Химикотехнологичният и металургичен университет

Universidade de Vigo

Departamento de Ingeniería Mecánica

3D DESIGN AND ANALYSIS OF PNEUMATIC DEVICE

Student: Jose Rodríguez Rodríguez
Tutor: Prof. Veselin Iliev
Assist. Iliyan Lesev

Academic year 2015-2016

ERROR! REFERENCE SOURCE NOT FOUND.

INDEX

ERROR! REFERENCE SOURCE NOT FOUND.

ERROR! REFERENCE SOURCE NOT FOUND.

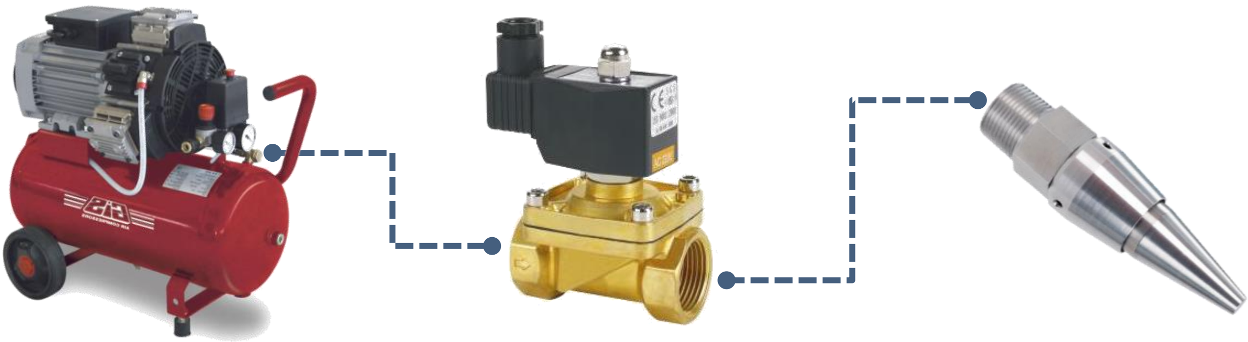
1 INTRODUCTION

Nowadays, compressed air is widely used, from a small compressor used to blow up wheels of cars to a big pneumatic hammer that the construction worker uses to get huge force and velocity at the same time. As well as in could be found both in factory tools (hammers, drills, painting machines...) as in leisure devices (compressed air tank to scuba diving, paintball equipment…).

The main reason why it is so used is because it has huge advantages as: cheap (almost free, we only spend money in the electricity consumed by our compressor), everywhere (wherever we can take a compressor, we can have compressed air) and clean and dry (we obtain only air, without water or dirt). In addition, it is safe to use (even if an accident happens), environment friendly (do not produce waste) and fast work (quickly switch in mechanical applications).

By another hand, in many processes in factories is needed get a clean surface (for example painting, glue or some chemical treatments). When it is only necessary clean the surface of dust, compressed air is one of the best choices. It is due to the fact that compressed air is free of another particles and dry, and with the suitable pressure, it has force enough to throw dust away.

In our case, we focused in a pneumatic system used to clean dust in factories; that include the compressor, a device to regulate and cur the airflow and the nozzle through the air go out, at it is shown in Fig. 1.



Air compressor ***Solenoid valve*** ***Nozzle***

Fig. 1: Dust blowing system

ERROR! REFERENCE SOURCE NOT FOUND.

1.1 Compressors

An air compressor is a machine that converts mechanical energy (provided by an electric, diesel or gasoline motor), into potential energy (specifically in compressed air).

By one of several methods (which will be explained in later), air compressor forces more and more air into a storage tank, increasing the pressure until its upper limit. Then, the compressed air is kept in the tank until we need use it.

Following are the most used methods of operation:

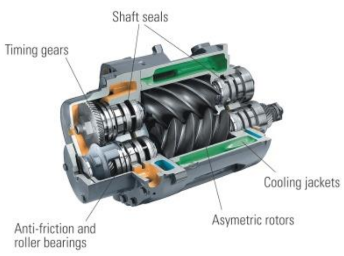


Fig. 3: Rotary screw compressor

-**Piston-type** *(*Fig. 2*)*: when the piston go down (inter valve opened, outer valve closed) air go into the cylinder, and then the inter valve is closed and the outer valve opened to push the air to the tank increasing the pressure in that *[01]*.

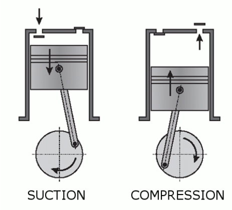


Fig. 2: Piston-type compressor

-**Rotary screw compressor** *(*Fig. 3*)*: use positive-displacement compression by matching two helical screws that, when turned, guide air into a chamber, whose volume is decreased as the screws turn *[01]*.

-**Vane compressor** *(*Fig. 4*)*: use a slotted rotor with varied blade placement to guide air into a chamber and compress the volume *[01]*.

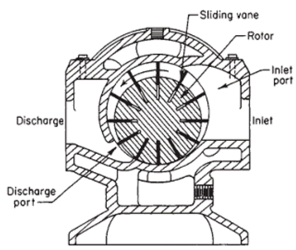


Fig. 4: Vane compressor

However, for us there are other features as pressure, power, air capacity or air flow that are more important than operation system when we need choose one.

The following table (Table 1) shows some commercial compressors with different features and prizes according with our project needs.

Table 1: Compressors

					
Model	Cevik Pro 40 Silent	CIS TOP300/ 100/CAR/M	Honda CTA5090412	John Deere AC2-80ES	Cevik AB500/10T
Presure	8 atm	10 atm	10 atm	12 atm	10 atm
Power	2 HP	2 HP	5 HP	5 HP	10 HP
Air Capacity	40 L	100 L	16 L	300 L	500 L
Air flow	335 l/min	280 l/min	195 l/min	480 l/min	912 l/min
Rpm	1400 rpm	-	2950 rpm	-	-
Power type	electric (230V)	electric (230V)	Petrol	electric (230V)	electric (400V)
Weight	38 kg	58 kg	34 kg	238 kg	260 kg
Prize	347,00 €	802,08 €	847,00 €	1.627,99 €	2.730,00 €

Also, factories sometimes have a compressed air system that provide whole factory by a big compressor which can supply a constant pressure without problems of capacity or shortage of air flow; but in our case we are going to suppose that we only need compressed air to this machine.

ERROR! REFERENCE SOURCE NOT FOUND.

1.2 Magnetic/solenoid valve

We have already air compressed in our tank, but now we need to be able to control it in the way we want to. This problem was solved setting a valve between the compressor's tank and the nozzle.

Compressors use to have a mechanic valve, but if we are thinking in use our nozzle in an automatic machine we should use a magnetic/solenoid valve which can be open or close by electrical current, becoming unneeded to have someone doing it.

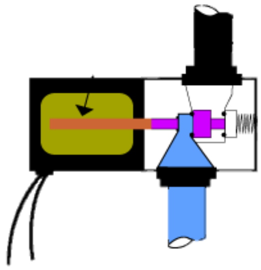


Fig. 5: valve NC no-energized

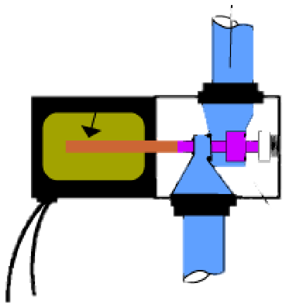


Fig. 6: valve NC energized

Operation mode of solenoid valves are simple, there is a valve actuator surrounding by a coil winding; when electricity go through the coil it generate motion that can be to close the valve or open it (that depends of our kind of valve: normally open (NO) or normally closed (NC)); and when the coil in this valve is de-energized the actuator come back to the first position due to the spring force. Fig. 5 shows a normally closed (NC) valve no-energized in which case the valve is closed don't allowing air go out; for the another hand when it is energized, the actuator move and flow can leave (Fig. 6) until the valve is de-energized again.

At the time to choose one, we should consider carefully the features of different valves. First, one is design, which is defined by two numbers: first means number of ports though air can go in and go out, and the second one is the number of positions (2/2, 3/2, 5/2…); this feature and choose between NC or NO depends of how are going to work.

Then, according with our flow, compressor, equipment and environment we have to see carefully other parameters as nominal diameter, operation pressure, operation temperature, flow rate, supply voltage or protection class.

Nowadays, cycle frequency (or switch rate) is usually above 100Hz, so that is enough for almost all cases; but even if we need a higher one, it would be easy to find it.

1.3 Nozzles

Nozzles are the last part before flow leaves the machine, so this make the parameters of the nozzle become the most important to define features of flow; therefore, we should choose it carefully.

If we are going to buy a nozzle or design one we need in both cases know some important information before do it:

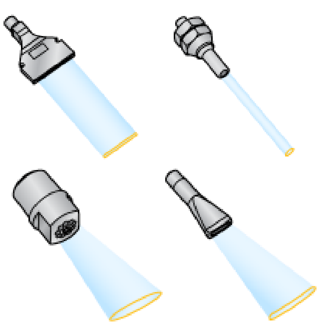


Fig. 7: Outlet shapes

First of all, we need know some characteristics of our air compressor, as work pressure and capacity. Each nozzle is designed for one range of pressure where it works better, so we have to know the maximum and minimum pressures between we are going to work. In addition, we should know the pressure in the walls to choose a material that can resist it safely, and depending of the fluid, could be important the surface of the material and its corrosion resistance (not in our case in which one we used air). Moreover, if we have a low tank capacity or we want to save compressed air (therefore energy), we should think in make a hole in our nozzle through more air can enter to get the maximum amount of air in the outlet of our nozzle with the minimum amount of air in the inlet.

ERROR! REFERENCE SOURCE NOT FOUND.

Another main feature of a nozzle is the shape in the outlet, which can be straight, circular, square, rectangular or another shape *[01]*[1], and should be chosen according to obtain the best results to our aim (Fig. 7).

1.4 Our case

To our research, we used the air compressor “CIS TOP300/100/CAR/M” with the following features *[02]*[2]:

- • Max. Pressure: 10 bar
- • Air flow: 280 L/min
- • Tank Capacity: 100 L
- • Power: 1500 W
- • Electrical connection: 230V/50Hz
- • Noisy: 69 dB
- • Weigh: 58 Kg
- • Price: 802.08 €

We chose this compressor because was the biggest one that we had available in university, which also research our needs about pressure, capacity and fluid flow.

With the aim of be able to cut and control our fluid flow in a better way, we used a solenoid valve “ASCO Series 353” *[03]*[3] because it had suitable features to this aim. We as looking for a valve that was 2/2 (that mean two ports and two positions), normally closed and that resist more than 10 atm of pressure. This valve had all these, and in addition, a proper price.

About nozzles, we made studies about several commercial nozzles that we had in the laboratory, which ones also are often used in factories.

1.5 Theoretical calculations *[04]*[4]

The fluid losses in the course of the motion of a fluid are due to the irreversible transformation of mechanical energy into heat. This energy transformation is due to the molecular and turbulent viscosity of the moving medium. There are two different types of fluid losses: frictional losses ($\mathbf{\Delta H_{fr}}$) and local losses ($\mathbf{\Delta H_{l}}$).

Frictional losses are due to the viscosity (molecular and turbulent) of the fluid and take place along the entire length of the pipe; on the another hand, local losses appear at a disturbance of the normal flow of the stream, such as its separation from the wall and the formation of eddies at places of alternation of the pipe configuration or at obstacles in the pipe.

The summing is conducted according to the principle of superposition of losses, according to which the total loss is equal to the arithmetic sum of the friction and local losses:

$$\boldsymbol{\Delta H_{sum} = \Delta H_{fr} + \Delta H_{l}\ [kg/m^2]}$$

In practice, it is necessary to take $\mathbf{\Delta H_{fr}}$ into account only for relatively long fittings or when its value is commensurable with $\mathbf{\Delta H_{l}}$.

Another important value to next equations is the fluid-resistance coefficient (ζ) which represents the ratio of pressure loss **ΔH** to the dynamic pressure in the section **F** considered:

$$\boldsymbol{\zeta = \frac{\Delta H}{\frac{\gamma \omega^2}{2g}}}$$

And if we change this formula properly we can obtain the summing of losses (**ΔH)** depending only of the fluid-resistance coefficient (ζ), mean stream velocity (**ω),** specific gravity of the flowing medium(**γ)** and gravitational acceleration (**g**).

$$\Delta H = \zeta \frac{\gamma \omega^2}{2g}$$

With these basics notions, we are going to study how would be able solve this flow case in a simple nozzle (without holes) as the next one (Fig. 8):

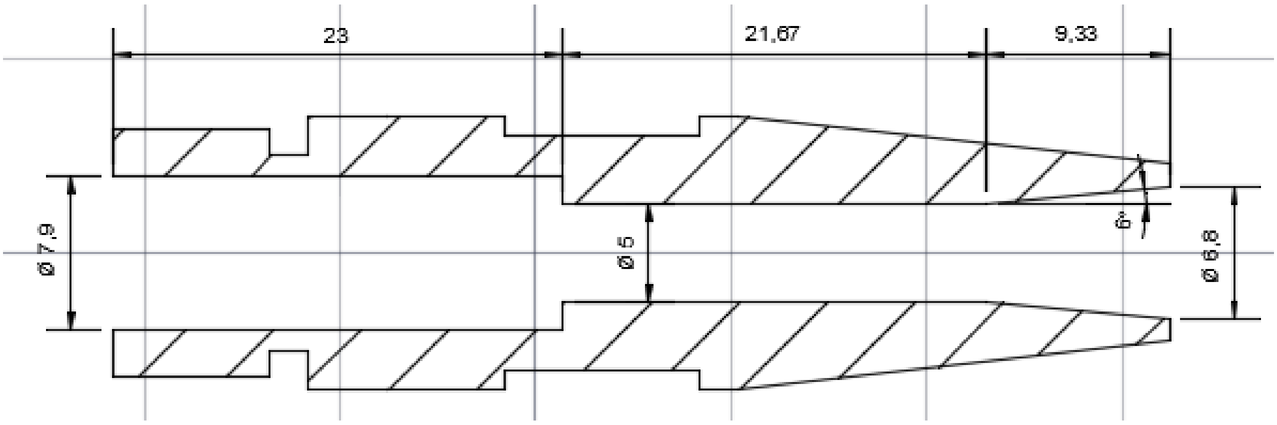


Fig. 8: Simple nozzle

As you can see, this nozzle has three different parts where we must use different equations: inlet where diameter is constant, the hole where diameter change suddenly and the exit with a diameter increasing slowly.

1.5.1 Inlet section

We suppose that the flow pattern is a sudden contraction (Fig. 9) due that the fact of our nozzle will be connected in the inner of an air hosepipe.

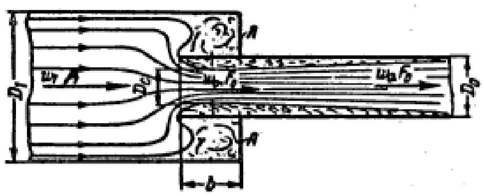


Fig. 9: Flow pattern at sudden contraction

The phenomenon observed in inlet stretches in which the stream suddenly contracts (passes suddenly from a large section $\mathbf{F_1}$ to a smaller section $\mathbf{F_0}$) is similar to the one observed at the entrance on a straight inlet from a very large volume; the only difference is that here the resistance coefficient is a function of the ratio $\mathbf{F_0/F_1}$. This coefficient would be calculated by the following formula:

$$\zeta = \frac{\Delta H}{\frac{\gamma \omega^2}{2g}} = \zeta' \left(1 - \frac{F_0}{F_1}\right)$$

where **ζ'** is a coefficient depending on the shape of the inlet edge of the narrow channel which can be obtained from a table.

ERROR! REFERENCE SOURCE NOT FOUND.

1.5.2 Hole

In general, the passage of a stream from one volume into another through a hole in a wall is accompanied by the phenomena illustrated in Fig. 10. The stream passes from channel 1; located before the partition **A** with orifice of diameter $\mathbf{D_e}$, into channel 2, located behind this partition (both channels cannot be smaller than the cross section of the orifice). The passage of the stream through the orifice is accompanied by the bending of the trajectories of the particles, the inertial forces causing them to continue their motion toward the orifice axis.

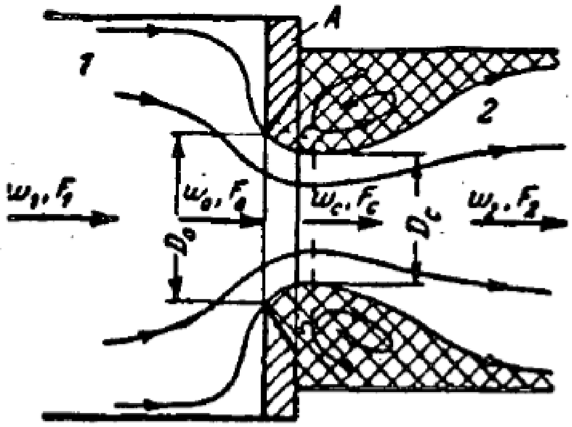


Fig. 10: Stream through a hole

The resistance coefficient of the stream passage through a sharp-edged orifice (Fig. 10) is calculated in the general case ($R_e = \frac{\omega_a D_h}{v} > 10^5$) by the formula:

$$\zeta = \frac{\Delta \mathrm{H}}{\frac{\gamma\omega^2}{2g}} = \left(1 + 0.707\sqrt{1 - \frac{F_0}{F_2}} - \frac{F_0}{F_2}\right)^2$$

At $R_e < 10^5$ the resistance coefficient may be calculated by the approximate formula:

$$\zeta = \frac{\Delta \mathrm{H}}{\frac{\gamma\omega^2}{2g}} = \left(\frac{1}{\varphi^2}\right) + \frac{0.342}{(\omega^{Re})^2}\left(1 + 0.707\sqrt{1 - \frac{F_0}{F_2}} - \frac{F_0}{F_2}\right)^2$$

where **φ**= velocity coefficient at discharge from a sharp-edged orifice.

1.5.3 Exit stretches

When a stream flows out from a pipe, independent of the exit conditions, the kinetic energy of discharged get is always lost to the pipe, and the resistance coefficient of the discharge in terms of velocity in the narrow section will be equal to:

$$\zeta = \frac{\Delta \mathrm{H}}{\frac{\gamma\omega^2}{2g}} = \frac{\Delta \mathrm{H}_{st}}{\frac{\gamma\omega^2}{2g}} + \frac{\Delta \mathrm{H}_{dyn}}{\frac{\gamma\omega^2}{2g}} = \zeta_{st} + \zeta_{dyn}$$

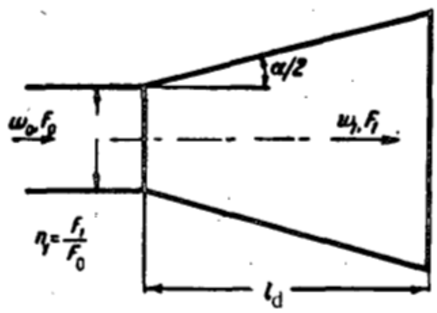


Fig. 11: Circular rectilinear diffuser

The pressure losses in a diffuser in the case of free discharge into a large volume (our case) are made up of the loss in the diffuser proper ζ_d and the loss of dynamic pressure at the exit ζ_{ex}:

$$\zeta = \frac{\Delta \mathrm{H}}{\frac{\gamma\omega^2}{2g}} = \zeta_d + \zeta_{ex} = \zeta_d + \frac{N}{n^2}$$

where $n = \frac{F_{ex}}{F_e}$ and $\mathbf{F_e}$ and $\mathbf{F_{ex}}$ are the area of narrowest and exit sections, respectively, in m^2.

ERROR! REFERENCE SOURCE NOT FOUND.

Velocity distribution at the discharge of a diffuser is assumed to be uniform **(N=1)**; to compensate for this assumption, a corrective coefficient in the form is introduced **(1+σ')**:

$$\boldsymbol{\zeta = \frac{\Delta H}{\frac{\gamma\omega^2}{2g}} = (1 + \sigma')\left(\zeta_d + \frac{1}{n^2}\right) = (1 + \sigma')\zeta_{cal}}$$

where **σ'** is the central angle of divergence of the diffuser, and $\boldsymbol{\zeta_{cal} = \zeta_{fr} + \zeta_{exp} + \frac{1}{n^2}}$ **,** where $\boldsymbol{\zeta_{fr}}$ and $\boldsymbol{\zeta_{exp}}$ are friction coefficient and resistance coefficient due to diffuser expansion, determined from the data of diagrams.

ERROR! REFERENCE SOURCE NOT FOUND.

2 PREPARING SIMULATIONS

2.1 Nozzle 3D Design

In this project, to make the 3D design we used a CAD program called Autodesk Inventor *[05]* [5], but could have been used another one like AutoCAD, SolidWorks or even in ANSYS.

First of all, we needed to measure our real nozzle (which we were going to simulate) with precision tools, as a measuring gauge (Fig. 12), and made the scale drawing with the measures taken (Fig. 13). Results depend from this design, so we should take measure several times to make sure that they are right.

In this case, we used AutoCAD to make drawings; it was due it is one of the most popular computer programs to do it.

Fig. 12: Taking measures

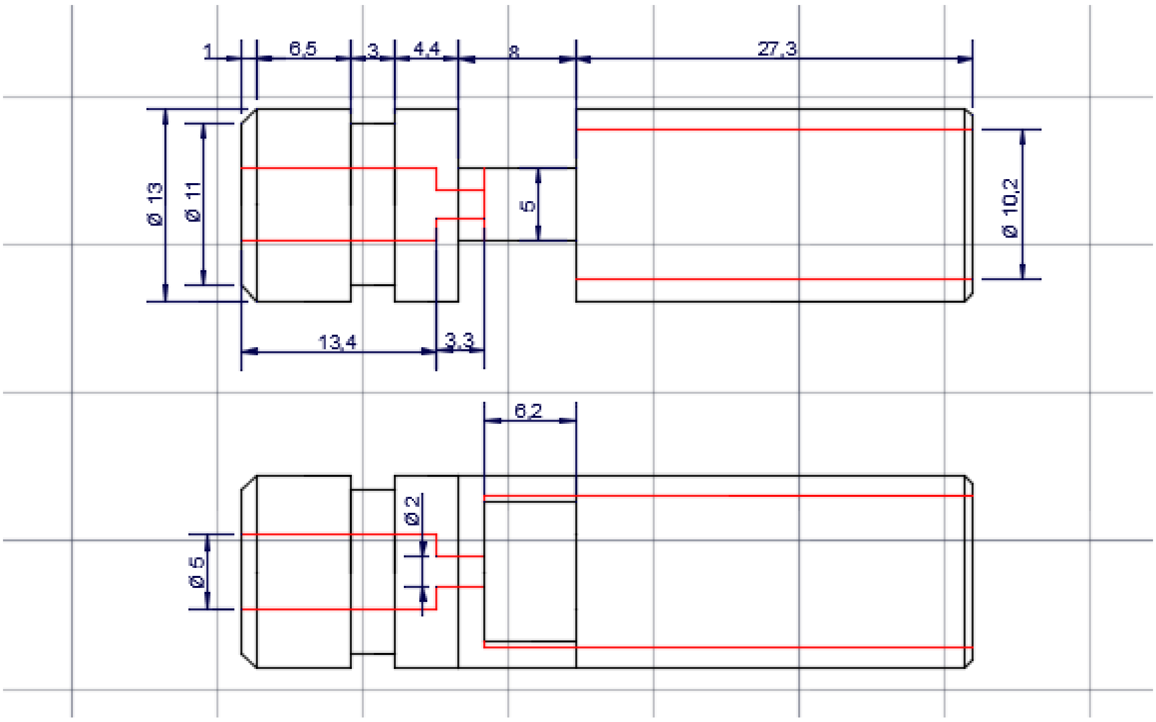


Fig. 13: Drawings in AutoCAD

After have the drawings with all measures clear, we could begin to make the 3D model in Autodesk Inventor.

The first step was to make the outer shape. To do that, we only had to choose our main plane of work and draw there a circle and extrude it; after that, we did two more next to the previous one until we had three cylinders joined (as in Fig. 14).

ERROR! REFERENCE SOURCE NOT FOUND.

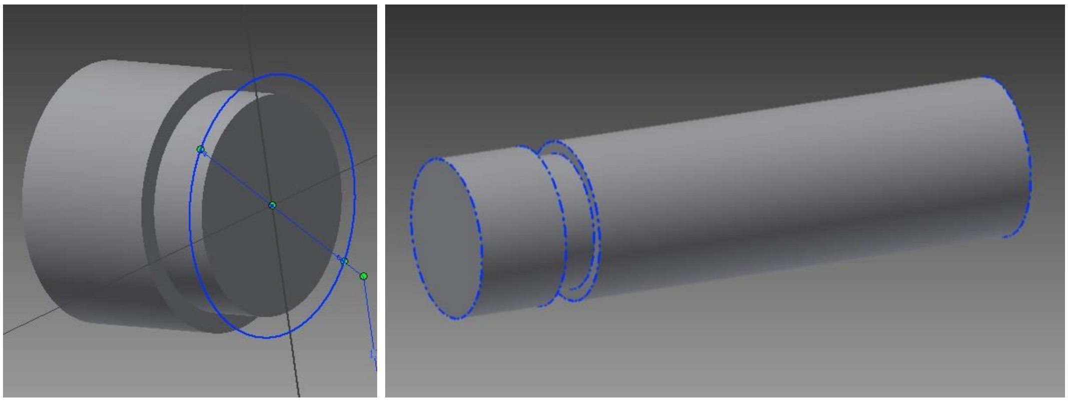


Fig. 14: Creating outer shape

Next step was to make the cylinder holes (CAD's programs usually have a specific tool to make it only in one step) (Fig. 15). In this case we made three holes, each one with a different size.

First one was the inlet of our nozzle (hole of 4.9 mm of diameter and 13.2 mm of length), second one was the hole through fluid flow go from inlet chamber to outlet chamber (diameter of 2 mm in this case); and the last hole was the outlet chamber, which one was created from the another side with a diameter of 10.2 mm and a length of 32.4mm. All these holes were done with a flat drill point and as a simple hole.

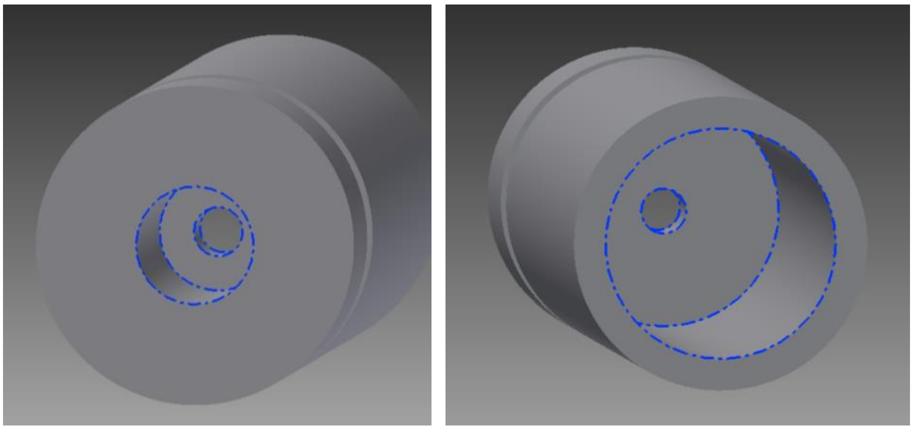


Fig. 15: Holes

In our nozzle, there were another two holes on both sides of it through the air enter. This operation was done drawing a sketch in a new plane that cuts our nozzle and using extrude, but in this case, we had to choose "cut" instead "join" in extrude options (Fig. 16).

Same operation was doing in the opposite side.

ERROR! REFERENCE SOURCE NOT FOUND.

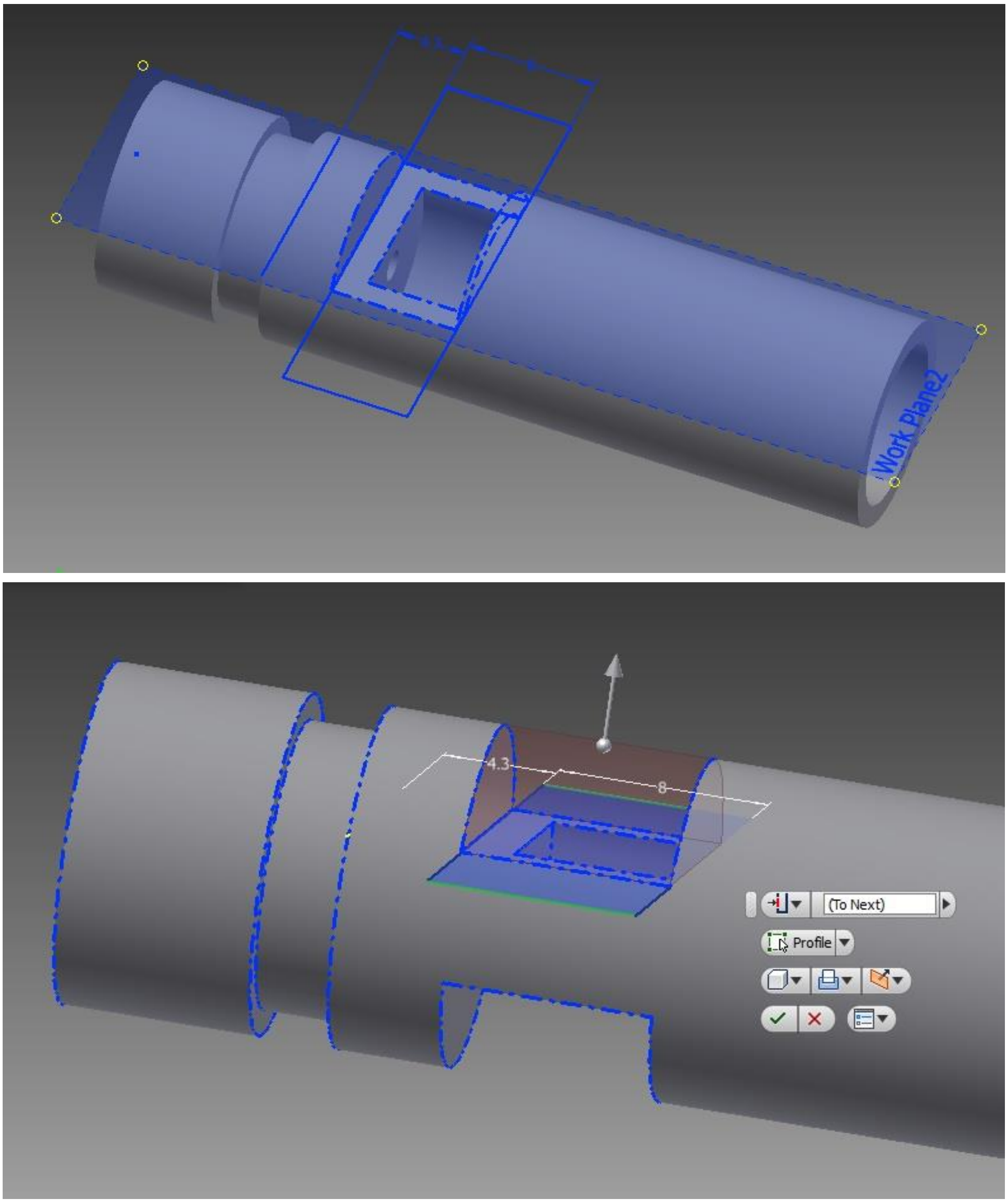


Fig. 16: Side holes operation

Finally, we made two chambers, one in the outer edge of the inlet and another one in the outer edge of the outlet; both with the same size, 0.5mm.

With this last step our nozzle model was finished (Fig. 17).

ERROR! REFERENCE SOURCE NOT FOUND.

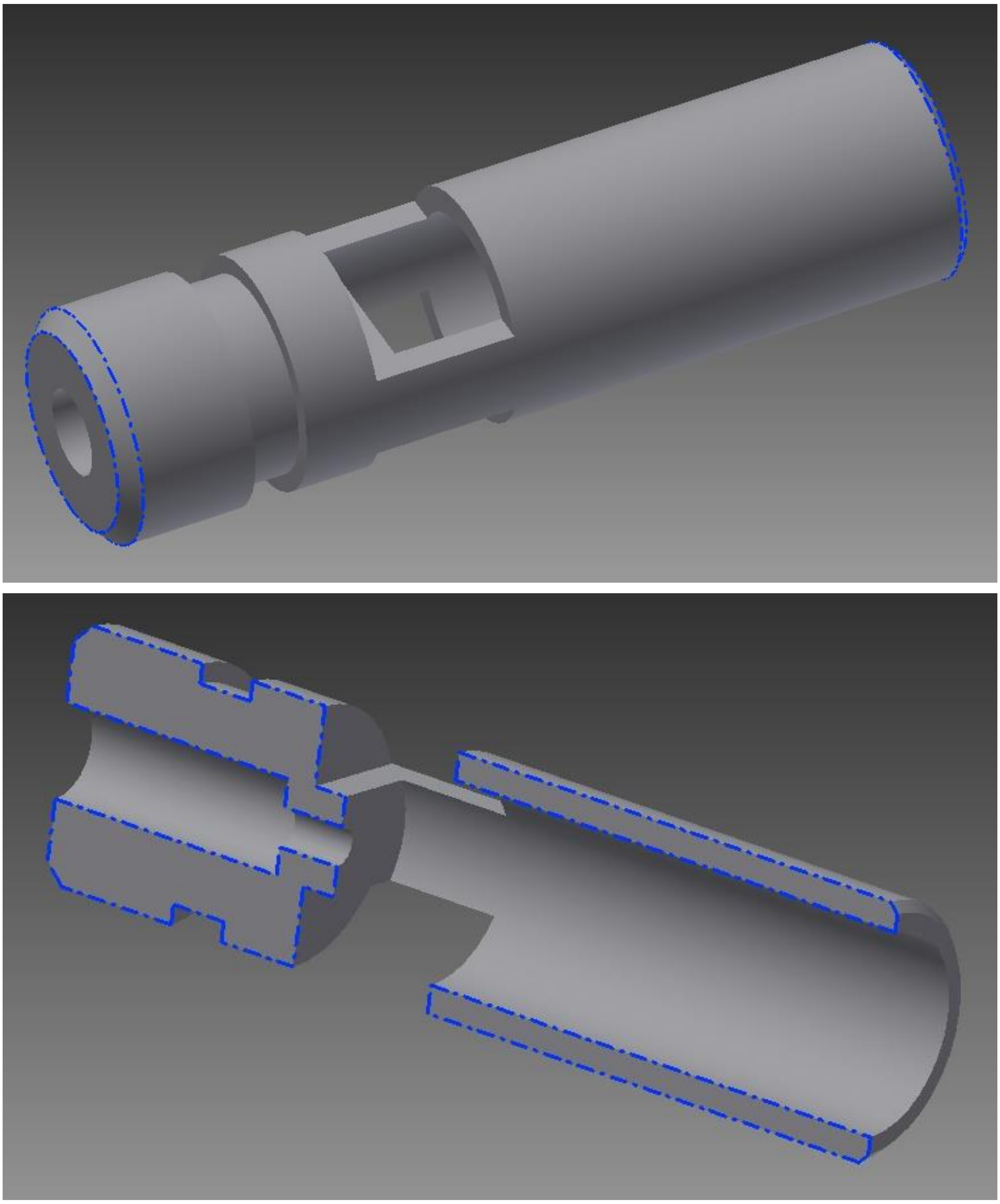


Fig. 17: Final geometry

2.2 ANSYS Geometry

Following researches were made in ANSYS, so we needed to export geometry to that computer program to be able to work in it.

Once we got the nozzle 3D design, we exported it to ANSYS (which is compatible almost with all the most famous CAD programs). The procedure was very simple: we selected export in Autodesk Inventor, saved the file, and after that, we clicked in import on ANSYS and selected that file to load it.

ERROR! REFERENCE SOURCE NOT FOUND.

After that, we created a rectangular body (0.25x0.25x0.3m) surrounding the nozzle, which became the region to analyse. This surrounding body must have the walls far enough from the nozzle to do not have influence of the boundaries on results of simulations. Finally we supressed the nozzle from the big body since we were only interested in fluid; not in the nozzle as a solid (at least at that moment).

In addition, when geometry is symmetric and process allow it (our case), we should split it by four equal quarters; so we did it and defined the symmetry faces (Fig. 18). This is very helpful because it reduces the geometry, therefore, number of elements and equations to solve, reducing processing time in the future simulations.

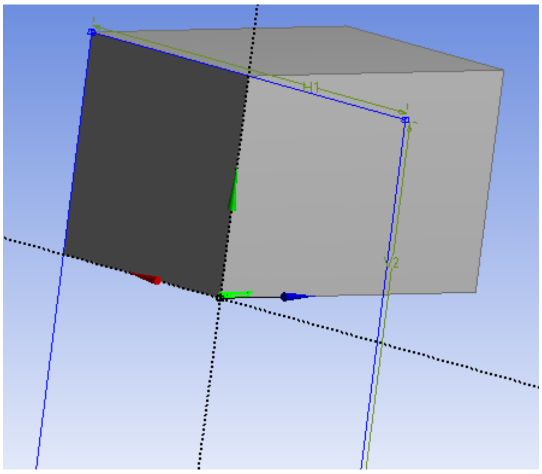


Fig. 18: Useful quarter

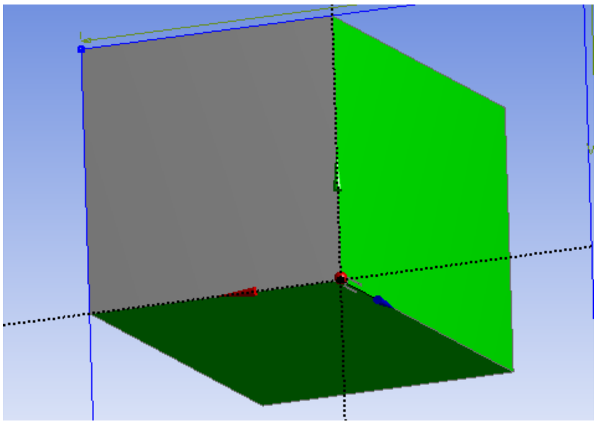


Fig. 19: Name selection: Symmetry

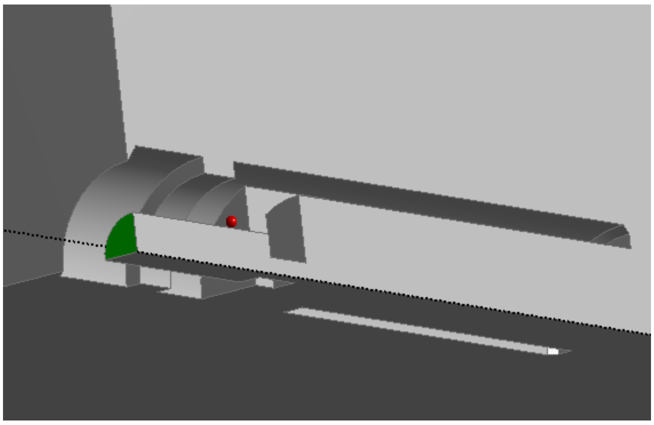


Fig. 20: Name selection: Inlet

Finally, to get easy next steps we defined in geometry some Name Selections as Symmetry (green coloured face in Fig. 19), Inlet (green coloured face in Fig. 20) and Opening (all faces but Inlet and Symmetry's faces).

