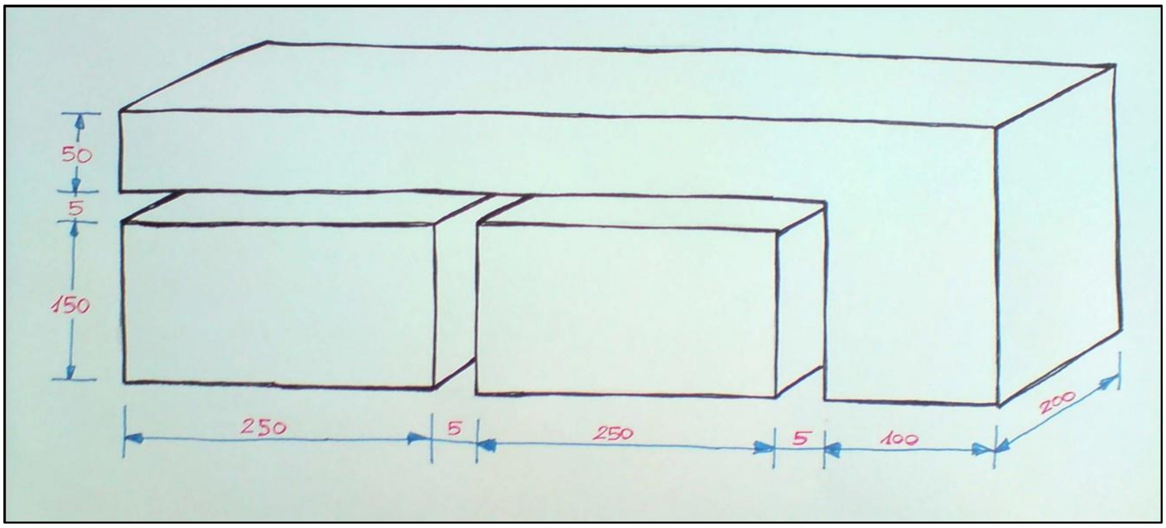


Figure 14. Model overview.

On one side there were made the holes in the plastic piece to set the control points. These control points are distributed in the following way: two in the main branch -one before the division and one after it-, and one in each secondary branch. In the following figure it is shown exactly where these points are (figure 2).

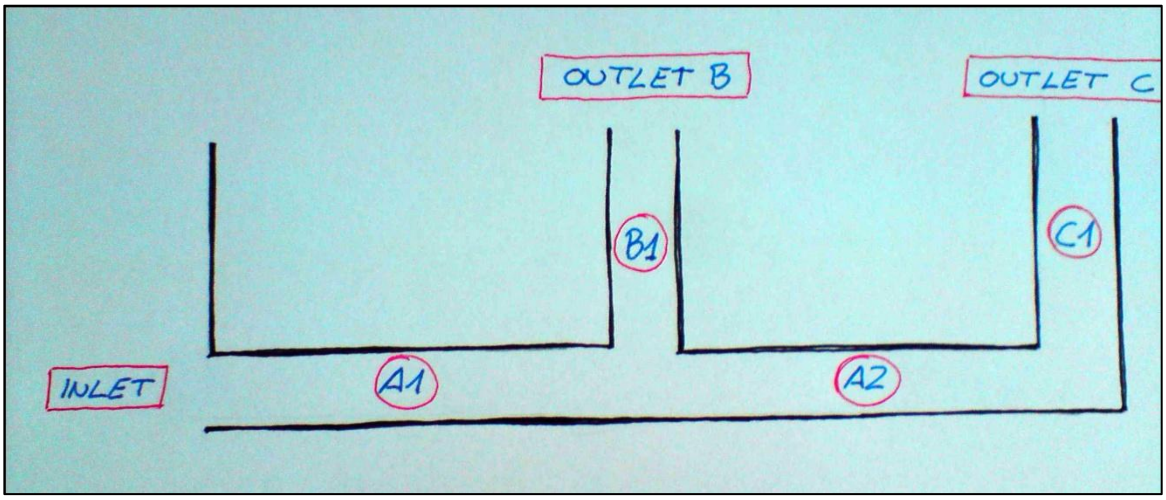


Figure 15. Holes situation.

In this new model it was used a different fan, with a two-phase electric motor which gives a rotational speed up to 16 000 rev/min at the shaft. This fan provides the air flow through a flexible plastic tube connected to the model. This fan is smaller than the other and it does not get the same power either, but as the conduits are also smaller the necessary flows were reached.

Just like the previous model, in this new one the diffusion plate was also used as a distributor of the air just before the inlet (figure 16).

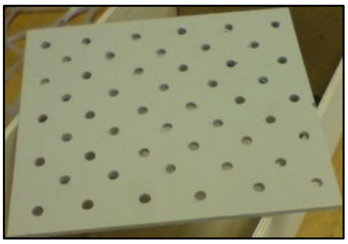


Figure 16. Diffusion plate

3.1.2 Test Description

Due to the narrowness of the branches, neither the Pitot tube nor the hot wire system could be used as measurers. The Pitot tube would distort too much the air flow, and the hot wire system did not fit inside the branches as its diameter was too wide.

Instead of these systems, the pressure values for each of the four control points are measured with a system consisting of a 0.5g lead ball hanging from a thin thread which moves a certain distance depending on the dynamic pressure. The thread is placed in each control point and its length allows the ball to be situated exactly in the middle of the branch. When the air is running along the manifold the ball is pushed and the measured displacement gives a relation to the dynamic pressure in that point.

Due to the system nature, it could only be done one measurement in each point, exactly in the middle of the branch, as it was not reliable measuring near the branch edges. Moreover, the ball was expected to move up to 8-9 cm, so there could only be set one control point in each branch as it was the only way to get the ball movement without flow interferences due to the divisions.

Figure 17. Ball and thread.

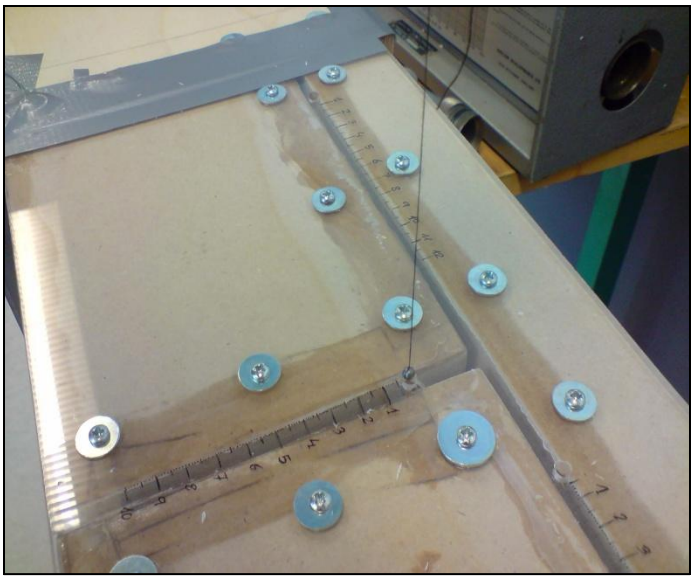


Figure 18. Model with scales, holes, and ball-threat system.

On the other hand, static pressures were measured by connecting the control point holes to the water column scale.

To make the test more accurate, it was done with three different wind speeds in the inlet: 10 m/s, 15 m/s and 20 m/s.

3.1.3 Test calibration

In order to use this system it was necessary to calibrate the ball displacement in relation to the dynamic pressure.

One option to calibrate the system was calculating the drag coefficient of the ball, knowing then the exerted force by measuring the displacement. With the exerted force it could be possible to calculate the speed and the dynamic pressure. This option was not considered due to the fact that in order to work with drag forces, the testing body must be surrounded by space enough in order to do not make any interferences; and as the conduit is not much wider than the ball diameter, the conditions could not be achieved. Moreover, the ball becomes an irregular sphere when it gets pressed to hold the thread, so the sum of these two factors would produce too much inaccuracy to the calibration.

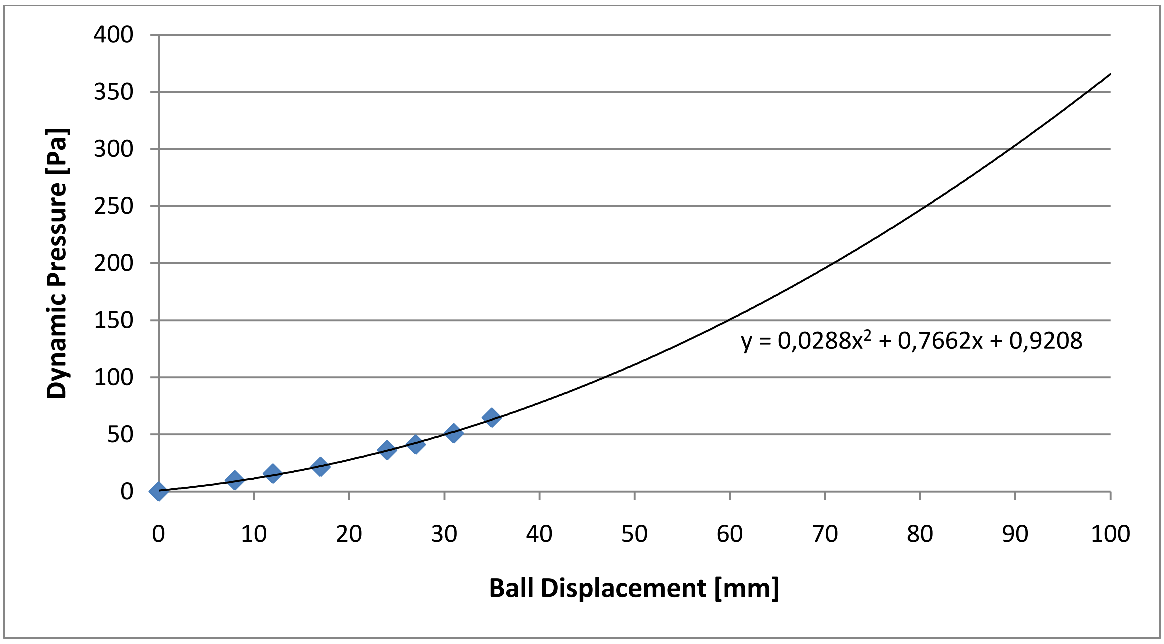
The next consideration for the calibration was testing directly the ball displacements, relating them with the dynamic pressure in the testing itself. To do so, the previous model was used, where the dynamic pressure could be measured using the Pitot tube right in the point where the ball was situated. The calibration point in the model was in the main branch, between the entrance and the first division. This point was chosen because it is where the flow is more even distributed and the ball is standing stable, making the measurements more accurate.

With the ball displacements and dynamic pressures, the following table was made.

Ball displ.	Dynamic Pressure	
mm	mm.w.c.	Pa
0	0	0
8	1	9,81
12	1,6	15,69
17	2,2	21,57
24	3,7	36,28
27	4,2	41,19
31	5,2	50,99
35	6,6	64,72

Table 13. Calibration values.

This table was entered in an Excel sheet and was used to make a graphic, necessary to calculate the tendency.



Graphic 25. Calibration tendency.

The tendency line, $\mathbf{y = 0.0288x^2 + 0.7662x + 0.9208}$, a 2nd order polynomial equation, relates the displacements of the ball directly to the dynamic pressure in that point. This equation was the one used to analyze the tests results.

It is necessary to say, however, that despite a relation between displacements and dynamic pressure was set, this method cannot be employed as an exact dynamic pressure measurer, as it carries much inaccuracy in all the tools used (Pitot tube, water column scale, fan, etc.). It was only used to analyze the flow tendency along the conduit and to have an overall idea of the fluid behaviour.

3.1.4 Measurements

The following results were taken in the laboratory, at an ambient temperature of 20 ºC (293.15 K).

Inlet Speed	Control Point	mm	Static P. [Pa]
10 m/s	A1	39	13,73
	A2	15	14,71
	B1	14	0,00
	C1	13	2,94

Table 14. Measurements 10 m/s.

Inlet Speed	Control Point	mm	Static P. [Pa]
15 m/s	A1	66	29,42
	A2	33	34,32
	B1	25	0,98
	C1	25	7,85

Table 15. Measurements 15 m/s.

Inlet Speed	Control Point	mm	Static P. [Pa]
20 m/s	A1	90	47,07
	A2	52	54,92
	B1	42	0,00
	C1	43	12,75

Table 16. Measurements 20 m/s.

Readings of ball displacements give an uncertain error of ±0.5 mm in each measurement.

3.1.5 Analysis of results

3.1.5.1 Pressures

These are the resulting pressure values from the ball displacements in each control point. Dynamic pressure values are taken using the tendency equation **$y = 0.0288x^2 + 0.7662x + 0.9208$**, being **x** the ball displacement and **y** the dynamic pressure value. Total pressure values are the sum of both dynamic and static pressure.

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]
10 m/s	A1	39	74,62	13,73	88,35
	A2	15	18,90	14,71	33,61
	B1	14	17,29	0,00	17,29
	C1	13	15,75	2,94	18,69

Table 17. Pressure values 10 m/s.

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]
15 m/s	A1	66	176,98	29,42	206,40
	A2	33	57,58	34,32	91,90
	B1	25	38,08	0,98	39,06
	C1	25	38,08	7,85	45,93

Table 18. Pressure values 15 m/s.

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]
20 m/s	A1	90	303,24	47,07	350,31
	A2	52	118,66	54,92	173,58
	B1	42	83,92	0,00	83,92
	C1	43	87,14	12,75	99,88

Table 19. Pressure values 20 m/s.

As there could be only four control points in the model, there are no control points enough to make an accurate pressure loss analysis along a single branch.

However, it can be noticed that from A1 to A2 there is a dynamic pressure loss of 185 Pa due to the division in the middle of the two points. Also, in the last bench between points A2 and C1 there is a dynamic pressure loss of approximately 30 Pa.

Regarding the static pressure, values in points B1 and C1 are zero or close to zero.

3.1.5.2 Speeds

Speeds are very useful to imagine and analyse how the fluid is behaving as it is running in the branches.

In order to calculate the speeds in our system, the formula below was used:

$$c = \sqrt{\frac{\rho_w \cdot 2 \cdot g \cdot \Delta h}{\rho_{air} \cdot 1000}}$$

Where:

c = speed, in [m/s]
ρ_w = density of water, being 1000 kg/m^3
g = gravity, being 9,80665 m/s^2
Δh = dynamic pressure in each point, in [mm.w.c.]
ρ_{air} = density of air, being 1,225 kg/m^3

The formula was applied to every dynamic pressure value taken from the tests. In the following table there can be seen all the results:

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Speed [m/s]
10 m/s	A1	39	74,62	11,04
	A2	15	18,90	5,55
	B1	14	17,29	5,31
	C1	13	15,75	5,07

Table 20. Speed values 10 m/s.

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Speed [m/s]
15 m/s	A1	66	176,98	17,00
	A2	33	57,58	9,70
	B1	25	38,08	7,89
	C1	25	38,08	7,89

Table 21. Speed values 15 m/s.

Inlet Speed	Control Point	mm	Dynamic P. [Pa]	Speed [m/s]
20 m/s	A1	90	303,24	22,25
	A2	52	118,66	13,92
	B1	42	83,92	11,71
	C1	43	87,14	11,93

Table 22. Speed values 20 m/s.

From the speed values in the tables it can be seen that the sum of the values in A2 and B1 is almost the value in A1. It can also be seen that the speed value in A2 remains almost the same in C1 just after the last bench.

3.1.5.3 Flows

Once the speeds were calculated, the last things to analyse were the flows. The flows allow checking the amount of air running through the manifold, and enable to prove if there is any air loss along the way.

The flows are calculated using the speed value in each control point and the branch section area in that point. For these calculations, the following data was used:

Branch	Width [m]	Depth [m]	Area [m^2]
Main	0,005	0,2	0,001
First	0,005	0,2	0,001
Second	0,005	0,2	0,001

Table 23. Geometry data.

And in the following table there can be seen the calculated values:

Inlet Speed	Control Point	mm	Flow [m3/s]
10 m/s	A1	39	0,01104
	A2	15	0,00555
	B1	14	0,00531
	C1	13	0,00507

Table 24. Flow values 10 m/s.

Inlet Speed	Control Point	mm	Flow [m3/s]
15 m/s	A1	66	0,01700
	A2	33	0,00970
	B1	25	0,00789
	C1	25	0,00789

Table 25. Flow values 15 m/s.

Inlet Speed	Control Point	mm	Flow [m3/s]
20 m/s	A1	90	0,02225
	A2	52	0,01392
	B1	42	0,01171
	C1	43	0,01193

Table 26. Flow values 20 m/s.

Analysing the flow values in the tables, it can be read that the flows keep constant all along the model, as the sum of the flows in B1 and C1 (outlets) is almost the value in A1 (inlet) in all three inlet speeds. As well as the outlets, the sum of the values in A2 and B1 (first division) is also almost the value in A1 considering the error.

3.2 EES REPORT

3.2.1 Model Description

The system introduced in EES in order to be analyzed is very similar to the model previously studied, with the only differences that in this case all the branches are narrower (5mm. instead of 20 and 30mm.) and the main branch has the same width as the secondary branches.

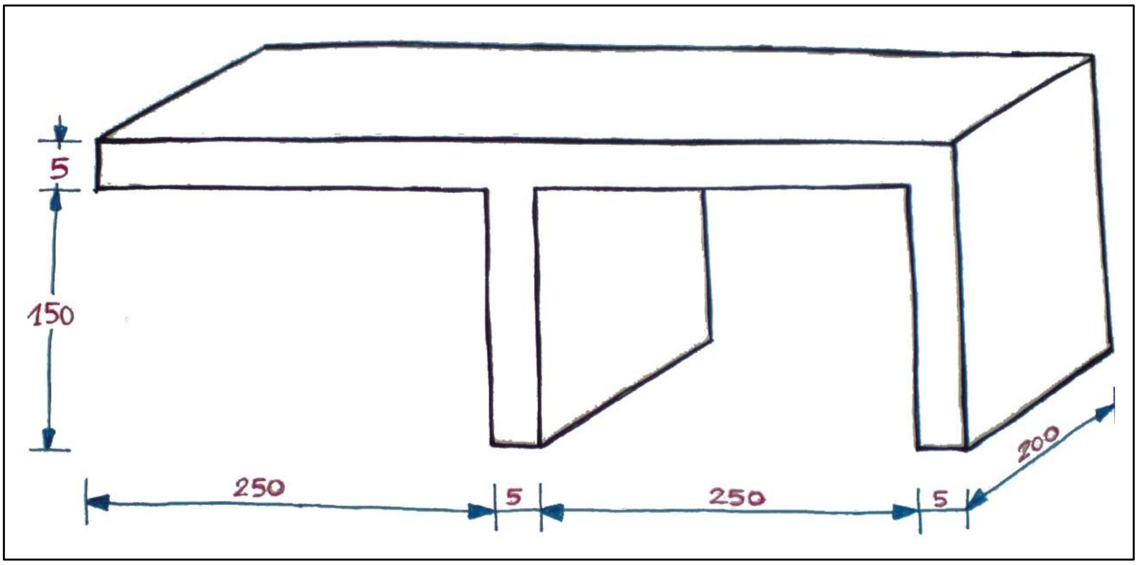


Figure 19. Model geometry

3.2.2 System Description

The 11 states defined in this case are the same as in the previous study (see Figure 4).

However, as it happened before, these states are only employed in EES, so it has been necessary to show the equivalent states employed in the laboratory test. In this study, as the number of points tested in the laboratory is very small (only four control points), the lab points do not coincide with the EES points, so the test points are placed between two EES points.

TEST Point	EES Point
A1	Between point 2 and point 3
A2	Between 5 and 6
B1	Between 4 and 9
C1	Between 7 and 8

Table 27. EES points equivalent to the TEST points

3.2.3 Program Construction

The Equations Window is almost the same shown in the Appendix B2 with one difference in the input data: the inlet flow. In this model three different cases have been studied depending on the inlet flow values, calculated by multiplying the inlet speed obtained in the lab by the branch area (0.001m^3). All the results shown in the next sections are related to the theoretical inlet speed.

Theoretical Inlet Speed (m/s)	Inlet Speed from the TEST (m/s)	Inlet Flow (m^3/s)
20	22.25	0.0223
15	17.00	0.0170
10	11.04	0.0110

Table 28. Inlet flows

3.2.4 Solution

All the solutions provided by the program in the Solution Window have been resumed in the tables below. The units are shown in SI units. The three whole Solution Window can be seen in Appendix B2.

- 1st case. Inlet speed = 20m/s.

	Solution Window				
State (EES)	Speed	Dynamic P.	Static P.	Total P.	Flow
1	1.50	1.38	489.80	491.20	0.0223
2	22.25	303.20	36.36	339.60	undefined
3	22.25	303.20	27.82	331.00	"
4	7.58	35.22	1.11	36.33	"
5	14.67	131.80	179.00	310.7	"
6	14.67	131.80	175.10	306.90	"
7	14.67	131.80	3.85	135.60	"
8	14.67	131.80	0	131.80	"
9	7.58	35.22	0	35.22	"
10	0	0	0	0	0.0076
11	0	0	0	0	0.0147

Table 29. Solution Window resume for 20m/s in EES states

	Solution Window				
State (Test)	Speed	Dynamic P.	Static P.	Total P.	Flow
A1	22.25	303.20	32.09	335.29	0.0223
A2	14.67	131.80	177.05	308.85	undefined
B1	7.58	35.22	0.56	35.78	0.0076
C1	14.67	131.80	1.93	133.73	0.0147

Table 30. Solution Window resume for 20m/s in TEST states

- 2nd case. Inlet speed = 15m/s.

	Solution Window				
State (EES)	Speed	Dynamic P.	Static P.	Total P.	Flow
1	1.50	1.38	282.60	284.00	0.0170
2	17.00	177.00	18.50	195.50	undefined
3	17.00	177.00	13.40	190.40	"
4	5.65	19.55	0.65	20.20	"
5	11.35	78.91	99.42	178.3	"
6	11.35	78.91	97.05	176.00	"
7	11.35	78.91	2.37	81.27	"
8	11.35	78.91	0	78.91	"
9	5.65	19.55	0	19.55	"
10	0	0	0	0	0.0057
11	0	0	0	0	0.0114

Table 31. Solution Window resume for 15m/s in EES points

	Solution Window				
State (Test)	Speed	Dynamic P.	Static P.	Total P.	Flow
A1	17.00	177.00	15.95	192.95	0.0170
A2	11.35	78.91	98.24	177.145	undefined
B1	5.65	19.55	0.33	19.88	0.0057
C1	11.35	78.91	1.18	80.09	0.0114

Table 32. Solution Window resume for 15m/s in TEST points

- 3rd case. Inlet speed = 10m/s.

	Solution Window				
State (EES)	**Speed**	**Dynamic P.**	**Static P.**	**Total P.**	**Flow**
1	1.50	1.38	118.60	119.90	0.0110
2	11.04	74.65	7.96	82.61	undefined
3	11.04	74.65	5.71	80.36	"
4	3.67	8.26	0.29	8.55	"
5	7.37	33.25	42.01	75.27	"
6	7.37	33.25	40.96	74.21	"
7	7.37	33.25	1.05	34.31	"
8	7.37	33.25	0	33.25	"
9	3.67	8.26	0	8.257	"
10	0	0	0	0	0.0037
11	0	0	0	0	0.0074

Table 33. Solution Window resume for 10m/s in EES points

	Solution Window				
State (Test)	**Speed**	**Dynamic P.**	**Static P.**	**Total P.**	**Flow**
A1	11.04	74.65	6.84	81.49	0.0110
A2	7.37	33.25	41.49	74.74	undefined
B1	3.67	8.26	0.15	8.41	0.0037
C1	7.37	33.25	0.53	33.78	0.0074

Table 34. Solution Window resume for 10m/s in TEST points

All the input and output data concerning to the three studies, as well as a sketch of the system are shown in the Diagram Window (see the figures below).

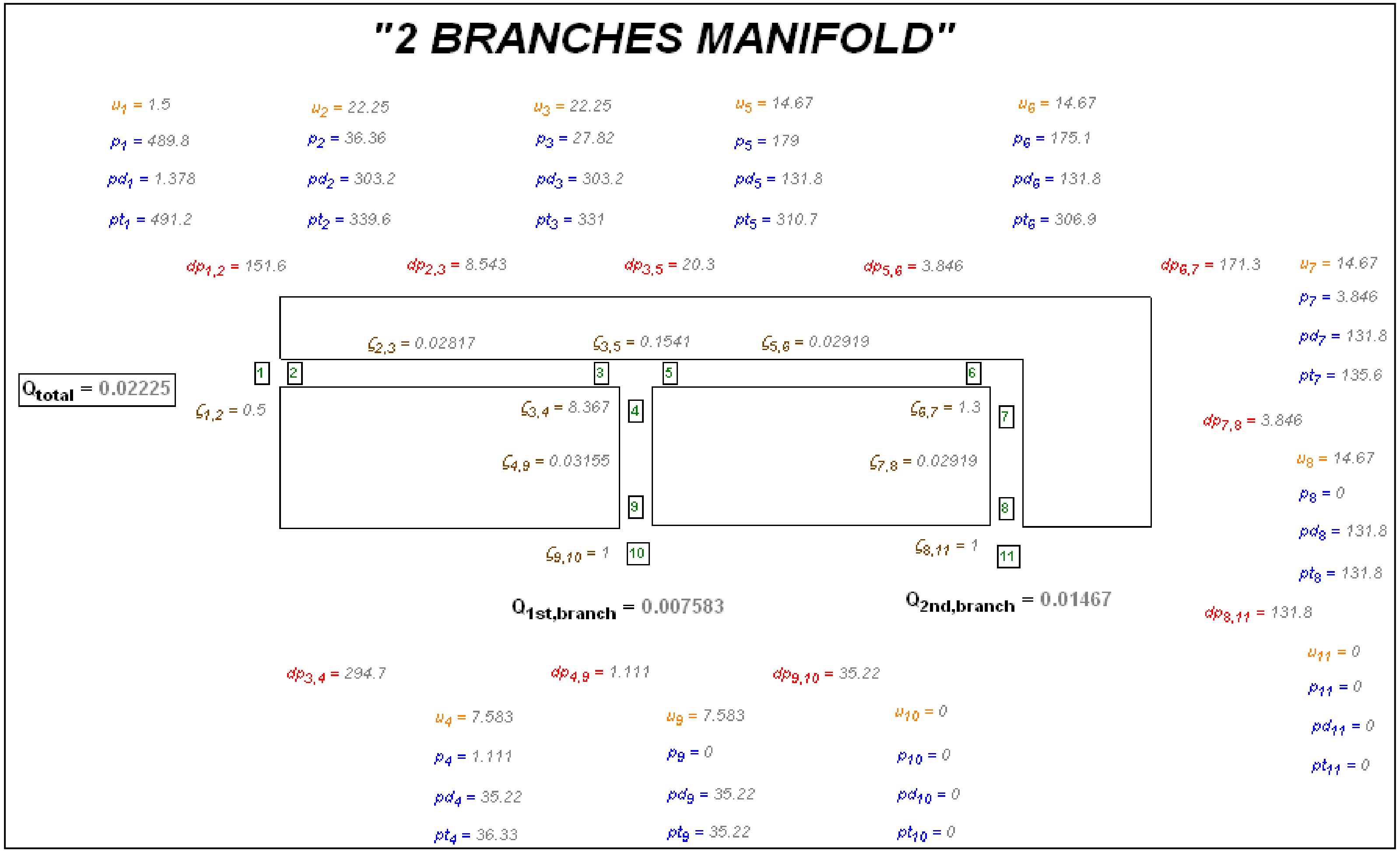


Figure 20. Diagram Window for 20m/s

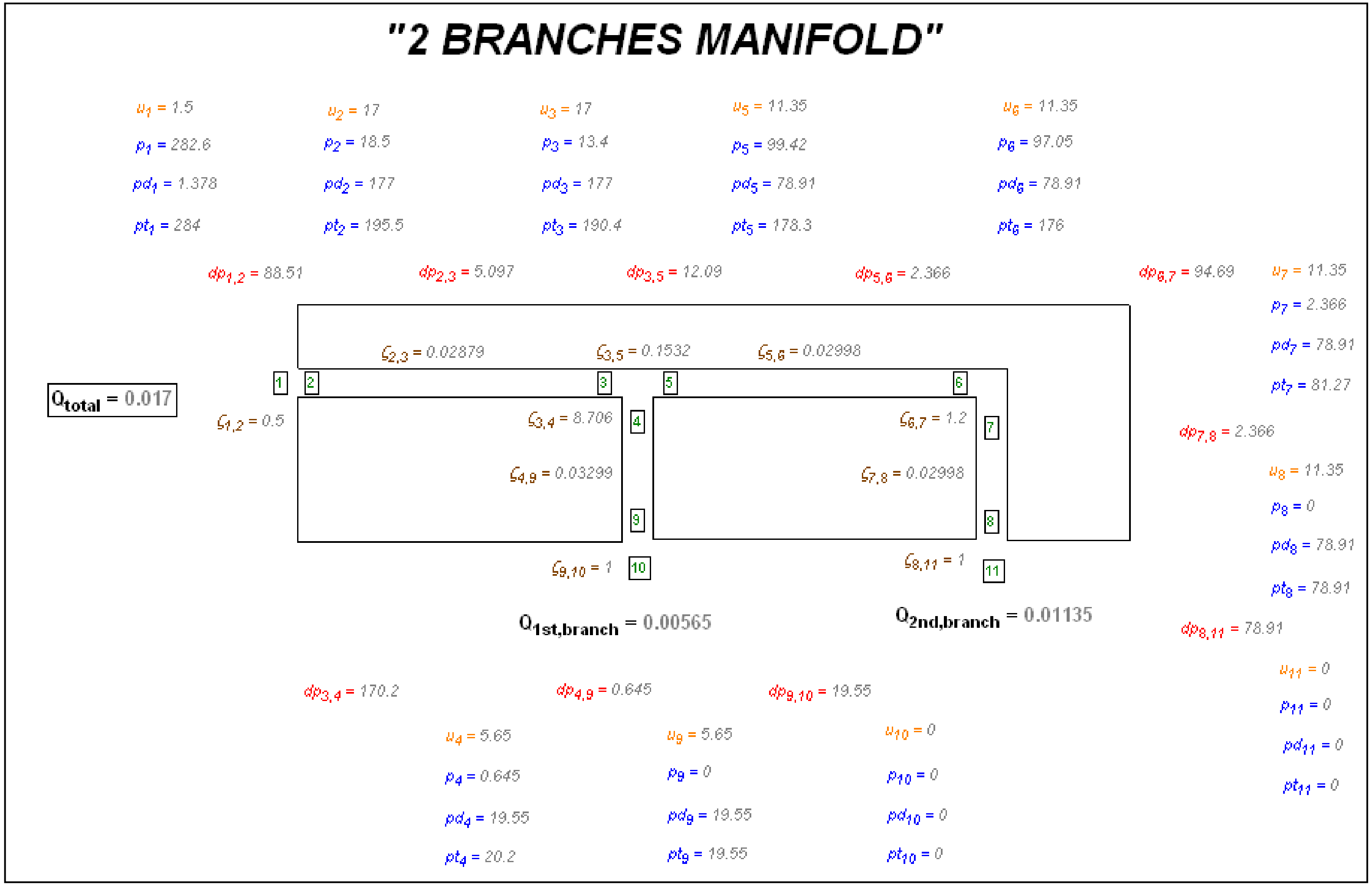


Figure 21. Diagram Window for 15m/s

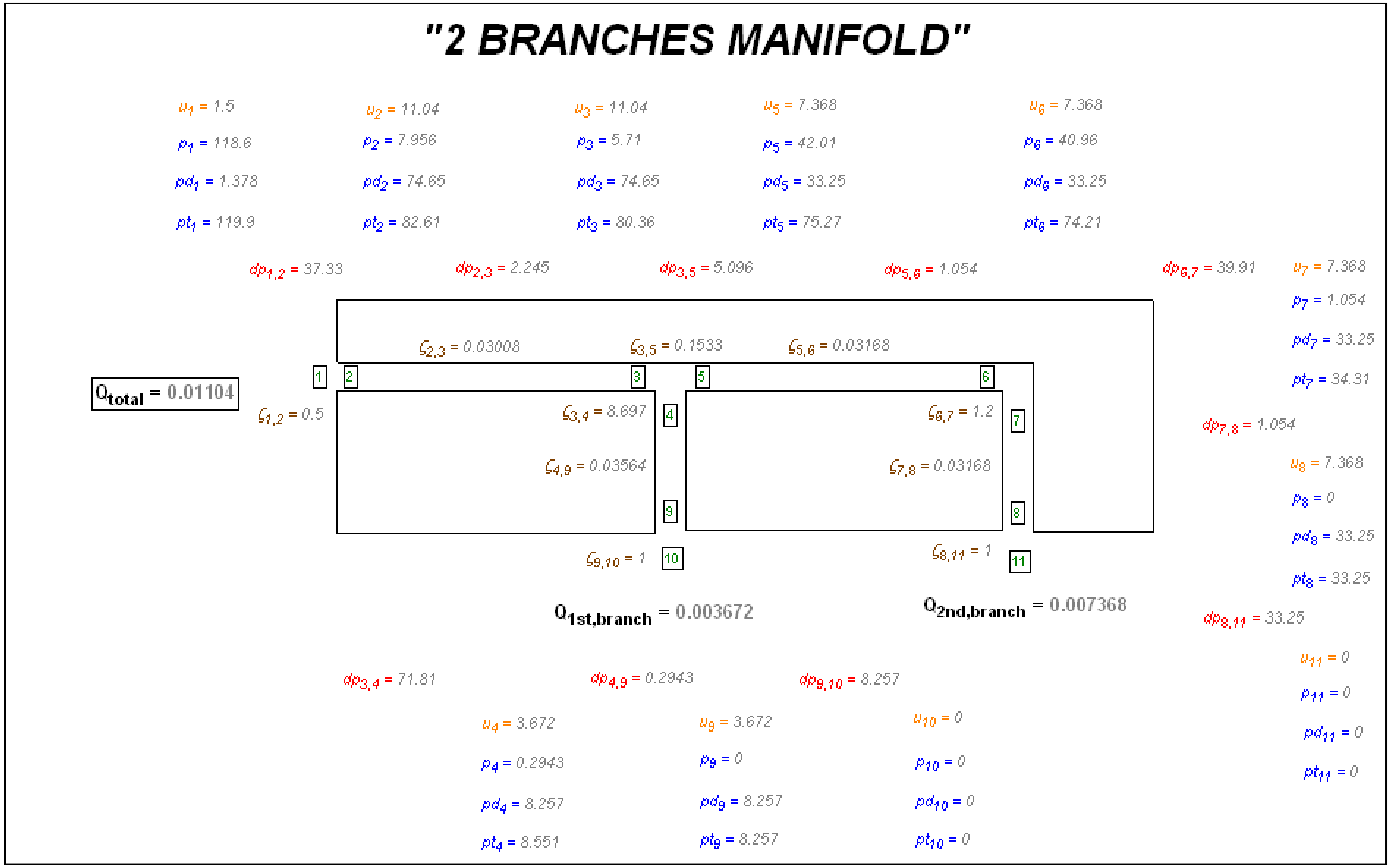
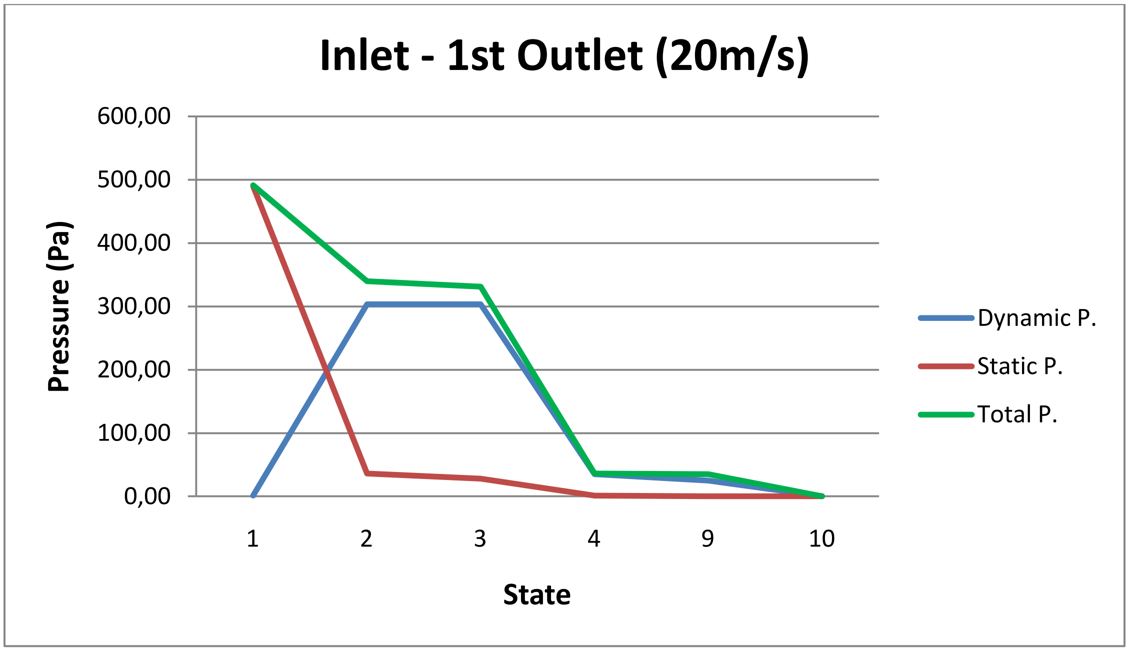


