



**UNIVERSITÀ DEGLI STUDI DI ROMA
TOR VERGATA**

FACOLTÀ DI INGEGNERIA

CORSO DI LAUREA IN INGEGNERIA MECCANICA

A.A. 2011/2012

Tesi di Laurea

Fluidodinamica dei collettori di scarico per motori a combustione interna. Analisi ed ottimizzazione della geometria attraverso un approccio numerico basato su un algoritmo di mesh morphing.

(Dynamic of internal flows in exhaust manifolds for combustion engines. Analysis and shape optimization through a CFD mesh-morphing-based approach)

RELATORE

Prof. Giuseppe Vairo

CANDIDATO

Walter Savastano
0144815

CORRELATORE

Prof. Marco E. Biancolini
Ing. Domenico Caridi

A Nonno Amedeo

Abstract

This thesis deals with a numerical parametric investigation aiming to optimize the fluid-dynamic performance of a manifold exhaust system for combustion engine.

Geometry and mesh have been directly provided by Ansys Italy and they refer to a four-cylinder internal combustion engine.

The model, generated and analysed by Ansys to simulate fluid-dynamics of internal flows, need to be tweaked.

Obviously in an exhaust system there are many parameters to consider; in our case, particular attention was placed on pressure drops among four engine exhaust valves and the outlet.

Specifically, it is need to prevent formation of turbulent areas. First thing to do is run a check on the baseline.

If the pressure drops across different ducts turns out to be different, this difference will be reduced as much as possible, trying to delete it altogether.

Following the baseline analysis, the changes necessary to apply to the geometry will be identified.

The analysis will be performed by Fluent, a commercial CFD core solver. Moreover, changes and optimization of the geometry will be developed through

the RBF-morph tool.

This software allows to significantly speed up calculation times. This is because normally, following an usual fluid dynamics simulation wherein changes to geometry are needed, other numerical external packages have to be employed, able to match, via an iterative data exchange process, with the CFD core solver.

With RBF-morph it is possible to create modifications and to wide these, even negatively, and setting output parameters (in our case the pressure differences). Moreover, it is possible to control the entire process from the Workbench platform;

noting how the output parameter changes, depending on the amplification of morph solutions.

Finally, numerical simulations will give indications on the values of the available parameters that allow to obtain an optimal geometry.

Contents

1	Understanding Exhaust	1
1.1	The How and Why	1
1.2	Flow Volume and Flow Velocity	5
1.3	Equal Length Exhaust	7
1.4	Exhaust Scavenging and Energy Waves	8
1.5	Materials	14
1.6	Pressure Loss	16
2	Computational Tools	18
2.1	Computational fluid dynamics	18
2.1.1	Discretization model	19
2.2	ANSYS FLUENT	22
2.2.1	Addressing the Complex Physics of a Complete System	24
2.2.2	CAD Import and Meshing	25
2.2.3	Multiphysics	25
2.2.4	Calculation procedure for CFD Fluent	26
2.3	Morphing with RBF Moph	38
2.3.1	Introduction	38

2.3.2	RBF Morph	40
2.3.3	conclusion	43
3	Model and Problems	45
3.1	Mesh	47
3.2	Fluent Settings	50
3.3	Baseline Results	54
3.4	CFD-Post Processing	55
3.4.1	Analyzing results	62
4	Morphing Solution	63
4.1	Test 1: Swelling & Necking Curve	64
4.1.1	Setup Details	64
4.1.2	Mesh Morphing	70
4.2	Test 2: Decreasing and Expanding section	74
4.2.1	Setup Details	74
4.2.2	Mesh Morphing	78
4.3	Test 3: Rotating outlet	81
4.3.1	Setup Details	81
4.3.2	Mesh Morphing	84
4.4	Test 4: Single Pipe Modifications	87
4.4.1	Setup Details Pipe1	87
4.4.2	Mesh Morphing Pipe1	90
4.4.3	Setup Details Pipe2	93

4.4.4 Mesh Morphing Pipe2	95
5 Results	98
5.1 Running and Results	99
5.1.1 “Swelling and Necking Curve” Results	100
5.1.2 “Decreasing manifold section” Results	102
5.1.3 “Rotating outlet” Results	104
5.1.4 “Single Pipe Modifications” Results	106
5.2 Coupled Results	108
 List of Figures	 116
 List of Tables	 119
 Bibliography	 120

Chapter 1

Understanding Exhaust

1.1 The How and Why

In automotive engineering, an exhaust manifold collects the exhaust gases from multiple cylinders into one pipe. The word manifold comes from the Old English word manigfeald (from the Anglo-Saxon manig [many] and feald [fold]) and refers to the folding together of multiple inputs and outputs. In contrast, an inlet manifold is the part of an engine that supplies the air to the cylinders. No exhaust system is ideal for all applications. Depending on their design and purpose, all exhaust systems compromise something to achieve something else. Before performing exhaust changes or modifications to increase performance, it is critical to determine what kind of performance you want:

- Do you want the best possible low-end and mid-range power or maximum top-end power?
- Will you be using an aftermarket cam with different lift, duration, timing and overlap?

- Have you investigated the relationship between torque (force) and horsepower (amount of work within time)?
- Do you want a cosmetic exhaust system or a performance exhaust system?



Figure 1.1: Mini Cooper exhaust manifold

Without careful thought about these variables, an exhaust system can yield very disappointing results. On the other hand, a properly designed and tuned exhaust system that is well-matched to the engine can provide outstanding power gains. The distinction between "*maximum power*" and "*maximum performance*" is significant beyond general discussion. Realistically, one exhaust system may not produce both maximum power and maximum performance.

For an automobile to cover "X" distance as quickly as possible, it is not the highest peak power generated by the engine that is most critical. It is the highest average power generated across the distance that typically produces

the quickest time. When comparing two horsepower curves on a dynamometer chart (assuming other factors remain constant), the curve containing the greatest average power is the one that will typically cover the distance in the least time and that curve may, or may not, contain the highest possible peak power.

In the strictest technical sense, an exhaust system cannot produce more power on its own. The potential power of an engine is determined by the proper amount of fuel available for combustion. However, the efficiency of combustion and engine pumping processes is profoundly influenced by the exhaust system. A properly designed exhaust system can reduce engine pumping losses. Therefore, the design objective for a high performance exhaust is (or should be) to reduce engine-pumping losses, and by so doing, increase volumetric efficiency. The net result of reduced pumping losses is more power available to move the automobile. As volumetric efficiency increases, potential fuel mileage also increases because less throttle opening is required to move the automobile at the same velocity.

Much controversy (and apparent confusion) surrounds the issue of exhaust "back-pressure". Many performance-minded people who are otherwise knowledgeable still cling tenaciously to the old school concept.... "You need more back-pressure for better performance". For virtually all high performance purposes, backpressure in an exhaust system increases engine-pumping losses and decreases available engine power. It is true that some engines are mechanically tuned to "X" amount of backpressure and can show a loss of low-end torque



Figure 1.2: Mazda mx5 racing beat exhaust manifold

when that backpressure is reduced. It is also true that the same engine that lost low-end torque with reduced back-pressure can be mechanically re-tuned to show an increase of low-end torque with the same reduction of back-pressure. More importantly, maximum mid-to-high RPM power will be achieved with the lowest possible backpressure.

The objective of most engine modifications is to maximize the proper air and fuel flow into, and exhaust flow out of the engine.

The inflow of an air/fuel mixture is a separate issue, but it is directly influenced by exhaust flow, particularly during valve overlap (when both valves are open for "X" degrees of crankshaft rotation). Gasoline requires oxygen to burn. By volume, dry, ambient air at sea level contains about 21% oxygen, 78% Nitrogen and trace amounts of Argon, CO₂ and other gases. Since oxygen is only about 1/5 of air's volume, an engine must intake 5 times more air than oxygen to get the oxygen it needs to support the combustion of fuel.

If we introduce an oxygen-bearing additive such as nitrous oxide, or use an oxygen-bearing fuel such as nitromethane, we can make much more power from the same displacement because both additives bring more oxygen to the combustion chamber to support the combustion of more fuel. If we add a supercharger or turbocharger, we get more power for the same reason, more oxygen is forced into the combustion chamber. Theoretically, in a normally aspirated state of tune without fuel or oxygen-rich additives, an engine's maximum power potential is directly proportional with the volume of air it flows. This means that an engine of 1300 cm^3 , has the same maximum power potential as an engine of 1600 cm^3 , if they both flow the same volume of air.

1.2 Flow Volume and Flow Velocity

One of the biggest issues with exhaust systems, is the relationship between gas flow volume and gas flow velocity (which also applies to the intake track):

- An engine needs the highest flow velocity possible for quick throttle response and torque throughout the low-to-mid range portion of the powerband;
- The same engine also needs the highest flow volume possible throughout the mid-to-high range portion of the powerband for maximum performance.

This is where a fundamental conflict arises.

For "X" amount of exhaust pressure at an exhaust valve, a smaller diameter exhaust pipe will provide higher flow velocity than a larger diameter pipe.

Unfortunately, the laws of physics will not allow that same small diameter pipe to flow sufficient volume to realize maximum possible power at higher RPM.

If we install a larger diameter pipe, we will have enough flow volume for maximum power at mid-to-high RPM, but the flow velocity will decrease and low-to-mid range throttle response and torque will suffer. This is the primary paradox of exhaust flow dynamics and the solution is usually a *design compromise* that produces an acceptable amount of throttle response, torque and horsepower across the entire powerband.

A very common mistake made by some performance people is the selection of an exhaust system with pipes that are too large in diameter for their engine's state of tune. Bigger is not necessarily better and is often worse.



Figure 1.3: Peugeot 206 16V GTI Stainless steel exhaust manifold

1.3 Equal Length Exhaust

The effectiveness of equal length exhaust is widely debated. Assuming that an exhaust system is otherwise properly designed, equal length pipes offer some benefits that are not present with unequal length pipes. These benefits are smoother engine operation, tuning simplicity and increased low-to-mid range torque. If the pipes are not equal length, both inertial scavenging and wave scavenging (par 1.4) will vary among engine cylinders, often dramatically.

This, in turn, causes different tuning requirements for different cylinders. These variations affect air/fuel mixtures and timing requirements, and can make it very difficult to achieve optimal tuning.

Equal length pipes eliminate these exhaust-induced difficulties. "Tuning", in the context used here, does not mean installing new sparkplugs and an air filter. It means configuring a combination of mechanical components to maximum efficiency for a specific purpose and it can not be overemphasized that such tuning is the path to superior performance with a combination of parts that must work together in a complimentary manner.

In an exhaust system that is properly designed for it's application, equal length pipes are generally more efficient. The lengths of both the primary and main section of pipes strongly influence the location of the torque peak(s) within the powerband. In street and track performance engines with longer pipes typically produce more low-to-mid range torque than shorter pipes and it is torque that moves a automobile. The question is:

Where in the powerband do you want to maximize the torque?

- Longer pipes tend to increase power below the engine's torque peak
- Shorter pipes tend to increase power above the torque peak.
- Large diameter pipes tend to limit low-range power and increase high range power.
- Small diameter pipes tend to increase low-range power and to some degree limit high-range power.
- "Balance" or "equalizer" chambers between the exhaust pipes tend to flatten the torque peak(s) and widen the powerband.

Among the more astute and responsible exhaust builders, it is more-or-less understood that pipe length variations should not exceed 1 inch to be considered equal. Even this standard can result in a 2 inch difference if one pipe is an inch short and another pipe is an inch long.

1.4 Exhaust Scavenging and Energy Waves

When an engine starts its exhaust stroke, the piston moves up the cylinder bore, decreasing the total chamber volume. When the exhaust valve opens, the high pressure exhaust gas escapes into the exhaust manifold or header, creating an exhaust pulse comprising three main parts: the high-pressure head is created by the large pressure difference between the exhaust in the combustion chamber and the atmospheric pressure outside of the exhaust system.

Engine needs an enormous amount of air and, therefore, needs a good suction system. The way in which the engine breathes depends on physical laws whose parameters are shape, width and length of intake pipes and exhaust. Varying length and shape of these ducts, it manages to improve (or worse) the engine, improving breathing (or worse) so its performances.

Inertial scavenging and wave scavenging, are different phenomena but both impact exhaust system efficiency and affect one another. Scavenging is simply gas extraction. These two scavenging effects are directly influenced, as already mentioned, by pipe diameter, length, shape and the thermal properties of the pipe material (stainless, mild steel, thermal coatings, etc.). When the exhaust valve opens, two things immediately happen:

an energy wave, or pulse, is created from the rapidly expanding combustion gases. The wave enters the exhaust pipe traveling outward at a nominal speed of 400-500 m/s (this speed varies depending on engine design, modifications, etc., and is therefore stated as a "nominal" velocity). This wave is pure energy, similar to a shock wave from an explosion.

Simultaneous with the energy wave, the spent combustion gases also enter the exhaust pipe and travel outward more slowly at 50 - 100 m/s nominal (maximum power is usually made with gas velocities between 80-100 m/s). Since the energy wave is moving about 5-10 times faster than the exhaust gases, it will get where it is going faster than the gases. When the outbound energy wave encounters a lower pressure area such as a second or larger diameter section of pipe, the muffler or the ambient atmosphere, a reversion wave

(a reversed or mirrored wave) is reflected back toward the exhaust valve without significant loss of velocity.

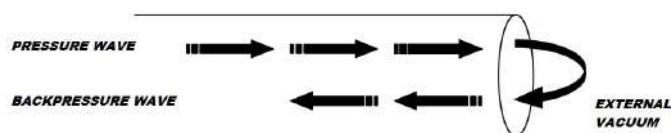


Figure 1.4: Backpressure

The reversion wave moves back toward the exhaust valve on a collision course with the exiting gases whereupon they pass through one another, with some energy loss and turbulence, and continue in their respective directions. What happens when that reversion wave arrives at the exhaust valve depends on whether the valve is still open or closed.

This is a critical moment in the exhaust cycle because the reversion wave can be beneficial or detrimental to exhaust flow, depending upon its arrival time at the exhaust valve. If the exhaust valve is closed when the reversion wave arrives, the wave is again reflected toward the exhaust outlet and eventually dissipates its energy in this back and forth motion. If the exhaust valve is open when the wave arrives, its effect upon exhaust gas flow depends on which part of the wave is hitting the open exhaust valve: a wave is comprised of two alternating and opposing pressures.

In one part of the wave cycle, the gas molecules are compressed. In the other part of the wave, the gas molecules are rarefied.

Therefore, each wave contains a compression area (node) of higher pressure and a rarefaction area (anti-node) of lower pressure. An exhaust pipe of the proper length (for a specific RPM range) will place the wave's anti-node at the exhaust valve at the proper time for its lower pressure to help fill the combustion chamber with fresh incoming charge and to extract spent gases from the chamber. This is wave scavenging or "wave tuning".

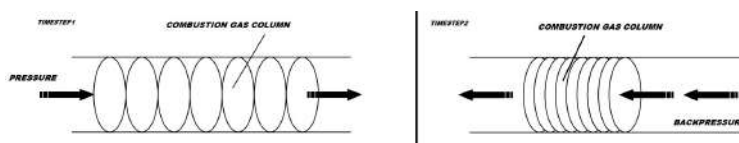


Figure 1.5: phases of pressure waves

The ideal condition for leaking exhaust gases into the combustion chamber (cylinder) is an empty on exit, just outside of the exhaust valve.

Vacuum aspires drain column gas from the cylinder.

Otherwise, if you were to make an excess pressure just outside of the exhaust valve, free breathing would be compromised, and with this also the performance of the engine. The engine, in this case, hasn't difficulty expelling gas, so expulsion takes place almost automatically.

Intervening on the geometry, varying the length and shape of exhaust pipe, managing to create an ideal environment for breathing engine, is improved its performance. More is the length of exhaust pipe, more time will put the pressure wave to complete his journey towards the exit, then bounce back, as wave of depression, to his point of departure. In an infinitely long tube there

are no phenomena of depression, for the simple fact that the pressure waves never reach the outside vacuum (for then bounced backwards). Conversely, in a very short exhaust system, pressure variations would feel very much because, just out of the cylinder, the wave would be the exit discharge. And bouncing, as a wave of depression, inwards, there would be only a moment to return to the cylinder, its point of departure.

From these cyclical engine events, one can deduce that the beneficial part of a rapidly traveling reversion wave can only be present at an exhaust port during portions of the powerband since it's relative arrival time changes with RPM.

This makes it difficult to tune an exhaust system to take advantage of reversion waves which is why there are various anti-reversion devices designed to improve performance. These anti-reversion devices are designed to weaken and disrupt the detrimental reversion waves (when the wave's higher-pressure node impedes scavenging and intake draw-through). Specifically designed performance baffles can be extremely effective, as well as heads with D shaped ports.

Unlike reversion waves that have no mass, exhaust gases do have mass. Since they are in motion, they also have inertia (or "momentum") as they travel outward at their comparatively slow velocity of 50 - 100 m/s. When the gases move outward as a gas column through the exhaust pipe, a decreasing pressure area is created in the pipe behind them. It may help to think of this lower pressure area as a partial vacuum and one can visualize the vacuous lower pressure "pulling" residual exhaust gases from the combustion chamber and

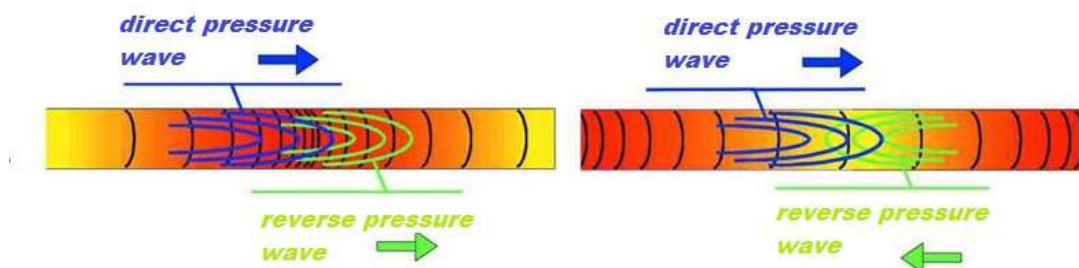


Figure 1.6: Pressure Waves

exhaust port. It can also help pull fresh air/fuel charge into the combustion chamber. This is inertial scavenging and it has a major effect upon engine power at low-to-mid range RPM.

There are other factors that further complicate the behavior of exhaust gases. Wave harmonics, wave amplification and wave cancellation effects also play into the scheme of exhaust events.

The interaction of all these variables is so abstractly complex that it is difficult to fully grasp. There does not appear to be any absolute formula that will produce the perfect exhaust design. Even super-computer designed exhaust systems must undergo dyno, track, and street testing to determine the necessary configuration for the desired results. Last but not least, the correct choices and combinations of carburetor, air cleaner, cam shaft, ignition, and exhaust used in the proper relationship to each other for the intended riding application will always produce the finest quality results.

Great care must be used when selecting the length and diameter of the primary tubes. Tubes that are too large will cause the exhaust gas to expand

and slow down, decreasing the scavenging effect.

Tubes that are too small will create exhaust flow resistance which the engine must work to expel the exhaust gas from the chamber, reducing power and leaving exhaust in the chamber to dilute the incoming intake charge. Since engines produce more exhaust gas at higher speeds, the header(s) are tuned to a particular engine speed range according to the intended application. Typically, wide primary tubes offer the best gains in power and torque at higher engine speeds, while narrow tubes offer the best gains at lower speeds.

Of course, exhaust system, must be held by the shape of the frame. Unloading must fit without being bulky, must have a suitable form. All these factors make it very difficult to design a highly competitive exhaust system.

1.5 Materials

Exhaust manifolds are generally simple cast iron or stainless steel units which collect engine exhaust from multiple cylinders and deliver it to the exhaust pipe. For many engines, there are aftermarket tubular exhaust manifolds known as headers in US English, as extractors in Australian English, and simply as "tubular manifolds" in UK English. These consist of individual exhaust headpipes for each cylinder, which then usually converge into one tube called a collector. Headers that do not have collectors are called zoomie headers, and are used exclusively on race cars.

The most common types of aftermarket headers are made of either ceramic or stainless steel. Ceramic headers are lighter in weight than stainless steel,

however, under extreme temperatures, they can crack, something stainless steel is not prone to.

Another form of modification used is to insulate a standard or aftermarket manifold. This decreases the amount of heat given off into the engine bay, therefore reducing the intake manifold temperature.

There are a few types of thermal insulation but three are particularly common:

- Ceramic paint is sprayed or brushed onto the manifold and then cured in an oven. These are usually thin, so have little insulatory properties however reduce engine bay heating by lessening the heat output via radiation;
- A ceramic mixture is bonded to the manifold via thermal spraying to give a tough ceramic coating with very good thermal insulation. This is often used on performance production cars and track-only racers;
- Exhaust wrap is wrapped completely around the manifold. Although this is cheap and fairly simple, it can lead to premature degradation of the manifold.

The goal of performance exhaust headers is mainly to decrease flow resistance (back pressure), and to increase the volumetric efficiency of an engine, resulting in a gain in power output. The processes occurring can be explained by the gas laws, specifically the ideal gas law and the combined gas law.

1.6 Pressure Loss

Let consider circular cross-section pipe, about length L and diameter D crossed by a viscous fluid with a volumetric flow Q ; assuming incompressible flow we can explain the relation

$$Q = \frac{\pi U D^2}{4} \quad (1.6.1)$$

So applying Bernoulli equation, and adding an h factor for viscous effect

$$\frac{p_1}{\rho} + \frac{U_1^2}{2} + gz_1 = \frac{p_2}{\rho} + \frac{U_2^2}{2} + gz_2 + gh \quad (1.6.2)$$

also suppose the height difference is unuseful, so we have

$$h = \frac{p_1 - p_2}{\rho g} \quad (1.6.3)$$

physic explanation is that effect of viscous effects is like a raising of the outlet of height h . This involves that to bring the fluid from inlet to outlet there is a need a greater pressure than non-viscous case. Using Buckingham Theorem it is easy put in an dimensionless relation, Pressure Loss with other factor from which depends

$$\frac{\Delta p}{\frac{1}{2}\rho U^2} = \phi\left(\frac{l}{D}, \frac{\epsilon}{D}, \frac{\rho U D}{\mu}\right) \quad (1.6.4)$$

Empirically it has been seen that the factor affects linearly then we can write

$$\frac{\Delta p}{\frac{1}{2}\rho U^2} = \frac{l}{D} \phi\left(\frac{\epsilon}{D}, \frac{\rho U D}{\mu}\right) \quad (1.6.5)$$

defining λ a factor as

$$\lambda = \frac{\Delta p}{\frac{1}{2}\rho U^2} \frac{D}{l} = \phi\left(\frac{\epsilon}{D}, \frac{\rho U D}{\mu}\right) \quad (1.6.6)$$

This is the most simplified case, we have also to consider some components, like junctions, curves, variations of sections et cetera, where the last discussion is totally wrong because of a separation of the flow.

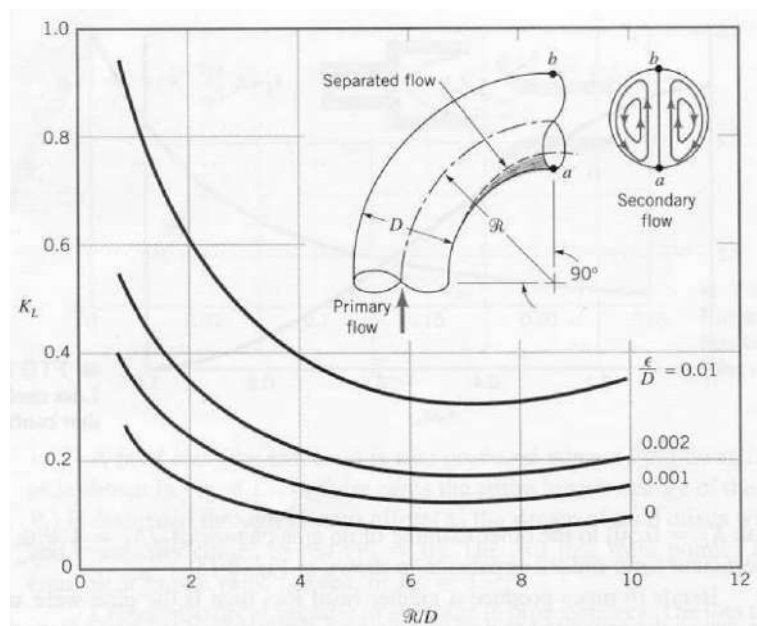


Figure 1.7: Concentrated Pressure Loss

Even in this case experimental analysis help us to find answers we want. Evidently this pressure loss localized depends on the component geometry in this way the effect of each component is a localized pressure loss, equivalent to a quantity of kinetic energy of the flow.

$$\Delta p = \frac{1}{2} \rho U^2 \lambda' \tag{1.6.7}$$

Chapter 2

Computational Tools

2.1 Computational fluid dynamics

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define any single-phase fluid flow:

$$\rho \frac{D\vec{u}}{Dt} = -\nabla p + \rho f - \frac{2}{3} \nabla \cdot [(\mu \nabla \cdot \vec{u}) \tilde{I}] + 2 \nabla \cdot (\mu \tilde{E}) \quad (2.1.1)$$

These equations can be simplified by removing terms describing viscosity to yield the Euler equations. Further simplification, by removing terms describing

vorticity yields the full potential equations. Finally, these equations can be linearized to yield the linearized potential equations.