When geometry was finished and saved, next step was creating the mesh.

2.3 Meshing

The partial differential equations that govern fluid flow and heat transfer are not usually amenable to analytical solutions, except for very simple cases. Therefore, in order to analyse fluid flows, flow domains are split into smaller subdomains (made up of geometric primitives like hexahedra and tetrahedral in 3D and quadrilaterals and triangles in 2D). The governing equations are then discretized and solved inside each of these subdomains. Typically, one of three methods is used to solve the approximate version of the system of equations: finite volumes, finite elements, or finite differences. Care must be taken to ensure proper continuity of solution across the common interfaces between two subdomains, so that the approximate solutions inside various portions can be put together to give a complete picture of fluid flow in the entire domain. The subdomains are often called elements or cells, and the collection of all elements or cells are called a mesh or grid *[06]* [6].

The process of obtaining an appropriate mesh (or grid) is termed mesh generation (or grid generation), and has long been considered a bottleneck in the analysis process due to the lack of a fully automatic mesh generation procedure. Specialized software programs have been developed for the purpose of mesh and grid generation, and access to a good software package and expertise

ERROR! REFERENCE SOURCE NOT FOUND.

in using this software are vital to the success of a modelling effort *[07]*[7]. As we had not access to this special programs, out mesh was done with ANSYS.

ANSYS Workbench Meshing creates a mesh by default, but it does not use to be enough for make simulations properly; as in our case; therefore we needed to make some adjustments.

First step was to generate the default mesh (Fig. 21), but this was too coarse to simulate a fluid flow of air with right results. It was clear that we needed to make some changes in mesh's options.

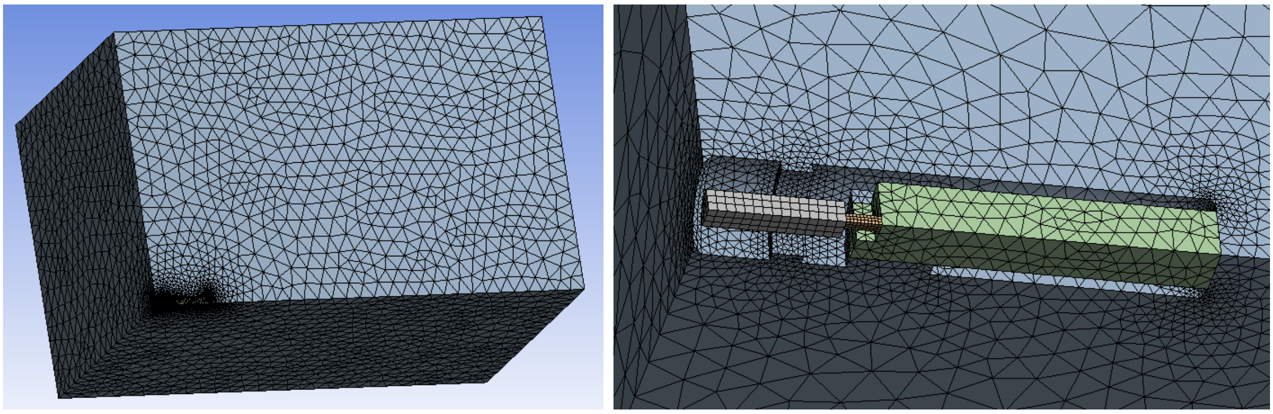


Fig. 21: Default mesh

We began changing general meshing options, as switch the relevance centre (mesh/sizing) to medium and the value of relevance from 0 to 100. Furthermore, we used advanced size function with proximity and curvature.

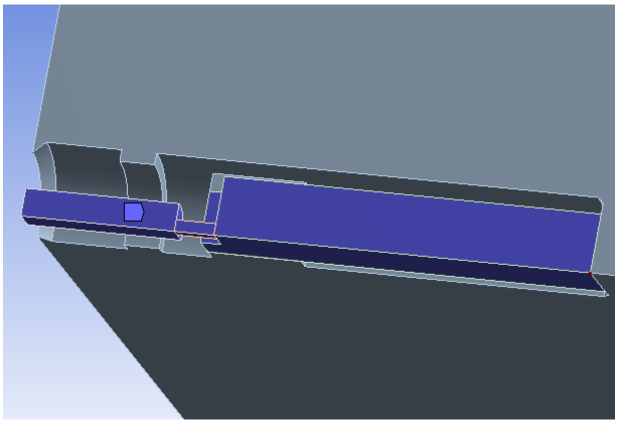


Fig. 22: Body sizing

Next step was defined the kind of mesh in our nozzle (Patch Conforming Method). We chose tetrahedrons, due to the fact that we couldn't obtain good results with another more regular mesh.

In addition, we defined the sizing of some parts to make our mesh finer in important places. In the three bodies of the nozzle was inserted a "body sizing" (Fig. 22); then, a "vertex sizing" in the outlet of the nozzle with a bigger radios than the outlet (8mm was the measure chosen in this case), and finally we inserted an "edge sizing" in the edge next to the outlet (Fig. 24). In all these adjustments were defined their element size as a parameter to be able to change it directly from the workbench.

Nevertheless, there was a zone where we need even a finer mesh due to the fact that there are supersonic velocities there, and that usually complicates the solving of simulations, so it is recommendable improve the mesh there. In consequence, we applied a refinement in that zone as it is shown in Fig. 23.

ERROR! REFERENCE SOURCE NOT FOUND.

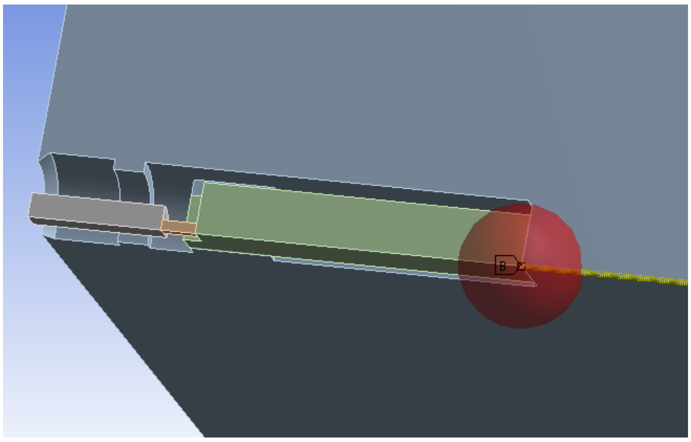


Fig. 24: Vertex and edge sizing

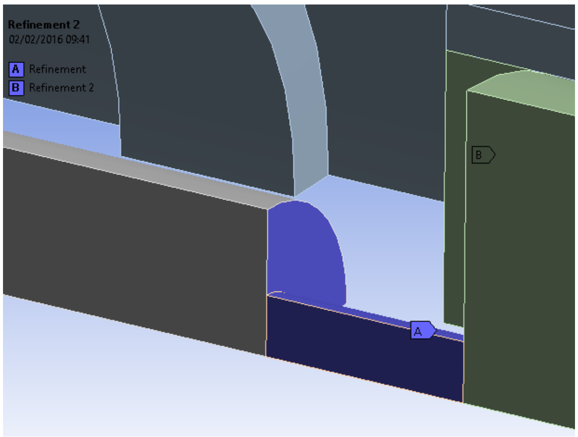


Fig. 23: Refinement

When we had our mesh defined, we went to parameters and set different elements sizes to make several simulations and can choose the right element size (Table 2). It is better to star with a course size and turning it into a finer one until we see that the results almost do not change with a finer mesh.

Then all design points were updated, and with the results we could create a char (Chart 1) where be able to see which size are fine enough for this simulation.

Some theories say us that if we increase the number of elements in a 30% and the results change less than a 5%, then our elements size are enough.

In Chart 1 is clear how below 0.22 mm of element size results almost didn't change, so we could use that size instead another finer, saving a significantly time of processing. Accordingly, from this moment we used always this mesh with 0.22 mm of element size in next simulations.

Table 2: Mesh parameter set

File View Tools Units Extensions Help
Project / Parameter Set
Update All Design Points

Outline of All Parameters

	A	B
1	ID	Parameter Name
2	Input Parameters	
3	Choose mesh (A1)	
4	P19	Edge Sizing Element Siz
5	P20	Body Sizing Element Siz
6	P21	Vertex Sizing 2 Element
7	P22	Face Sizing Element Siz
8	P23	Vertex Sizing Element S
*	New input parameter	New name
10	Output Parameters	
11	Choose mesh (A1)	
12	P24	Mesh Elements
13	P25	VelInlet
14	P26	VelOulet

Table of Design Points

	C	D	E	F	G	H	I	J	K	L
1	P19 - Edge Sizing Element Size	P20 - Body Sizing Element Size	P21 - Vertex Sizing 2 Element Size	P22 - Face Sizing Element Size	P23 - Vertex Sizing Element Size	P24 - Mesh Elements	P25 - VelInlet	P26 - VelOulet	P27 - VelClose	P2...
2	mm	mm	mm	mm	mm		m s^-1	m s^-1	m s^-1	N
3	1	1	1	1	1	4.461E+05	174.66	162.26	67.412	0.049834
4	0.5	0.5	0.5	0.5	0.5	7.1429E+05	176.74	171.92	69.213	0.054916
5	0.3	0.3	0.3	0.3	0.3	1.2415E+06	175.51	168.57	14.731	0.01564
6	0.25	0.25	0.25	0.25	0.25	1.5824E+06	175.66	170.31	67.132	0.05563
7	0.22	0.22	0.22	0.22	0.22	1.9055E+06	175.51	172.98	67.704	0.056956
8	0.2	0.2	0.2	0.2	0.2	2.2133E+06	174.67	172.87	66.871	0.055692
*										

Properties of Schematic: Parameter Set

	A	B
1	Property	Value
2	Design Point Update Process	
3	Update Option	Run in Foreground
4	Design Point Update Order	Update from Current
5	License Checkout	On-demand
6	Partial Update	None
7	Retain Partial Update	None
8	Retained Design Point	Update parameters

ERROR! REFERENCE SOURCE NOT FOUND.

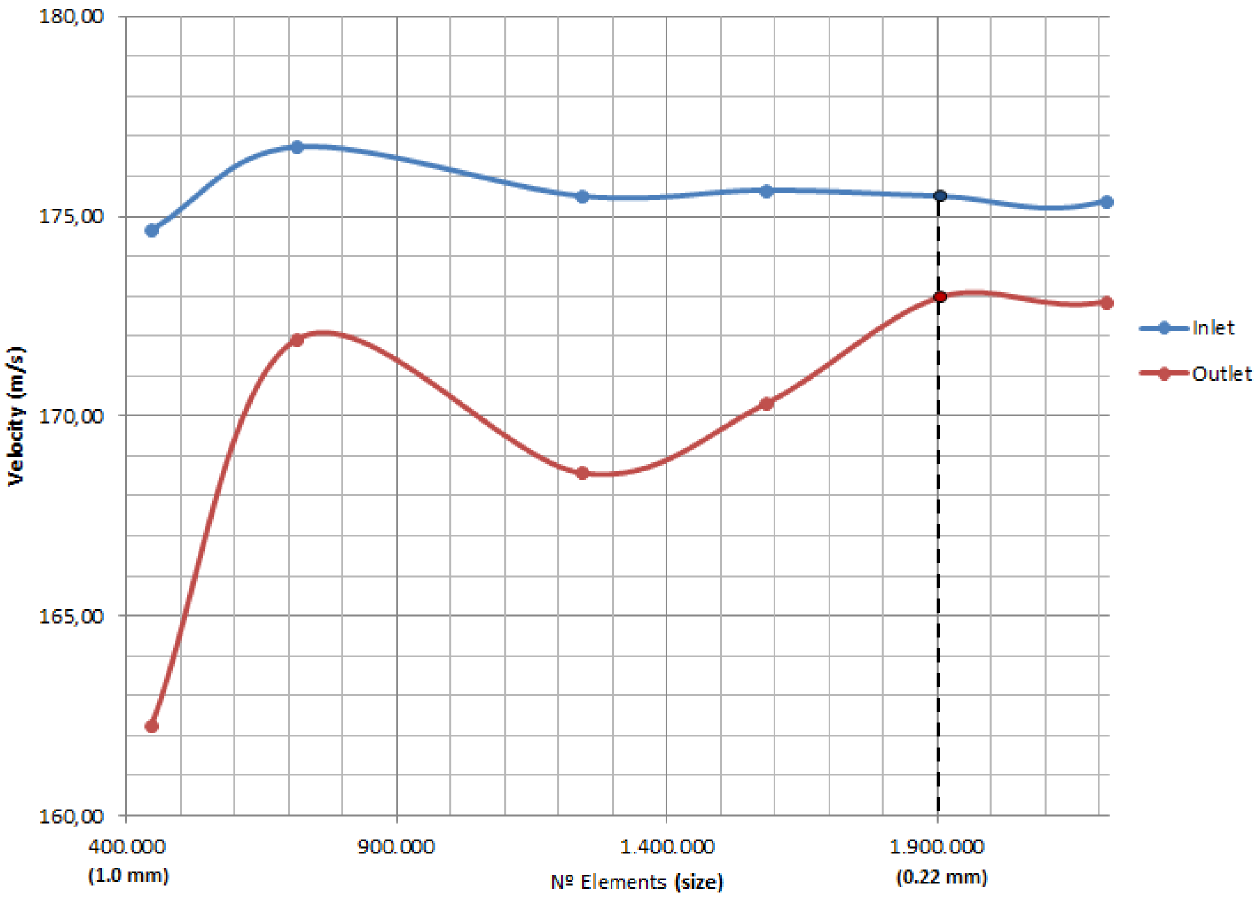


Chart 1: Velocity-Nº of elements

On the next Fig. 25 and Fig. 26 it is show the final mesh with the suitable element size.

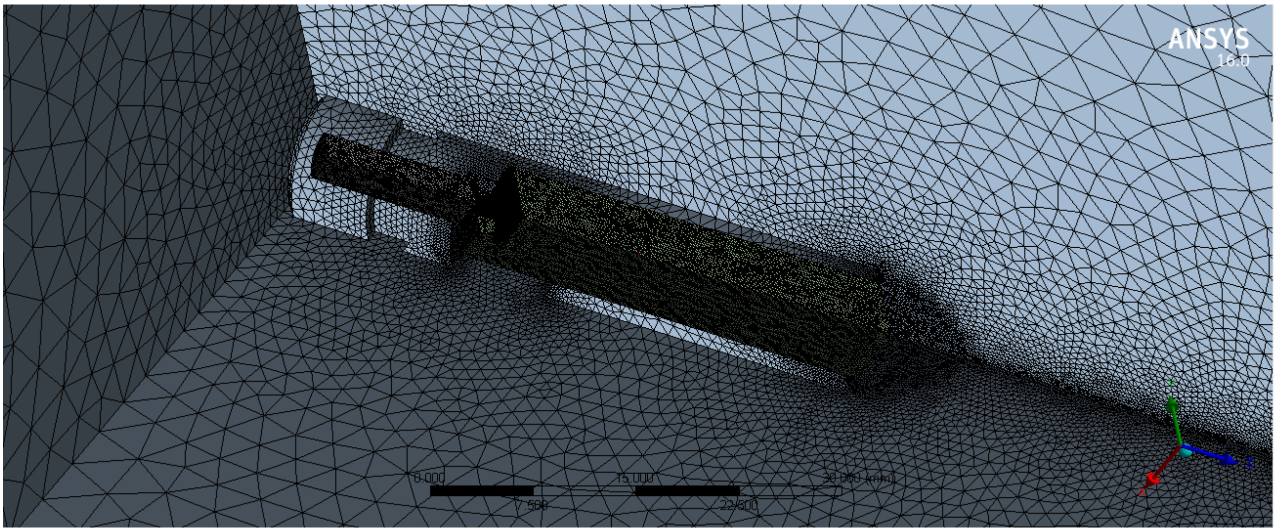


Fig. 25: Final mesh nozzle

ERROR! REFERENCE SOURCE NOT FOUND.

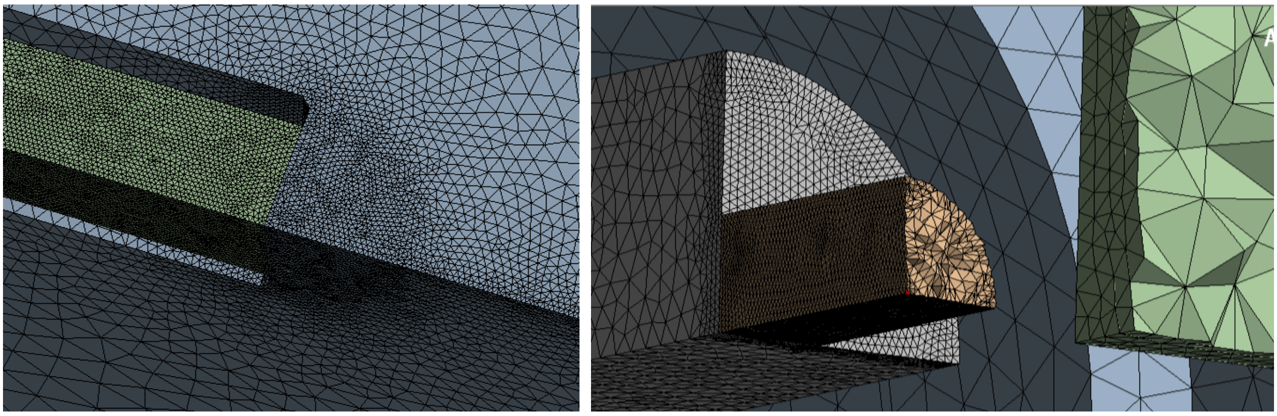


Fig. 26: Final mesh detailed

The process to obtain a good mesh is long and complicated and it is necessary a long experience to know how get it quickly.

This was the right way to obtain a good mesh for our simulation, but to get it we spent many time trying different meshes. Sometimes it produces errors during the simulation and it is necessary change it for another one, and another times results are not rational and it is clear that it is not the right way.

2.4 Set-Up [CFX-Pre]

After meshing, we loaded the place to physics-definitions called "Setup" with the aim of define all the necessary to make a correct analysis. Automatically, the setup after importing the mesh creates the domain and the respective interface between them.

In this first simulation, we defined the basics setups ("main setups" since now). In following simulations, we changed some parameters but always starting from these ones, so we only will mention the differences between those ones and the setups defined here or which were stablished by default.

We started with Steady State analysis type (selected by default); it means that the magnitudes are constants with the time and we are going to receive results of the moment when all parameters are constants ("infinite" time). Then we created and set some domains:

Default Domain (Fig. 27)**:**

Here is where we defined our fluid and its parameters:

- Material: Air at 25 ºC and Continuous Fluid
- Reference pressure = 1 [atm]
- Non Buoyant and Stationary
- Model Isothermal (25 ºC)
- Turbulence: Shear Stress Transport, with Automatic wall function

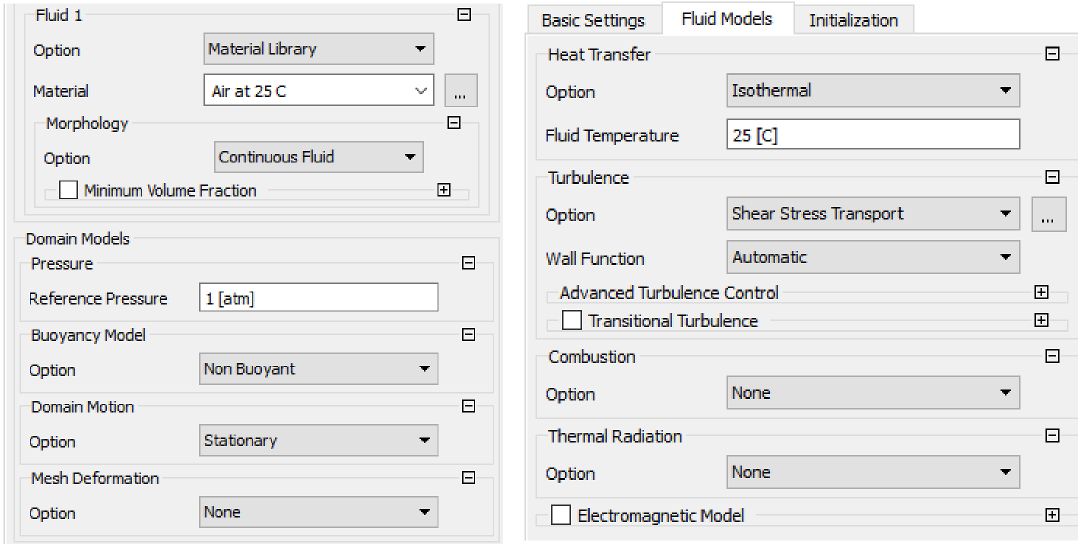


Fig. 27: Default Domain setups

Inlet (Fig. 28)**:**

To add this domain we had to click in “Boundary”, to write “Inlet” in name and the right location was chosen automatically. Then we changed some parameters:

• Mass and Momentum: Total Pressure (stable)

• Relative Pressure: “Press”

*”Press” is an expression previously created with 10 [atm] by default.

• Flow direction: Normal to Boundary Condition

• Turbulence: Medium

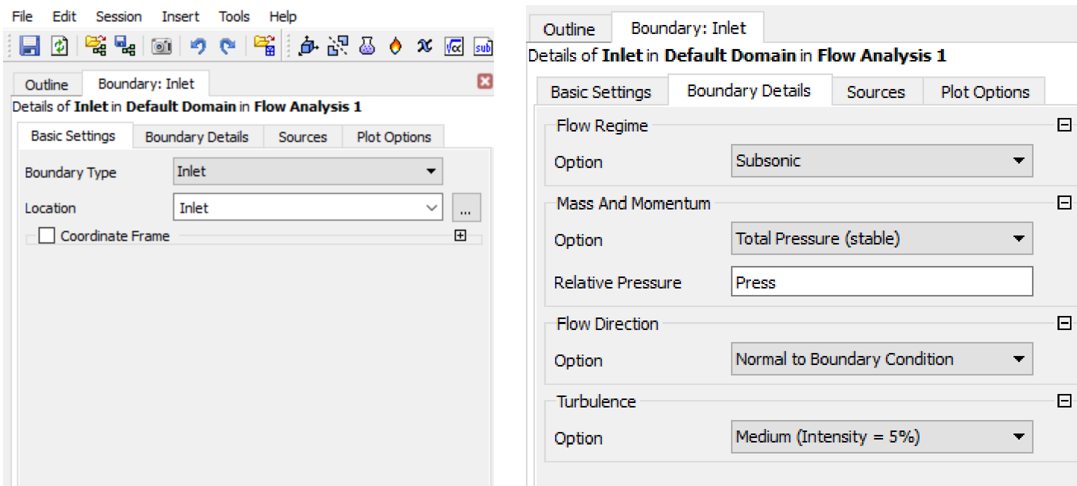


Fig. 28: Inlet setups

Opening (Fig. 29)**:**

Again, we had to click in “Boundary”, wrote “Opening” this time, and the right location was chosen automatically. Then we changed some parameters:

• Mass and Momentum: Opening Pres. and Dirn.

• Relative Pressure: 0 [atm]

• Flow Direction: Normal to Boundary Condition

•Turbulence: Medium

ERROR! REFERENCE SOURCE NOT FOUND.

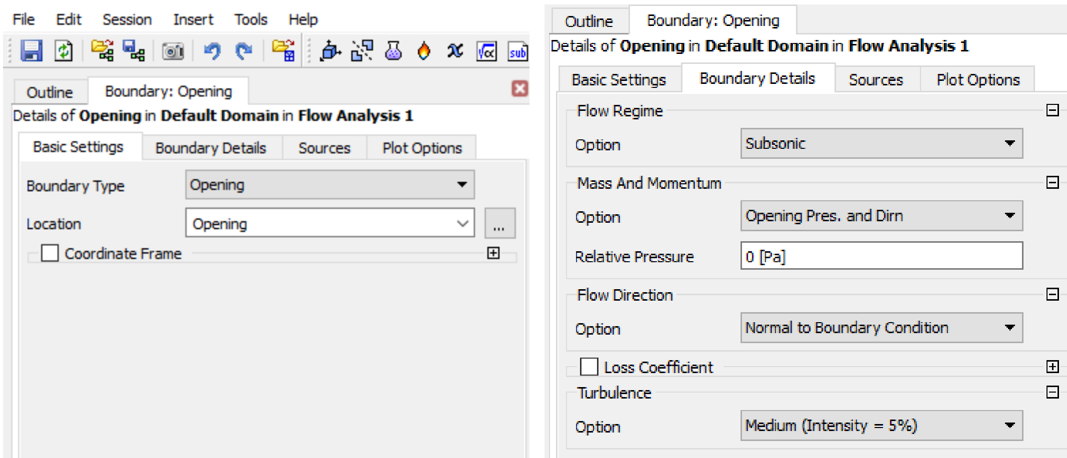


Fig. 29: Opening setups

Symmetry (Fig. 30)**:**

Last boundary was Symmetry; in this step, we only clicked in boundary and wrote Symmetry in name to have all right parameters chosen.

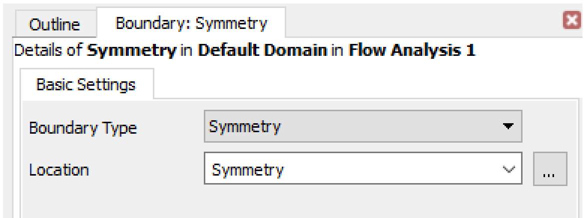


Fig. 30: Symmetry setups

In addition, we needed to change some parameters in Solver Control, where we could modify the way to solve the equations. In this simulation, we only increased the maximum of interactions from 100 to 300, because 100 could be not enough to solve this simulation.

Once we had all setup defined we could jump from Setup to Solution, the next and last step before run the simulation.

2.5 Run settings [CFX-Solver Manager]

Run settings are the last parameters that are be able to change before run the simulation. This setting have a relevant influence in the way of solve our problem, time of processing and even in the results.

In our case, we always used double precision (it gets more accurate results, but slower solver at the same time) and initial conditions as initial values (the solver always starts from the same point: initial values, which ones were defined in setup).

About run mode, we used two different settings depending in which computer we ran the solver, in mine one or in the computer of university. In both we used Platform MPI Local Parallel, but with different number of partitions. It was chosen according to the number of cores that the computer had: two in my laptop and six in the computer of the university (Fig. 31).

ERROR! REFERENCE SOURCE NOT FOUND.

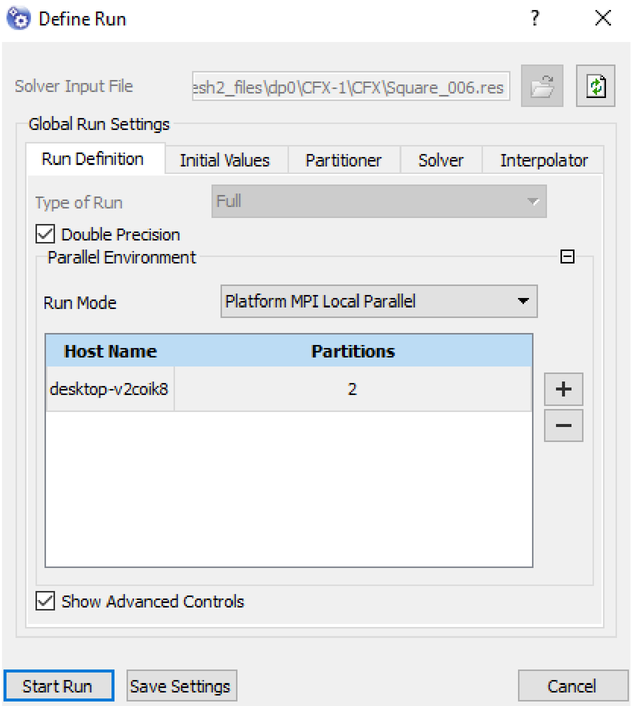


Fig. 31: Solution control

With the option Platform MPI Local Parallel instead "serial" (by default); we could speed up the run time drastically depending on the model we are running.

Computer used to run the solver play a crucial role in the time of processing.

Here we can see the time of processing to a same simulation with the two computers that was able to run them in our case, applying the previous same settings.

My laptop (CPU: 2 x 2.53GHz; RAM: 4 GB) = 7.170s = 119.5 min = **2 hours**

University's computer (6 x 1.6GHz; RAM: 128 GB) = 860s = **14.8 min**

The previous cases were with a simple simulation that was solve quickly, but in another cases times became longer; also we needed to make many simulations, furthermore, a high percentage of them give us back wrong results or errors and it was necessary doing it many times changing little parameters until research the right one. Therefore, it is essential to have a powerful computer at the time to make simulations.

In addition, it is more efficient spend more time choosing the properly mesh and setups than try to do it faster and make mistakes, forcing you to repeat the simulation again and again, that means a very long time wasted.

ERROR! REFERENCE SOURCE NOT FOUND.

2.6 Turbulence Model

Turbulence modelling is a key issue in most CFD simulations.

ANSYS offers a number of advanced turbulence models in the form of algebraic, one-equation, two-equation and Reynolds stress models. These models are integrated into state-of-the-art CFD solvers. The most widely used turbulence models are Reynolds-averaged Navier–Stokes (RANS) models that are based on time averaging of the equations. Time averaging filters out all turbulent scales from the simulation, and the effect of turbulence on the mean flow is then re-introduced through appropriate modeling assumptions.

The standard k-ε model is used in the prediction of most turbulent flow calculations because of its robustness, economy, and reasonable accuracy for a wide range of flows. However, the model performs poorly when faced with non-equilibrium boundary layers. It tends to predict the onset of separation too late and to under-predict the amount of separation. Separation influences the overall performance of many devices, such as diffusers, turbine blades and aerodynamic bodies. It also has a strong influence on other effects, such as wall heat transfer and multi-phase phenomena *[08]*[8].

Predicting reduced separation usually results in an optimistic prediction of machine performance. To solve this problem, new models have been developed. One of the most effective is the Shear Stress Transport (SST) model. For flow separation, the shear–stress transport (SST) model has become accepted as the two-equation model industry standard. The SST model unifies the advantages of the most widely employed two-equation (k-ω and k-ε) models and is the most reliable model for fluids with flow separation *[09]*[9]. This model works by solving a turbulence/frequency-based model (k–ω) at the wall and k-ε in the bulk flow. A blending function ensures a smooth transition between the two models.

Although according our simulation the most recommendable turbulence model was Shear Stress Transport (SST) model, we decided check if there were significant variance among different models. This would save us much time if a simpler model as k-epsilon gave us similar results.

Then, we chose some different turbulence models and ran their simulations to obtain the measures in each one obtaining the following results (Table 3 and Chart 2):

Table 3: Turbulence models data

	SST	K-Epsilon	K-Omega	BSL
Vel Inlet (m/s)	175,51	176,71	176,30	175,34
Vel Outlet (m/s)	172,98	163,87	169,77	153,77
Force (N)	0,056956	0,091165	0,062518	0,028160

ERROR! REFERENCE SOURCE NOT FOUND.

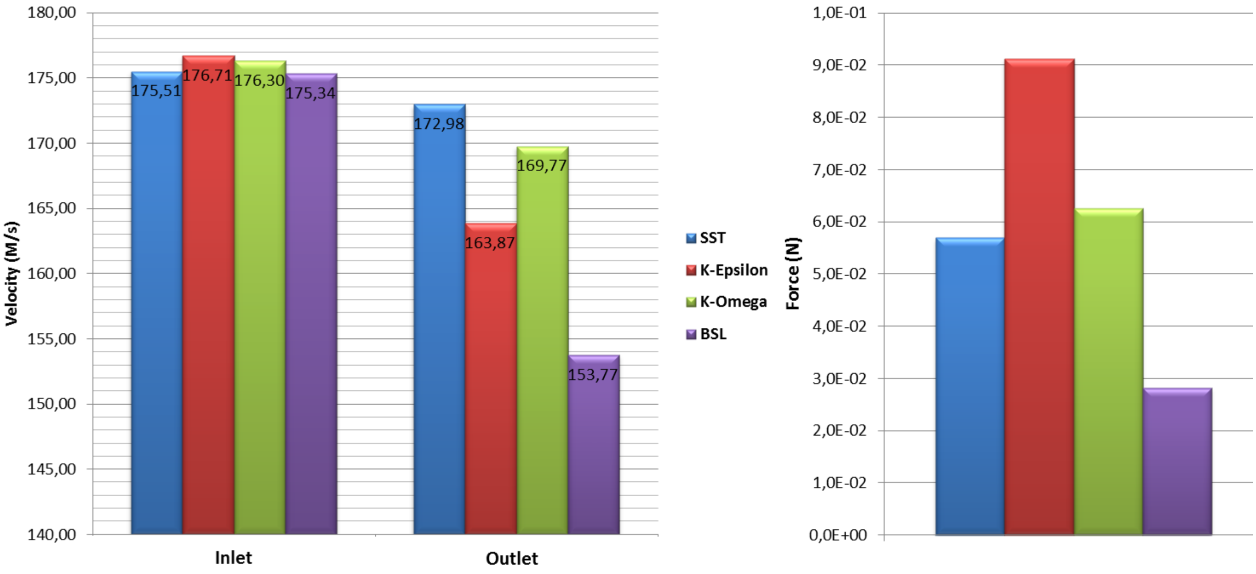


Chart 2: Velocity and Force in different turbulence models

In our case, differences between these models were big enough to have to choose the most accuracy (although more time expensive too) turbulence model; which in our case was Shear Stress Transport (SST) due it is the widely model used for air simulations.

ERROR! REFERENCE SOURCE NOT FOUND.

3 VALIDATION

Simulations usually are a good way to know the behaviour of a fluid, a mechanic piece or each another thing that we could simulate. However, not every computer simulations are right and give us suitable results. For this reason, before start with the researches, it is strongly recommendable make an experiment with the most similar parameters between the experiment and computer simulation, and then compare both results to know if our simulation model are close to the real case. According with the results, we can assert that our simulation is valid for this problem or not. In the case, that computer simulation was not right; we should go on working in it until reach the right way.

In our case, we did an experiment to validate our simulation too, in that we wanted to measure velocity of our airflow in different distances from the nozzle and compare them with results obtained in computer simulations. To do it, we had to get some additional stuff, as a device to measure the airflow velocity (which includes an electronic device where data are processed and showed by a scream), a clamping jaw to hold the air compressed gun and a video camera to record the experiment and be able to analyse it carefully.

The experiment assembly was as is shown in Fig. 32: On a table, we placed the clamping jaw holding the air-compressed gun, which was connected for one side to the air compressor and in the other side we screwed the nozzle that we wanted to check (nozzle 2 in this first case). In addition, opposite to the air-compressed gun, we placed the measurer of velocity, which we connected with its data processor. It was very important have the outlet of the nozzle and the inlet of the measurer in the same plane and direction, the way that fluid flow go straight from the nozzle to the measurer.

Fig. 32: Experiment

The way that we followed to make the experiment was to run the air compressor until get the maximum pressure in it, and when it was full, open the valve of the air-compressed gun until the pressure in it was below 3 bar, all this while we was recording with a video camera.

We did this process several times placing the velocity measurer in different distances from the nozzle (0.2; 0.4; 0.6; 0.8 and 1 m).

After do all the experiments, we could analyse them carefully in the videos and take the measures of velocity in the precise moment when the pressure in nozzle inlet was the pressure that we wanted. Taking those measures, we obtained the following table (Table 4).

Table 4: Real experiment results, nozzle 2

	Press (bar)	Distance (cm)	Velocity (m/s)
Nozzle 2 (2 holes)	3,5	60	13,97
	3,5	80	10,67
	3,5	100	8,25

In this case velocity only were measured from 60 cm of distance because our velocity measurer only could take measures below 20 m/s, and in distances below 60 cm it was above it.

For another hand, we had to create a new computer simulation the most similar possible to this experiment. Starting from the previously prepared simulation, we made some changes in geometry, mesh and setups to get a solution.

About geometry, we extended it to can take faraway measures due last one was shorter because we only needed take it near to the nozzle. As it was necessary take measures 1 m far from the nozzle, we change the value of the extrusion from 0.3 to 1.1 m length, getting the geometry shown in Fig. 33.

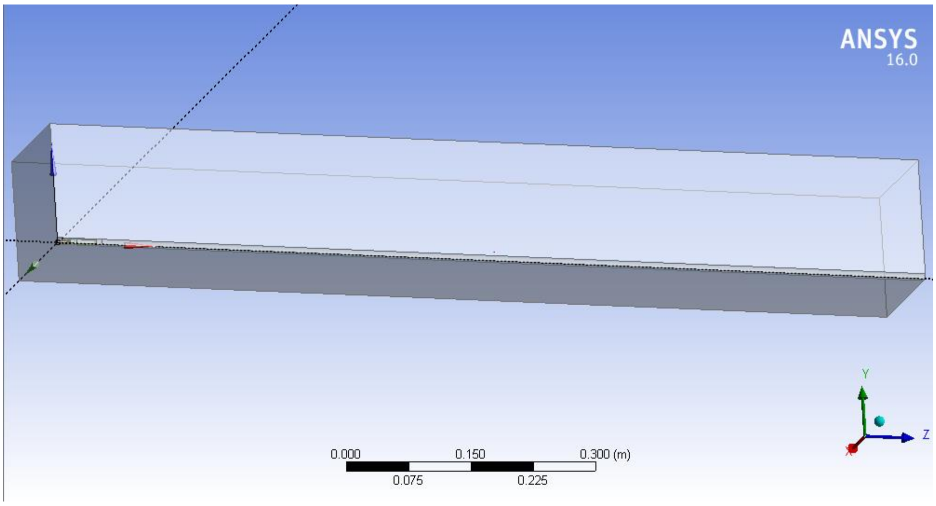


Fig. 33: experiment geometry

After that, we updated the mesh and opened setups. In setups, after try with different configurations we get obtain suitable results only with transient analysis (due the length of the geometry with steady state fluid flow disappeared). So we chose transient analysis, stablish the total time in 2s (it was enough to converge the solution) and time step in 0.1 s. In addition, we turned down the pressure in inlet to 3.5bar to simulate the same conditions than in the experiment. To this simulation only was necessary to save the velocity results and each five time steps (actually, we only needed the final one, but it was good have some previous steps to see that the solution converged). Finally, we ran the solver and got the results (Table 5).

ERROR! REFERENCE SOURCE NOT FOUND.