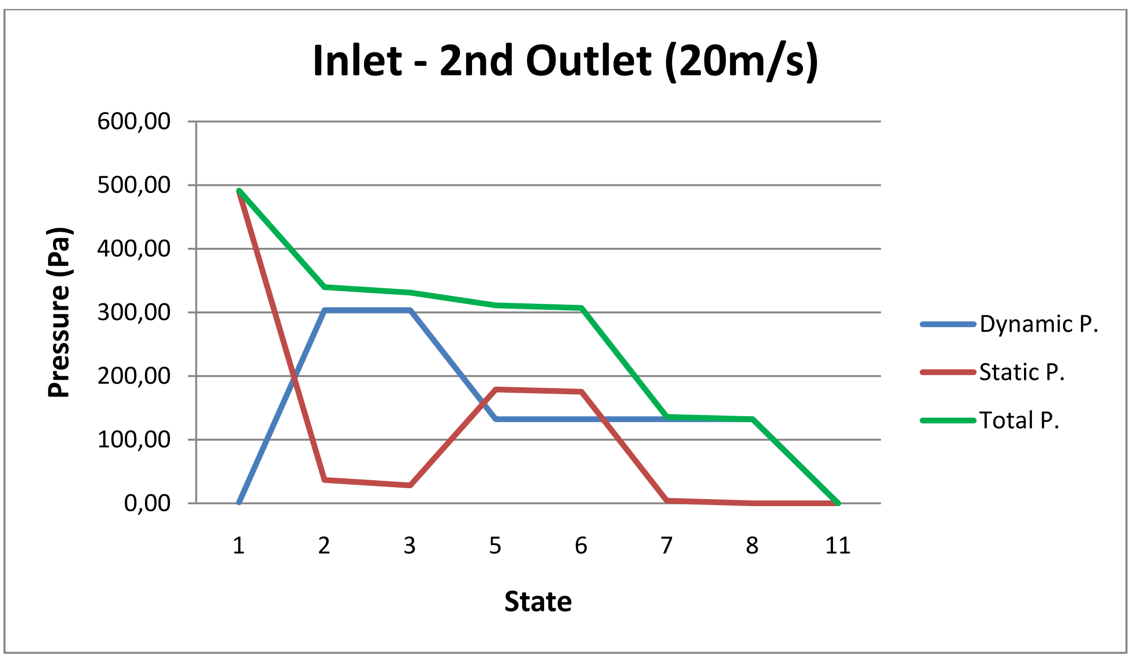
Figure 22. Diagram Window for 10m/s

3.2.5 Analysis of results

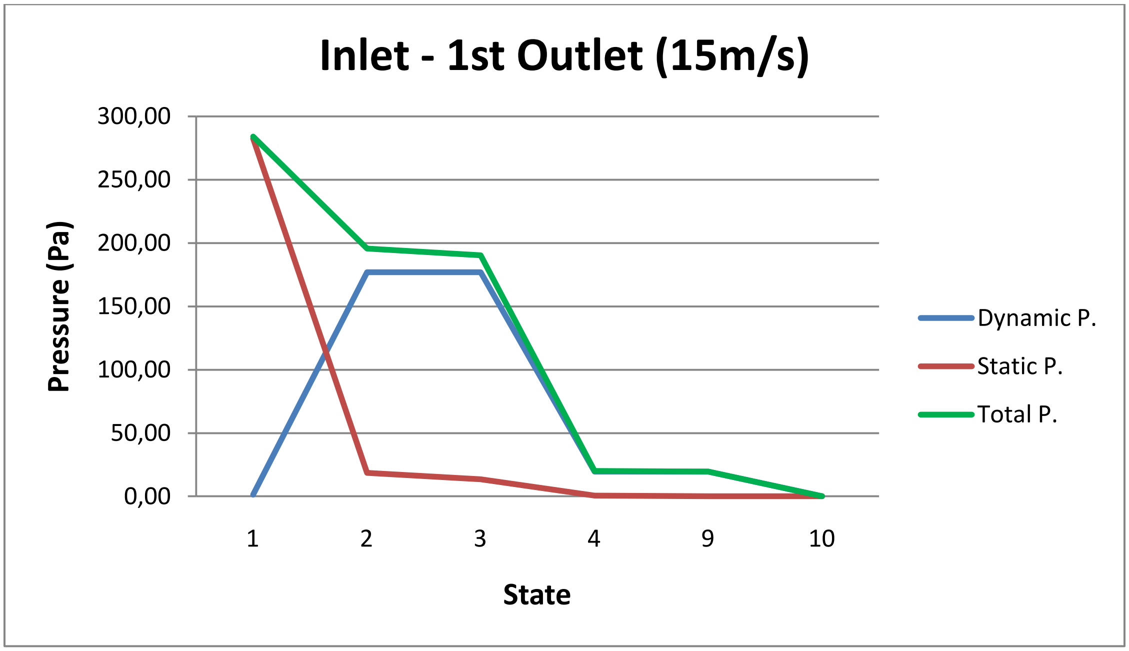
3.2.5.1 Pressure



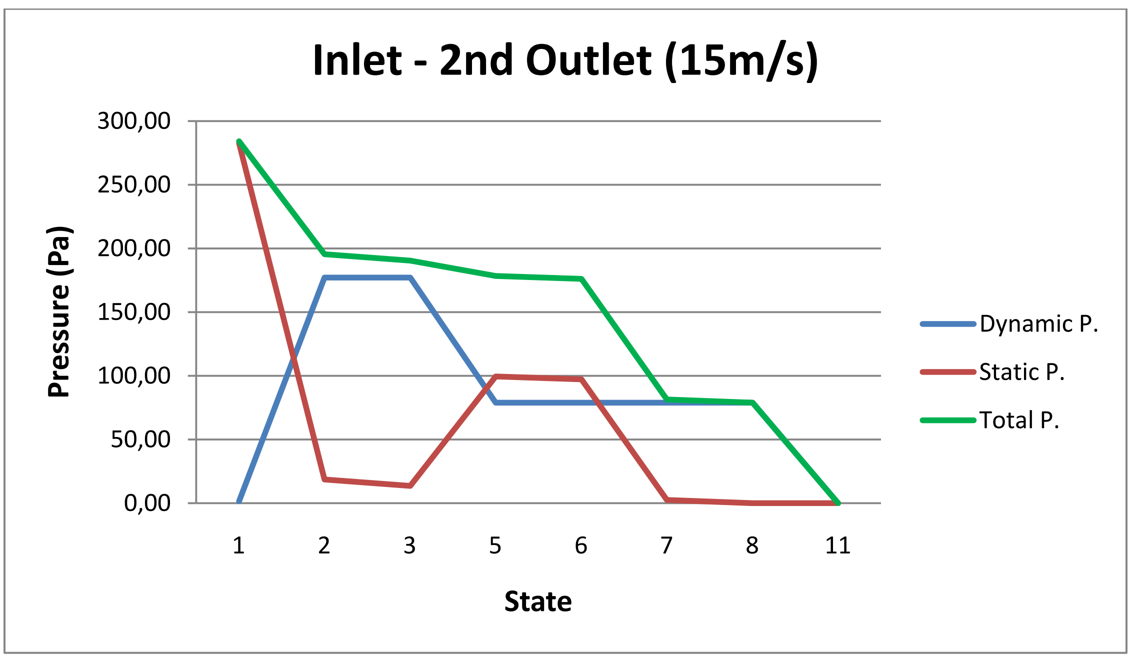
Graphic 26. EES pressures for 20m/s between the inlet and the 1st outlet



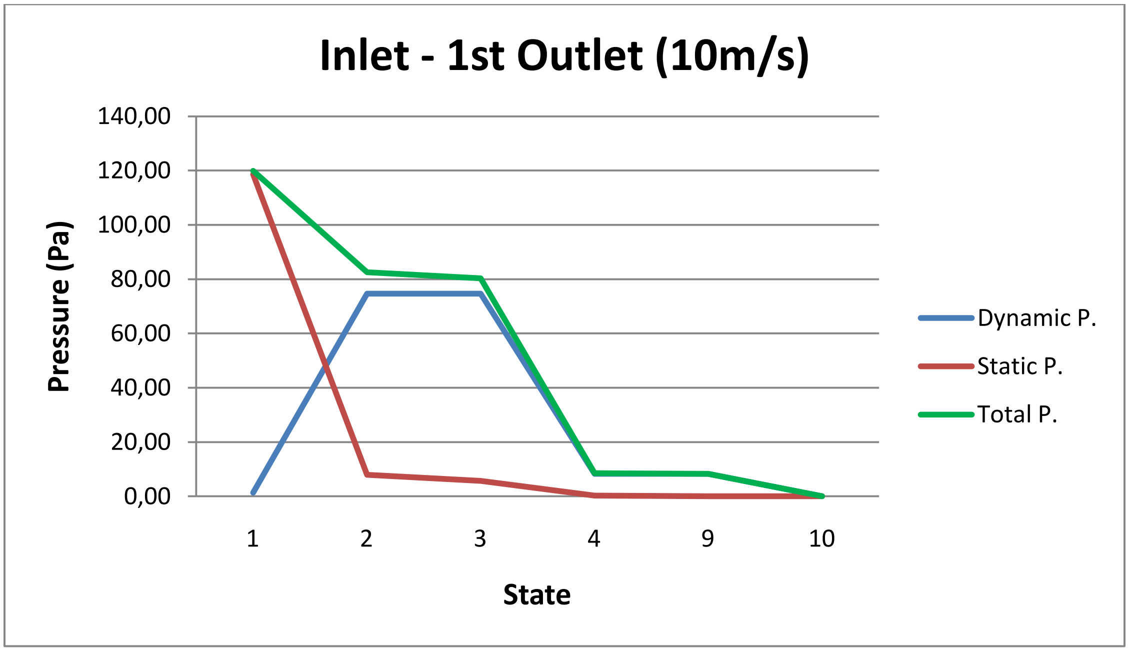
Graphic 27. EES pressures for 20m/s between the inlet and the 2nd outlet



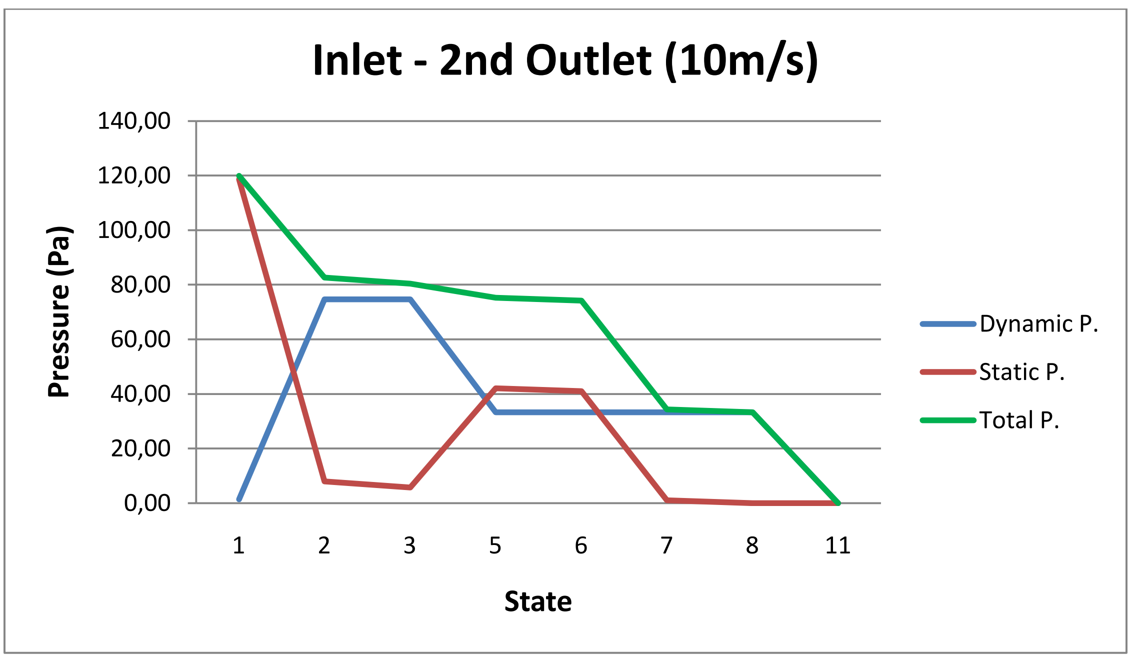
Graphic 28. EES pressures for 15m/s between the inlet and the 1st outlet



Graphic 29. EES pressures for 15m/s between the inlet and the 2nd outlet



Graphic 30. EES pressures for 10m/s between the inlet and the 1st outlet



Graphic 31. EES pressures for 10m/s between the inlet and the 2nd outlet

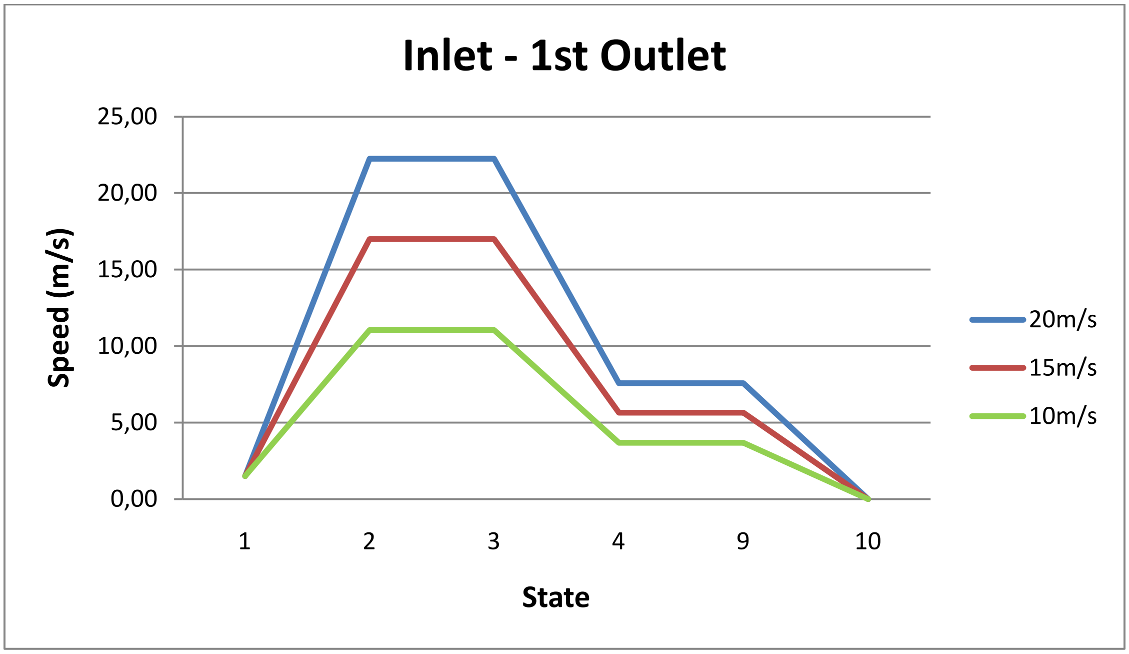
As it can be observed, all the graphics showing the pressures data between the inlet and the first outlet have the same shape and so does happen with the graphics between the inlet and the second outlet. This indicates that the pressures behaviour is the same whatever the inlet speed is, and the only difference is the pressure values, which are higher when the inlet speed is 20m/s and lower when this is 10m/s.

In the case of the Inlet - 1st Outlet graphics, the most significant control point is state 4 (right at the beginning of the first branch) because there is a very important pressure loss as a result of the very high zeta coefficient (around 8.5 in the three cases) between state 3 and state 4 calculated with the double interpolation “zeta_branch”.

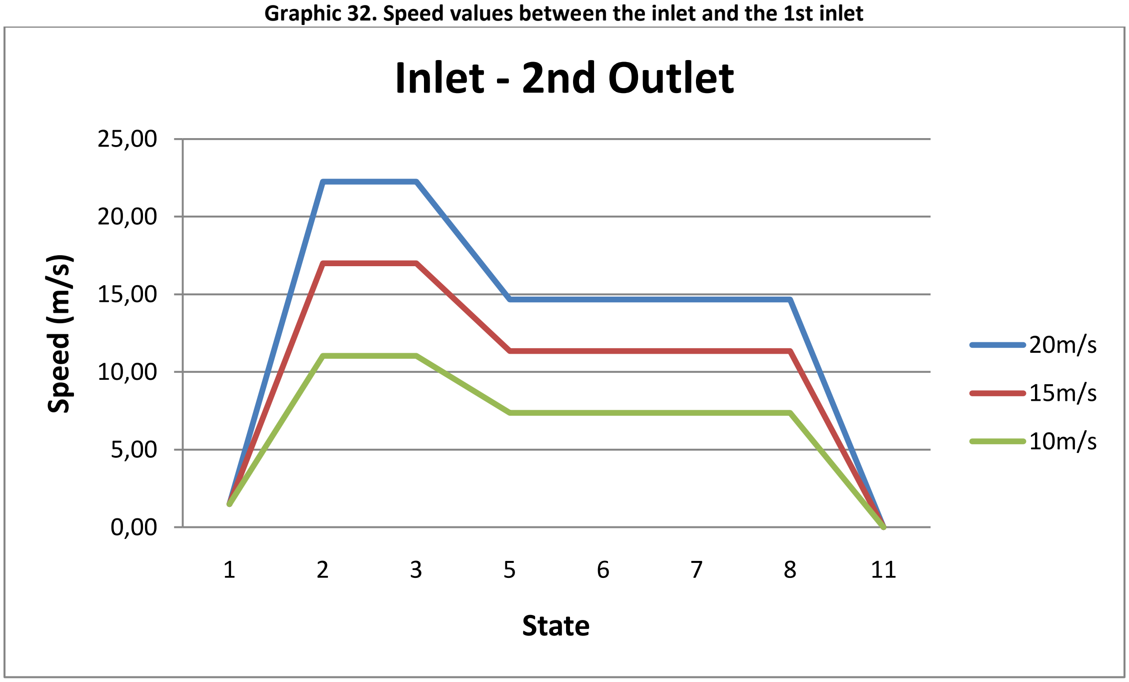
In the Inlet – 2nd Outlet graphics state 5 (after the flow division) and state 7 (first point in the second branch) are the most important points. In state 5, the total pressure does not drop very much as a consequence of the low value (ς=0.15) calculated by using the double interpolation “zeta_through”. However, in the control point 7 the situation is quite different because the zeta value is higher (ς=1.3), so there is an important loss pressure.

The rest of the pressure behaviours are the same as in the model of wide branches, so it has already been explained (see EES REPORT, page 21).

3.2.5.2 Speed



Graphic 32. Speed values between the inlet and the 1st inlet



Graphic 33. Speed values between the inlet and the 2nd outlet

In the previous model, there were two reasons to explain the speed variations: the flow division and the area changes. However, in this case, as the area is constant because all the branches have the same dimensions, only the flow separation which takes place in state 4 (graphic

32) and sate 5 (graphic 33) explain the speed changes shown in the graphics . The fact that the 10m/s graphic values are lower than the 20m/s values is obviously a consequence of the smaller air flow.

3.2.5.3 Flow

The inlet flows, which are obtained in the laboratory tests (see table 24, 25 and 26), are introduced as an input value in the Equation Window. After reaching the state 3 the flow is divided and a 34.1% goes through the first branch while a 65.9% goes through the second one. The percentages are the same in the three studies.

	Flow (m^3/s)		
Inlet Speed (m/s)	Main branch	1st Branch	2nd Branch
20	0.0223	0.0076	0.0147
15	0.0170	0.0057	0.0114
10	0.0110	0.0037	0.0074

Table 35. Flow division

Using the Uncertainty Propagation tool (only in the case of the highest flow because the results are very similar in the three studies), it can be appreciated how big is the influence of each one of the variables. The most remarkable conclusion is that in the model with narrow branches the inlet flow is not such as relevant factor as could be expected. However the width of the first branch is much more important than in the model with wide branches (where its uncertainty value is around 5%).

Variable	**Variable ± uncertainty**	**% of uncertainty**
Q 1st branch	0.007583 ± 0.004718	
1st branch width	0.005 ± 0.002	37.45%
2nd branch width	0.005 ± 0.002	57.78%
Main branch width	0.005 ± 0.002	2.18%
Q total	0.02225 ± 0.002225	2.58%
Device depth	0.2 ± 0.001	0.00%
Q 2nd branch	0.03558 ± 0.00427	
1st branch width	0.005 ± 0.002	34.97%
2nd branch width	0.005 ± 0.002	53.96%
Main branch width	0.005 ± 0.002	2.04%
Q total	0.02225 ± 0.002225	9.04%
Device depth	0.2 ± 0.001	0.00%

Table 36. Flow uncertainty values for 20m/s

3.3 CFD REPORT

3.3.1 Model Description

In this second test, the model is almost the same than the one used for the first test. The differences between them are the conduit dimensions. It is still a main conduit deviated in two outlet branches, but in this case the area remains invariant throughout the hole model. If in the previous model the conduits were 30, 19.5 and 21.5 mm thick, now the three of them are only 5 mm. The conduits are considerably narrower this time, to check if the flow behaves the same way after de division. The main branch is still 200 mm wide and the first division is at 250 mm from the inlet, but in this case the whole branch is a bit shorter, it is reduced to 510 mm. The outlet branches are both 150 mm long and 250 mm wide as in the first model.

Figure 23. Model overview.

3.3.2 System description

The control points used for this second test will be the same used in the first one, from one to eight. But in this case the test control points (A1, A2, B1 and C1) are displaced in the middle of the branches as figure 24 shows. The values taken from the CFD program will be operated and an average value will be taken to determine the value in the middle of each branch.

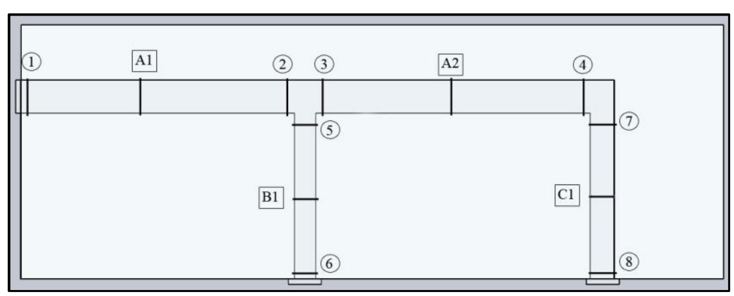


Figure 24. Control points.

The lids will be placed again at the inlet and the two outlets to set the inlet flow up and to get the flow rates in each one of the outlets.

3.3.3 Program Construction

The program is made the same way than the first time. Starting with the assembly and setting up the Flow Simulation Studio and following with the wizard. This test will be done working with air again, and with a wall roughness of 0.0001 micrometer.

In this case, the inlet flows will depend on the inlet speed we are working with. For a flow speed of 10 m/s the flow rate will be 0.0110 m^3/s. When the speed is increased to 15 m/s the flow will be 0.0170 m^3/s. And finally with a 20 m/s speed at the inlet, the flow will increase to 0.0223 m^3/s. The air temperature for this test is set to 293.15 K (20 ºC). The outlets are again under the effect of the atmospheric pressure.

3.3.4 Analysis of results

As in the first test, once the simulation is run we can get the first results. However, it is needed to place several point meshes to get the control point values and later compare them to the test results.

The following tables show the results from the SolidWorks' CFD program for the three different speeds tested. As it can be seen the flow is not even distributed through the branches, but more flow is going to the second branch even the thickness is the same in both outlet branches.

	Solution Window				
CFD State	Speed [m/s]	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]	Volume Flow [m^3/s]
I1	--	--	--	--	0,0110
1	8,96	49,13	25,78	74,91	--
2	8,74	46,80	-7,17	39,63	--
3	5,50	18,49	37,62	56,11	--
4	5,39	17,77	23,76	41,54	--
5	4,06	10,09	1,36	11,45	--
6	4,06	10,09	-0,72	9,37	--
O1	--	--	--	--	0,0042
7	5,12	16,07	9,42	25,49	--
8	5,14	16,16	0,78	16,94	--
O2	--	--	--	--	0,0069

Table 37. SolidWorks' Flow Simulation results for 10 m/s.

	Solution Window				
CFD State	Speed [m/s]	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]	Volume Flow [m3/s]
I1	--	--	--	--	0,0170
1	13,87	117,83	43,03	160,86	--
2	13,57	112,68	-18,98	93,70	--
3	8,89	48,43	81,18	129,61	--
4	8,74	46,78	53,52	100,30	--
5	5,83	20,83	1,91	22,73	--
6	5,83	20,83	-1,76	19,07	--
O1	--	--	--	--	0,0060
7	8,28	41,93	17,88	59,81	--
8	8,34	42,62	1,55	44,17	--
O2	--	--	--	--	0,0110

Table 38. SolidWorks' Flow Simulation results for 15 m/s.

	Solution Window				
CFD State	Speed [m/s]	Dynamic P. [Pa]	Static P. [Pa]	Total P. [Pa]	Volume Flow [m3/s]
I1	--	--	--	--	0,0223
1	18,21	203,12	91,23	294,35	--
2	17,97	197,66	-24,20	173,46	--
3	11,83	85,70	129,51	215,22	--
4	11,65	83,12	88,86	171,98	--
5	7,43	33,82	3,22	37,04	--
6	7,44	33,85	-2,09	31,76	--
O1	--	--	--	--	0,0077
7	10,98	73,87	24,61	98,48	--
8	11,12	75,69	2,10	77,80	--
O2	--	--	--	--	0,0147

Table 39. SolidWorks' Flow Simulation results for 20 m/s.

Figures 25, 26 and 27 show the pressure variations for a 10, 15 or 20 meters per second flow. The differences between them are the highness of the values, but the behaviour is the same in the three cases. The pressure in the inlet is reduced while the flow is moving through the conduit, but increases again in the main branch after the division. And in the two branches the pressure decreases again influenced by the atmospheric pressure in the outlets.

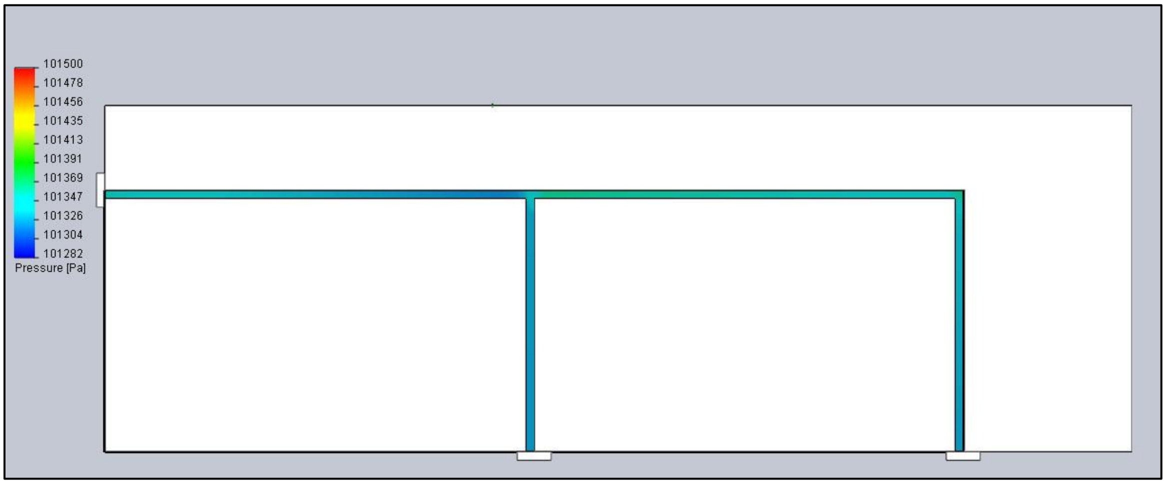


Figure 25. Pressure changes along the conduit at 10 m/s.

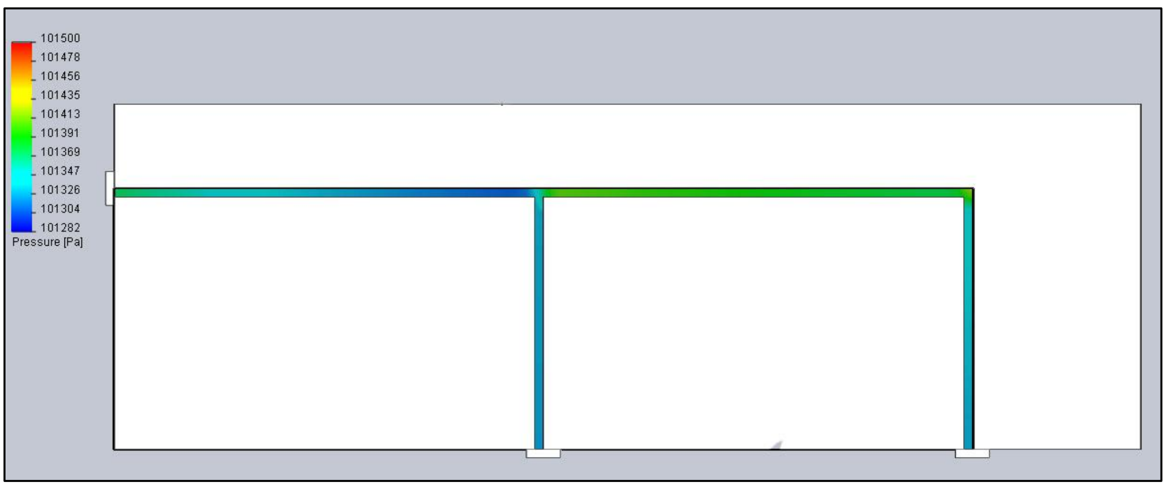


Figure 26. Pressure changes along the conduit at 15 m/s.

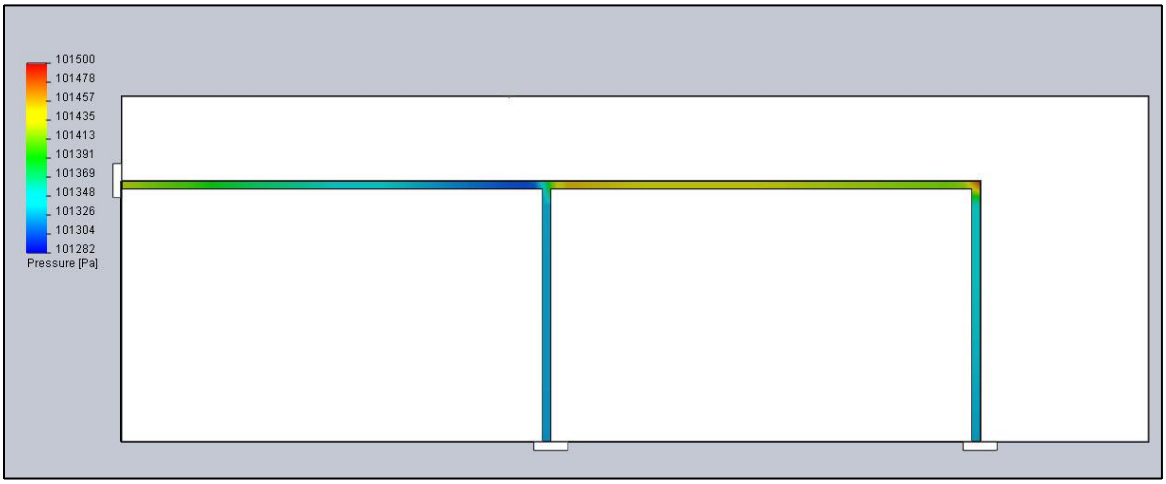


Figure 27. Pressure changes along the conduit at 20 m/s.

In figures 28, 29 and 30 is represented the speed along the branches. Again it can be seen how the tendency in the three cases is maintained; only being changed the magnitude of the values. The speed in the inlet is maintained until the division; there, the fluid going to the first branch decreases more the speed than the fluid going through the main branch. This is due to the fact that there is more fluid going through the second branch than through the first and the areas remain constant.

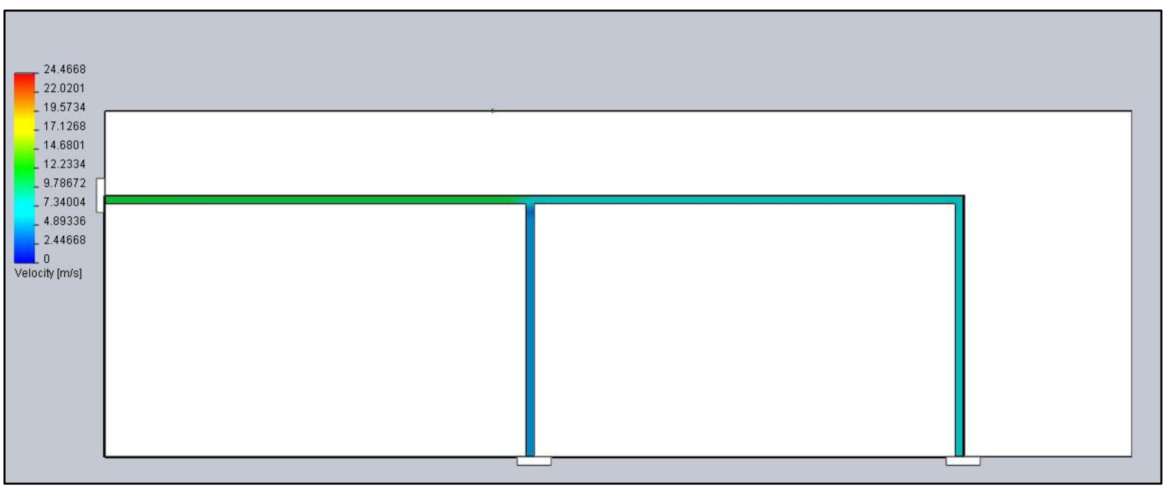


Figure 28.Velocity changes along the conduit at 10 m/s.

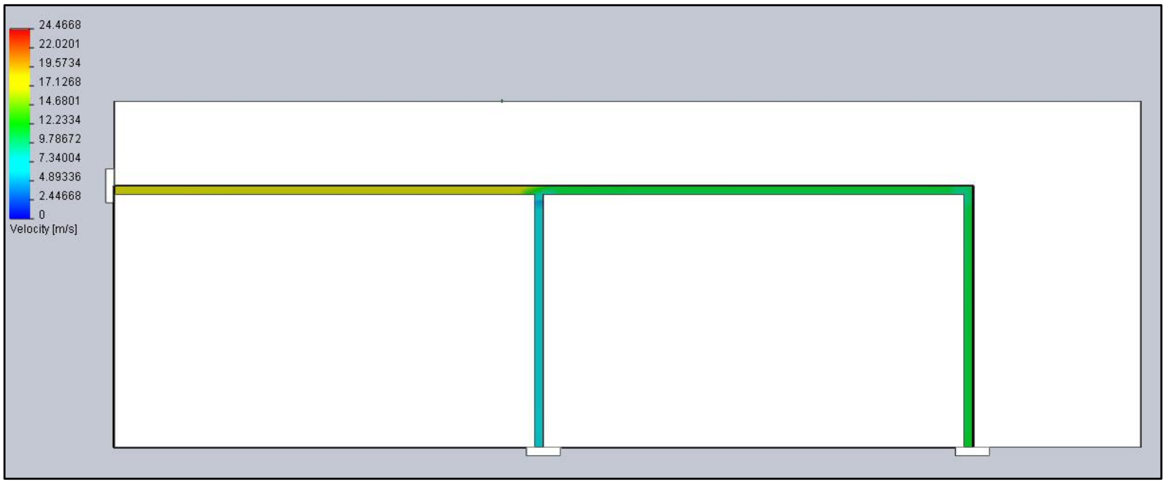


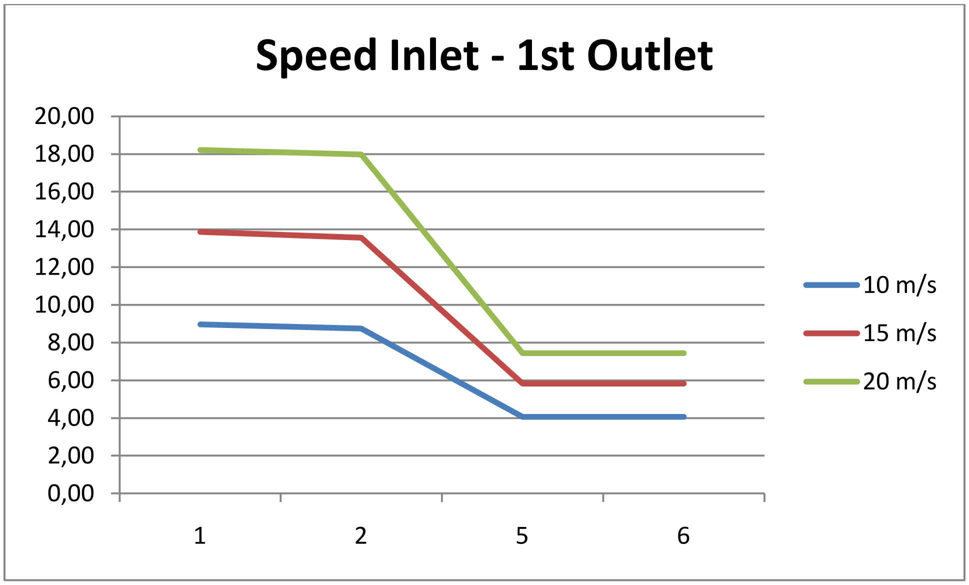
Figure 29.Velocity changes along the conduit at 15 m/s.

Figure 30.Velocity changes along the conduit at 20 m/s.

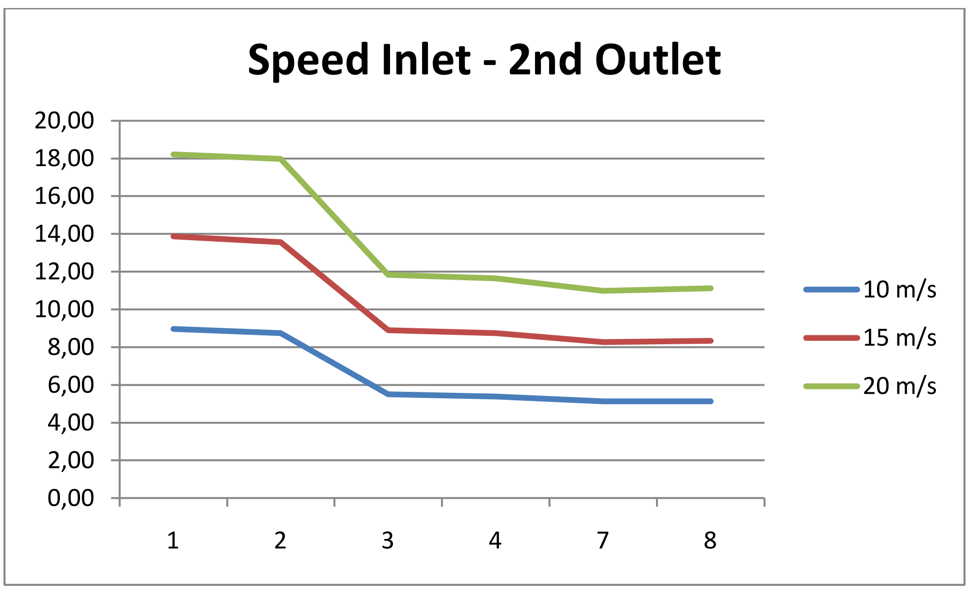
Graphics 34 and 35 show the speed variances from the inlet to each one of the outlets for the three speeds.

It can be appreciated how the speed from 1 to 2 is lightly reduced; this is due to the friction between the walls and the flow. The same is happening from 3 to 4, when the flow remaining in the main branch reduces its speed a very few points, and from 7 to 8.

From 2 to 5 the speed gap is considerably bigger than before, as the flow is reduced to its half and the area remains constant. The same happens from 2 to 3, when the flow is reduced in the same way than from 2 to 5 and the area does not change.