2.1.1 Discretization model

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are:

Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes. The governing equations are solved over discrete control volumes.

Finite volume methods recast the governing partial differential equations (typically the Navier-Stokes equations) in a conservative form, and then discretize new equation. This guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form:

$$\frac{\delta}{\delta t} \iiint \mathbf{Q} \delta V + \iint \mathbf{F} \delta A = 0 \quad (2.1.2)$$

where \mathbf{Q} is the vector of conserved variables, \mathbf{F} is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the volume of the control volume element, and A is the surface area of the control volume element.

Finite element method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach. However, FEM can require more memory than FVM. In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q \delta V^e \quad (2.1.3)$$

where R_i is the equation residual at an element vertex i , Q is the conservation equation expressed on an element basis, W_i is the weight factor, and V^e is the volume of the element.

Finite difference method

The finite difference method (FDM) has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries, making these codes highly efficient and accurate. Other ways to handle geometries include use of overlapping grids, where the solution is interpolated across each grid.

$$\frac{\delta \mathbf{Q}}{\delta t} + \frac{\delta F}{\delta x} + \frac{\delta G}{\delta y} + \frac{\delta H}{\delta z} = 0 \quad (2.1.4)$$

where \mathbf{Q} is the vector of conserved variables, and F , G , and H are the fluxes in the x , y , and z directions respectively.

Spectral element method

Spectral element method is a finite element type method. It requires the mathematical problem (the partial differential equation) to be casted in a weak formulation. This is typically done by multiplying the differential equation by an arbitrary test function and integrating over the whole domain. Purely mathematically, the test functions are completely arbitrary, they belong to an infinitely dimensional function space. Clearly an infinitely dimensional function space cannot be represented on a discrete spectral element mesh. And this is where the spectral element discretization begins. The most crucial thing is the choice of interpolating and testing functions. In a standard, low order FEM in 2D, for quadrilateral elements the most typical choice is the bilinear test or interpolating function of the form:

$$v(x, y) = ax + by + cxy + d \quad (2.1.5)$$

In a spectral element method however, the interpolating and test functions are chosen to be polynomials of a very high order (typically e.g. of the 10th order in CFD applications). This guarantees the rapid convergence of the method. Furthermore, very efficient integration procedures must be used, since the number of integrations to be performed in a numerical codes is big. Thus, high order Gauss integration quadratures are employed, since they achieve the highest accuracy with the smallest number of computations to be carried out. At the time there are some academic CFD codes based on the spectral element method and some more are currently under development, since the new time-stepping schemes arise in the scientific world.



2.2 ANSYS FLUENT

There are huge numbers of engineering applications that can benefit from computational fluid dynamics simulation. Whether are analyzed commonplace fluid flow and heat transfer or work with complex transient reacting flows, Ansys Fluent software should be an integral part of product design and optimization process.

A fully featured fluid dynamics solution for modeling flow and other related physical phenomena, Fluent offers unparalleled analysis capabilities. It provides all the tools needed to design and optimize new equipment and to troubleshoot existing installations. The versatile technology offers insight into how a product design will behave in the real world, all before a single prototype is built.

Fluent's capabilities are developed by world-renowned experts and supported by extremely experienced engineers so you can have confidence in the solution as you develop higher quality products faster, decrease time to market, reduce risk and increase innovation. To provide high productivity, the Ansys

Workbench platform directly couples with any CAD software and automatically extracts and meshes fluid volumes. You control this easy-to-perform operation via a small set of parameters; the yield is the high-quality meshes critical for both accurate and fast CFD simulation.

CFD package includes solvers that accurately simulate behavior of the broad range of flows that engineers encounter daily from Newtonian to non-Newtonian, from single-phase to multi-phase, and from subsonic to hypersonic. Each solver is highly robust, well tested, validated and optimized for fast simulation time. Time tested and part of a single environment, the highly efficient solvers deliver both accuracy and speed. For deeper insight such as making informed decisions about small adjustments that yield large performance improvements you can increase the granularity of the analysis.

Such improved resolution requires more computational resources and parallel computing. Fluent has a record of outstanding parallel scalability, ranging from two processors to thousands, giving you high-fidelity results in the shortest possible time.

Optimizing your product requires evaluating a large number of designs. Capabilities within Ansys Workbench enable efficient, fully automated optimization (or design of experiments) for tens or hundreds of design points; the technology can evaluate many design points concurrently. Workbench makes the process easy by controlling the execution, results data and file management for each set of design point.

2.2.1 Addressing the Complex Physics of a Complete System

Over the years, products have become extremely complex. So too have done fluid dynamics problems: systems with moving parts (such as pistons and valves) require transient analysis; systems with phase changes due to heating or cooling liquids (for example, heat exchangers) require accurate, multi-phase capabilities; and systems with challenging multiphysics phenomena (such as fluid–structure interaction) call for easy-to-use, accurate advanced capabilities.

To gain more insight into product behavior, you need to consider the full range of physics present in the system you are designing. Fluent offers state-of-the-art advanced capabilities to model laminar and turbulent flows as well as more-complex physics including multiphase flows, chemical reactions, radiation and particulate dynamics. You can be confident that Fluent will accurately predict product behavior because all models are thoroughly tested and validated. For best-in-class products, engineers can no longer rely on analyzing one type of physics (fluids, structural or electromagnetics). Instead, you must study all physics along with their interactions. You can seamlessly couple Fluent with Ansys structural mechanics or electromagnetics simulation tools to gain insight into the entire system. For example, you can study how the fluids system deforms the structure that contains it, or how the heat generated by an electronic component affects fluids temperature.

2.2.2 CAD Import and Meshing

From CAD import to geometry meshing, flexible tools allow to automatically create meshes or hand-craft them.

Ansys meshing can extract fluid volume from a CAD assembly and automatically create tetrahedral or hexahedral meshes with inflation layers.

Ansys also offer advanced repair tools so can import and prep geometry for partly or fully manual meshing.

Ansys pre-processing tools provide the high-quality meshes project needs, so you obtain accurate results. Simulation-Driven Product Development relies on design process compression, using solutions that fully automate your simulation workflow so you can focus on your engineering goals. Advanced simulation workflow and application technologies are key to accelerating processes and gaining the necessary insight into your product so can quickly make the right design decisions

2.2.3 Multiphysics

The Ansys portfolio of simulation tools enables to accurately predict real-world, multiphysics behavior of industrial designs.

Phenomena such as flow-induced vibration and material deformation induced by fluid can be readily captured using our multiphysics tools. Ansys provide comprehensive technologies for all physics disciplines: structural mechanics, heat transfer, fluid flow and electromagnetics.

By combining these, it can solve complex industrial engineering challenges

to optimize entire product.

Ansys provides a powerful integrated solution for automating different physics simulations. Data transfer to and from multiple physics eliminates the time required to manually convert output from one stage of work process into input for another.

Optimization: The inherent and integrated parametric capabilities of our simulation framework enables optimization for all types of engineering applications. We in this case used RBF Morph to optimize the geometry and a its tool that allow to optimize geometry taking into account external constraints

Post-Processing and Archival: The powerful post-processor for Ansys CFD provides advanced quantitative and high-quality visual post-processing capabilities, including easy creation of charts, high-quality images and videos. Simulation engineers generate large volumes of data that must be archived in a searchable format.

2.2.4 Calculation procedure for CFD Fluent

Fluent CFD has a very complex structure, at the base of which there are several mathematical models needed to define the physical properties and flow characteristics of the fluid.

It is essential, before starting any simulation with Fluent software, follow some basic points. The first thing doing after have imported the geometry model developed in any software, it is to check that there are no errors in the

grid (e.g. parts of mesh finite element with distorted, negative volume, border zones not defined properly, etc.).

Check When Fluent start in the command line shows information regarding: the grid item type, the number of cells, the number of nodes, the minimum and maximum size of the volumes of the elements, the minimum and maximum spatial coordinates of geometric model along the axis x, y, z.

For a better view the other information can also be displayed by the command ”General → Check.

```

Mesh
Agri 10.1
ANSYS FLUENT 13.0 (3d, pbrs)

> Reading "C:\Users\Walt\Desktop\Thesis\mesh\maniFold_cutcell_ske.cas"...
Using buffering for scanning file.
397892 mixed cells, zone 2, binary.
1036912 mixed interior faces, zone 1, binary.
  70 mixed velocity-inlet faces, zone 5, binary.
 380 mixed pressure-outlet faces, zone 6, binary.
 427 mixed pressure-outlet faces, zone 7, binary.
 484 mixed pressure-outlet faces, zone 8, binary.
 483 mixed pressure-outlet faces, zone 9, binary.
 87895 mixed wall faces, zone 10, binary.
126384 mixed interior faces, zone 11, binary.
447991 nodes, binary.
447991 node flags, binary.

Building...
  mesh

Mesh
ANSYS FLUENT 13.0

Mesh Check

Domain Extents:
x-coordinate: min (m) = -2.059723e-01, max (m) = 1.699978e-01
y-coordinate: min (m) = -1.000000e-02, max (m) = 6.058343e-01
z-coordinate: min (m) = -8.099721e-02, max (m) = 1.999888e-02

Volume statistics:
  minimum volume (m3): 6.020139e-13
  maximum volume (m3): 9.222114e-07
  total volume (m3): 3.033195e-03

Face area statistics:
  minimum face area (m2): 6.576930e-09
  maximum face area (m2): 1.476299e-04

Checking mesh.....
Done.

```

Figure 2.1: Fluent window: “Grid” menù

General setup In solver panel is possibly choose if the model will Run in Steady or in Transient. If the velocity formulation is Absolute or Relative, and if Solver Type is Pressure-Based or Density-Based

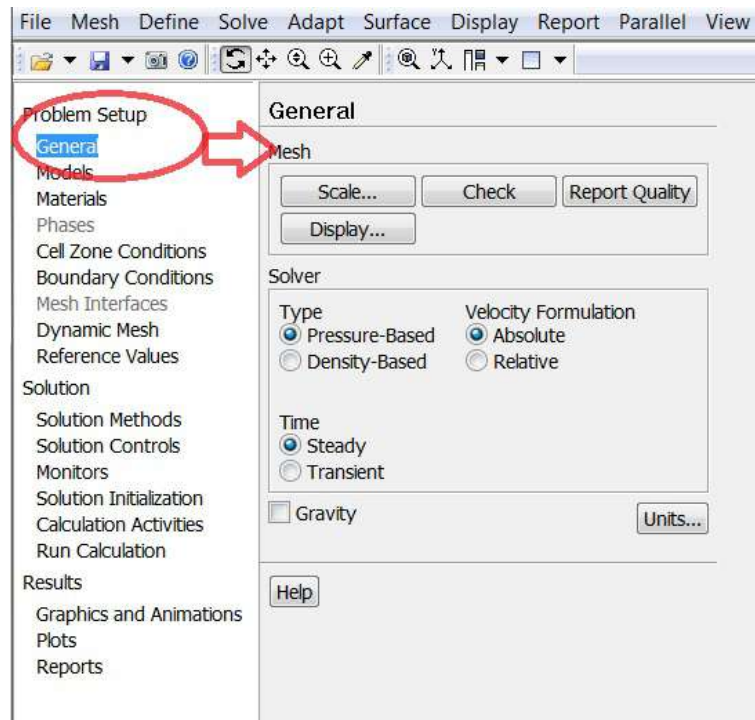


Figure 2.2: Fluent window: Solver Panel

Viscous Model: The next step now is to define the type of viscous model for runoff. From the main menu you can do this by selecting “Define” → “Models” → “Viscous”.

Fluent software contains a wide range of viscous models: Inviscid, Laminar Spalart-Allmaras,, $k - \varepsilon$, $k - \omega$, etc.

In the ”Viscous Model” are also displays the values of the constants in the

mathematical equations of viscous model chosen.

It can also determined the type of condition to the wall.

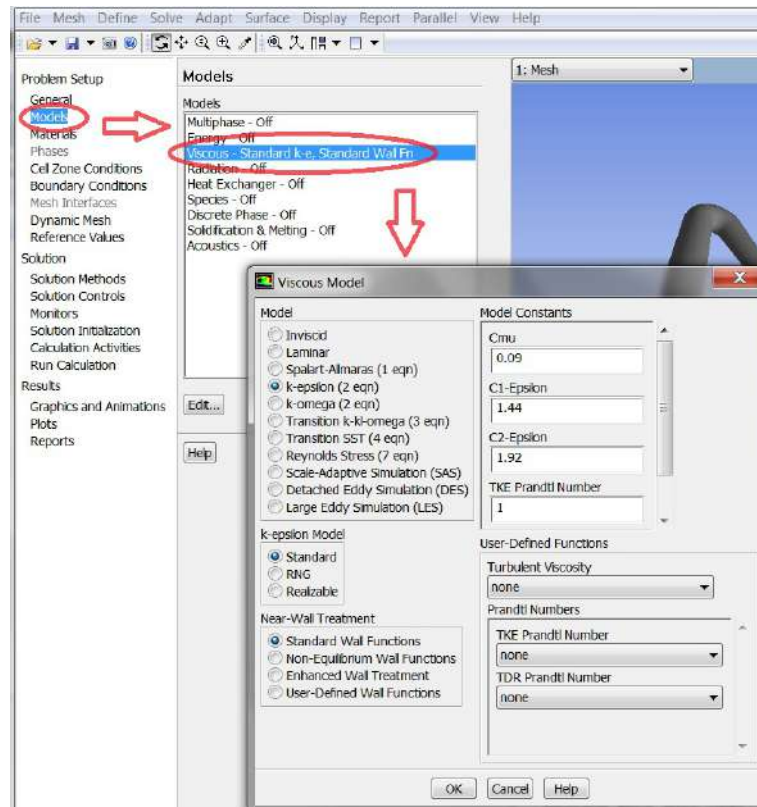


Figure 2.3: Fluent window: Viscous model

The energy equation could be activated selecting “Define” → “Models” → “Energy”.

Inside the “Define” → “Models” there are many other physical models involving the thermal radiation, heat exchanger, etc.

Fluid Type Chosen the viscous model it is necessary to establish the operating fluid type during simulations. It is sufficient in this case select “Define” → “Materials” to define the physical characteristics of the fluid.

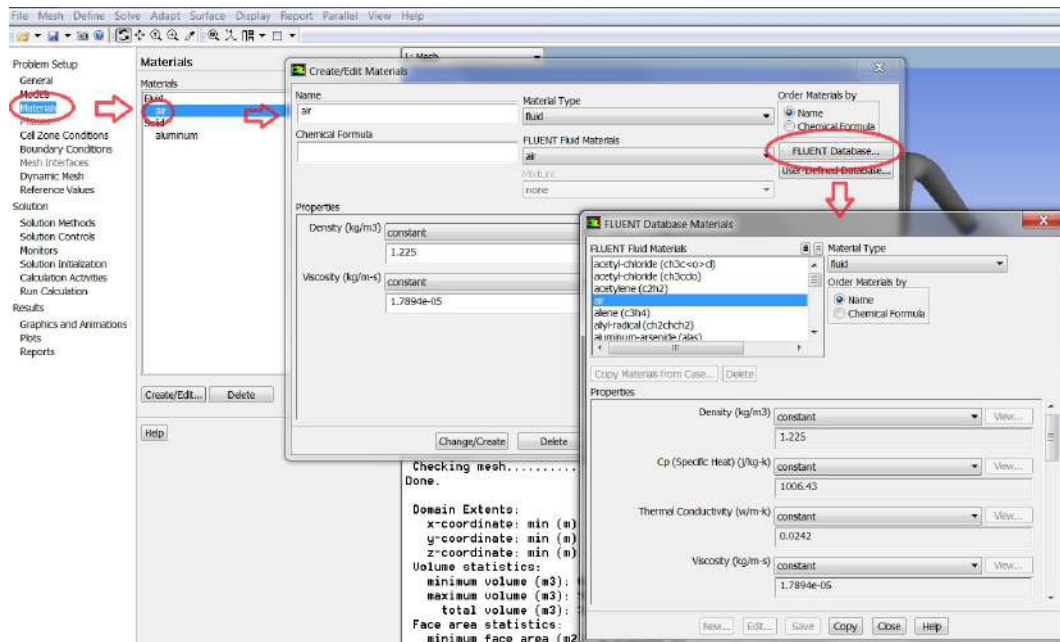


Figure 2.4: Fluent Database Materials

Operative & Boundary Condition Before moving on to ”Solve” it is necessary to establish operating conditions and the boundary conditions system to simulate.

Operating Condition can be set in “Cell Zone Condition”.

Boundary conditions must be defined for any area, from the “Boundary Conditions” you select the area of interest and then the type of boundary condition). In some cases, such as for sections of entry or exit, choose the type of boundary condition, another window opens in which you are prompted more

physical variables (eg. speed, temperature, hydraulic diameter section, etc.).

The operative condition must be modifies only not atmospheric condition case

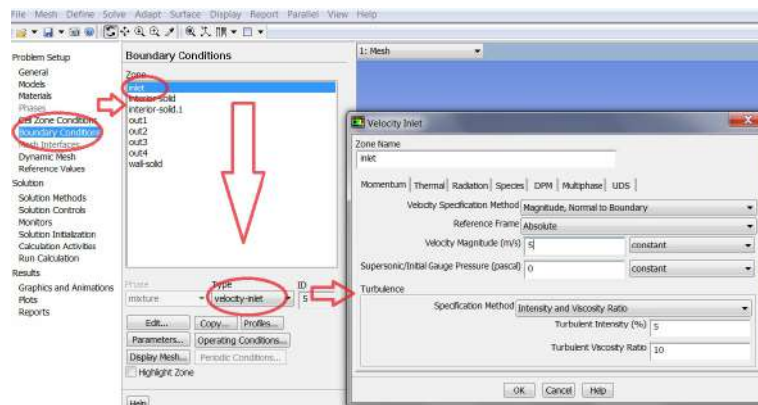
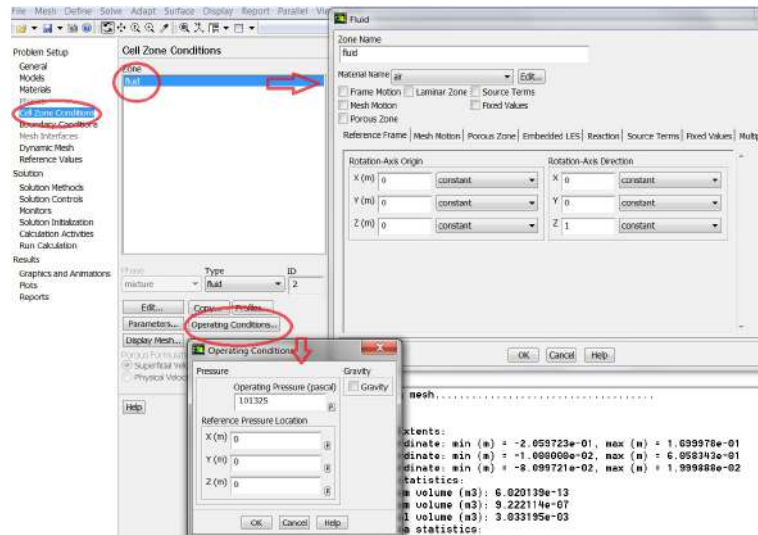


Figure 2.5: Operating Condition & Boundary Condition

Solution Control The later stage, before moving to iterations, it is to control the parameters present in the mathematical equations of viscous model previously defined. Selecting path “Solution Control” is shown a window that is composed mainly by subwindow: “Explicit Relaxation Factors, Under Relaxation Factors”. By clicking the text “Equation”, there are mathematical formulas that will iterate. The subwindows “Explicit Relaxation Factor” “Under Relaxation Factor” contain values of relax in the formulas of the moment of momentum in the continuity equation, the equation turbulence model, etc. These values should not be changed, usually at least that the simulations diverges, in this circumstance is convenient to reduce to promote convergence.

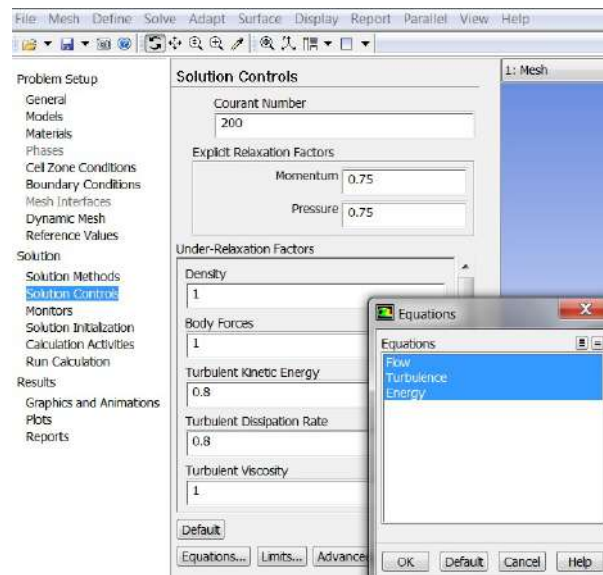


Figure 2.6: Solution Controls

Solution Method Selecting patch “Solution Method” it is allowed choosing the control parameters of the discretization solving equations.

In the subwindow “Scheme” is possible set the most appropriate algorithm for coupled equations of speed and pressure. The solver “Simple” is indicated for incompressible fluids or slightly compressible. Type “Coupled” instead is indicated for high speed compressible fluid outflow.

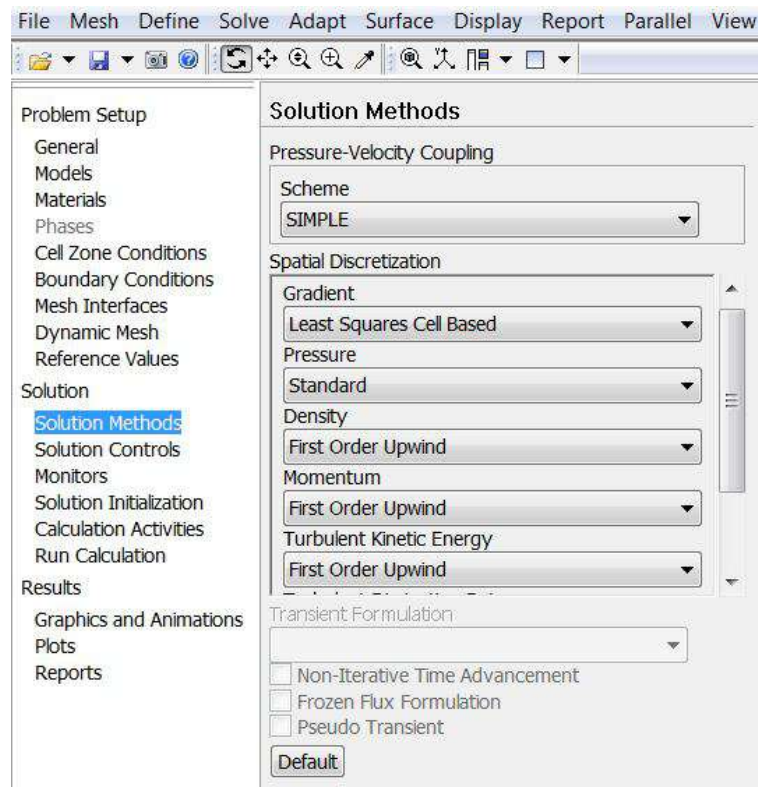


Figure 2.7: Solution Methods

Monitors Fluent software also allows to change the values of residue after each iteration.

The command “Solution” → “Monitors” → “Residual” → “Edit” displays the “Residual Monitors” in which values are assigned to the residual equation of continuity, the components of the speed, the energy equation, coefficient of turbulent kinetic energy k at the rate of dissipation ε . Usually the residual values, except the continuity equation, are on the order of 10^{-6} . The evolution of debris over time, or with each iteration, it can also be displayed on the monitor; to enable this command, select the command is simply “Plot” under “Option” window “Residual Monitors” and indicate the number of the view window under the heading “Plotting Window”



Figure 2.8: Monitors

Solution Inizialization Before starting simulation is necessary initialize the flow field in the entire domain. There are two methods to initialize the system. The first is to Hybrid Inizialization; the second defining initial values in some areas. The “Solution Initialization” sets the initial values for variables and initialize the fluid dynamics solution according to these values. They can be entered manually in the invoked by selecting or area of interest under “Compute From” and through the “Initialize” command you can initialize the system.