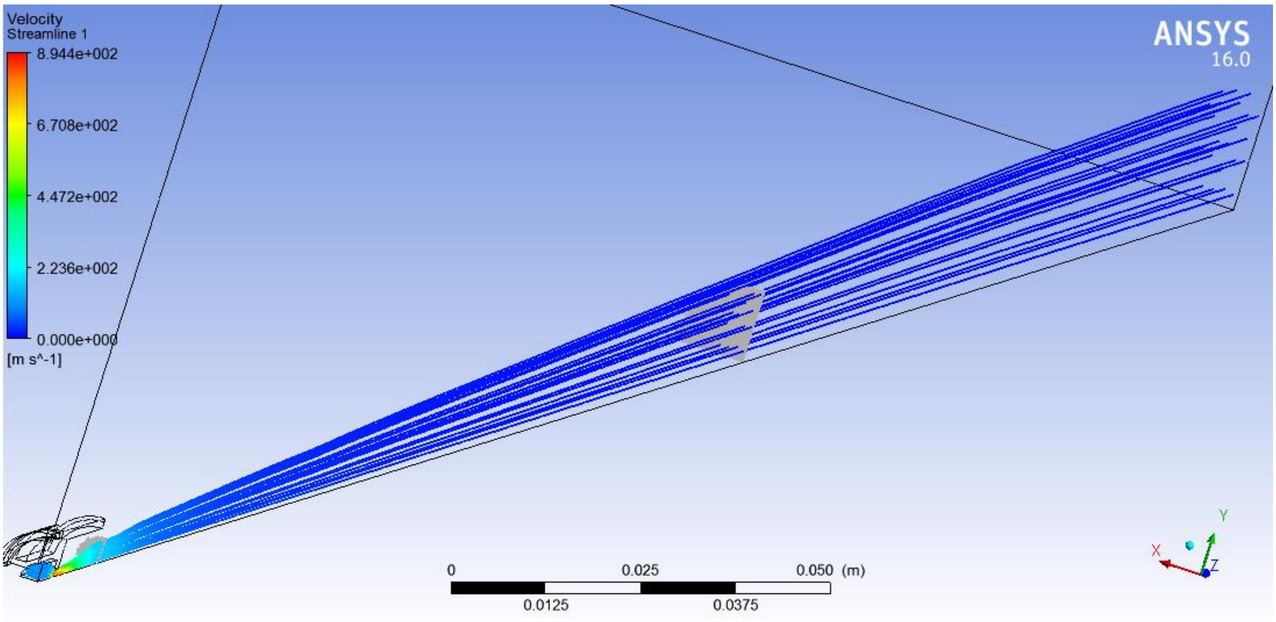


Fig. 34: Streamlines in experiment simulation

Once we obtained results, the best way to compare it with the ones measured in the real experiment was collect the velocities in different distances and creating a table and a chart where compare the velocities of real experiment and simulated experiment (Table 5 and Chart 3).

Table 5: Real and simulated results

			Velocity (m/s)	
	Press (bar)	Distance (cm)	Real	Simulation
Nozzle 2 (2 holes)	3,5	60	13,97	7,45
	3,5	80	10,67	5,02
	3,5	100	8,25	4,07

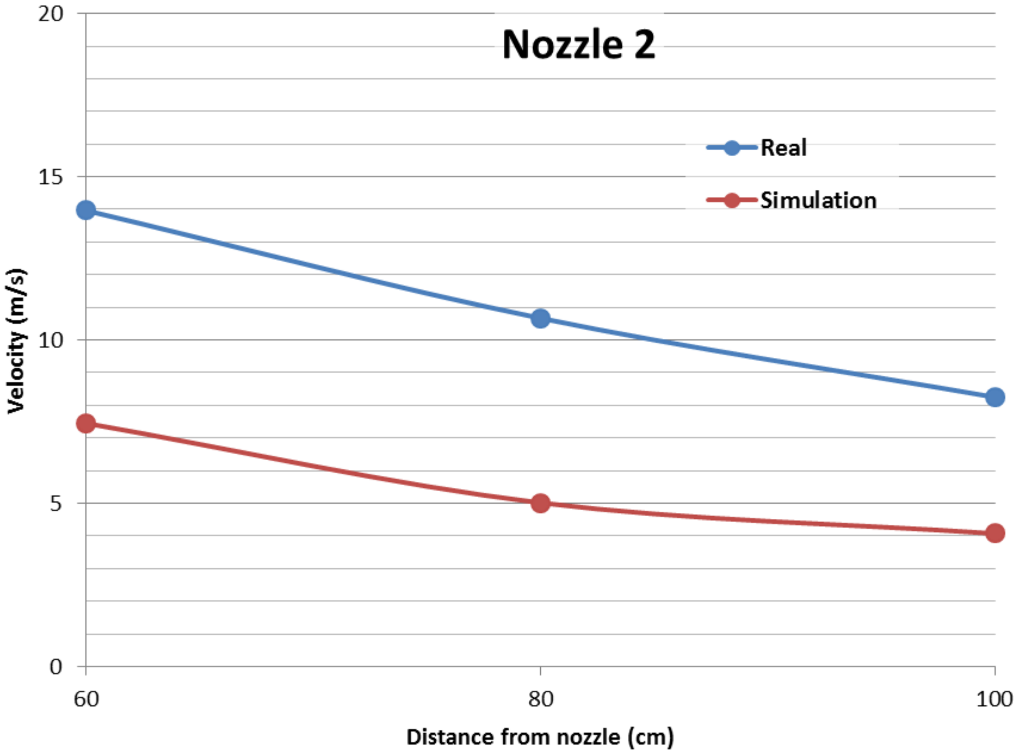


Chart 3: Comparison experiment - simulation results

As we can see in Chart 3, results were quite similar with low differences in figures and same behaviour about changes on distance from nozzle; therefore, we can conclude that the simulation was valid and we can go on with the researches in computer simulations.

ERROR! REFERENCE SOURCE NOT FOUND.

4 RESEARCHES

When we want to know more about any device or we want to improve it, we need to do some deeply research before it: how it is, how it works, how its parameters influence in its work…

Another important question is "why"; why it works like that or why we have problems with that device. To discover what trigger that right or wrong behaviour we must analyse different parts and parameter of our device separately.

In this case, we wanted to know how to work with this device and how maybe we could improve it. Therefore, we had to solve some questions before know it. For example, how the flow works inside the nozzle, how is the influence of the pressure or distance from the nozzle in the fluid flow, or which time step are the best to work with.

4.1 Fluid flow through the nozzle

Before researching the influence of different parameters and ways to work with our nozzle, we analysed what really happen inside our nozzle and how fluid cross through it. This helped us to understand later analysis and results, as well as why some problems appeared and how could be able to solve them.

4.1.1 Preparing simulation

In this simulation, we started from the main setups without any shift for the time being. Therefore, we only defined the solve parameters and run the solver manager until obtain the solution.

Once we got it, we opened CDF-Post to can take some measures. However, before take measures in one place, we have to define the point, surface or volume where doing it.

In this case, we wanted to measure velocity in the outlet; so first, we created a new circular plane in the outlet of our nozzle with the same diameter than our nozzle (5 mm). After that, we could use the function calculator to calculate some data. In this case, we calculated average velocity in inlet, average velocity in outlet, area of inlet face and area of outlet face; coping in all these cases the equivalent expression and creating a new "expression" with each one. Then, using these new expressions, we created another ones to measure the amount of fluid flow crossing inlet and outlet surfaces: **Qinlet** and **Qoulet** (velocity in respective places multiplied by its area), **Qhole** (*Qoutlet* minus *Qinlet*), and finally **Qrel** (ratio between fluid flow that we provide in inlet and fluid flow that leave our nozzle, *Qoutlet* / *Qinlet*).

The last but one step was to mark all these new expressions and inlet pressure parameter (**Press**) as Output Workbench Parameters.

Finally, we created several design parameters with different pressures in inlet and updated all designed points, obtaining the following results:

4.1.2 Results

Table 6: Nozzle parameter set

C	D	E	F	G	H	I
P39 - Press	P29 - VelInlet	P30 - VelOulet	P40 - QInlet	P41 - Qhole	P42 - QOulet	P43 - Qrel
atm	m s^-1	m s^-1	litre s^-1	litre s^-1	litre s^-1	
12	192.27	189.63	0.86858	2.8841	3.7526	4.3204
10	175.51	173.06	0.79286	2.632	3.4249	4.3197
9	166.5	164.16	0.75215	2.4965	3.2487	4.3192
8	156.97	154.75	0.70911	2.3532	3.0624	4.3186
7	146.85	144.72	0.66341	2.2006	2.864	4.3171
6	135.96	133.95	0.6142	2.0367	2.6509	4.316

Table 6 gave us some relevant information. One was that velocities and flow rate were directly related with the pressure provided in inlet (we studied it deeply in following simulations). Another one was seeing that fluid flow rate in outlet was more than four times higher than fluid flow rate in inlet. That is an important data, because it tell us how efficient is this nozzle saving air. By contrast, it was noticed too that this parameter did not depend of inlet pressure.

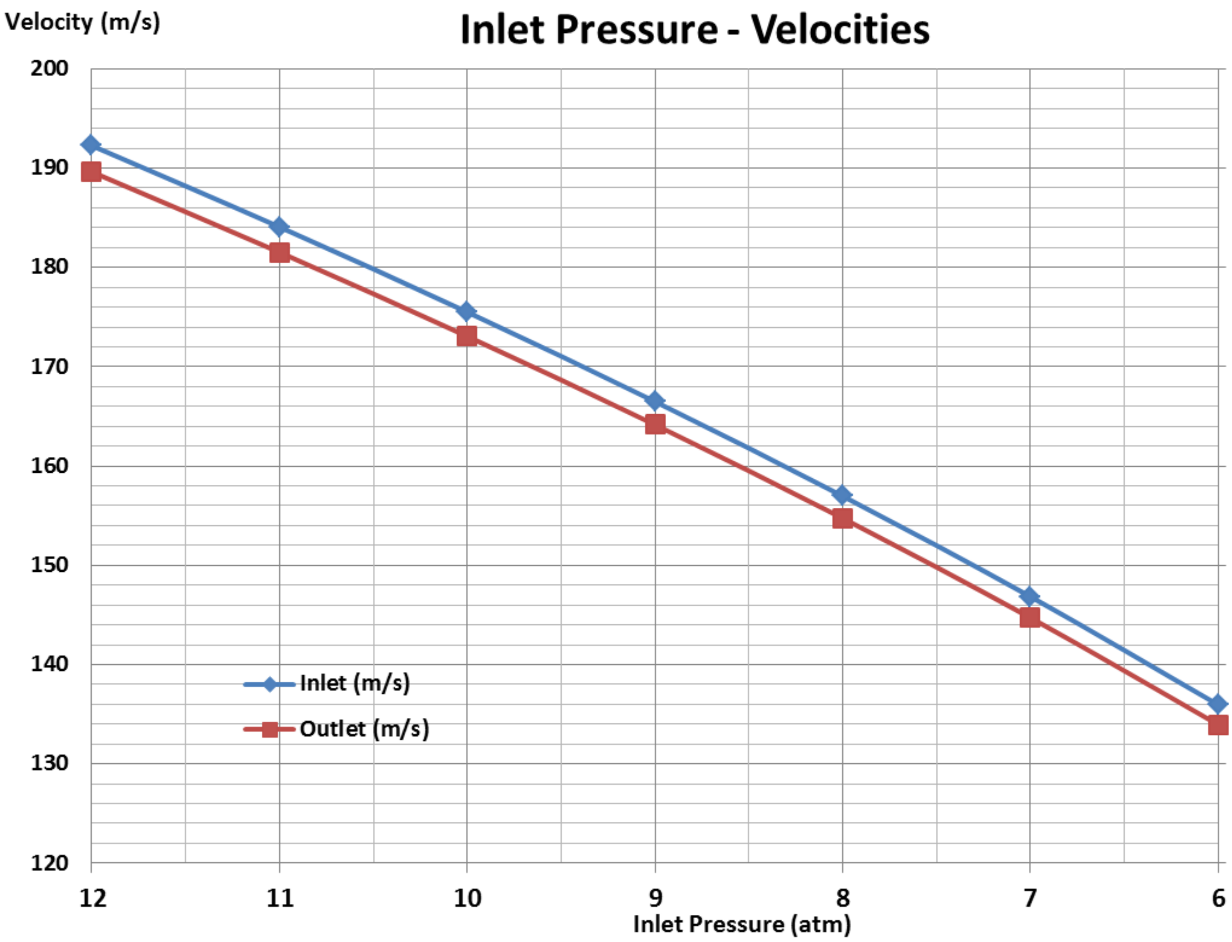


Chart 4: Velocity – Inlet pressure

ERROR! REFERENCE SOURCE NOT FOUND.

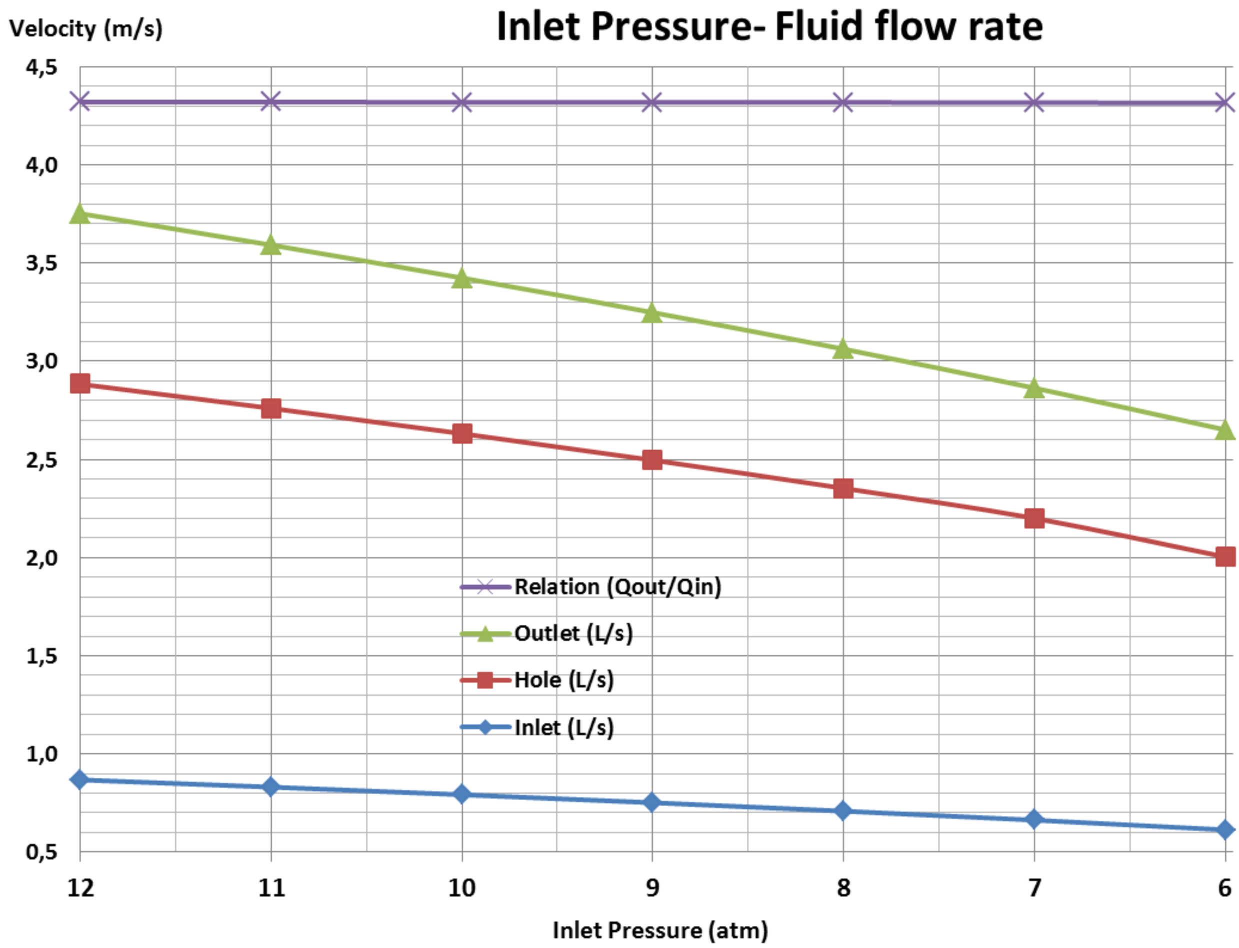


Chart 5: Flow rates – Inlet pressure

As we could see in Chart 4 and Chart 5 the fall down of velocities and fluid flow rates were directly influenced by the decrease of the pressure in inlet.

Therefore, we can assert that there is not shift in the behaviour of our flow between 12 and 6 atmospheres, so we can work in those pressures without problem. If we increase or decrease pressure, we obtain higher or lower velocity, flow rate and forces, but the behaviour and relations between then carry on being the same.

After analysed these figures, we applied several useful locations and plots where was easy seeing how the fluid cross through the nozzle and what effects produced.

For example, in Fig. 35 is showing how total pressure is in each part of the nozzle and in outer place. As you can see all the inlet chamber has around 10 atm of pressure, as well as in almost all the hole chamber. Then, when fluid flow leave that hole, pressure begin to fall down until research atmospheric pressure.

ERROR! REFERENCE SOURCE NOT FOUND.

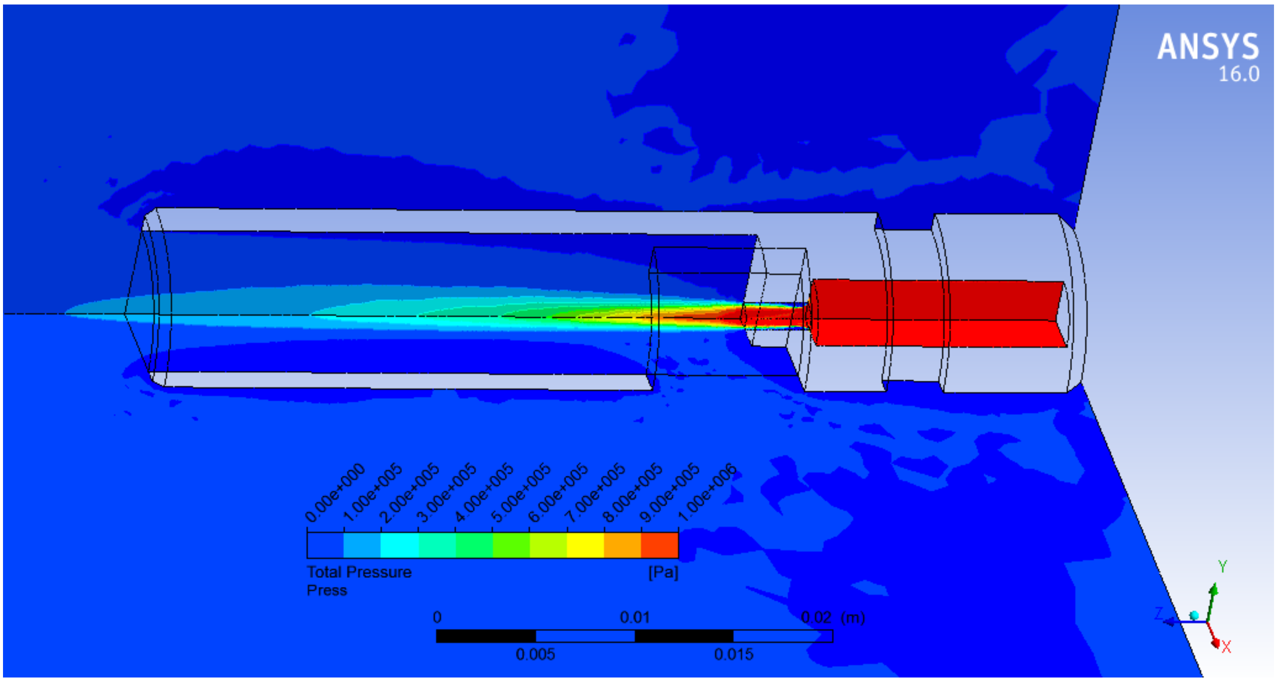


Fig. 35: Total Pressure

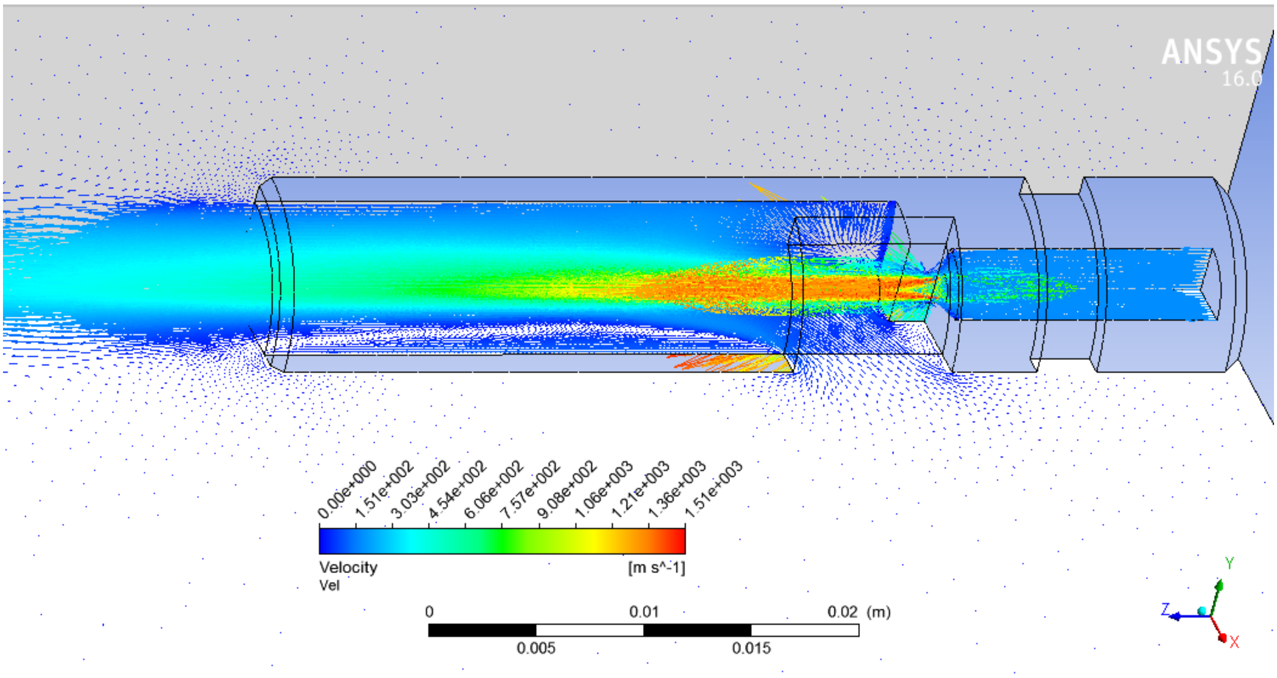
Another fact that is worth bearing in mind is velocity, both amount and direction. To be able to see this, we created the Fig. 36 where colour shows how high velocity was, and the arrows represented its direction.

As this figure shows us, the highest velocity was in the smallest hole between the inlet chamber and the outlet one. There, fluid flow researched velocity above sound's velocity, having a supersonic zone. This was a very important point because when in a simulation we have subsonic and supersonic zones, usually getting a right solution becomes very difficult.

In addition, velocity in the inlet chamber was lower than in the outlet chamber, despite the diameter was smaller in the first one. This happened because through the outlet chamber had to cross air flow from the inlet and, in addition, air flow from the lateral holes. This lateral flow had not a high velocity, however, the holes had enough size to a big amount of fluid was able enter as we could notice in previously analysis.

Moreover, in this figure was very clear how velocity was higher in the centre than close to walls, where airflow velocity decreased according as approaching to them. This is a normal behaviour in fluid streams. In this case, it was even more significantly, because the stream comes from another smaller hole that was placed in the centre.

ERROR! REFERENCE SOURCE NOT FOUND.



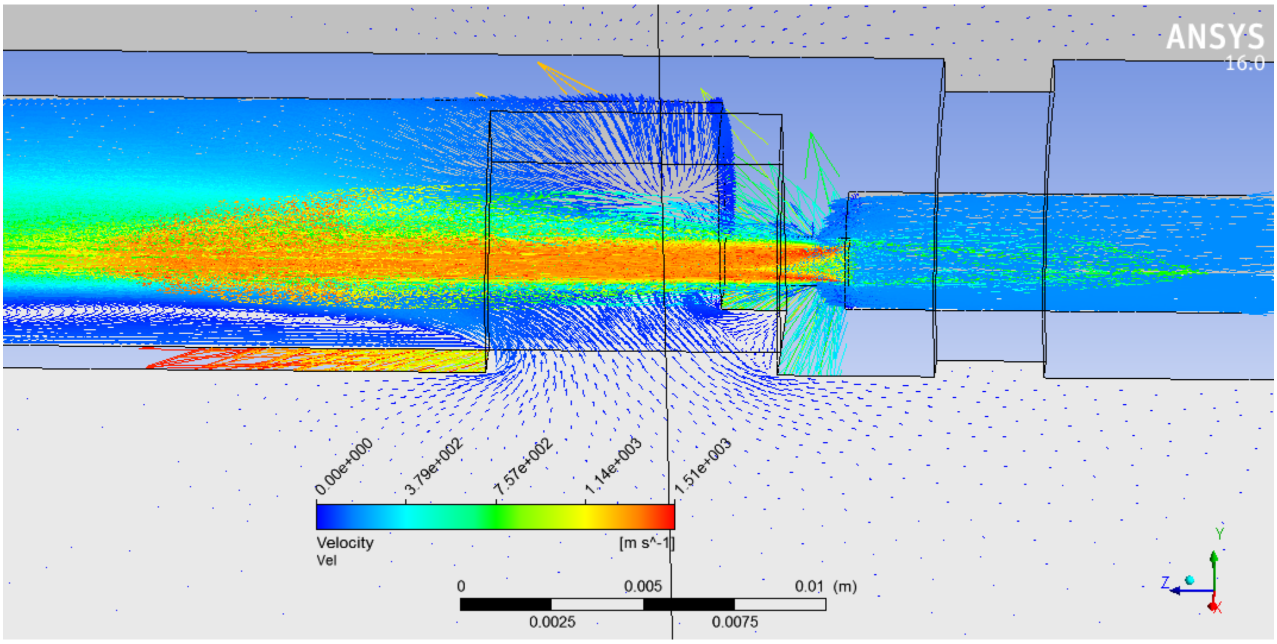


Fig. 36: Velocity

Next one was streamline (Fig. 37). In this kind of study it is able to watch which path fluid particles followed. To make it in a properly way, we created two streamlines, one to flow from the inlet and another one to flow from the lateral hole. Also, how we knew that the amount of air that enter in the nozzle from the lateral hole was 4.3 times bigger than the inlet one, we defined the number of points according which that ratio: 4.3 times higher in the streamline which start from the lateral hole (86 points) than the another one (20 points).

ERROR! REFERENCE SOURCE NOT FOUND.

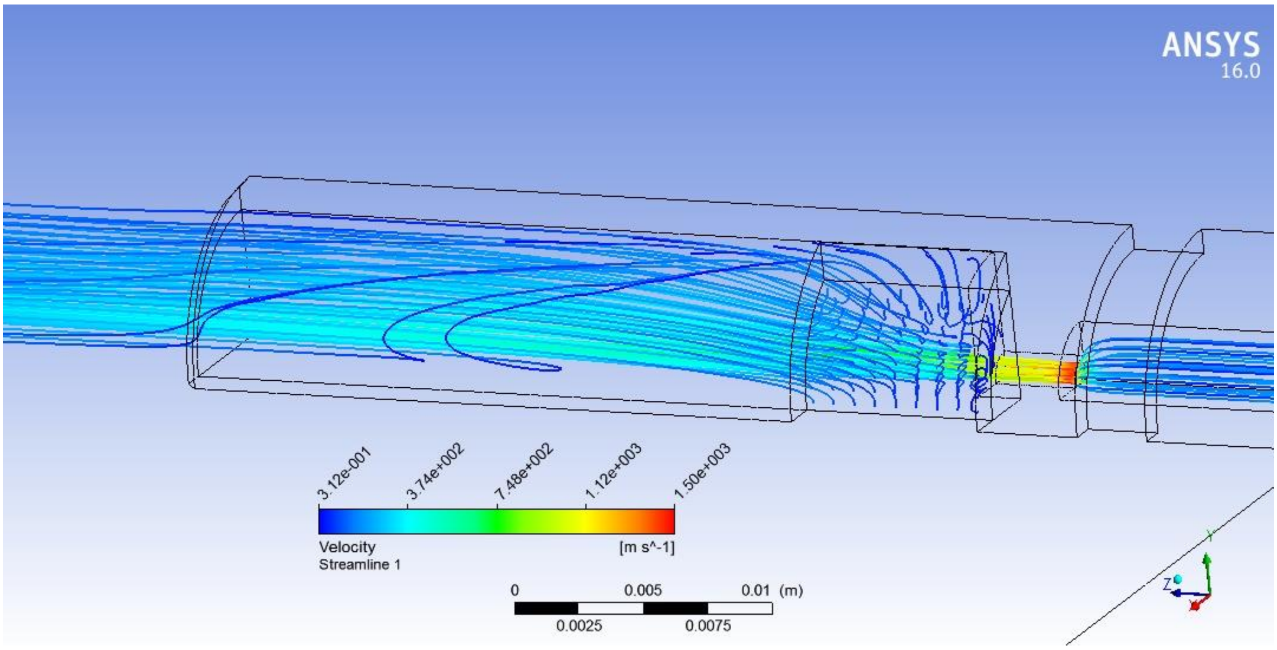


Fig. 37: Velocity streamlines

In addition, it was interested too printing zones with different velocities through where fluid flow entered in our nozzle from the lateral hole (Fig. 38).

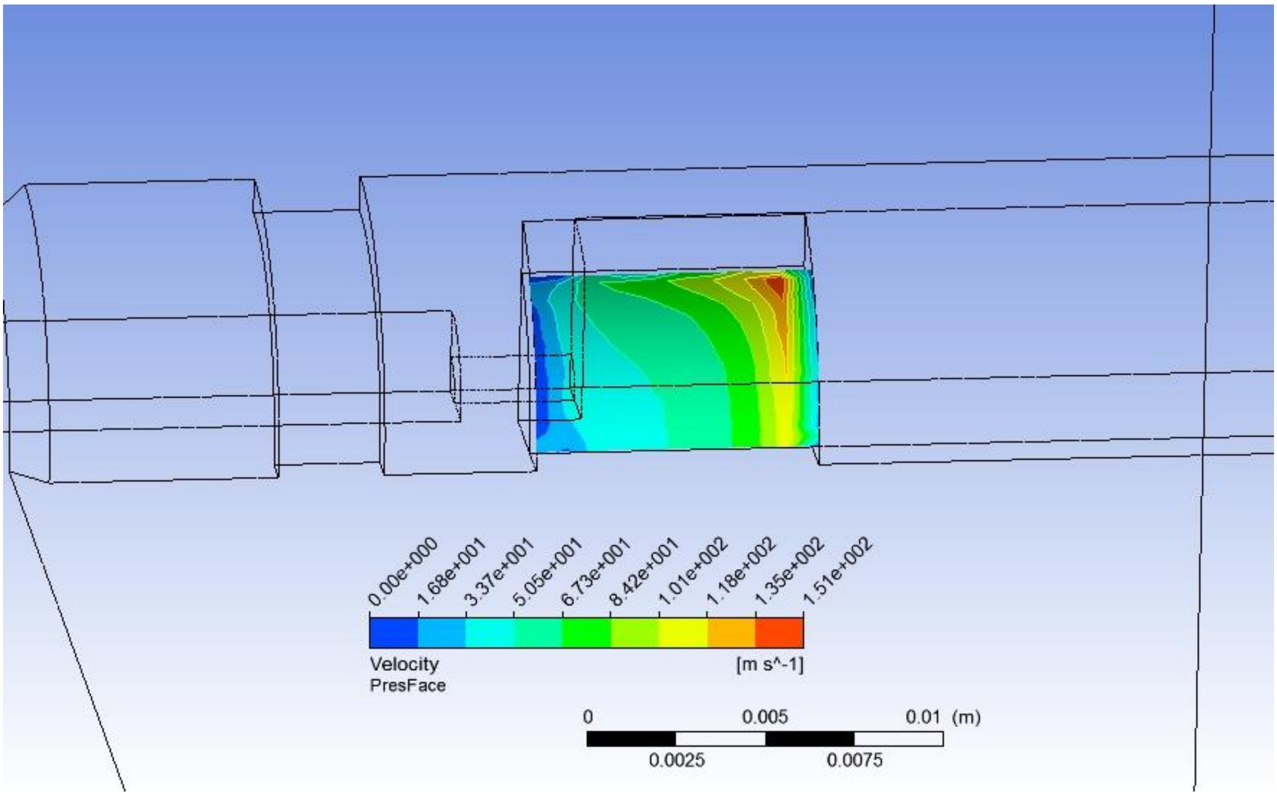


Fig. 38: Velocity in lateral hole

ERROR! REFERENCE SOURCE NOT FOUND.

4.2 Influence of particle’s shape (and size)**

Dust is not something easy to simulate due its particles can have whatever shape.

As it is impossible simulate a real dust particles, we studied how different are forces in three basic particle shapes (square, square inclined and sphere) if we keep all another parameters constant.

4.2.1 Designs and mesh

Starting from the previously design which we did to choose the mesh; we duplicated it two times to have three different CFX analysis in Workbench.

Then, they were modified in ANSYS to have a different shape of particle in each one.

<u>Square:</u>

In Geometry, we created a new plane parallel to the nozzle axis and flow direction 10 cm far from the nozzle. In that plane was drawn a square with equal size than nozzle outlet hole: 5x5 mm (we was working with a quarter). Then, we extruded that sketch another 5 mm to have a regular cube, but using the operation “slice material” instead “add material” (Fig. 40). After that, we got two bodies, the fluid and the cube, so we deleted the last one with a boolean operation to have only fluid volume. Finally we defined the face where the flow runs into as a named selection (“FaceSquare” was named in this case) to could measure easily there some parameters (Fig. 41).

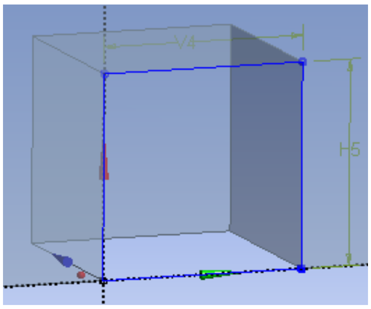


Fig. 40: New plane, square sketch and extrusion

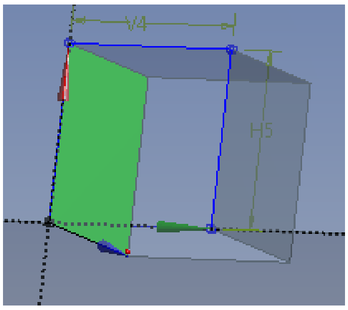


Fig. 41: Named selection

After had designed it, we needed make some adjustments in mesh to improve it around the particle. In this case, we decided to make a sphere of influence (radius= 8 mm) around the particle, also it was done with the same element size that in the nozzle (as is show in Fig. 39).

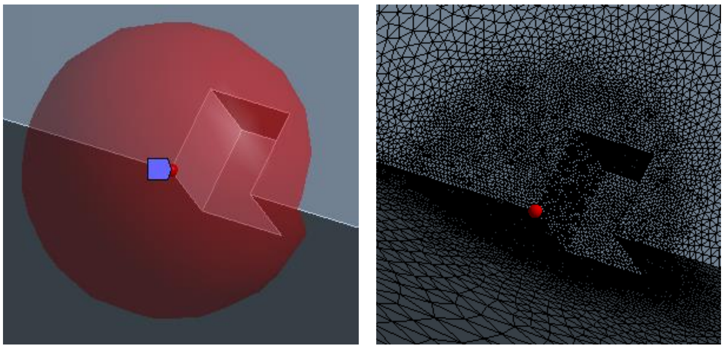


Fig. 39: Mesh in square particle

<u>Square inclined:</u>

The way to make the same cubed particle but turned 45 degrees was very similar than the previous one, but drawing a triangle instead a square in the sketch. This triangle represents the previous square but inclined 45 degrees. Therefore, it was drawn as a triangle with two equal sides with the same size that the square sides (5 mm) and an inclination of 45 degrees from the axis, as it is shown in Fig. 42. Then, it was extruded 5 mm with slice material operation as in the previous geometry.

Once again, we deleted this body and selected the face where our fluid flow run into, that was named “FaceSquare”.

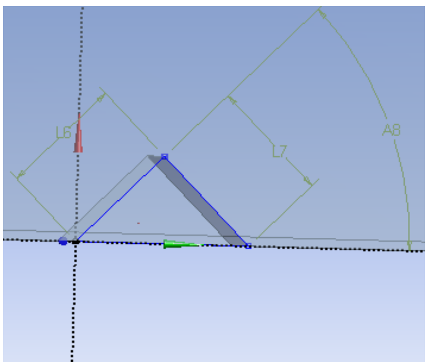


Fig. 42: Square inclined sketch

ERROR! REFERENCE SOURCE NOT FOUND.

Finally we opened the mesh editor to create a sphere of influence surrounding this new particle (centre in the nearest corner and radius= 8 mm, Fig. 43).

With these changes, we got a finer mesh there and to be able to obtain right results in next analysis.

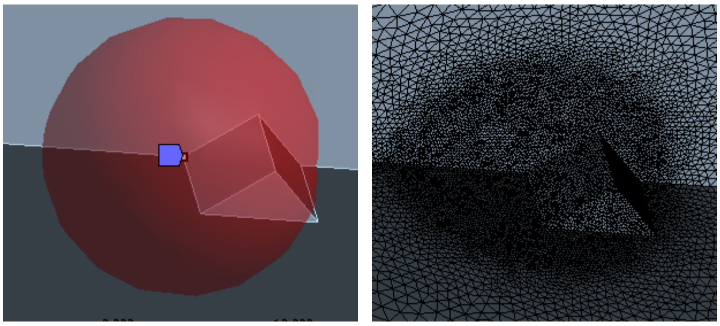


Fig. 43: Mesh in square inclined particle

Sphere:

Last case was a particle with a spherical shape. As we were working with symmetry planes, we only had to draw a quarter of that sphere.

To do that, the best way was to draw a quarter of circle (Fig. 44) and then use it with the revolve tool, choosing the sketch as tool geometry and Y-axis as axis of revolution. Once again, we selected “slice material” in operation. With this operation, we got a quarter of sphere in a new body, which one was deleted later to obtain only the fluid geometry (Fig. 45).

The most difficult part in this case was create the name selection to can measure different parameter in it, because to do that we had to slice our body by the last plane created (getting split the sphere face) and after that create the selection named “FaceSphere” in the right half (Fig. 46).