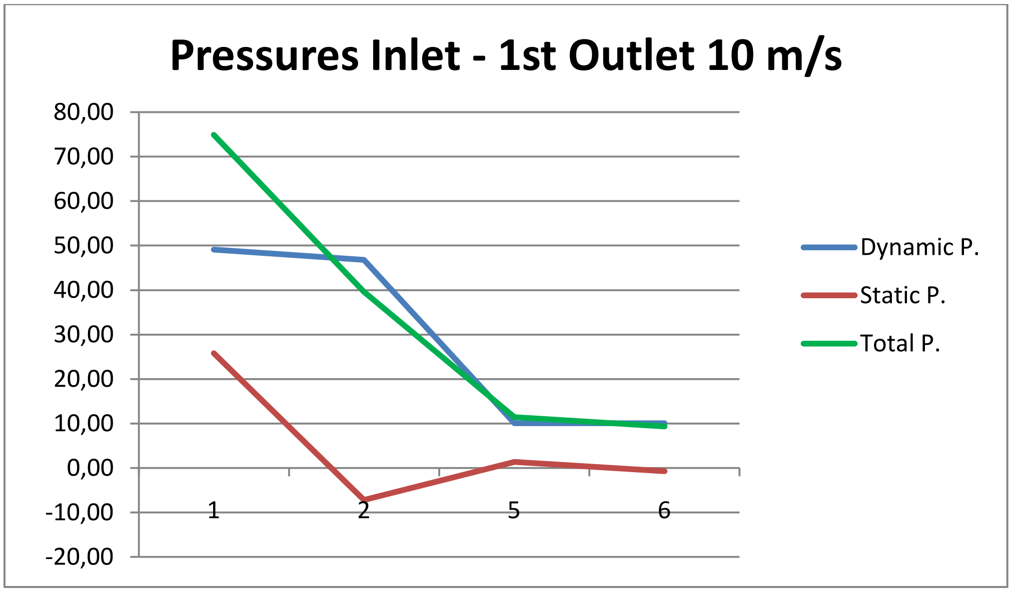


Graphic 34. CFD fluid speed from inlet to first outlet for the three different inlet speeds.

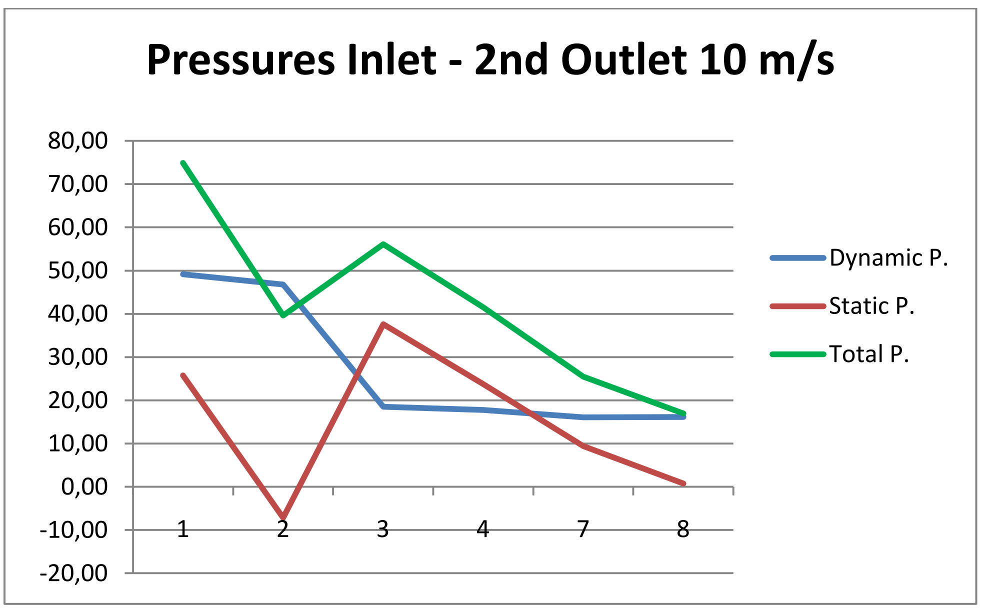


Graphic 35. CFD fluid speed from inlet to second outlet for the three different inlet speeds.

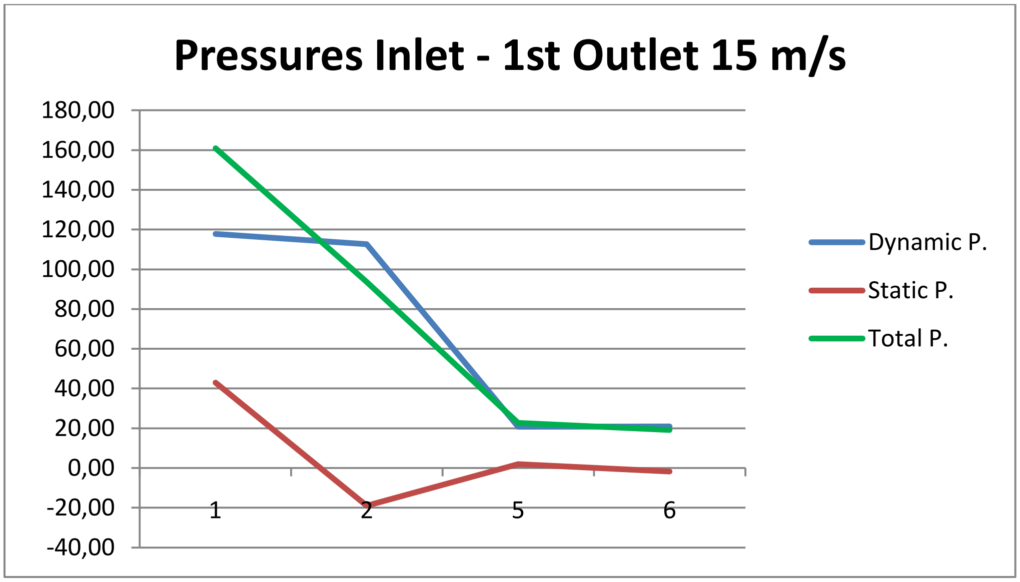
In the following six graphics are represented the dynamic, static and total pressures from the inlet to each of the outlets. As it can be appreciated the flow pressure behaviour with the three speeds is very similar.



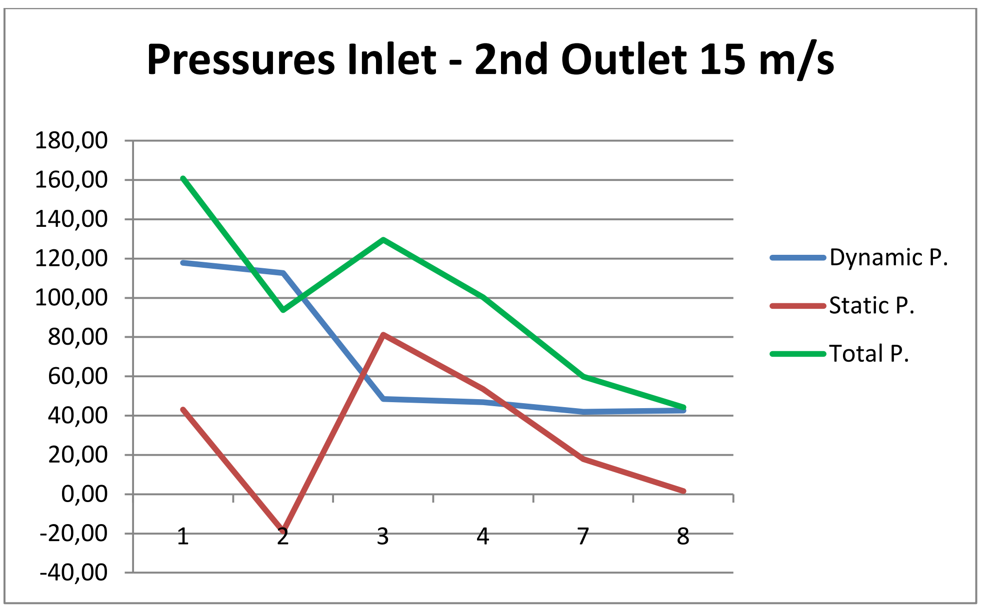
Graphic 36. CFD fluid pressure between inlet and first outlet for a 10 m/s inlet speed.



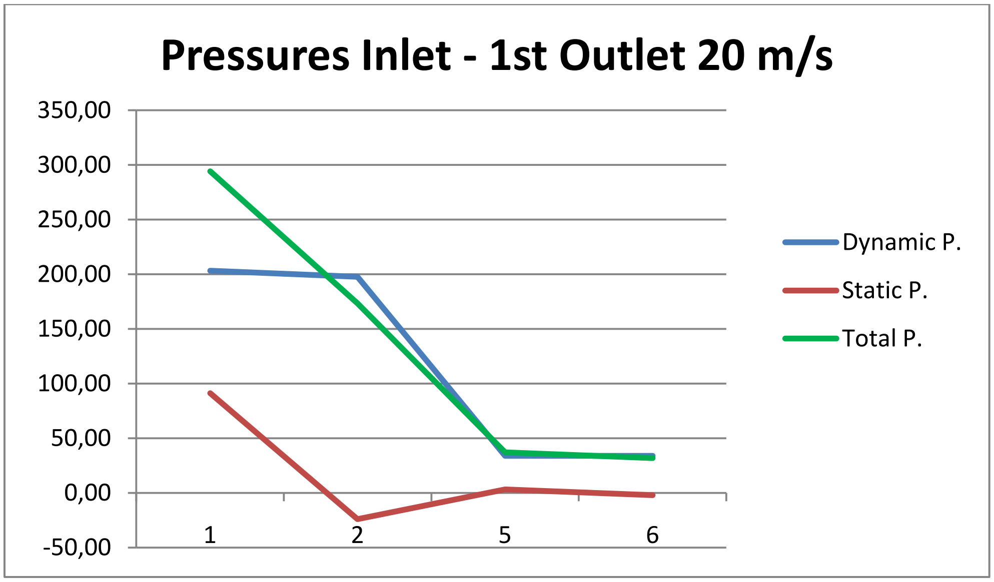
Graphic 37. CFD fluid pressure between inlet and second outlet for a 10 m/s inlet speed.



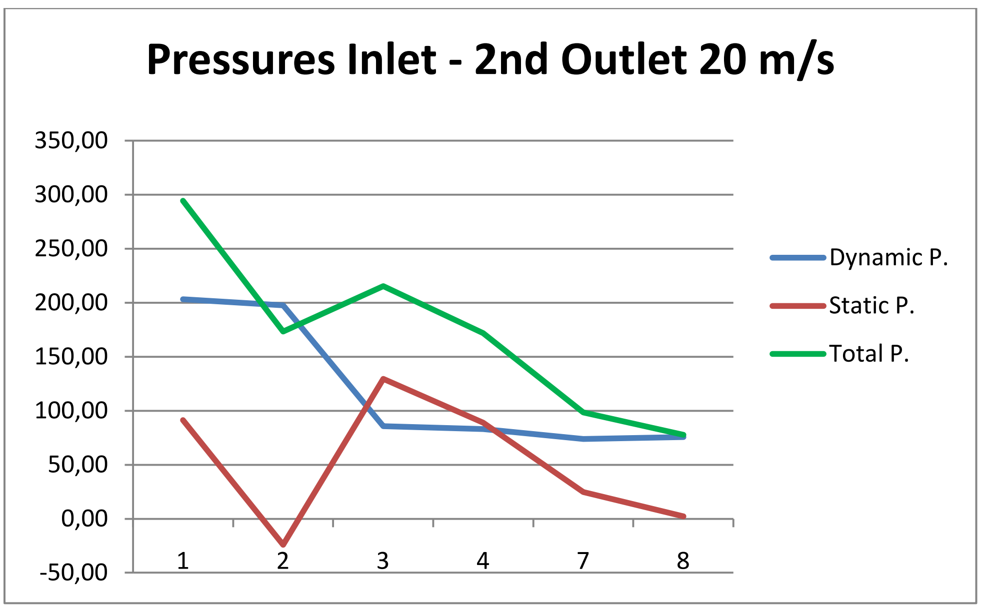
Graphic 38. CFD fluid pressure between inlet and first outlet for a 15 m/s inlet speed.



Graphic 39. CFD fluid pressure between inlet and second outlet for a 15 m/s inlet speed.



Graphic 40. CFD fluid pressure between inlet and first outlet for a 20 m/s inlet speed.



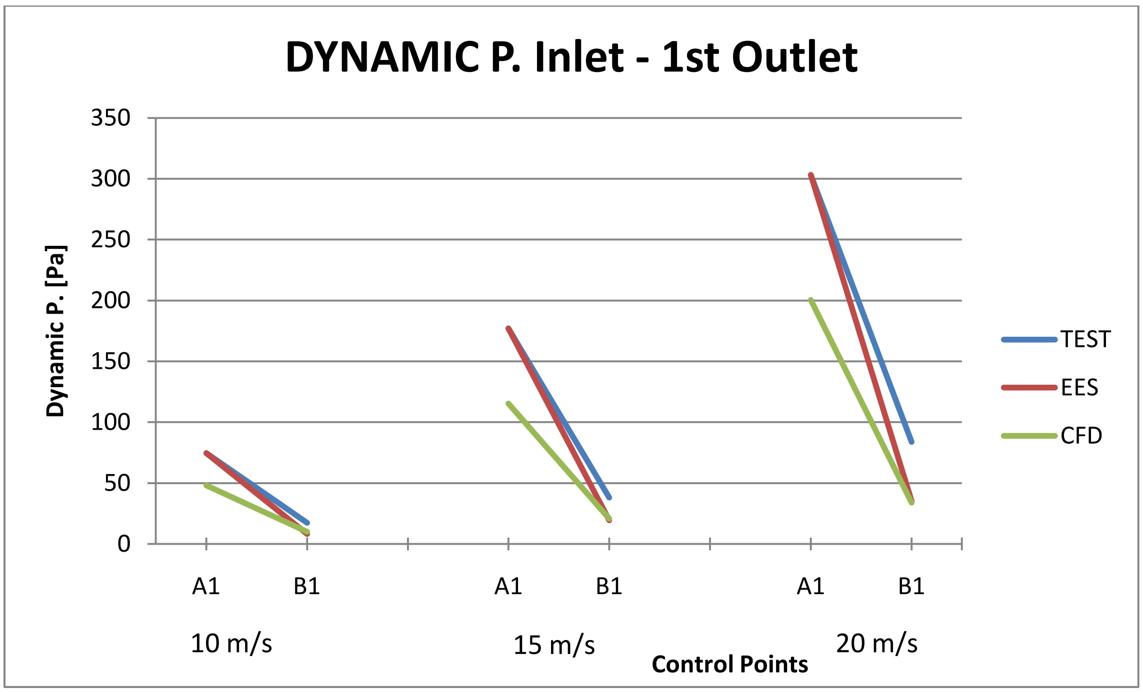
Graphic 41. CFD fluid pressure between inlet and second outlet for a 20 m/s inlet speed.

3.4 NARROW BRANCHES MODEL CONCLUSIONS

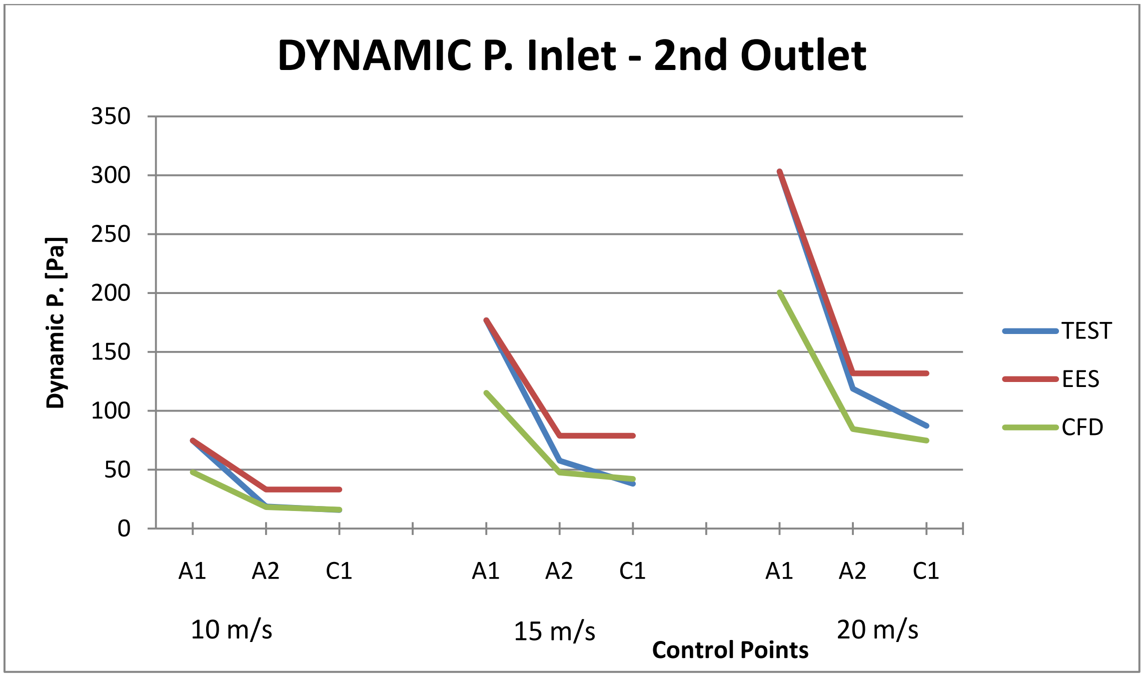
In the following graphics it is shown a comparison between the results of the three different study methods employed for the three different inlet speeds. It is assumed that the test values representation might defer from the other two computational methods, as they both are more accurate in the internal flow analysis.

It is necessary to remark that the analysis are not very accurate, as there are only two points in main branch and one in each branch.

3.4.1 Pressures

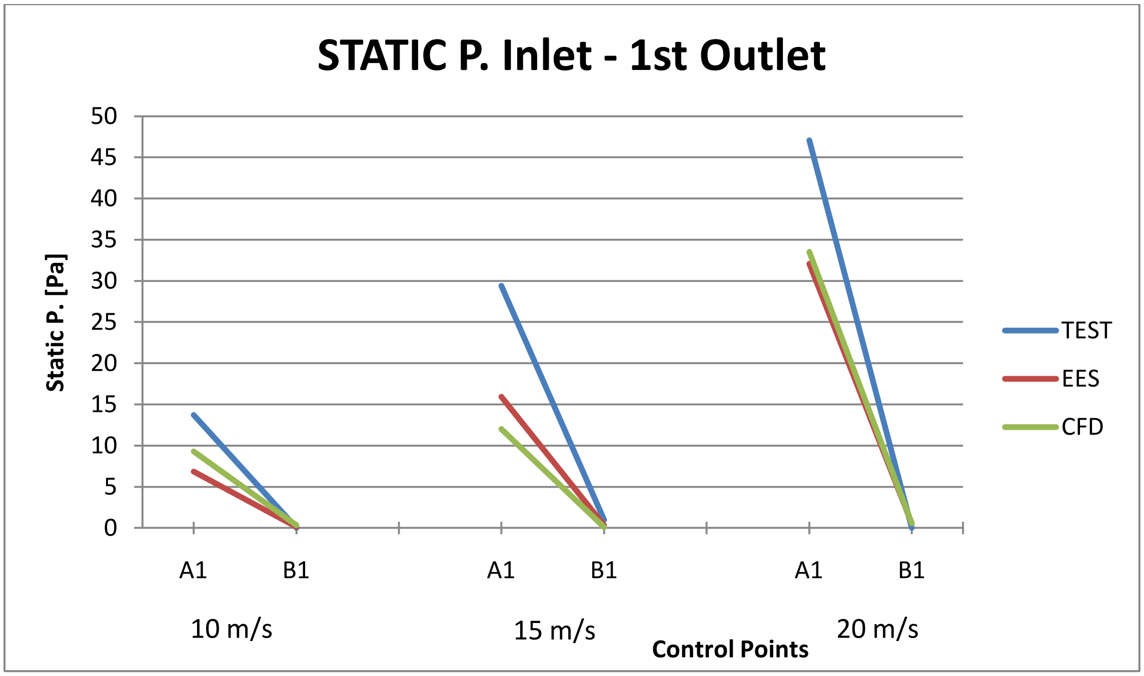


Graphic 42. Dynamic pressure Inlet – 1st Outlet.

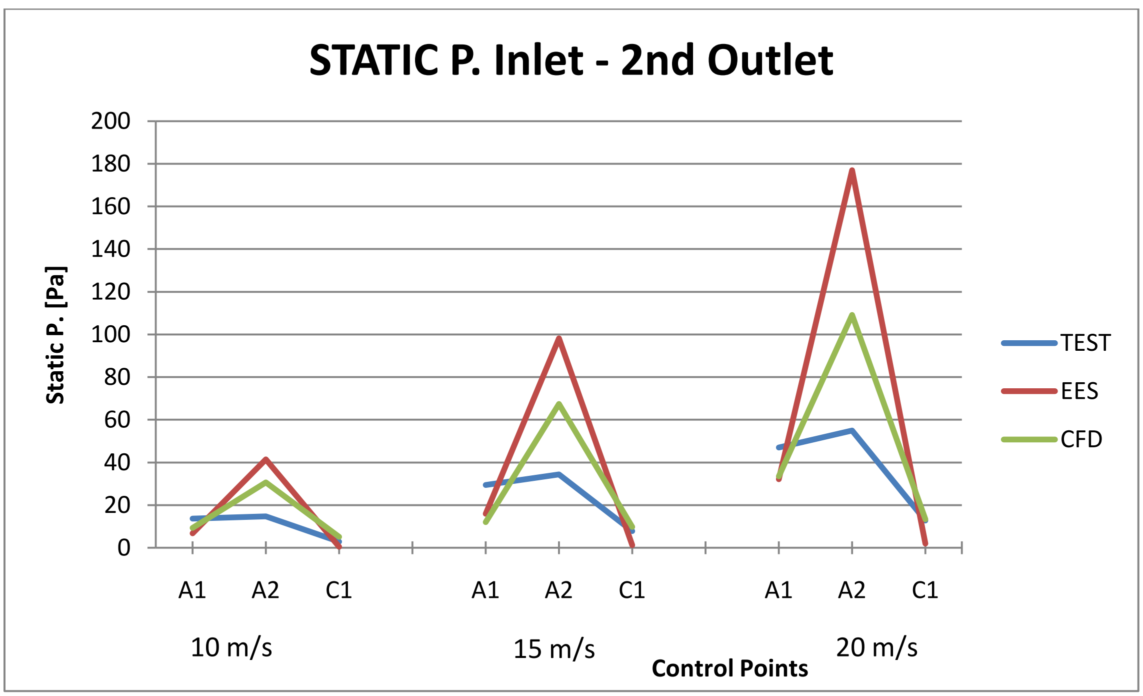


Graphic 43. Dynamic pressure Inlet – 2nd Outlet.

Concerning to the dynamic pressure in first outlet, test and EES values are almost coincident at the beginning, while EES and CFD are almost equal at the end. While in the second outlet are test and CFD the ones that nearly coincide at the end. The reason to explain the fact that the A1 values are different in EES and CFD is that CFD works with average speeds, so the value is lower and so it is the dynamic pressure.

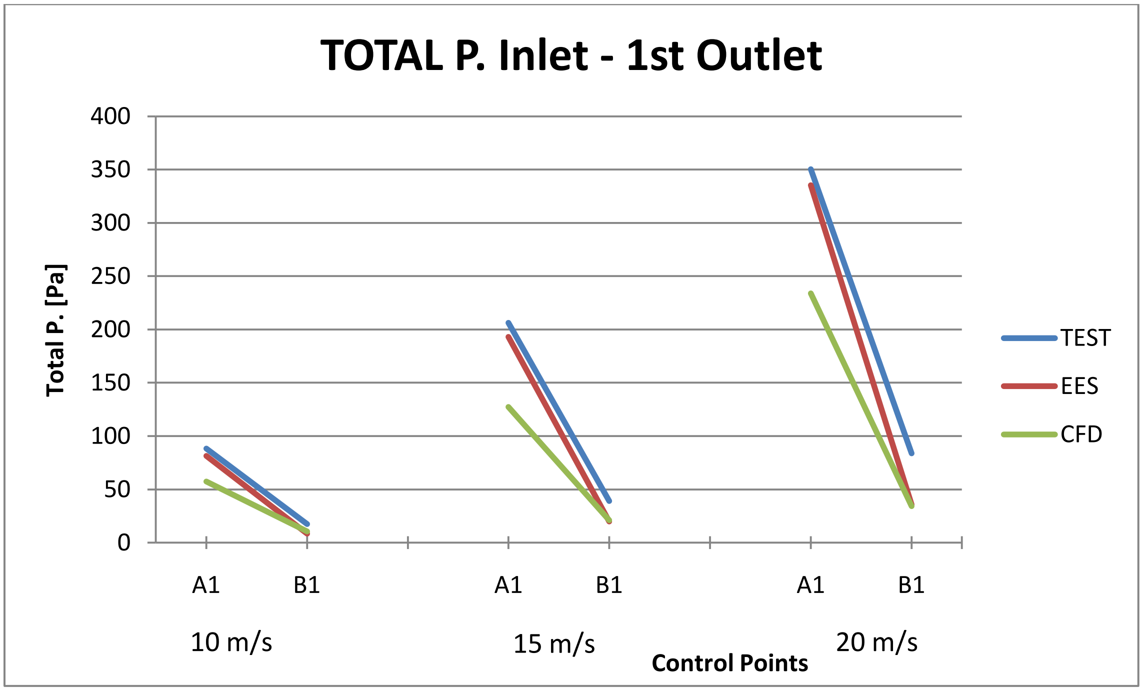


Graphic 44. Static pressure Inlet – 1st Outlet.

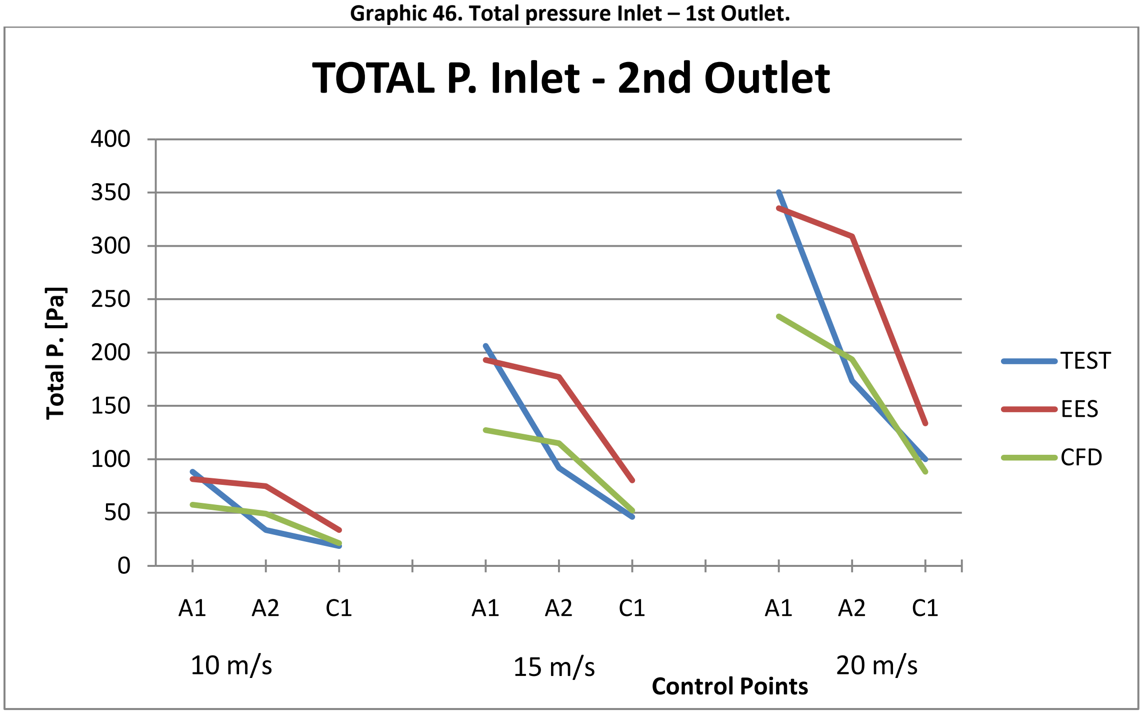


Graphic 45. Static pressure Inlet – 2nd Outlet.

In these graphics there are two facts to be remarked. On the one hand, the test values are always higher than the two softwares, except for A2. On the other hand, EES values are higher than CFD. However, despite these two differences, a similar tendency can be appreciated.



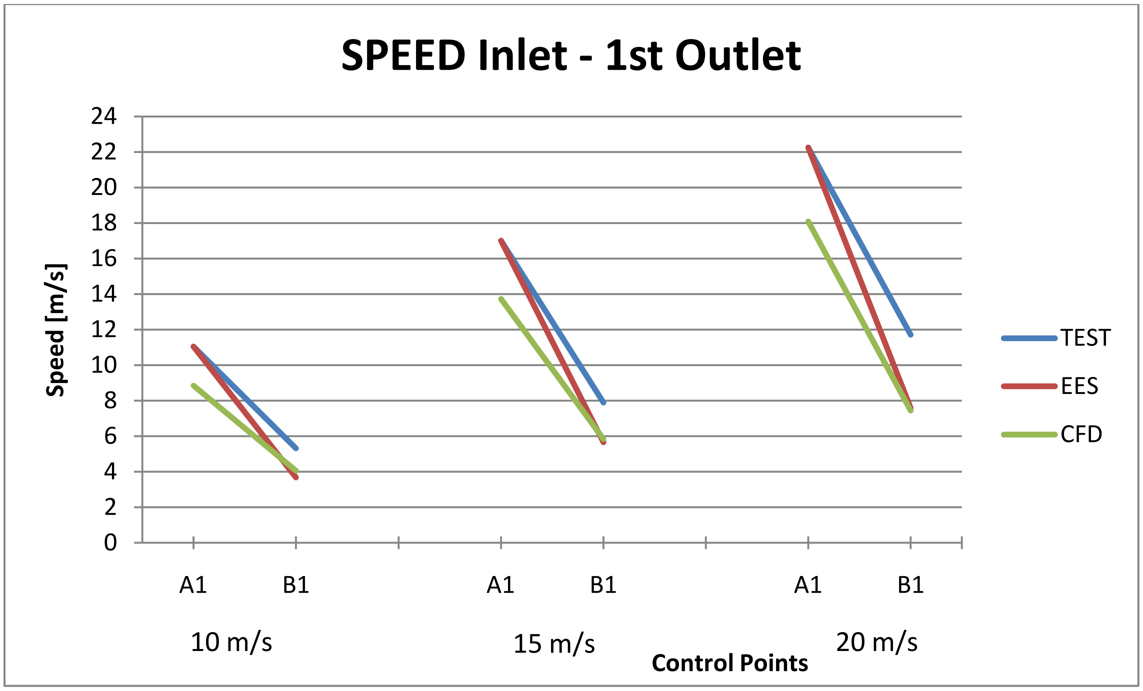
Graphic 46. Total pressure Inlet – 1st Outlet.



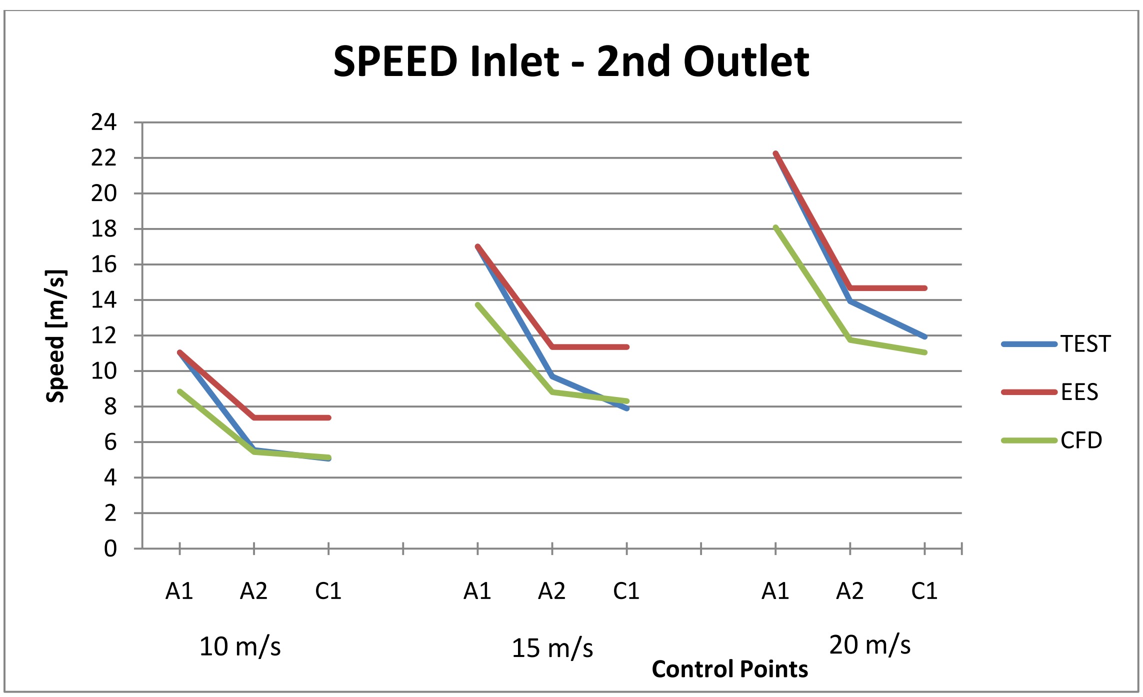
Graphic 47. Total pressure Inlet – 2nd Outlet.

In the case of the graphic which shows the total pressures from the inlet to the first outlet the results are very satisfying because in the three cases al the lines are almost alike. However, in the second graphic the test value shows a different behaviour which is a consequence of the lower values of the static pressure in A2.

3.4.2 Speeds



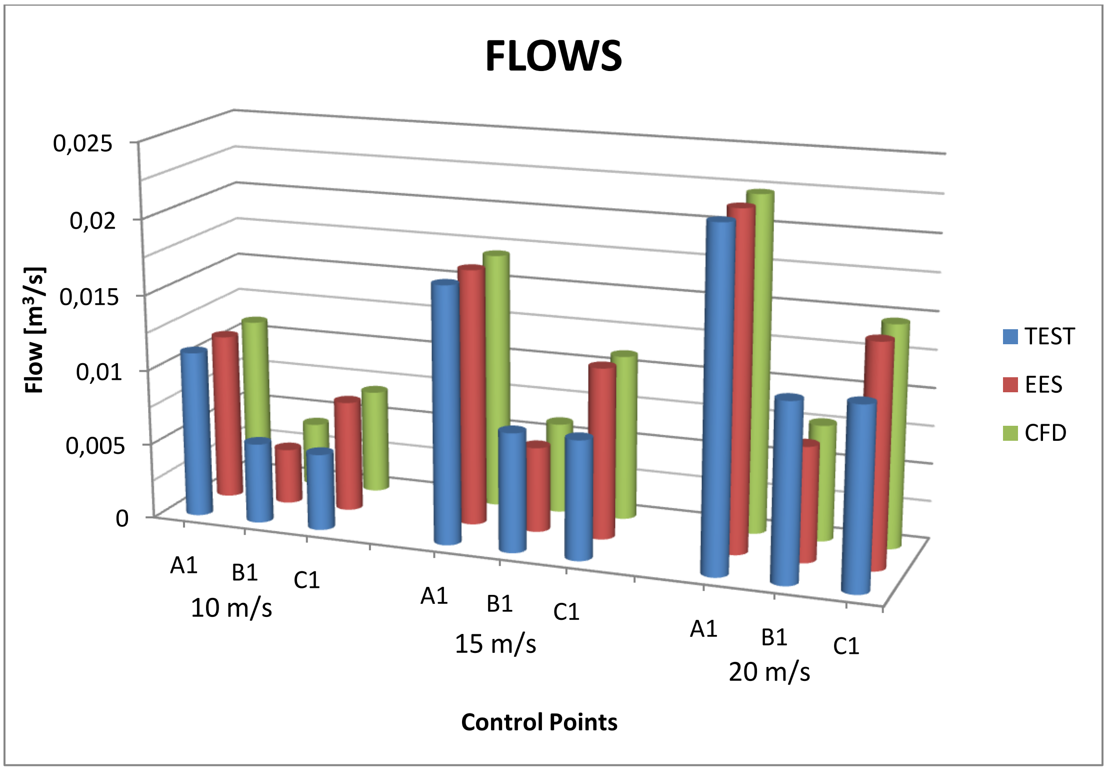
Graphic 48. Speed Inlet – 1st Outlet.



Graphic 49. Speed Inlet – 2nd Outlet.

Regarding to the speeds, the tendency is the same in both graphics. However, in the first graphic inlet speed in CFD is lower than both test and EES, while in the outlet it is the test value the one which is higher than the others. In the second graphic it occurs the same with the difference that in the outlet the higher value is the EES one.

3.4.3 Flows



Graphic 50. Flows in different control points and inlet speeds.

In this graphic, there is a very important factor to be commented: the flow proportion which goes along each branch after the division. It can be observed, that in the computer methods the results are very similar and a bigger amount goes through the second branch. However, in the test results, the air flow through both outlets is the same.

To check this behaviour, a second test was carried out. The dynamic pressure was measured three times in each outlet with a Pitot tube. Once the dynamic pressure was taken, the speed and the flow were calculated. The results are shown in the table below and, as happened in the sphere method, the flow was the same in outlet B and outlet C.

	Outlet B				Outlet C			
Speed (m/s)	mm.w.c	Average	Speed	**Flow**	mm.w.c	Average	Speed	**Flow**
20	3,4 3,8 3,4	3,533	7,523	**0,00752**	3,9 4,2 3,9	4	8,004	**0,00800**
15	1,8 1,8 1,8	1,800	5,369	**0,00537**	1,7 1,9 1,9	1,83	5,419	**0,00542**
10	0,8 0,8 0,7	0,767	3,504	**0,00350**	0,8 0,8 0,8	0,8	3,580	**0,00358**

Table 40. Pitot tube values in outlets.

Considering all the data taken in the three different methods and taking a general overview of the comparisons, we can say that the values obtained in the test are pretty acceptable, and even more if we consider that the sphere method was thought as an orientative tool.

The differences in the results from the EES and the CFD programmes may be due to the sometimes too much theoretical method of the EES and the imprecision of the control area in the CFD. Anyway, these differences between two trustable programmes make more insignificant the distance from the test results to the computational methods.

3.5 SPHERE METHOD IMPROVEMENTS

After studying the results of the tests, it has been necessary to study some improvements to the sphere method. After discussing how this method could be improved so that the results were more accurate four decisions were made.

- In the first test, the inlet dynamic pressure used to calculate the velocity and the flow was measured with the Pitot tube. We realized that it was blocking a 42% of the inlet area and the results obtained could be wrong. So, in the next test the inlet flow should be calculated in a different way. The best option is to calculate the millimeters of water column to be introduced in the scale[2] depending on the inlet speed needed and then check the flow in the manufacturer graphic according to the inlet (A, B or C) employed.
- As the test had been decided to be repeated, a new and more accurate calibration was regarded as very appropriate. The calibration took place in the wind tunnel where the thread with the sphere was hanged from a nail. The nail was hammered into a piece of wood, so that it was placed in the beginning of a scale (see figure 32). When the wind tunnel is turned on, the sphere moves and it can be seen in the scale how big the displacement is in function of the dynamic pressure. Afterwards, the process is the same as in the first test, it is necessary to create a tendency line in Excel.