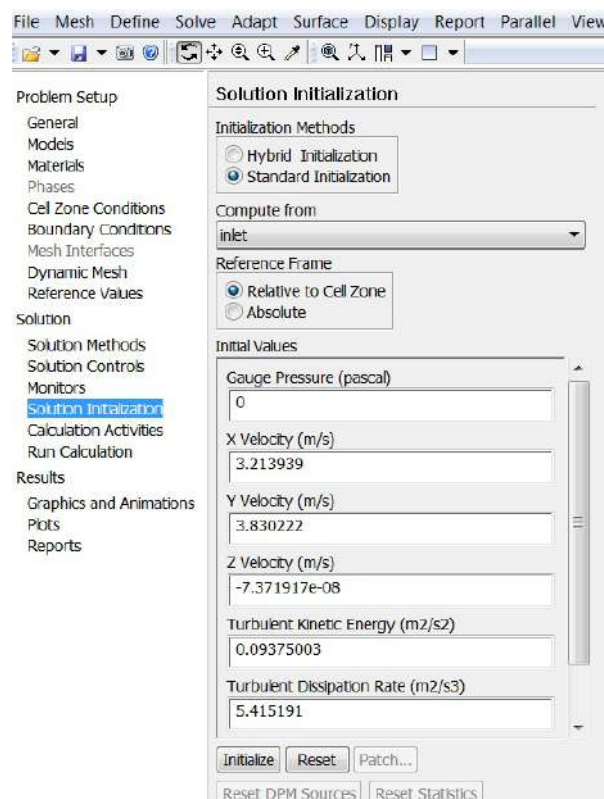


Figure 2.9: Solution Inizialization

Run Calculation Once the initialization phase you move to iterations. Through the command "Run Calculation", "Calculate" button is displayed. In the "Calculate" are requested: the number of intervals of time Δt or "time steps", the duration in seconds of each time interval and the maximum number of iterations for "time step".

Always "Calculate" window you can change the number of updates planned for the "UDF (User Defined Function) for each" time step ". Very important is the choice of time interval Δt . The time interval must be small enough so you don't miss any fast transients; usually the time interval must be less than one order of magnitude of time constant used by modelling system. A valid method choosing the amplitude is based on the number of iterations.

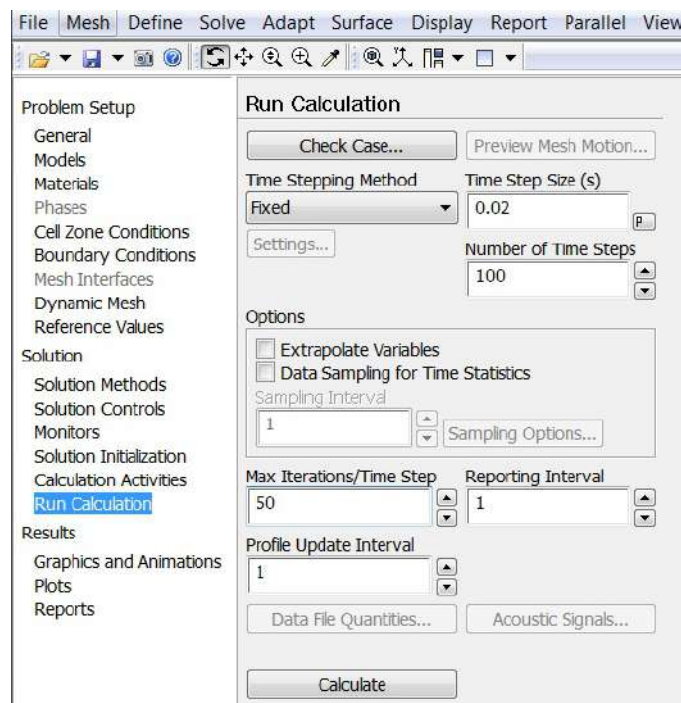


Figure 2.10: Run Calculation

Usually, a proper time interval Solver convergence leads to after about 20 iterations. If the number of iterations is greater need to reduce "time step", if it is less, it is necessary to increase the amplitude of time step ".

In the steady flow case the "Calculate" takes a simpler form. In it are required only the number of iterations, the number of intervals to update the display of charts and the number of updates for each iteration of the UDF. The simulation can now be started by clicking the button "iterate" of the respective window. Many other commands in Fluent software can be activated before proceeding to a simulation.



2.3 Morphing with RBF Moph

2.3.1 Introduction

Morphing is the ability to change one thing into another smoothly. In modern computer graphics 3D character animation uses morphing for several reasons. The movement of characters is usually gained through motion capture techniques while the backbone of the character is considered to be constrained to rigid motions. However the characters are not rigid, with the obvious exclusion of characters representing mannequins or humanoid robots.

Morphing transforms the surfaces of the original model into a new position or shape. In computer graphics however the accuracy of the movement is not important because it just needs to look good. Morphing the mesh required for a numerical simulation is a more complicated and quite a delicate task, especially for a 3D CFD mesh. In this case morphing, also termed smoothing, is not limited to the surface but has to be extended to the entire volume of the mesh and the solver suffers dramatically. Although the concept is

basically the same as morphing in computer graphics, i.e. morphing defines the motion of a set of points and moves them accordingly to the action of a motion field. The way the task is accomplished, depends on which smoothing algorithm is selected and on the definition of the control points criteria which can substantially change the result.

A good morpher is one that preserves the exact shape that the user wants (i.e. it undergoes a rigid motion where there is a steady object and a null rigid motion prescribed) and gently deforms the surface and volume elements that are within the deformation field and minimises the distortion of each element. In general any mesh distortion introduces a reduction in the quality of the mesh, so the morpher is required to minimize this effect, still allowing for significant modification of the shapes.

The presented software is based on a well known technique related to the use of the so called Radial Basis Functions (RBF). The method is based on the use of a system of radial functions to produce a solution for the mesh movement/morphing, from a list of source points and their displacements. This approach is valid for both surface shape changes and volume mesh smoothing. The software allows the user to modify the shape of the surface in a user-friendly way, interacting directly with the graphical representation of the surface and being fully integrated with FLUENT. Then the smoother is capable to reduce the related distortion of the volume mesh and preserve an acceptable mesh quality, which is fundamental for a reliable flow solution. The impact on the CFD process is that, instead of re-generating the mesh, it is possible to specify

several shape modifications (compatible with the mesh topology which cannot be changed) directly in the solving stage inside FLUENT, without the need to step back to the meshing tools and so saving a significant amount of human and CPU time. Moreover the shape changes can be parameterized in order to perform parametric studies of shapes and component positions typical of the fluid dynamics development. Finally the morpher can be coupled with external optimizer tools to perform automatic optimizations.

2.3.2 RBF Morph

RBF Morph is a unique morpher that combines a very accurate control of the geometrical parameters with an extremely fast mesh deformation, fully integrated in the CFD solving process.

RBF Morph is the meeting point between state-of-the-art scientific research and top-level industrial needs. The industrial need was very simple: move an object inside a very large CFD mesh (many millions of cell), preserving the original geometry of the other parts, and preserving the quality of the volume mesh without the need for remeshing, i.e. only by updating the node positions. To complete the specification of such a tool the following extra features were required: working in parallel, handling every kind of volume elements (tetrahedrons, prisms, hexahedrons, polyhedrons, preserving also non conformal interfaces) and working inside ANSYS Fluent with a simple interface.

The aim

The aim of RBF Morph is to perform fast mesh morphing using a mesh-independent approach based on the state-of-the-art RBF (Radial Basis Functions) techniques. The use of RBF Morph allows the CFD user to perform shape modifications, compatible with the mesh topology, directly in the solving stage, by just adding one single command line to the input file. The most important requirements are:

- a mesh-independent solution;
- parallel morphing of the grid;
- large size models (many millions of cells) to be morphed in a reasonable short time;
- and management of every kind of mesh element type (tetrahedral, hexahedral, polyhedral, prismatic, hexcore, non-conformal interfaces, etc.).

The final goal is to perform parametric studies of component shapes and positions typical of the fluid-dynamic design such as:

- design developments;
- multi-configuration studies;
- sensitivity studies;
- DOE (Design Of Experiment);
- and optimization.

Usage Steps

RBF Morph is based on a three-step procedure:

- SERIAL: general setup (definition of the source points);
- SERIAL: solution of the RBF system;
- SERIAL or PARALLEL: morphing of the surface/volume mesh.

The serial setup requires an extensive use of the RBF Morph GUI. The GUI offers all the tools required for the definition of the problem. It is composed by a main panel that, acting on the radio buttons on the left, offers several different operative modes.

The first four panels (*Config, Encaps, Surfs, Points*) are addressed to the problem setup and the definition of the source points; *Solve and Multi-Sol* panels allow to calculate or combine the RBF solutions; *Preview and Morph* panels let the user to preview or apply the morphing to the mesh; in the CAD panel is possible to apply the morphing modification to a given STEP CAD file; the *Tools* panel contains various utilities and settings.

After completing **Step 1** it is possible to proceed to **Step 2** and calculate the RBF solution. The effect of the solution (often referred as *modifier or parameter*) can be verified using the Preview feature. It allows to pre-morph on-the-fly an arbitrary number of surfaces, without altering the actual mesh, overlaying them on the current graphical viewport. In alternative, it is also possible to morph the actual mesh, check the result in terms of shape deformation, mesh quality, etc. and then to go back to the original

configuration using the Undo capability. Once that the solution is satisfactory, it can be saved on file. The operation is then repeated for each desired modifier.

Step 3 can be performed in serial or in parallel with or without the GUI. Once the solutions are available, they can be loaded and used to morph the mesh using the *Morph* panel of the GUI or using TUI commands that allow to prescribe a single morph, a multi-morph or a sequential morph, combining the effect of multiple modifiers. Given that each modifier can be applied with the desired magnitude (*Amplification*), a parametric FLUENT model results.

2.3.3 conclusion

In this chapter RBF theory has been presented showing how these methods can be used for a very efficient solution of one of the most challenging application in the field of CFD: mesh morphing and smoothing of very large models. The complete research path from the industrial need to a software solution has been discussed. The RBF based approach for mesh morphing and smoothing has proven to have one of the best performances of any techniques in this field. The main reason is that it combines several advantages that are individually the key strength of the other methods: mesh quality preservation, the ability to handle very complex free form shape modifications, the ability to exactly prescribe rigid movement, element type independence, parallelism and reusability on different meshes.

The importance of a high quality and very fast mesh morpher in CAE is clear and the benefit that comes from its application strongly relates to the size of the problem. For small problems, where a rebuild of the mesh is not an

issue, the main advantages are

- the simplicity of the geometry parameterization;
- the flexibility that make it good to use for multi-physics problems

These advantages are linked to the ease of synchronization of different calculation models to the same geometry. For medium sized models, where the CAD to model automation is still possible, an extra benefit comes from saving calculation time in rebuilding the mesh. For large problems where this automation is not feasible the benefits are very high:

- a "what if" analysis can be performed with minimum effort for the set-up (hours vs days are required for the rebuild of a new model);
- a parametric analysis (for the shape optimization or for the set-up of an investigation with adjustable parts) that with the traditional approach is limited to a small number of solutions.

Chapter 3

Model and Problems

This thesis describes the steps and the results of a shape optimization project for an Exhaust System. The CAD geometry of the model provided by Ansys UK, realized with CATIA V5 to help accomplish the vision by providing integration of computational fluid dynamics (CFD) into product lifecycle management (PLM).



Figure 3.1: Cad Design with CATIA

It is a manifold exhaust system cutcell-mesh for a four-engine cylinders, typically used in the automotive industry, and it needed to be optimized reducing the *pressure drop* between the inlet and the outlet. As could be seen in fig.(3.1), the four-cylinder flow is conveyed in the manifold and must follow a curve approximately $2,44 \text{ rad}$.

In addition to knowledge and data management capabilities, PLM companies need fully generative relationships between the design of their manufacturing-ready geometric models and their mathematical simulation models, which include structural, thermal, and CFD analysis. Simply having the CFD model show up in the same graphics window as the CAD model is not enough, says ANSYS. The CFD data has to fully reside in the PLM software's data management system so engineers can turn their CAD models into flow models, and perform knowledge-based optimization studies.

Despite this is a very innovative idea, using RBF-Morph is even more simply, intuitive and practical, directly inside Fluent in consequence of CFD simulation.

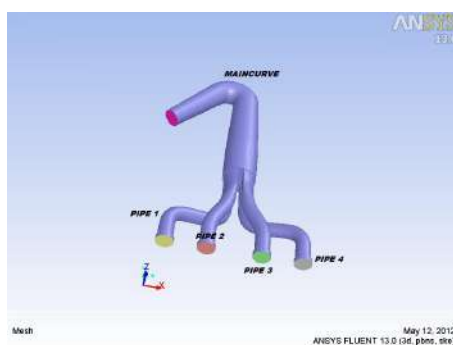


Figure 3.2: Manifold cutcell mesh

3.1 Mesh

Grid generation is of extreme importance in CFD, and can be defined as the process of breaking the physical domain into small discrete sub-domains, in order to calculate numerical partial differential equations on a discrete number of points. Cutcell mesh has the main advantage is that you can easily (push button mesher) get a predominantly hexa mesh with inflation layers to capture your viscous boundary effects. Studies have shown that the solver likes the mesh and converges more quickly and accurately than with tetra/prism meshes elements.

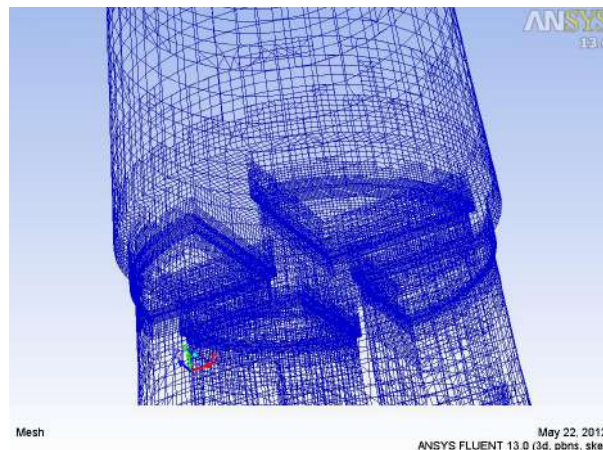


Figure 3.3: Manifold cutcell detail

The full model of the system was meshed so that the flow entering the duct inlet would be representative of the actual flow rather than just entering a flow velocity at the duct inlet.

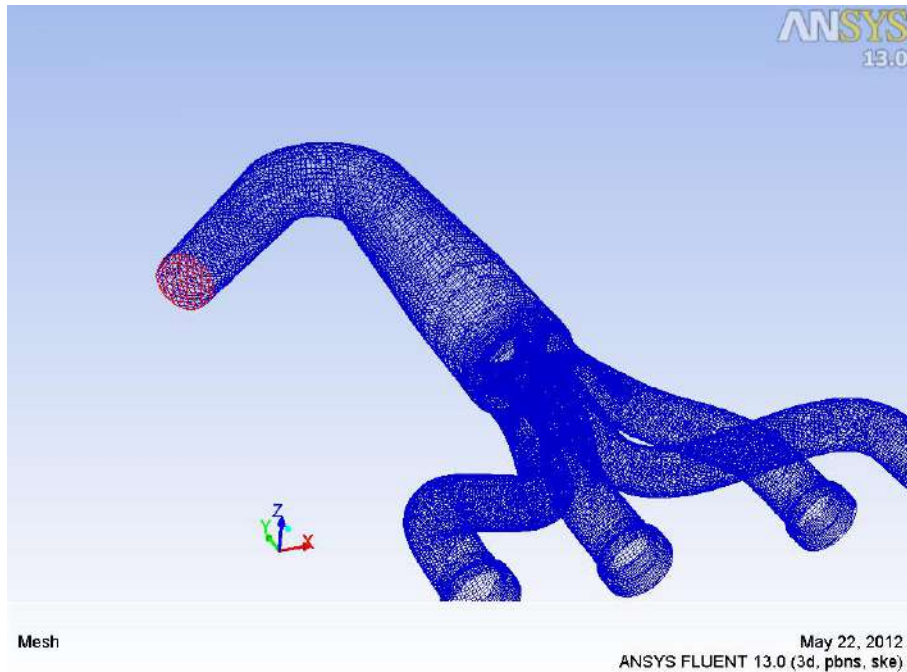


Figure 3.4: Manifold cutcell mesh 2

The mesh created has the following features

- 397892 Mixed Cells
- 1252785 Faces
- 447991 Nodes.

In addition the *mesh check* give us some information about it:

1. The Domain is Extended in
 - x-coordinate from $-2.059723 \cdot 10^{-01}m$ to $1.699978 \cdot 10^{-01}m$
 - y-coordinate from $-1.000000 \cdot 10^{-02}m$ to $6.058343 \cdot 10^{-01}m$
 - z-coordinate from $-8.099721 \cdot 10^{-02}m$ to $1.999888 \cdot 10^{-02}m$

2. Volume statistics

- minimum volume = $6.020139 \cdot 10^{-13} m^3$
- maximum volume = $9.222114 \cdot 10^{07} m^3$
- total volume = $3.033195 \cdot 10^{-03} m^3$

3. Face area statistics

- minimum face area = $6.570930 \cdot 10^{-09} m^2$
- maximum face area = $1.470299 \cdot 10^{-04} m^2$

The surface is divided into 6 surfaces:

- inlet 1,
- inlet 2,
- inlet 3,
- inlet 4,
- outlet,
- wall-solid.

3.2 Fluent Settings

The procedure for definition of fluid-dynamic model, implemented for the simulations conducted with Fluent software, is the following.

The fluid used in simulation is air considered ideal gas. The operating conditions are those atmospheric. The equations of continuity and momentum of are coupled. The mathematical model is the viscous $k - \epsilon$ (turbulent flow), two differential equations, one for the turbulent kinetic energy k , the other to define the ϵ turbulence. The boundary conditions are adiabatic walls and external environment at a temperature of 300 K.

- viscous standard $k - \epsilon$ model with standard wall function (Fig 3.5);

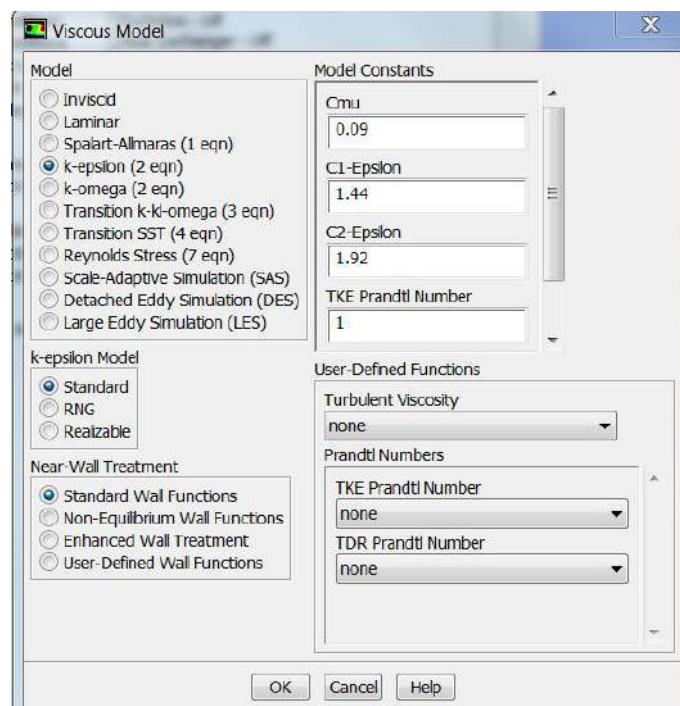


Figure 3.5: Turbulence Model

- boundary condition for the inlet is velocity-inlet. Because the model has four inlet, then four parallel simulations are performed, making flow of gasses in conducts once at a time, setting others as wall (Fig 3.6)

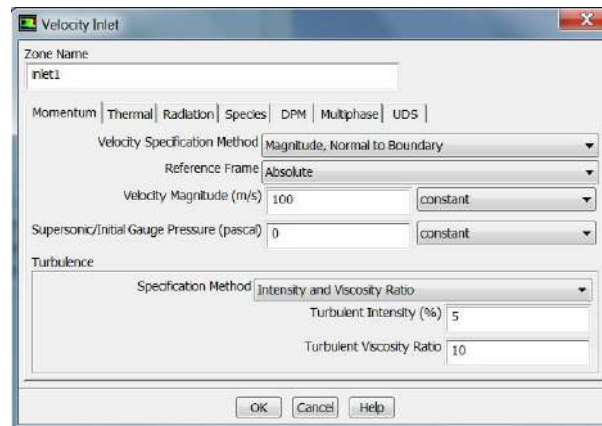


Figure 3.6: Velocity-Inlet BC

- boundary condition for the outlet is Pressure-Outlet, and the Gauge pressure is set to 0 Pa, cause we are interested only in Pressure Drop.

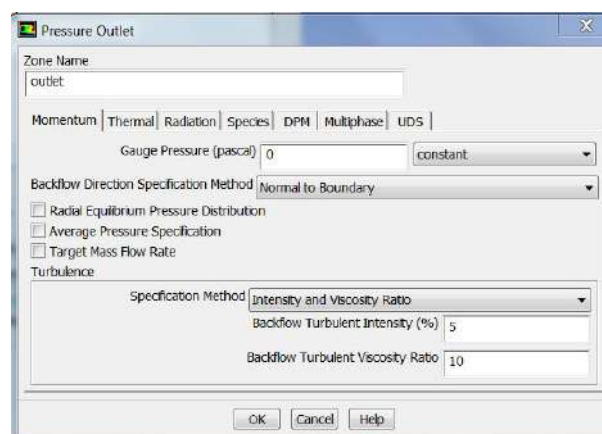


Figure 3.7: Pressure-Outlet BC

- Boundary condition for the exhaust surface is Wall, with shear condition "No Slip" (Fig 3.8)

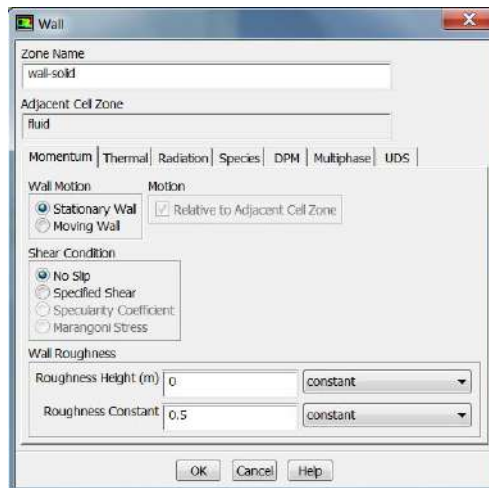


Figure 3.8: Wall solid BC

- The parameter controlled is the Static pressure on the Inlet, where gas is flowing in the simulation (Fig.3.9).