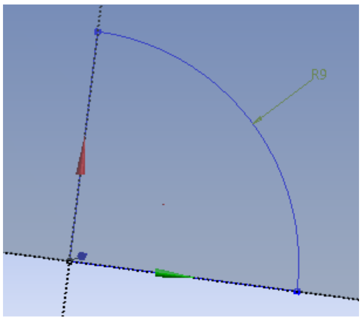


Fig. 44: Sphere sketch

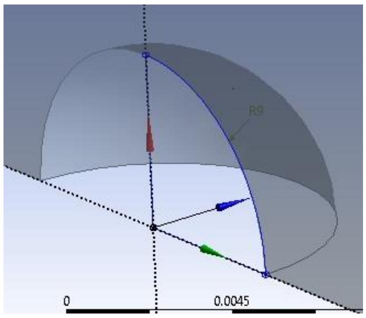


Fig. 45: Revolved body

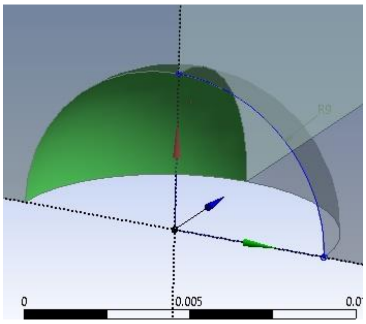


Fig. 47: FaceSphere

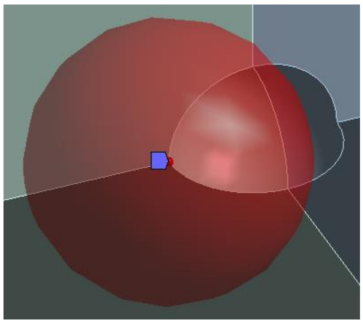


Fig. 46: Vertex sizing

Finally, we defined a sphere of influence in the mesh to improve it around our particle (Fig. 47). The vertex selected was the most closest to our nozzle, due that that was the most interesting side of our particle (where airflow crashed and force was applied).

4.2.2 Setup

About setup, but for some little differences, setups were the same in tree cases.

Starting from the main setups, we only had to define new named selections (FaceSquare or FaceSphere, according with each case) as no slip and smooth wall.

Another new faces and interfaces were defined automatically by default.

In addition, the previous pressure parameter called "Press" was marked as workbench output parameter (Fig. 48). This let us to analyse the behaviour of each particle with different pressures in only one simulation, making it faster and simpler.

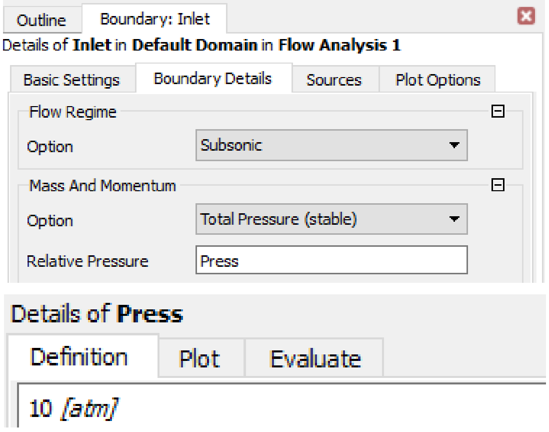


Fig. 48: Press parameter

Finally we defined the design points with pressures that we wanted to simulate (in this case from 6 to 10 atmospheres was enough) and updated all design points.

4.2.3 Results

Following figures show us the results obtained with 10 atmospheres of pressures in different particle's shapes. To analysed the behaviour of the flow and its influence in our particle we decided to apply streamlines of flow from inlet and side holes (as in the previous research, number of streamlines from each place were proportional to the amount of air that went into the nozzle), and measured total pressure in the face where flow run into. This let us see how is the way that fluid took when it was near to the particle and on which zones of the nozzle highest pressures were applied.

Square:

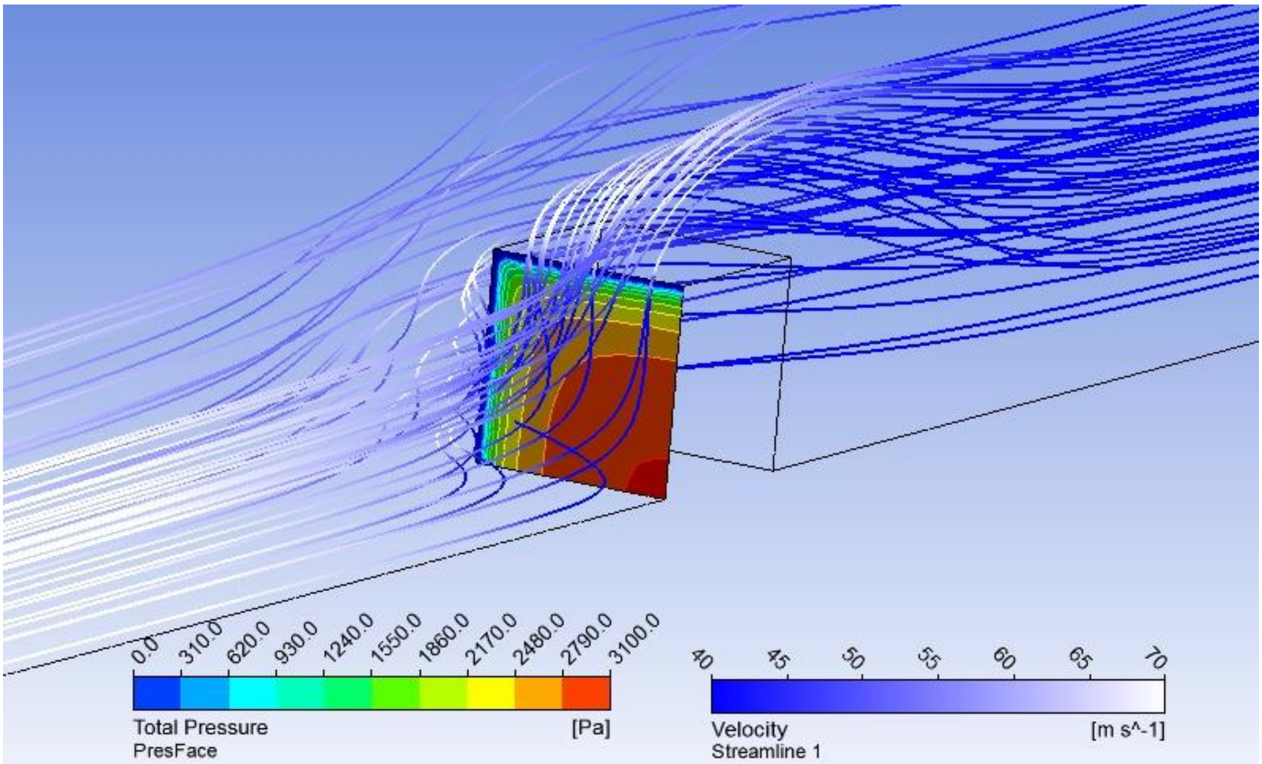


Fig. 49: Square results

In Fig. 49 we could see that the highest pressure is in the centre of square (remember that we were working with symmetry), and that the decreasing rate is not constant when we go away from the centre. It decrease slowly in most of the face until the zone close to the edge, where the decreasing rate is very high (even researching negative pressure).

By another hand, fluid velocity has a different behaviour; it is low near to the centre of particle and higher near to the edge. This is the normal behaviour according physics' laws: when velocity of fluid increase, pressure decreases, and the opposite.

Square inclined:

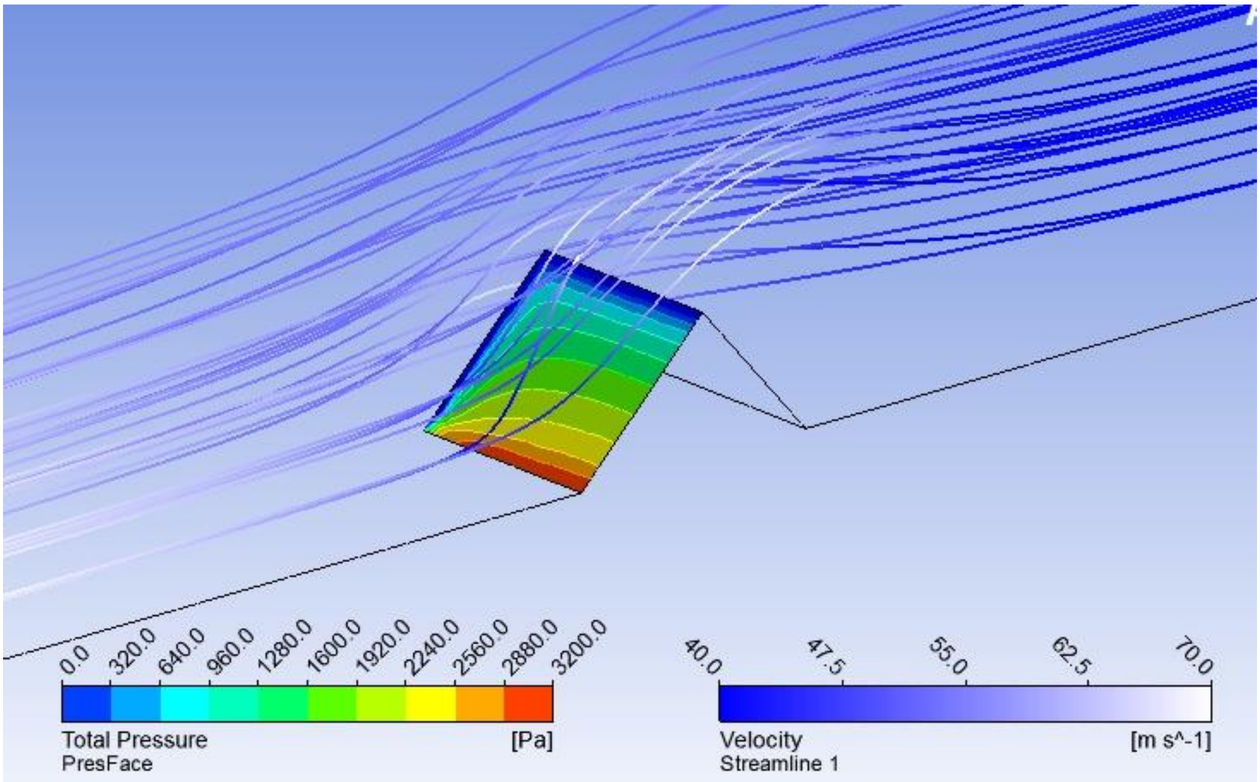


Fig. 50: Square inclined results

In Fig. 50 we can notice the different between when the cube shape are at 45º with the fluid flow, from when it is at 90º (previous figure, Fig. 49). Maximum pressure in face is almost equal, but in this case, it decreases with a higher rate (as result average pressure and force are lower).

About fluid flow, it increases once again close to the edges and after leaves the particle's surface.

Sphere:

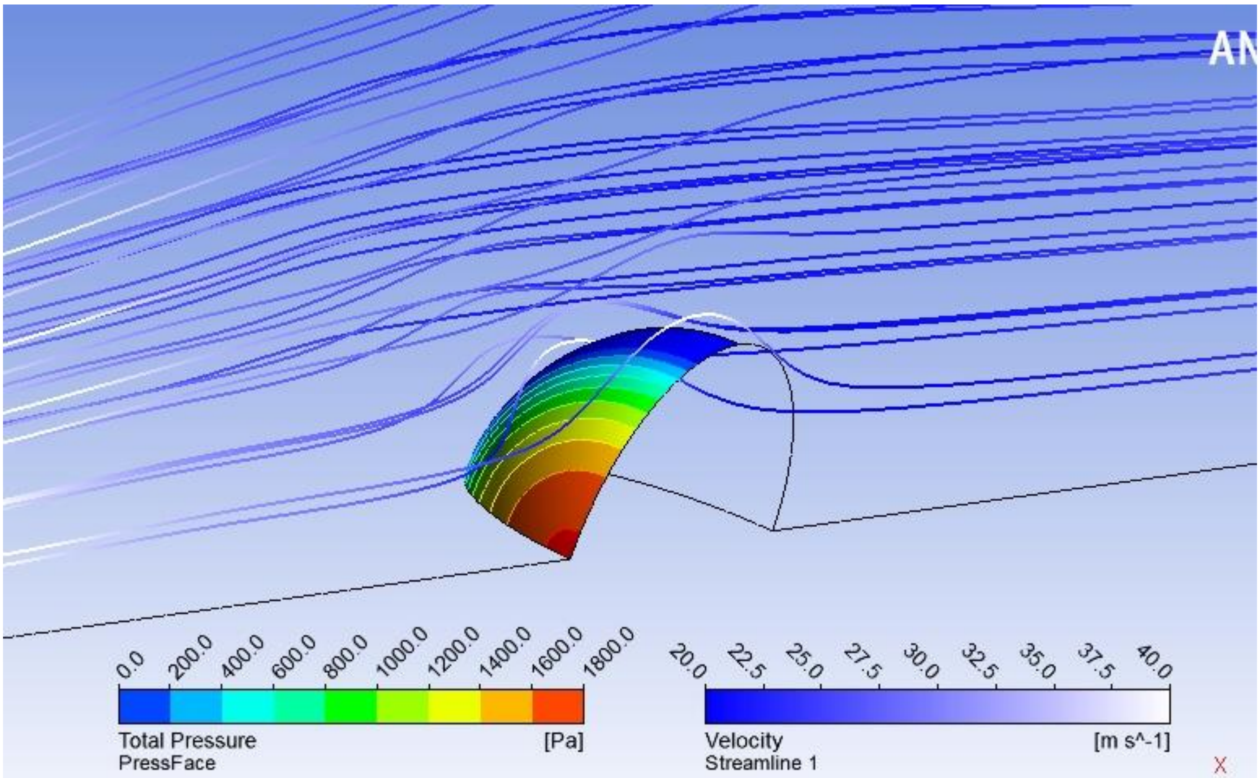


Fig. 51: Sphere results

The last shape simulated was the sphere (Fig. 51), and it was the most different case. Maximum pressure is around half of maximum pressure in square cases; and decreasing rate is similar to square inclined case (so average pressure was even lower than in the previous case).

Once again, airflow velocity increase where pressure is lower, but in this case the difference is the way; here flow go on close to the sphere surface after overcome the middle of the particle, instead go on in a straight path (and separated of particle surface) as in previous cases.

To have a clearer idea of how pressure and force change with particle shape we created a chart to compare all particles' results (Chart 6).

ERROR! REFERENCE SOURCE NOT FOUND.

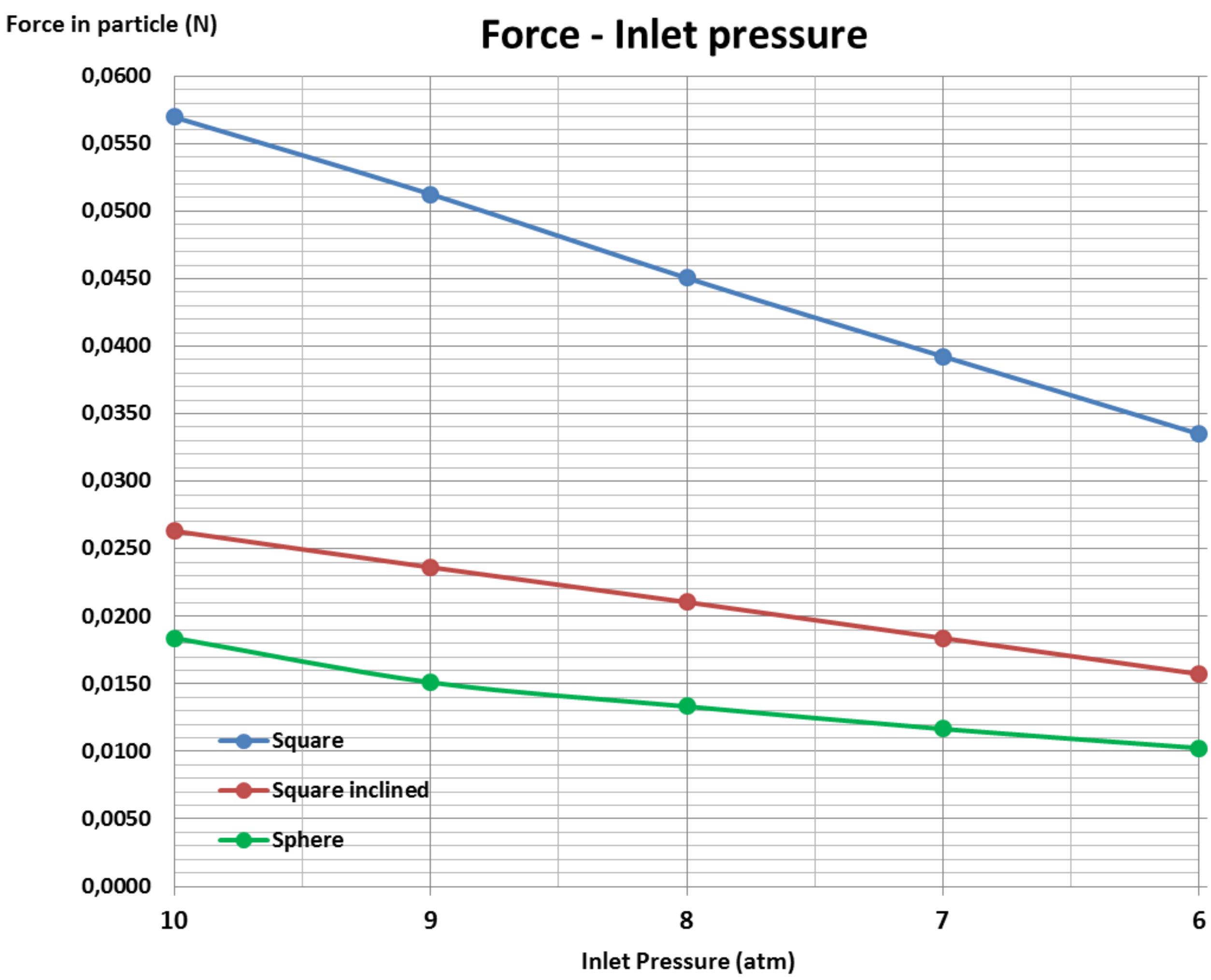


Chart 6: Force – Inlet pressure, different shapes

*4.2.4 Conclusion***

In summary, we had two significantly conclusions:

First one, shape and specially angle of the particle have an important influence in pressures and forces. Therefore, the best way of work would be blow the dust perpendicular to their main face, but since shape of dust use to be irregular and too little, it is difficult to do.

Second one, it is that SIZE….

*****

****

***

ERROR! REFERENCE SOURCE NOT FOUND.

4.3 Distance influence

Next analysis dealt about how high is the influence of distance between the nozzle and the particle in the force applied in it.

In this research, we used the square particle due the fact that it had the highest forces, giving us clearer results.

4.3.1 Geometry and mesh

As we had made the geometry of the nozzle and the square particle in a previous research, we took advantage of it and used it. As result, we only had to do a few changes to obtain the geometry and mesh for this case.

Starting from the geometry previously mentioned, we selected the plane where was drawn the square and changed its parameters, creating two offsets from the base plane. The first one was made with the value of length of the nozzle (0.049 m), staying the plane now just in the outlet of the nozzle; and after that, we defined another offset, now with the distance between the outlet of our nozzle and the particle (Fig. 52 and Fig. 53). This last value was defined as a workbench output parameter making easier and faster futures simulations.

Details View

Details of Plane6	
Plane	Plane6
Sketches	1
Type	From Plane
Base Plane	YZPlane
Transform 1 (RMB)	Offset Y
FD1, Value 1	0.049 m
Transform 2 (RMB)	Offset Y
D FD2, Value 2	0.15 m
Transform 3 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Fig. 52: Parameters in the plane of the

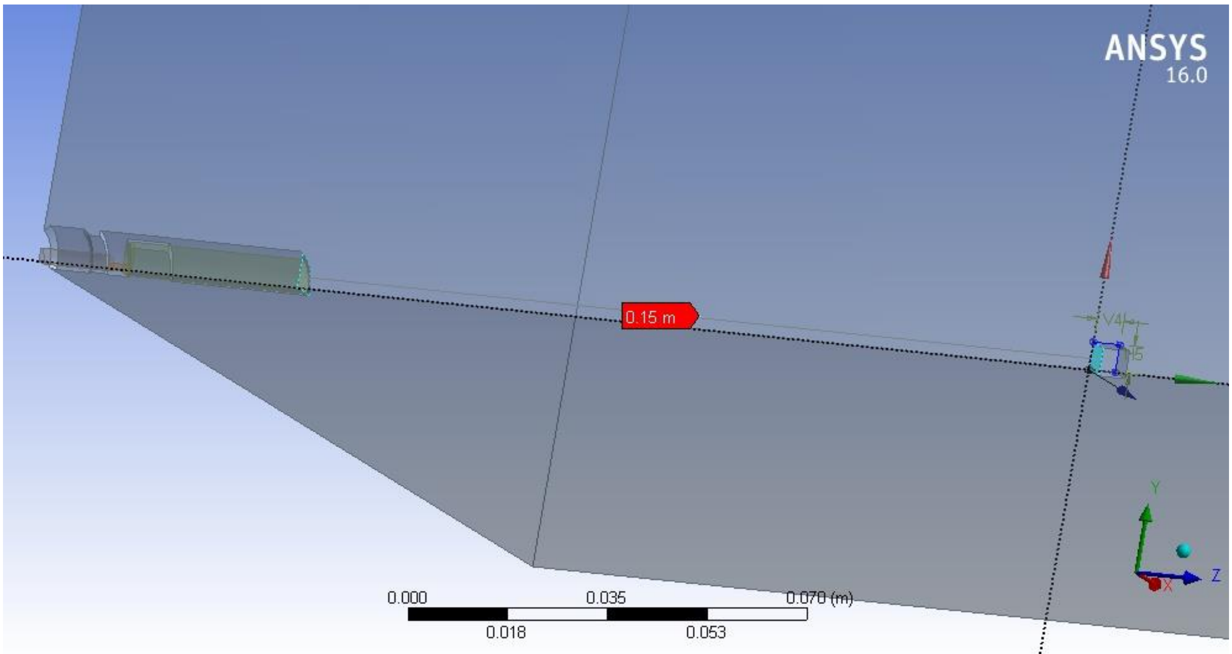


Fig. 53: Offset from nozzle

About meshing, we really did not need make changes, but it was necessary to open the mesh tool and check that every parameters were properly defined (Fig. 54). Setbacks sometimes appear when we modify the geometry and it is necessary redefine the mesh, so is always strongly recommendable to check that the mesh go on being the properly one after any change in geometry.

ERROR! REFERENCE SOURCE NOT FOUND.

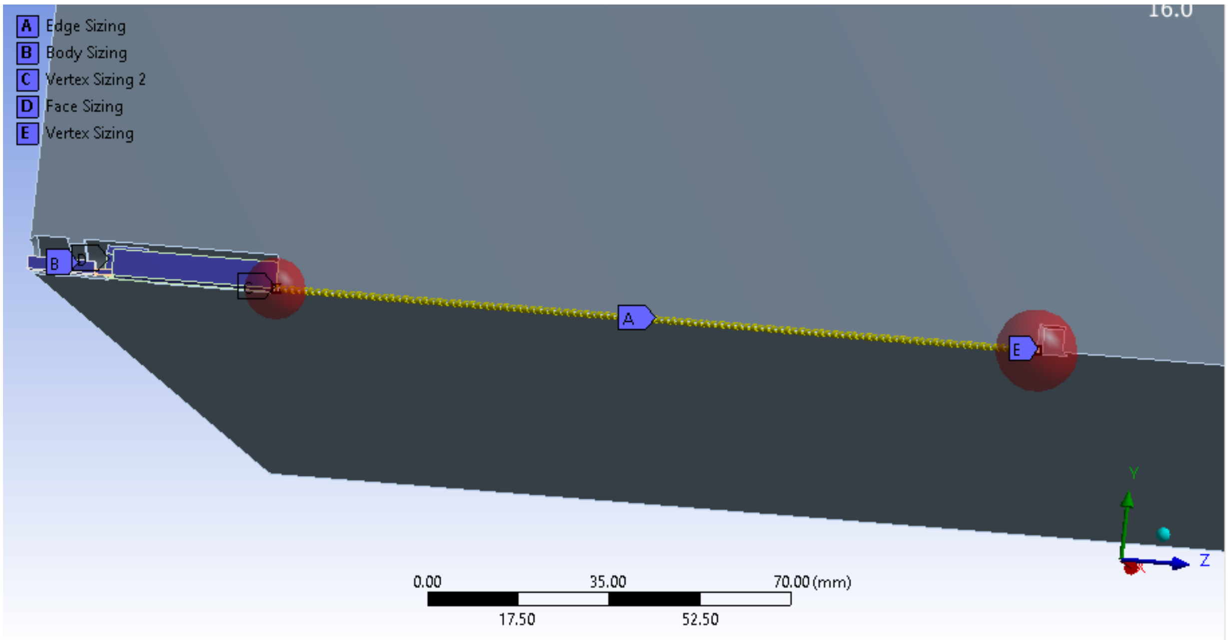


Fig. 54: Mesh sizing

4.3.2 Setup and parameters set

Such as in the previous researches, we began from the main setups. Conditions went on being the same, so we only marked the expression "Press" as Workbench Output Parameter and setups was ready.

Next step was to choose which pressures and distances we wanted to analyse. We though those three different distances were enough to study their influence. We chose 10, 15 and 20 cm of distance from the nozzle, and at 10, 9 and 8 atmospheres in each distance.

Finally, we defined the run settings according with our computer, and finally updated all designed points.

4.3.3 Results and Conclusion

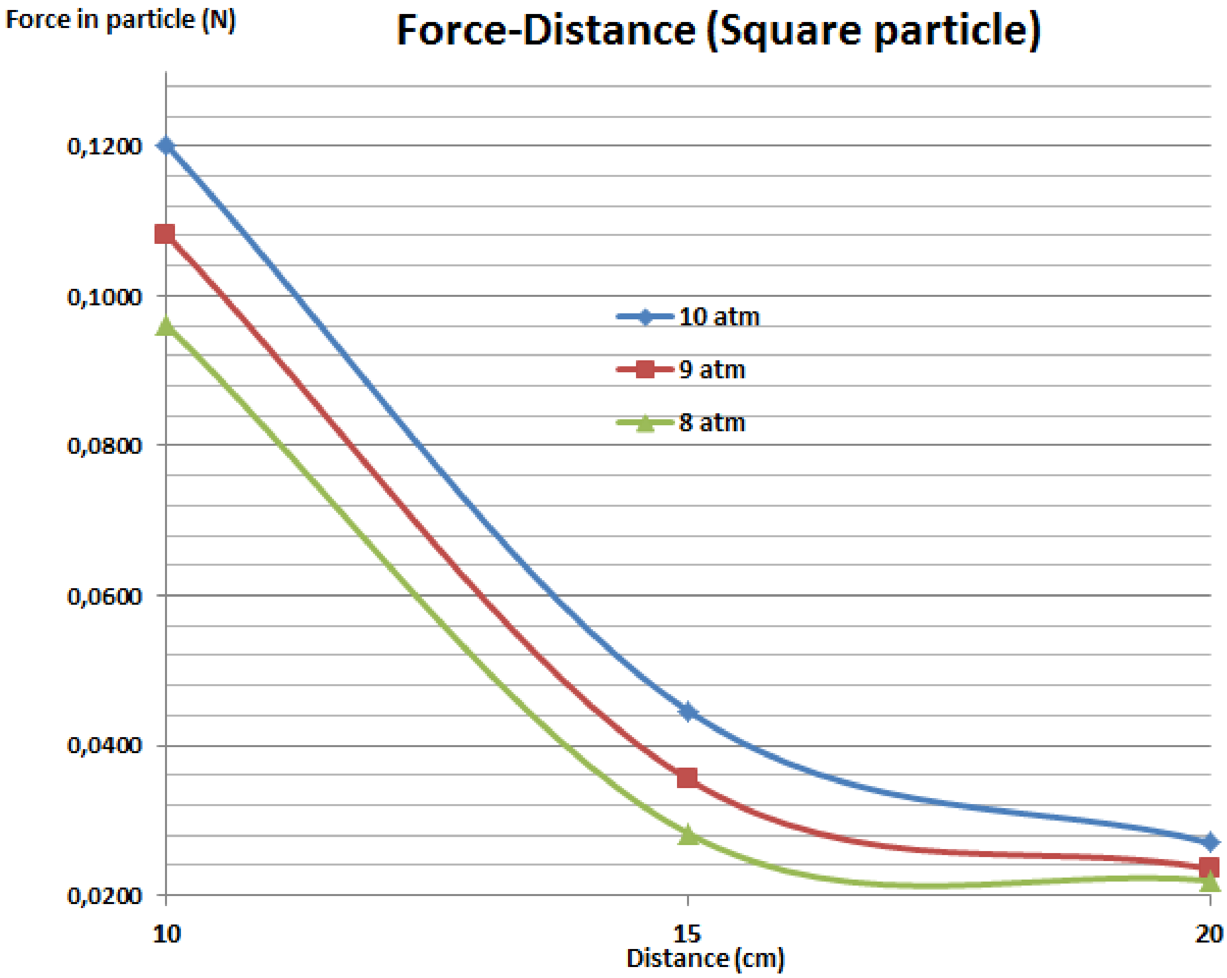


Chart 7: Force- Distance

This simulation gave us back many data, so we thought that the best way to understand the results was creating a chart (Chart 7). On it, we represented force in particle and distance between it and the nozzle. Three lines were drawn (one to each pressure) to check if there was difference in the behaviour from one to another one.

Watching results, we could conclude that distance between particles and nozzle plays a crucial role in forces on dust particles. It should be noted that this influence is exponential instead linear, having more and more influence as we approach.

If we compare how much increase the force when we get close to the nozzle to particles (dust) and how much increase when we turn up the pressure, is easily noticed that influence of distance is bigger than pressure. Therefore, if we want to get higher forces to clean dust, it would be by far more efficient try to bring closer the nozzle to the surface where the dust stays in than increase the pressure (as well as cheaper).

4.4 Roughness influence

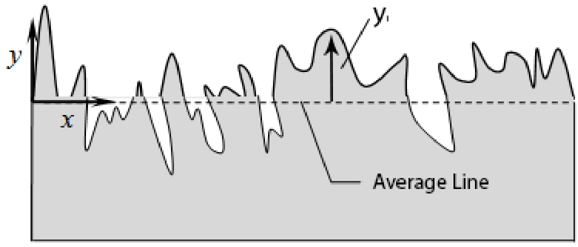


Fig. 55: Roughness

Roughness is a component of surface texture. It is quantified by the deviations in the direction of the normal vector of a real surface from its ideal form (Fig. 55). If these deviations are large, the surface is rough; if they are small, the surface is smooth. Although a high roughness value is often undesirable, it can be difficult and expensive to control in manufacturing. Decreasing the roughness of a surface will usually increase its manufacturing costs. This often results in a trade-off between the manufacturing cost of a component and its performance in application. There are many different roughness parameters in use and each one of the roughness parameters is calculated using a formula for describing the surface but R_a is by far the most common, though this is often for historical reasons and not for particular merit, as the early roughness meters could only measure R_a. Other common parameters include R_z, R_q, and R_{sk} *[10]*[10].

R_a parameter is easily calculated as we can see in the next equation, where "n" is the number of peaks and "y_i" distance between the average line and each peak (Fig. 55).

$$R_a = \frac{1}{n}\sum_{i=1}^{n}|y_i|$$

However, ANSYS uses a different parameter of roughness called "sand-grain roughness". This equivalent roughness supposes that spheres (as sand grains) compose the surface and this parameter define that spheres radio. Therefore, in our case we need convert $\mathbf{R_a}$ parameter to sand-grain roughness parameter **(ε)**.

4.4.1 Conversion of parameters

To convert the normal roughness parameter $\mathbf{R_a}$ into sand-grain parameter **(ε)** we can use the following way *[11]*[11]:

$$\boldsymbol{R_a} = \frac{1}{\varepsilon}\int_{x=0}^{\varepsilon}|\boldsymbol{y}-\bar{\boldsymbol{y}}|\mathrm{dx}$$

For the profile in Fig. 56

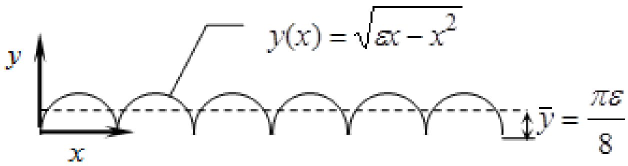


Fig. 56 : Sand grains surface

$$\boldsymbol{y}(\boldsymbol{x}) = \sqrt{\varepsilon\mathbf{x} - \boldsymbol{x}^2}$$

And

$$\bar{\boldsymbol{y}} = \frac{\pi\varepsilon}{8},$$

Substituting the previous equations:

$$\boldsymbol{R_a} = \frac{\varepsilon}{2}\left(\frac{\pi}{2} - \cos^{-1}\left(1-\frac{\pi^2}{16}\right)^{\frac{1}{2}} - \frac{\pi}{4}\left(1-\frac{\pi^2}{16}\right)^{1/2}\right)$$

Finally solving for ε and simplifying gives

ERROR! REFERENCE SOURCE NOT FOUND.

$$\boldsymbol{\varepsilon = 5.863R_a}$$

Now we can convert the typical roughness factor that manufacturers give us of different materials (R_a) into sand-grain parameter and simulate in ANSYS the behaviour of airflow inside our nozzle with different surface roughness (Table 7).

Table 7: Roughness conversion

	Roughness parameter ,Ra (mm)	Sand-grain parameter ,ε (mm)
Stainless Steel	0,0150	0,0879
Steel commercial pipe	0,0450	0,2638
Galvanized steel	0,1500	0,8795
Aluminium	0,0015	0,0088
Plastic	0,0050	0,0293

4.4.2 Set-up and Solution

After convert all parameters we were ready to put it in ANSYS set-up. To do this, the best way was creating an expression called **Roughness**, put it in Sand Grain Roughness (Fig. 57) and used it as Workbench Input Parameter. In this way, we could change it in parameters place and update all solutions in only one-step.

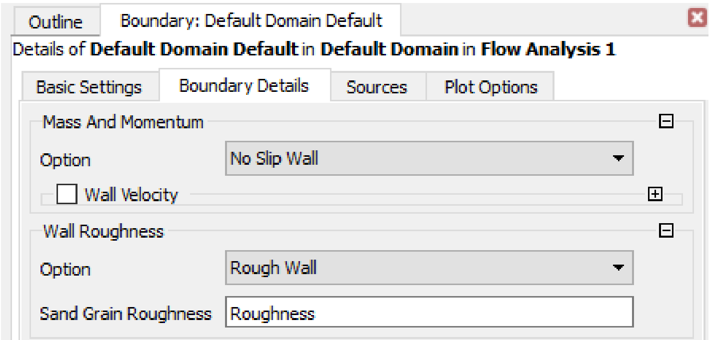


Fig. 57: Set-up roughness

However, before update the project we created three circular planes with 5 mm of radio where it was able measure the average velocity: inlet, outlet and 5 cm from the outlet; and finally we only had to write the new expression to measure average velocity in those planes (VelInlet, VelOulet and VelNear) and marked them as Workbench Output Parameters.

Finally, we updated our project and waited for the solution.

ERROR! REFERENCE SOURCE NOT FOUND.

4.4.3 Results and conclusion

Results obtained after perform the simulation are showing in the following table:

Table 8: Roughness influence

P33 - Roughness	P29 - VelInlet	P30 - VelOulet	P31 - VelClose	P32 - force
mm	m s^-1	m s^-1	m s^-1	N
0.8795	174.42	179.12	68.074	0.054737
0.2638	174.66	177.16	67.855	0.055974
0.0293	174.81	172.79	67.585	0.056679
0.0088	175.2	172.94	67.656	0.056844
0	175.51	173.06	67.712	0.056965

As we can see in Table 8 that velocity at 5 cm from the outlet of our nozzle change less than 1m/s with different roughness, so we can assert that the roughness of our material surface is not relevant. Therefore, we could avoid this parameter in the choice of materials and choose the cheapest option without care about the surface finish.

This happens due our flow is air, which has a low density; if we work with water or another liquid the influence of roughness would be higher and we should be aware about how change in each case to choose the material carefully.

4.5 Mechanical analysis

One very useful analysis system in ANSYS is the Static Structural Analysis. It let us analyse forces, pressures, temperature and some other parameters in the structure of a solid.

In this case, we analysed the resistance of our nozzle the forces due to the airflow and in the case of something hit it.

4.5.1 Preparing analysis

First step was open a solved simulation with right results. Then, we opened geometry and switch from ON to OFF where we chose supress the tool body in the first boolean operation (where we subtract the nozzle from the big cube of fluid). Next was supress all another bodies to have only the nozzle geometry (solid).

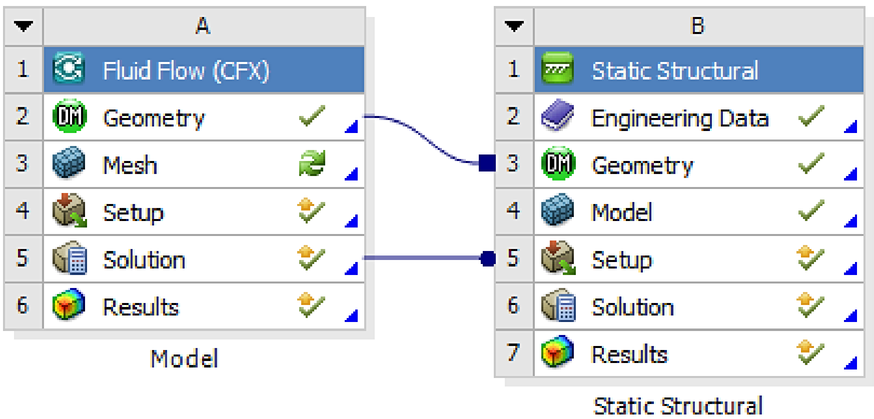


Fig. 58: Static Structural connections

Later, we dragged “Static Structural” from toolbox to workbench scream, and then connected geometry from first simulation to the new one, and the same from old results to setup of the new simulation, as its show in Fig. 58. This make possible use the same geometry, and even most important, to import results from airflow analysis to this static structural analysis.

In structural analysis, it is necessary to choose or create the material of the solids. Our nozzle was made in aluminium, so we chose it. To do it, first it was necessary open “Engineering Data Sources”, activate “Outline” view, and in that scream select and add “aluminium alloy” from general materials. After that, already was allowed to select aluminium as material in our model. As Fig. 59 show, the way to set the material was click in the nozzle body and there select the material in Assignment and apply.

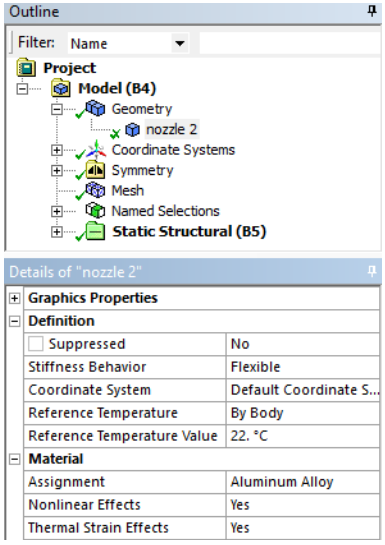


Fig. 59: Select material

Next step was to enter in “model” and define simulation conditions. As in fluid flow simulation, next step after have the geometry was to define the mesh. In this case was enough to increase the relevance to one hundred and switch relevance centre and angle centre from curse to fine (Fig. 60).