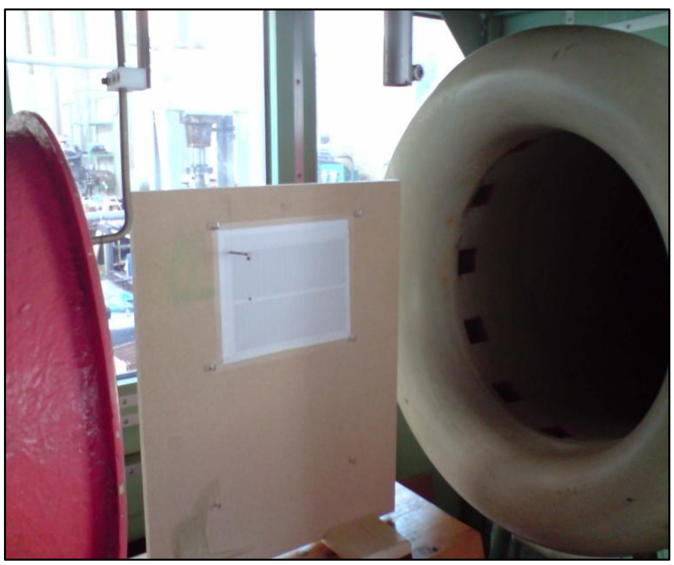


Figure 31. Wind tunnel calibration.

[2] $\Delta h = \frac{c^2 \cdot \rho_{air} \cdot 1000}{\rho_{H2O} \cdot 2 \cdot g}$

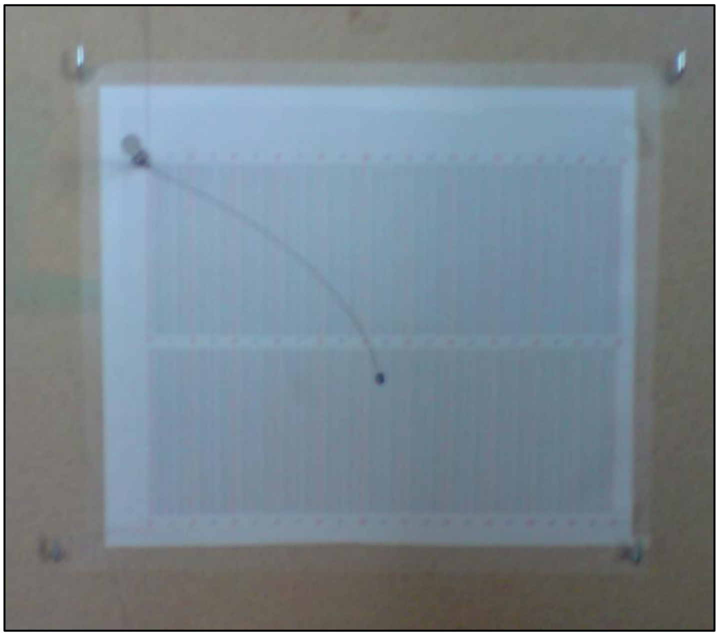


Figure 32. Wind tunnel calibration II.

- There was another point to be revised: the thread length. The first test was made using only a 13cm. thread. So, taking advantage of repeating the test, it was made with three different threads of 6, 13 and 18cm. in order to calculate a tendency line for each one of them and, therefore, a more complete data is obtained.

- Finally, the outlets were modified. Previously, the branches went directly to the atmosphere and, owing to the geometry (a rectangle 5 x 200mm.) it is rather inaccurate to measure the outlet flow. An adaptor made of paper was made to change the rectangular shape into a circular one (d = 4.15 cm, area = 13.52 cm^2). With this adaptor the air flow is more concentrated and the dynamic pressure is easier to be measured.

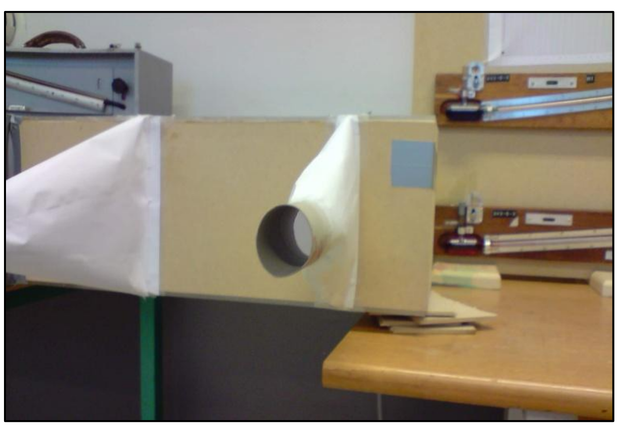


Figure 33. Paper adaptor.

3.5.1 Calibration process

The results of the wind tunnel calibration are resumed in the table below. The dynamic pressure is measured with the Pitot tube installed in the wind tunnel.

	Dynamic Pressure [Pa]				
cm thread	**78.45**	**117.68**	**156.91**	**215.75**	**274.59**
6	17	28	36	43	48
13	74	92	102	113	122
18	106	125	139	154	162
	Ball Displacement [mm]				

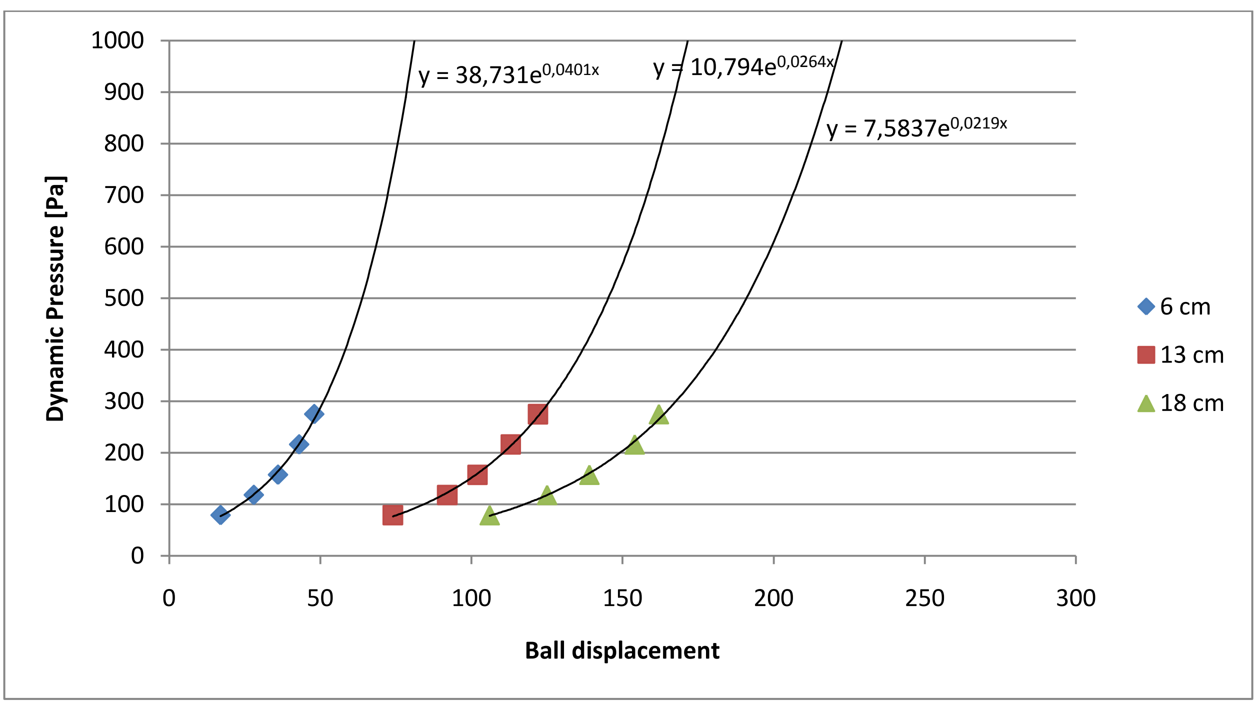
Table 41. Sphere method new calibration.

In the table below it can be seen the speed calculated in function of the measured dynamic pressure.

Dynamic P.	Wind Speed
78.45	11.32
117.68	13.86
156.91	16.01
215.75	18.77
274.59	21.17

Table 42. Speed values.

The table 41 was entered in an Excel sheet and was used to make a graphic with a tendency line for each one of the thread lengths.



Graphic 51. Sphere calibration tendency lines.

Three exponential tendency lines were obtained: $\mathbf{y = 38.071 \cdot e^{0.0401x}}$ in the case of the 6 cm. thread, $\mathbf{y = 10.794 \cdot e^{0.0264x}}$ for 13 cm. and finally, $\mathbf{y = 7.5837^{0.0219x}}$ when the 18 cm. thread is calibrated; being **x** the ball displacement and **y** the dynamic pressure.

3.5.2 Test results

After making the sphere test again, three tables are obtained for the every inlet speed (one table for each thread). The dynamic pressure in the modified outlets was checked with the Pitot tube.

- Inlet Speed = 10 m/s

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
6	10	A1	19	82.97	11.64	0.0116
		A2	7	51.28	9.15	0.0092
		B1	4	45.47	8.62	0.0086
		C1	6	49.27	8.97	0.0090

Table 43. Sphere test. Inlet speed = 10 m/s. 6 cm thread.

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
13	10	A1	69	66.73	10.44	0.0104
		A2	29	23.21	6.16	0.0062
		B1	25	20.88	5.84	0.0058
		C1	27	22.02	5.99	0.0060

Table 44. Sphere test. Inlet speed = 10 m/s. 13 cm thread.

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
18	10	A1	105	75.60	11.11	0.0111
		A2	46	20.77	5.82	0.0058
		B1	42	19.03	5.57	0.0056
		C1	48	21.70	5.95	0.0060

Table 45. Sphere test. Inlet speed = 10 m/s. 18 cm thread.

	Outlet	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
PITOT	B	9.81	4.00	0.0054
	C	12.75	4.56	0.0062

Table 46. Sphere test. Inlet speed = 10 m/s. Outlet measurements.

- Inlet Speed = 15 m/s

Thread [cm]	Inlet Speed [m/s]	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
6	15	A1	30	128.98	14.51	0.0145
		A2	11	60.20	9.91	0.0099
		B1	10	57.84	9.72	0.0097
		C1	11	60.20	9.91	0.0099

Table 47. Sphere test. Inlet speed = 15 m/s. 6 cm thread.

Thread [cm]	Inlet Speed [m/s]	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
13	15	A1	89	113.14	13.59	0.0136
		A2	43	33.59	7.41	0.0074
		B1	37	28.67	6.84	0.0068
		C1	43	33.59	7.41	0.0074

Table 48. Sphere test. Inlet speed = 15 m/s. 13 cm thread.

Thread [cm]	Inlet Speed [m/s]	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
18	15	A1	133	139.59	15.10	0.0151
		A2	72	36.70	7.74	0.0077
		B1	63	30.14	7.01	0.0070
		C1	70	35.13	7.57	0.0076

Table 49. Sphere test. Inlet speed = 15 m/s. 18 cm thread.

	Outlet	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
PITOT	B	17.65	5.37	0.0073
	C	23.54	6.20	0.0084

Table 50. Sphere test. Inlet speed = 15 m/s. Outlet measurements.

- Inlet Speed = 20 m/s

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
6	20	A1	36	164.06	16.37	0.0164
		A2	18	79.71	11.41	0.0114
		B1	13	65.23	10.32	0.0103
		C1	18	79.71	11.41	0.0114

Table 51. Sphere test. Inlet speed = 20 m/s. 6 cm thread.

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
13	20	A1	105	172.60	16.7869	0.0168
		A2	66	61.64	10.0322	0.0100
		B1	57	48.61	8.9084	0.0089
		C1	64	58.47	9.7708	0.0098

Table 52. Sphere test. Inlet speed = 20 m/s. 13 cm thread.

Thread [cm]	**Inlet Speed [m/s]**	Control Point	Ball displ. [mm]	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
18	20	A1	151	207.034	18.39	0.0184
		A2	100	67.76	10.52	0.0105
		B1	91	55.64	9.53	0.0095
		C1	99	66.29	10.40	0.0104

Table 53. Sphere test. Inlet speed = 20 m/s. 18 cm thread.

	Outlet	Dynamic P. [Pa]	Speed [m/s]	Flow [m^3/s]
PITOT	B	29.42	6.93	0.0094
	C	39.23	8.00	0.0108

Table 54. Sphere test. Inlet speed = 20 m/s. Outlet measurements.

All the data concerning to the air flows calculations and measurements shown in the previous tables is resumed in the table 55.

3.5.3 Velocity curves

As a in the new test three different threads were used, the possibility of studying the shape of the velocity curve appeared. According to the theory this curve has to be a parabola with velocity equal to zero in the walls and maximum velocity in the center of the conduit (see the figure below).

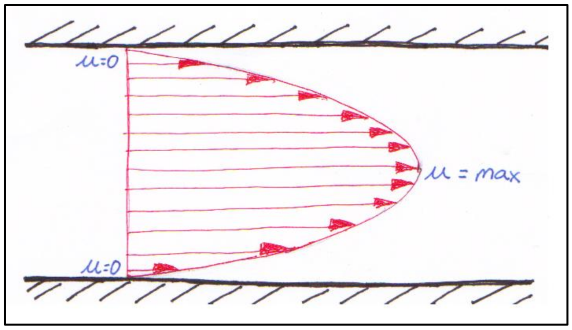


Figure 34. Velocity profile.

To do the study the inlet and outlets speed values were studied with the three thread lengths (see figure 35):

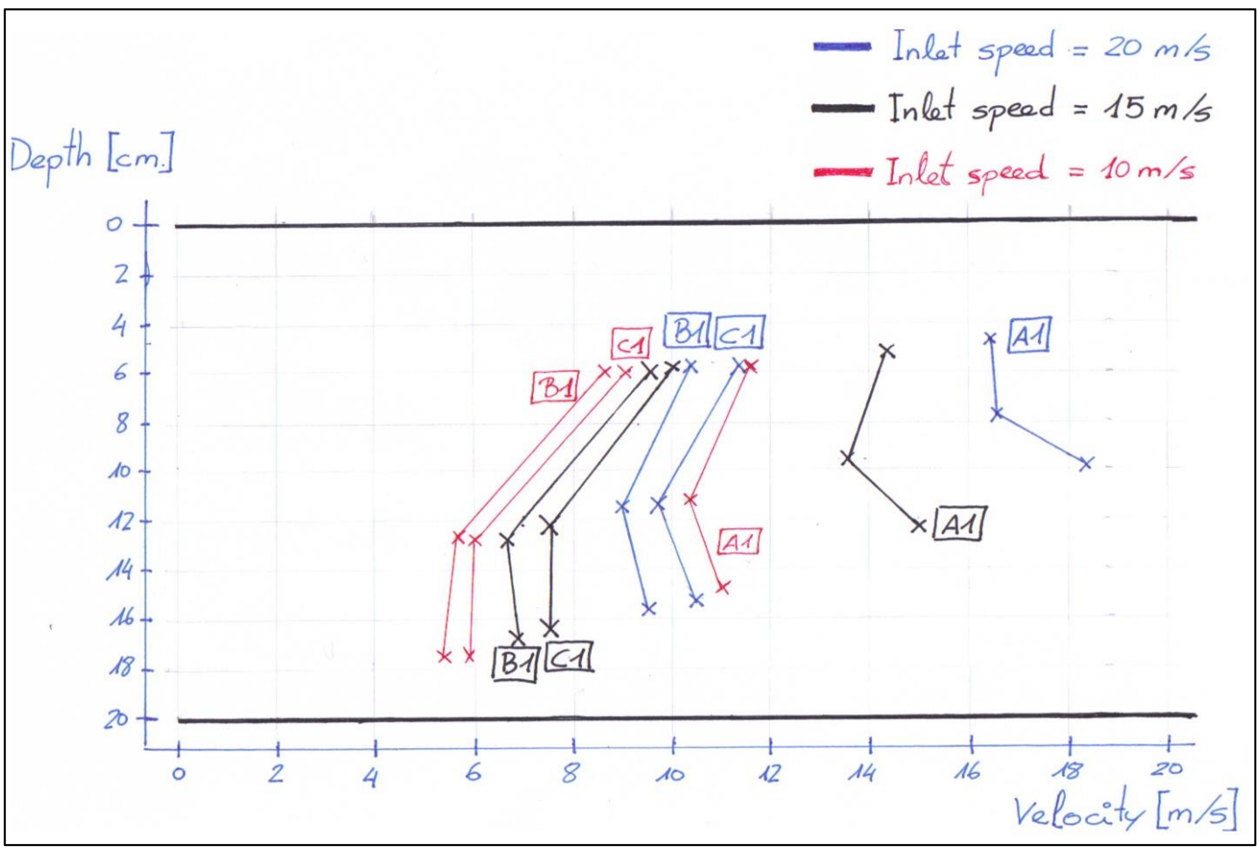


Figure 35. Sphere method velocity profiles.

Having a look at the figure 35, it can be concluded that the sphere method cannot be employed to draw the speed parabola. However, a tendency can be observed. In the three cases the speed is higher in A1 than in C1 and C1 is higher than B1, as well. In the outlets (B1 and C1) the lines are very similar, with a higher speed for the 6 cm. thread. The 13 and 18 cm. threads give a very similar result for low velocities.

3.5.4 Flow analysis

This table is very useful to compare all the inlet and outlets flows. The fan inlet flow observed in the manufacturer graphic is shown in the third column. The column called “Inlet [theory]” is the result of multiplying the inlet speed by the branch area (0.001 m^2). All the results of the sphere tests are shown with the word “ball” between square brackets, as well as the Pitot measurements are indicated with the word “Pitot”.

		Flow [m^3/s]								
Inlet Speed [m/s]	**Thread [cm]**	**Inlet [manufact.]**	**Inlet [theory]**	**Inlet [ball]**	**Outlet B [ball]**	**Outlet C [ball]**	**Outlets sum [ball]**	**Outlet B [Pitot]**	**Outlet C [Pitot]**	**Outlet sum [Pitot]**
10	6	**0.0096**	**0.0100**	**0.0116**	0.0086	0.0090	**0.0176**	0.0054	0.0062	**0.0116**
	13			**0.0104**	0.0058	0.0060	**0.0118**			
	18			**0.0111**	0.0056	0.0060	**0.0115**			
15	6	**0.0146**	**0.0150**	**0.0145**	0.0097	0.0099	**0.0196**	0.0073	0.0084	**0.0156**
	13			**0.0136**	0.0068	0.0074	**0.0142**			
	18			**0.0151**	0.0070	0.0076	**0.0146**			
20	6	**0.0191**	**0.0200**	**0.0164**	0.0103	0.0114	**0.0217**	0.0094	0.0108	**0.0202**
	13			**0.0168**	0.0089	0.0098	**0.0187**			
	18			**0.0184**	0.0095	0.0104	**0.0199**			

Table 55. Sphere test. Flows comparison.

In general, it can be concluded that the results are very satisfying in most of the cases. The results are all around a 10-15 % error respecting to the theoretical values. The worst cases occur when the 6 cm. thread is tested in the outlets (especially with an inlet speed of 10 and 15 m/s); however, these cases are not over 20% error. The explanation to this error is might due to the instability of the sphere during the test, as it vibrates too much to be measured accurately.

Another inaccuracy can be observed when the inlet flow is measured with the sphere method when the inlet speed value is 20 m/s. The most likely explanation is that the drag force on the thread is too high to be neglected with high speeds, so it could affect the calibration and the test itself.

An expected change in the results, in comparison with the first sphere test, is that here it can be appreciated that the flow through the second branch is higher than in the first one, as EES and CFD had shown. This behaviour has already been explained before as a consequence of the air tendency to continue going straight when it circulates along a conduit, what happens when the main branch is divided.

In conclusion, the sphere method has proved to be a practical method more reliable than expected (especially with the 13 and 20 cm. threads), as it can be seen when it is compared with the Pitot measurements, manufacturer graphic and theoretical flow.

3.5.5 Test vs. EES and CFD

The difference between this table and the previous one is that here the EES and CFD results are also shown (see the last four columns). It makes the comparison with the test results easier.

		Flow [m^3/s]								
Inlet Speed [m/s]	**Thread [cm]**	**Inlet [manufact.]**	**Outlet B [ball]**	**Outlet B [Pitot]**	**Outlet B [EES]**	**Outlet B [CFD]**	**Outlet C [ball]**	**Outlet C [Pitot]**	**Outlet C [EES]**	**Outlet C [CFD]**
10	6	0.0096	0.0086	0.0054	0.0031	0.0037	0.0090	0.0062	0.0061	0.0059
	13		0.0058				0.0060			
	18		0.0056				0.0060			
15	6	0.0146	0.0097	0.0073	0.0050	0.0053	0.0099	0.0084	0.0096	0.0093
	13		0.0068				0.0074			
	18		0.0070				0.0076			
20	6	0.0191	0.0103	0.0094	0.0065	0.0067	0.0114	0.0108	0.0126	0.0125
	13		0.0089				0.0098			
	18		0.0095				0.0104			

Table 56. Sphere test. Flows comparison with EES and CFD.

Despite the test results were very satisfying, when they are compared with the softwares these are a bit different, as in both EES and CFD the air flow in the second outlet is around the double than in the first inlet. Only the outlet C results are very close in all cases. However, the outlet B measurements are rather different in comparison to the programs, as the measurements are around a 50% higher.

After checking all the previous studies (wide and narrow branches), it can be seen that this error takes place in all cases, so it can be concluded that maybe the theory is different from the practice in some aspects and, as EES and CFD works according to the theory, this explains the different values, especially in the first branch.

4 PRESSURE LOSS IN A TUBE SYSTEM

The aim of this chapter is to show a comparison between the results obtained in the same exercise with three different softwares: EES (Engineering Equation Solver), SolidWorks Flow Simulation Studio and CFDesing.

The exercise consists of calculating the pressure loss in a tube system as shown below:

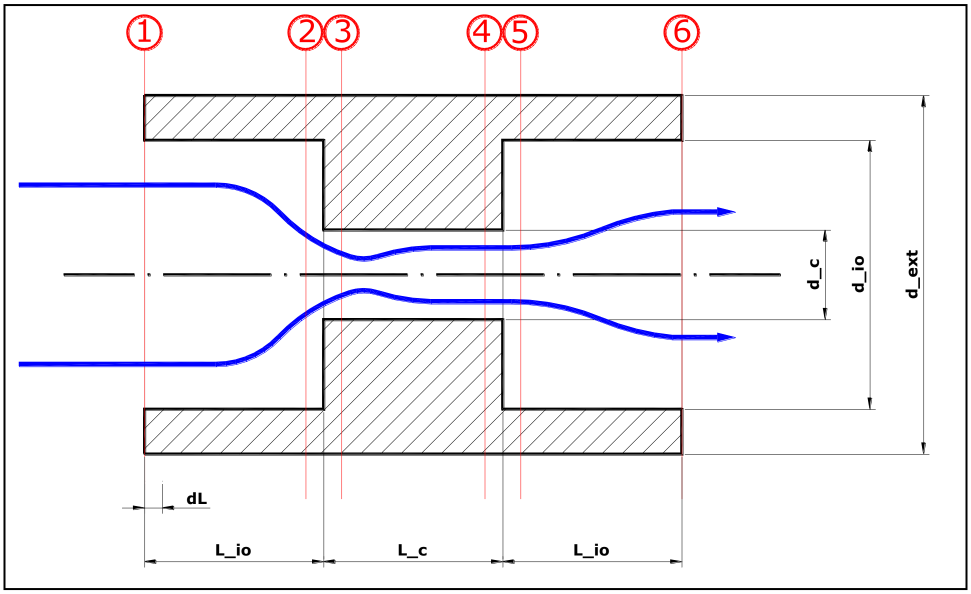


Figure 36. Statement.

The figure 36 shows a tube where the diameter of the centre part is reduced

The "edges" at the change of diameters are to be considered as"sharped edged"!

Geometry:
d_c = 20 mm
d_io = 40 mm
d_ext = 50 mm (But this is not used)
L_c = 10 x d1 = 200 mm
L_io = 10 x d2 = 400 mm
Roughness of the tube: k = 0 mm (i.e. "smooth")

Properties of fluid:
Medium: Water
Temperature: 20^0 C => Density = 1000 kg/m^3 and Kin. visc. = $1.0 \cdot 10^{-6}$ m^2/s

States:
States 1 and 6: Placed at the inlet and outlet openings
State 2, 3, 4 and 5: Placed ± 5mm upstream and downstream the contraction/expansion

Inlet:

Speed at inlet u_1 = 0.25 m/s

Speed distribution at inlet: Even distributed, i.e. 0.25 m/s all over the cross flow area.

To do:

Calculate

- Static and total pressure (= p + ½ ρ u^2) in all 6 states, where the static pressure in state 1 is equal to 0 Pa
- Pressure loss from 1 to 6

Solution

Static pressure 1 should be 0 Pa according to the problem instructions. However, in both CFD programs there is no possibility of setting two boundary conditions in the same point, so static pressure equal to 0 Pa is set as an outlet (state 6) boundary condition and speed equal to 0.25 m/s is set as an inlet boundary condition.

	EES		SolidWorks FS		CFDesign	
State	Static P.	Total P.	Static P.	Total P.	Static P.	Total P.
1	629.8	661.05	794.45	821.77	697.64	728.89
2	620.10	651.35	746.65	794.93	677.90	711.7
3	-48.63	451.37	71.93	402.42	105.60	732.8
4	-177.80	322.20	-170.88	185.70	-131.17	462.88
5	9.68	40.93	-183.25	-143.57	-170.64	329.36
6	0.00	31.25	0.00	27.27	0.00	33.8
Pressure loss 1_6	**629.80**		**794.5**		**695.09**	

Table 57. Static and Total pressure values.

Having a look at table 57, it can be noticed that in SolidWorks Flow Simulation and in CFDesign the tendency is exactly the same, but SolidWorks Flow Simulation calculates higher values. However, in the Equations Editor Solver, the results are very different because, while in the CFD programs a depression in states 4 and 5 can be appreciated, in EES is in 3 and 4 where this depression values are obtained.

The reason to explain this behaviour is that EES is only based on Mathematics and all the states are defined just as a group of equations. As it can be seen in graphic 52 the pressure drop after entering in the narrowing is almost immediate. There is no transition time from one pressure to the low one, and the pressure decreases almost 700 Pa just after the 0.4 meters value in the X-axis.

In both CFD programs the pressure decreasing takes a little bit longer than in the EES, and it is more progressive. This explains the differences between the values from the Equation solver and the other two programs in point 3. As you can appreciate in figure 42 it is really important the point where you place the control points in the CFD programs, especially in the narrowing. Only a few millimetres can make the pressure reading vary up to 100 Pa.

At the widening, in state 5, there is an interesting difference in the static pressure results. There are negative values for both CFD programs; however, this value is positive in EES, as it can be seen in table 57. This is due to the fact, as explained before, that while in EES the pressure just after the area increasing changes suddenly, in CFDesign and SolidWorks Flow Simulation it is still decreasing and takes more time to start increasing again.

Concerning to the velocities, it can be appreciated a very similar behaviour to the static pressure reading. The velocity in state 5 is very different depending on the program employed. In CFDesign the velocity lasts more time than in the other programs to decrease in the widening. This fact, explains the high total pressure value in 5. However, in state 3, the velocity increases very fast when CFDesign is read.

In EES, the speed values change from 0.25 to 1 m/s in less than 1 cm. That makes the dynamic pressure values differ to the ones read from the CFD programs, and consequently the total pressure.

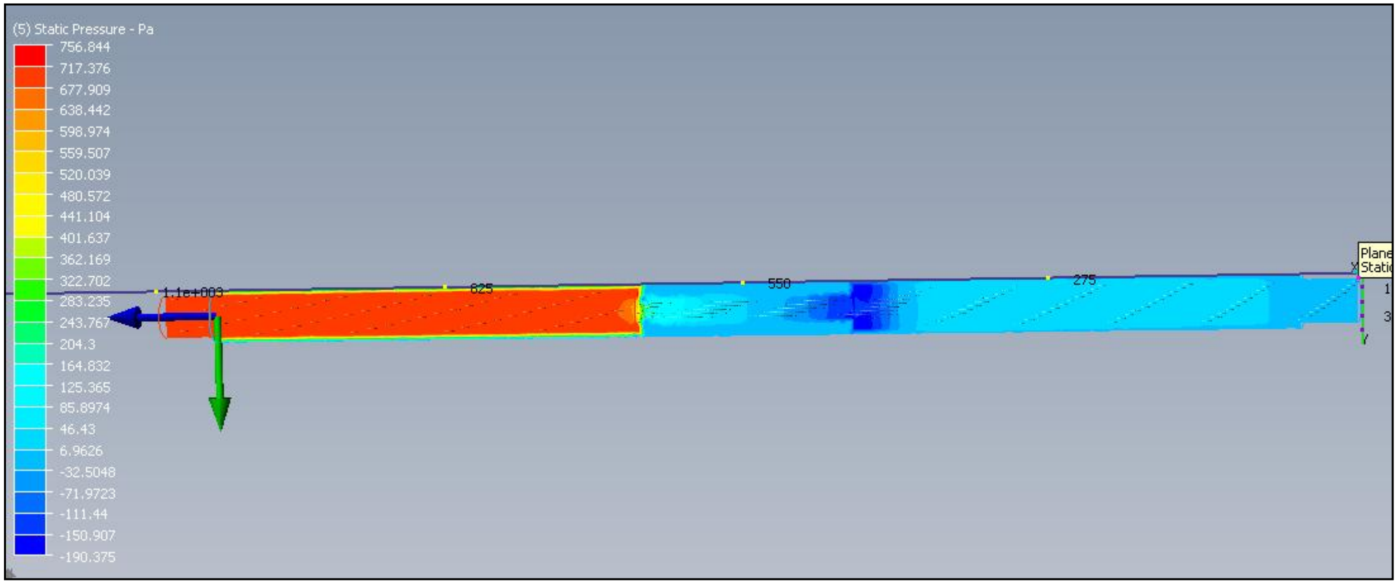


Figure 37. CFDesign Static pressure plot.

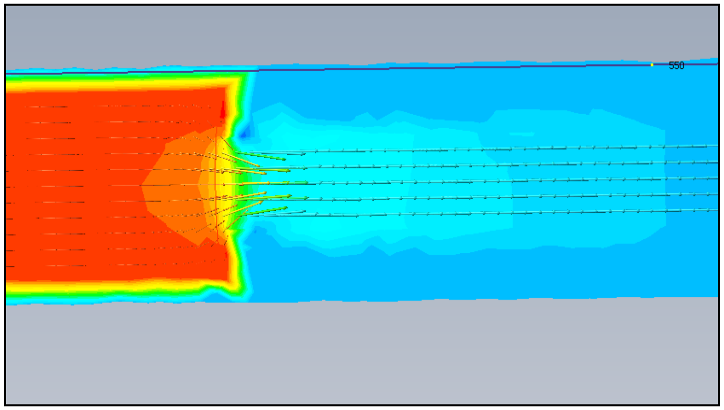


Figure 38. CFDesign narrowing Static pressure plot.

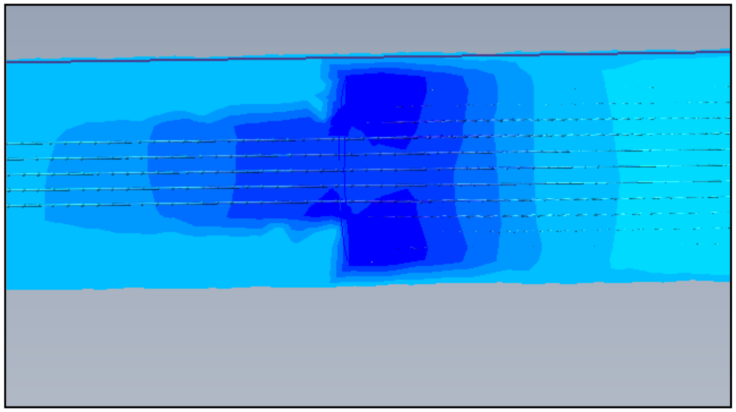


Figure 39. CFDesign widening Static pressure plot.

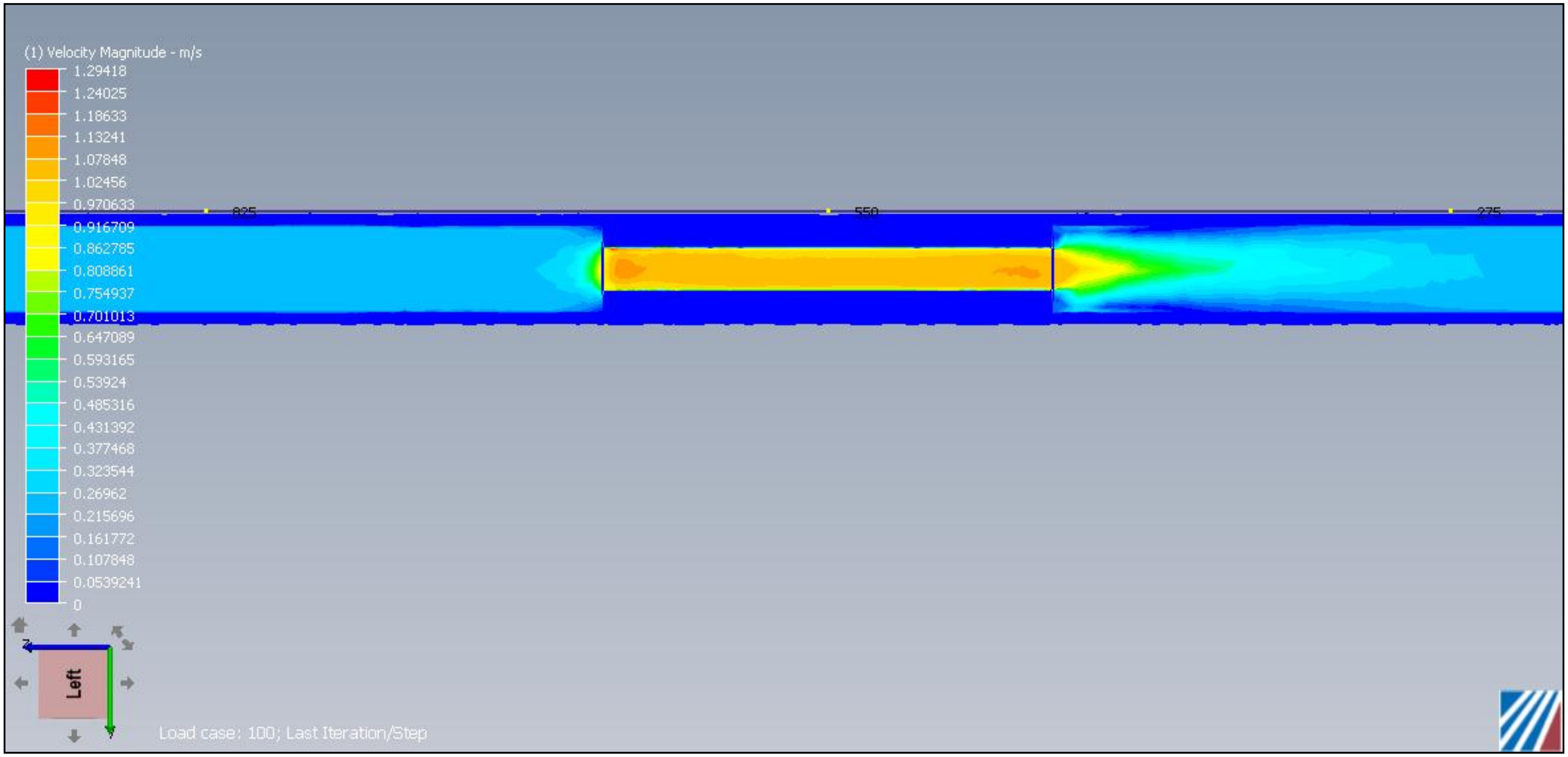


Figure 40. CFDesignn Velocity plot

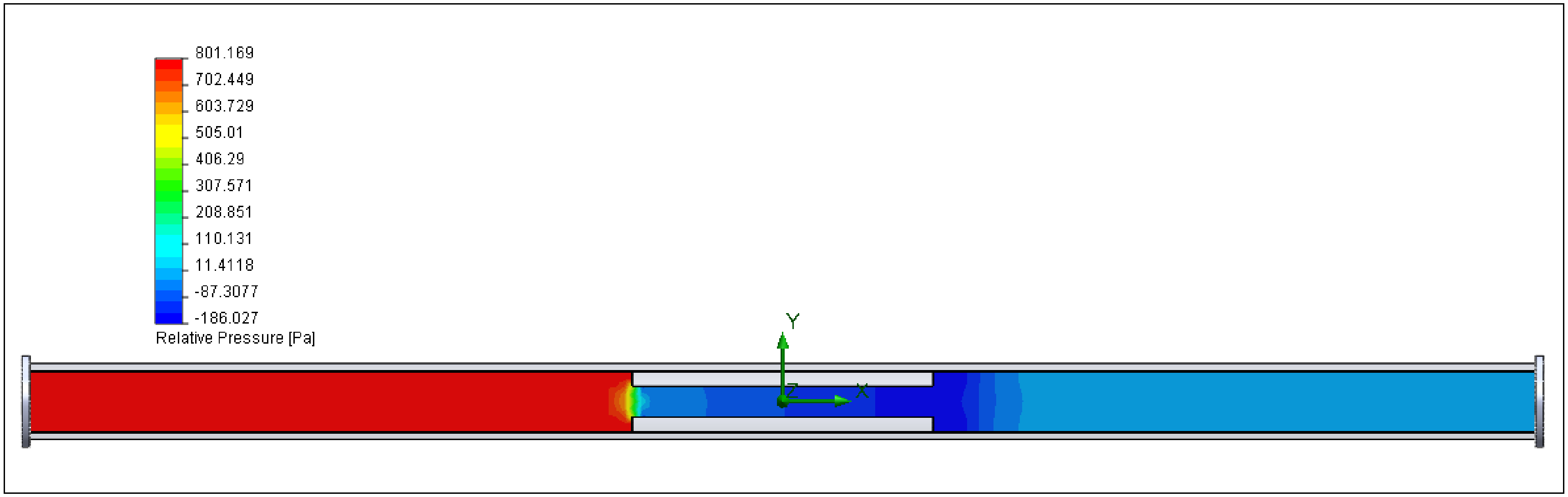


Figure 41. SolidWorks FS Static pressure plot.

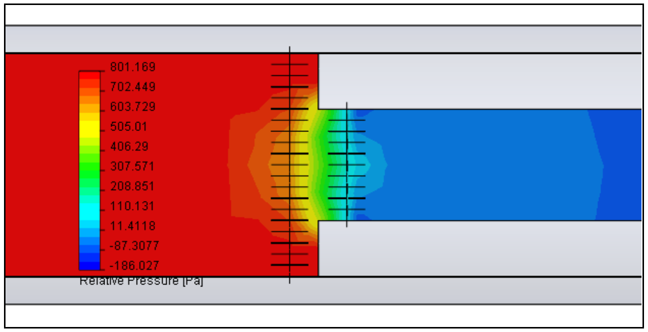


Figure 42. SolidWorks FS narrowing Static pressure plot.