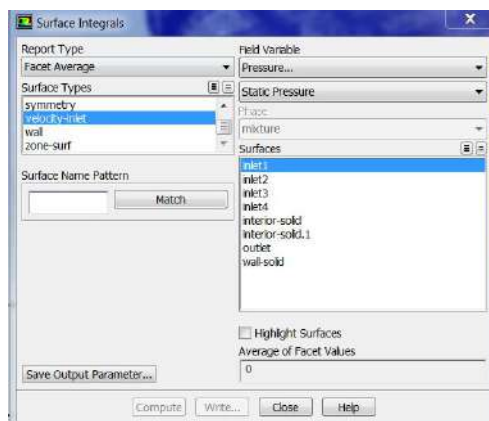


Figure 3.9: Output Parameter

- After set the boundary condition, must be set the Solution Method:

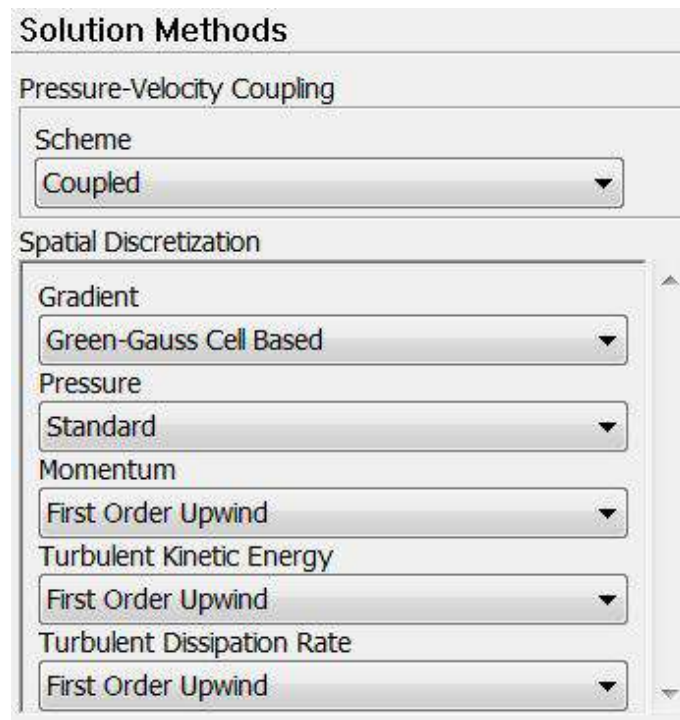


Figure 3.10: Solution Method

- The finally operations are:
 - set Residuals Monitors to 10^{-3}
 - Solution Initialization with Hybrid Initialization
 - and then Run calculation with 100 number of Iteration

3.3 Baseline Results

Running four simulation, one for every inlet, we obtain that the Pressure Drop isn't homogeneous for the four Pipe. The simulation results are the following:

The difference between the first pipe and the third pipe, is about 2108 Pa ,

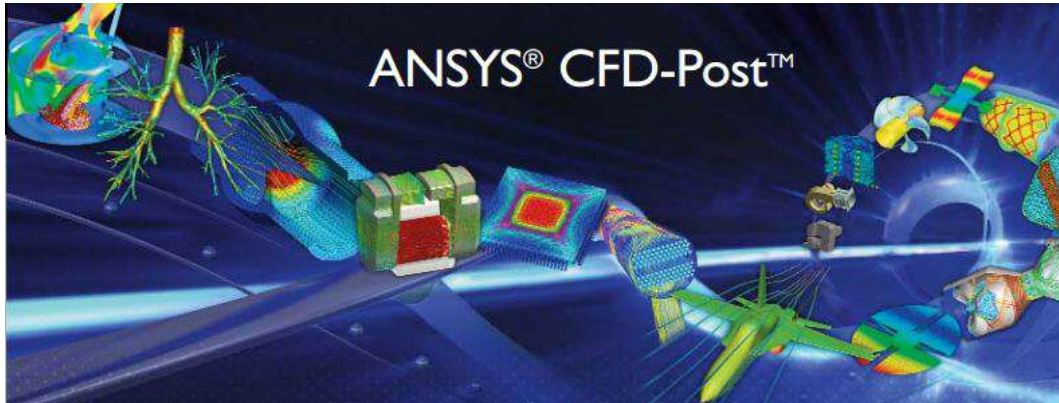
Pressure Drop	Inlet 1	Inlet 2	Inlet 3	Inlet 4
Static Pressure (Pa)	21658,3	20507,1	19550,8	19853,4

Table 3.1: Baseline Results

i.e. 20%, too much. Even more troubling is the difference between the first and the fourth pipe i.e. 2075 Pa and between the second and the third pipe almost 1000 Pa .

Apparently we can't explain this big difference, whereas the paths less of Maincurve, at the bottom of the duct, are virtually symmetrical two by two, we should have the same pressure drop at least between symmetric pipes. So we must import the result, obtained in Fluent, in CFD-Post, and using Ansys Workbench this is so immediate, creating a standalone system, with analysis system *Fluid Flow Fluent* toolbox. In this way it is automatic keep linked in sequence Geometry, Mesh, Setup, Solution, and Results, of our Project, despite those part are managed with different software. Using the Workbench platform, after calculated the solution with Fluent, it is necessary only refresh the project and click on Results and open CFD-Post to post-processing the result and understand why the four pressure drop are so different.

3.4 CFD-Post Processing



CFD simulations don't end with the fluid flow prediction. To benefit from the prediction requires post-processing that provides complete insight into fluid dynamics simulation results. ANSYS CFD-Post software, the common post-processor for all ANSYS fluid dynamics products, delivers everything needed to visualize and analyze fluid dynamics results. These powerful capabilities include image generation to communicate results visually, qualitative post-processing to display and calculate data, automation to ease repetitive tasks, and the ability to run in batch mode.

The abstract of this step is understanding how is possible a significant pressure drop difference between the four pipe, and in particular between the symmetric pipe.

Now we must to check the contour of Pressure in the Exhaust System and the Stream Line. After this, it is need to found the points where there are high pressure loss, and meanwhile the points where there is a posting flow.

Opening the CFD-Post there is the mesh on the right, the toolbar on the top, and some control panel on the left:

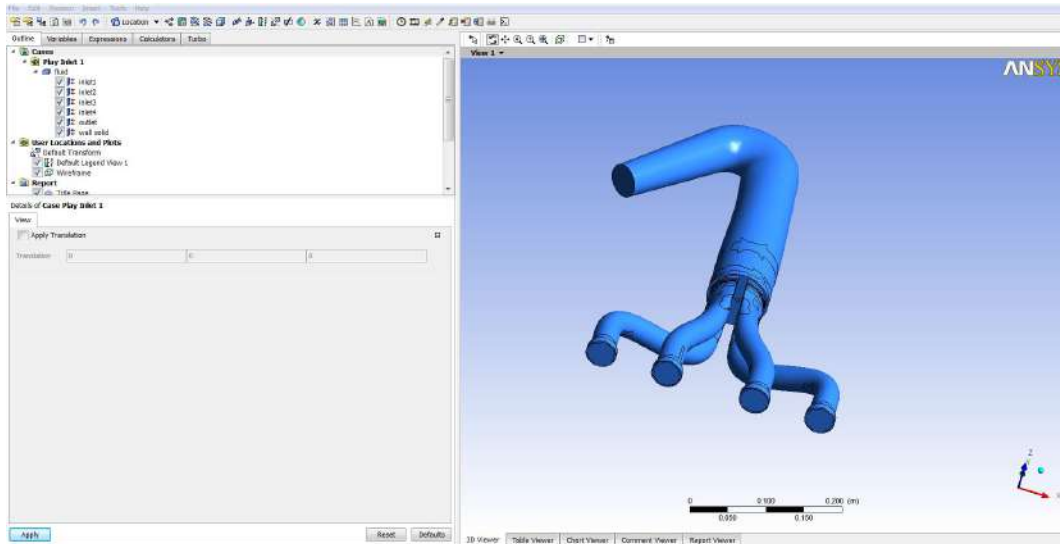


Figure 3.11: CFD Post Window

Clicking on Pressure Contour, will be open its relative control panel.

Now it must be set:

- Domains: All Domains;
- Location: wall solid
- Variable: Pressure;
- Color scale: Linear.

and we'll get the Contour Map of the Pressure all over the wall.

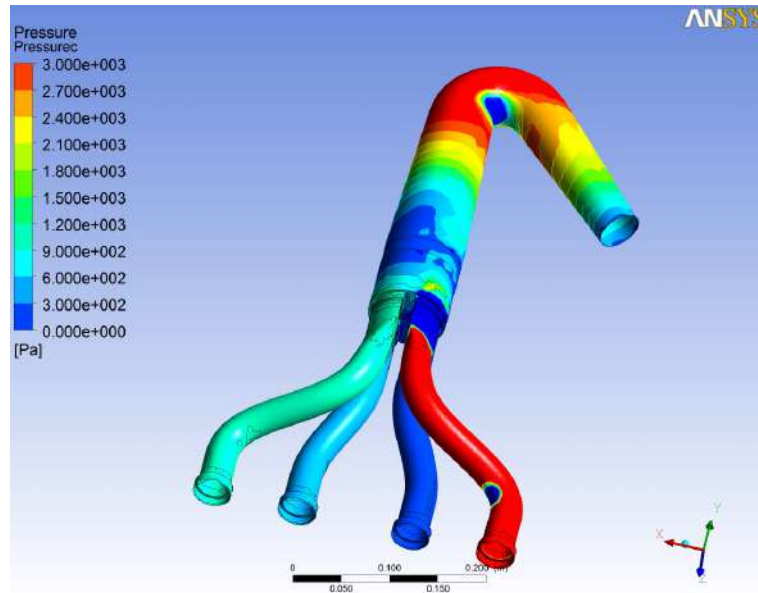


Figure 3.12: Contour of Pressure Inlet1

Same thing must be done to see the Streamline. Clicking on Streamline button it will be opened its control panel. In this case the set are

- Type: 3D Streamline;
- Domains: All Domains;
- Start From: Inlet 1;
- Number of Points: a representative number of Streamline to see what want we see, for example 50;
- Variable: Velocity;
- Direction: Forward....and Apply!

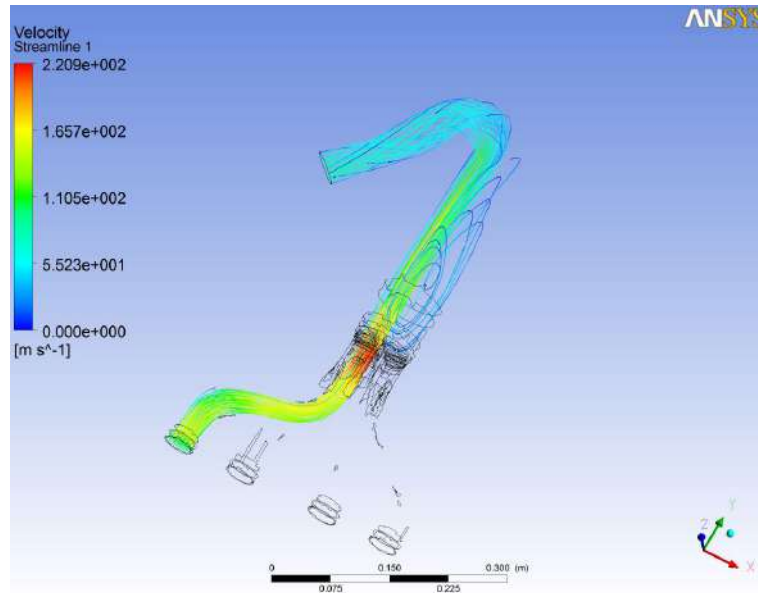


Figure 3.13: Stream Line, Inlet1

By analyzing the baseline streamlines, a turbulent zone clearly appears after the flow enters the main pipe, as well as an hint of vorticity occurs when the stream met the main curve..

Watching the Contour of Pressure is easy to see that there is a Pressure Loss in the inner side of the first curve of the pipe, before entering in the Manifold. And a more big Pressure loss even in the inner side of the Main Curve. In addition the most important Pressure Drop is in the Manifold, where there is a increase of Pressure Concurrently with the turbulent area seen in Fig. 3.13.

Doing the same process, there are checked the Streamline and Contour of Pressure in the other three cases, streaming the other three inlet.

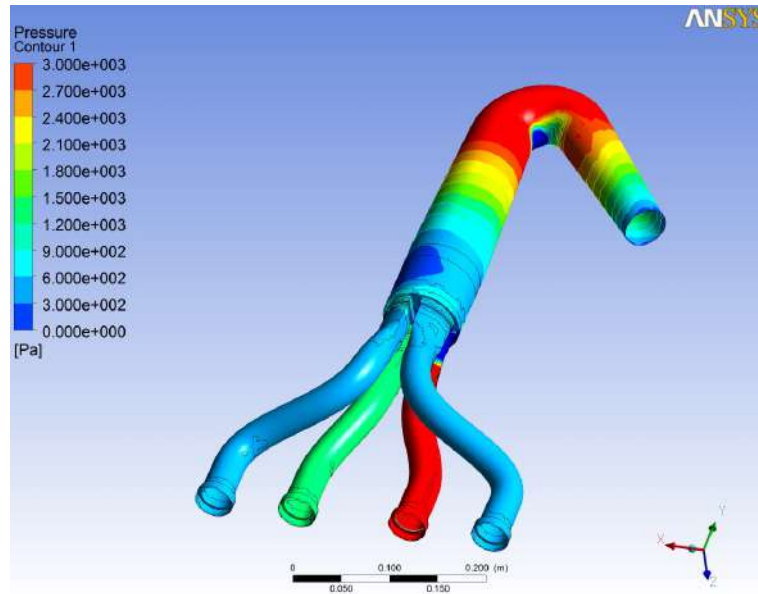


Figure 3.14: Contour of Pressure Inlet2

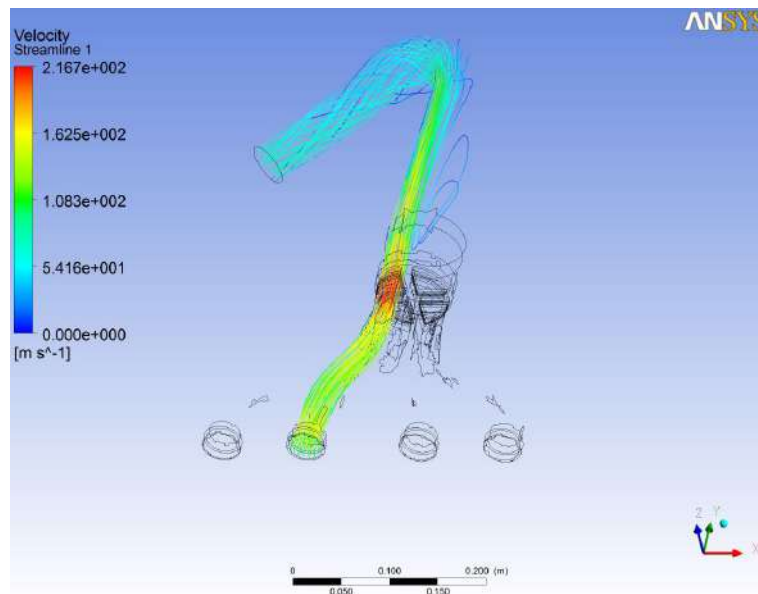


Figure 3.15: Stream Line, Inlet2

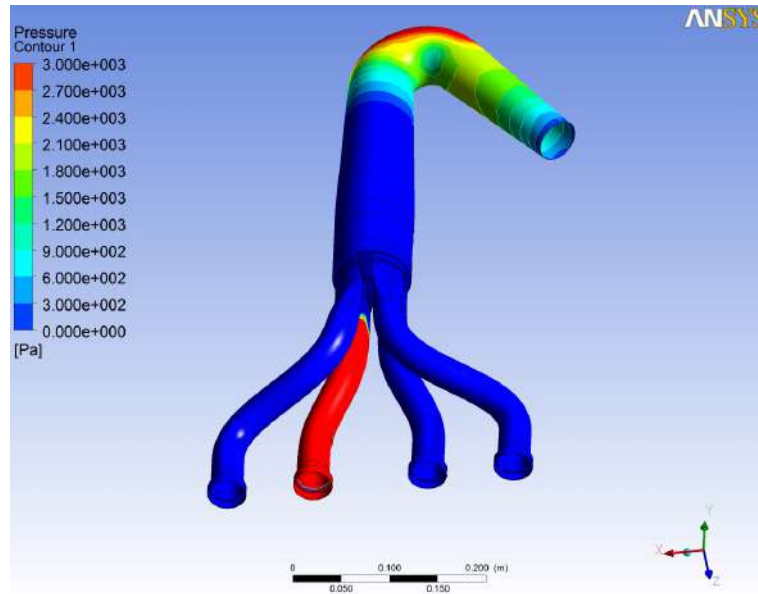


Figure 3.16: Contour of Pressure Inlet3

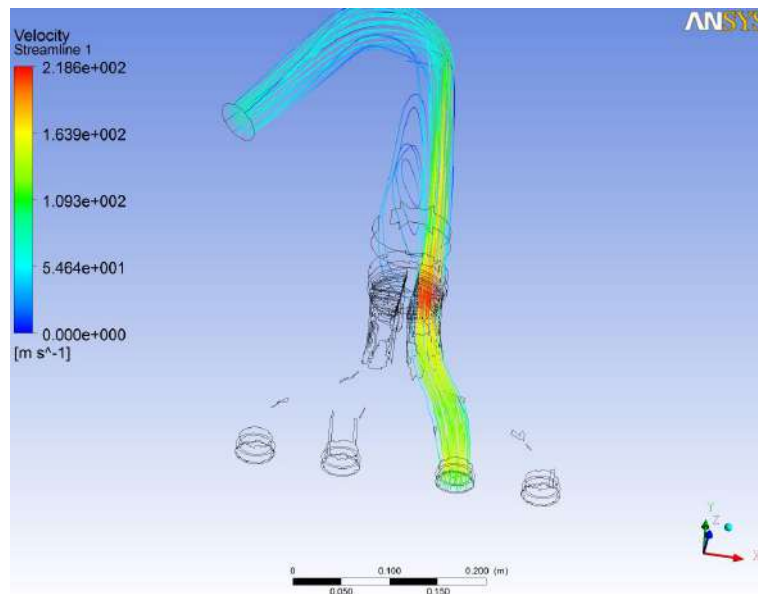


Figure 3.17: Stream Line, Inlet3

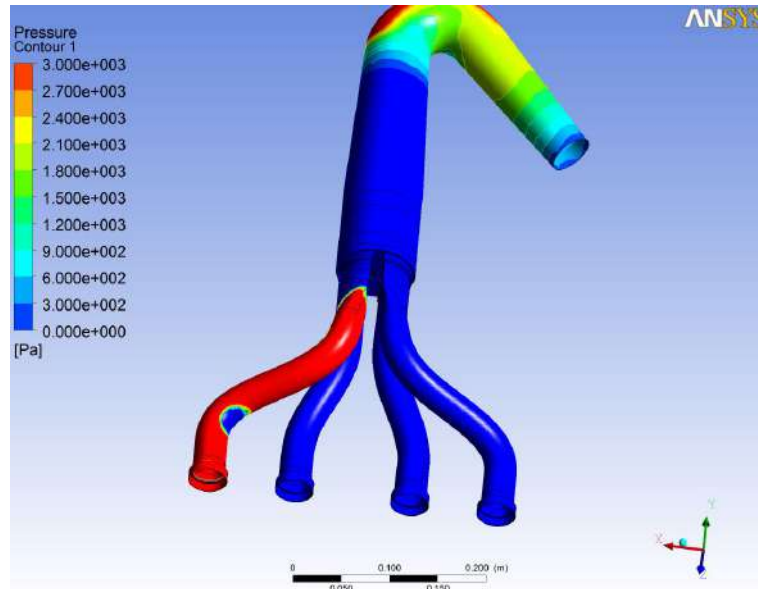


Figure 3.18: Contour of Pressure Inlet4

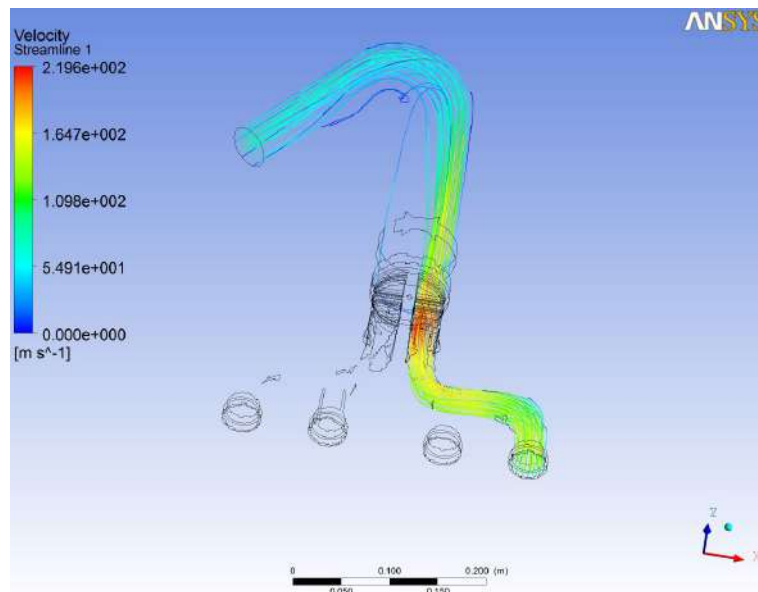


Figure 3.19: Stream Line, Inlet4

3.4.1 Analyzing results

The vortex area in proximity of the Main Curve is more accentuated in the simulations where the flow come from Inlet 1, then from Inlet 2, and less from Inlet 4 and Inlet 3. All four case show that there is consistency between Numerical Results, and Post-Processing Results.

In fact, flow that comes from the Inlet, 1 and 2, has more Pressure Drop rather than the flow comes from Inlet, 3 and 4. This can be easy explained because of, intuitively, the development of pressure and flow lines depending on how the flow “sees” the curve;

i.e. in the cases 3 and 4 the flow follows the exhaust geometry, while in cases 1 and 2, the flow improperly impacts against the curve generating vorticity.

It is also noteworthy that in cases 1 and 4, where the curve made by the flow is more pronounced than the other two cases, there is a Pressure loss, visible in contour of pressure image, as a blue zone within a red zone. Appropriate amendments must be done.

The answer to the question due to read only the pressure jumps, in four cases, has now been comprehensively expressed, with the help of graphical reports. The new question is how to standardise the pressure drop, and at the same time lower it, optimizing the geometry?

Chapter 4

Morphing Solution

In the last chapter was shown the type of problem to be addressed.

There isn't a traditional way to do this. The "literature" doesn't speak about problem like this. Only thing to do is developing some sensitivity about the problem, making different tests, observing the results about it, and take considerations how much a specific shape modification affects our parameters of interest.

As already mentioned, vorticity is the most important reason of the pressure loss, and this is caused by the different way in which the streamline arrive at the curve.

Another important reason of the pressure loss, is the turbulence.

This happens because of exhaust gases flow entering the manifold lies in duct with section four times larger than the pipe from which arrives, and this causes fluid vein detachment.

Following these assumptions, in the next few pages will be submitted the manner in which it was developed the project; choosing carefully the sensitivity tests to perform, and showing how they have been developed through RBF-

Morph, and highlighting as this system would speed up the time in the design phase, and get better results, without returning to edit case, starting with geometry by re-executing mesh, and whole process.

4.1 Test 1: Swelling & Necking Curve

The first solution implemented is swelling and necking the duct in the Main Curve.

This solution is designed to reduce pressure losses, due to how the curve is traversed by streams coming from four cylinders. In addition is noteworthy to emphasize that it is not obvious that the excellent solution is the one with a constant section along the curve, on the contrary in the most of case it isn't the best solution.

The main goal is avoid separation of the flow.

4.1.1 Setup Details

RBF-Morph can be open in the Menù “*Define*” → “*RBF-Morph*” First thing to do is defining a domain. It is possible define a domain in three ways:

- box;
- cylinder;
- sphere.

In this case is useful define a box domain;

meshes out of domain are non affected by morph solution.

Define a box is done defining maximum and minimum coordinates along the principal axes. It is now obvious observe that it is possible only have a box along xyz axes oriented.

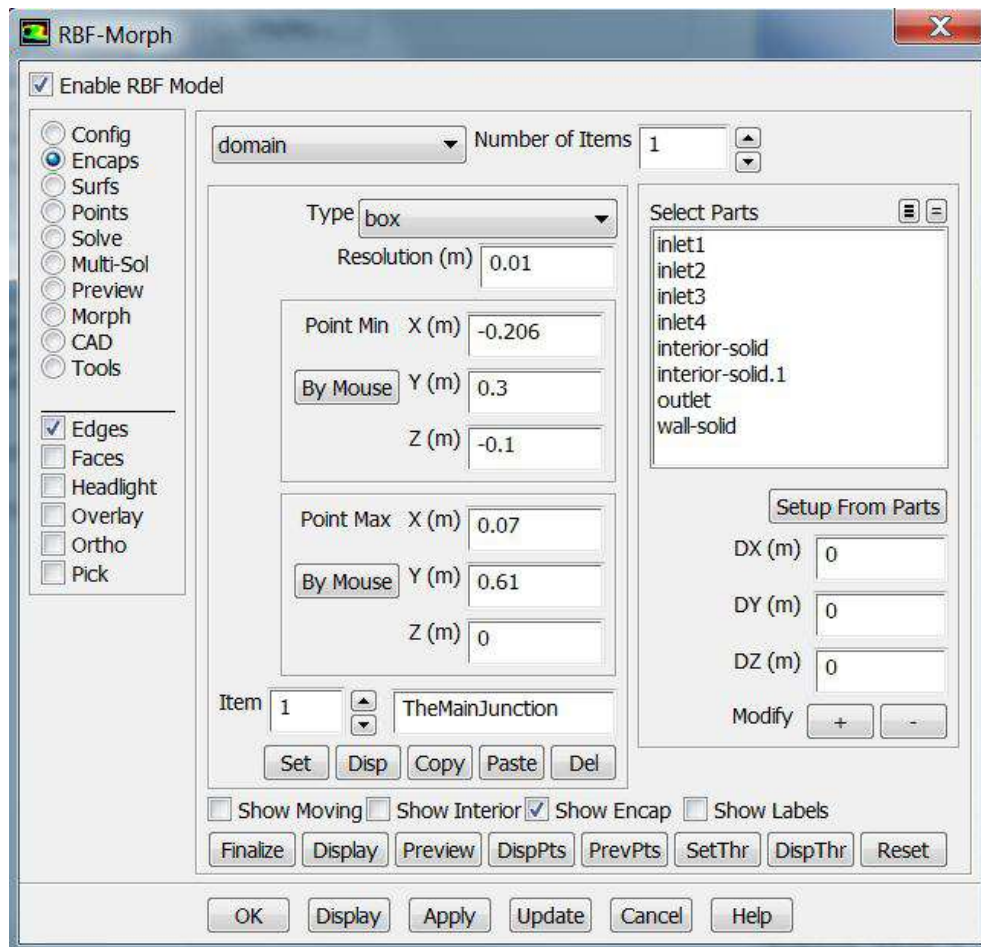


Figure 4.1: Domain Set Necking

In addition it is possible verify graphically if the coordinates setted are or not consistent.