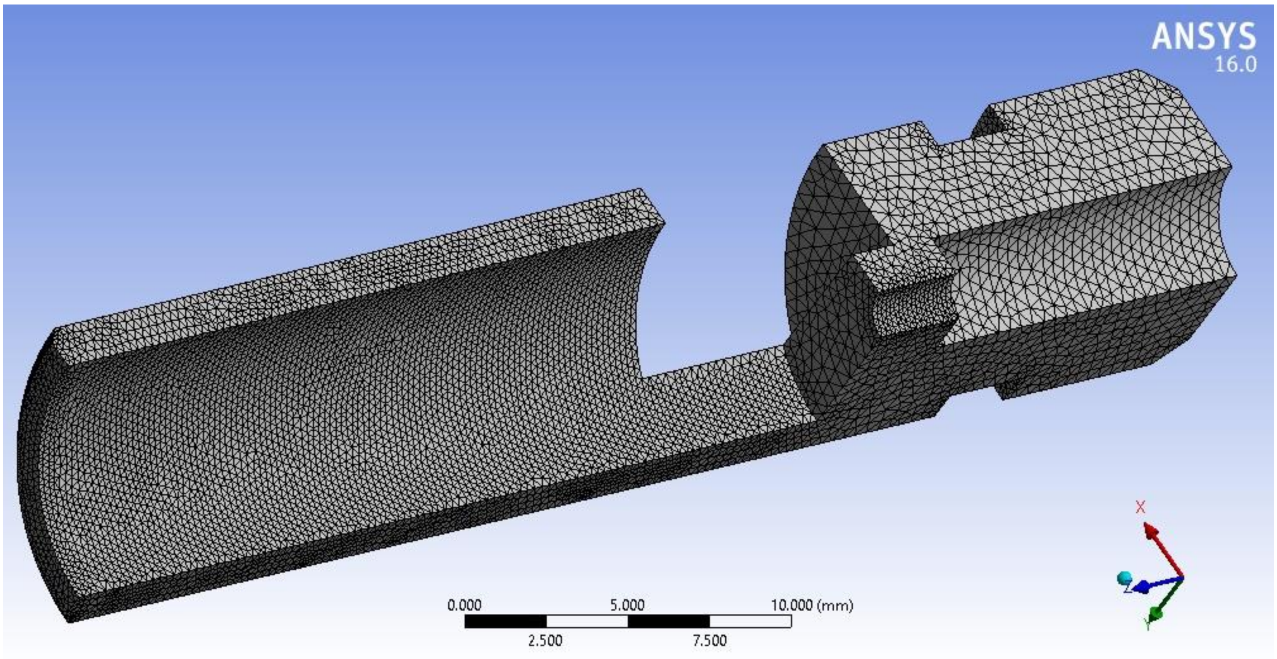


Fig. 60: Mechanical mesh

Later, we imported “Pressure” in “Imported Load” and selected all faces where we wanted import the pressure from airflow. This was in all faces that are in contact with air. Then we inserted a “Fixed support” in the face where would be screwed as is show in Fig. 61.

With the previous conditions defined, was time to apply the solutions that we need to study.

ERROR! REFERENCE SOURCE NOT FOUND.

4.5.2 Results and conclusions

In this case we started with Equivalent Stress (von-Mises) showed in Fig. 62. Von-Mises criteria is the most used in engineering, especially in ductile materials; and it give us clearly how high is the stress in each part of our material. In this case, it is obtained in the end of the inlet chamber with a value around three MPa. This value is insignificant when for example the tensile ultimate strength of aluminium alloy is above 300 MPa.

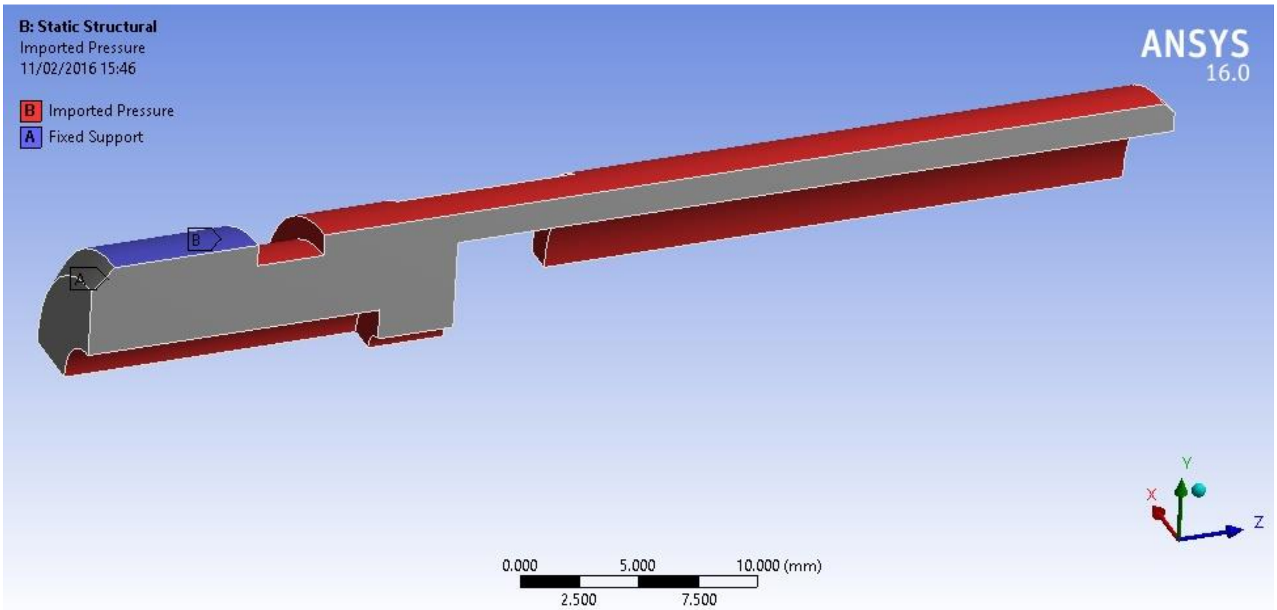


Fig. 61: Selected faces

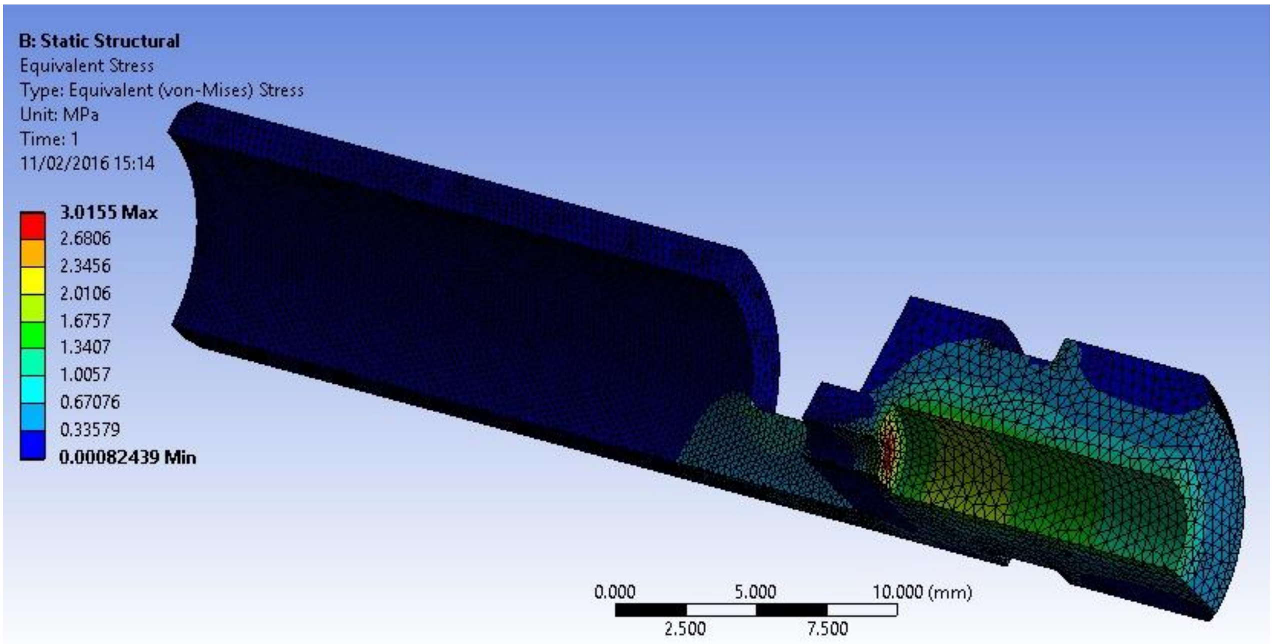


Fig. 62: Equivalent Stress (von-Mises)

If we mix equivalent stress with the properties of our material it is be able to obtain the total deformation. It show us how high are deformation and in which direction it is. In this case, as the values are very low, we applied an extension scale to could see direction of deformation (Fig. 63). Anyway, due the low values of the deformation (maximum was $1.33e^{-4}$), direction is not relevant either.

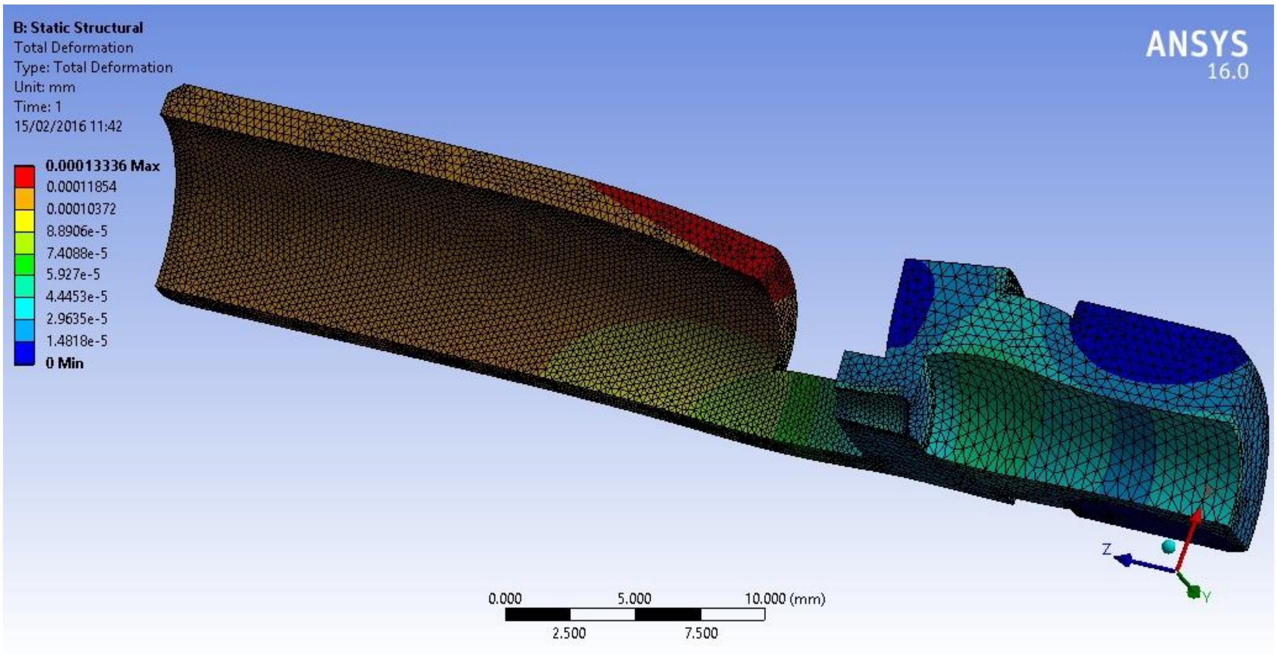


Fig. 63: Total deformation (2e^{-4} scale)

As conclusion, work forces are not relevant, so we could have used another material without worry about it. As forces are not significant as well as roughness, material would be chosen according with prize and ease of machining.

4.5.1 Crash postulation

Sometimes it is interested to make other tests with terms different from work terms; for example to know if the piece would resist the impact of an object, a fall or where it would break.

In this case, we applied a force on the outlet of the nozzle to simulate an able impact of our nozzle with the piece that is cleaning or another one. The force was applied in an angle between perpendicular a parallel to the nozzle axis and with a value of 500 newton (around 50 kgf) that would be a strong hit. The procedure to add an external force is simply, we only click insert, force and wrote the value of the force in each coordinate (Fig. 64).

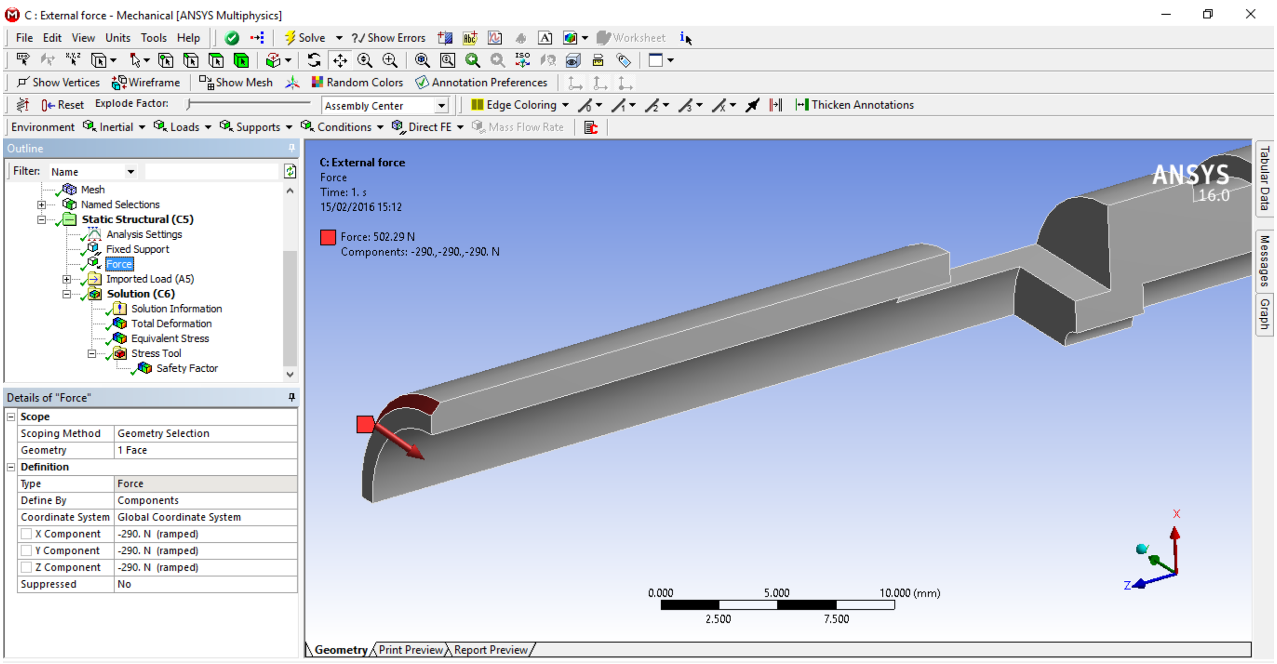


Fig. 64: Applying external force

After that, we inserted some solutions: Total deformation, Equivalent Stress (von-Mises) and Safety Factor and finally solve them.

Once again, deformation was very low (0.02mm) so we needed to apply a scale to could watch its direction (Fig. 65).

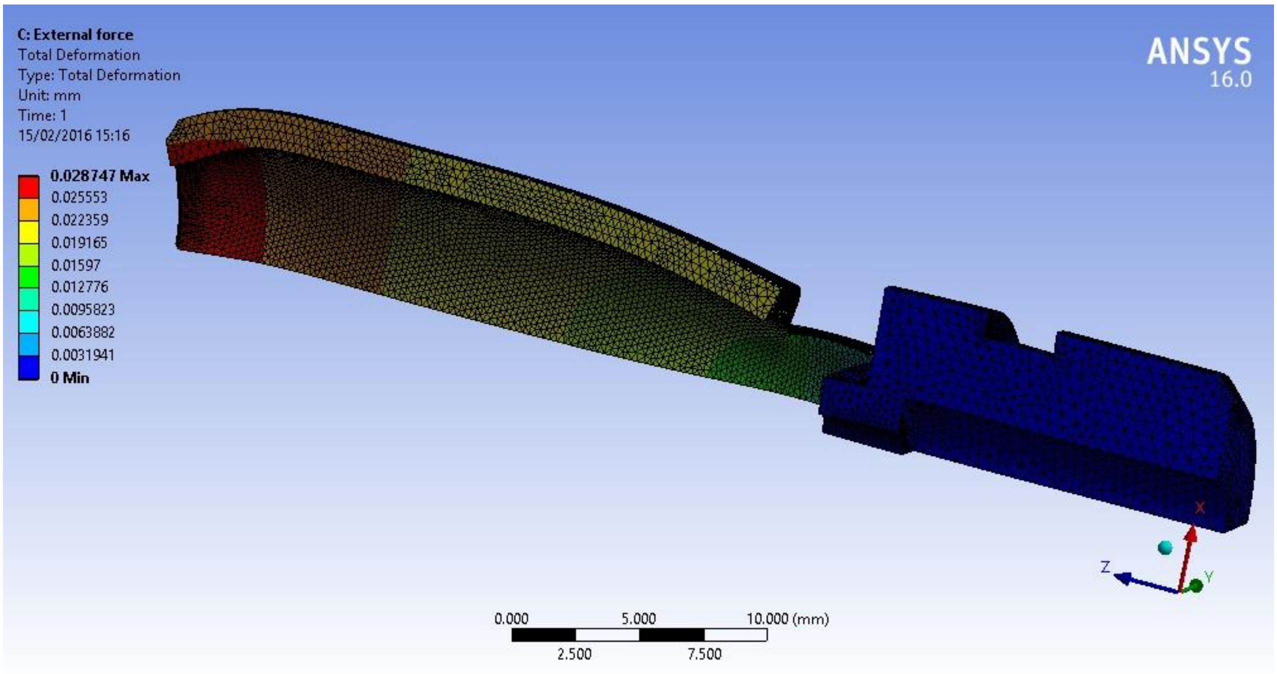


Fig. 65: Deformation

In the other hand, this time the equivalent stress was relevant. We could see that it researched almost 250 MPa. In addition, we could identify the most dangerous zones where equivalent stress was higher (yellow zones shows in Fig. 66).

ERROR! REFERENCE SOURCE NOT FOUND.

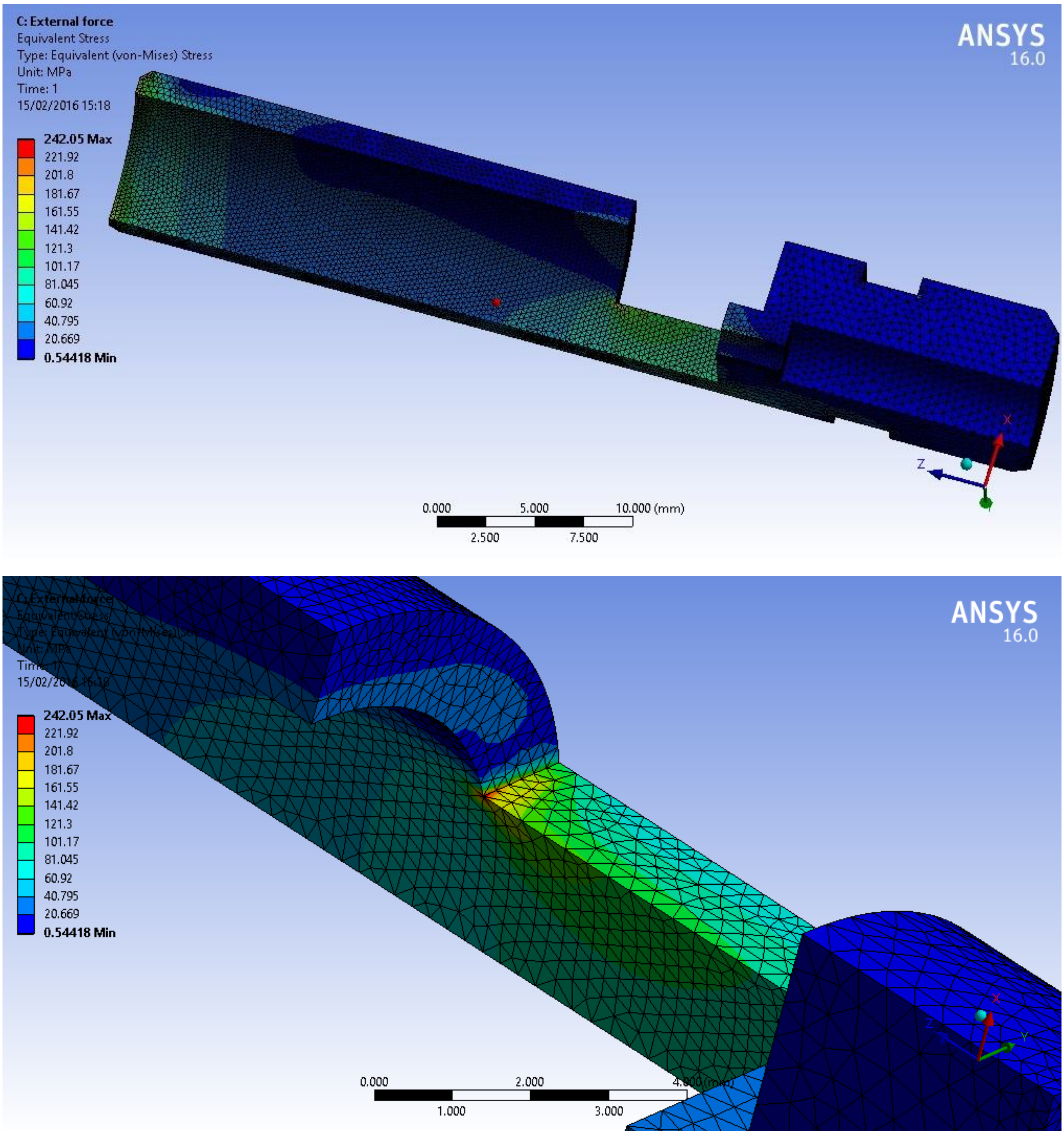


Fig. 66: Equivalent stress

Nevertheless, the most representative simulation to watch if the material would break and where is the Safety Factor one. In that one is possible watch in that zones the stress get over the resistance of the material (so, break) or how many times highest must be the stress to break in that zone (safety factor).

In this case, Fig. 67 shows us that even applying this external force our nozzle would resist without break due there is no red zones (safety factor equal or below one).

In addition is possible guess that if that force was higher enough to break it, it would break in the part from where the outer air go in the nozzle.

ERROR! REFERENCE SOURCE NOT FOUND.

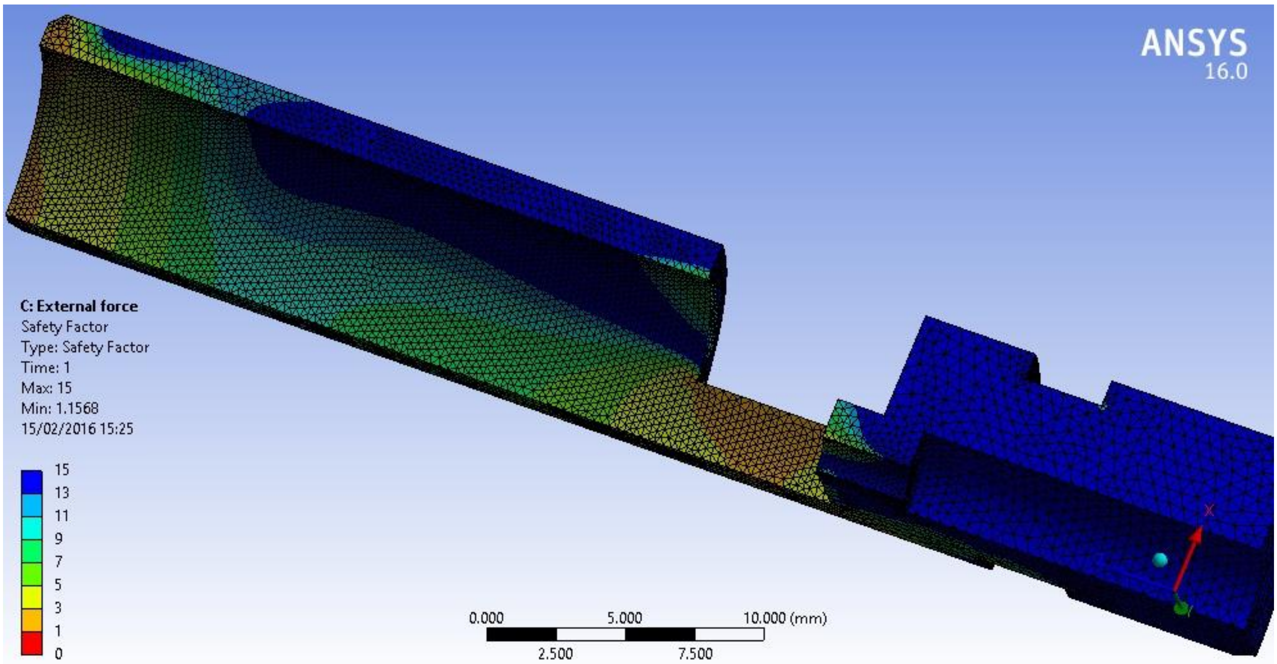


Fig. 67: Safety Factor

As conclusion, we have here a strong nozzle that will not be broken easily in normal conditions.

However, it should be noted that sometimes we wish have a device that break easily for the zone where we want. For example, when we have an expensive machine connected to it, it is better having a weak nozzle that breaks in case of some works wrong than break the machine or the junction; which ones use to be more expensive to repair or replace than a nozzle.

4.6 Non continuous flow (Transient)

Machines of dust cleaning sometimes works with a non-continuous airflow. This means that they blow compressed air for a moment, and next moment there is not airflow. It is a way of work that can be positive because it saves air and it is be able to give us same effectiveness cleaning dust.

In ANSYS, there is another analysis type different from Steady State, which is called Transient Analysis and is used to simulate the behaviour of our model for a part time.

Once again, we started from the model of our nozzle blowing to a square particle because it is the most representative case. Therefore, we did not need to modify anything in geometry or mesh.

4.6.1 Model and Setup

In this case there was a previous step before define the setup. It was necessary to define first what model of pressure in inlet we wanted to simulate, and then, to elaborate some equations which representing mathematically that model.

To make the later work easier, we decided to create the equations using some expressions (Fig. 68); thereby only changing them, we could obtain another model without have to do a new equation.

ERROR! REFERENCE SOURCE NOT FOUND.

Parameters defined for us in expressions were the following: **t1** (time step of blowing, in seconds); **t2** (time step of break, in seconds); **ta** (time that nozzle takes to get the maximum pressure, in seconds) and **Pmax** (maximum or main pressure of work, in atmospheres). In addition, another expressions were defined with the aim of have a clearer equation and easily of understand: **Press2** (main parameter, which define the pressure in inlet and contain the all other equations); **time** (equation needed to turn the time into a dimensionless value); **Pin2**, **Pin3** and **Pin4** (expressions to get the jumps smoothly).

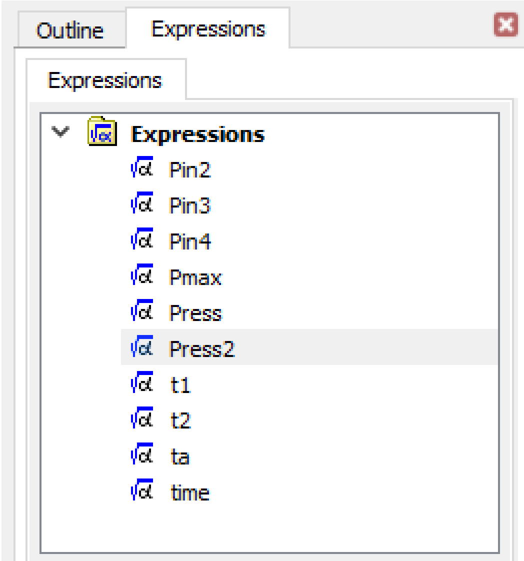


Fig. 68: expressions

As tip, when we make new expression, as in this case, it is strongly recommended do not define the parameters with the same value; this do easier notice if there are any mistake.

According with this, we decided to begin with an analysis in which one time of blowing (***t1***) was one seconds and time rest (***t2***) equal to an a half seconds. Moreover, pressure (***Pmax***) was defined as 10 atmospheres and time of switch (***ta***) was 0.1 seconds.

We began creating the equation that gave us a square signal that switches times with full pressure and times with any pressure.

$$\mathbf{Press2} = Pmax * \Big(\mathbf{step}(t1-time)+\mathbf{step}(time-(t1+t2))*\mathbf{step}(2*t1+t2-time)+ \mathbf{step}(time-2*(t1+t2))*\mathbf{step}(3*t1+2*t2-time)\Big)$$

“Step” is a predefined ANSYS’ function that gives us back a “0” if the value inside brackets is negative and “1” if it is equal to zero or positive.

Parameter “time” was defined as “time= *t/1[s]*”; where “t” is the time on simulation is working in each moment. We needed divided it by “1[s]” to turn it into a dimensionless parameter and can operate with another parameters and numbers of equations.

As we mentioned before, ***Pmax***, ***t1*** and ***t2*** were defined as numbers (*10 [atm]*, *1* and *0.5* respectively in this case).

This equation could have been enough simulate this case, but it is not real at all that pressure increase suddenly and also, it made that ANSYS did not work well and gave us back some wrong results. Therefore, to solve this problem, we create a equations to introduce a little slope in the increasing of pressure.

We began creating the first slope (***Pin2***). In this equation what we did was to split ***ta*** time in ten equal parts to get a smooth slope. As show the following equation, the value of ***Pin2*** increase 0.1 in each step, from 0 to 1, getting a slowly increasing along the time defined in ***ta***.

$$\mathbf{Pin2} = \Big(\mathbf{step}(time-0.1*ta)+\mathbf{step}(time-0.2*ta)+\mathbf{step}(time-0.3*ta)+ \mathbf{step}(time-0.4*ta)+\mathbf{step}(time-0.5*ta)+\mathbf{step}(time-0.6*ta)+ \mathbf{step}(time-0.7*ta)+\mathbf{step}(time-0.8*ta)+\mathbf{step}(time-0.9*ta)+ \mathbf{step}(time-ta)\Big) *0.1$$

This equation only have influence in when time is between 0 and ***ta***, after that time is bigger than ***ta***, so ***Pint2*** is equal to 1 and do not affect the equation of pressure.

ERROR! REFERENCE SOURCE NOT FOUND.

Similar equations were used to the next slopes (***Pin3*** and ***Pin4***) but with a few changes due now the slope are in later times, so we had to add *"-t1-t2"* in all steps in Pin3, and *"-2*t1-2*t2"* in all steps in Pin4.

$$\mathbf{Pin3} = \Big(\mathbf{step}(time-0.1*ta-t1-t2)+\mathbf{step}(time-0.2*ta-t1-t2)+\mathbf{step}(time-0.3*ta-t1-t2)+\mathbf{step}(time-0.4*ta-t1-t2)+\mathbf{step}(time-0.5*ta-t1-t2)+\mathbf{step}(time-0.6*ta-t1-t2)+\mathbf{step}(time-0.7*ta-t1-t2)+\mathbf{step}(time-0.8*ta-t1-t2)+\mathbf{step}(time-0.9*ta-t1-t2)+\mathbf{step}(time-ta-t1-t2)\Big)*0.1$$

$$\mathbf{Pin4} = \Big(\mathbf{step}(time-0.1*ta-2*t1-2*t2)+\mathbf{step}(time-0.2*ta-2*t1-2*t2)+\mathbf{step}(time-0.3*ta-2*t1-2*t2)+\mathbf{step}(time-0.4*ta-2*t1-2*t2)+\mathbf{step}(time-0.5*ta-2*t1-2*t2)+\mathbf{step}(time-0.6*ta-2*t1-2*t2)+\mathbf{step}(time-0.7*ta-2*t1-2*t2)+\mathbf{step}(time-0.8*ta-2*t1-2*t2)+\mathbf{step}(time-0.9*ta-t2*t1-2*t2)+\mathbf{step}(time-ta-2*t1-2*t2)\Big)*0.1$$

Finally, we add these equations (***Pin2***, ***Pin3*** and ***Pin4***) to the main equation ***Press2*** getting the final equation:

$$\mathbf{Press2} = Pmax * \Big(Pin2*\mathbf{step}(t1-time)+Pin3*\mathbf{step}(time-(t1+t2))*\mathbf{step}(2*t1+t2-time)+Pin4*\mathbf{step}(time-2*(t1+t2))*\mathbf{step}(3*t1+2*t2-time)\Big)$$

Last step was to check that our expressions were defined in a right way. To do it, ANSYS has a tool that allow us plot the expressions and watch their shapes (Fig. 70). This made it easier to know if we have the right shape or a wrong one.

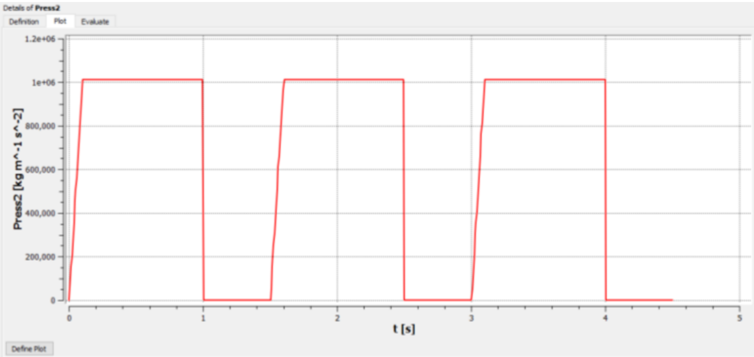


Fig. 69: Press2 plot

Once that we had all expression defined and checked, we could start with the setups.

First was switching the analysis type from Steady State to Transient. In the same tab, we had to define some times. For example, time duration was set as total time and 4.5 seconds (just the time that we needed to simulate three impulses and its breaks, 3 * 1s + 3 * 0.5s). Another very important parameter is "Time Step", which defines the points that are going to be simulated; in our case, we considered that each 0.05 seconds was enough (Fig. 69).

Later, we used the new expression ***Press2*** to define the relative pressure in Inlet, instead the previous Press.

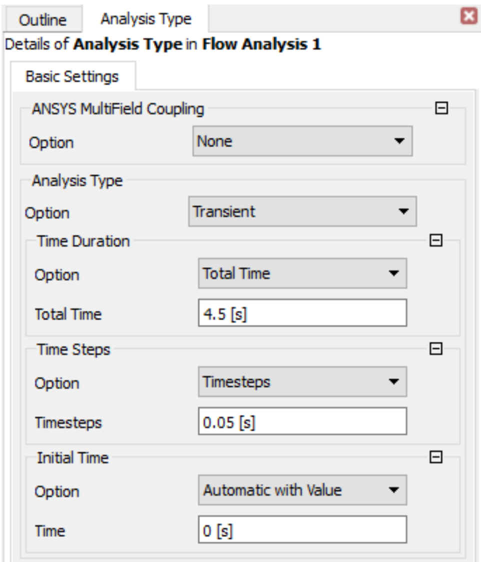


Fig. 70: Analysis type setup

ERROR! REFERENCE SOURCE NOT FOUND.

About solver controls (Fig. 71), after try the default number of loops (10) we could notice that solution was not right at all, so go on increasing the maximum coefficient of loops until get the right solution (60 in this case). It is important keep in mind that increasing the number of loops, time of simulation is raised in the same ratio.

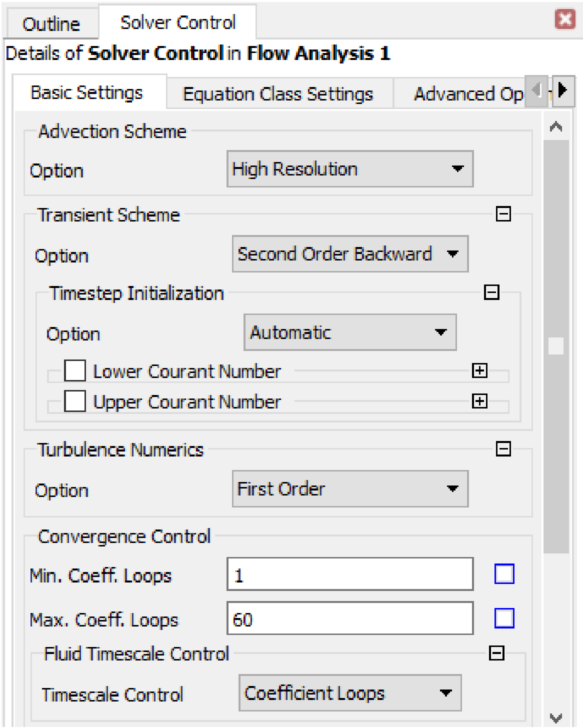


Fig. 71: Solver control setup

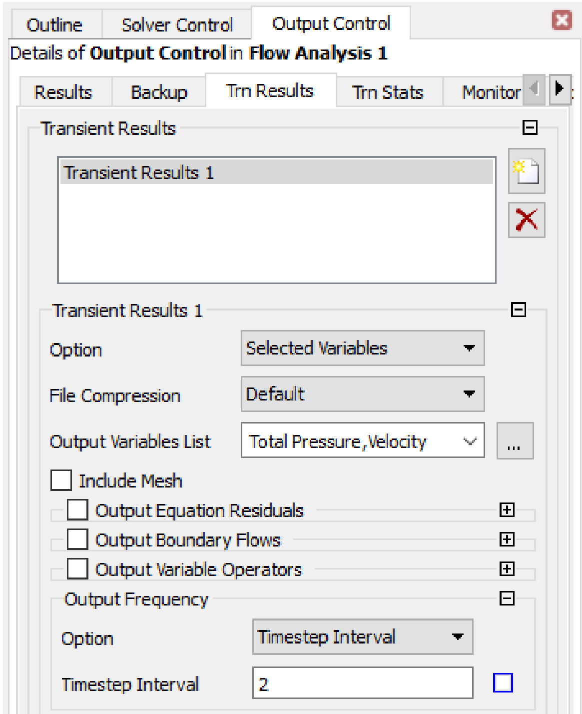


Fig. 72: Output control setup

Finally, in this kind of analysis we must create in Output Control the settings about what information we want to keep. Obviously, how much more information we save, bigger will be the file. Therefore, it is important to select only the variables that we really need analyse, and the maximum time step interval of savings that let us have all the useful information. For this research, we only wanted to know the variables Total Pressure and Velocity; and how hot time step was 0.05s, we saved results each two time steps (this means, each 0.1s), (see Fig. 72).

With these adjustments, setup is ready. Later we set the solution controls according our computer and run the solver.

It is noteworthy that in this kind of simulation we needed a long time to solver it due the high number of loops for each time step, which ones were a high number too.

Finally, we duplicated this simulation three times to make it with different times. Another three cases were with following parameters: ***t1***=0.5s and ***t2***=0.5s; ***t1***.5=1s and ***t2***=0.5a; ***t1***=0.2s and ***t2***=0.2s.

4.6.2 Results and conclusion

Transient results let us watch all previous selected variables in the time step that we want. Thanks of this, we could know how the behaviour of our airflow in different times (Fig. 73).