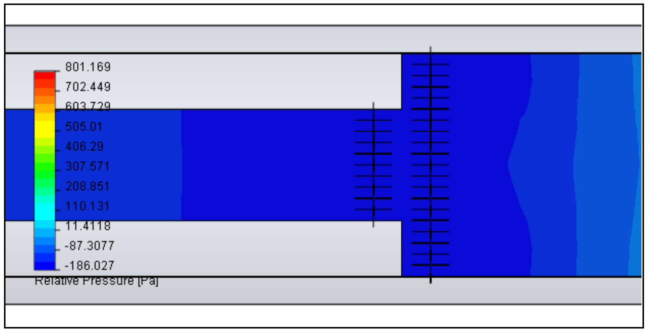


Figure 43. SolidWorks FS widening Static pressure plot.

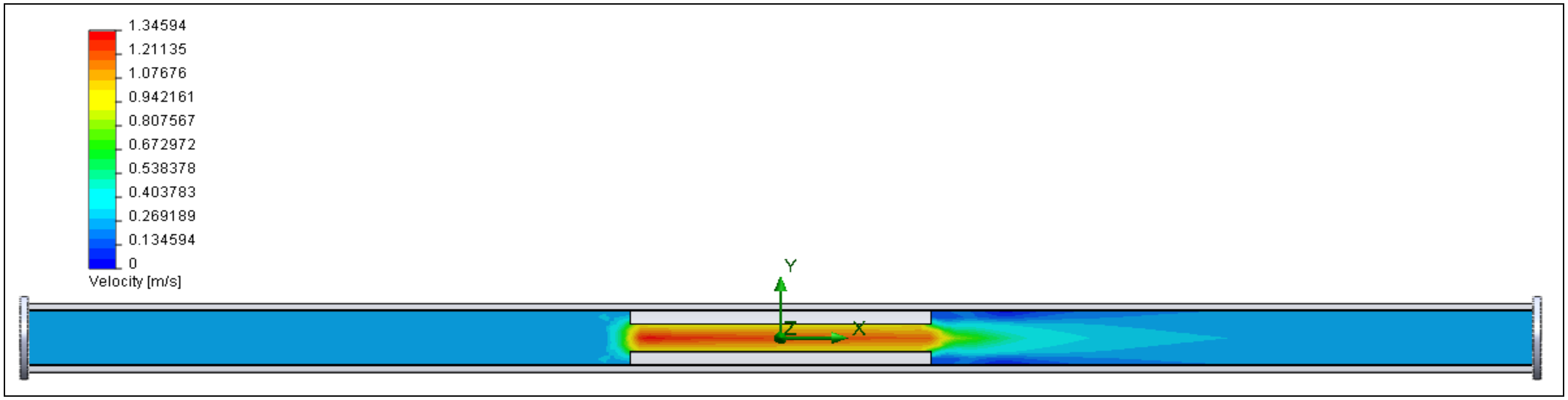


Figure 44. SolidWorks FS Velocity plot

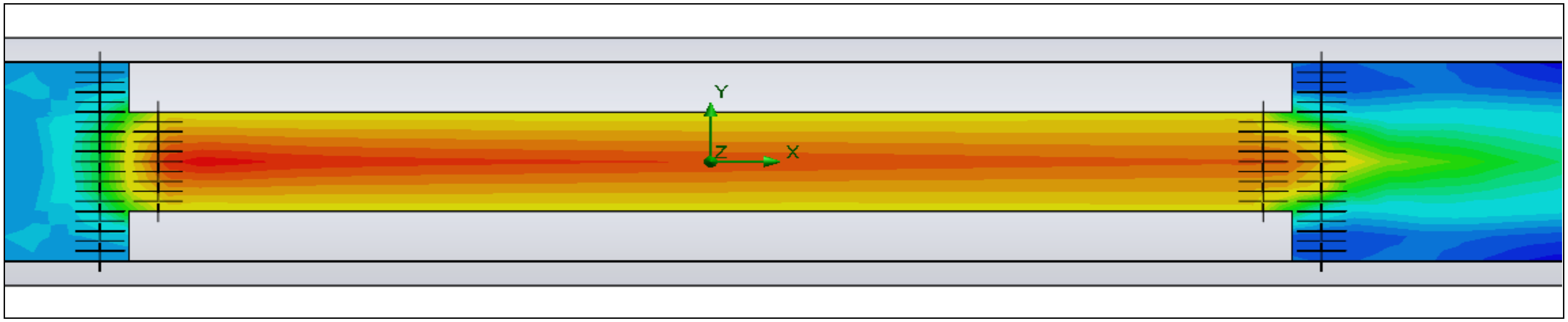
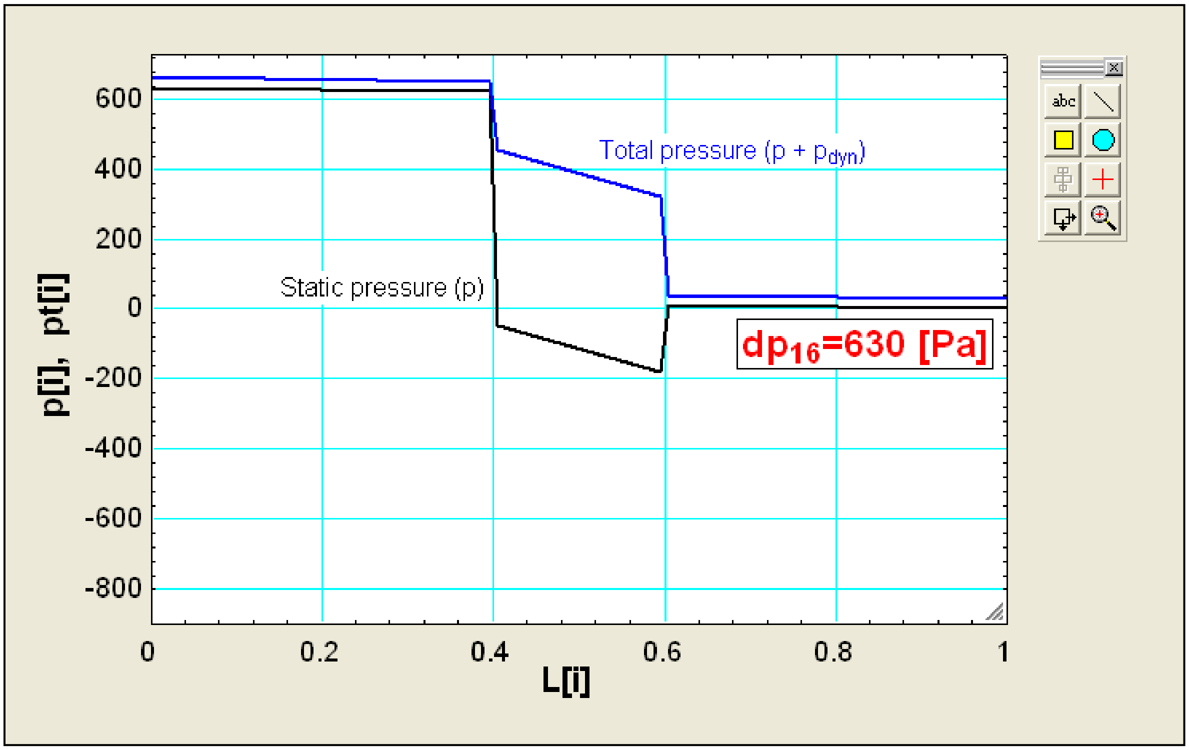
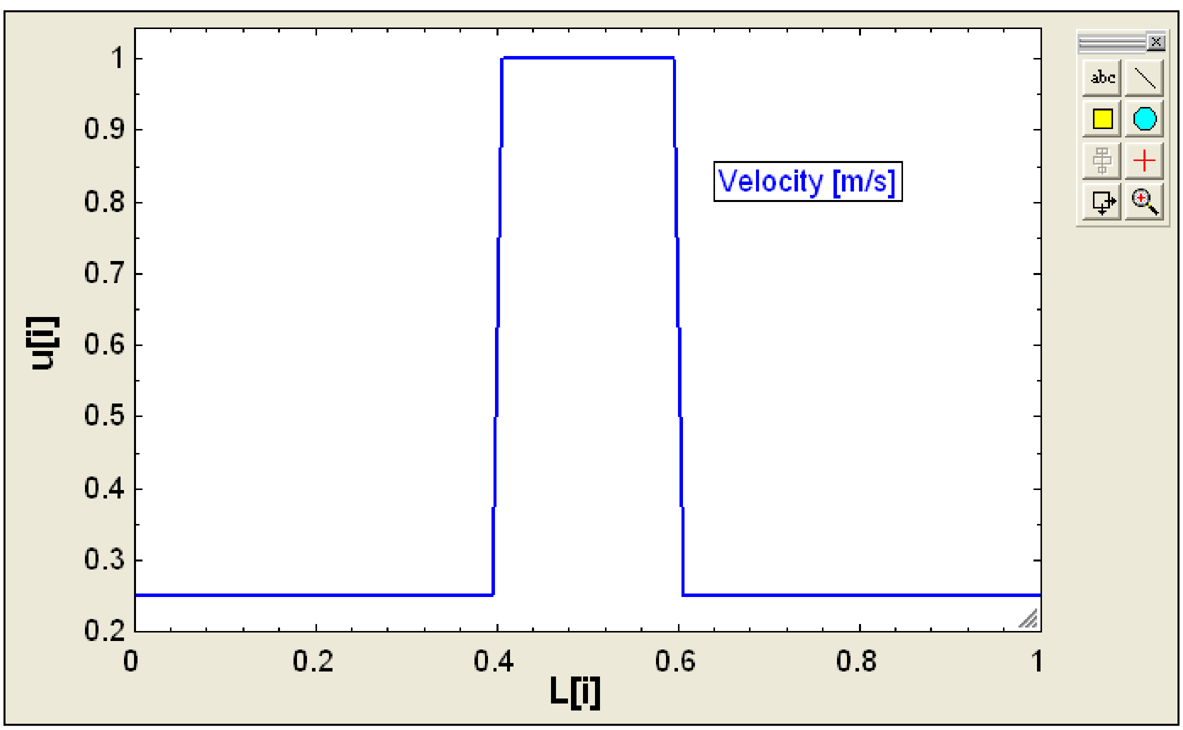


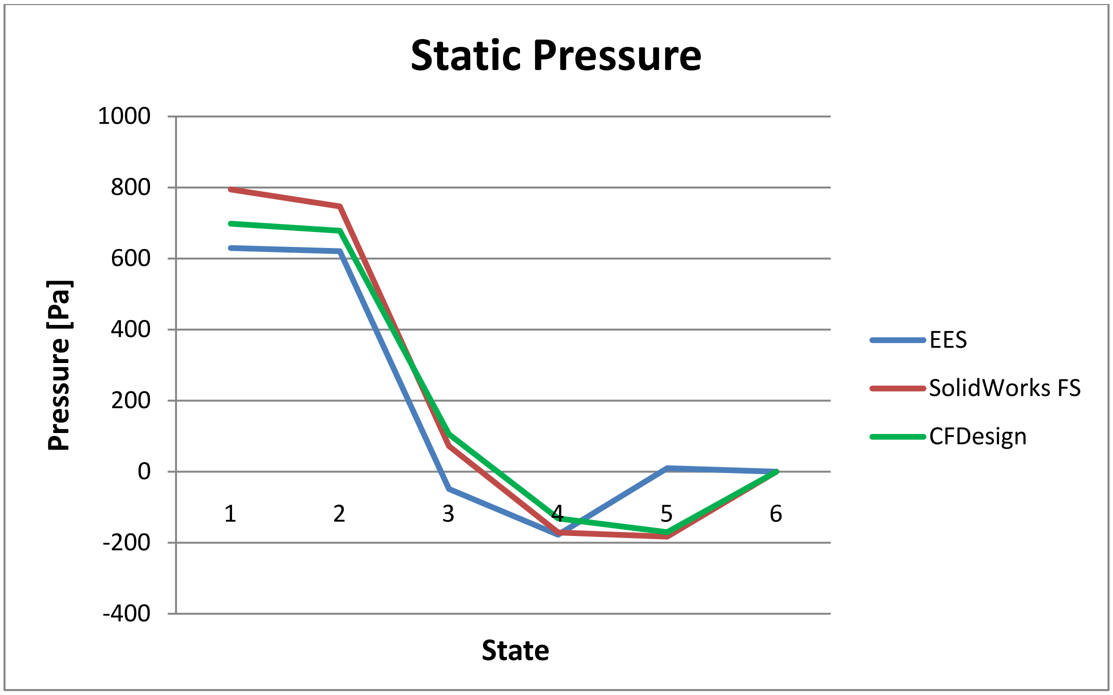
Figure 45. SolidWorks FS Centre part Velocity plot



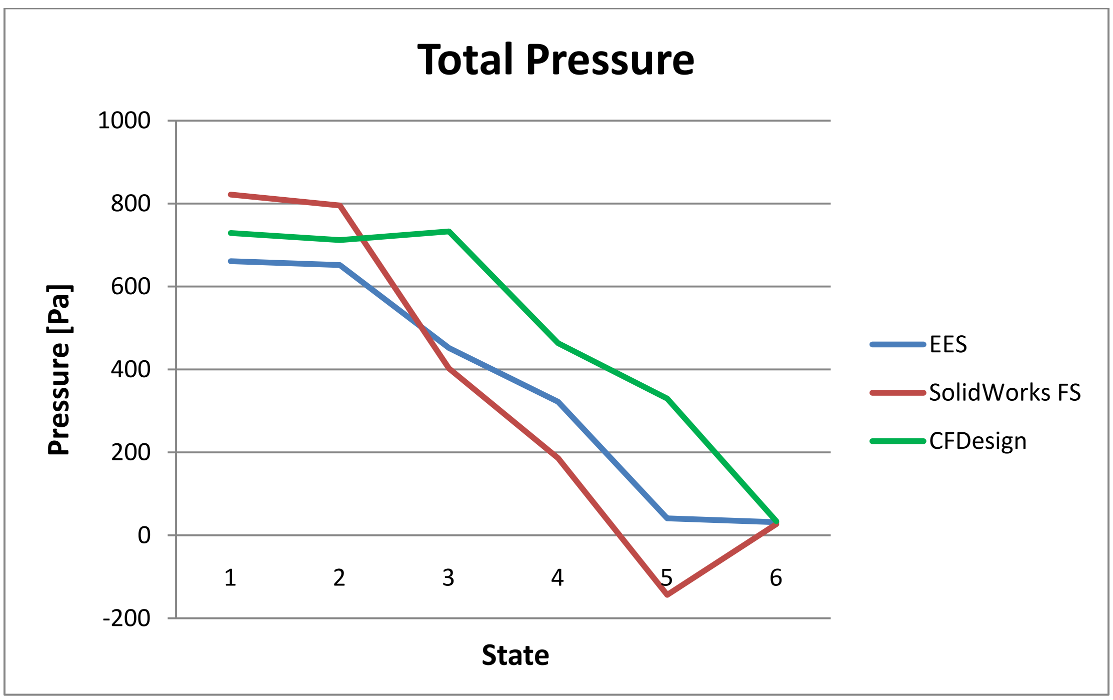
Graphic 52. EES Static and Total pressure values



Graphic 53. EES Velocity values.



Graphic 54. Static pressure comparison.



Graphic 55. Total pressure comparison.

In the previous graphics the static pressure values are very similar in the three cases, with the only difference on state 5 when it is read in EES. The total pressure graphics are not as coincident because the speed changes are not studied in the same way: in EES the speed changes almost immediately when the area varies; in CFDesign the changes are slower than in SolidWorks Flow Simulation when the velocity decreases, but they are faster when the velocity increases.

To conclude, we can say that even working with two CFD programs which are supposed to use the same principles when simulating a flow study, the results obtained are similar, but not the same (for example, the pressure loss difference is around 12.5%). This variation between two similar programs could be explained as a consequence of the mesh accuracy employed. The difference with EES is due to the fact that EES is just a mathematical program which does not take into account points such as the fluid inertia and the speed curve shape.

5 PROJECT CONCLUSIONS

Once the project is over, is time to make an overview, analyse and take some conclusions about what we have been doing for the last half year.

After studying two different two-branched models and analyzing the values in the outlets, we can say that the results are not the expected before doing the project, as we thought that the air flow would be the same in both outlets when their geometry is the same. However, once studied the first model results, we realized that the flow inertia along the main branch takes part on the flow distribution, so the second outlet flow rate is higher than the first one. If we had studied a bigger model with higher number of branches, we could have understood a little better this interesting behaviour of the air flow in the "T" divisions.

It must also be commented that the models assembly exceeded all the expectations. As we used materials such as wood pieces, plastic plates, screws and washers, the isolating methods were pretty homemade using bicycle tires trimmings for the first model, and the assembly was made by non-professional people, we thought there would be a lot of flow loss between the joints along the conduits comparing the inlet and the two outlets. However, there was a little flow loss in both models, which probably took place in the diffusion box.

Another point to be explained is the result differences depending on the tool employed in the study: EES, CFD or test. In our opinion, the different results are due to the fact that both programs use a lot of theoretical equations to obtain the results, especially EES, which actually is just an equations solver. SolidWorks Flow Simulation goes one step further and shows a more real analysis than EES, but is still closer to the theory than to the reality.

We could not conclude the project conclusions without expressing our satisfaction with the results obtained by employing the sphere method. At the beginning we were not sure at all that it would work. Although we were very confident on the method, we were not sure to be able to make it accurate enough to get some satisfactory results. After analysing the first test results, we noticed that with some improvements this method would be accurate enough and we were encouraged by the supervisors to do so.

When we were told to make this project, the initial goal was to design a cooling system for a generator. After the semester has past we have realized the complexity and the time you have to spend to make a project of this importance. So far, we have understood how the air behaves when it is divided and now should be time to start evaluating the cooling power of the flow. This could take, for us, another entire project.

Perhaps we would have liked to get to know how this kind of projects are faced and developed on a company, and work in close cooperation with them. But anyway, the IHA environment proved to be very professional, serious and we have had everything we needed to make our project succeed.

To conclude, the last five months we have learnt to work with two different programmes, new for us, and analyse the results and compare it to a real test. This has shown us the importance of complementing the theoretical analysis with practical tests. Moreover, because of the magnitude of the project, we have learnt to divide and assign tasks to the components of the group, and to work in an international environment.

6 BIBLIOGRAPHY

- Donald S. Miller, *Internal Flow Systems*
BHRA Fluid Engineering
1978

- D.J. Tritton, *Physical Fluid Dynamics*
Van Nostrand Reinhold Company Ltd
1977

- Joseph P. DeCarlo, *Fundamentals of flow measurement*
Instrument Society of America
1984

- Richard W. Miller, *Flow measurement engineering handbook*
McGraw-Hill Publishing Company
1983

- Solid Works Flow Simulation 2009 Online Tutorial

- P. Charbonneau, B. Knapp, *Engineering Equation Solver for Microsoft Windows Operating Systems*
Online Tutorial.

7 INDEX OF ELEMENTS

7.1 INDEX OF FIGURES

7.2 INDEX OF GRAPHICS

7.3 INDEX OF TABLES

APPENDICES

A. TEST APPENDICES

In order to develop the analysis of wide branches model test's results, many values were taken during the test. The followings are all these values, ordered according to the different depths in the control points.

1/8		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	8,6	7,4	16	--	--	--
	A1	0	1,2	1,2	9,5	1,1	10,6	7,3	0,8	8,1
	A2	6,2	1,2	7,4	7,2	1,2	8,4	6,6	1	7,6
	A3	6,3	4,8	11,1	6,9	4,3	11,2	5,4	4	9,4
	A4	7,2	4,5	11,7	6,9	4,4	11,3	4,8	4,3	9,1
	A5	6,6	4,6	11,2	6,8	4,7	11,5	5,4	4,5	9,9
	A6	6,4	4,2	10,6	6,5	4,6	11,1	5,5	5	10,5
	B1	-3,8	-2,3	-6,1	1,7	-2,7	-1	8,4	-2,5	5,9
	B2	0	0	0	2,5	0	2,5	4,5	0,1	4,6
	B3	2,1	0	2,1	2,4	0	2,4	3,1	0	3,1
	O1	--	--	--	1,9	0	1,9	--	--	--
	C1	-3,5	-0,4	-3,9	6,6	-0,4	6,2	7	-0,4	6,6
	C2	2,8	0,1	2,9	5,4	0,2	5,6	5,9	0,1	6
	C3	3,6	0	3,6	4,9	-0,1	4,8	6	0	6
	O2	--	--	--	3,8	0	3,8	--	--	--

1/4		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	9,3	7,4	16,7	--	--	--
	A1	3,2	1	4,2	8,2	1	9,2	6,6	0,7	7,3
	A2	7,2	1,2	8,4	8	1,2	9,2	6	1	7
	A3	7	5	12	7,5	4,4	11,9	5	4,1	9,1
	A4	7,5	4,5	12	7,2	4,5	11,7	4,8	4,4	9,2
	A5	6,9	4,6	11,5	6,8	4,6	11,4	5,4	4,5	9,9
	A6	6,5	4,2	10,7	6,9	4,5	11,4	5,5	4,8	10,3
	B1	-3,8	-2,5	-6,3	-3,6	-2,7	-6,3	8,6	-2,3	6,3
	B2	-0,3	0	-0,3	1,9	-0,1	1,8	4,3	0,1	4,4
	B3	1,8	0	1,8	1,8	0	1,8	1,8	0	1,8
	O1	--	--	--	1,8	0	1,8	--	--	--
	C1	-3,3	-0,5	-3,8	6,1	-1	5,1	7	-0,7	6,3
	C2	2	0,2	2,2	4	0,2	4,2	5,8	0,3	6,1
	C3	2,8	-0,1	2,7	3,4	-0,1	3,3	4,4	-0,1	4,3
	O2	--	--	--	3,4	0	3,4	--	--	--

3/8		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	10	7,8	17,8	--	--	--
	A1	3,4	1	4,4	10	1	11	6,6	0,7	7,3
	A2	7,2	1,2	8,4	7,8	1,3	9,1	6,6	1	7,6
	A3	6,8	5,2	12	7,8	4,5	12,3	5,1	4,1	9,2
	A4	7,8	4,4	12,2	7,4	4,5	11,9	4,7	4,4	9,1
	A5	7,1	4,6	11,7	7,1	4,6	11,7	5,5	4,6	10,1
	A6	6,7	4,4	11,1	6,9	4,6	11,5	5,5	4,8	10,3
	B1	-3,9	-2,1	-6	-2	-2,5	-4,5	9,1	-2	7,1
	B2	-0,2	0,2	0	0,9	0	0,9	0,2	5,3	5,5
	B3	1,4	0	1,4	1,4	0	1,4	1,4	0	1,4
	O1	--	--	--	1,8	0	1,8	--	--	--
	C1	-3,2	-0,5	-3,7	7,2	-1	6,2	7,4	-0,6	6,8
	C2	1,8	0,4	2,2	3,6	0,3	3,9	6,4	0,4	6,8
	C3	2,7	-0,1	2,6	3,5	0	3,5	5,1	0	5,1
	O2	--	--	--	3,4	0	3,4	--	--	--

1/2 (I)		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	10,5	7,4	17,9	--	--	--
	A1	2,6	1	3,6	10,4	1	11,4	5,4	0,8	6,2
	A2	6,9	1,2	8,1	7,8	1,3	9,1	6,6	1,1	7,7
	A3	6,4	5,1	11,5	7,8	4,7	12,5	5	4	9
	A4	7,8	4,4	12,2	7,4	4,4	11,8	4,8	4,3	9,1
	A5	6,9	4,6	11,5	7,2	4,6	11,8	5,6	4,6	10,2
	A6	6,8	4,3	11,1	6,8	4,7	11,5	5,7	5	10,7
	B1	-3,8	-2,1	-5,9	-3	-1,9	-4,9	9,2	-1,8	7,4
	B2	0,1	0,2	0,3	3,3	0	3,3	6,4	0.3.	6,4
	B3	1,6	0	1,6	1,6	0	1,6	1,6	0	1,6
	O1	--	--	--	2,3	0	2,3	--	--	--
	C1	-2,6	-0,4	-3	7,2	-1	6,2	7,8	-0,4	7,4
	C2	1,7	0,5	2,2	3,2	0,4	3,6	7	0,6	7,6
	C3	2,2	0,1	2,3	3,4	0	3,4	5,3	0	5,3
	O2	--	--	--	3,4	0	3,4	--	--	--

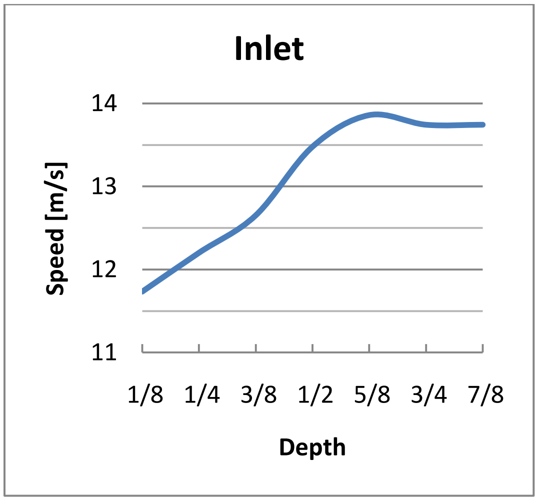
1/2 (II)		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	12,2	7,5	19,7	--	--	--
	A1	6,2	1,7	7,9	11,6	1,5	13,1	8,2	1,5	9,7
	A2	8,6	1,8	10,4	10,4	1,8	12,2	8,8	2	10,8
	A3	7,8	7	14,8	8,8	5,5	14,3	7,4	4,5	11,9
	A4	9	5,2	14,2	8,8	5,1	13,9	5,9	5,1	11
	A5	8,8	5,5	14,3	8,4	5,5	13,9	6,9	5,6	12,5
	A6	8	5	13	8,5	5,2	13,7	7	5,8	12,8
	B1	-4	-1,3	-5,3	7,9	-2,7	5,2	8,6	-1,7	6,9
	B2	-0,2	0,4	0,2	2,4	0	2,4	5,8	0,2	6
	B3	1,2	0	1,2	2,3	0	2,3	3	-0,1	2,9
	O1	--	--	--	2	0	2	--	--	--
	C1	-3,2	-0,2	-3,4	5,2	-0,4	4,8	7,8	-0,4	7,4
	C2	1,5	0,7	2,2	4,8	0,4	5,2	6,9	0,7	7,6
	C3	2,4	0	2,4	3,3	0	3,3	5,1	0	5,1
	O2	--	--	--	3,2	0	3,2	--	--	--

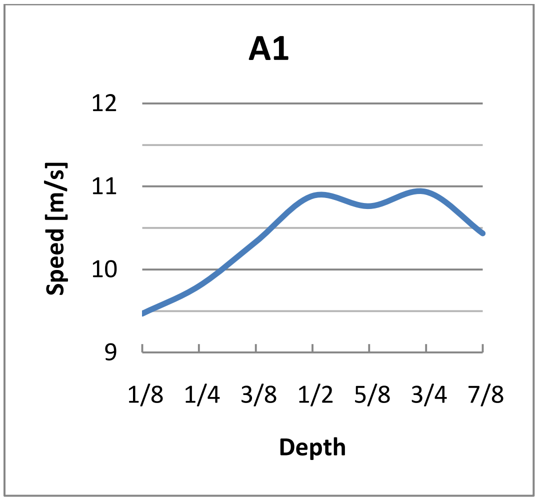
5/8		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	12	7	19	--	--	--
	A1	1,5	1,5	3	11,4	1	12,4	8,8	1,1	9,9
	A2	8,8	1,7	10,5	9,5	1,6	11,1	8,3	1,7	10
	A3	7,7	6,8	14,5	8,7	5	13,7	7	4,3	11,3
	A4	9	5,3	14,3	8,4	5	13,4	5,7	5,1	10,8
	A5	8,3	5,4	13,7	8,4	5,5	13,9	7	5,6	12,6
	A6	7,9	5	12,9	8,2	5,1	13,3	6,8	6	12,8
	B1	-4,2	-1,4	-5,6	7	-2,1	4,9	9,1	-2,1	7
	B2	0,1	0,4	0,5	2,4	0,3	2,7	5,2	0,3	5,5
	B3	1,4	0	1,4	2,2	0	2,2	3	0	3
	O1	--	--	--	2,2	0	2,2	--	--	--
	C1	-3,4	-0,3	-3,7	7,8	-0,6	7,2	7,8	-0,4	7,4
	C2	1,4	0,6	2	3,6	0,4	4	3,8	0,6	4,4
	C3	2,6	0	2,6	3,5	0	3,5	5,3	0	5,3
	O2	--	--	--	3,2	0	3,2	--	--	--

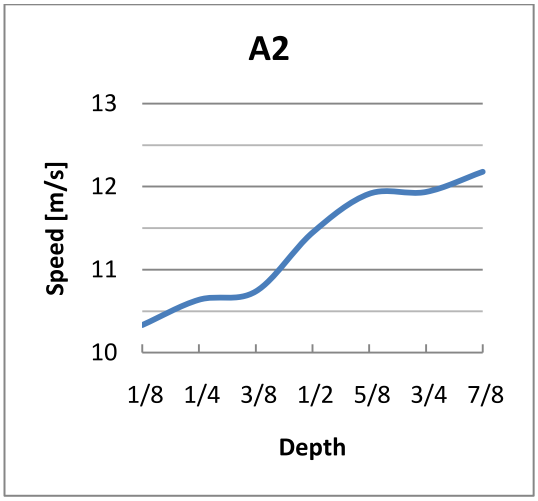
3/4		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	11,8	8	19,8	--	--	--
	A1	3	1	4	11,2	0,6	11,8	8,2	1	9,2
	A2	9	1,7	10,7	9,2	1,7	10,9	8,5	1,7	10,2
	A3	8,4	6	14,4	8,6	4,7	13,3	6,6	4,3	10,9
	A4	8,6	5	13,6	8	5	13	5,2	5	10,2
	A5	8,3	5,5	13,8	8	5,5	13,5	6,4	5,5	11,9
	A6	7,6	4,9	12,5	7,7	5,1	12,8	6,5	5,5	12
	B1	-4,3	-1,8	-6,1	2,5	-1,7	0,8	8,4	-2	6,4
	B2	0	0,2	0,2	2,4	0	2,4	4,8	0	4,8
	B3	1,4	0	1,4	1,7	0	1,7	2,6	0	2,6
	O1	--	--	--	1,7	0	1,7	--	--	--
	C1	-3,8	-0,1	-3,9	5,8	-0,6	5,2	7,6	-0,4	7,2
	C2	2	0,6	2,6	3,6	0,6	4,2	6,4	0,4	6,8
	C3	2,7	0,1	2,8	3,4	0	3,4	5,2	0	5,2
	O2	--	--	--	3,2	0	3,2	--	--	--

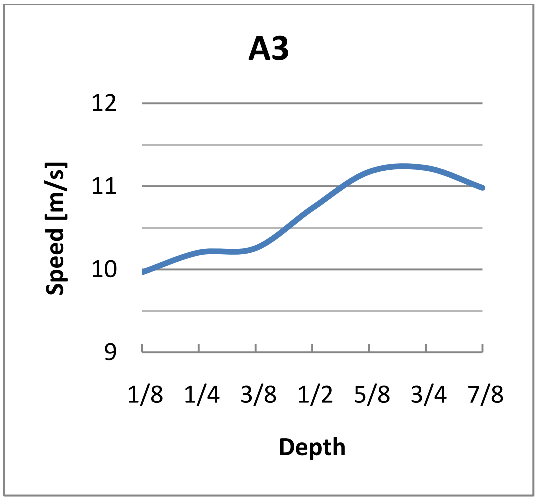
7/8		Testing values [mmH2O]								
		1st/upper wall			Middle			2nd/lower wall		
		Dyn	Stat	Total	Dyn	Stat	Total	Dyn	Stat	Total
Control Points	Fan	--	--	--		6,5	6,5	--	--	--
	I1	--	--	--	11,8	8	19,8	--	--	--
	A1	3,8	0,8	4,6	11,2	0,7	11,9	5,4	0,4	5,8
	A2	10	1,6	11,6	9,6	1,6	11,2	8,2	1,7	9,9
	A3	8,4	5,2	13,6	8,4	4,6	13	5,8	4	9,8
	A4	8,2	5	13,2	6,8	4,8	11,6	3,6	5	8,6
	A5	7,3	5,4	12,7	7,1	5,5	12,6	5,8	5,4	11,2
	A6	7,1	4,7	11,8	6,9	5,1	12	6,3	5,5	11,8
	B1	-4,4	-2	-6,4	4	-2	2	8,4	-2	6,4
	B2	0	0	0	1,3	-0,1	1,2	4,8	-0,1	4,7
	B3	1,2	0	1,2	1,6	0	1,6	2,6	-0,1	2,5
	O1	--	--	--	1,6	0	1,6	--	--	--
	C1	-3,6	0,3	-3,3	6,4	-0,3	6,1	6,8	-0,3	6,5
	C2	3	0,3	3,3	5	0,4	5,4	6	0,3	6,3
	C3	3,2	0	3,2	4,3	0	4,3	5,4	0	5,4
	O2	--	--	--	4,1	0	4,1	--	--	--

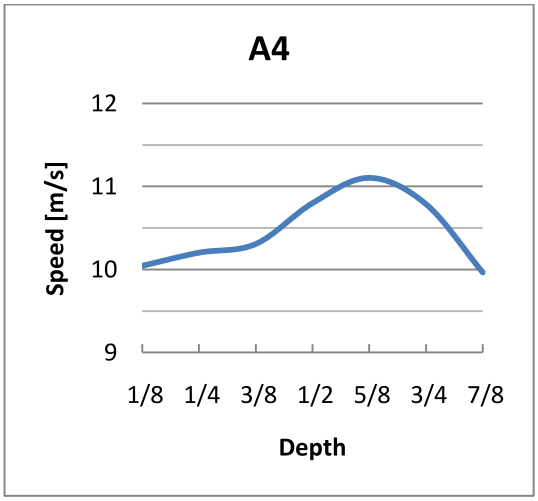
For the speed analysis, there were made speed profiles in each control point. With these profile graphics it could be done a better visual comprehension of how the flow speed behaves inside the manifold. All these speed profiles are below.