It is possible clicking on *Set* and then on *Disp*, and in Fluent window, will

appears box setted previously, coupled with the mesh: allowing us to have a better understanding of the assembly.

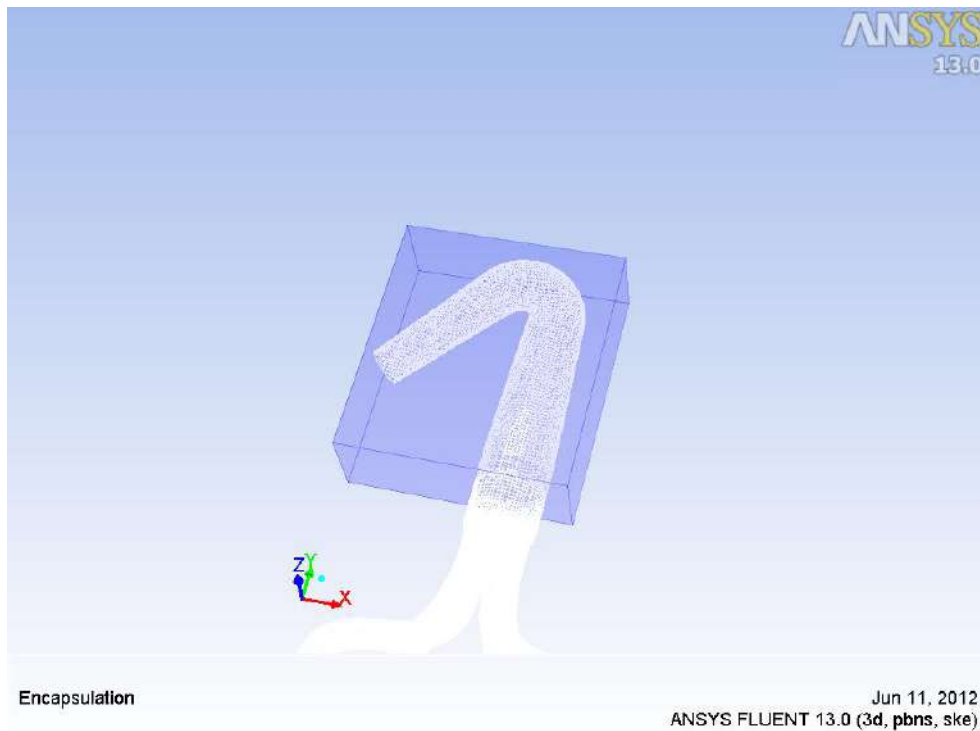


Figure 4.2: Domain Display Necking

Chosen domain you want to modify, it must set geometry modification. This can be achieved by inserting a solid in *Moving* panel.

In this case, is useful to put cylinder near the curve, oriented with the axis orthogonal to the plane on which the curve lies and assigning rigid motion in the direction of the curvature centre.

A cylinder is simply defined through two points and radius.

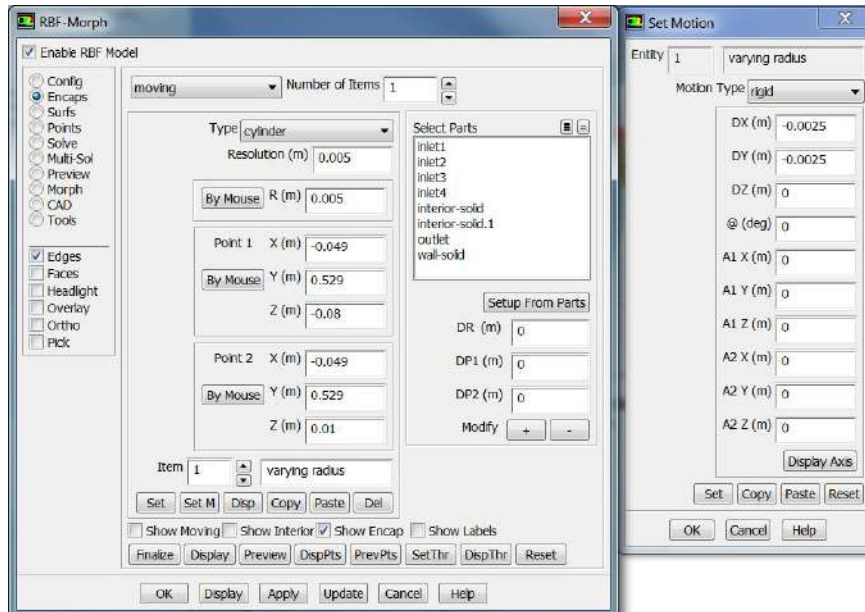


Figure 4.3: Moving Set Necking

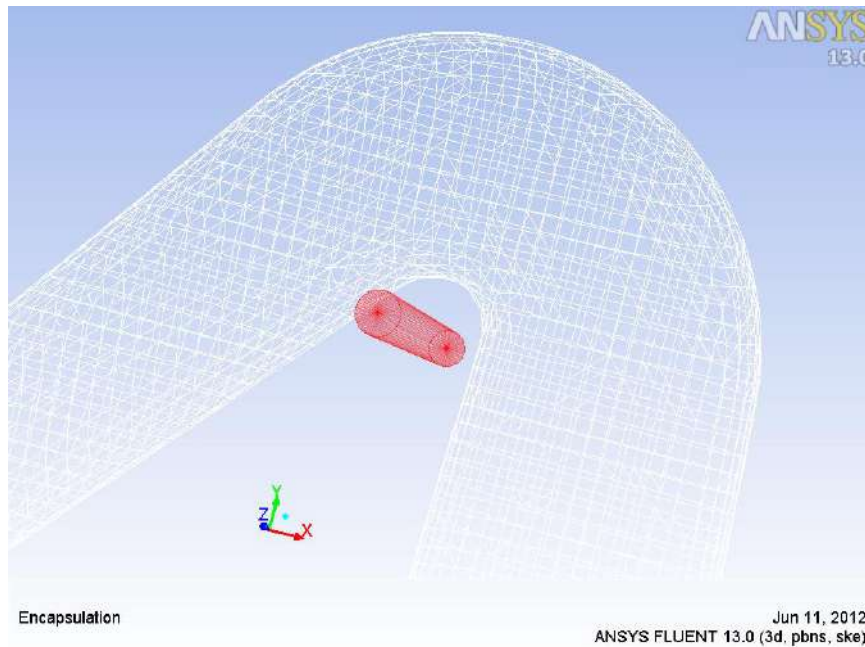


Figure 4.4: Moving Display Necking

Verified all parameters are what we wanted, it is required to “Finalize” what we just set. We can now “Display Points” involved in shape modification, and display “Preview Points” i.e. motion assigned in “Moving” panel.

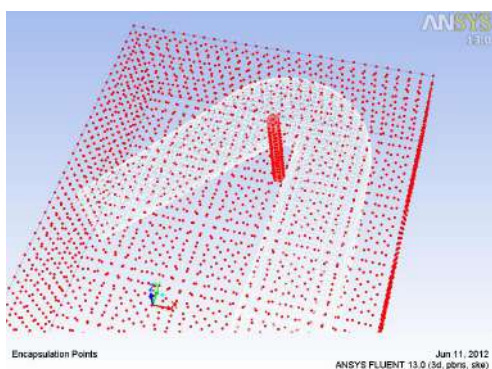


Figure 4.5: Display Points Necking

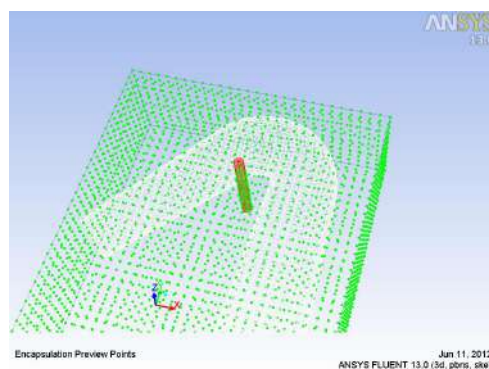


Figure 4.6: Preview Points Necking

Once all settings of solution set-up are completed, it is possible to switch to the solution panel by selecting “Solve” in the Main Sidebar.

The panel shown in following figure appears. After pressing the “Source Points” button, all source points are collected, the “Solution”, “DispPts” and “PrevPts” buttons become active, and some information are printed in the Fluent shell.

Final step is Morphing the geometry and it is possible now choosing “Morph” in the sidebar and pressing the “Morph” Button.

It’s also possible to see preview geometry in the panel “Preview”, inserting code “range 0 1 11” and click first on “Init” and than “Play”. So it will appear the animation of transition from initial to final mesh, through 11 frames.

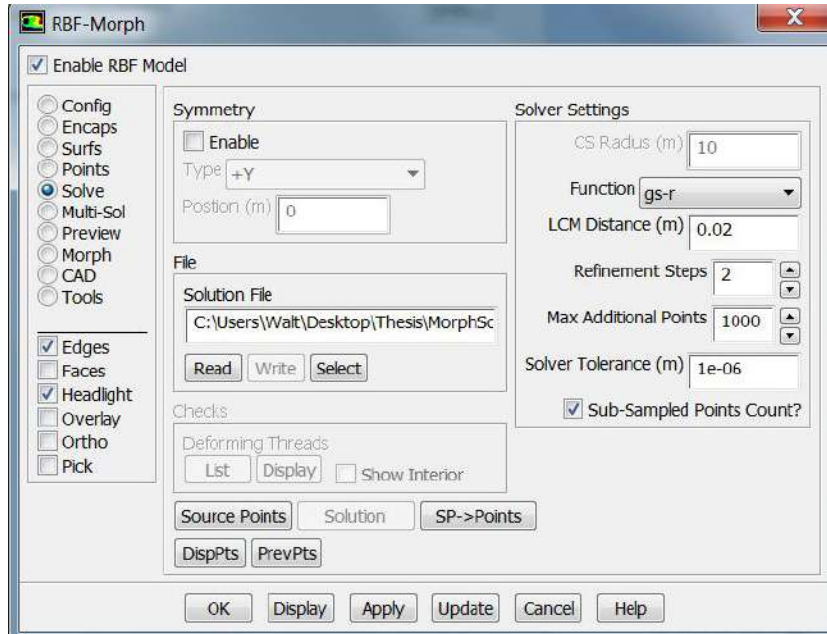


Figure 4.7: Solve Panel

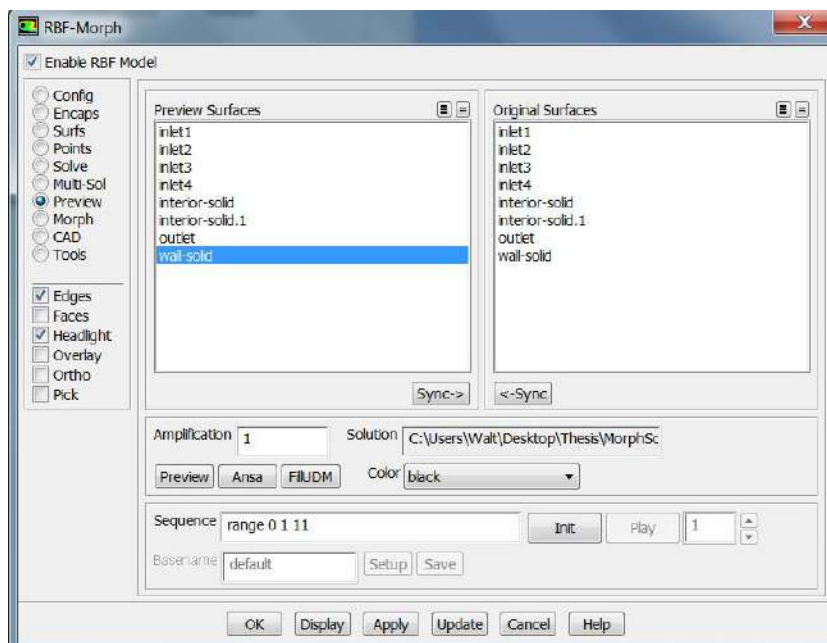


Figure 4.8: Preview Panel

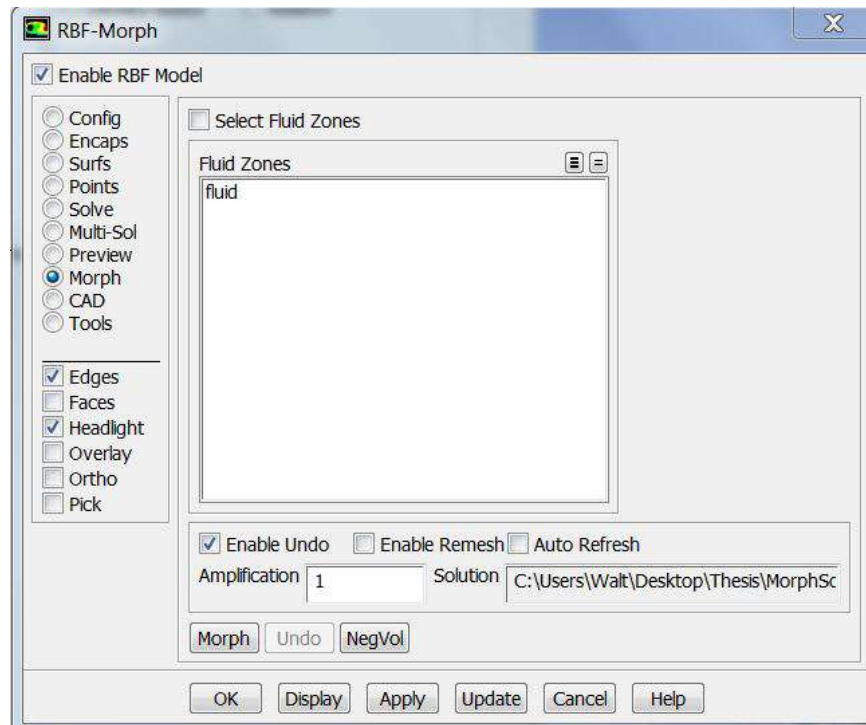


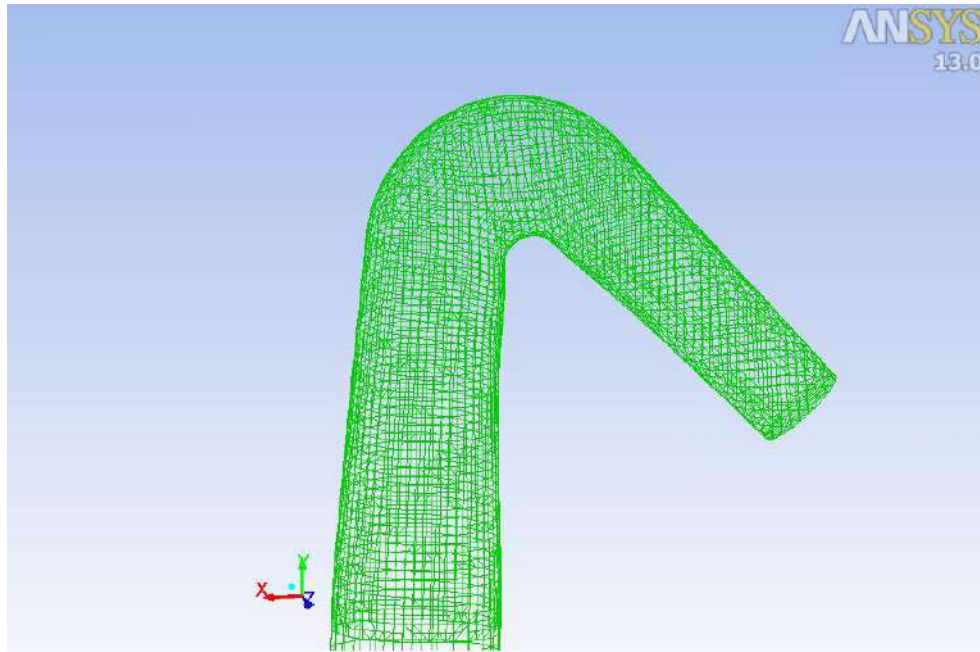
Figure 4.9: Morphing Panel

4.1.2 Mesh Morphing

Morphing or simply previewing, the next step is Display morphed geometry, amplifying solution within the project limits.

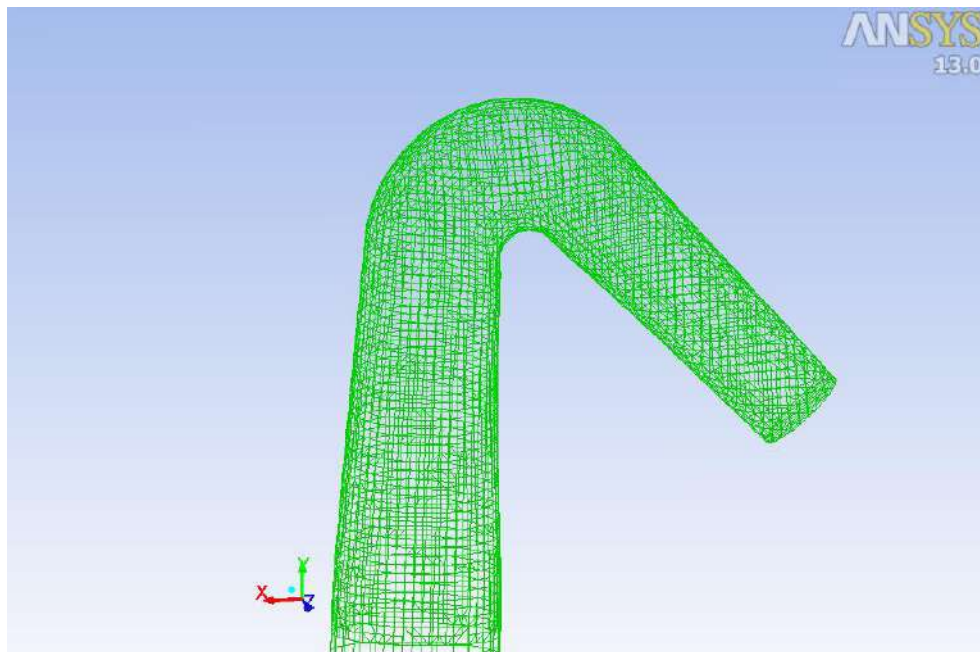
Observe that doing only a preview there are no negative volume cells detected, and is impossible detect errors.

In opposite doing morph process, the geometry will be modified and it will be displayed mesh-morphing errors, if there are.



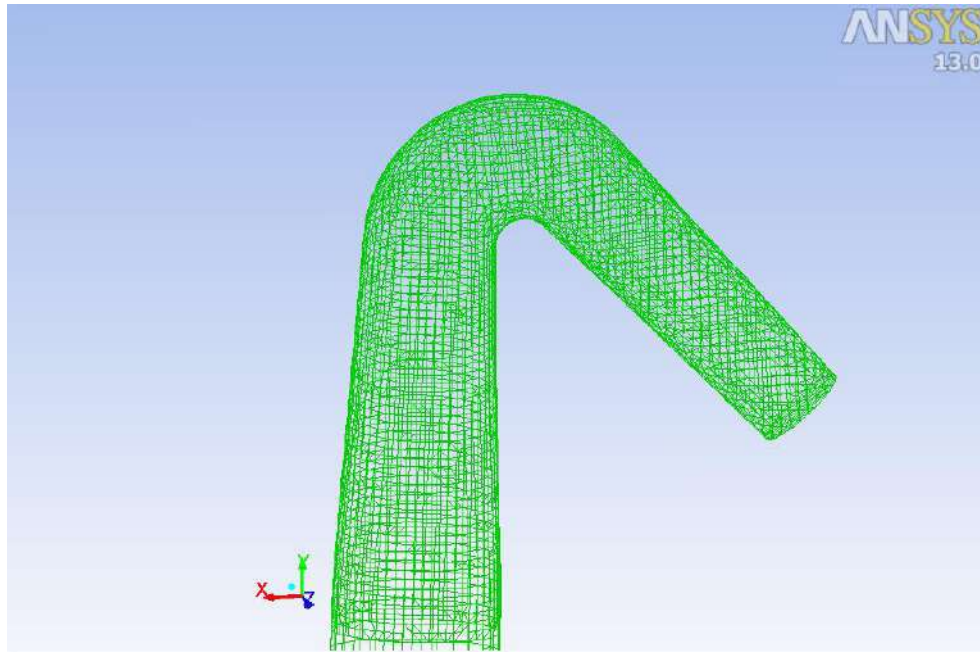
Morphing Preview (A=3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



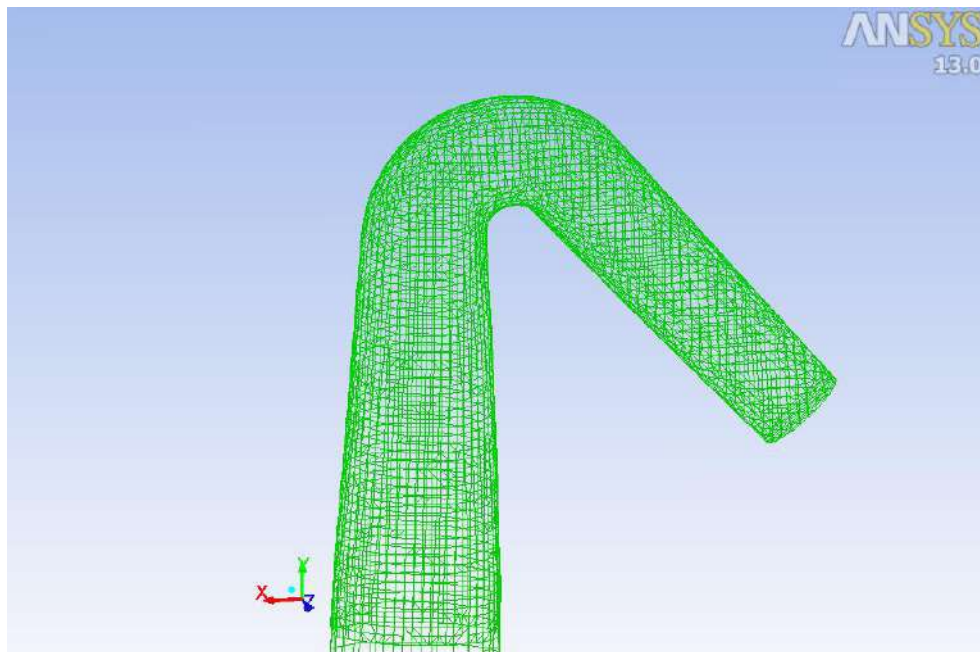
Morphing Preview (A=2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



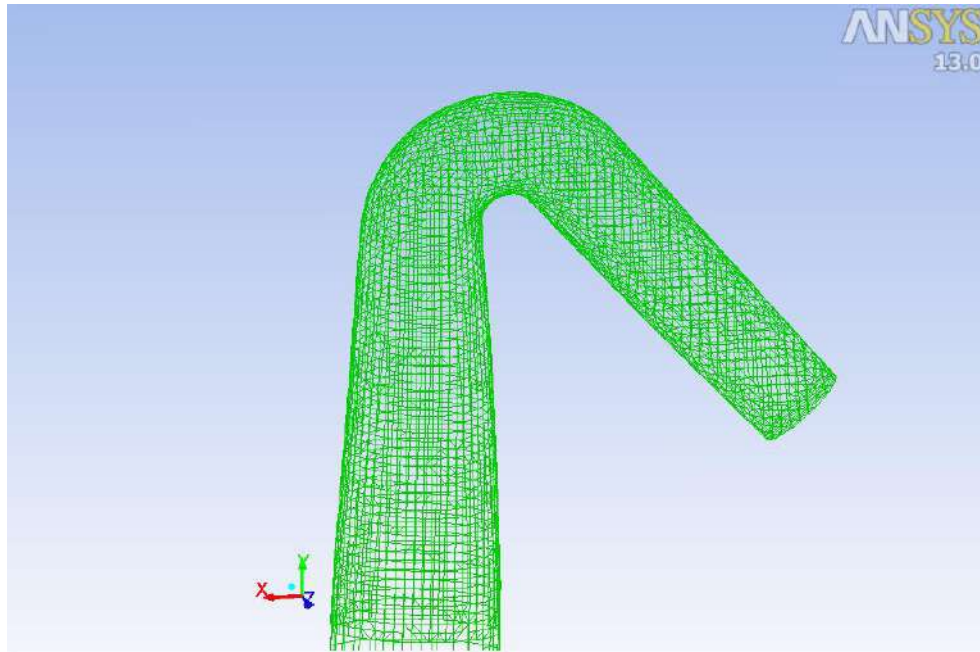
Morphing Preview (A=1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



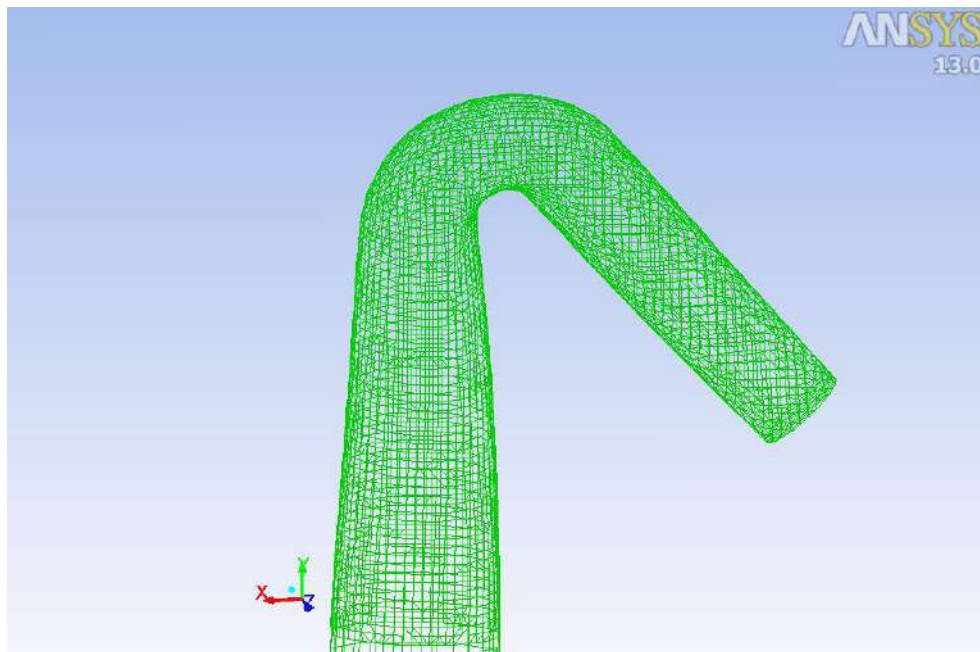
Morphing Preview (A=-1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)

4.2 Test 2: Decreasing and Expanding section

Another solution to implement is decreasing manifold section.