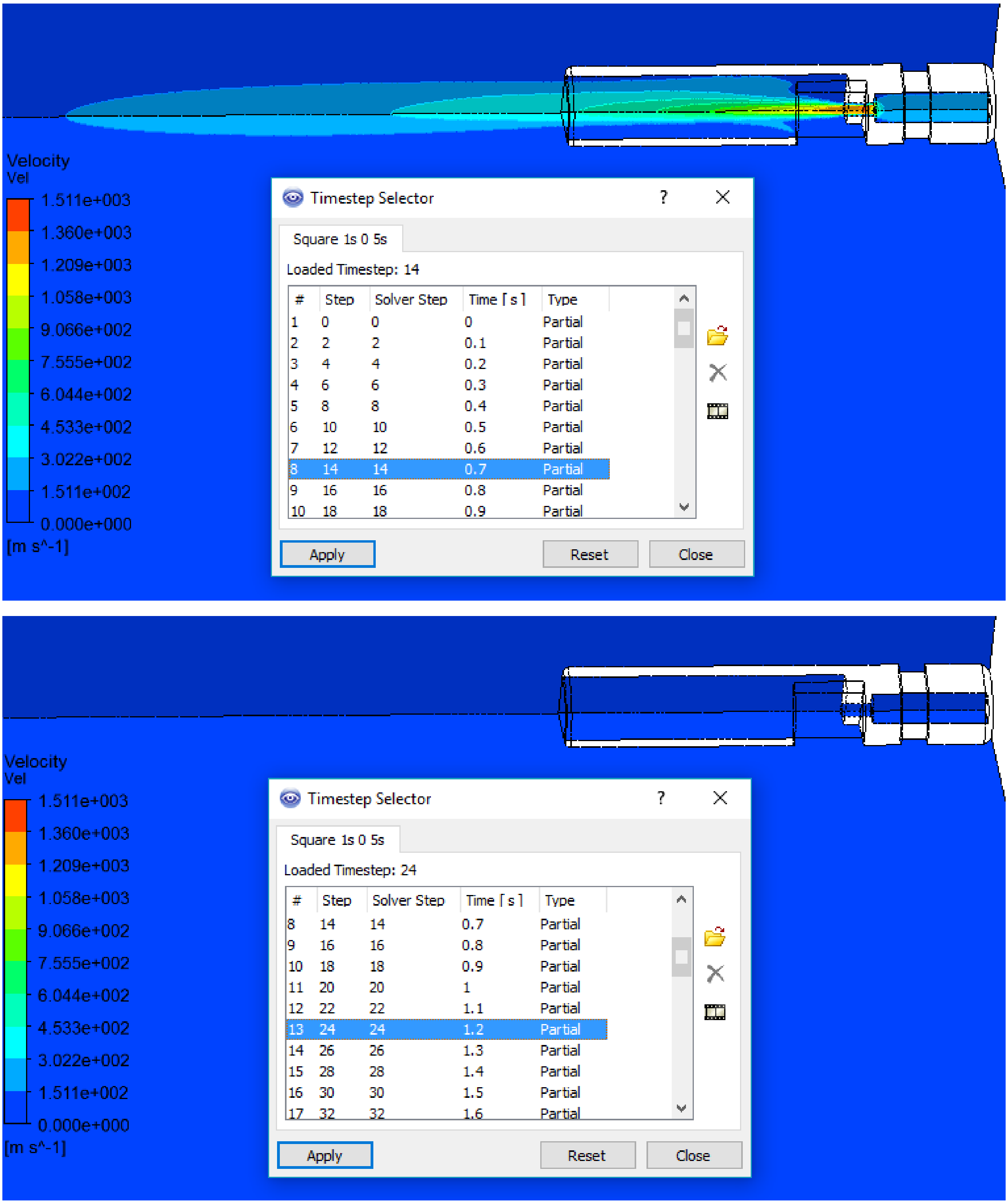


Fig. 73: Transient results

Could be useful to see what happen on a certain moment (especially if we have detected some problem or different behaviour on that moment), but it is generally more useful create a chart where see the behaviour of some parameter or data along time.

In our case, we did one with force in particle and time due it was our main aim. Charts obtained from the result of the three different cases that we ran are shown below (Chart 9, Chart 8 and Chart 10).

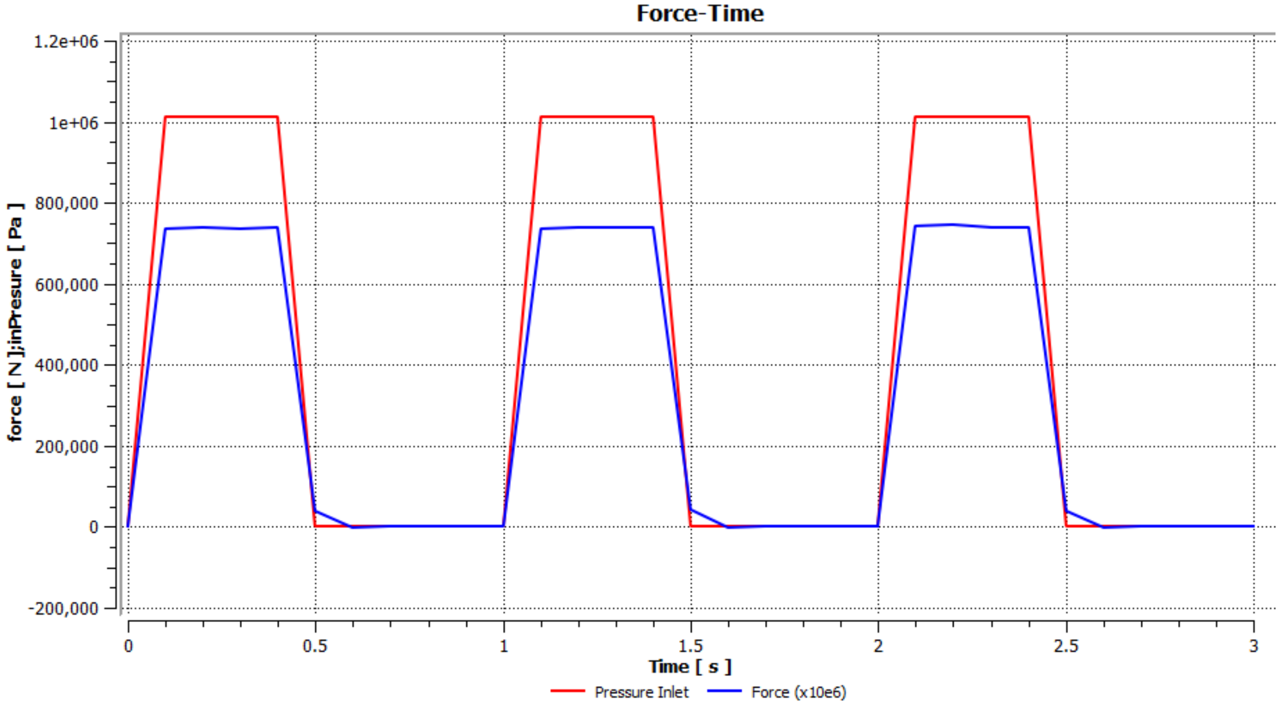


Chart 8: Force - Time (0.5s air / 0.5s rest)

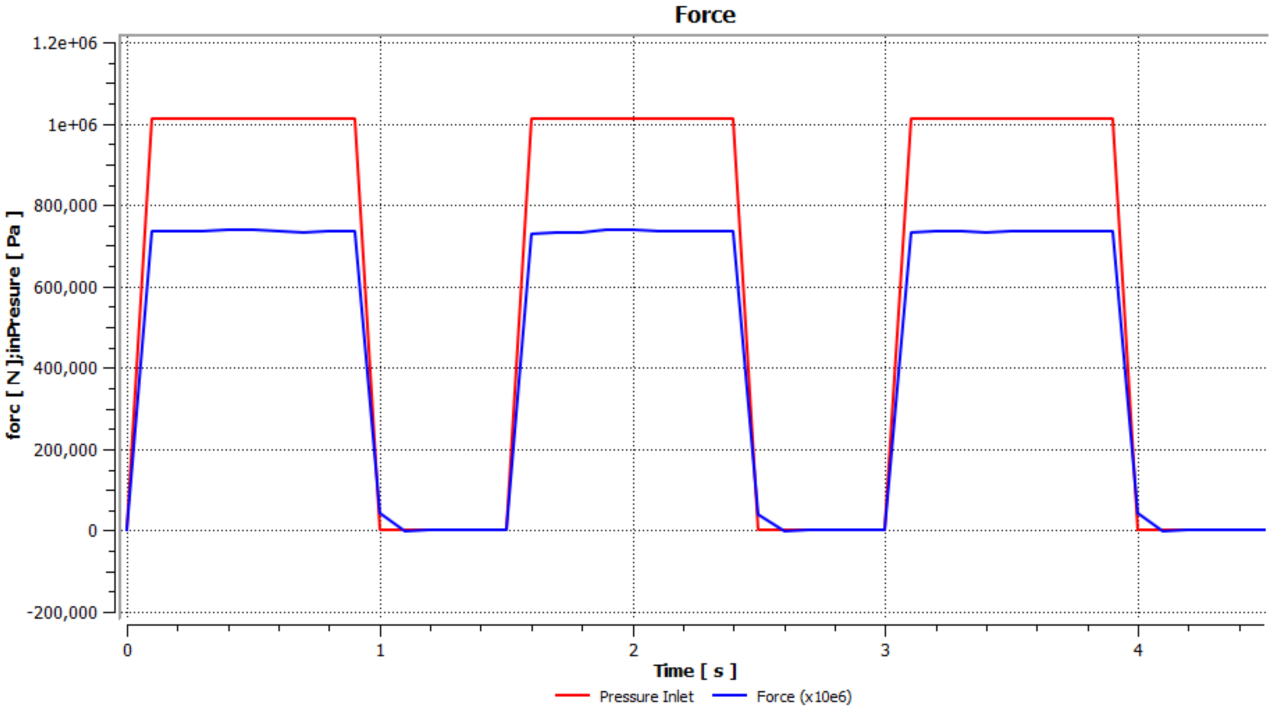


Chart 9: Force - Time (1s air / 0.5s rest)

ERROR! REFERENCE SOURCE NOT FOUND.

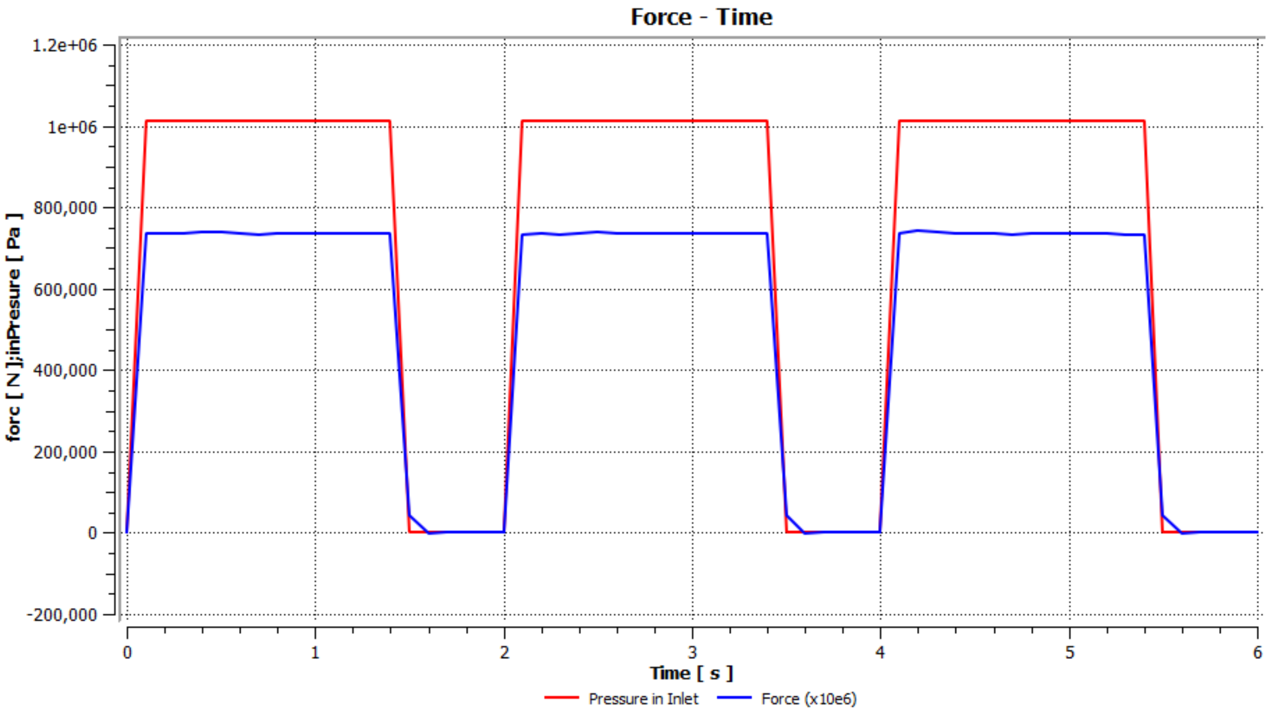


Chart 10: Force - Time (1.5s air / 0.5s rest)

After analyse previous charts, we concluded that time applying airflow and time of rest are not relevant on force exercised in the particle due maximum force was research in a short time and then it went on constant. However, this is not true at all, because although the force was constant we did not considered the movement of the particle in our simulation. Therefore, we should have done another kind of simulation or experiment to establish the minimum time applying airflow to those particles does not fall again on our surface after stop to apply airflow.

Time of work and rest is a very important parameter in the way of working due two significantly reasons: First, this system could not work without breaks because even with a big tank, if it does not stop it would run out of air in a very short time. Secondly, would be a waste of air and energy apply airflow every time if we can get the same results applying only short impulses.

ERROR! REFERENCE SOURCE NOT FOUND.

5 COMPARISON OF DIFFERENT NOZZLES

We had already analysed the behaviour and influence of different parameters in our first nozzle. Then, it was time to compare it with other nozzles. In this case two nozzles more: one a nozzle following the England standard and the other one a nozzle created by Proff. XX trying to improve the previous one.

The most important feature that we was interested in improve was the ration between air in inlet and air in outlet, keeping velocity in outlet equal or higher. It would let us save compressed air, which also means energy and money.

5.1 England Standard (4 holes)

This nozzle has four holes through air from outside enter in the nozzle, increasing the air in the outlet. This hole is very different from the holes in the first nozzle; in this case, they are circular and with an angle of inclination refer to axial axis (instead perpendicular as in the other nozzle).

5.1.1 Geometry, Mesh and Setup

In this case, Autodesk Inventor Geometry's file was given us (Fig. 74). Therefore, the first step was open that file, supress the thread (due it have not influence in our simulation and it would become the geometry more complicated), and finally we exported the geometry in a file compatible with ANSYS, in this case STEP file (.stp).

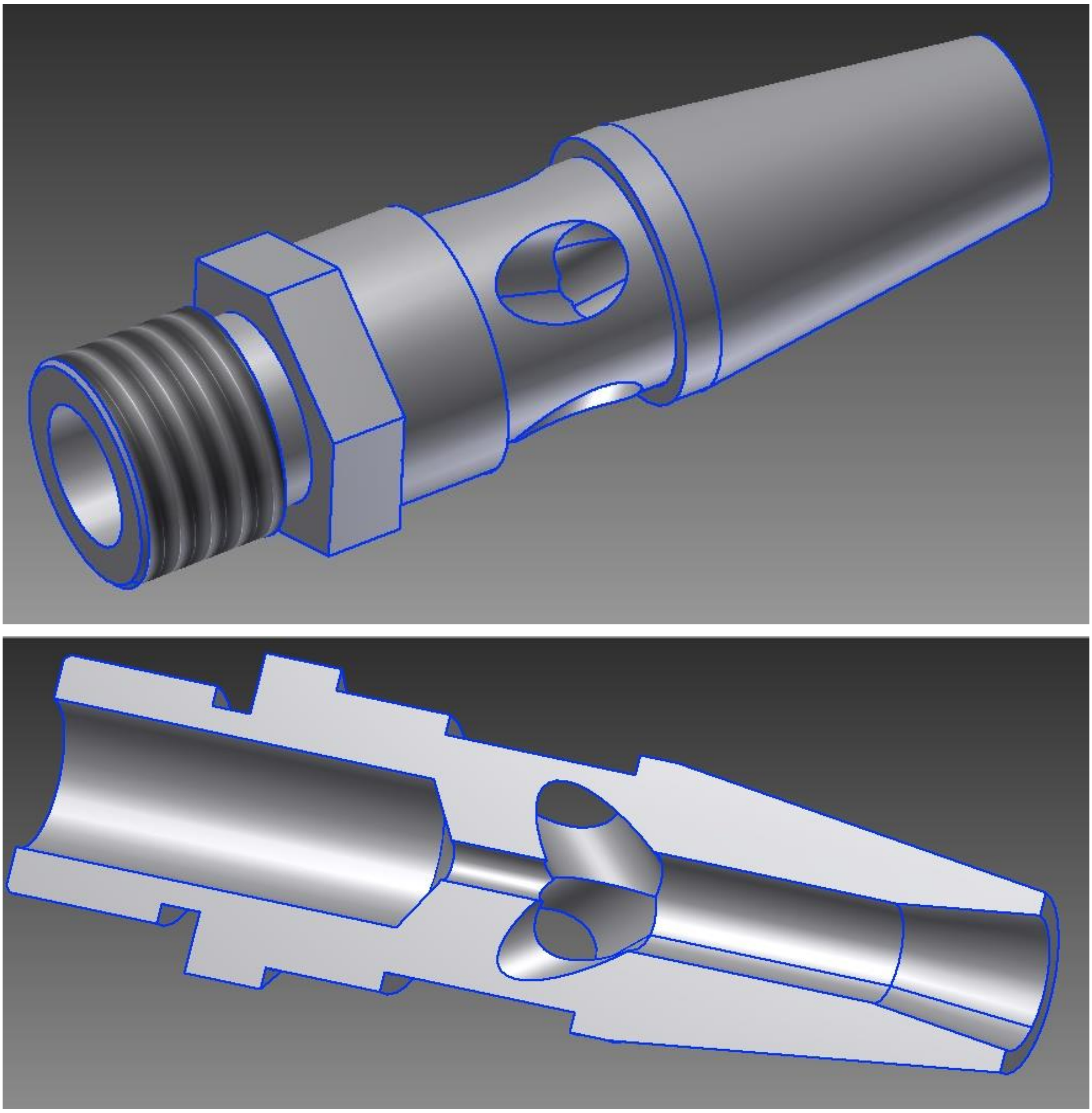


Fig. 74: Geometry of the new nozzle (4 holes)

After that, we followed the same steps that in the previous nozzle: open a new workbench file, drag a Fluid Flow (CFX) analysis system and open Geometry where we imported the file that we had just created in the previous step.

As in the other simulation, we created a cube that surrounded the nozzle with the wall far enough (60x60x30 cm) and then supressed the nozzle from that new body obtaining only one body, which represented our airflow. Thanks to this geometry was symmetrical, as well as the boundary conditions, we could apply symmetry in XZ and YZ planes, working since that moment only with a quarter of the problem.

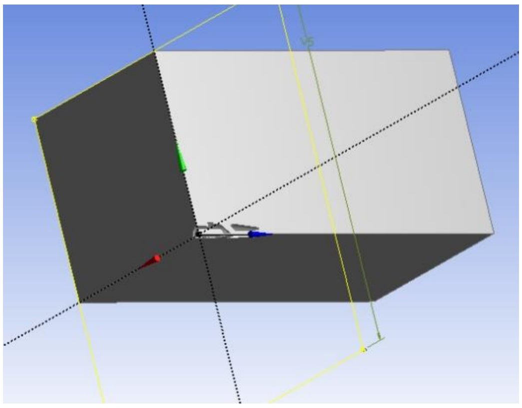


Fig. 75: A quarter of symmetry

ERROR! REFERENCE SOURCE NOT FOUND.

Then, with the aim of be able to define our mesh in a better way, we sliced the geometry in several bodies. First, we cut the original body for the nozzle outer edge to split it in inside and outside bodies. Then, the inner part was spited in three bodies: one from inlet to almost the end of the inlet chamber (we let a few millimetres to the mesh geometry change stand out of a zone where are airflow changes), another from the last one edge to the end of the small hole chamber and the last one was the rest of them (as is shown in Fig. 76).

It was important to form a new part with all these bodies due all of them are the same fluid, so it needs be notice to the program for create the right setups.

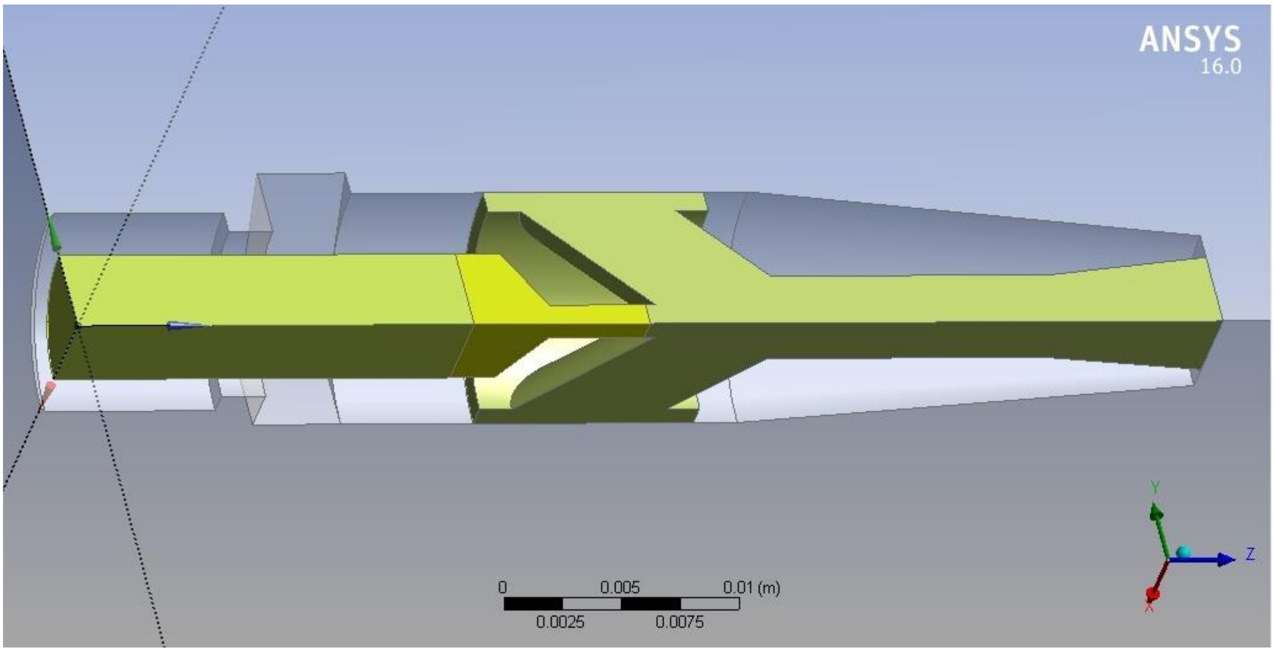


Fig. 76: Geometry sliced

After it, geometry was ready and we could start with the mesh.

In this case we applies two body sizes: one for the inlet and outlet chambers bodies (A in Fig. 77), and another one in small hole body with half the value of element size (D in Fig. 77). It was due the hole chamber had a tiny diameter so it needed a finer mesh to solve properly our simulation.

In addition, we applied a vertex sizing in the outlet of our nozzle with 4.5mm of radio and same element size (C in Fig. 77) and a face sizing in the surface from the outlet airflow come in (B in Fig. 77).

ERROR! REFERENCE SOURCE NOT FOUND.

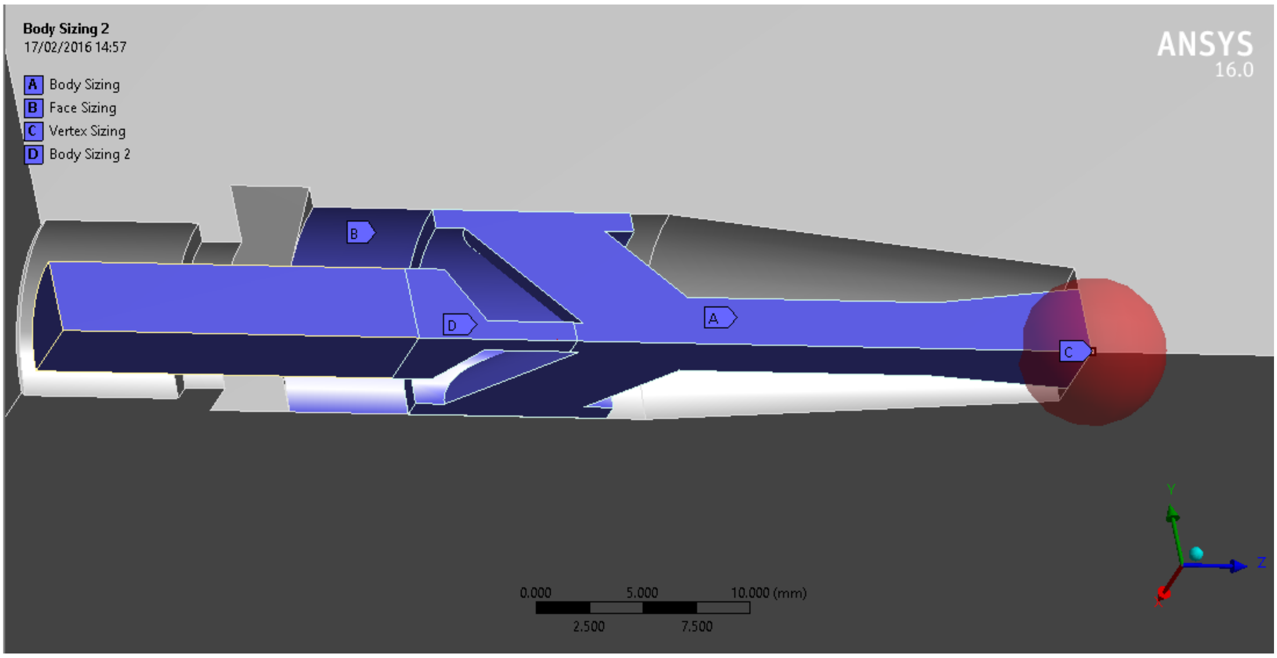


Fig. 77: Sizes on mesh

After that, we created the mesh to watch that all parameters were right and the mesh works well, obtaining the mesh shows in Fig. 78. There we could see that meshes of different bodies were well joined and as we wanted.

ERROR! REFERENCE SOURCE NOT FOUND.

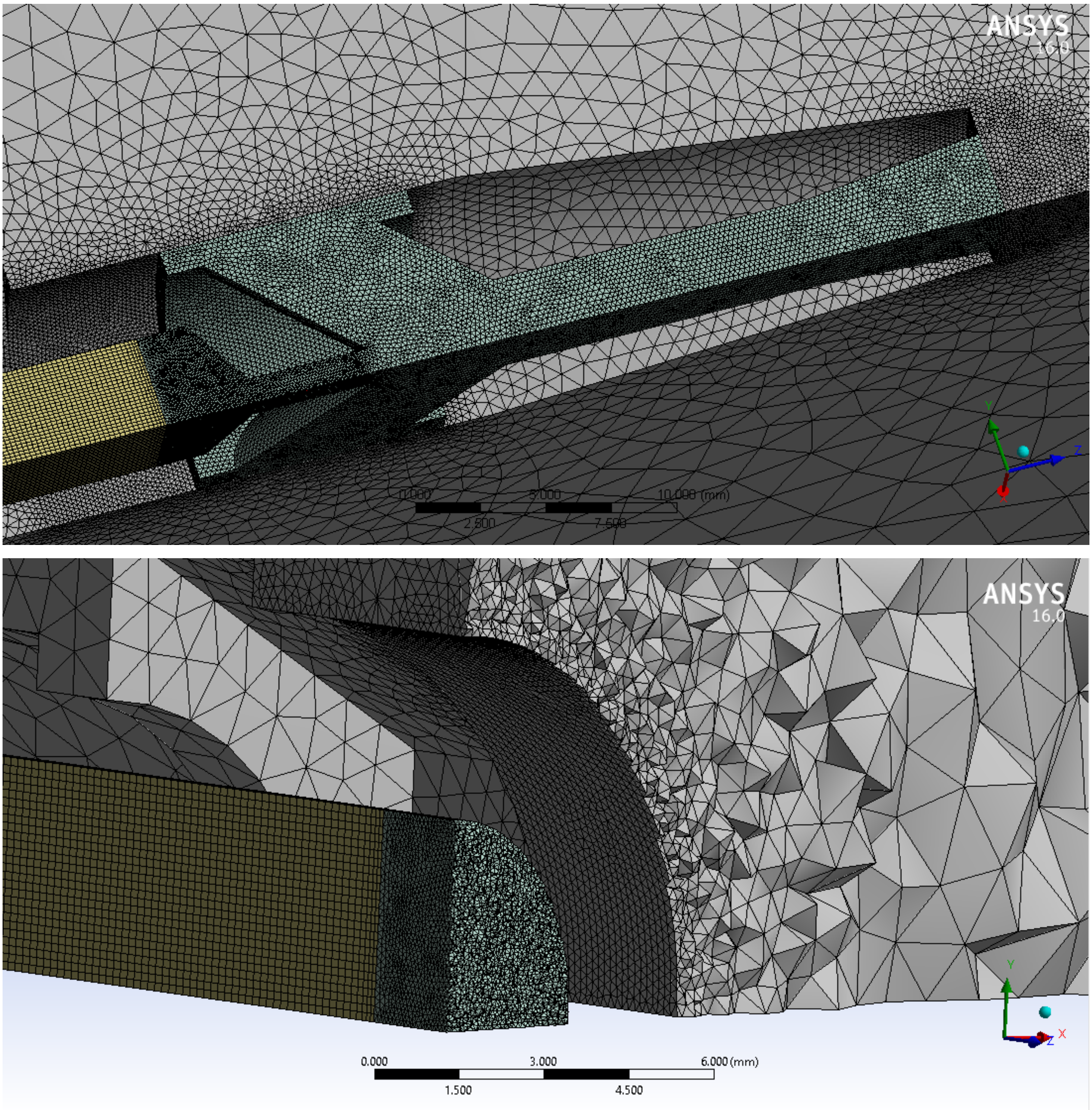


Fig. 78: Mesh generated

Although we watch that the mesh seems good, it is always high recommendable take a look of some quality parameters. For example, there are on very useful in statistics of mesh called “element quality” that show as the following chart:

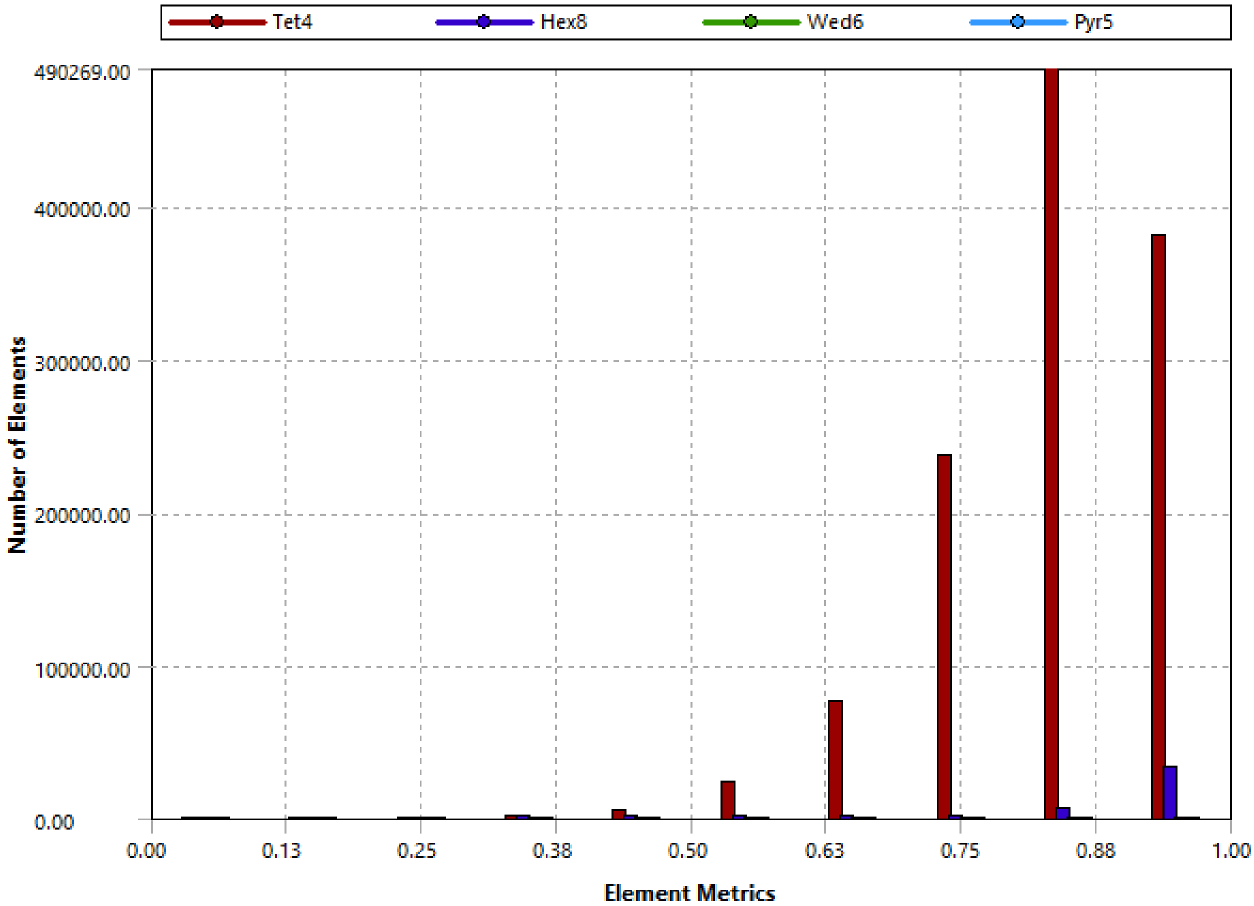


Chart 11: Element Quality (Nozzle 6)

As we can see in Chart 11, mesh quality was quite good (zero is the worst and one is the best) due most of our elements had a quality above 0.75, and, being the highest amount 0.88.

Mesh was not still finished, but to go on with the choosing of the mesh we needed to define setups before.

5.1.2 Setup

To can compare properly the differences between several nozzles, domain setups and boundary conditions must be the same. Therefore, we applied same setups:

Default Domain:

- Material: Air at 25 ºC and Continuous Fluid
- Reference pressure = 1 [atm]
- Non Buoyant and Stationary
- Model Isothermal (25 ºC)
- Turbulence: Shear Stress Transport, with Automatic wall function

Inlet:

- Location: Inlet
- Mass and Momentum: Total Pressure (stable)
- Relative Pressure: “Press”

*”Press” is an expression previously created with 10 [atm] by default.

ERROR! REFERENCE SOURCE NOT FOUND.

• Flow direction: Normal to Boundary Condition
• Turbulence: Medium

Opening:
• Location: Opening
• Mass and Momentum: Opening Pres. and Dirn.
• Relative Pressure: 0 [atm]
• Flow Direction: Normal to Boundary Condition
•Turbulence: Medium

Symmetry:
• Location: Symmetry

5.1.3 Mesh size choice and validation

Until here, all would be ready to run the simulation and obtain the results; but before do that, we need to find an accurate mesh size that give us right results.

Accurate mesh size might be found changing the mesh size until reach the point that making it finer, results almost do not change. The best and fastest way to do it was set the element sizes of the mesh as Input Workbench Parameters, as well as the number of elements as an output parameter.

For another hand, results had to be created (with one interaction was enough) to be able to create the planes where take measures and their output parameters. Once results were created, we opened and inserted several locations. For this case was enough to insert a circular plane in the outlet of our nozzle (planed based in XY plane with offset in Z of 42 mm and a radius of 3.4 mm). Then, calculated average velocity in the locations Inlet and Outlet con copied their equivalent expressions in news expressions called VelInlet and VelOutel, which ones were marked as Workbench Output Parameter.

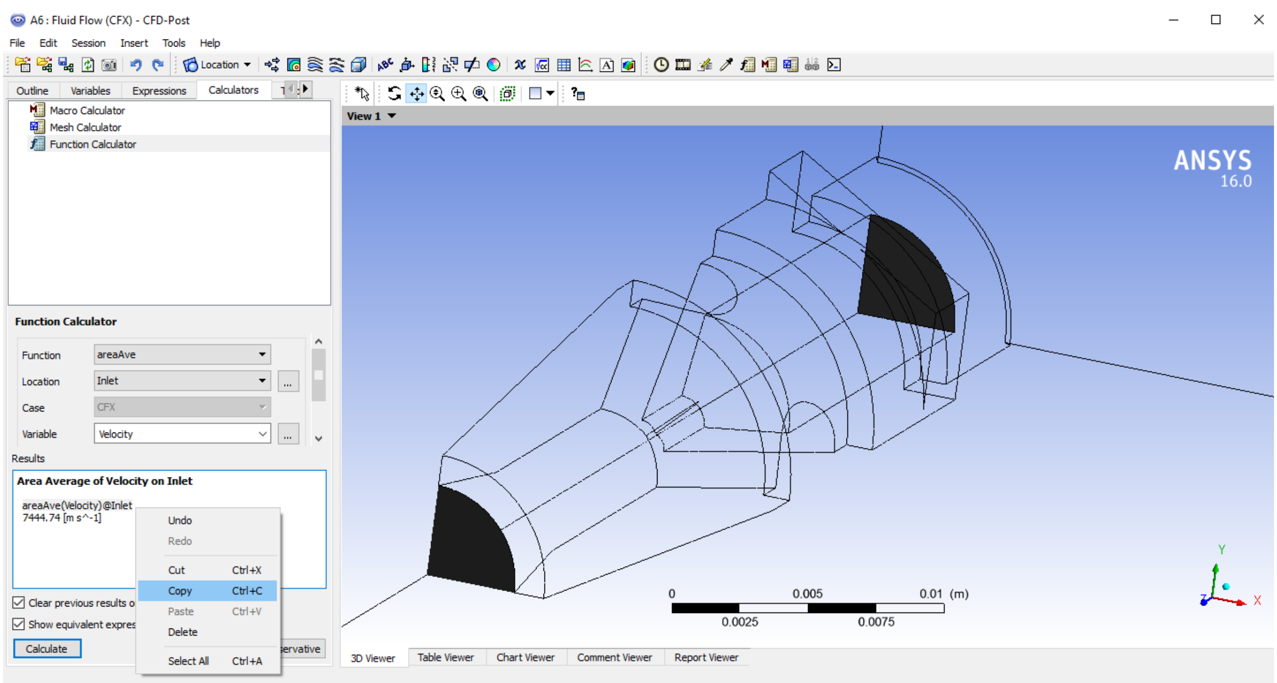


Fig. 79: Inlet and Output planes

ERROR! REFERENCE SOURCE NOT FOUND.

When we were sure that we were working in the right way with the mesh and had all parameters and expressions defined, we finally could look for the proper element size of our mesh. To do that, we applied some different sizes in the parameters previously defined and updated all design points to obtain results (Table 9).