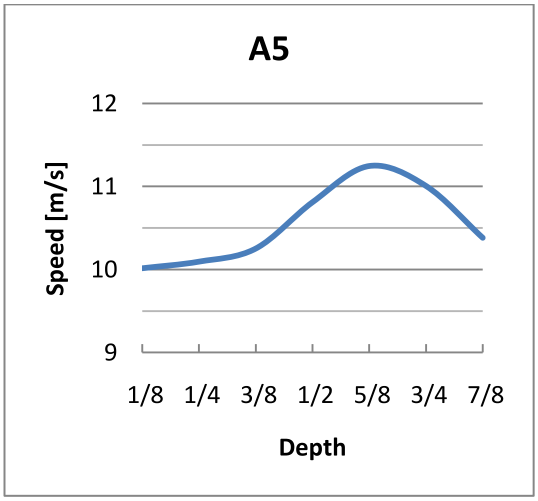


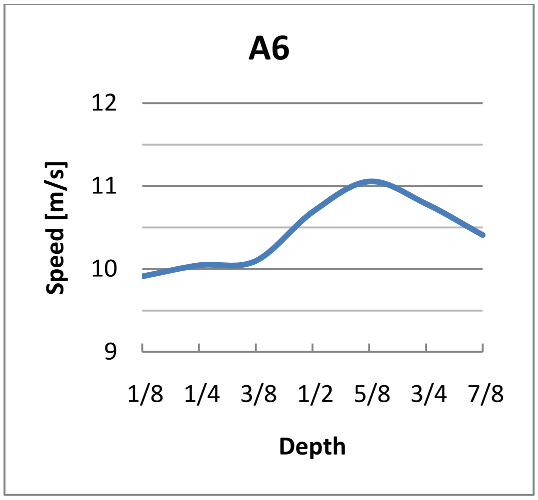


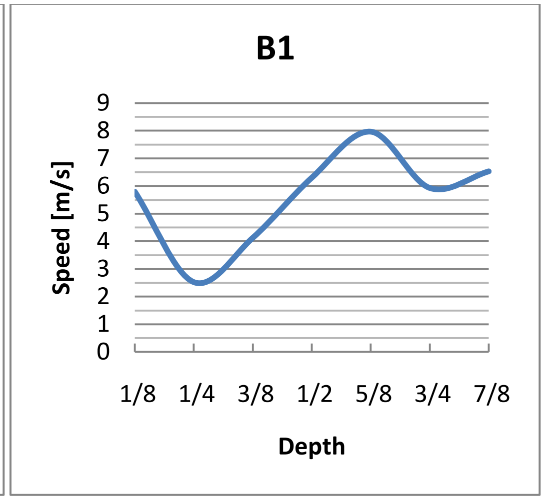


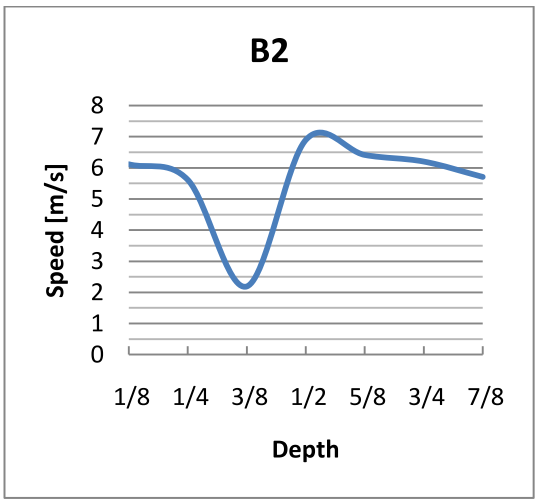


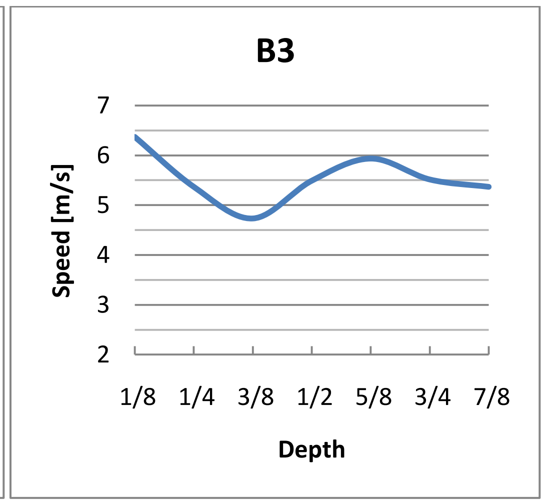


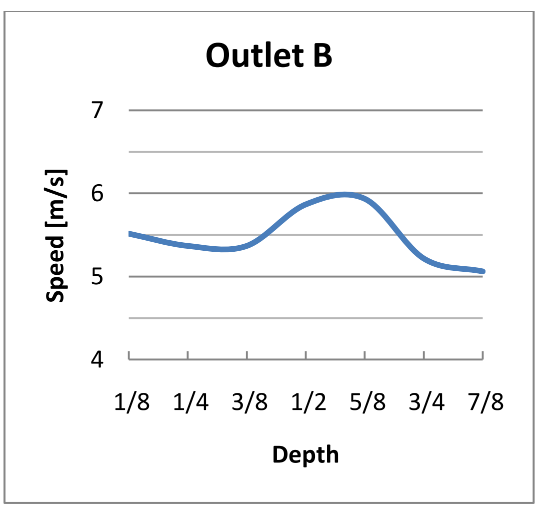


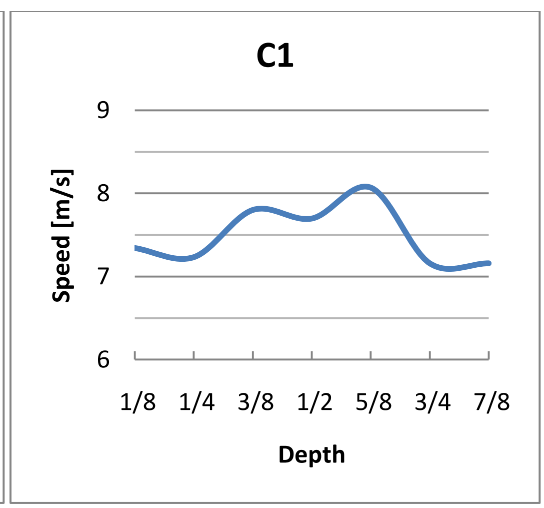

A6
Speed [m/s]
12
11
10
9
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth

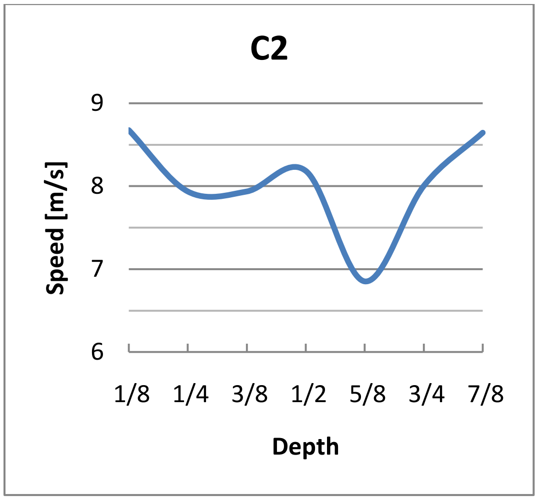

B1
Speed [m/s]
9
8
7
6
5
4
3
2
1
0
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth

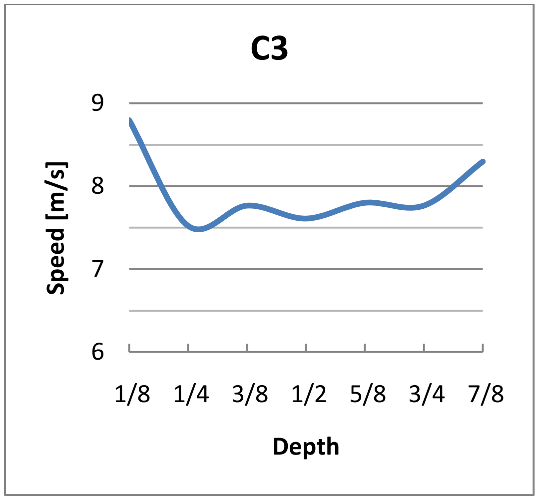

B2
Speed [m/s]
8
7
6
5
4
3
2
1
0
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth

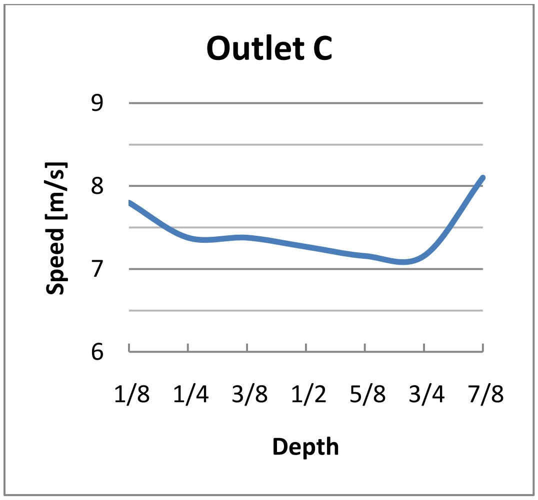

B3
Speed [m/s]
7
6
5
4
3
2
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth


Outlet B
Speed [m/s]
7
6
5
4
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth


C1
Speed [m/s]
9
8
7
6
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth


C2
Speed [m/s]
9
8
7
6
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth


C3
Speed [m/s]
9
8
7
6
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth


Outlet C
Speed [m/s]
9
8
7
6
1/8
1/4
3/8
1/2
5/8
3/4
7/8
Depth

B. EES APPENDICES

B1. EES introduction

One of the tools employed in this project has been the software EES (Engineering Equation Solver).

The main function provided by this program is the solution of a set of algebraic equations. However, it can also solve differential equations, equations with complex variables, do optimization, provide linear and non-linear regression, generate publication-quality plots, simplify uncertainty analyses and provide animations.

There are two major differences between EES and the existing numerical equation-solving programs. On the one hand, EES automatically identifies and groups equations that must be solved simultaneously. This feature simplifies the process for the user and ensures that the solver will always operate at optimum efficiency. On the other hand, EES provides many built-in mathematical and thermophysical property functions useful for engineering calculations.

The first exercise solved by using EES was a tube as shown in the picture below where four states were defined:

- State 1: 0.1 m. before the inlet.
- State 2: Right after the inlet.
- State 3: Before the outlet.
- State 4: 0.1 m. after the outlet.

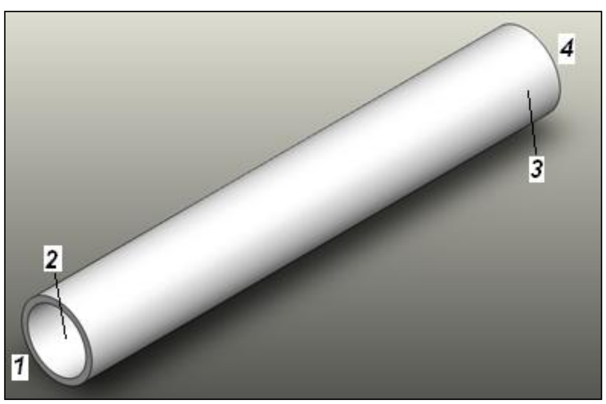


Figure 46. Tube model: 4 states

To be able to find a solution the following assumptions were used:

- P_1 = 0 bar.
- Q = 0.015 m^3/s.
- L = 1 m.

- $\zeta_{inlet} = 0.5$
- $\zeta_{outlet} = 1$

The mathematical expressions employed were:

- $Q = v \cdot A$
- $Re = u \cdot \frac{d}{v}$
- $P_t = P + P_d$
- $P_d = \frac{1}{2} \cdot \rho \cdot u^2$
- $dP_{1,2} = \zeta_{1,2} \cdot P_{d_{1,2}}$
- $P_{t_2} = P_{t_1} - dP_{1,2}$

Equations Window

The Equations Window operates like a Word processor with commands such as Cut, Copy and Paste. The equations that EES is to solve are entered in this window.

In the case of the tube model, , it was necessary to include a little function, apart from the previously mentioned equations, which makes possible to calculate the pressure loss between two points taking into account the roughness and diameter of the tube and the Reynolds number.

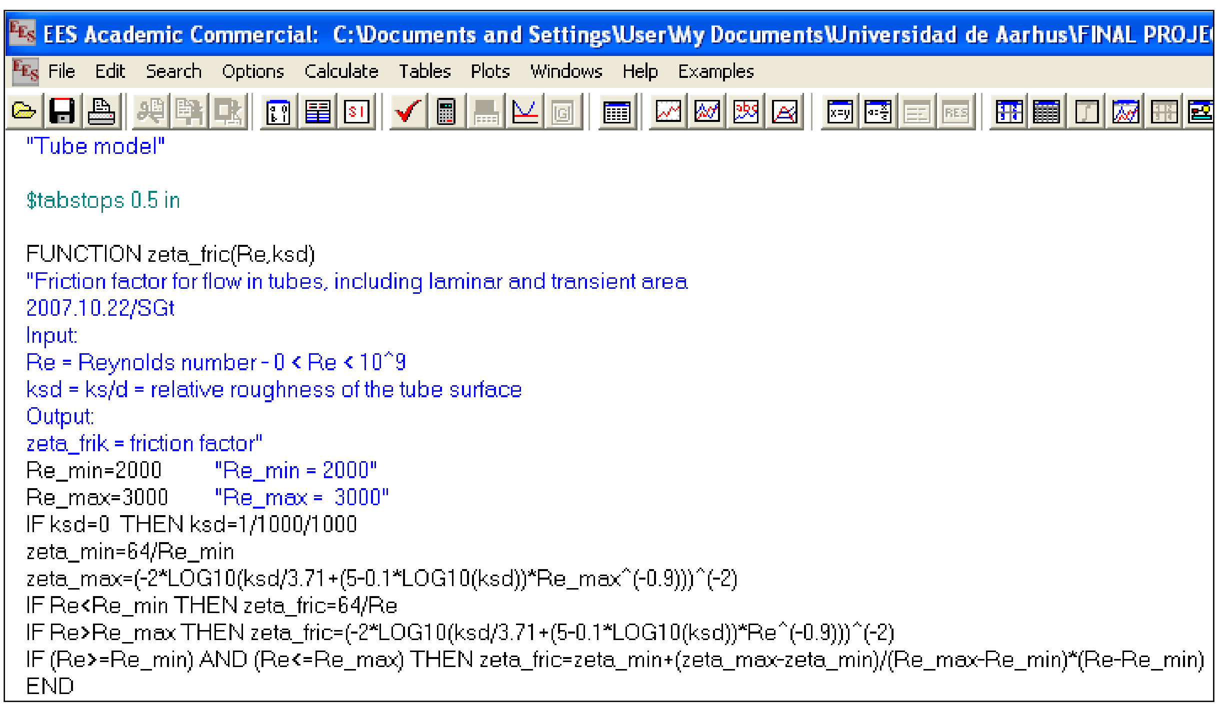


Figure 47. Friction function

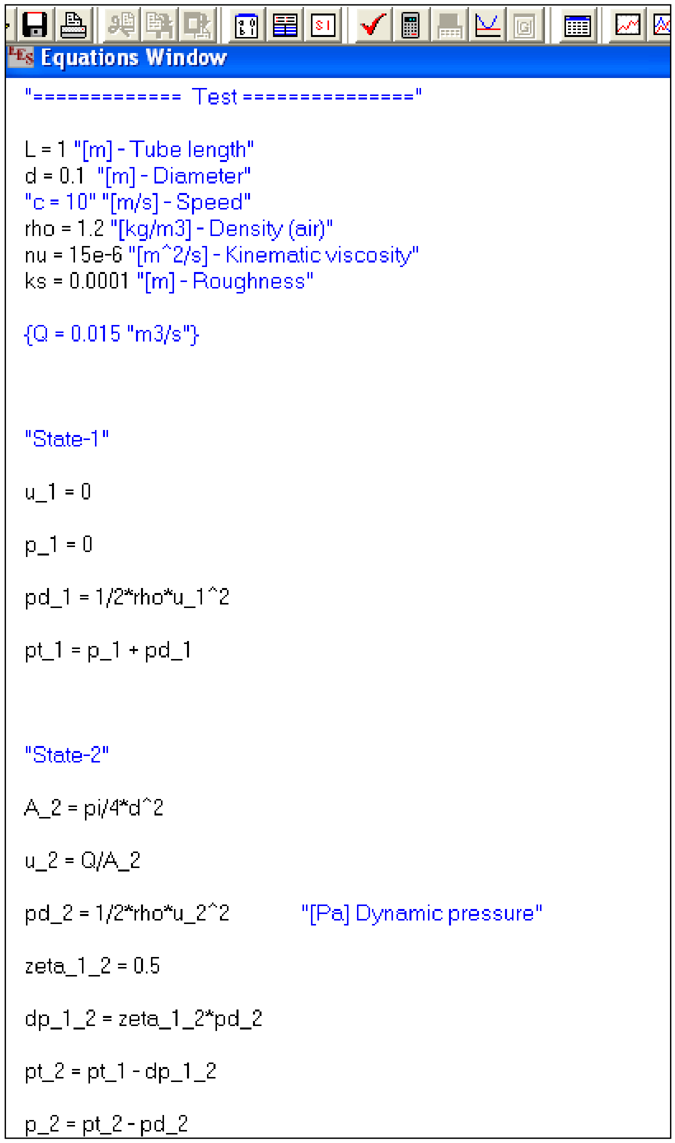


Figure 48. Equations Window

Solution Window

The Solution Window will automatically appear in front of all other windows after the calculations, initiated with the Solve icon or by pressing F2 on the keyboard, are completed. The values and units of all variables appearing in the Equations window will be shown in alphabetical order.

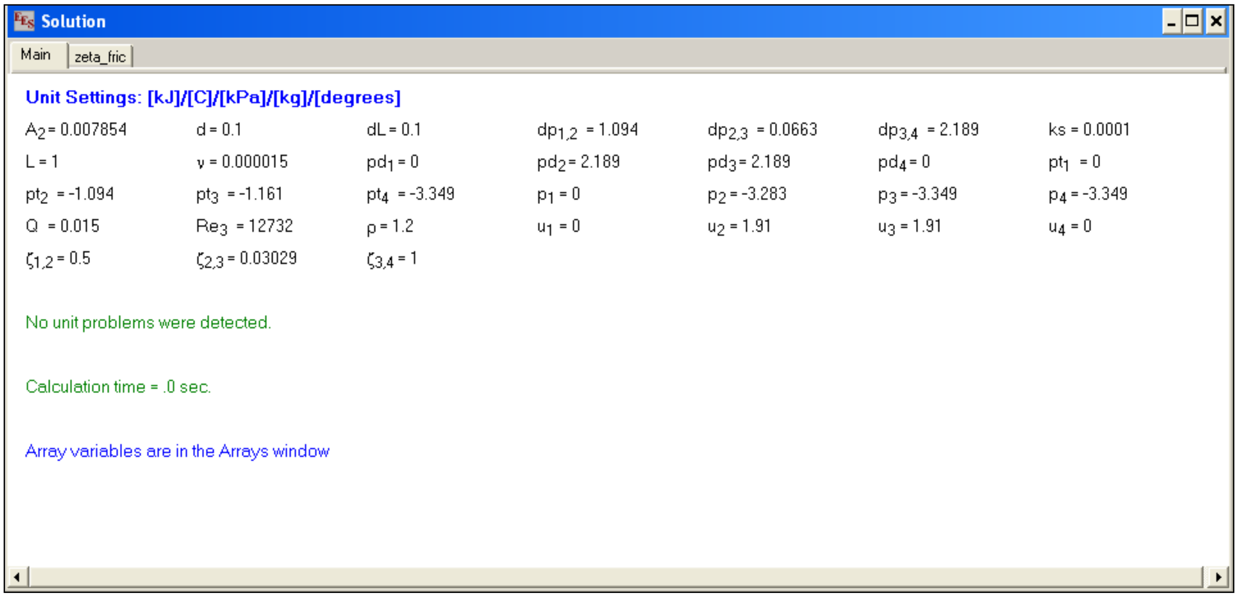
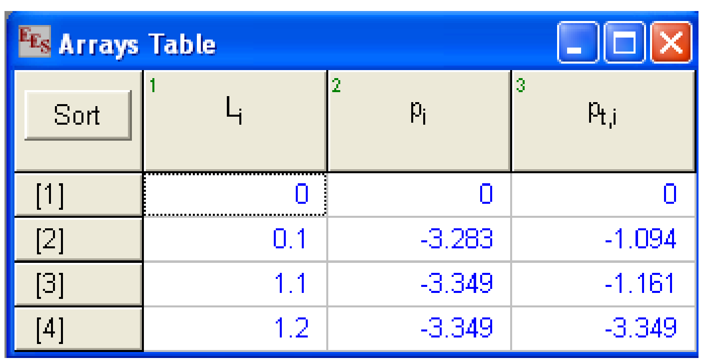


Figure 49. Tube model: Solution Window

Arrays Window and Plot Window

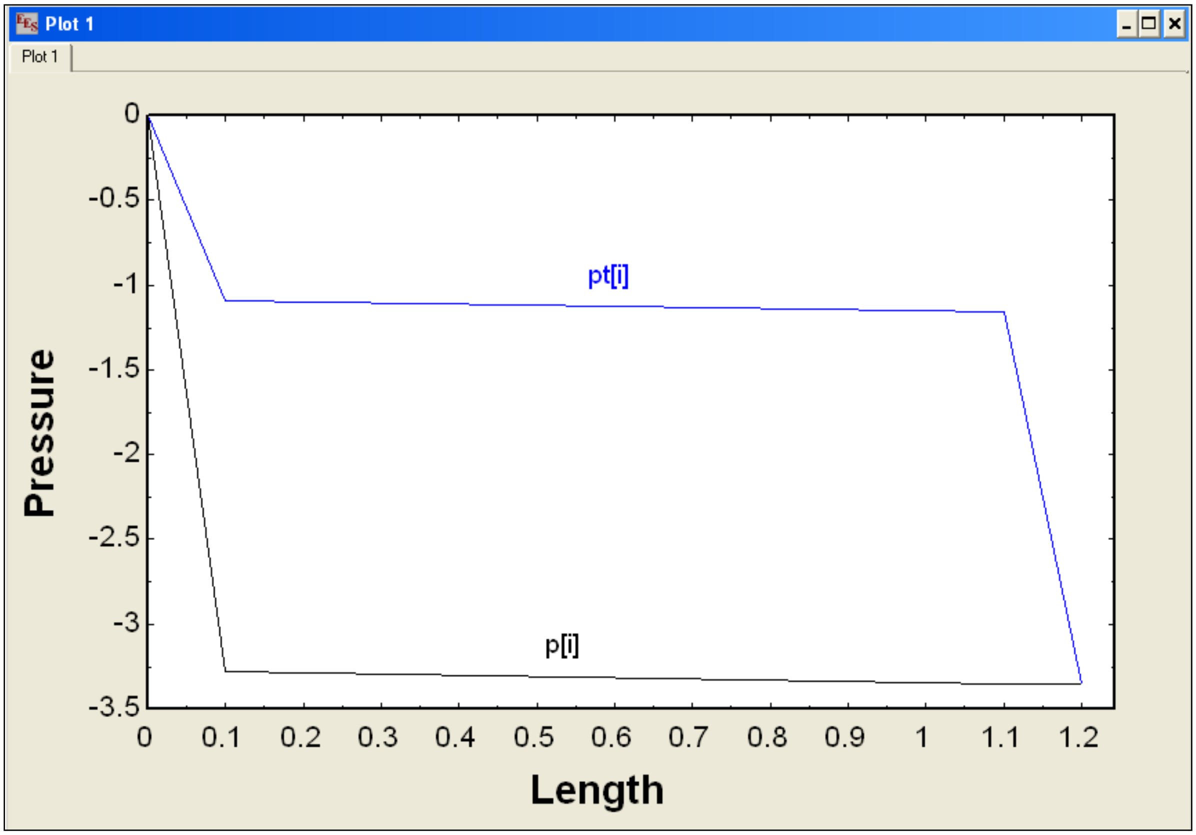
EES allows the use of array variables. EES array variables have the array index in square brackets in the Equations Window. In most ways, array variables are just like ordinary variables. The values of all variables including array variables are normally displayed in the Solution Window after calculations are completed. However, array variables may optionally be displayed in a separate Arrays Window, rather than in the Solution Window. If this is the case, an Arrays Window will automatically be produced. The values in the Arrays Window may be plotted using the New Plot Window command in the Plot menu.



Arrays Table

Sort	1 L_i	2 p_i	3 $p_{t,i}$
[1]	0	0	0
[2]	0.1	-3.283	-1.094
[3]	1.1	-3.349	-1.161
[4]	1.2	-3.349	-3.349

Table 58. Tube model: Arrays table



Graphic 56. Tube model: Plot Window

Diagram Window

The Diagram Window provides a place to display sketches and texts relating with the program being solved. Furthermore, it can be used to provide convenient input and output information. To do that, it is necessary press on Add Text and click on Input (or Output) variable. Then, a list with all the variables appears and the user choose the variables which wants to show in the Diagram Window.

In the case of the tube model all the pressures are shown as output variables as well as the zeta coefficients and the pressure loss. The flow is defined as input variable, so that changing its value in the Diagram Window all the output variables change.

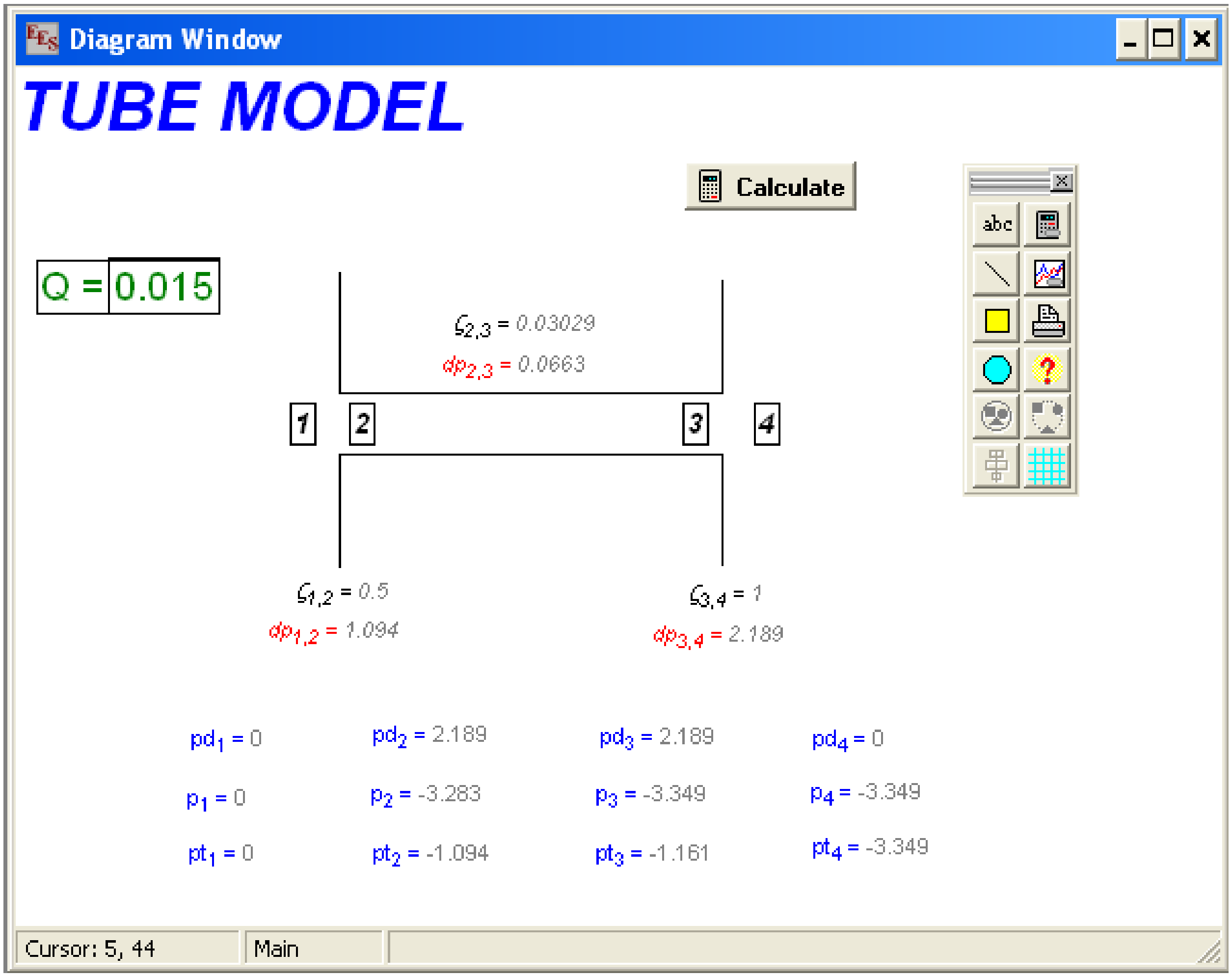


Figure 50. Tube model: Diagram Window

B2. Program description

The Equations Window defined in EES in the case of the manifold consisting of a 30mm. main branch and 19.5 and 21.5mm. branches is show in this appendix. The input data are introduced as highlighted values yellow.

```
"Tube model"

$tabstops 0.5 in

FUNCTION zeta_fric(Re,ksd)
"Friction factor for flow in tubes, including laminar and transient area
2007.10.22/SGt
Input:
Re = Reynolds number - 0 < Re < 10^9
ksd = ks/d = relative roughness of the tube surface
Output:
zeta_frik = friction factor"
Re_min=2000       "Re_min = 2000"
Re_max=3000       "Re_max =  3000"
IF ksd=0  THEN ksd=1/1000/1000
zeta_min=64/Re_min
zeta_max=(-2*LOG10(ksd/3.71+(5-0.1*LOG10(ksd))*Re_max^(-0.9)))^(-2)
IF Re<Re_min THEN zeta_fric=64/Re
IF Re>Re_max THEN zeta_fric=(-2*LOG10(ksd/3.71+(5-0.1*LOG10(ksd))*Re^(-0.9)))^(-2)
IF   (Re>=Re_min)   AND   (Re<=Re_max)   THEN   zeta_fric=zeta_min+(zeta_max-zeta_min)/(Re_max-
Re_min)*(Re-Re_min)
END


"============= Test ==============="

"PROPERTIES"

rho = 1.225 "[kg/m3] - Density (air)"
nu = 15e-6 "[m^2/s] - Kinematic viscosity"
ks = 0.0001 "[m] - Roughness"


"GEOMETRY"

h_1st = 0.0195 "m"
h_2nd = 0.0215 "m"
h_main = 0.03 "m"
width = 0.2 "m^2"
A_1st_branch = h_1st * width "m^2"
```

```
A_2nd_branch = h_2nd * width "m^2"
A_main = h_main * width "m^2"
d_main = ((A_main * 4)/pi)^(1/2) "m" "EQUIVALENT DIAMETER"
d_1st_branch = ((A_1st_branch * 4)/pi)^(1/2) "m" "EQUIVALENT DIAMETER"
d_2nd_branch = ((A_2nd_branch * 4)/pi)^(1/2) "m" "EQUIVALENT DIAMETER"
L_branch = 0.15 "m"
L = 0.25 "m"
dL = 0.1 "m"

Q_total = 0.0622 "m3/s" "VALUE OBTAINED IN THE TESTS"

"State-1"

u_1= 1.5 "m/s" "GUESS"

pd_1 = 1/2 * rho * u_1^2

p_1 = pt_1 - pd_1

"State-2"

u_2 = Q_total /A_main

pd_2 = 1/2 * rho * u_2^2

zeta_1_2 = 0.5 "ZETA IN THE INLET"

dp_1_2 = zeta_1_2 * pd_2

pt_2 = pt_1 - dp_1_2

p_2 = pt_2 - pd_2

"State-3"

u_3 = u_2

Re_3 = u_3 * d_main/nu

zeta_2_3 = zeta_fric(Re_3,ks/d_main)

pd_3 = pd_2
```

```
dp_2_3 = zeta_2_3 * pd_3

pt_3 = pt_2 - dp_2_3

p_3 = pt_3 - pd_3


"State-4"

"m/s - GUESS"

Q_1st_branch = u_4 * A_1st_branch

pd_4 = 1/2 * rho * u_4^2

flow_ratio_1 = Q_1st_branch / Q_total  "column"

area_ratio_1 = A_1st_branch / A_main  "row"

zeta_3_4 = Interpolate2DM('zeta_branch',flow_ratio_1,area_ratio_1) "INTERPOLATION"

dp_3_4 = zeta_3_4 * pd_4

pt_4 = pt_3 - dp_3_4

p_4 = pt_4 - pd_4


"State-5"

Q_2nd_branch = Q_total - Q_1st_branch

u_5 = Q_2nd_branch / A_main

flow_ratio_2 = Q_2nd_branch / Q_total  "column"

area_ratio_2 = A_main / A_main  "row"

zeta_3_5 = Interpolate2DM('zeta_through',flow_ratio_2,area_ratio_2)  "INTERPOLATION"

pd_5 = 1/2 * rho * u_5^2

dp_3_5 = zeta_3_5 * pd_5

pt_5 = pt_3 - dp_3_5

p_5 = pt_5 - pd_5
```

```
"State-6"

u_6 = u_5

Re_6 = u_6 * d_main/nu

zeta_5_6 = zeta_fric(Re_6,ks/d_main)

pd_6 = pd_5

dp_5_6 = zeta_5_6 * pd_6

pt_6 = pt_5 - dp_5_6

p_6 = pt_6 - pd_6

"State-7"

u_7 = Q_2nd_branch / A_2nd_branch

zeta_6_7 = 1.3 "INTERNAL FLOW SYSTEMS D.S.Miller page 149"

pd_7 = 1/2 * rho * u_7^2

dp_6_7 = zeta_6_7 * pd_7

pt_7 = pt_6 - dp_6_7

p_7 = pt_7 - pd_7

"State-8"

u_8 = u_7

Re_8 = u_8 * d_2nd_branch/nu

zeta_7_8 = zeta_fric(Re_8,ks/d_2nd_branch)

pd_8 = pd_7

dp_7_8 = zeta_7_8 * pd_8

pt_8 = pt_7 - dp_7_8

p_8 = pt_8 - pd_8
```

"State-9"

```
u_9 = u_4

Re_9 = u_9 * d_1st_branch/nu

zeta_4_9 = zeta_fric(Re_9,ks/d_1st_branch)

pd_9 = pd_4

dp_4_9 = zeta_4_9 * pd_9

pt_9 = pt_4 - dp_4_9

p_9 = pt_9 - pd_9
```

"State-10"

```
u_10 = 0

zeta_9_10 = 1 "ZETA IN THE OUTLET"

dp_9_10 = zeta_9_10 * 1/2*rho*u_9^2

pd_10 = 1/2 * rho * u_10^2

pt_10 = pt_9 - dp_9_10

p_10 = pt_10 - pd_10

p_10 = 0
```

"State-11"

```
u_11 = 0

zeta_8_11 = 1 "ZETA IN THE OUTLET"

dp_8_11= zeta_8_11 * 1/2 * rho * u_8^2

pd_11 = 1/2 * rho * u_11^2

pt_11 = pt_8 - dp_8_11

p_11 = pt_11 - pd_11
```

p_11 = 0

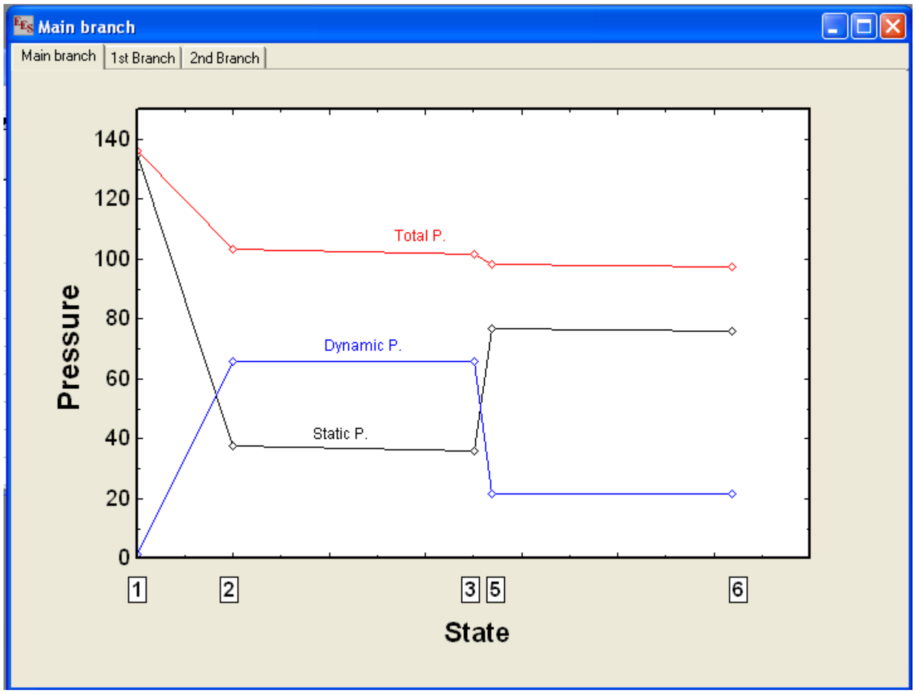
The following equations are the definition of the Arrays Window calculated in order to plot the results obtained in the Solution Window by using the Plot menu.

"Chart - Main branch"

p[1] = p_1
p[2] = p_2
p[3] = p_3
p[5] = p_5
p[6] = p_6

p_t[1] = pt_1
p_t[2] = pt_2
p_t[3] = pt_3
p_t[5] = pt_5
p_t[6] = pt_6

p_d[1] = pd_1
p_d[2] = pd_2
p_d[3] = pd_3
p_d[5] = pd_5
p_d[6] = pd_6



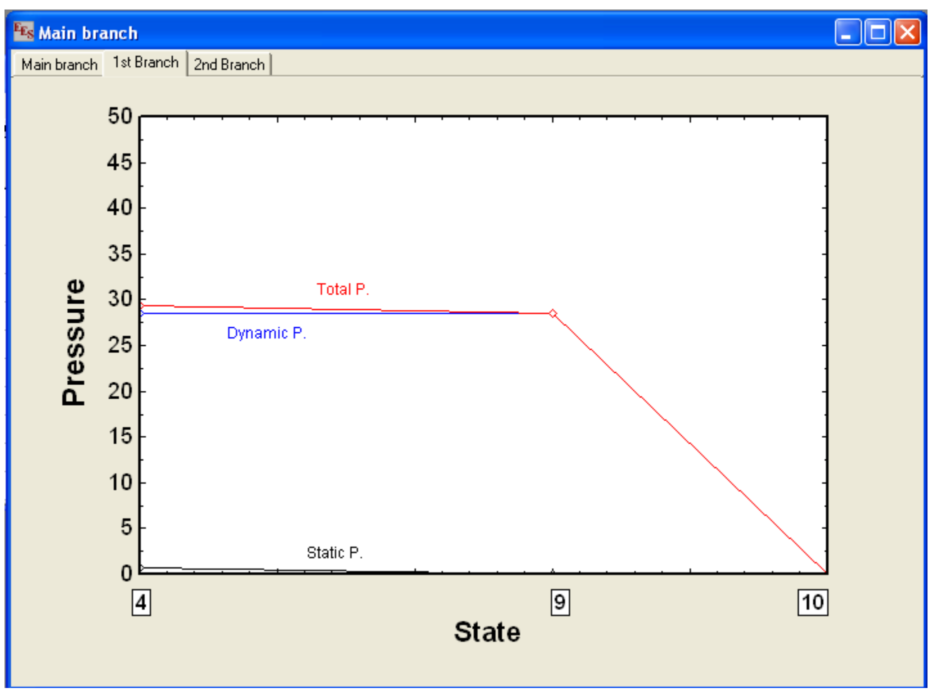
Graphic 57. Plot Window: Main branch

L[1] = 0
L[2] = L[1] + dL
L[3] = L[2] + L
L[5] = L[3] + h_1st
L[6] = L[5] + L

"Chart - 1st branch"

p[4] = p_4
p[9] = p_9
p[10] = p_10

p_t[4] = pt_4
p_t[9] = pt_9
p_t[10] = pt_10



Graphic 58. Plot Window: 1st Branch

p_d[4] = pd_4
p_d[9] = pd_9
p_d[10] = pd_10

L[4] = 0
L[9] = L[4] + L_branch
L[10] = L[9] + dL

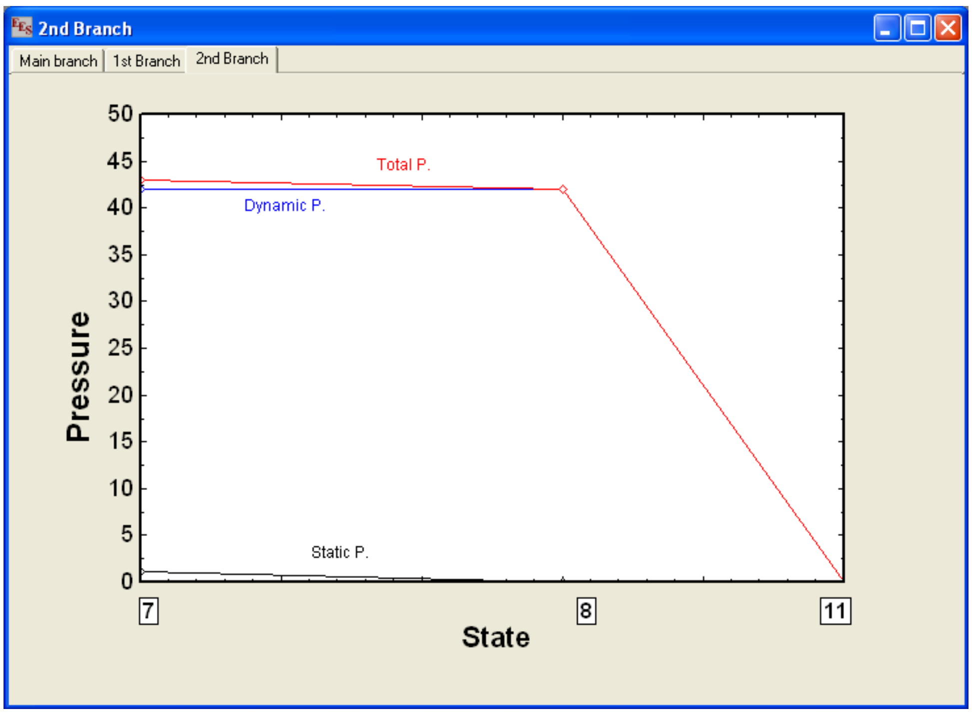
"Chart - 2nd branch"

p[7] = p_7
p[8] = p_8
p[11] = p_11

p_t[7] = pt_7
p_t[8] = pt_8
p_t[11] = pt_11

p_d[7] = pd_7
p_d[8] = pd_8
p_d[11] = pd_11

L[7] = 0
L[8] = L[7] + L_branch
L[11] = L[8] + dL



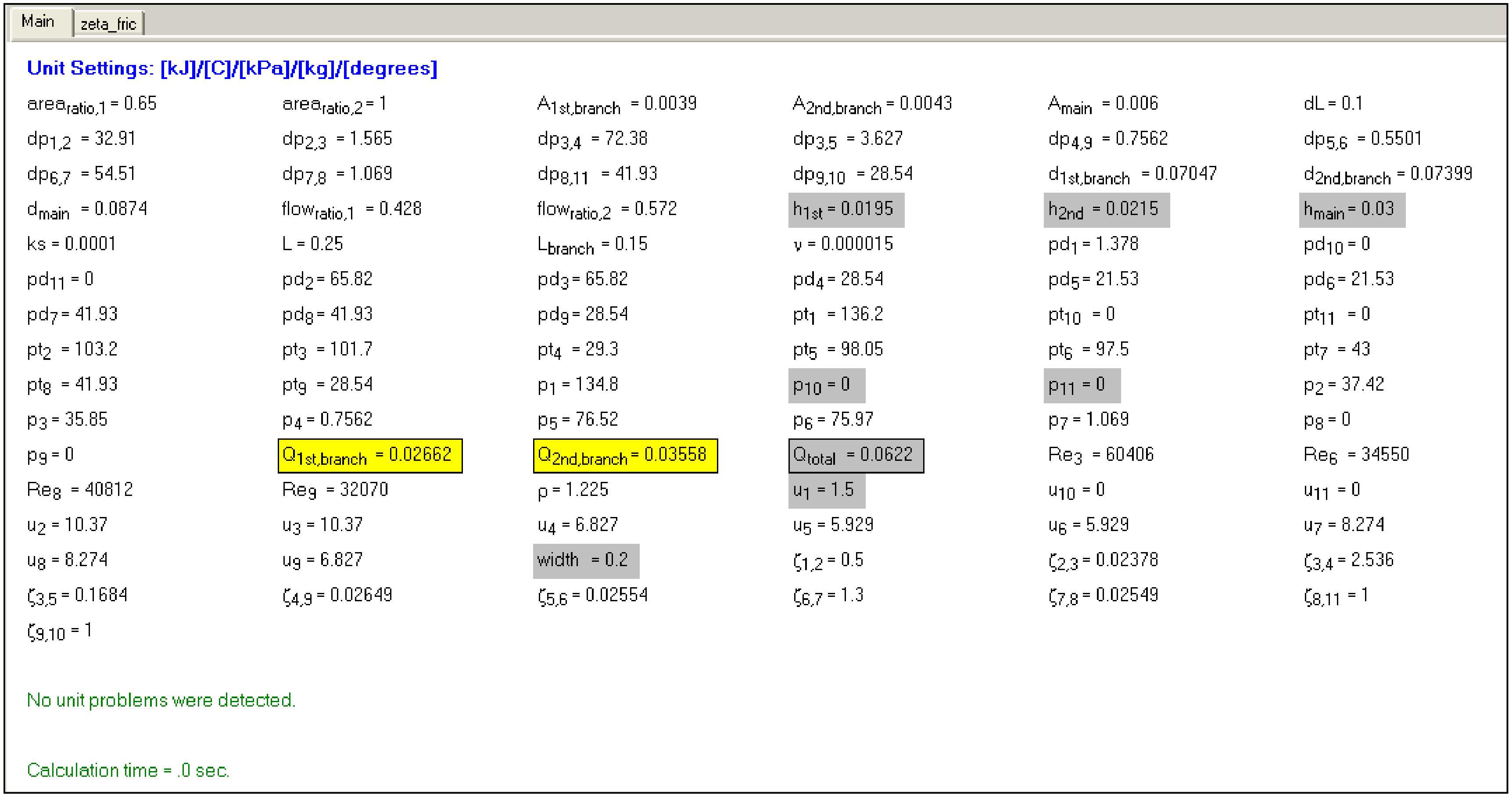
Graphic 59. Plot Window: 2nd branch

Arrays Table

Sort	1 L_i	2 p_i	3 $p_{d,i}$	4 $p_{t,i}$
[1]	0	134.8	1.378	136.2
[2]	0.1	37.42	65.82	103.2
[3]	0.35	35.85	65.82	101.7
[4]	0	0.7562	28.54	29.3
[5]	0.3695	76.52	21.53	98.05
[6]	0.6195	75.97	21.53	97.5
[7]	0	1.069	41.93	43
[8]	0.15	0	41.93	41.93
[9]	0.15	0	28.54	28.54
[10]	0.25	0	0	0
[11]	0.25	0	0	0

Table 59. Wide branches model: Arrays table

The solution table shown in both the "Wide Branches Model"and "Narrow Branches Model" reports is a resume of the Solution Window calculated by the program (see the figures below).