In the last chapter it was underlined that a big part of pressure loss is caused by turbulence phenomena. In particular big difference between single pipe section and manifold section could cause separation of flow. For this reason a shape optimization could be decreasing section of manifold, shortly after the four streams converge in the main. To see better, how this is true, giving a negative amplification to the Morph, manifold section will be expanded, and ensues a higher pressure drop.

4.2.1 Setup Details

Even in this case is useful define a box domain; setting and display are the following

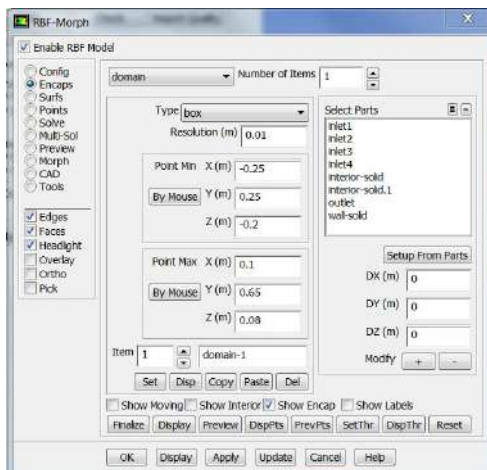


Figure 4.10: Domain Set

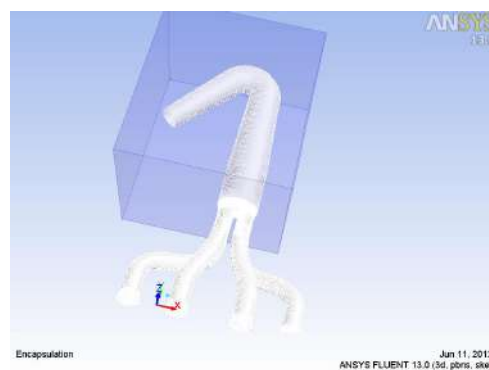


Figure 4.11: Domain Disp

In contrast to the previous, the “Moving” is setted through “*Surfs*” Panel. In this panel it is possible select a surface, previous defined, and apply a motion, without define box, cylinder or sphere. Motion applied isn’t a rigid one, but a scale motion.

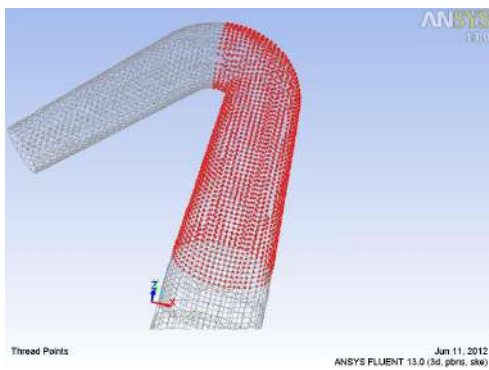


Figure 4.12: Surface Point Display

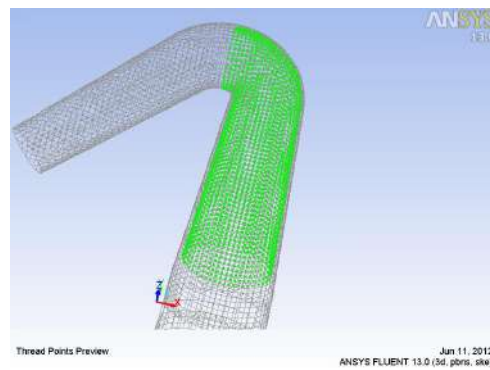


Figure 4.13: Surface Point Preview

To define a scale motion it must be indicated two axis, the third is auto-generated, a origin point, and the proportions applied along the orthonormal basis choice.

Clearly set a motion to entire wall-solid is useless, it must be selected only the main part of the duct.

RBF-Morph is also in capable to apply solution only on a part of points, in the surface selected, through “*Selection*” in *Encap* panel.

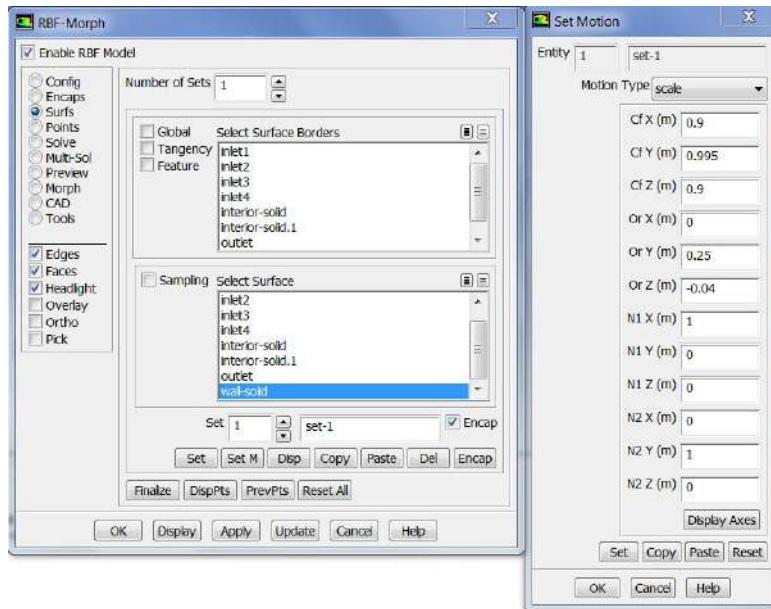


Figure 4.14: Surface Motion Set

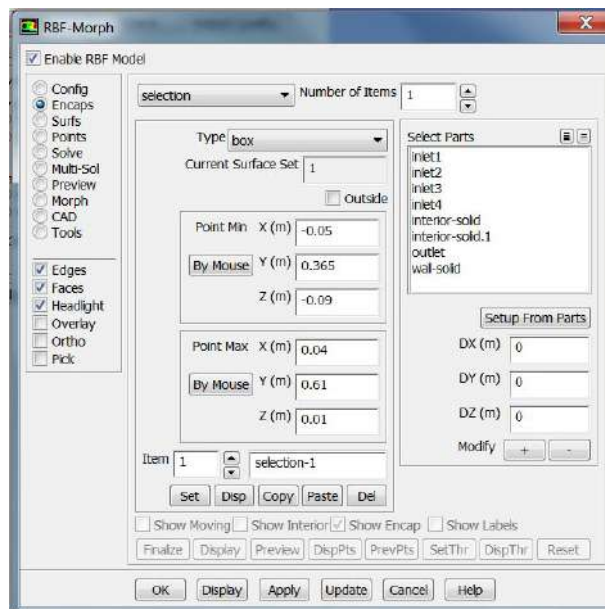


Figure 4.15: Surface Selection Set

Last consideration must be done is the sequent:

if we apply to the geometry, solution just set, some points near domain details, could have strange movement, creating in some case negative volume.

Avoiding this problem, is simply and rapid. It is sufficient create a box, with no moving, in proximity of domain details, as in figure.

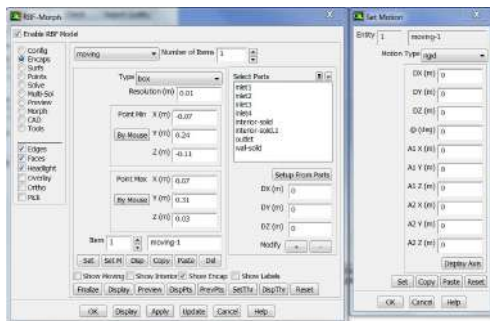


Figure 4.16: Set Moving

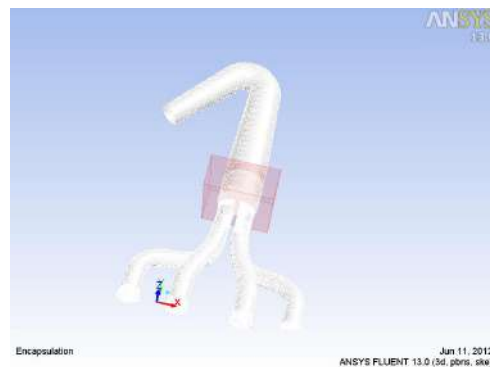


Figure 4.17: Display Moving

For a better view a “Display Points” of Domain and Moving is the following:

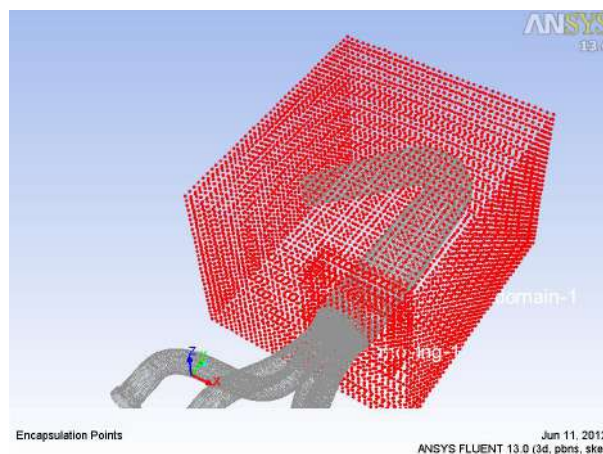
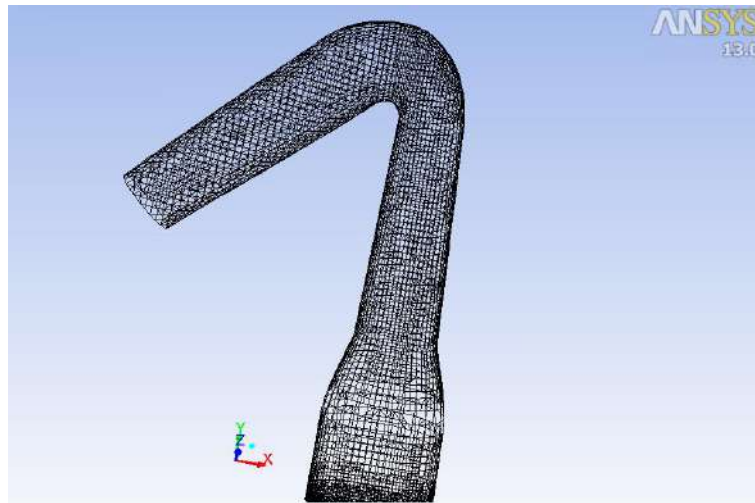


Figure 4.18: Display Points

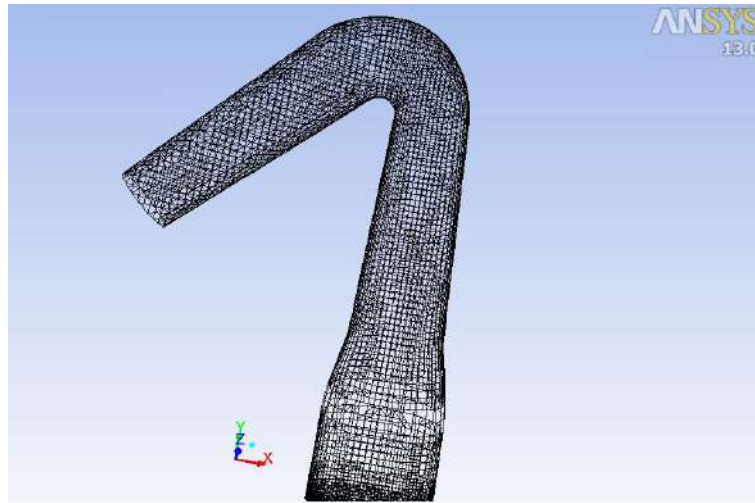
4.2.2 Mesh Morphing

Solution amplification in range $[-3; +3]$ (Excluded baseline):



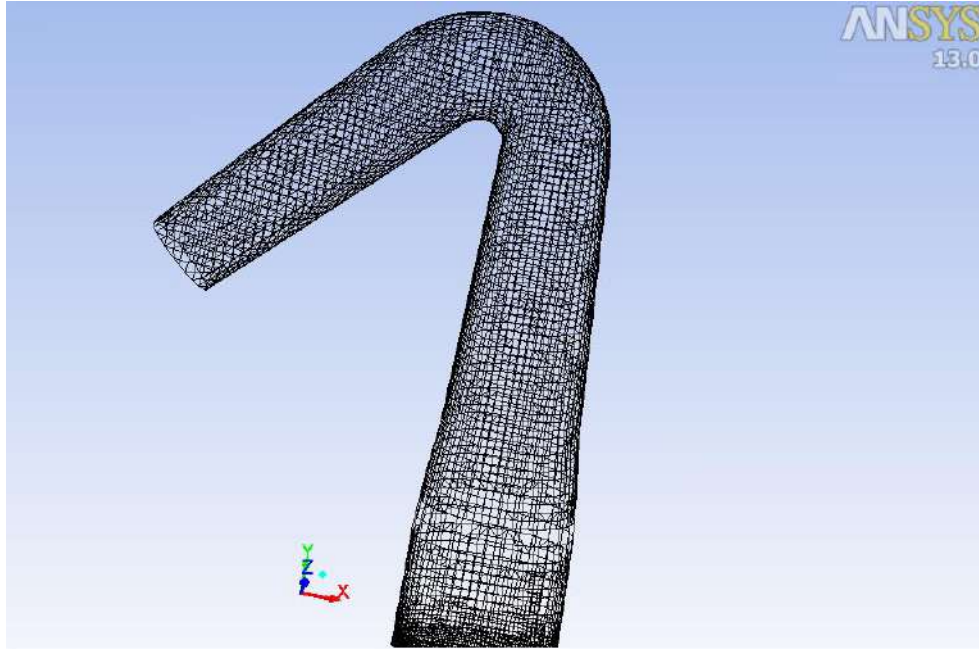
Morphing Preview (A=3)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



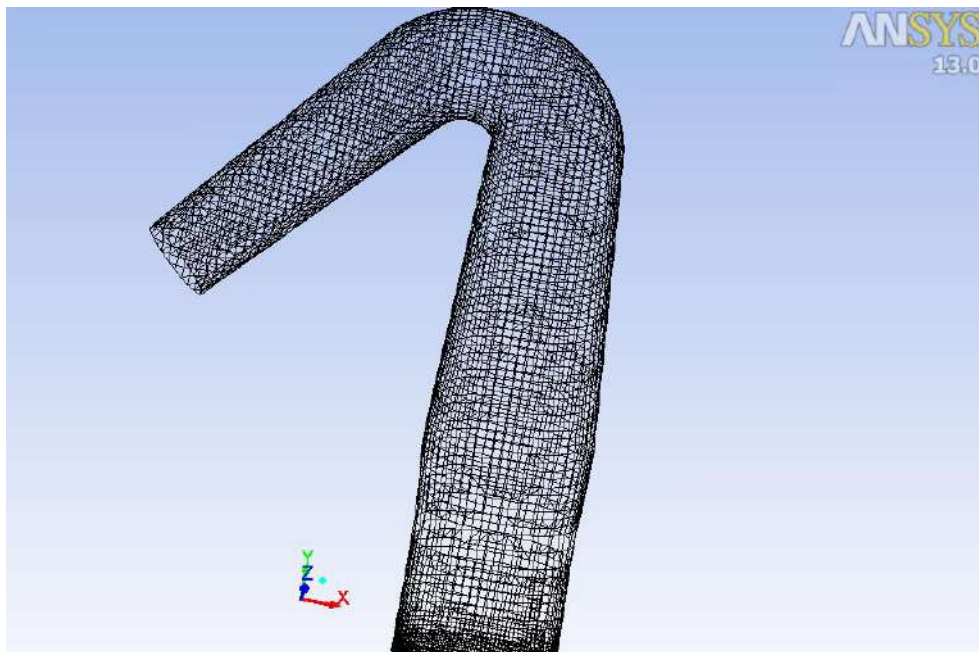
Morphing Preview (A=2)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



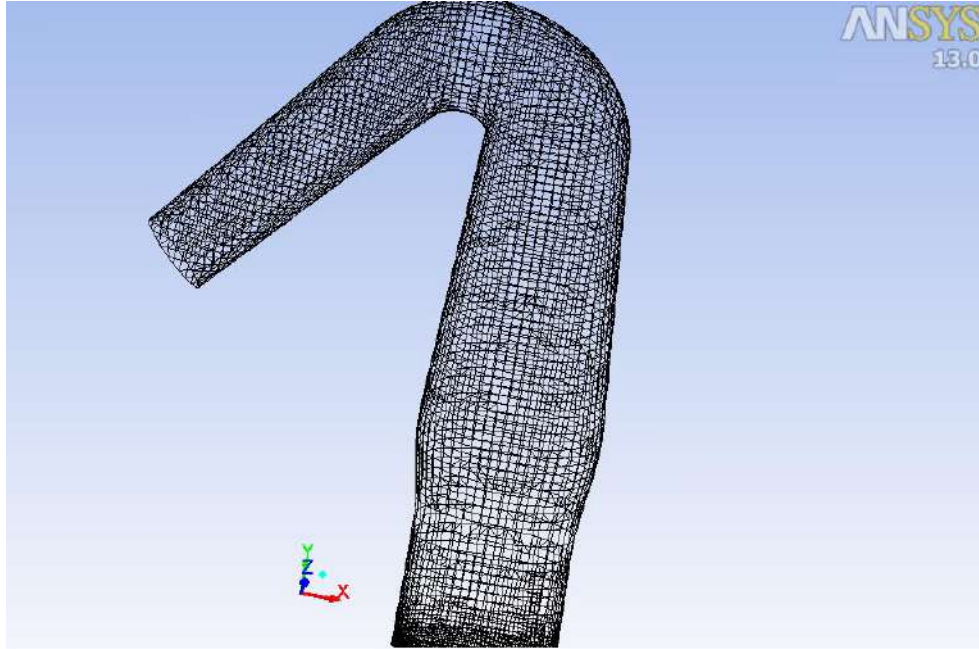
Morphing Preview (A=1)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



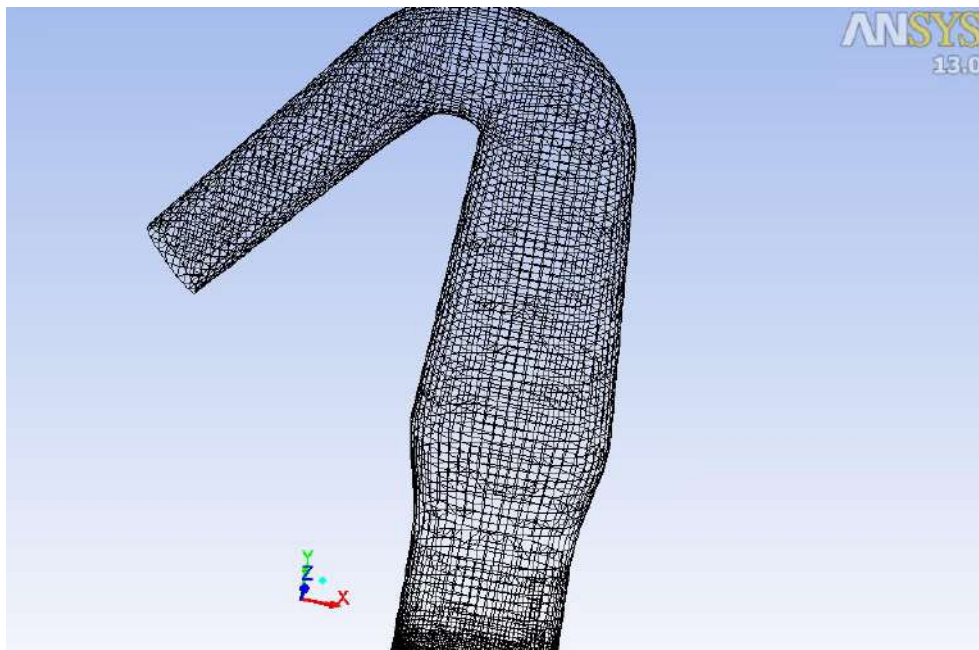
Morphing Preview (A=-1)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-2)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-3)

Jun 11, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)

4.3 Test 3: Rotating outlet

Last shape modification developed is rotation of outlet pipe, changing angle flow will follow, and thus changing the trajectory.

This could have positive repercussions on vorticity born during bending.

4.3.1 Setup Details

Setup of this solution is developed differently from both previous. No domain is created, and when aren't domains, all mesh is affected by RBF-morph solution. Since the goal is this morph is a rotation, and to make a rotation it should be defined a domain too particular, otherwise could be born negative volumes; it is preferred enter any domain, and stop the rest of the template with appropriate "moving" set to not move.

Evidently rotation is limited by external constraints, this is part of an exhaust system, in this report is optimized this model, the rest of the exhaust system must be tuned according to the same.