Table 9: Velocities with different mesh element sizes

C	D	E	F	G	H	I
P12 - Body Sizing Element Size	P13 - Face Sizing Element Size	P14 - Vertex Sizing Element Size	P15 - Body Sizing 2 Element Size	P4 - Mesh Elements	P8 - VelInlet	P10 - VelOutlet
mm	mm	mm	mm		m s^-1	m s^-1
0.2	0.2	0.2	0.1	1.2372E+06	83.312	274.63
0.18	0.18	0.18	0.09	1.5765E+06	83.445	274.91
0.16	0.16	0.16	0.08	2.0556E+06	83.907	275.73
0.14	0.14	0.14	0.07	2.8276E+06	83.995	277.05
0.12	0.12	0.12	0.06	4.19E+06	84.171	282.56
0.1	0.1	0.1	0.05	6.7884E+06	84.414	284.55
0.08	0.08	0.08	0.05	1.0598E+07	84.559	286.45

As in a table was not pretty clear how velocity changes with size, we made a chart where compare the velocity obtained in outlet (in inlet it almost do not changed) with the number of elements of our mesh, getting the following Chart 12.

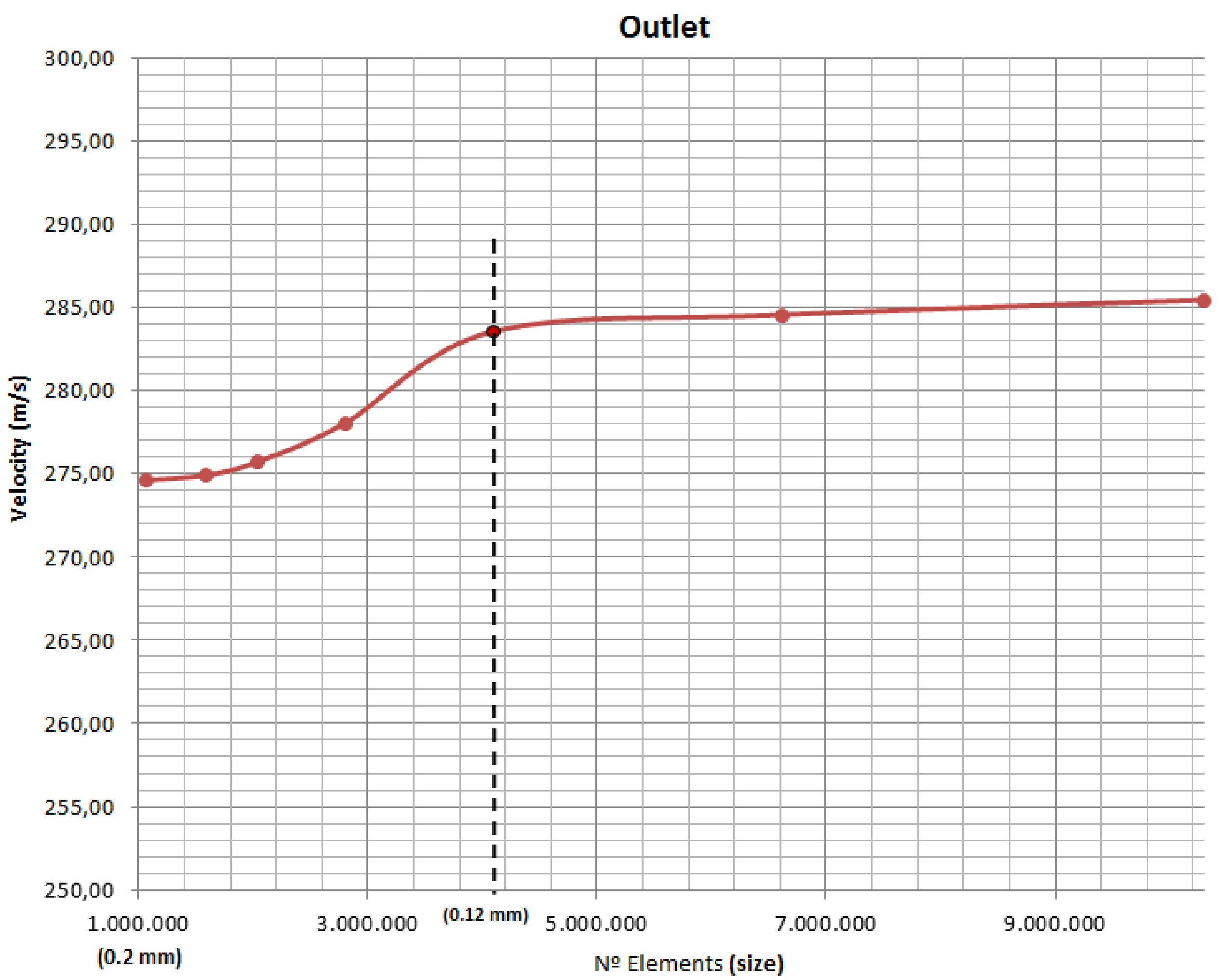


Chart 12: Velocity - Nº elements, Nozzle 4

Here was clear which one was the number of element that had balance between an accurate results and a low number of elements in mesh.

Last step to obtain the final results was applied the size of elements chosen and run the solver, this time until obtain a RSM residuals below $10e^{-4}$.

5.1.4 Validation

Although we had already done a validation in this research, with a new model is always recommendable validate it again because geometry and some parameters have changed.

The experiment was the same performed in the previous case, but this time with the new nozzle 4 and applying 4 bar in the inlet instead 3.5 as in the previous experiment.

Table 10: experiment results, nozzle 4

			Velocity (m/s)	
	Press (bar)	Distance (cm)	Real	Simulation
Nozzle 4 (4 holes)	4	20	16,15	29,32
	4	40	7,95	16,54
	4	60	5,58	10,44
	4	80	4,42	7,81
	4	100	3,96	6,55

ERROR! REFERENCE SOURCE NOT FOUND.

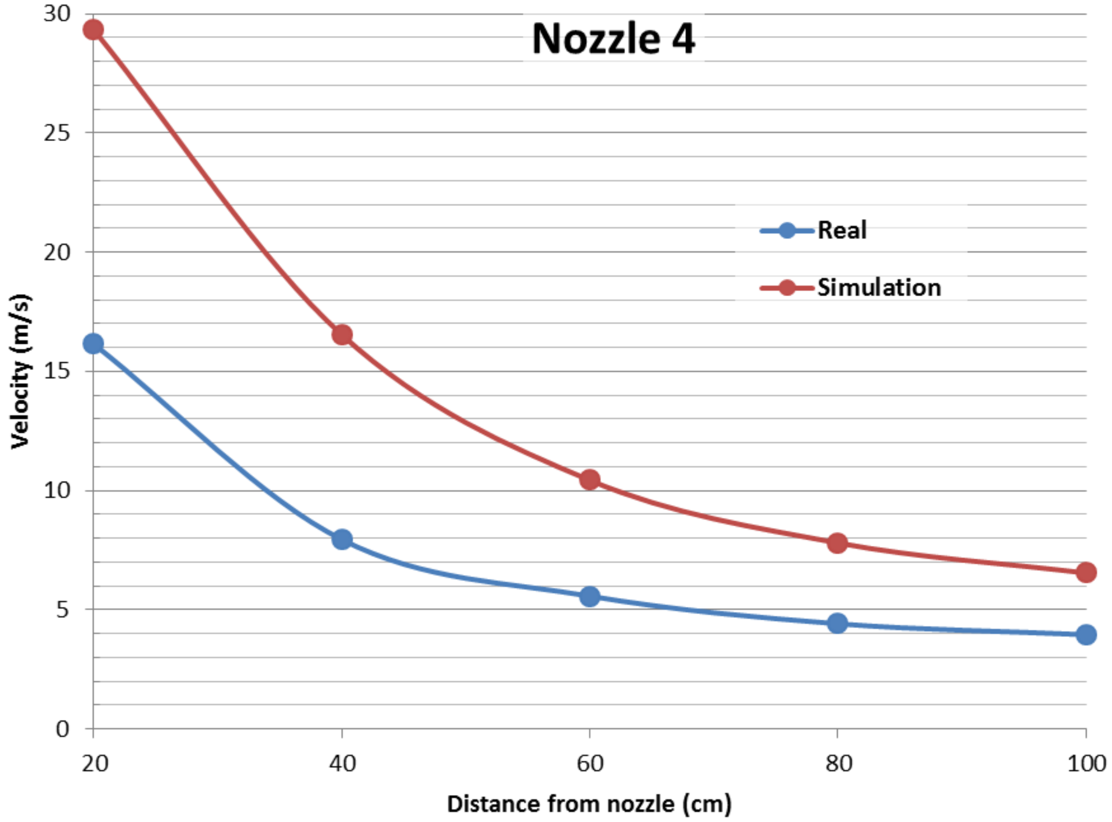


Chart 13: Comparison experiment - simulation results

Table 10 and Chart 13 confirm us that this simulation is suitable, because although figures was not equal, they was close enough and with a very similar behaviour to be able to assert that this simulation can be used to predict the behaviour of our nozzle.

5.1.5 Final Results

In results, we there are two ways to understand how the nozzle works: by plots and by calculations. The most recommended is use both, plots to see the behaviour of fluids and calculator to take measures that help us to know how exactly are that behaviour and can compare with another models or cases.

About plots, we applied some different options to see the results, for example, contours of pressure and velocity (Fig. 80), streamlines or vectors (Fig. 81).

ERROR! REFERENCE SOURCE NOT FOUND.

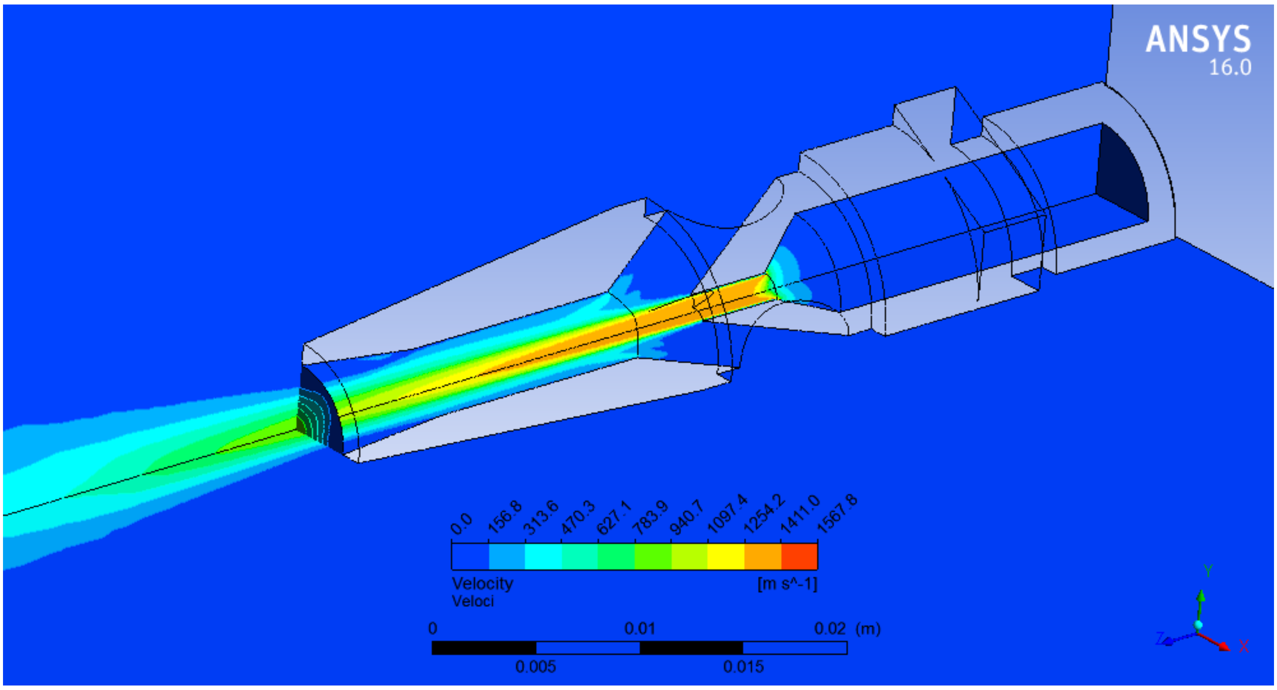


Fig. 80: Velocity, Nozzle 4

In Fig. 80 is shown a typical behaviour of fluids through hosepipes, where velocity is higher in the centre than close to walls. In addition here we could see that the highest velocity was in the hole through airflow go from the inlet chamber to the outlet chamber, and that then it went decreasing.

This contour let us to see how high velocity was, but it did not let us see the direction, so we applied a vector plot (Fig. 81).

ERROR! REFERENCE SOURCE NOT FOUND.

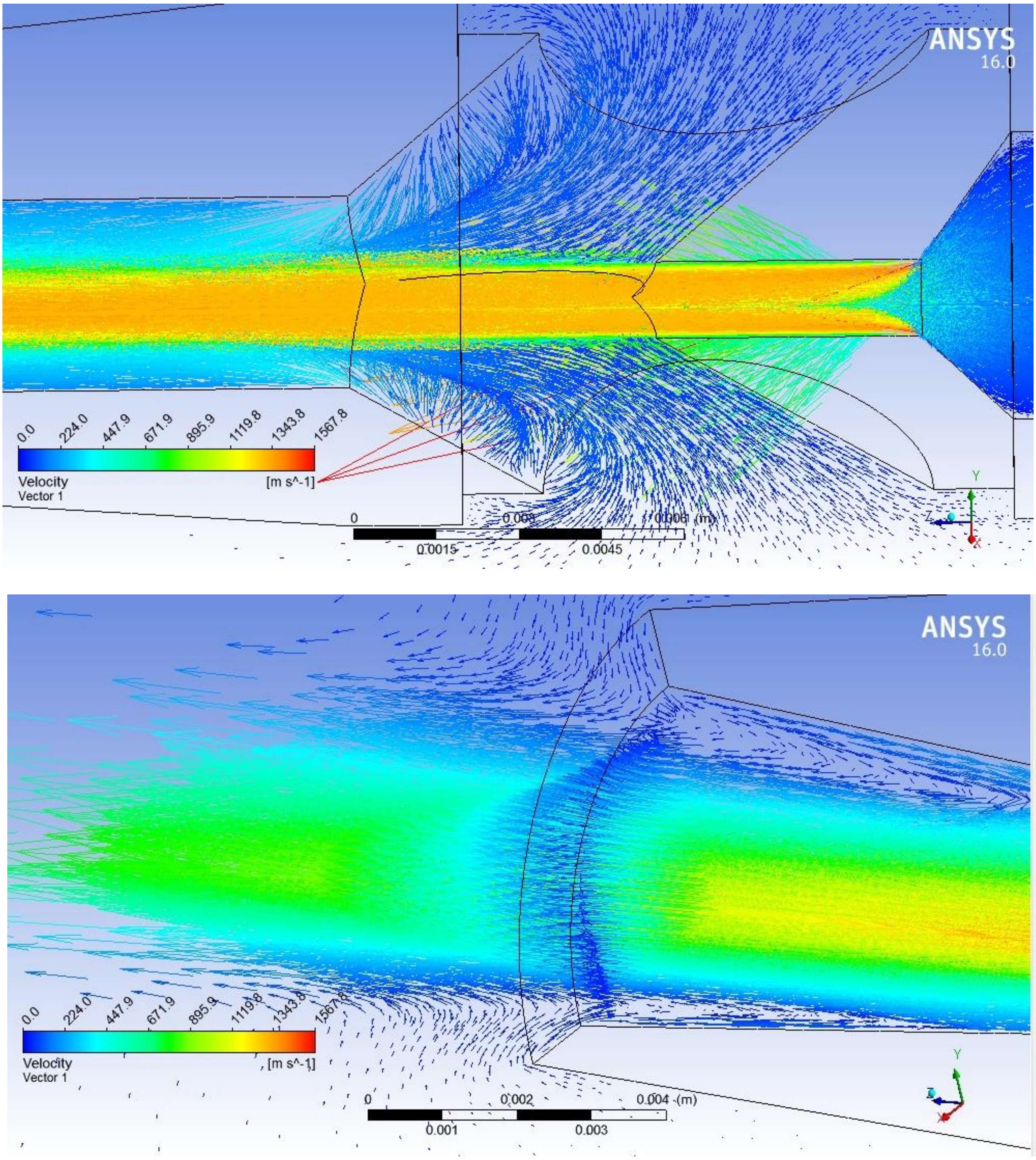


Fig. 81: Zoom of velocity, Nozzle 4

Making zoom in the interesting zones was possible watch the behaviour of airflow in outlet and side inlet.

For another hand, we took some measures in Inlet and Outlet planes. With them, we could calculate important data as fluid flow rate or the ratio between this one on inlet and outlet.

The way to do that was using the function calculator and creating new expressions. Doing that we obtained the following results (see Table 11).

Table 11: Measures taken in Nozzle 4

Name Expresion	Definition	Results	Units
VelInlet	*areaAve(Velocity)@Inlet*	84,17	m/s
VelOutlet	*areaAve(Velocity)@Oulet*	282,56	m/s
QInlet	*area()@Inlet*VelInlet*	0,000930	m^3/s
QOutlet	*area()@Oulet*VelOulet*	0,002565	m^3/s
Qside	*QOutlet -QInlet*	0,001635	m^3/s
Qrel	*QOutlet /QInlet*	2,76	-
QInlet2	*massFlow()@Inlet*	0,001102	Kg/s
QSide2	*QOutlet2-QInlet2*	0,001858	Kg/s
QOutlet2	*massFlow()@Outlet*	0,002959	Kg/s
Qrel2	*QOutlet2/QInlet2*	2,7	-

5.2 Nozzle 6 holes

After realize that with the new nozzle we got higher velocities in outlet with the same pressure in inlet, we thought that maybe a new nozzle with more holes could improve it and obtain even better results.

Starting from the previous nozzle but increasing the number of holes until six PROF. created this new nozzle (Fig. 82).

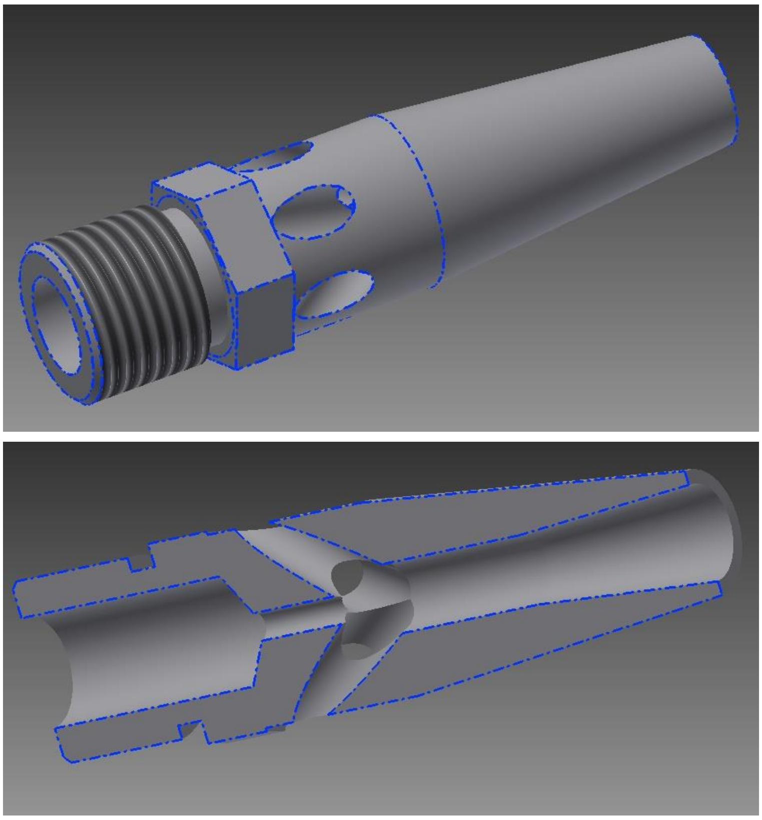


Fig. 82: Nozzle 6 geometry

The geometry had been already given us in an Autodesk Inventor's file. Then, it was saved in a new file compatible with ANSYS for then imports it.

ERROR! REFERENCE SOURCE NOT FOUND.

5.2.1 Geometry and Meshing

Once in ANSYS the new geometry was made as in the other cases; we created as a cube with the walls faraway form our nozzle and subtracted the nozzle body from that cube.

To can make the mesh as we wanted, we used the slice tool to make some cuts in the geometry. First on was around all the edge of the nozzle, separating the airflow inside it from outside one. After that, we divided the inside body in three ones: inlet chamber, outlet chamber and a body between them where is the small hole through airflow go from on to another one.

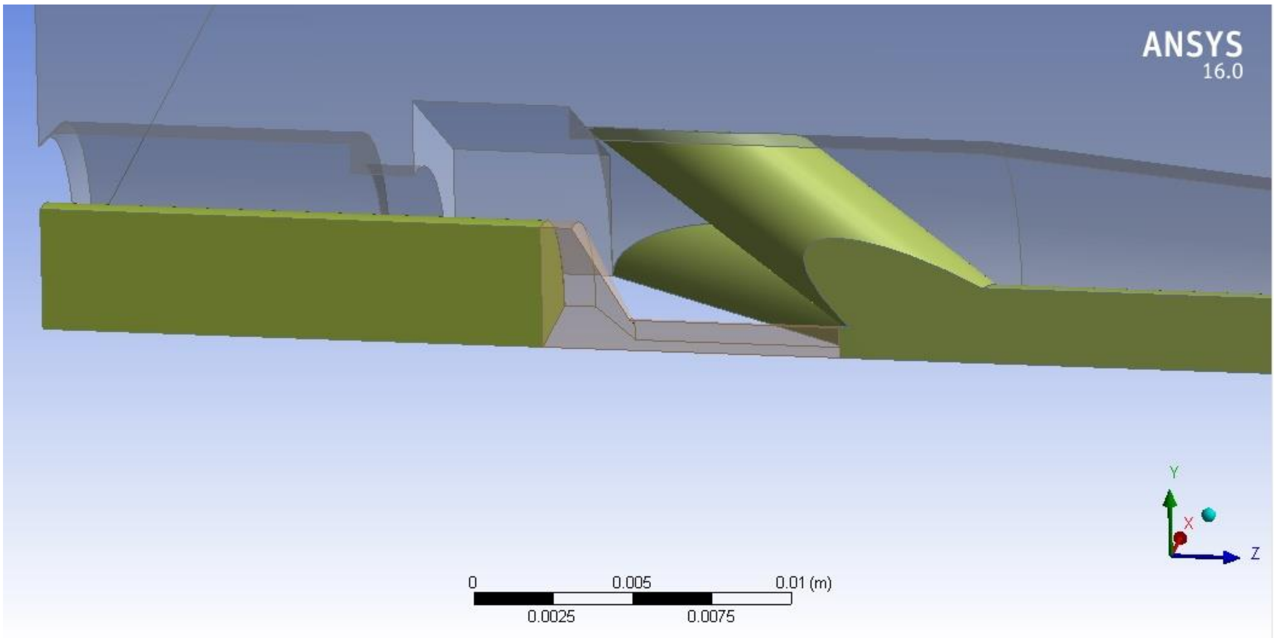


Fig. 83: Body sliced

As Fig. 83 show, the middle body included a little part of the inlet chamber; this was done because it is recommendable do not make changes of mesh geometry in places where flow proprieties are changing (in this case compression).

In addition, we created the Name Selections Inlet, Symmetry and Opening in the same places that in the first case.

Finally, we form a new part with all these bodies due all of them are the same fluid (so, the same part on the simulation).

Mesh was done in a very similar way than in the previous nozzle. We applied again a body sizing in inlet and outlet chamber bodies (“A” in Fig. 84) and other body sizing but this with different element size in the middle chamber body (“D” in Fig. 84). Besides we created a vertex sizing in the outlet of the nozzle with a radius of 4.5mm (“B” in Fig. 84) and finally a face sizing in in the edge of the nozzle from where the outer air go inside it (“C” in Fig. 84).

ERROR! REFERENCE SOURCE NOT FOUND.

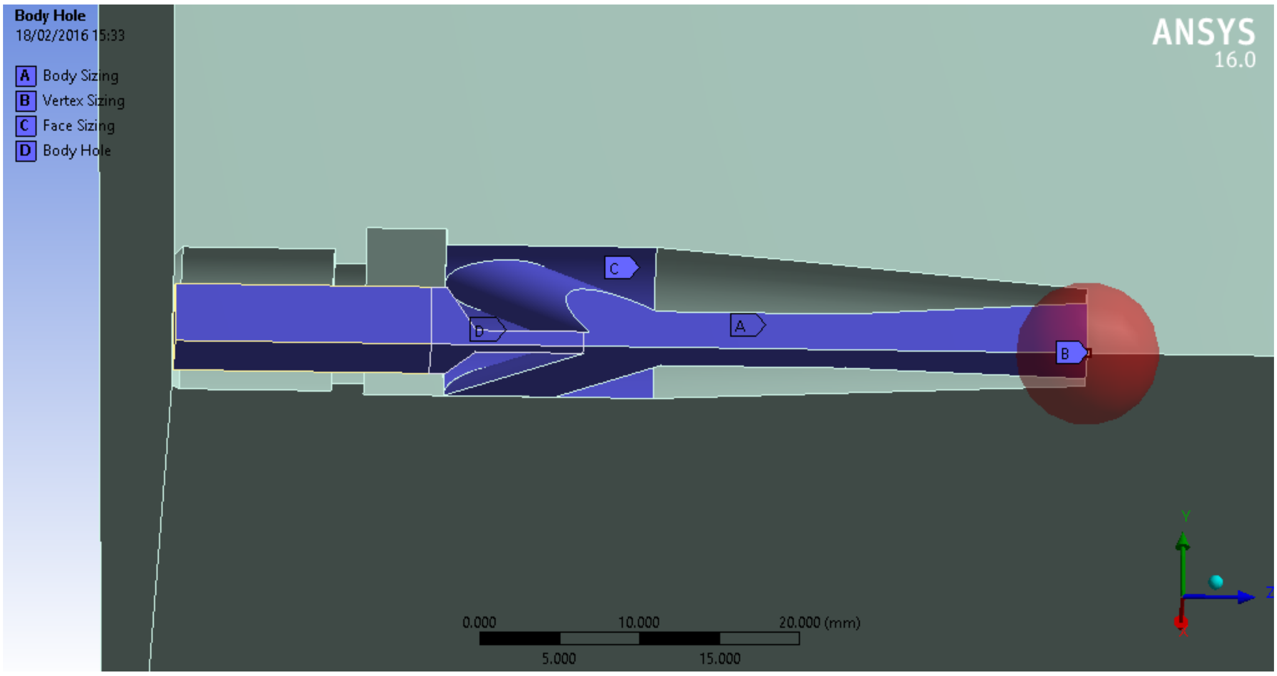


Fig. 84: Mesh sizing (Nozzle 6)

All the element sizes of these sizing were marked as workbench input parameters to be able to do several simulations in only one step and can compare all results together.

After that, we applied a suitable element size in all these parameters and updated the mesh to see if its behaviour was properly and check the mesh quality (Fig. 85).

ERROR! REFERENCE SOURCE NOT FOUND.

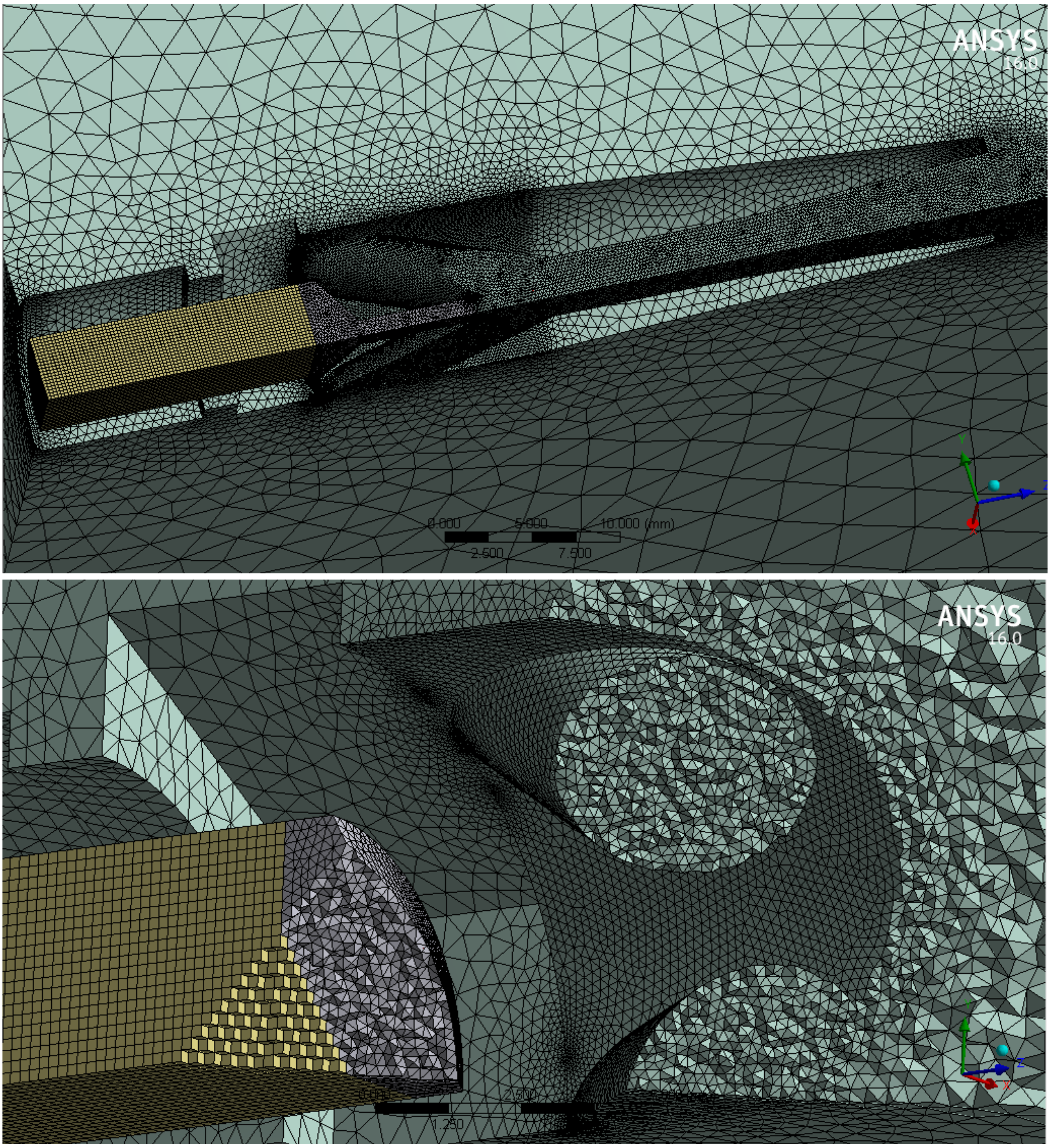


Fig. 85: Mesh Nozzle 6

Taking a look mesh seemed good, but it was not enough and we checked the mesh quality with and additional tool called "element quality" located in mesh statistics.

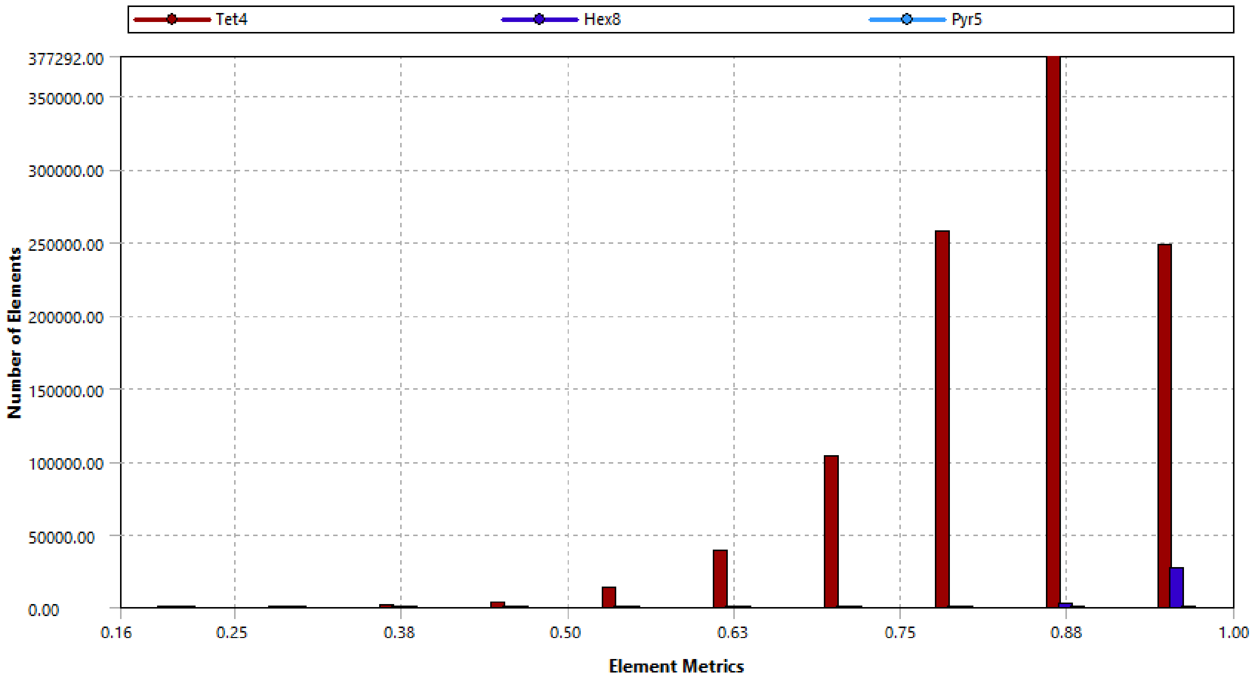


Fig. 86: Mesh quality

As it is show in Fig. 86, quality of this mesh was really good due most of the elements had a high quality (above 0.75) and there were a little amount of them with bad quality (below 0.5).

Next step was to apply setups, which were defined as following:

Default Domain (Fig. 87**):**

• Material: Air at 25 ºC and Continuous Fluid
• Reference pressure = 1 [atm]
• Non Buoyant and Stationary
• Model Isothermal (25 ºC)
• Turbulence: Shear Stress Transport, with Automatic wall function

Inlet:

• Location: Inlet
• Mass and Momentum: Total Pressure (stable)
• Relative Pressure: "Press"

*"Press" is an expression previously created with 10 [atm] by default.

• Flow direction: Normal to Boundary Condition
• Turbulence: Medium

Opening:

• Location: Opening
• Mass and Momentum: Opening Pres. and Dirn.
• Relative Pressure: 0 [atm]
• Flow Direction: Normal to Boundary Condition
•Turbulence: Medium

ERROR! REFERENCE SOURCE NOT FOUND.

Symmetry:

• Location: Symmetry

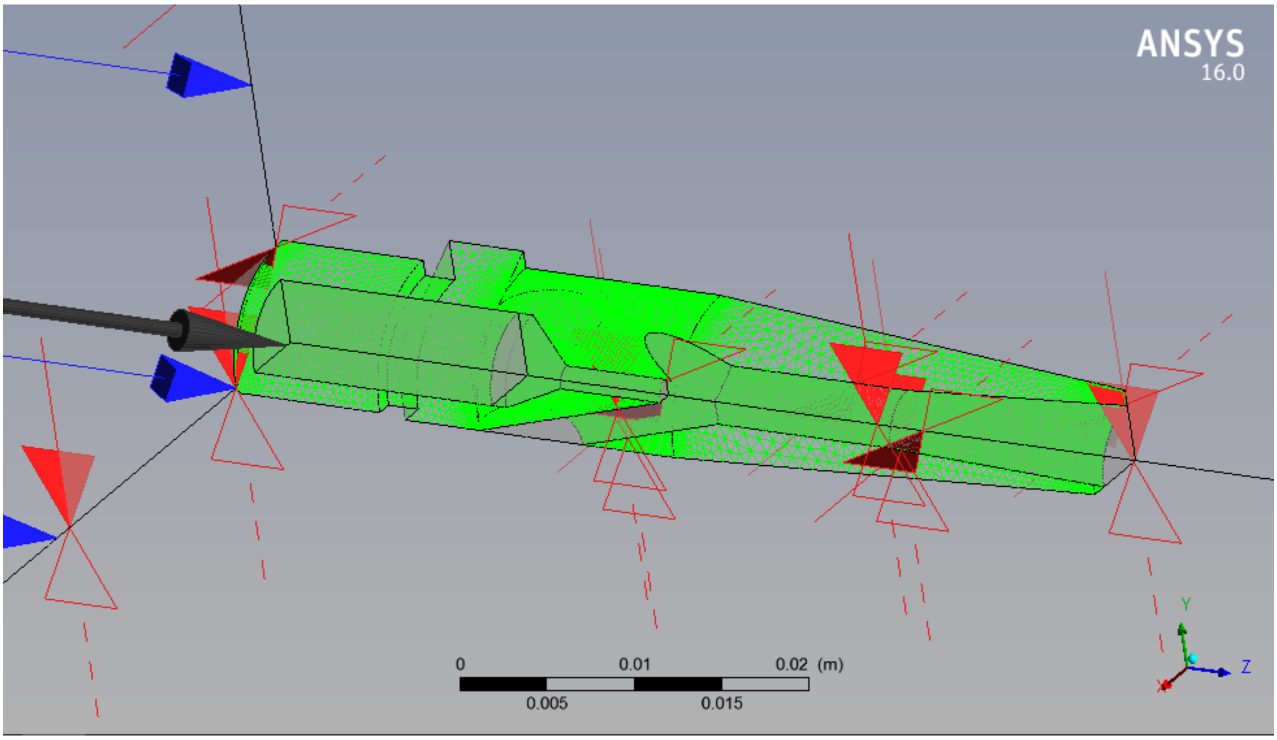


Fig. 87: Default Domain

With the mesh and setups defined, we ran the solver to get any results and could create output parameters.

The parameters those we needed to compare different mesh and choose the right one were velocity in inlet and in outlet. To obtain it was necessary having two planes where take those measures. One was the Inlet named selection already created and another one we had to create it. It was located in the outlet of our nozzle (defined as based in XY plane with an offset of 57 mm and as circular type with a radius of 3.5 mm) as is show in Fig. 88.