Main | zeta_fric

Unit Settings: [kJ]/[C]/[kPa]/[kg]/[degrees]

$area_{ratio,1} = 0.65$	$area_{ratio,2} = 1$	$A_{1st,branch} = 0.0039$	$A_{2nd,branch} = 0.0043$	$A_{main} = 0.006$	$dL = 0.1$
$dp_{1,2} = 32.91$	$dp_{2,3} = 1.565$	$dp_{3,4} = 72.38$	$dp_{3,5} = 3.627$	$dp_{4,9} = 0.7562$	$dp_{5,6} = 0.5501$
$dp_{6,7} = 54.51$	$dp_{7,8} = 1.069$	$dp_{8,11} = 41.93$	$dp_{9,10} = 28.54$	$d_{1st,branch} = 0.07047$	$d_{2nd,branch} = 0.07399$
$d_{main} = 0.0874$	$flow_{ratio,1} = 0.428$	$flow_{ratio,2} = 0.572$	$h_{1st} = 0.0195$	$h_{2nd} = 0.0215$	$h_{main} = 0.03$
$ks = 0.0001$	$L = 0.25$	$L_{branch} = 0.15$	$\nu = 0.000015$	$pd_1 = 1.378$	$pd_{10} = 0$
$pd_{11} = 0$	$pd_2 = 65.82$	$pd_3 = 65.82$	$pd_4 = 28.54$	$pd_5 = 21.53$	$pd_6 = 21.53$
$pd_7 = 41.93$	$pd_8 = 41.93$	$pd_9 = 28.54$	$pt_1 = 136.2$	$pt_{10} = 0$	$pt_{11} = 0$
$pt_2 = 103.2$	$pt_3 = 101.7$	$pt_4 = 29.3$	$pt_5 = 98.05$	$pt_6 = 97.5$	$pt_7 = 43$
$pt_8 = 41.93$	$pt_9 = 28.54$	$p_1 = 134.8$	$p_{10} = 0$	$p_{11} = 0$	$p_2 = 37.42$
$p_3 = 35.85$	$p_4 = 0.7562$	$p_5 = 76.52$	$p_6 = 75.97$	$p_7 = 1.069$	$p_8 = 0$
$p_9 = 0$	$Q_{1st,branch} = 0.02662$	$Q_{2nd,branch} = 0.03558$	$Q_{total} = 0.0622$	$Re_3 = 60406$	$Re_6 = 34550$
$Re_8 = 40812$	$Re_9 = 32070$	$\rho = 1.225$	$u_1 = 1.5$	$u_{10} = 0$	$u_{11} = 0$
$u_2 = 10.37$	$u_3 = 10.37$	$u_4 = 6.827$	$u_5 = 5.929$	$u_6 = 5.929$	$u_7 = 8.274$
$u_8 = 8.274$	$u_9 = 6.827$	$width = 0.2$	$\zeta_{1,2} = 0.5$	$\zeta_{2,3} = 0.02378$	$\zeta_{3,4} = 2.536$
$\zeta_{3,5} = 0.1684$	$\zeta_{4,9} = 0.02649$	$\zeta_{5,6} = 0.02554$	$\zeta_{6,7} = 1.3$	$\zeta_{7,8} = 0.02549$	$\zeta_{8,11} = 1$
$\zeta_{9,10} = 1$					

No unit problems were detected.

Calculation time = .0 sec.

Figure 51. Wide branches model: Solution Window

Main | zeta_fric

Unit Settings: [kJ]/[C]/[kPa]/[kg]/[degrees]

$area_{ratio,1} = 1$	$area_{ratio,2} = 1$	$A_{1st,branch} = 0.001$	$A_{2nd,branch} = 0.001$	$A_{main} = 0.001$	$dL = 0.1$
$dp_{1,2} = 37.33$	$dp_{2,3} = 2.245$	$dp_{3,4} = 71.81$	$dp_{3,5} = 5.096$	$dp_{4,9} = 0.2943$	$dp_{5,6} = 1.054$
$dp_{6,7} = 39.91$	$dp_{7,8} = 1.054$	$dp_{8,11} = 33.25$	$dp_{9,10} = 8.257$	$d_{1st,branch} = 0.03568$	$d_{2nd,branch} = 0.03568$
$d_{main} = 0.03568$	$flow_{ratio,1} = 0.3326$	$flow_{ratio,2} = 0.6674$	$h_{1st} = 0.005$	$h_{2nd} = 0.005$	$h_{main} = 0.005$
$ks = 0.0001$	$L = 0.25$	$L_{branch} = 0.15$	$\nu = 0.000015$	$pd_1 = 1.378$	$pd_{10} = 0$
$pd_{11} = 0$	$pd_2 = 74.65$	$pd_3 = 74.65$	$pd_4 = 8.257$	$pd_5 = 33.25$	$pd_6 = 33.25$
$pd_7 = 33.25$	$pd_8 = 33.25$	$pd_9 = 8.257$	$pt_1 = 119.9$	$pt_{10} = 0$	$pt_{11} = 0$
$pt_2 = 82.61$	$pt_3 = 80.36$	$pt_4 = 8.551$	$pt_5 = 75.27$	$pt_6 = 74.21$	$pt_7 = 34.31$
$pt_8 = 33.25$	$pt_9 = 8.257$	$p_1 = 118.6$	$p_{10} = 0$	$p_{11} = 0$	$p_2 = 7.956$
$p_3 = 5.71$	$p_4 = 0.2943$	$p_5 = 42.01$	$p_6 = 40.96$	$p_7 = 1.054$	$p_8 = 0$
$p_9 = 0$	$Q_{1st,branch} = 0.003672$	$Q_{2nd,branch} = 0.007368$	$Q_{total} = 0.01104$	$Re_3 = 26262$	$Re_6 = 17528$
$Re_8 = 17528$	$Re_9 = 8734$	$\rho = 1.225$	$u_1 = 1.5$	$u_{10} = 0$	$u_{11} = 0$
$u_2 = 11.04$	$u_3 = 11.04$	$u_4 = 3.672$	$u_5 = 7.368$	$u_6 = 7.368$	$u_7 = 7.368$
$u_8 = 7.368$	$u_9 = 3.672$	$width = 0.2$	$\zeta_{1,2} = 0.5$	$\zeta_{2,3} = 0.03008$	$\zeta_{3,4} = 8.697$
$\zeta_{3,5} = 0.1533$	$\zeta_{4,9} = 0.03564$	$\zeta_{5,6} = 0.03168$	$\zeta_{6,7} = 1.2$	$\zeta_{7,8} = 0.03168$	$\zeta_{8,11} = 1$
$\zeta_{9,10} = 1$					

No unit problems were detected.

Calculation time = .0 sec.

Figure 52. Narrow branches model: Solution Window (Inlet speed = 10m/s)

Main | zeta_fric

Unit Settings: [kJ]/[C]/[kPa]/[kg]/[degrees]

$area_{ratio,1} = 1$	$area_{ratio,2} = 1$	$A_{1st,branch} = 0.001$	$A_{2nd,branch} = 0.001$	$A_{main} = 0.001$	$dL = 0.1$
$dp_{1,2} = 88.51$	$dp_{2,3} = 5.097$	$dp_{3,4} = 170.2$	$dp_{3,5} = 12.09$	$dp_{4,9} = 0.645$	$dp_{5,6} = 2.366$
$dp_{6,7} = 94.69$	$dp_{7,8} = 2.366$	$dp_{8,11} = 78.91$	$dp_{9,10} = 19.55$	$d_{1st,branch} = 0.03568$	$d_{2nd,branch} = 0.03568$
$d_{main} = 0.03568$	$flow_{ratio,1} = 0.3323$	$flow_{ratio,2} = 0.6677$	$h_{1st} = 0.005$	$h_{2nd} = 0.005$	$h_{main} = 0.005$
$ks = 0.0001$	$L = 0.25$	$L_{branch} = 0.15$	$\nu = 0.000015$	$pd_1 = 1.378$	$pd_{10} = 0$
$pd_{11} = 0$	$pd_2 = 177$	$pd_3 = 177$	$pd_4 = 19.55$	$pd_5 = 78.91$	$pd_6 = 78.91$
$pd_7 = 78.91$	$pd_8 = 78.91$	$pd_9 = 19.55$	$pt_1 = 284$	$pt_{10} = 0$	$pt_{11} = 0$
$pt_2 = 195.5$	$pt_3 = 190.4$	$pt_4 = 20.2$	$pt_5 = 178.3$	$pt_6 = 176$	$pt_7 = 81.27$
$pt_8 = 78.91$	$pt_9 = 19.55$	$p_1 = 282.6$	$p_{10} = 0$	$p_{11} = 0$	$p_2 = 18.5$
$p_3 = 13.4$	$p_4 = 0.645$	$p_5 = 99.42$	$p_6 = 97.05$	$p_7 = 2.366$	$p_8 = 0$
$p_9 = 0$	$Q_{1st,branch} = 0.00565$	$Q_{2nd,branch} = 0.01135$	$Q_{total} = 0.017$	$Re_3 = 40440$	$Re_6 = 27000$
$Re_8 = 27000$	$Re_9 = 13440$	$\rho = 1.225$	$u_1 = 1.5$	$u_{10} = 0$	$u_{11} = 0$
$u_2 = 17$	$u_3 = 17$	$u_4 = 5.65$	$u_5 = 11.35$	$u_6 = 11.35$	$u_7 = 11.35$
$u_8 = 11.35$	$u_9 = 5.65$	$width = 0.2$	$\zeta_{1,2} = 0.5$	$\zeta_{2,3} = 0.02879$	$\zeta_{3,4} = 8.706$
$\zeta_{3,5} = 0.1532$	$\zeta_{4,9} = 0.03299$	$\zeta_{5,6} = 0.02998$	$\zeta_{6,7} = 1.2$	$\zeta_{7,8} = 0.02998$	$\zeta_{8,11} = 1$
$\zeta_{9,10} = 1$					

No unit problems were detected.

Calculation time = .0 sec.

Figure 53. Narrow branches model: Solution Window (Inlet speed = 15m/s)

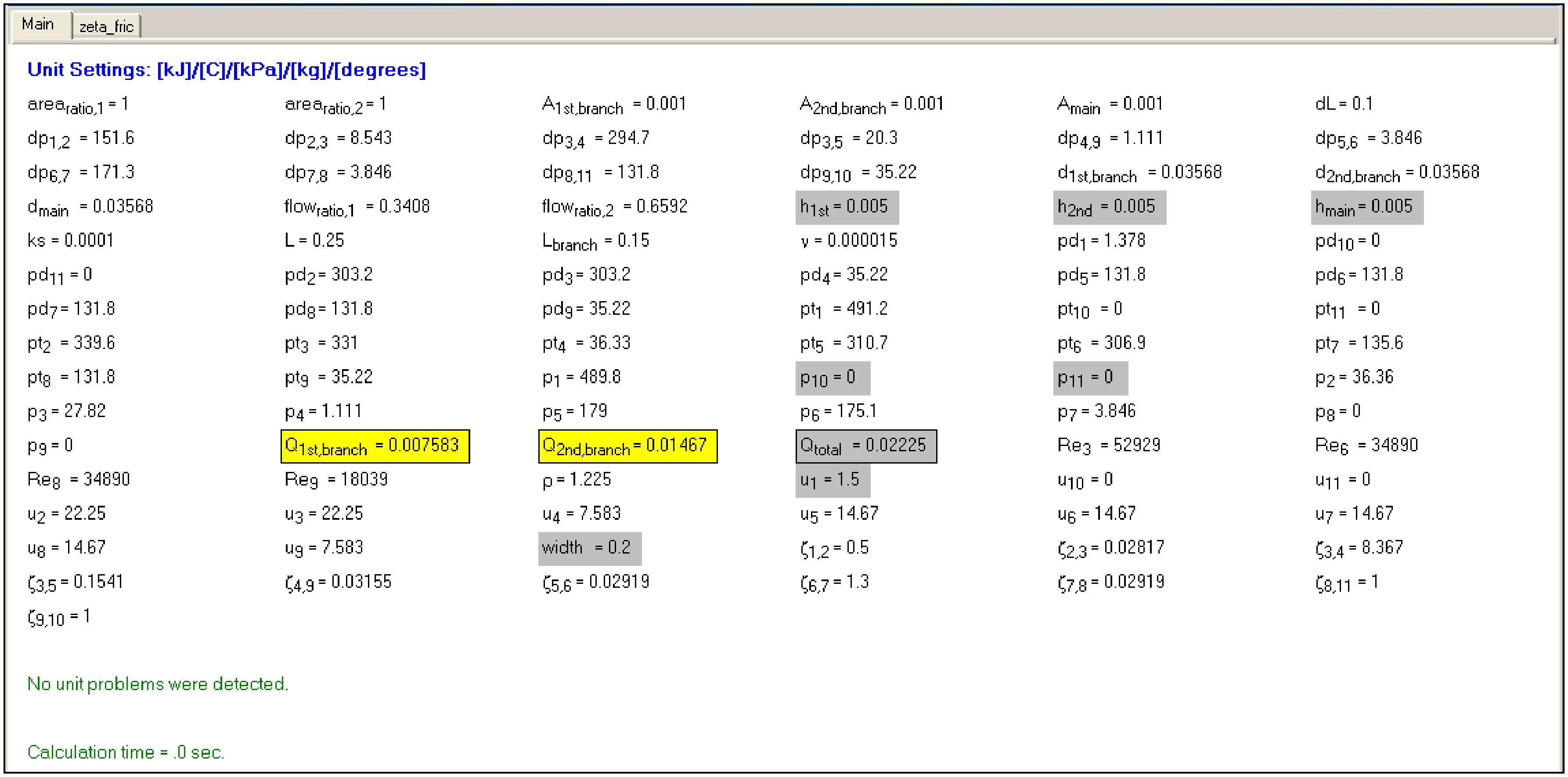


Figure 54. Narrow branches model: Solution Window (Inlet speed = 20m/s)

C. SolidWorks' Flow Simulation Studio Tutorial. Internal flows.

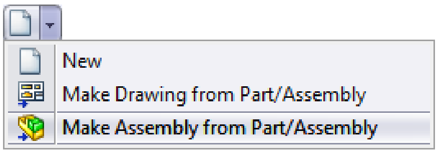
- **Model preparation.**
- **Create and set up a new flow simulation project.**
- **Set up boundary conditions and goals.**
- **Results visualization.**

Model preparation

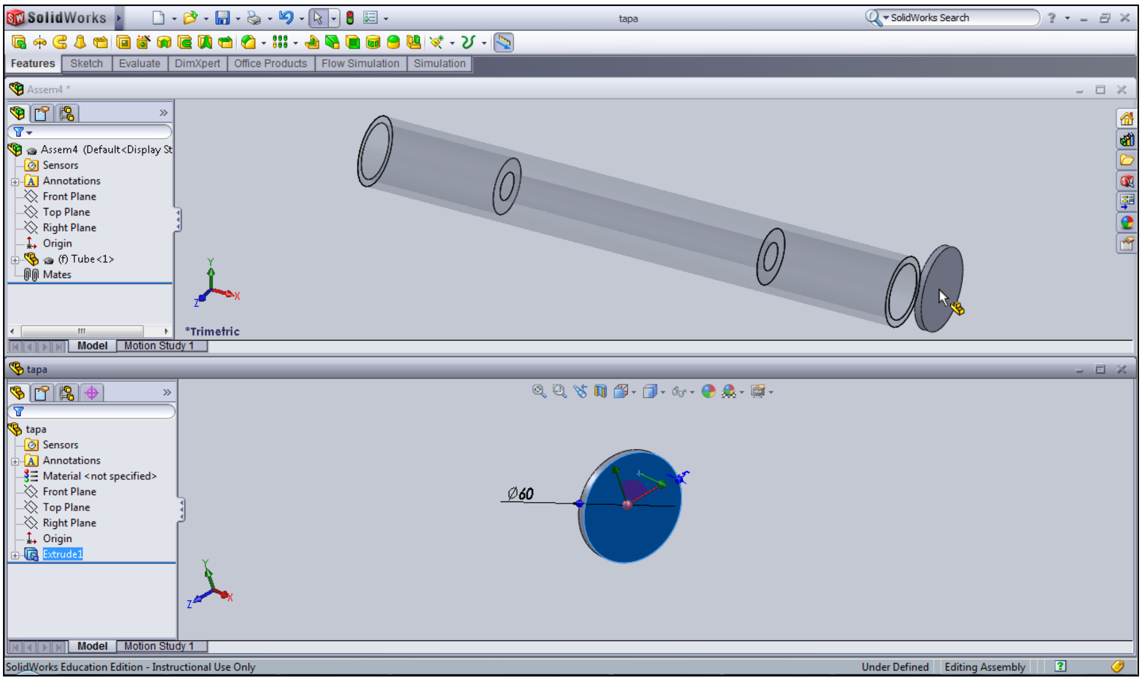
To start the SolidWorks' Flow Simulation Study it is needed to have a SolidWorks' part or assembly to work with. To make an internal flow studio it is needed a part or assembly with an empty cavity, while for an external flow studio it is only needed a part or assembly with any wanted shape.

In this tutorial we are going to work with a tubular pipe as an example of how the Flow Simulation works.

To start with the internal flow analisys open the part which you are going to work with and click **New** and **Make Assembly from Part/Assembly** as shown in the figure to the right .



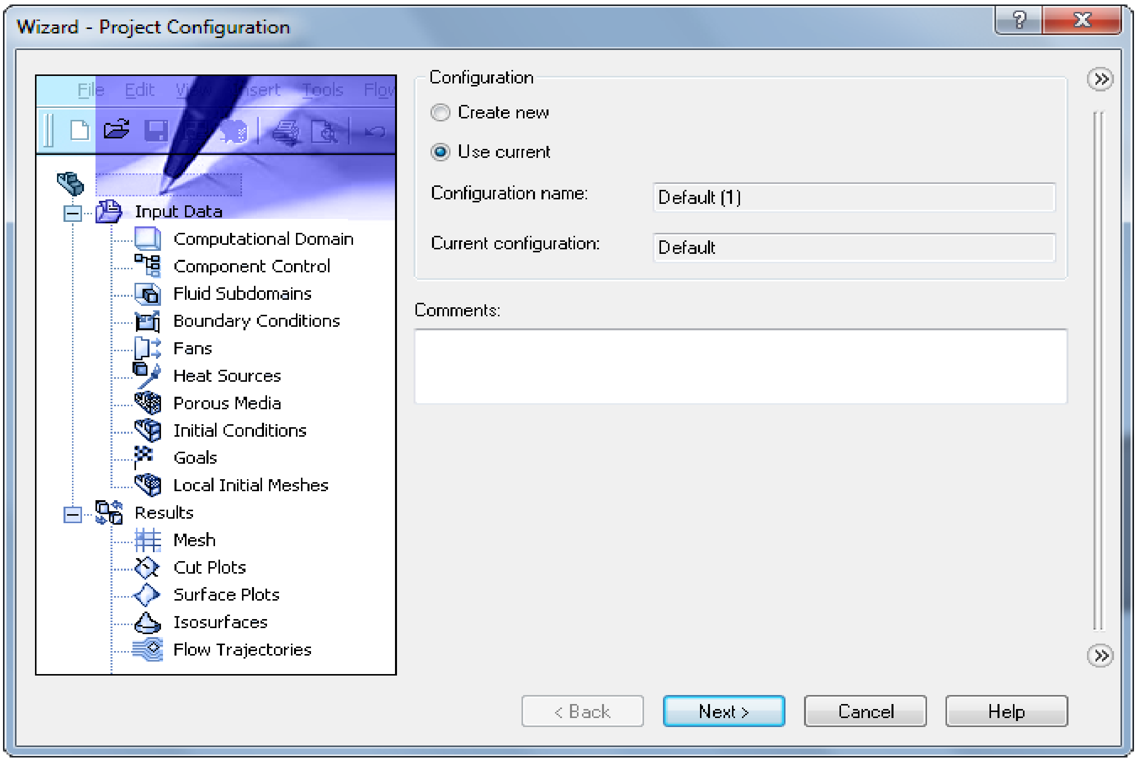
Then it is necessary to create a lid as a new part to enclose the flow field. Open the assembly and the lid part and tile them so you can drag the lids into the assembly.



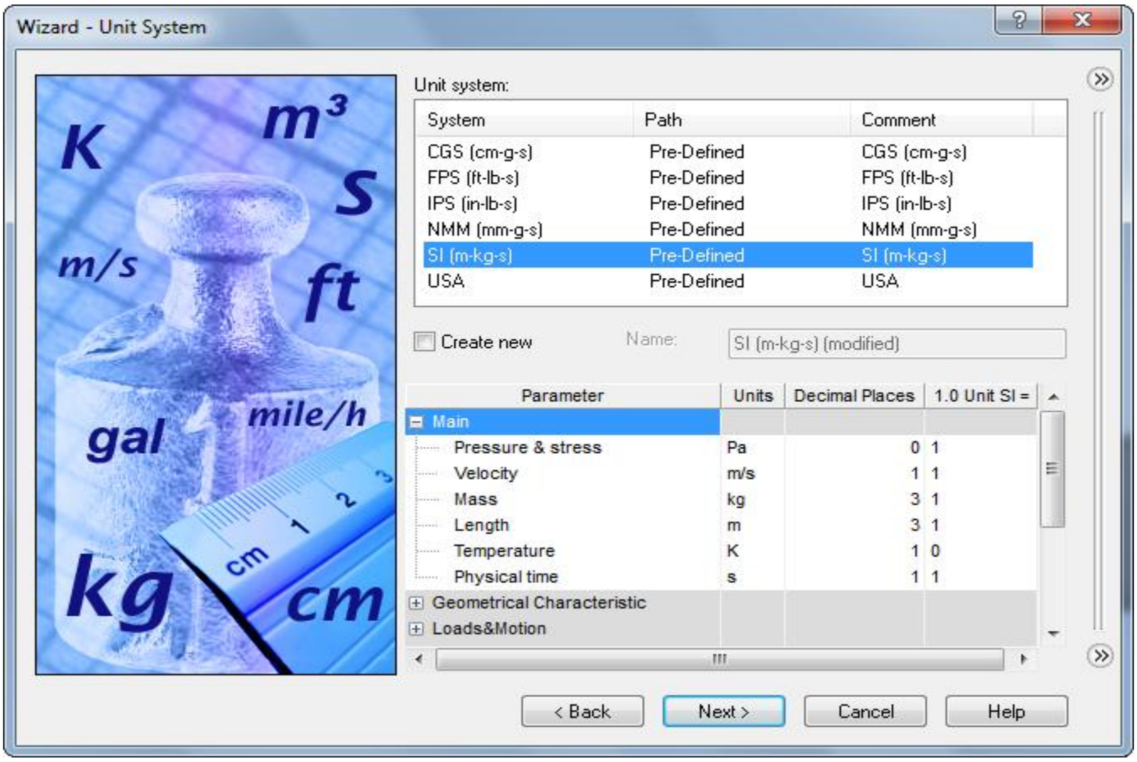
Then make the lids concentric and coincident with the ends of the pipe by adding mates  between them.

Create and set up a new flow simulation project

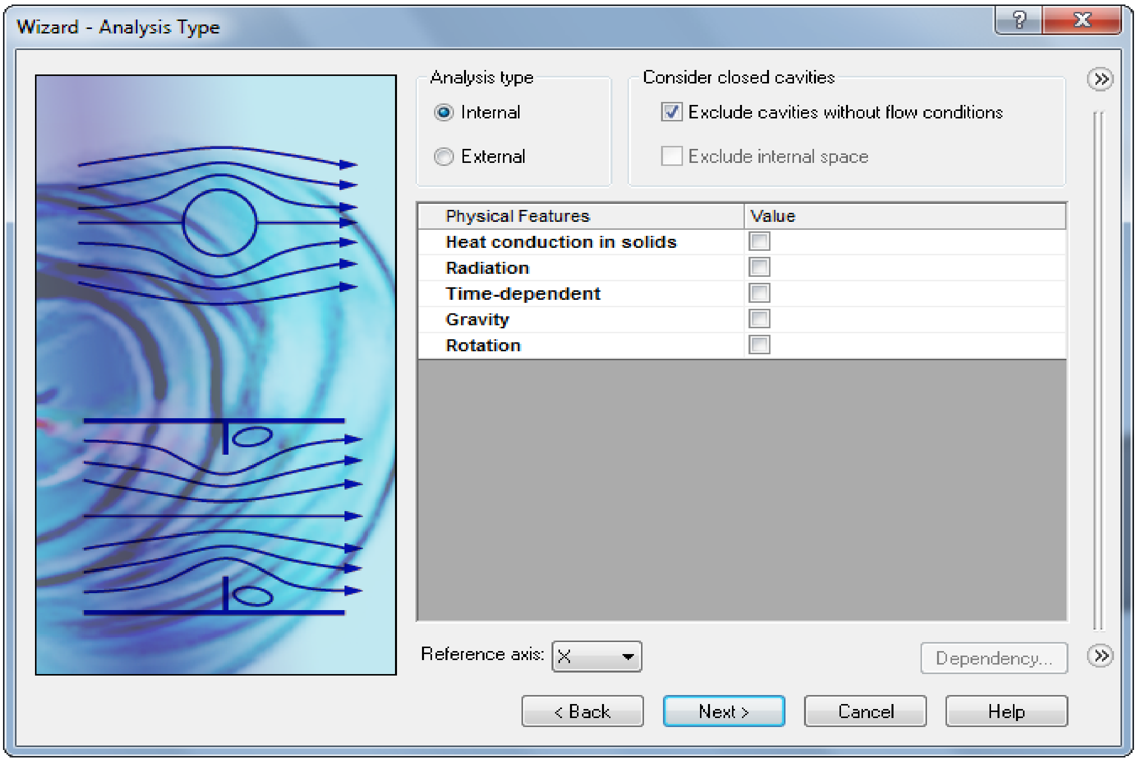
Click on **Flow Simulation** in the **CommandManager** and then click **Wizard**  to start configuring a new project. The **Flow Simulation Wizard** dialog box appears, as shown in the next figure:



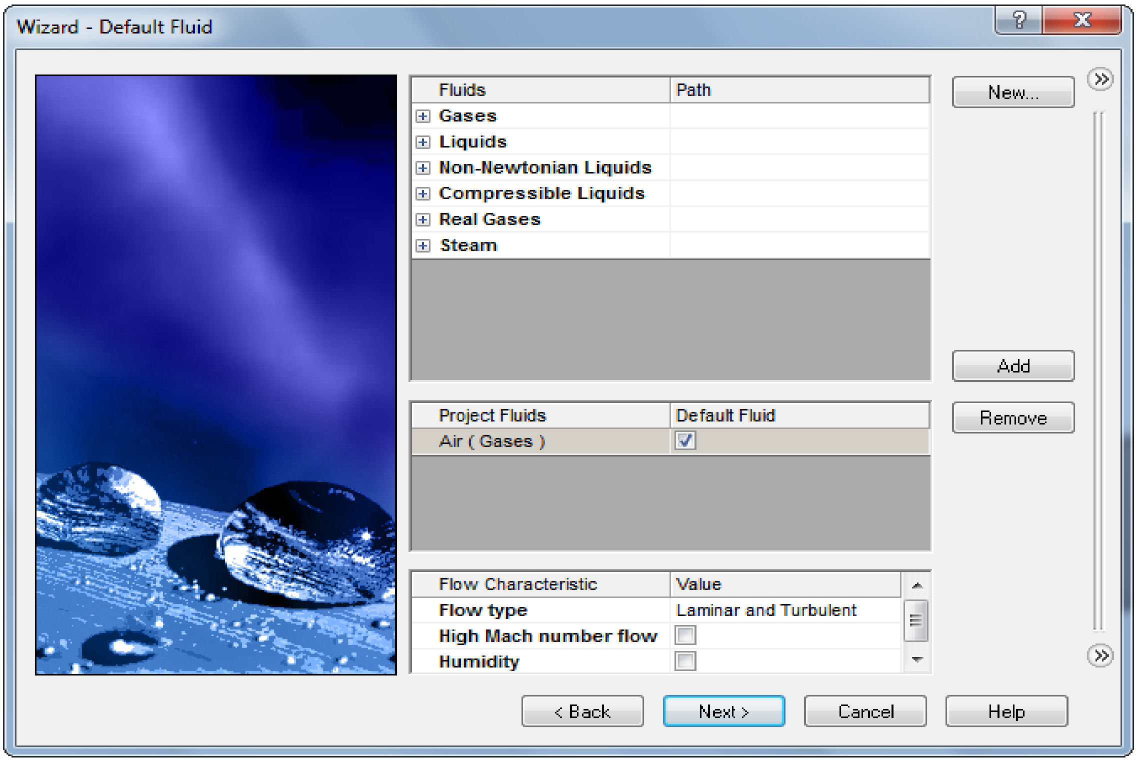
Select **Use current** configuration and click **Next**.



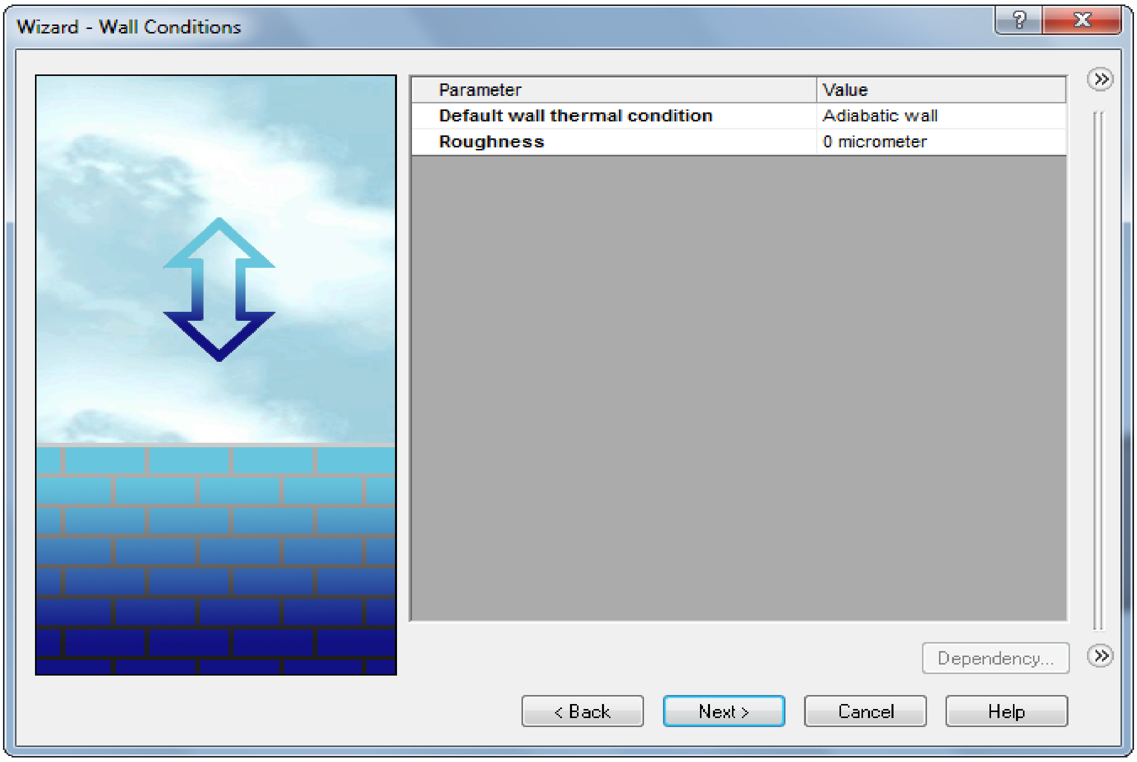
Now is time to choose the unit system. Select the **SI (m-kg-s)** units by clicking on it and click **Next**.



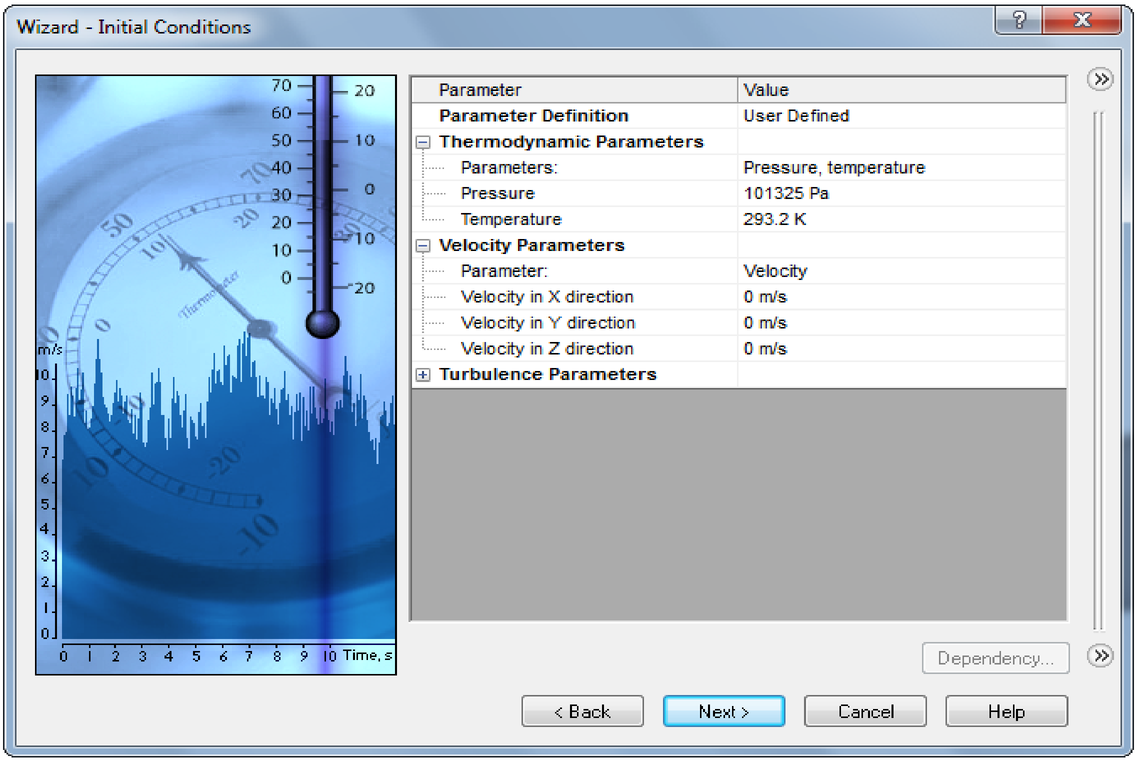
Here you can choose whether internal or external flow and the physical features involving the system. In our case is an **Internal** flow and does not involve heat conduction or any other options, click **Next**.