Constraints given are following:

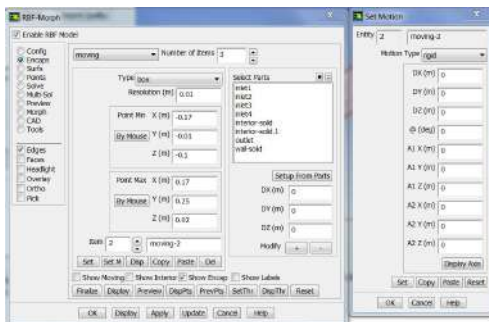


Figure 4.19: Set Moving 2

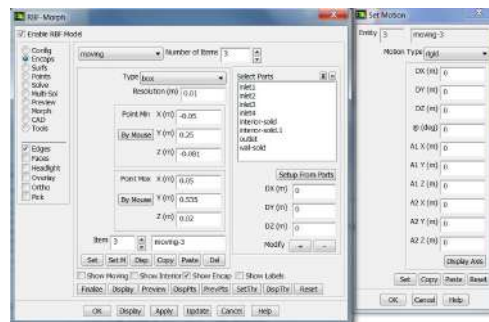


Figure 4.20: Set Moving 3

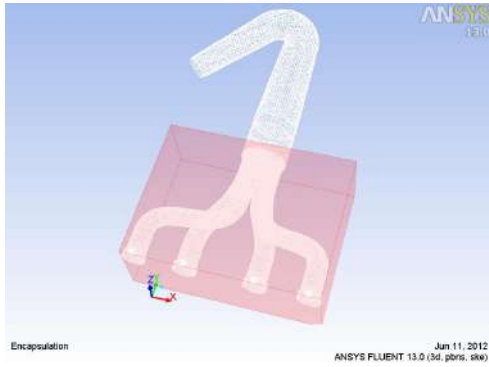


Figure 4.21: Display Moving 2

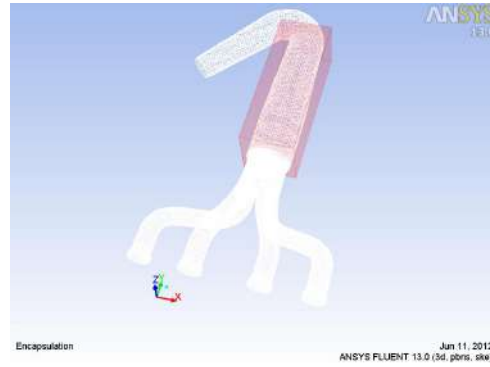


Figure 4.22: Display Moving 3

Now outlet pipe is encapsulated in a cylinder, and it is assigned rotation axis and angle (5°). Define cylinder to rotate it is not so easy, it needs some experience and instrument sensibility.

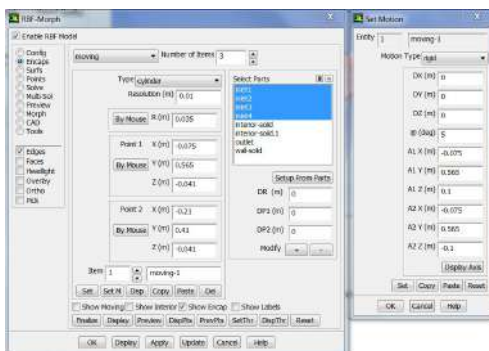


Figure 4.23: Set Moving 1

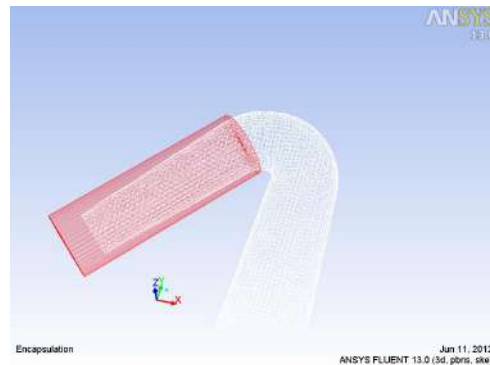


Figure 4.24: Display Moving 1

In the end, Preview and Display points about solution just developed:

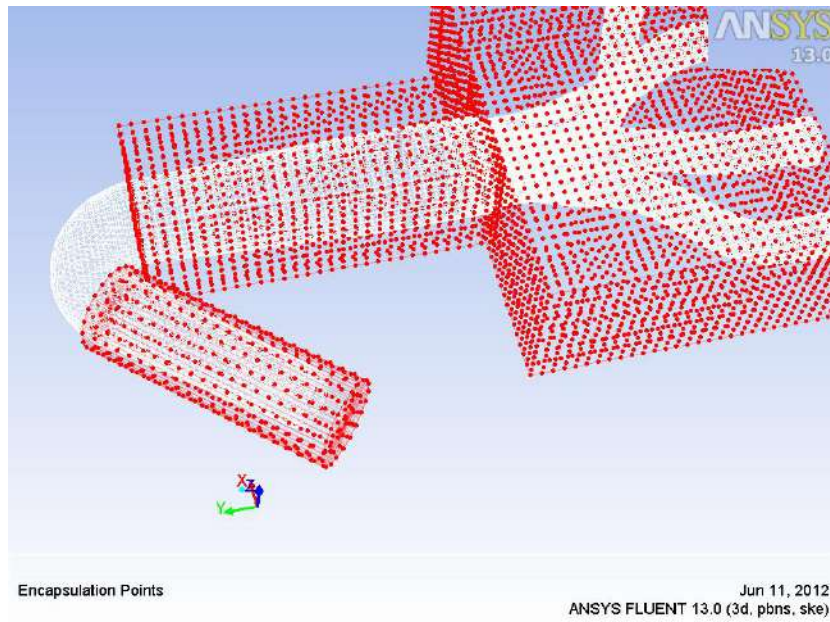


Figure 4.25: Display Points

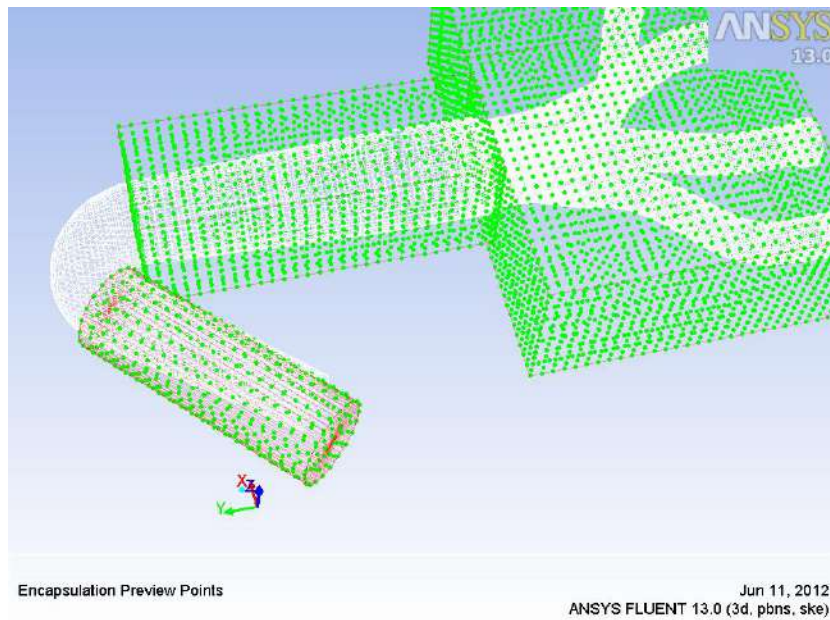
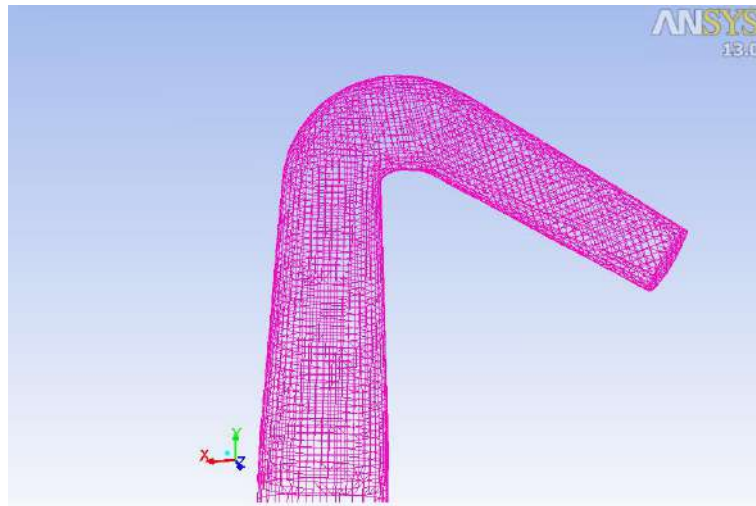


Figure 4.26: Preview Points

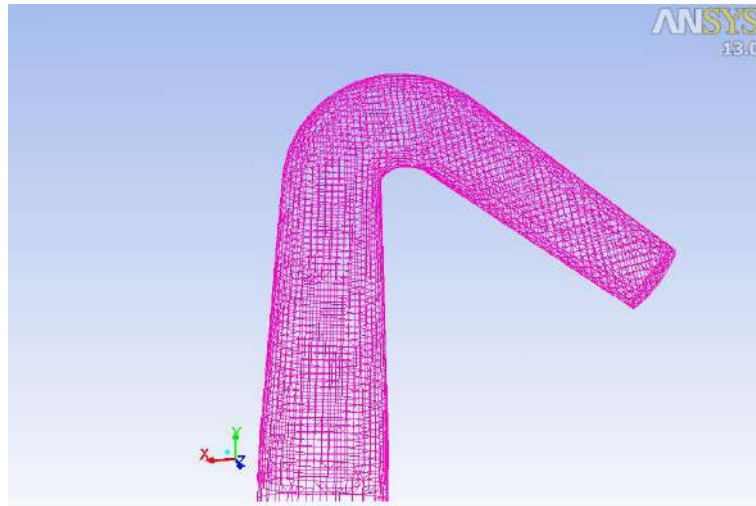
4.3.2 Mesh Morphing

Solution amplification in range $[-3; +3]$ (Excluded baseline):



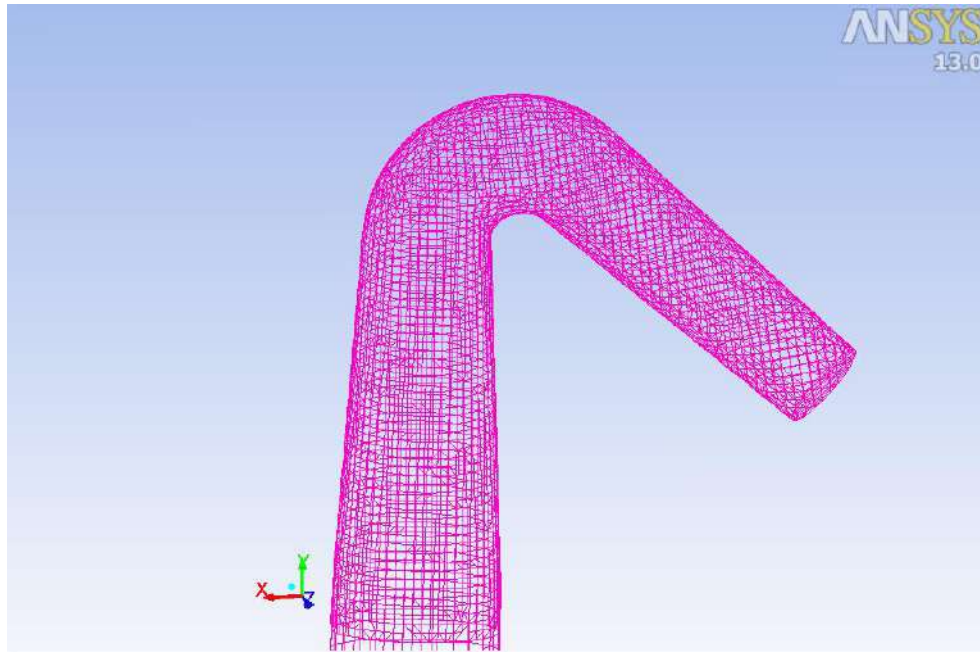
Morphing Preview (A=3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



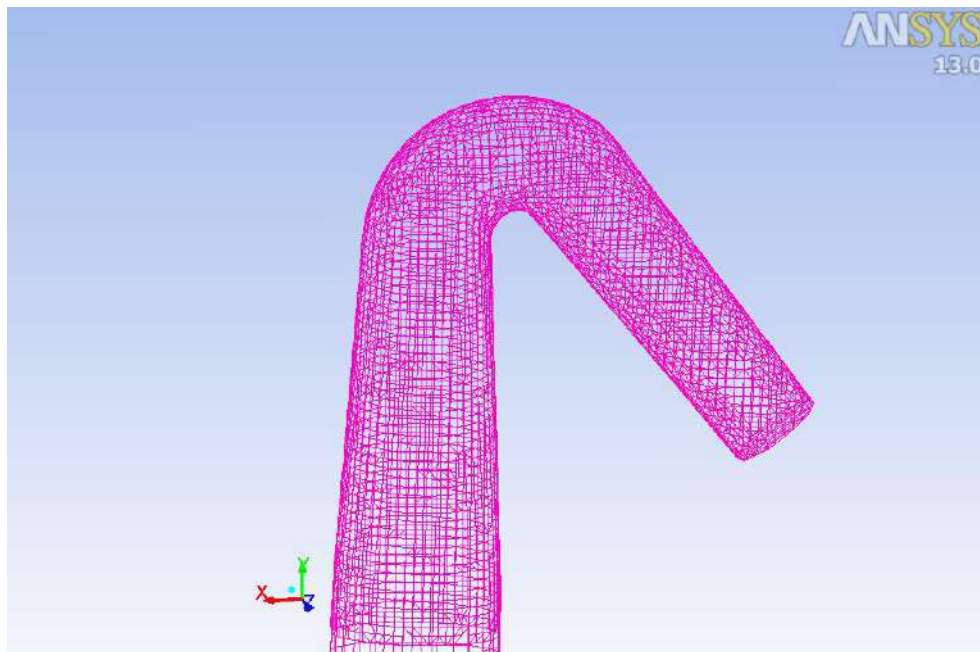
Morphing Preview (A=2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



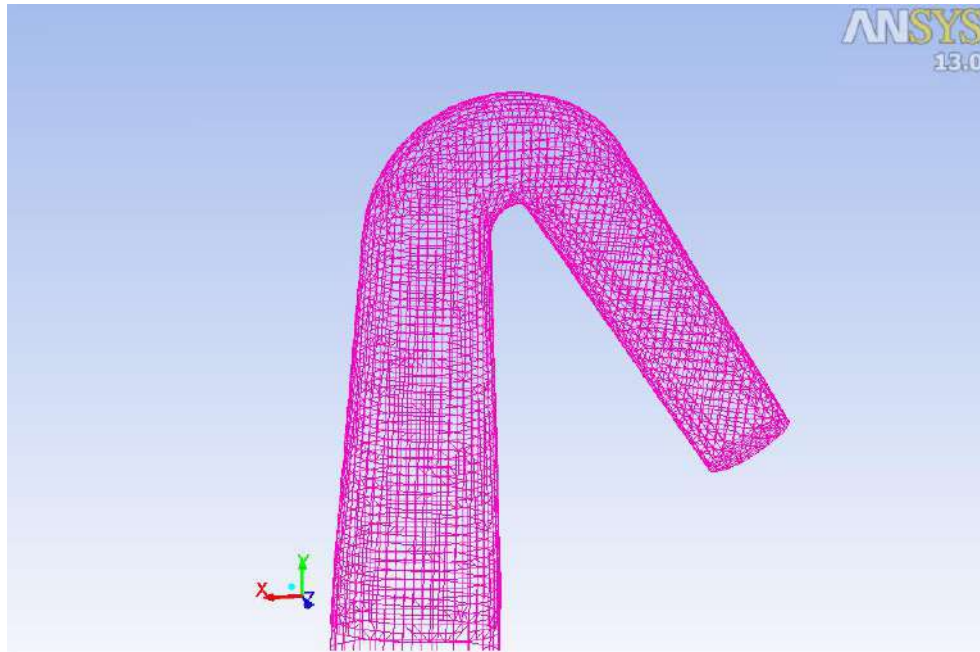
Morphing Preview (A=1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



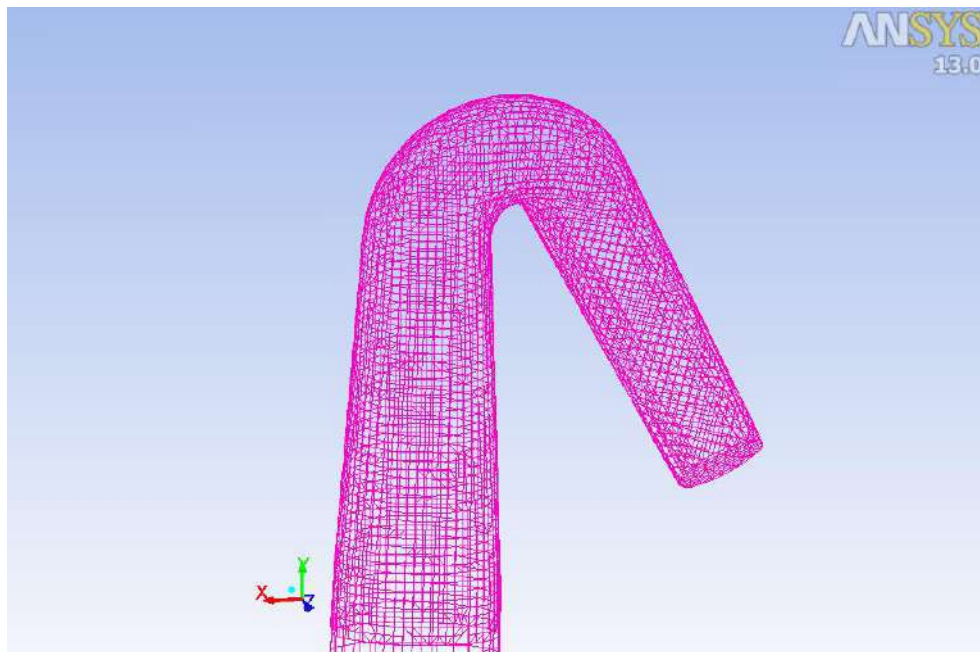
Morphing Preview (A=-1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)

4.4 Test 4: Single Pipe Modifications

In additions to manifold solutions, it is needed create some solutions to apply to the single pipes.

In this way is possible also to reduce pressure drop between four pipe, giving every one, appropriate shape modifications.

Since the geometry, apart from final curve, presents a symmetry; changes can be made also symmetric, respectively for pipe 1 and pipe 4 in one hand, for pipe 2 and pipe 3 in the other.

Despite symmetry of pipes, and symmetry of solutions developed, best solutions will not necessarily be those with equal amplifications: e.g. it might happen that in order to achieve the optimum condition, it must tighten a duct and enlarge the symmetrical, thus losing the symmetry. But this will be possible to say only after run simulations with Fluent.

4.4.1 Setup Details Pipe1

As already seen in other tests, it must be defined a *Domain*, to modify only Pipe of interest. Like done in test N.1, it is defined a *moving encap*. In this case is defined a sphere, the most simply encap to create.

In fact it must be defined a point and a radius, and afterwards a rigid motion is to be set. Mesh will follows sphere rigid motion.

In following page there are shown settings for Domain and Moving, and relative display. For greater precision, there are also shown *Display Points* and *Preview Points* of RBF-Morph solution.

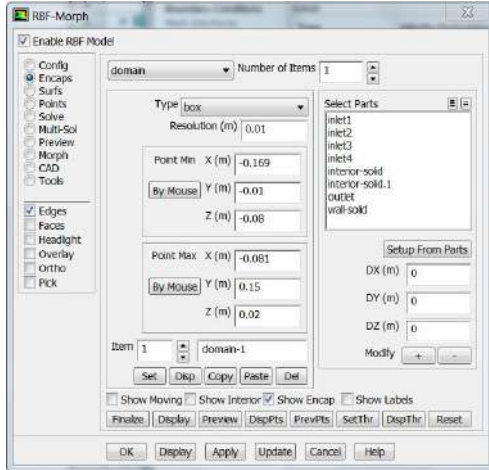


Figure 4.27: Set Domain

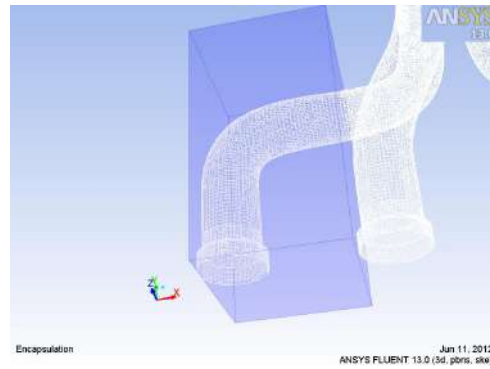


Figure 4.28: Display Domain

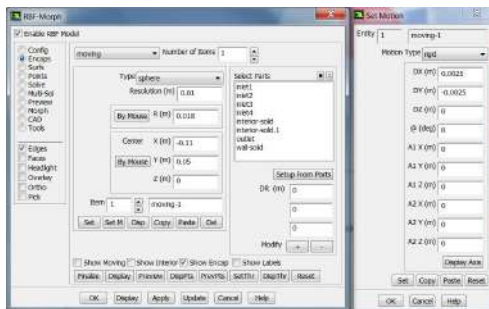


Figure 4.29: Set Moving

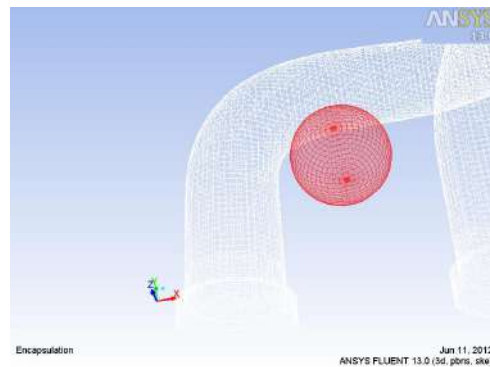


Figure 4.30: Display Moving

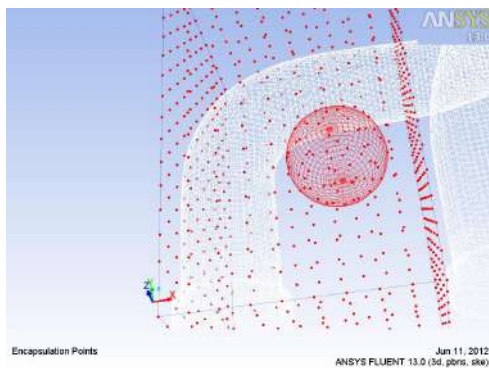


Figure 4.31: Display points

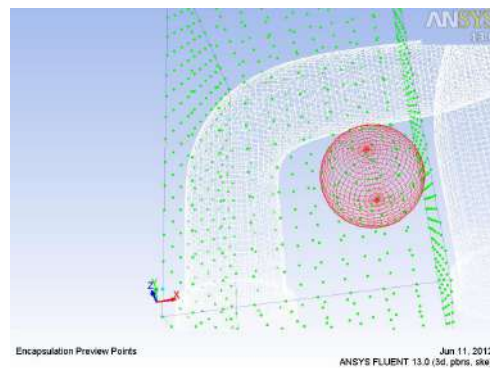


Figure 4.32: Preview Points

It is important not forget assign to Inlet 1 a rigid movement set to zero, which is equivalent to stillness.

This could also be done with a ploy to exclude it from the domain, but this way the solution is more complete.

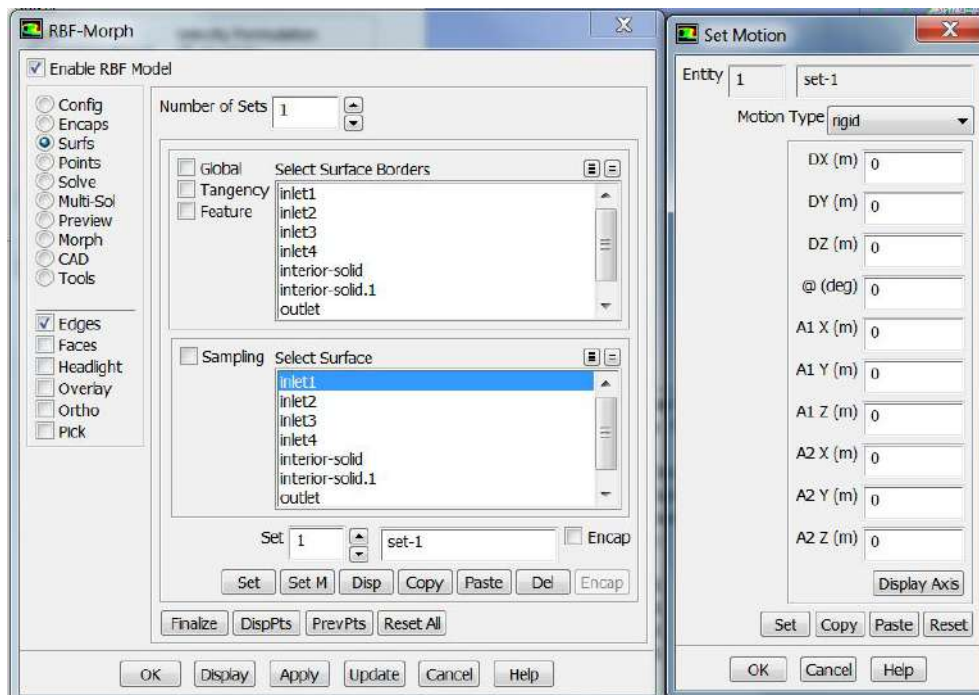


Figure 4.33: Surf Set

This is clearly essential cause of respect of constraints, in this case engine cylinders are fixed, and if Inlet is moved this work becomes useless.