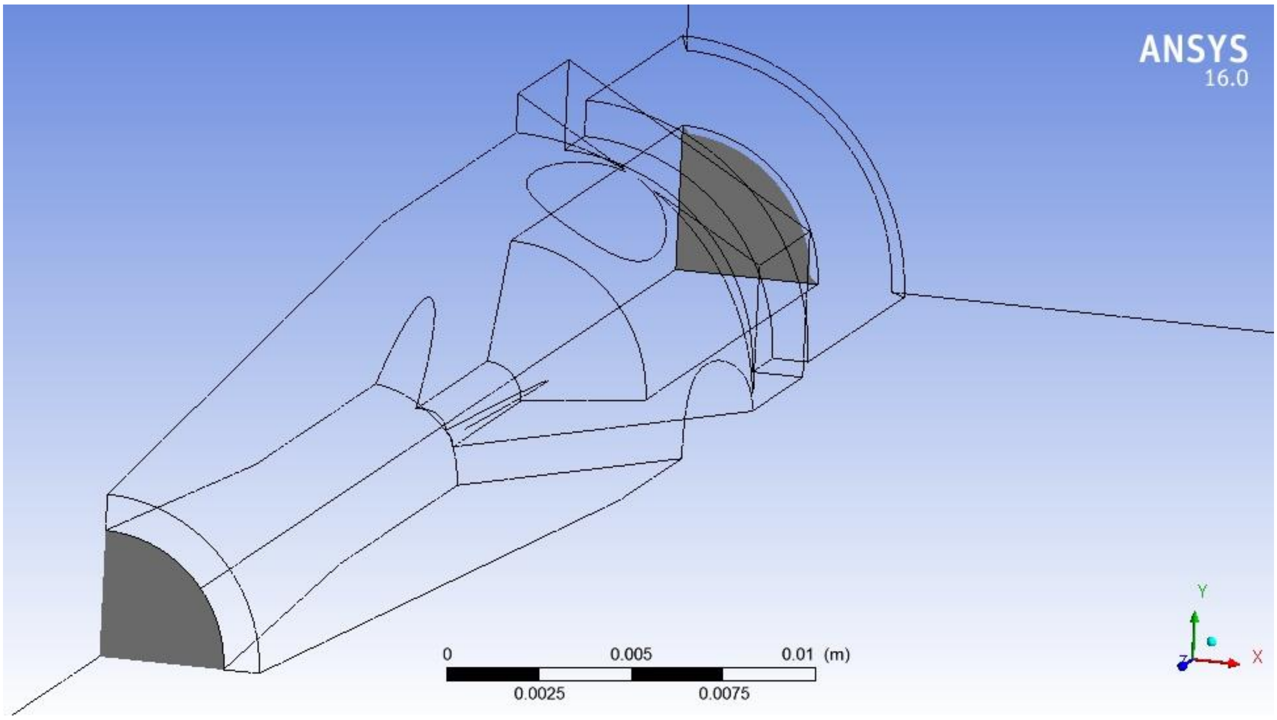


Fig. 88: Inlet and Outlet planes

We measured the velocity in these new planes with the Function Calculator, and copied their equivalent expressions and pasted them in new expressions. Finally, we marked these new expressions as Workbench Output Parameter.

Last step was open Parameters, set up several designed points with different sizes of mesh elements and solve all designed points. We used the following sizes and obtained these results (see Table 12):

Table 12: Parameter Set (Nozzle 6)

C	D	E	F	G	H	I	J
P3 - Body Sizing Element Size	P4 - Vertex Sizing Element Size	P5 - Face Sizing Element Size	P11 - Body Super Element Size	P1 - Mesh Elements	P2 - Mesh Average	P8 - VelInlet	P9 - VelOutlet
mm	mm	mm	mm			m s^-1	m s^-1
0.2	0.2	0.2	0.2	1.0683E+06	0.84588	72.65	269.08
0.2	0.2	0.2	0.1	1.2645E+06	0.83622	73.453	273.25
0.18	0.18	0.18	0.09	1.5875E+06	0.8438	73.472	277.07
0.16	0.16	0.16	0.08	2.0398E+06	0.84453	73.672	280.08
0.14	0.14	0.14	0.07	2.7997E+06	0.84671	73.798	285.28
0.12	0.12	0.12	0.06	4.0987E+06	0.8496	73.852	289.08
0.1	0.1	0.1	0.05	6.6136E+06	0.85112	73.953	291.91
0.08	0.08	0.08	0.05	1.0292E+07	0.85331	74.204	290.16

With the aim of seeing results become easier, we created a chart (Chart 14) where watch clearly how velocity in outlet change along the different element sizes (we skipped velocity in inlet because it almost did not change).

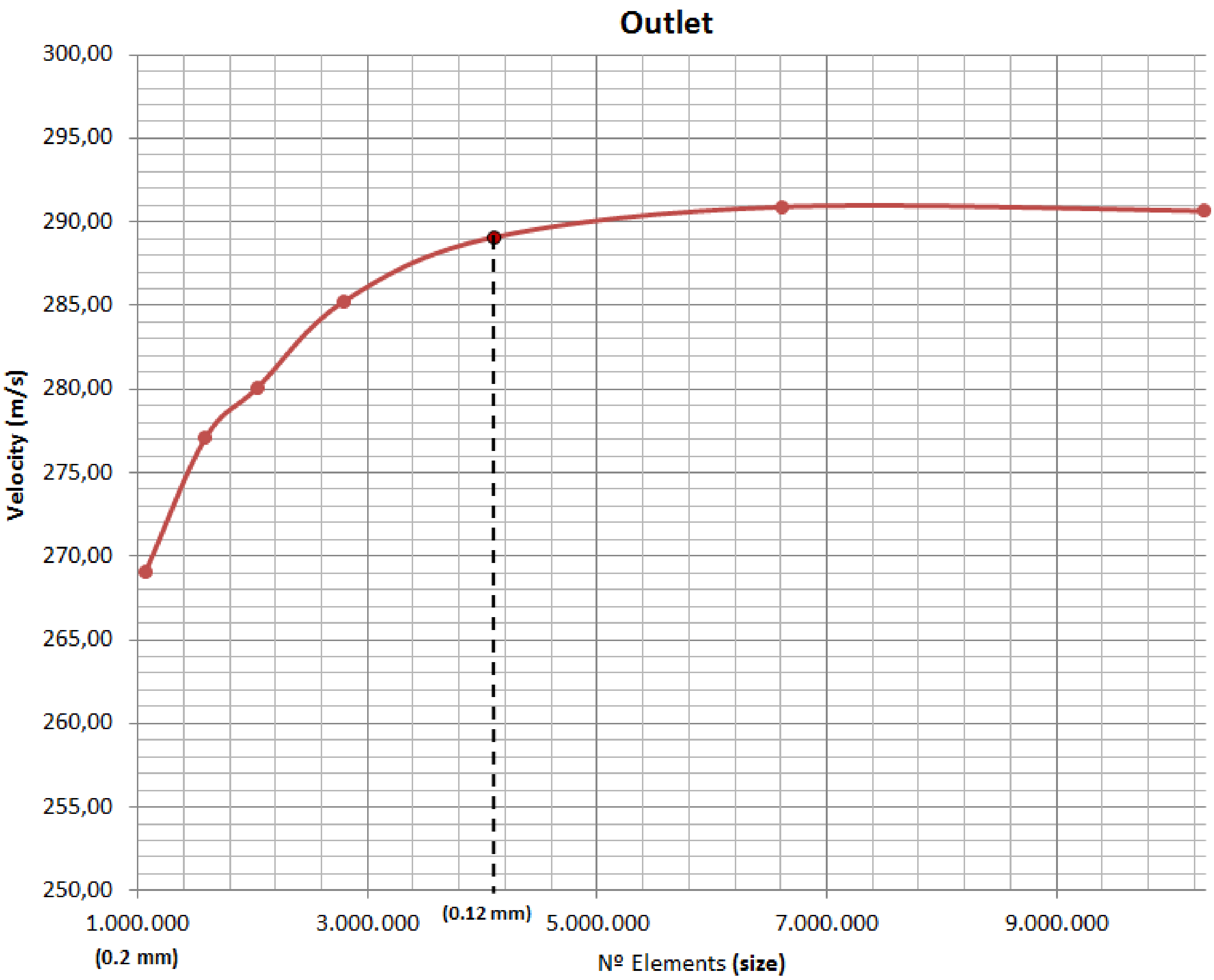


Chart 14: Velocity - Nº elements in Nozzle 6

According with this chart, we finally chose 0.12 as the properly size of the biggest bodies (0.6 to body between both chambers) because it was the point where was the balance between accurate results and reduced number of elements.

5.2.2 Validation

If the shape and features of our model did not change too much, we usually do not need to a validation again if we use the same kind of mesh and setups. This is very helpful due sometimes we have not the new model because we are still designing it; for example is we are trying to improve a device (as in our case) we want to make several models and check in ANSYS which is better before build it. In these cases, making a validation of the previous model (which we want to improve) must be enough because design is similar enough to assert that the simulation will be valid.

The way to make the experiment was the same: with the new nozzle in the air gun, we blew the compressed air for a while measuring the velocity in different distances from the nozzle. All experiments were recorded to be able watch it carefully and take measures.

After watch all video records, we noticed that the pressure more clear to take measures of velocity was 4bar, so we took that measure in the different videos and created a table. In this table we added the measures that were taken in the ANSYS simulation with the new nozzle and 4 bar of pressure (Table 13).

Table 13: Experiment results, Nozzle 6

	Press (bar)	Distance (cm)	Velocity (m/s) Real	Velocity (m/s) Simulation
Nozzle 6 (6 holes)	4	20	xxxx	31,62
	4	40	15,39	14,60
	4	60	11,17	8,75
	4	80	8,97	6,07
	4	100	7,26	5,47

In addition, a chart with this data was created too with the aim of make easier see the results (Chart 15)

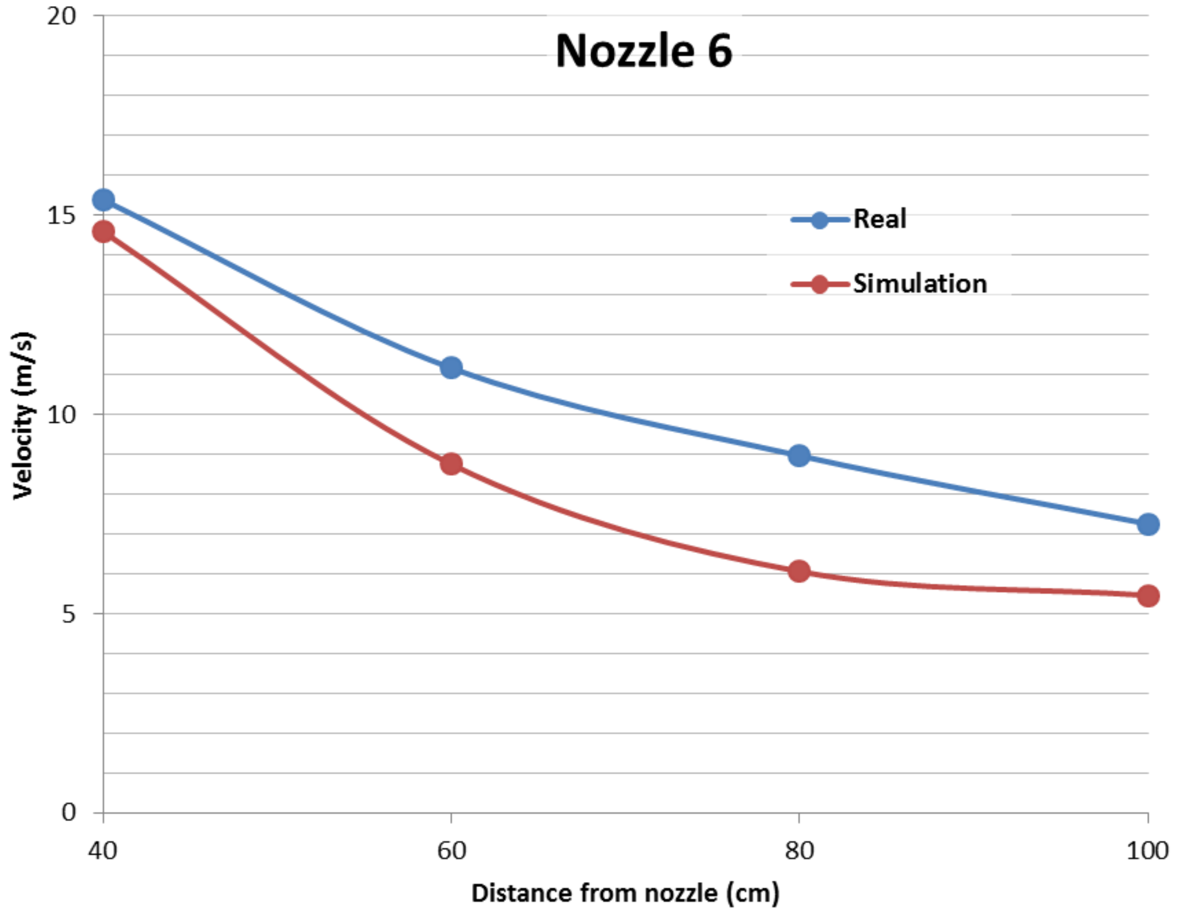


Chart 15: Velocity - Distance from nozzle, Nozzle 6

As we hoped, velocities and behaviour according distance were quite similar between both (real experiment and simulated experiment). Therefore, we could conclude that this simulation was accurate enough to be trusted and to considerate right its results.

ERROR! REFERENCE SOURCE NOT FOUND.

5.2.3 Final results

Once again, we created some plots where see the behaviour of our fluid flow.

As geometry of the nozzle was very similar, results were similar too in the way of airflow and its behaviour, but with some differences in the data.

Airflow had higher velocity in the centre than in the walls, and decreasing from the beginning of the outlet chamber (where is the highest velocity) to the end of the outlet (Fig. 89).

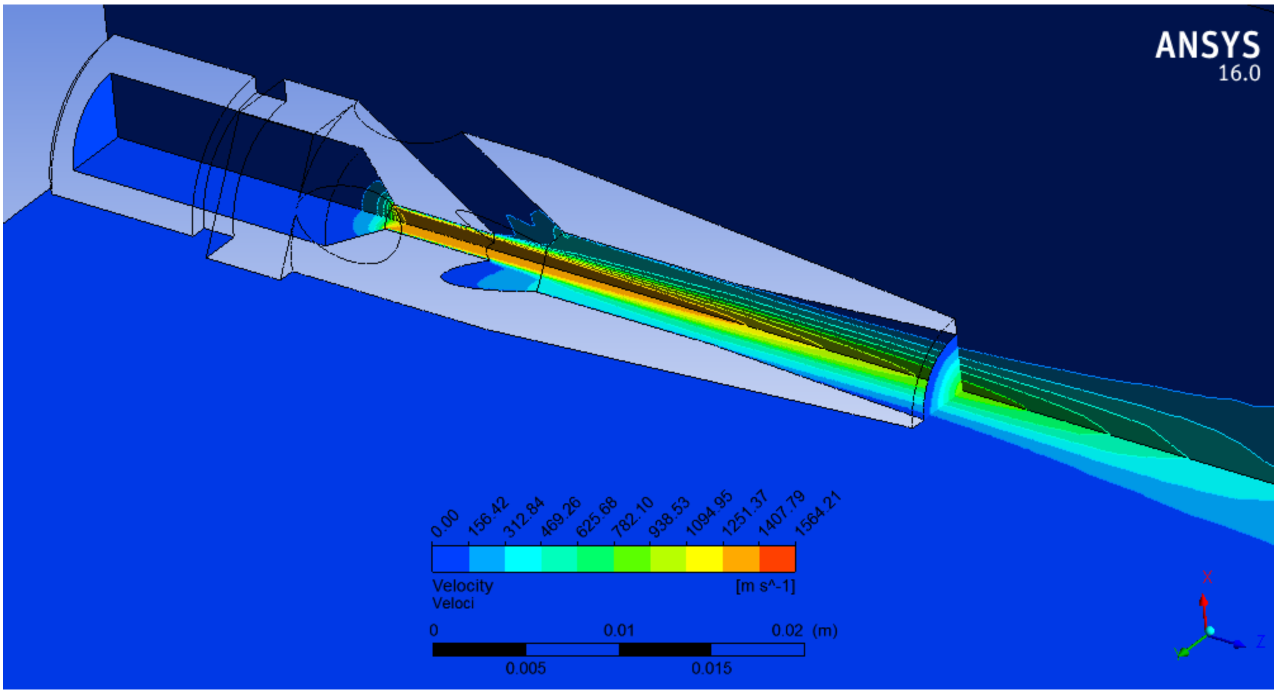


Fig. 89: Velocity contour, Nozzle 6

The same happened with the behaviour of the airflow in side inlets and in outlet due geometry is almost the same, only changed the number of holes. It can be easily seen in Fig. 90.

ERROR! REFERENCE SOURCE NOT FOUND.

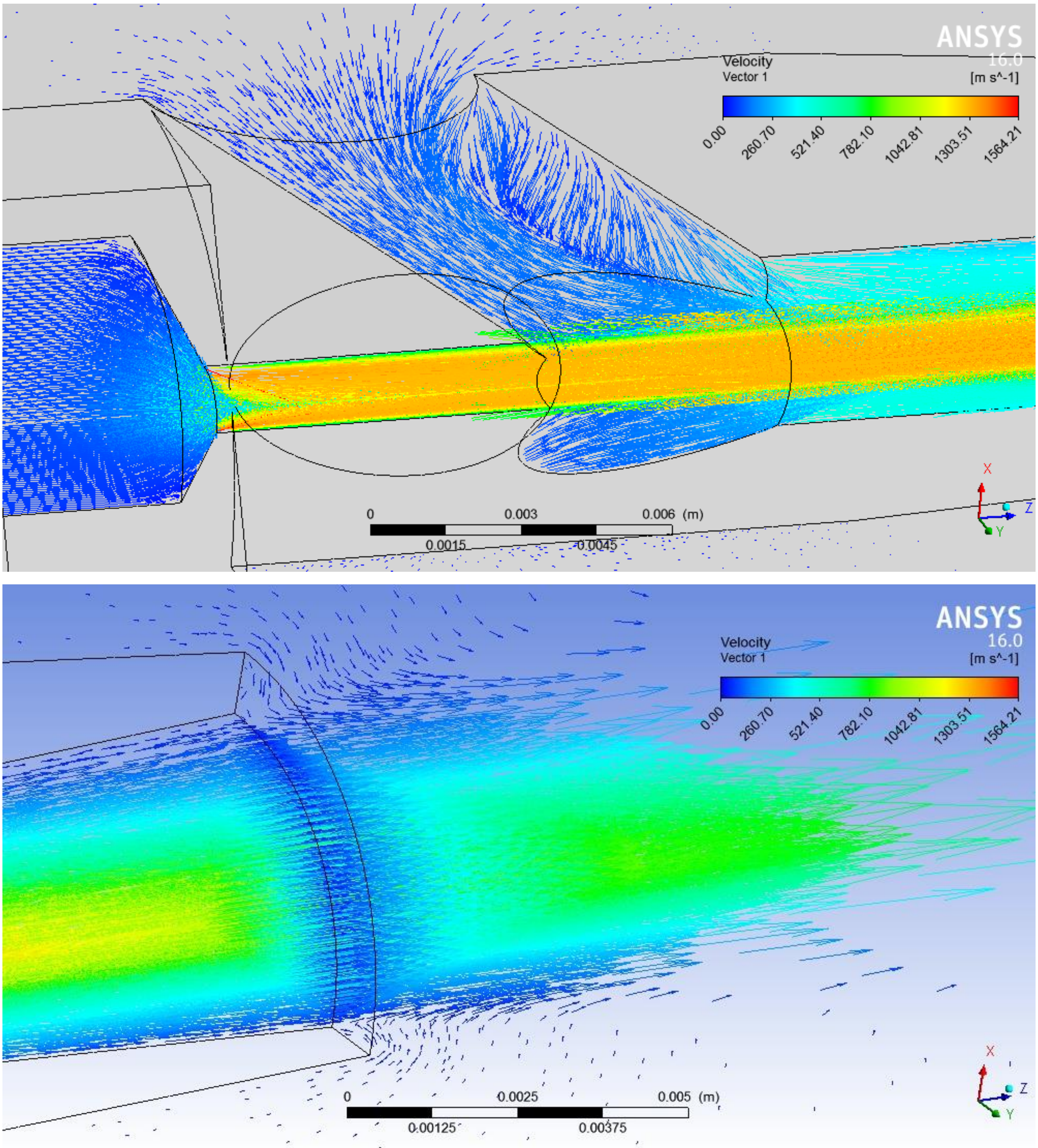
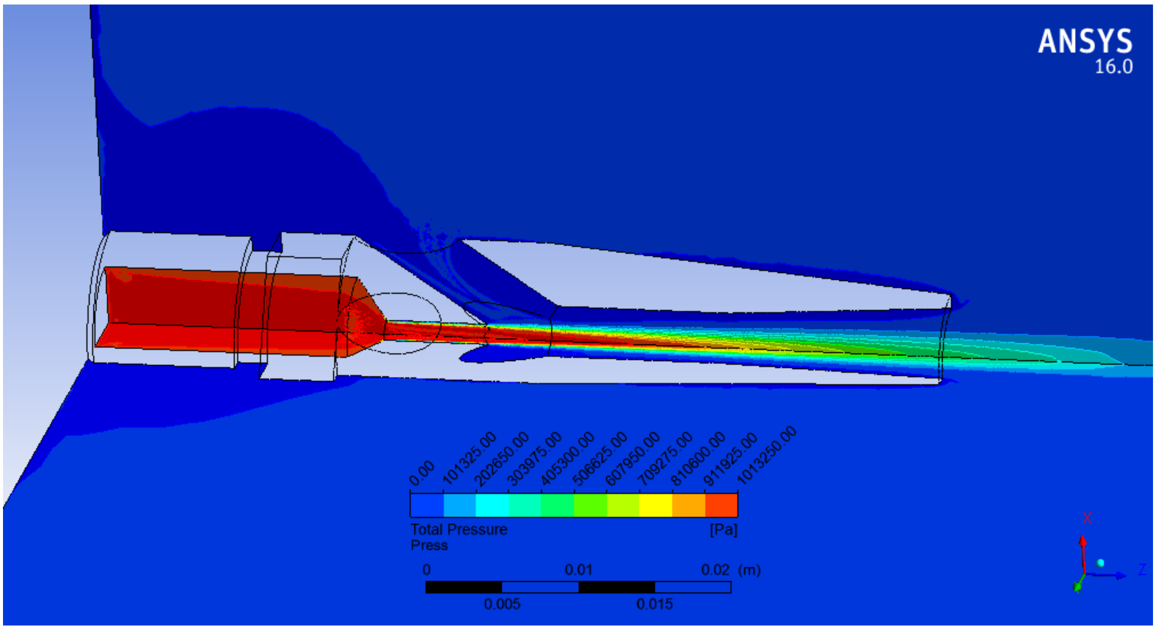


Fig. 90: Velocity vectors in inlet and outlet, Nozzle 6

ERROR! REFERENCE SOURCE NOT FOUND.



By another hand, we took some measures that could help us to realize that what change from the nozzle with four holes. It was done using the calculator in placed CFX-Post and creating new equations that helped us to keep the measures and make other ones easily.

All the data obtained were kept in the following table (Table 14)

Table 14: Measures taken in Nozzle 6

Name Expresion	Definition	Results	Units
VelInlet	*areaAve(Velocity)@Inlet*	73,89	m/s
VelOutlet	*areaAve(Velocity)@Oulet*	291,97	m/s
QInlet	*area()@Inlet*VelInlet*	0,000928	m^3/s
QOutlet	*area()@Oulet*VelOulet*	0,002809	m^3/s
Qside	*QOutlet -QInlet*	0,001880	m^3/s
Qrel	*QOutlet /QInlet*	3,0	-
QInlet2	*massFlow()@Inlet*	0,001100	Kg/s
QSide2	*QOutlet2-QInlet2*	0,002227	Kg/s
QOutlet2	*massFlow()@Outlet*	0,003327	Kg/s
Qrel2	*QOutlet2/QInlet2*	3,0	-

ERROR! REFERENCE SOURCE NOT FOUND.

5.3 Comparison nozzles

Until here we have studied each nozzle deeply, but we wanted to know how are better for our case. The best way to do it was comparing all data obtained in the simulations from different nozzles.

First at all we put all of them in a table (Table 15) and then with these data we could create some charts (Chart 16 and Chart 17) where see the behaviour of different nozzle clearly.

Table 15: Comparison

	Nozzle 2	Nozzle 4	Nozzle 6
Vel Inlet (m/s)	175,51	84,17	73,89
Vel Outlet (m/s)	172,98	282,56	291,97
Q Inlet (m3/s)	0,000793	0,000930	0,000928
Q Outlet (m3/s)	0,003423	0,002565	0,002809
Qdif (m3/s)	0,002630	0,001635	0,001880
Rel Q. in volume	4,3	2,76	3,0
Q Inlet (Kg/s)	0,000940	0,001102	0,001100
Q Side (Kg/s)	0,003130	0,001858	0,002227
Q Outlet (Kg/s)	0,004069	0,002959	0,003327
Rel Q. in mass	4,3	2,7	3,0

In this table, it should be noted that the ratio between air that enter from main inlet and air that enter from side holes is quite higher in the first nozzle over others. This is due to the fact that the holes in the nozzle with two holes are much bigger than in the other ones letting that a bigger amount of air go in the nozzle form sides. About the same data, we can see that increasing number of holes we got improve this parameter.

Others result were easier to understand watching them in charts.

ERROR! REFERENCE SOURCE NOT FOUND.

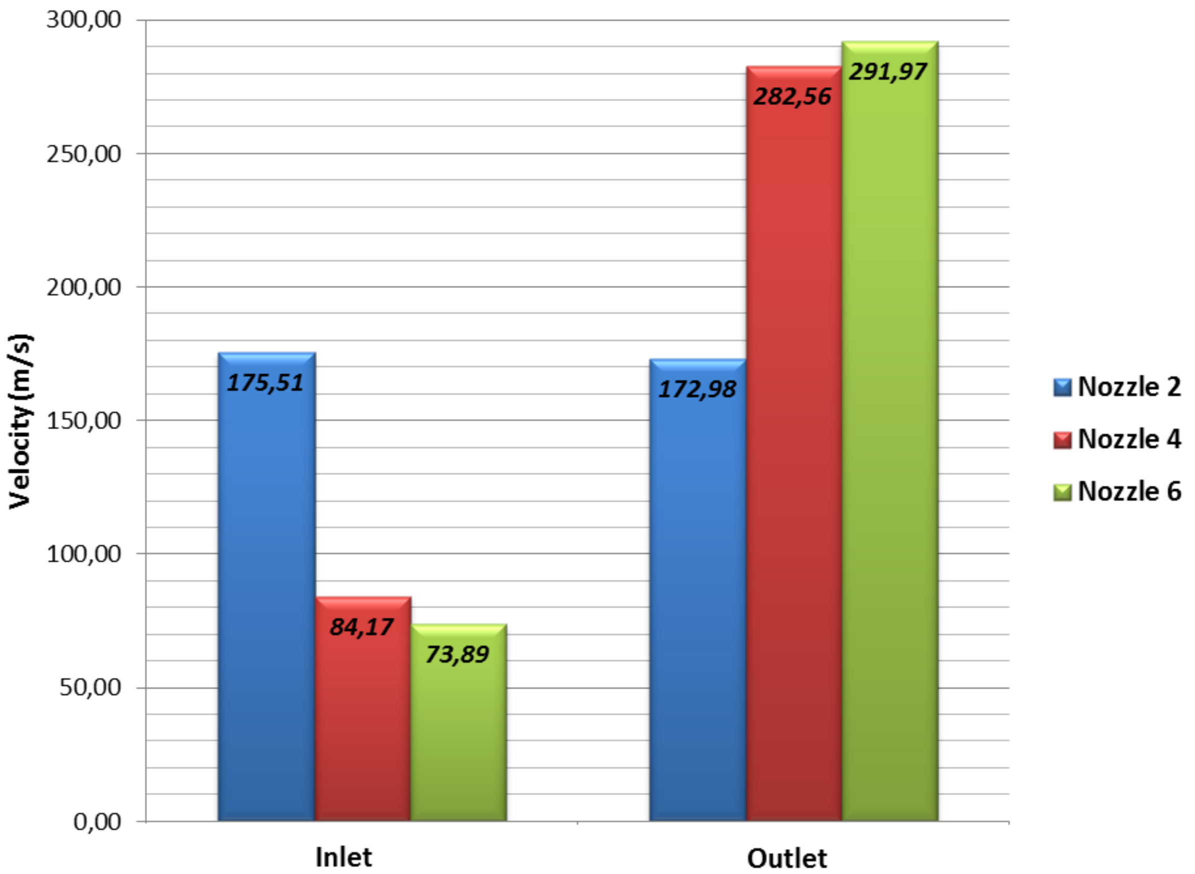


Chart 16: Velocities

About velocities, there were two main conclusions. First, the nozzle with two holes researches a higher velocity in inlet but lower in outlet than other nozzles. This is a bad feature because we want to have the highest velocity possible in the outlet, which means a higher pressure in particles.

Second, with a lower velocity in inlet, the nozzle with six holes researched higher velocity in inlet than other ones; therefore, behaviour improved increasing the number of holes.

ERROR! REFERENCE SOURCE NOT FOUND.

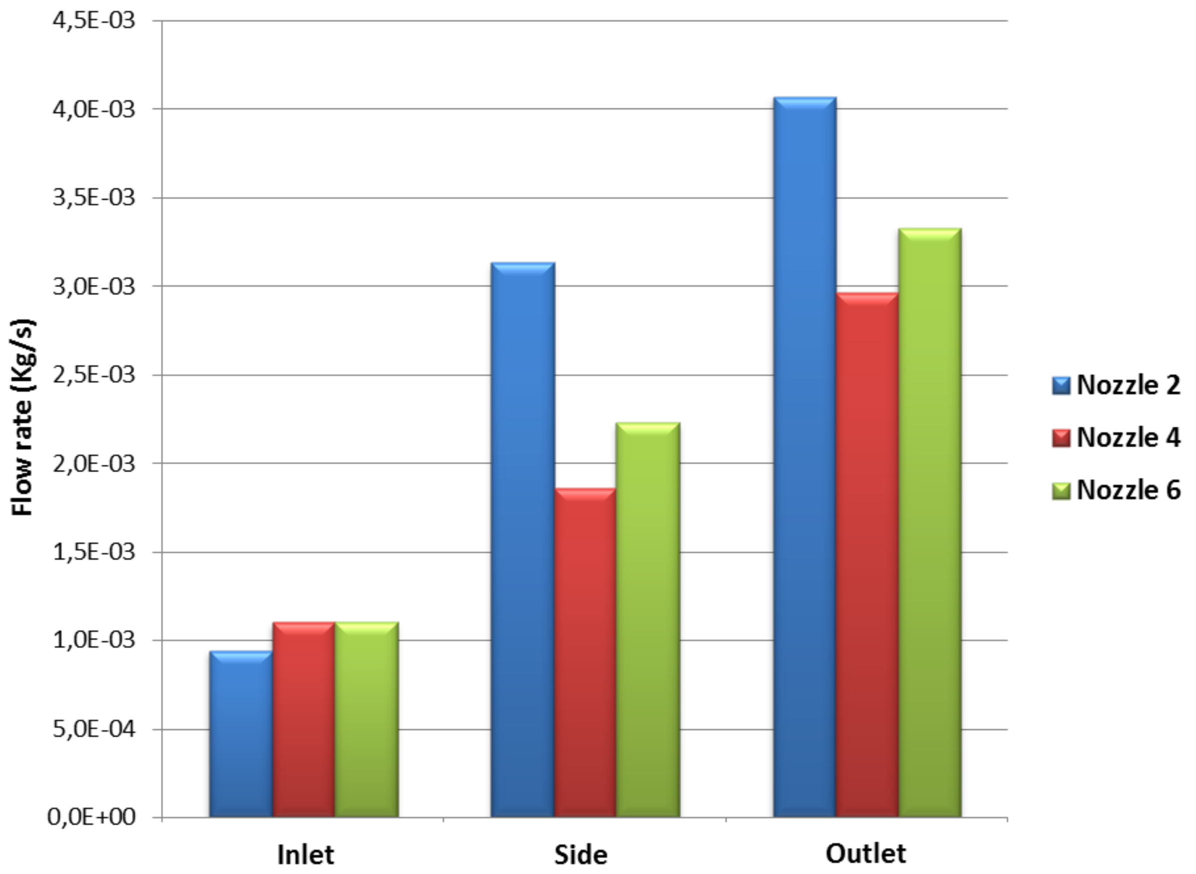


Chart 17: Flow rates

Finally, we compared flow rates. Although in inlet it was very similar (only nozzle with two holes had a little bit lower flow rate), they were significantly different in outlet and in sides.

This was due holes area differences between nozzles. Watching this chart was clear that the more area of holes, more air enters from outside by the holes, therefore, more flow rate in the outlet too.

5.4 Conclusions

We can conclude that if we look for saving the most amount of air without care about velocity of air (or force in particles) because we do not need a high force to clean our dust, the best option would be the Nozzle 2. Thanks to it have the biggest holes, it have too the highest rate between flow rate in inlet and flow rate in outlet, saving a huge amount of compressed air.

By another hand, if we need high velocity or a high force in particles of dust, we should use the Nozzle 6, with provide us the highest velocities and forces; but spending a little bit more of air.

In addition, it was clear that the last design (Nozzle 6), which adds two additional holes, improve the mean features regarding the previous one (Nozzle 4). It reaches higher velocities in the outlet with less flow rate in the inlet, saving more air than the previous design.

6 REFERENCE

ANSYS DesignModel User Guide. Release 16.0 - © SAS IP, Inc., 2014

ANSYS Workbench Meshing User´s Guide. Release 16.0 - © SAS IP, Inc., 2014

ANSYS CFX – Pre User´s Guide. Release 16.0 - © SAS IP, Inc., 2014

6.1 Information

***01.** Air Nozzles. MISUMI USA, Inc., (last visited on 25.01.2016) http://us.misumi-ec.com/vona2/mech/m2000000000/m2005000000/m2005020000*

***02.** Compressors, Pazaruvaj S.A, https://www.pazaruvaj.com/kompresori-c3765/gis/top300-100-car-m-p291878171 (last visited on 18/02/2016*

***03.** Solenoid Valve - 2 Way, Emerson Electric Cohttp://www.asco.com/en-us/Pages/solenoid-valve-series-353.aspx*

***04.** Handbook of Hydraulic Resistance, Gosudarstvennoe Energeticheskoe Izdatel'stvo, Moskva-Leningrad, 1960*

***05.** "Autodesk Inventor 2016 - A Tutorial Introduction" - L. Scott Hansen*

***06.** Meshing, CFD Online, http://www.cfd-online.com/Wiki/Meshing (last visited on 02.02.2016)*

***07.** Meshing, CFD Online, http://www.cfd-online.com/Wiki/Meshing (last visited on 02.02.2016)*

***08.** Turbulence Modeling, ANSYS Inc.; http://fluid.itcmp.pwr.wroc.pl/~pblasiak/CFD/UsefulInformation/sst.pdf (last visited on 21/02/2016)*

***09.** Turbulence Modeling, ANSYS Inc.; http://resource.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/brochure/ANSYS-Turbulence.pdf (last visited on 21/02/2016)*

***10.** Surface roughness, Wikipedia, https://en.wikipedia.org/wiki/Surface_roughness#cite_note-1 (last visited in 26.01.2016)*

***11.** Proposed Roughness Conversion Algorithm, Avestia Publishing, International ASET Inc., http://ijmem.avestia.com/2012/008.html (last visited on 26.01.2016)*

6.2 Figures

***Fig. 2** : The piston compressor as airbrush compressor, Airbrush-Compressor.info, http://airbrush-compressor.info/airbrush-compressor-piston/ (last visited on 25.01.2016)*

***Fig. 3**: Rotary Screw Air Compressors, Air Compressor Guide. By C. Beld, https://www.air-compressor-guide.com/learn/compressor-types/rotary-screw-compressor (last visited on 25.01.2016)*

***Fig. 4**: Vane Compressors, Compressor Pro, http://compresss.net/sliding-vane-compressors (last visited on 25.01.2016)*

***Fig. 5** and **Fig. 6** : Solenoid Actuator, About Air Compressors Bill's, http://www.about-air-compressors.com/solenoid-actuator.html, (last visited on 26.01.2016)*

ERROR! REFERENCE SOURCE NOT FOUND.

Fig. 7: *Air Nozzles. MISUMI USA, Inc., (last visited on 25.01.2016) http://us.misumi-ec.com/vona2/mech/m2000000000/m2005000000/m2005020000*

Fig. 8: *Simple nozzle, created by myself from a real nozzle drawing.*

***Fig. 9**, **Fig. 10** and **Fig. 11:** Handbook of Hydraulic Resistance, Gosudarstvennoe Energeticheskoe Izdatel'stvo, Moskva-Leningrad, 1960*

***Fig. 55**, **Fig. 56:** Proposed Roughness Conversion Algorithm, Avestia Publishing, International ASET Inc., http://ijmem.avestia.com/2012/008.html (last visited on 27.01.2016)*

6.3 Tables

***Table 1**: information to elaborate this table was taken from: Ventageneradores.net, Barcelona, http://www.ventageneradores.net/compresores-aire (last visited on 26.01.2016)*

***Table 7:** information to elaborate this table was taken from: The Engineering ToolBox, http://www.engineeringtoolbox.com/surface-roughness-ventilation-ducts-d_209.html (last visited on 26.01.2016)*

6.4 Abbreviations and symbols

$\mathbf{R_a}$: roughness normal parameter

ε: sand-grain radius parameter

ΔH: pressure loss or resistance (kg/m^2)

$\mathbf{\Delta H_{fr}}$**:** frictional losses (kg/m^2)

$\mathbf{\Delta H_l}$**:** local losses (kg/m^2)

ζ: coefficient of fluid resistance

ω= stream velocity (m/s)

γ: specific gravity of the flowing medium (kg/m^3)

g: gravitational acceleration (m/s^2)

φ= velocity coefficient at discharge from a sharp-edged orifice

$\mathbf{F_e}$, $\mathbf{F_{ex}}$ = area of narrowest and exit sections, respectively, (m^2)

σ' = central angle of divergence of the diffuser

ERROR! REFERENCE SOURCE NOT FOUND.

1 Air Nozzles. MISUMI USA, Inc., (last visited on 25.01.2016) http://us.misumi-ec.com/vona2/mech/m2000000000/m2005000000/m2005020000

2 Compressors, Pazaruvaj S.A, https://www.pazaruvaj.com/kompresori-c3765/gis/top300-100-car-m-p291878171 (last visited on 18/02/2016)

3 Solenoid Valve - 2 Way, Emerson Electric Cohttp://www.asco.com/en-us/Pages/solenoid-valve-series-353.aspx

4 Handbook of Hydraulic Resistance, Gosudarstvennoe Energeticheskoe Izdatel'stvo, Moskva-Leningrad, 1960

5 "Autodesk Inventor 2016 - A Tutorial Introduction" - L. Scott Hansen

6 Meshing, CFD Online, http://www.cfd-online.com/Wiki/Meshing (last visited on 02.02.2016)

7 Meshing, CFD Online, http://www.cfd-online.com/Wiki/Meshing (last visited on 02.02.2016)

8 Turbulence Modeling, ANSYS Inc.; http://fluid.itcmp.pwr.wroc.pl/~pblasiak/CFD/UsefulInformation/sst.pdf (last visited on 21/02/2016)

9 Turbulence Modeling, ANSYS Inc.; http://resource.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/brochure/ANSYS-Turbulence.pdf (last visited on 21/02/2016)

10 Surface roughness, Wikipedia, https://en.wikipedia.org/wiki/Surface_roughness#cite_note-1 (last visited in 26.01.2016)

11 Proposed Roughness Conversion Algorithm, Avestia Publishing, International ASET Inc., http://ijmem.avestia.com/2012/008.html (last visited on 26.01.2016)