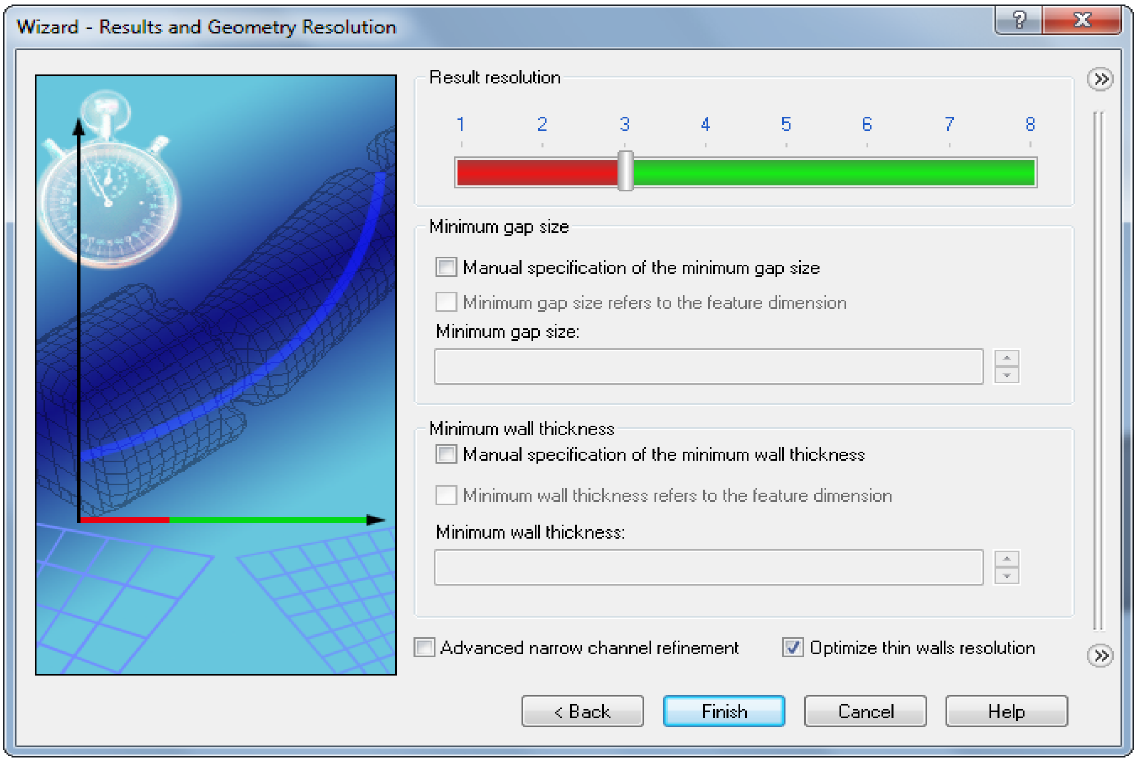
Now is time to choose the fluid. Open the **Gases** menu, double click on **Air** and click **Next**.



Here you can change the wall thermal conditions and its roughness. We will use the default **Wall** and **Roughness**, click **Next**.

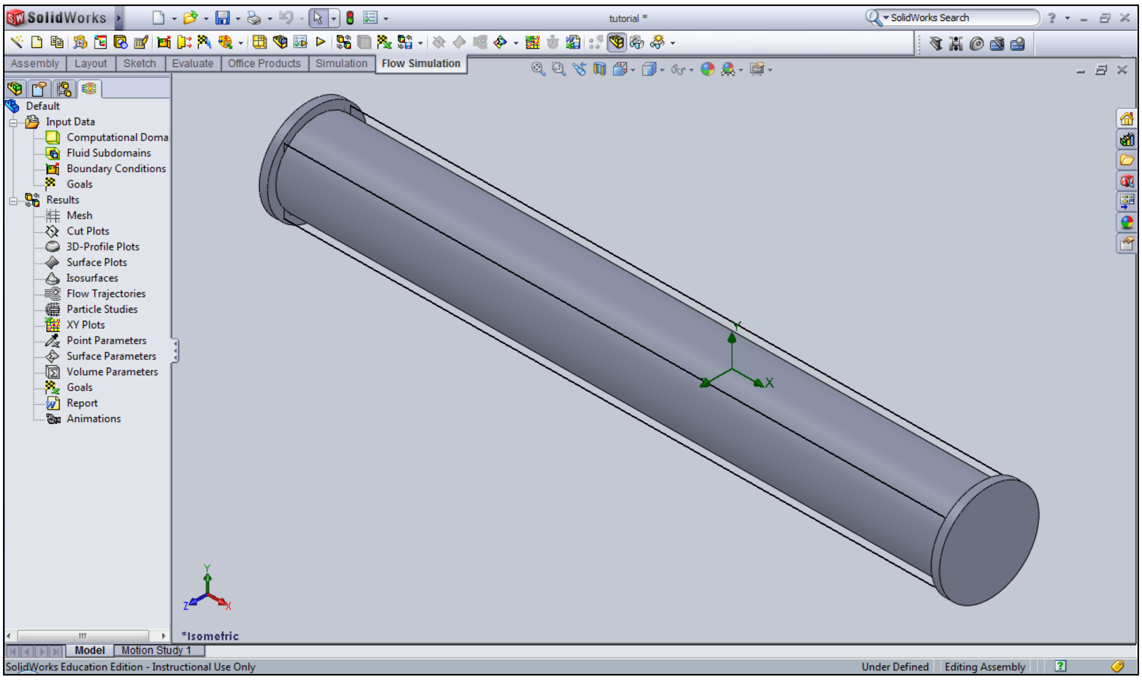


This is a resume table to check that all the initial conditions are ok. Click **Next**.



In this last step of the wizard you can change the result resolution. We will choose the default settings, click **Finish**.

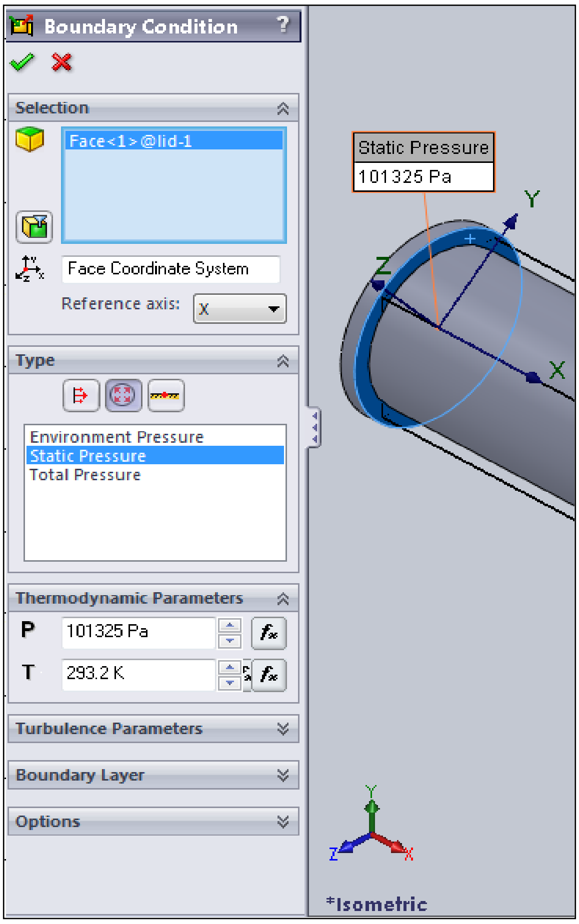
Notice the **Bounding Box** around the model. Flow Simulation has found the interior of your flow field and it is called the **Computational Domain**.

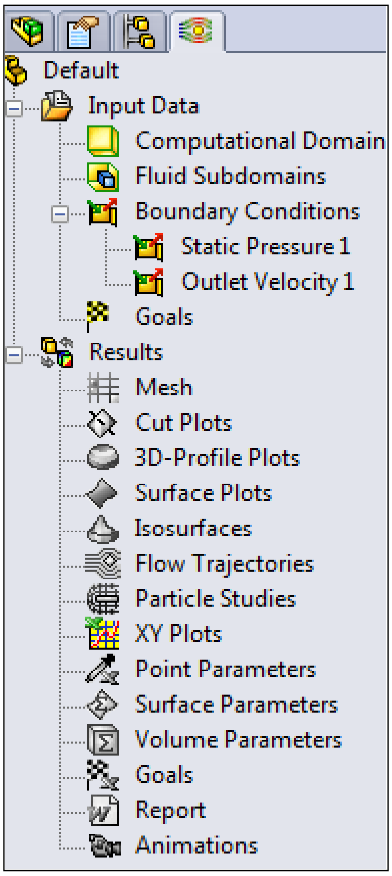


Notice as well the new tab, this is the flow simulation tree and you can select it. You will see two main branches. The first one is used to input data. The second one, labelled Results, has its leaves greyed out for the moment.

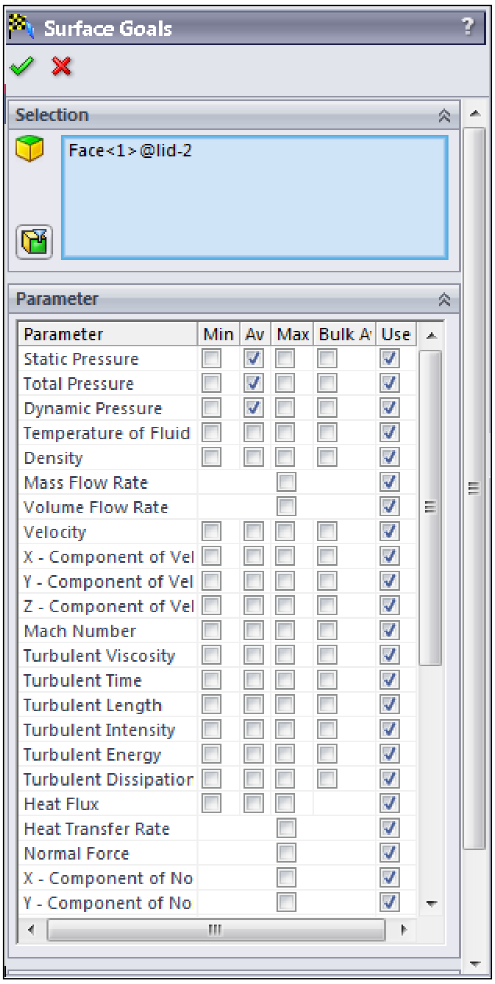
Set up boundary conditions and goals

Click on the **Input Data** branch and enter an inlet flow condition. Right click on **Boundary Conditions** and then click on **Insert Boundary Condition**. Here you can choose between Flow Opening, Pressure Opening and Wall. Each type of has several condition which can be chosen. Pick the inside face of the inlet lid, then select **Pressure Openings** and choose a **Static Pressure** of 101325 Pa. and a temperature of 293,15 K as shown in the figure to the left and then click **OK**.





Note the branch in the tree representing the boundary condition, which you can change. Now let's do outlet lid. It will be set to an outlet velocity of 4 m/s. Click on **Flow Opening**, then click **Outlet Velocity**, write 4 m/s and click **OK**.

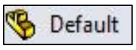


Now we need to set the goals. We need the static, dynamic and total pressures at the outlet so choose the outlet boundary condition icon

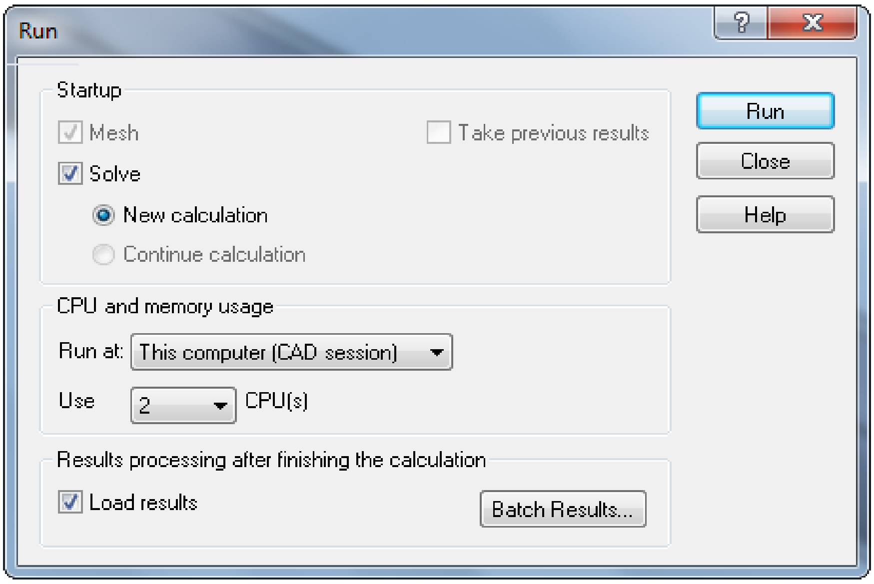
and right click on goals. Insert a surface goal and then pick the option to make average values for the three pressures a goal as shown below. Click **OK**

.

Now the program is ready to run the simulation. Right click on the top tree icon



and choose **Run**. You get a dialog box that allows you to use one or more CPU's, if you have them, for parallel processing.

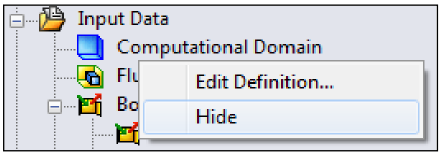


When you clock on the run button you will see the software monitoring window. After the measuring progress is completed you can graph the goals to monitor the progress. Click on the graph icon and select the three goal items specified earlier.

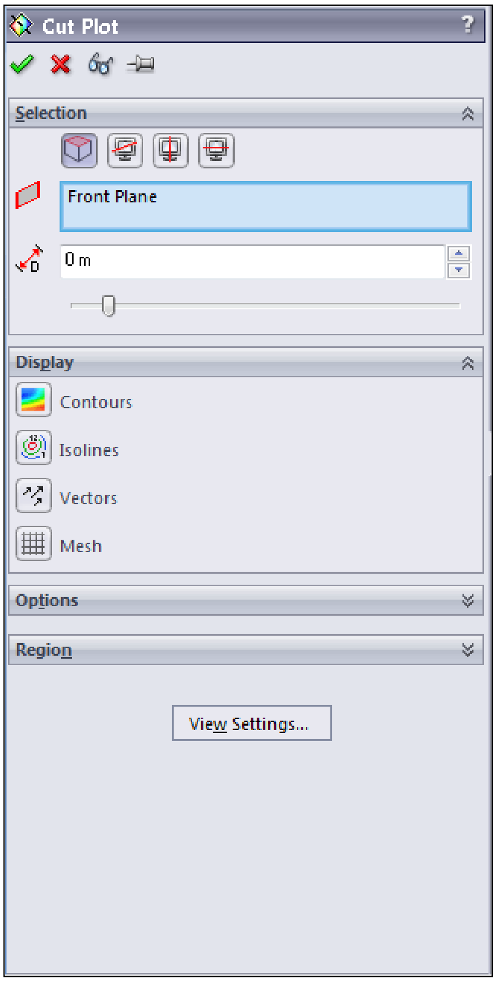
Now you can close the monitor window. Notice that the results tree has filled with colour indicating that the results are loaded.

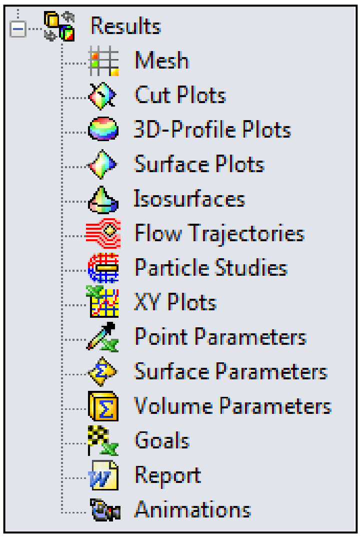
Results visualization.

First of all, hide the computational domain by right clicking on the icon Computational Domain and then click **Hide**. Now make the pipe transparent, so we can see the flow. Right click on the pipe and click **Change Transparency**.

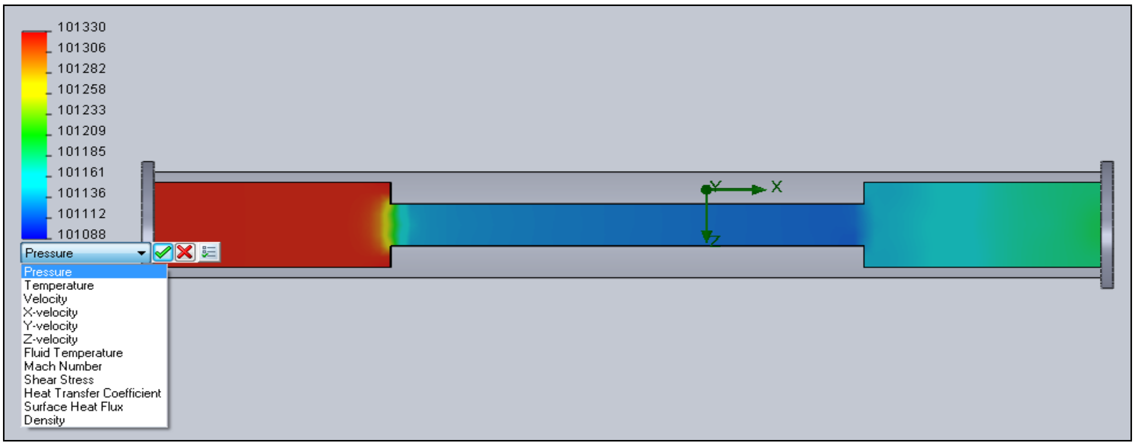


On the Results tree you can see many different ways to see the results. We are going to use some of them to see what the SolidWorks' Flow Simulation can offer. Right click on **Cut Plots** Cut Plots and insert a new cut plot. Now you can select on which plane you want to see the cut and which way you want to display the results; contours , isolines , vectors or mesh .

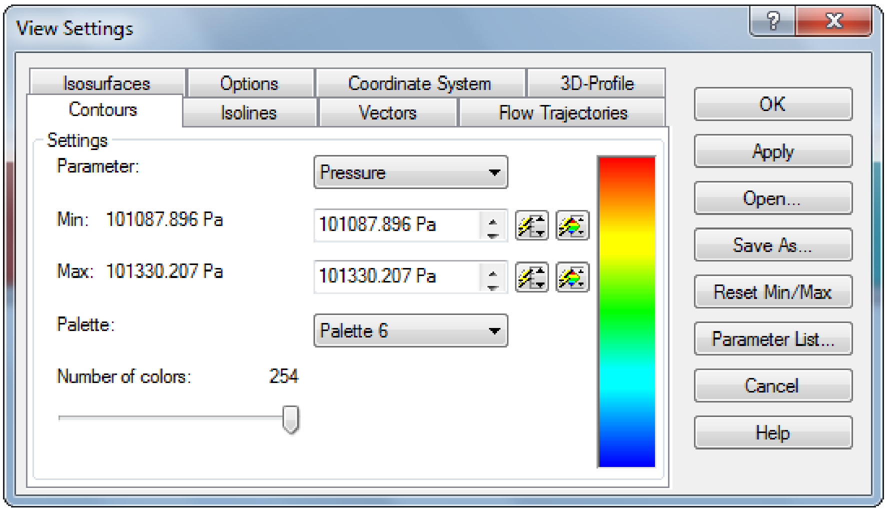


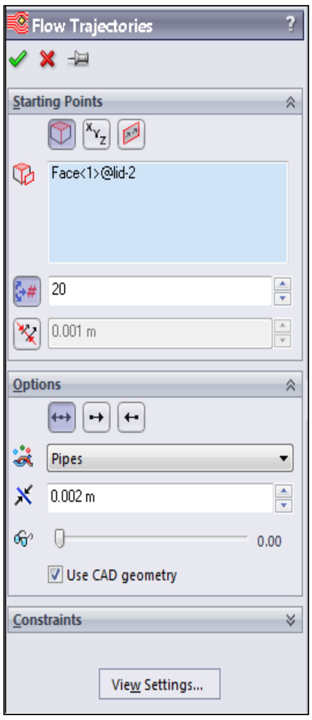


Select the **Top Plane** and dislplay **Contours** and then click **OK** . You can click on **Pressure** under the legend to change it to many other parameters you can show.



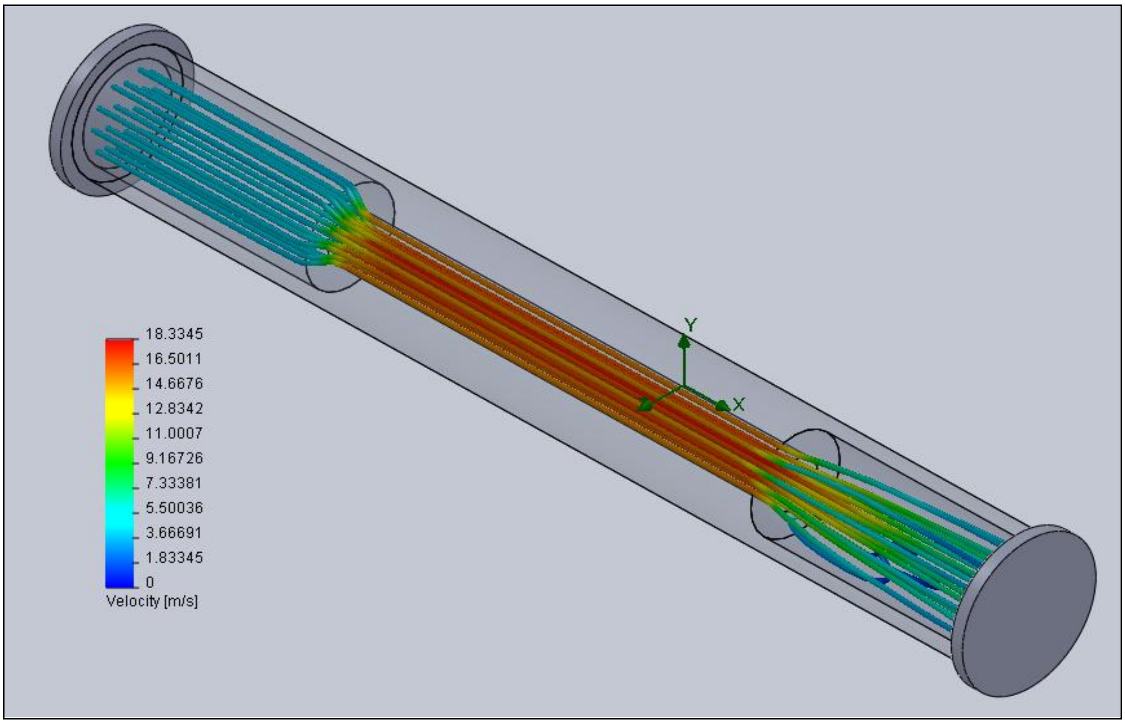
You can also double-click on the legend to open the cut plot **Settings** box and modify the legend, the magnitudes you want to see and some other options.



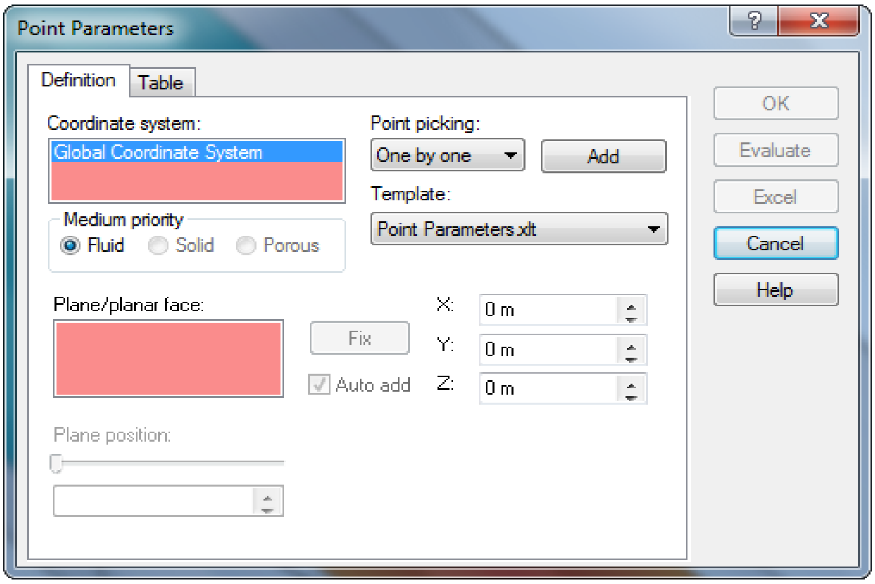


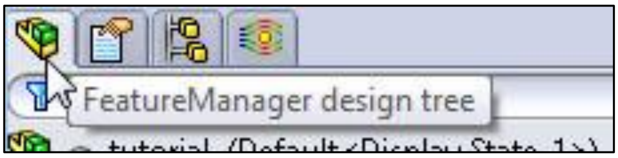
Now let's make a velocity flow trajectory starting from the inlet. First hide the **Cut Plot**, right click on [Cut Plot 1] and the select **Hide**. Select the inlet boundary condition [Static Pressure 1] and right click on the [Flow Trajectories] item to choose **Insert**. Leave the number of trajectories at 20 and draw the trajectories as pipes. Click **OK**.

This is how it should look like. It is possible to animate the flow trajectories. To do so is better to change the way the trajectories are drawn from pipes to spheres. Right click on Flow Trajectories 1 and select **Edit Definition**. Change to **Spheres** and select the number of trajectories you want (i.e. 50). You can see an example in Appendix X.x.x (CD).

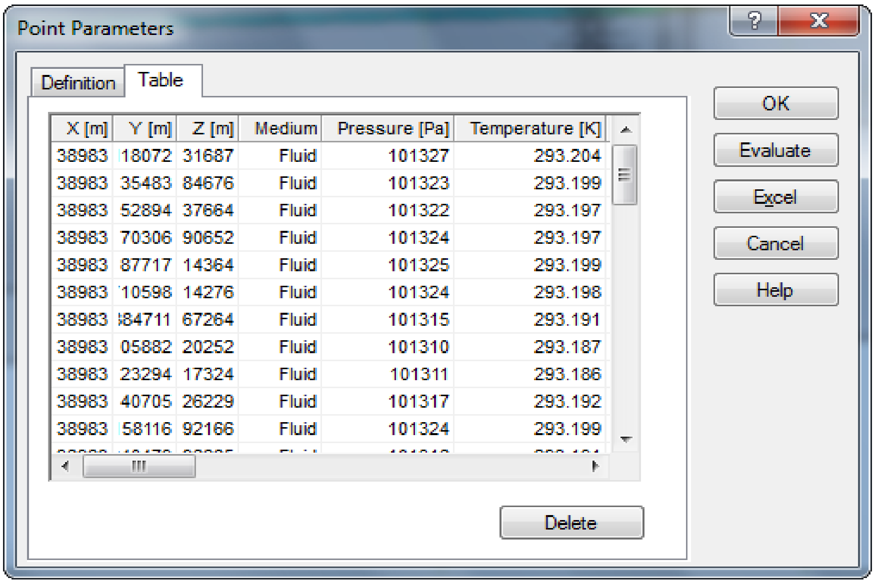


It is possible to determine the parameters in a concrete point inside the bounding box, and consequently the parameters on a determined area inside the computational domain. Right click on Point Parameters and the click on **Insert**. On the **Point Parameters** box, like the one below, you can choose to pick the points one by one or you can select a plane or a face and let the Flow Simulator make a point grid on it.



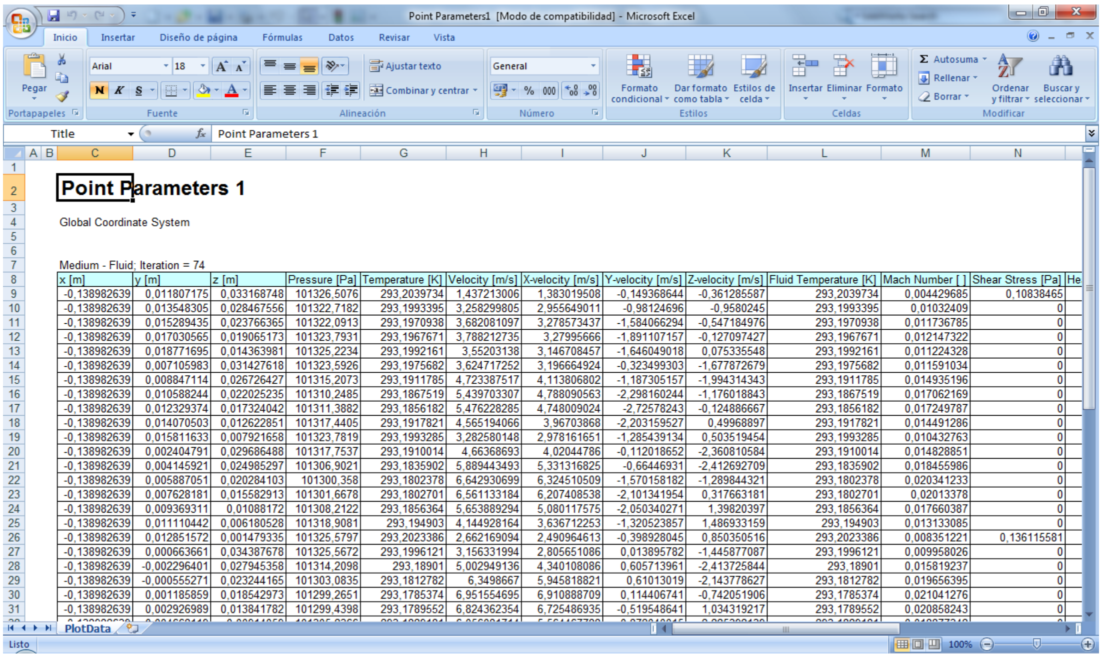


Select **Grid** in the **Point picking** menu and go to the **FeatureManager design tree** and select the **Right Plane**. Set the **Number of points** to 50 and the **Plane position** to -0.139. This point is placed 5 mm before the pipe narrowing. Now click on **Add** to add the point grid and then click on **Evaluate**. If you change to the **Table** tab you will see all the parameters detailed as it follows.



X [m]	Y [m]	Z [m]	Medium	Pressure [Pa]	Temperature [K]
38983	18072	31687	Fluid	101327	293.204
38983	35483	84676	Fluid	101323	293.199
38983	52894	37664	Fluid	101322	293.197
38983	70306	90652	Fluid	101324	293.197
38983	87717	14364	Fluid	101325	293.199
38983	10598	14276	Fluid	101324	293.198
38983	84711	67264	Fluid	101315	293.191
38983	05882	20252	Fluid	101310	293.187
38983	23294	17324	Fluid	101311	293.186
38983	40705	26229	Fluid	101317	293.192
38983	58116	92166	Fluid	101324	293.199

By clicking on **Excel**, Flow Simulator gives a Microsoft Excel™ sheet with all de data calculated from the points in the grid.

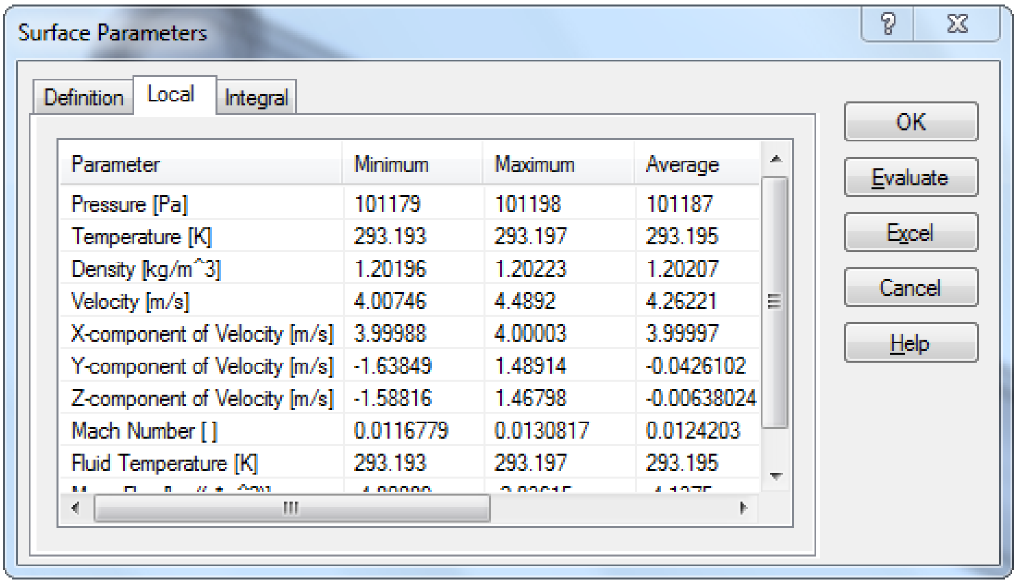


x [m]	y [m]	z [m]	Pressure [Pa]	Temperature [K]	Velocity [m/s]	X-velocity [m/s]	Y-velocity [m/s]	Z-velocity [m/s]	Fluid Temperature [K]	Mach Number [ ]	Shear Stress [Pa]
-0,138982639	0,011807175	0,033168748	101326,5076	293,2039734	1,437213006	1,383019508	-0,149368644	-0,361285587	293,2039734	0,004429685	0,10838465
-0,138982639	0,013548305	0,028467556	101322,7182	293,1993395	3,258299805	2,955649011	-0,98124696	-0,9580245	293,1993395	0,01032409	0
-0,138982639	0,015289435	0,023766365	101322,0913	293,1970938	3,682081097	3,278573437	-1,584066294	-0,547184976	293,1970938	0,011736785	0
-0,138982639	0,017030565	0,019065173	101323,7931	293,1967671	3,788212735	3,27995666	-1,891107157	-0,127097427	293,1967671	0,012147322	0
-0,138982639	0,018771695	0,014363981	101325,2234	293,1992161	3,55203138	3,146708457	-1,646049018	0,075335548	293,1992161	0,011224328	0
-0,138982639	0,007105983	0,031427618	101323,5926	293,1975682	3,624717252	3,196664924	-0,323499303	-1,677872679	293,1975682	0,011591034	0
-0,138982639	0,008847114	0,026726427	101315,2073	293,1911785	4,723387517	4,113806802	-1,187305157	-1,994314343	293,1911785	0,014935196	0
-0,138982639	0,010588244	0,022025235	101310,2485	293,1867519	5,439703307	4,788090563	-2,298160244	-1,176018843	293,1867519	0,017062169	0
-0,138982639	0,012329374	0,017324042	101311,3882	293,1856182	5,476228285	4,748009024	-2,72578243	-0,124886667	293,1856182	0,017249787	0
-0,138982639	0,014070503	0,012622851	101317,4405	293,1917821	4,565194066	3,96703868	-2,203159527	0,49968897	293,1917821	0,014491286	0
-0,138982639	0,015811633	0,007921658	101323,7819	293,1993285	3,282580148	2,978161651	-1,285439134	0,503519454	293,1993285	0,010432763	0
-0,138982639	0,002404791	0,029686488	101317,7537	293,1910014	4,66368693	4,02044786	-0,112018652	-2,360810584	293,1910014	0,014828851	0
-0,138982639	0,004145921	0,024985297	101306,9021	293,1835902	5,889443493	5,331316825	-0,66446931	-2,412692709	293,1835902	0,018455986	0
-0,138982639	0,005887051	0,020284103	101300,358	293,1802378	6,642930699	6,324510509	-1,570158182	-1,289844321	293,1802378	0,020341233	0
-0,138982639	0,007628181	0,015582913	101301,6678	293,1802701	6,561133184	6,207408538	-2,101341954	0,317663181	293,1802701	0,02013378	0
-0,138982639	0,009369311	0,01088172	101308,2122	293,1856364	5,653889294	5,080117575	-2,050340271	1,39820397	293,1856364	0,017660387	0
-0,138982639	0,011110442	0,006180528	101318,9081	293,194903	4,144928164	3,636712253	-1,320523857	1,486933159	293,194903	0,013133085	0
-0,138982639	0,012851572	0,001479335	101325,5797	293,2023386	2,662169094	2,490964613	-0,398928045	0,850350516	293,2023386	0,008351221	0,136115581
-0,138982639	0,000663661	0,034387678	101325,5672	293,1996121	3,156331994	2,805651086	0,013895782	-1,445877087	293,1996121	0,009958026	0
-0,138982639	-0,002296401	0,027945358	101314,2098	293,18901	5,002949136	4,340108086	0,605713961	-2,413725844	293,18901	0,015819237	0
-0,138982639	-0,000555271	0,023244165	101303,0835	293,1812782	6,3498667	5,945818821	0,61013019	-2,143778627	293,1812782	0,019656395	0
-0,138982639	0,001185859	0,018542973	101299,2651	293,1785374	6,951554695	6,910888709	0,114406741	-0,742051906	293,1785374	0,021041276	0
-0,138982639	0,002926989	0,013841782	101299,4398	293,1789552	6,824362354	6,725486935	-0,519548641	1,034319217	293,1789552	0,020858243	0

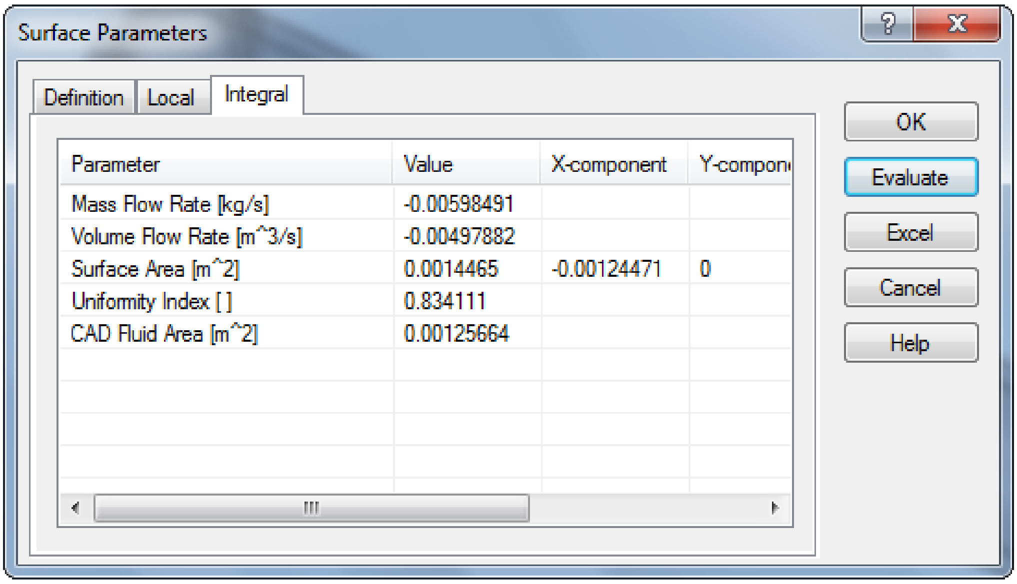
Now let's check the outlet parameters. Select the outlet lid 

and right click on **Surface Parameters** 

and click on **Insert**. Now you can see the local and integral values at the outlet. Click on **Evaluate** and the on the **Local** tab. You can appreciate that the velocity on the lid gets 4.26 m/s and we configured 4 m/s at the outlet. This is due to the fact that the velocity is a vector, and the X-component of it is the one we configured to be 4 m/s, and it is 3.99 m/s, so the value is the expected.



Clicking on the **Integral** tab will show the integrated values for the outlet lid.



D. DRAWINGS

In the following pages there are some drawings of the models we have been working with, as well as the diffusion box.

E. CD

In the attached CD-Room it can be found, in digital format, some extra data and information about the project.

Included, there are:

- Whole report in PDF format
- Complete EES model
- Pictures of the project realization
- Videos of the project realization