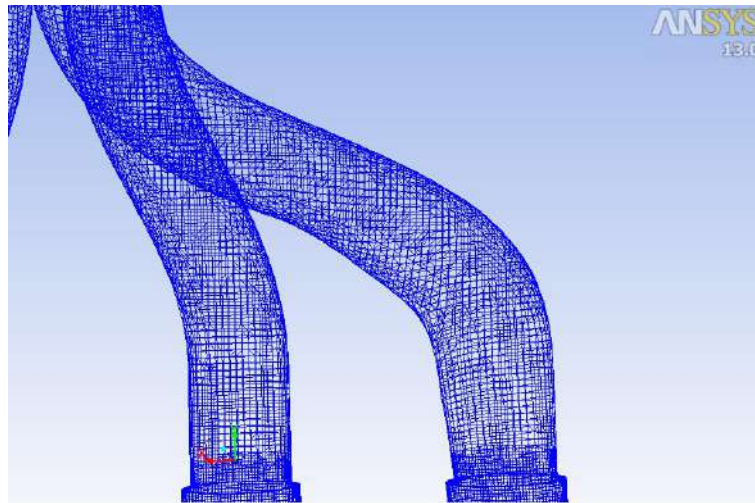
Now a careful observer could say that some speech is valid also in *Test 3* where is rotated outlet.

In those test the difference is that this model is a “cutcell” and where there is the outlet, the pipe continues. So it is editable, in project limits.

As just said, solution developed for Inlet 4, is totally symmetric.

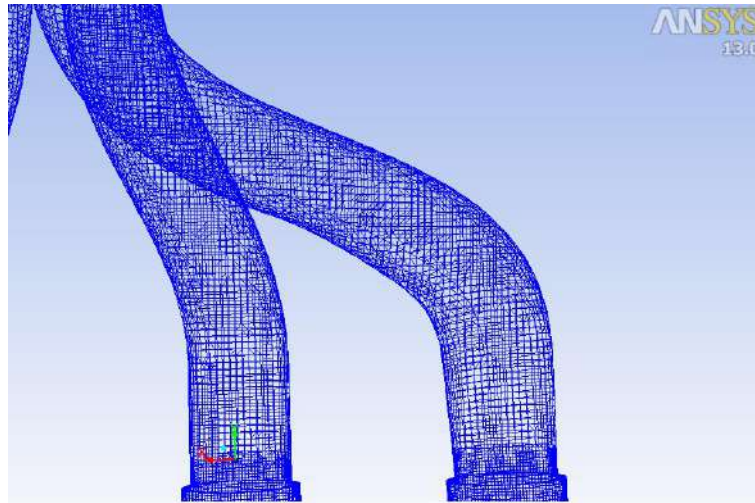
4.4.2 Mesh Morphing Pipe1

Solution amplification in range $[-3; +3]$ (Excluded baseline):



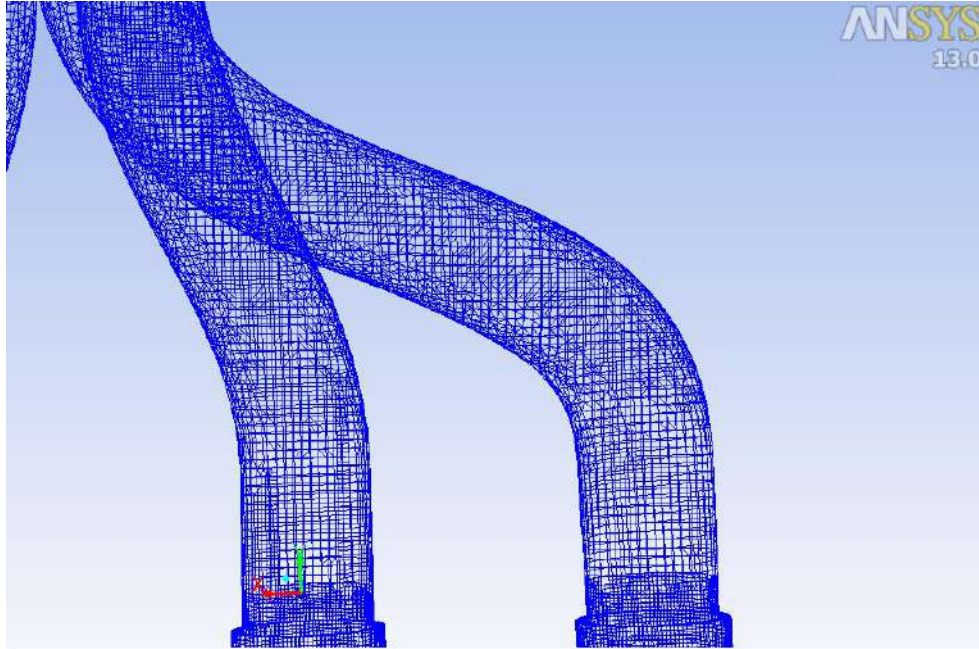
Morphing Preview (A=3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



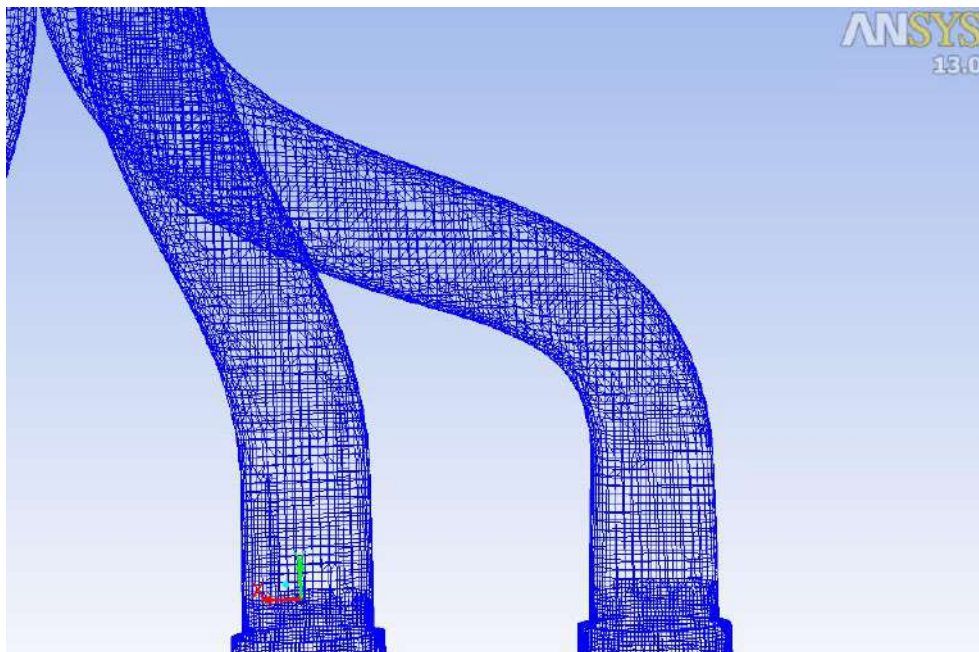
Morphing Preview (A=2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



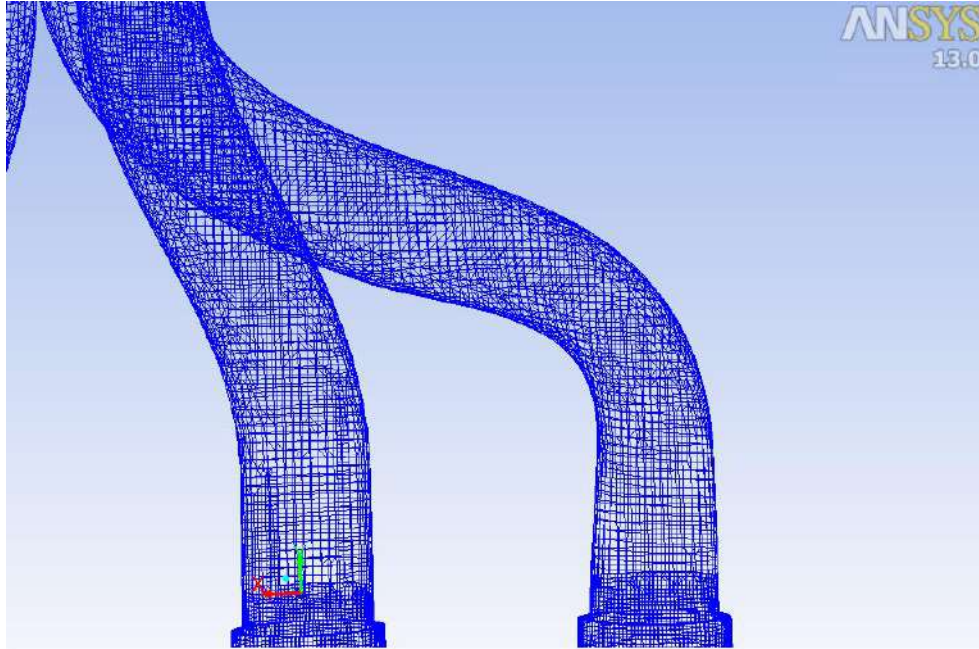
Morphing Preview (A=1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



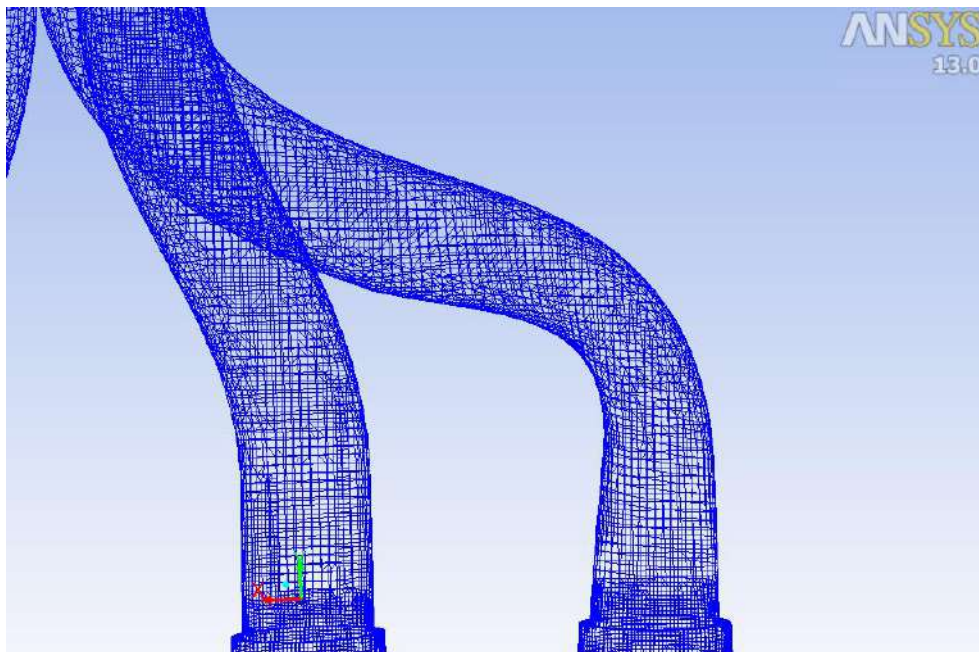
Morphing Preview (A=-1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)

4.4.3 Setup Details Pipe2

Guidelines developing morph of second pipe solution, are the same as before, except that instead of a rigid movement is assigned a “scale” motion type.

This choice was made to emphasize all the possibilities that RBF-Morph offers, but overall because in this way solution developed is better, if it would be like previous case, mesh would have changed too much.

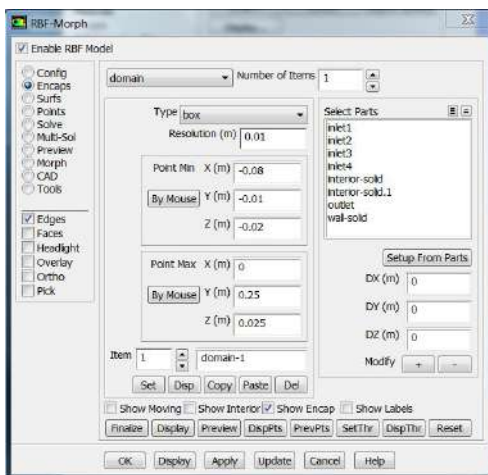


Figure 4.34: Set Domain

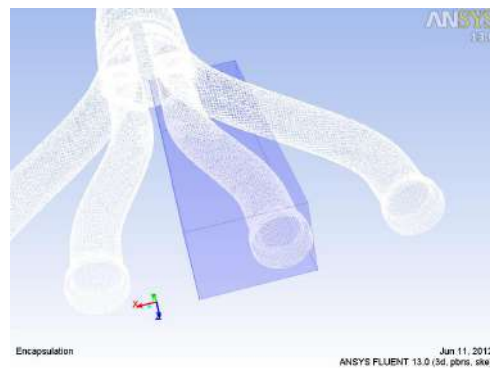


Figure 4.35: Display Domain

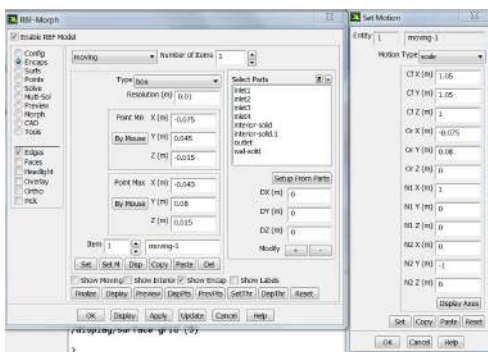


Figure 4.36: Set Moving

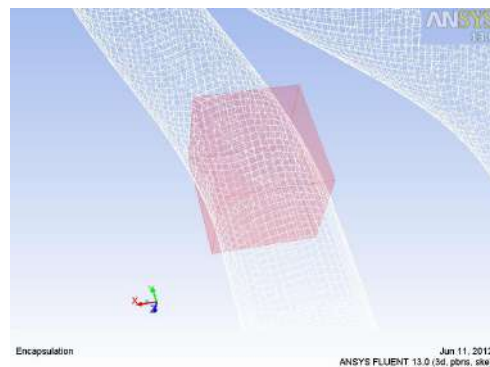


Figure 4.37: Display Moving

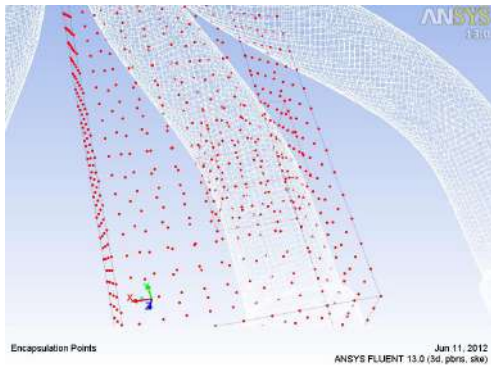


Figure 4.38: Display Points

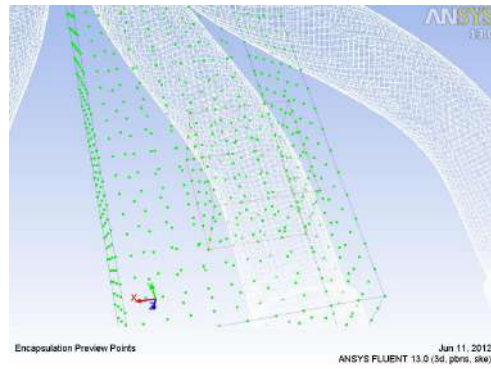


Figure 4.39: Preview Points

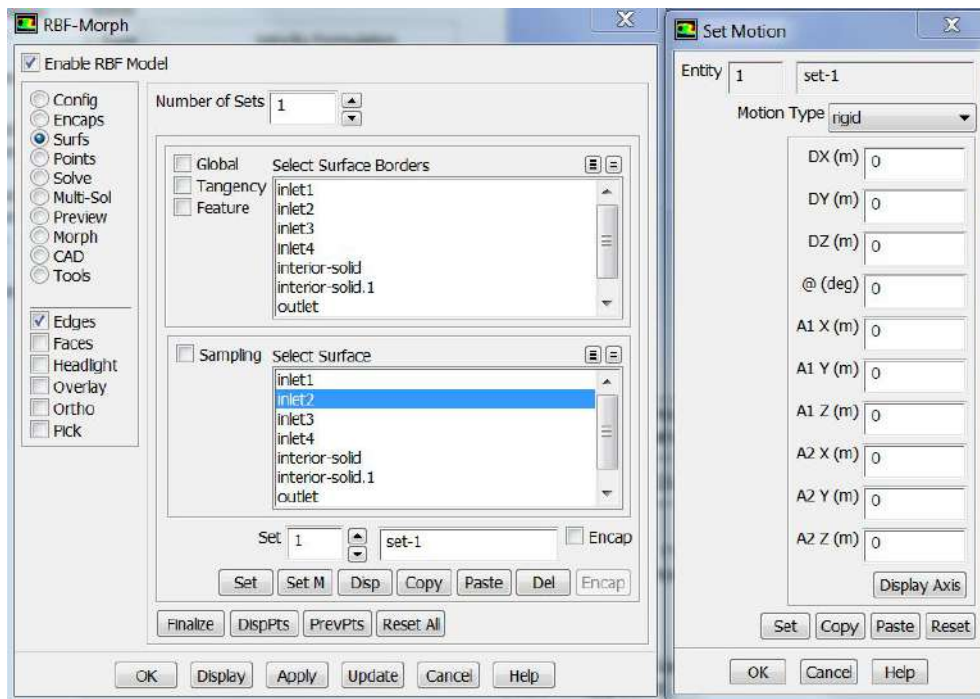
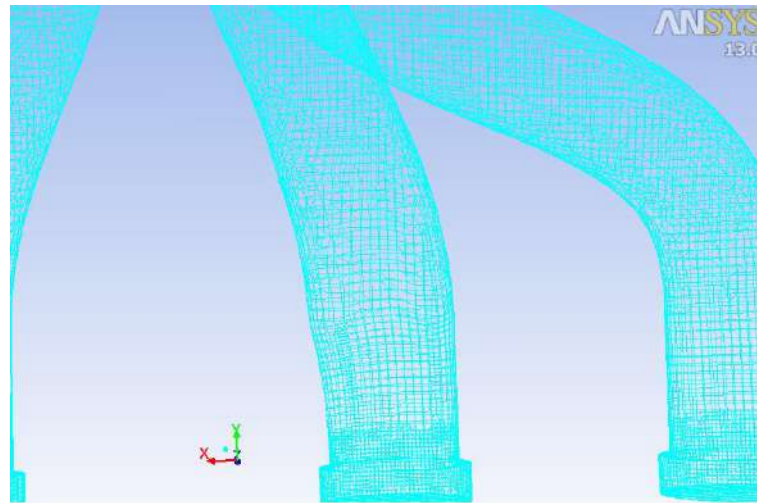


Figure 4.40: Surf Set

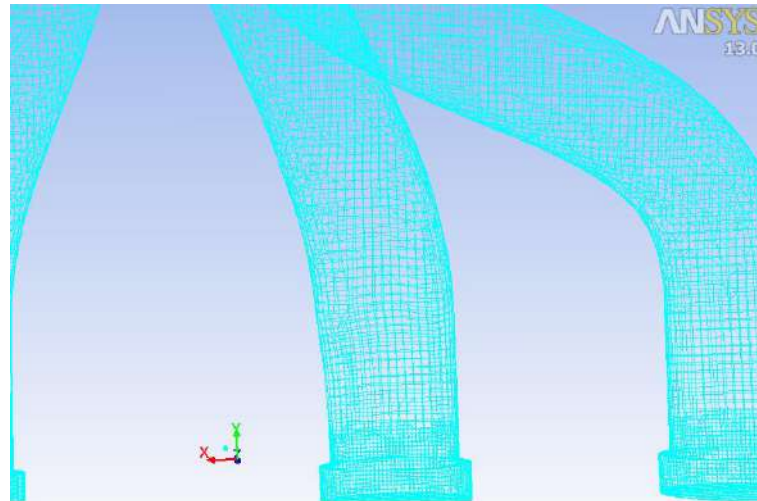
4.4.4 Mesh Morphing Pipe2

Solution amplification in range $[-3; +3]$ (Excluded baseline):



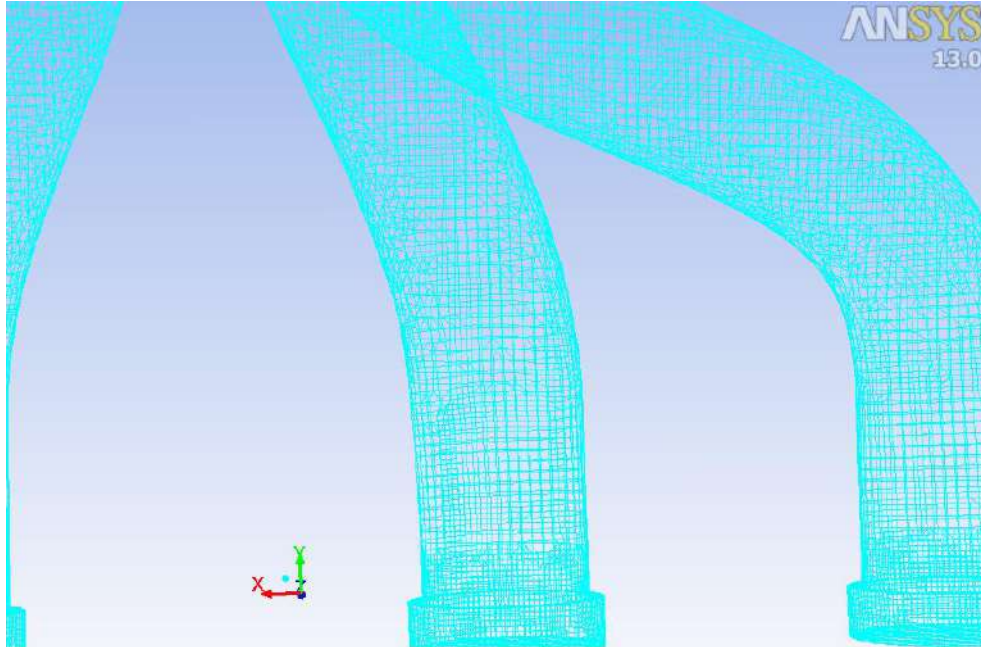
Morphing Preview (A=3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



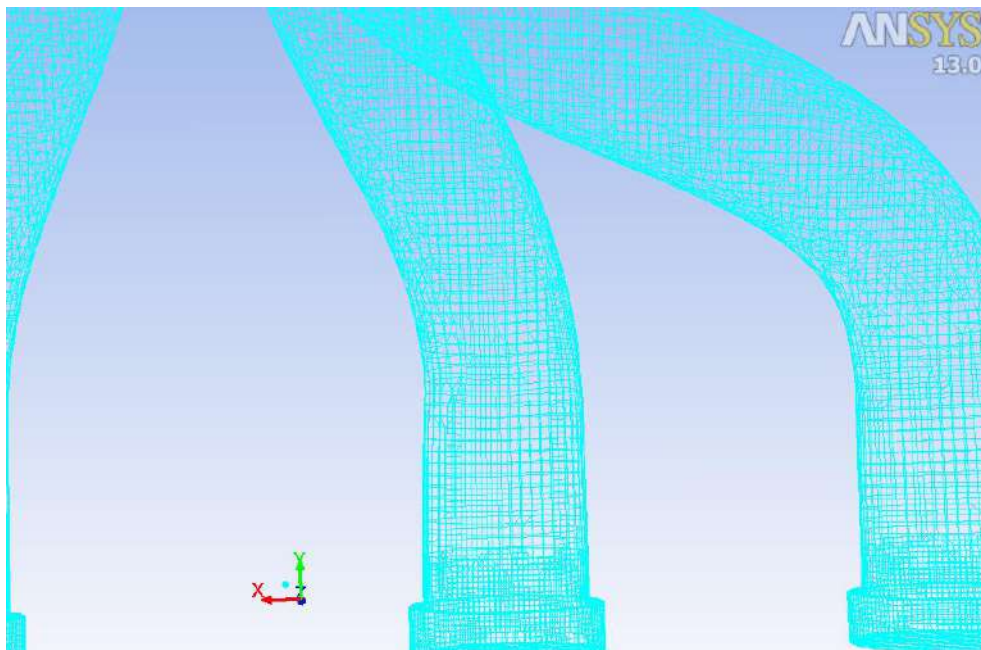
Morphing Preview (A=2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



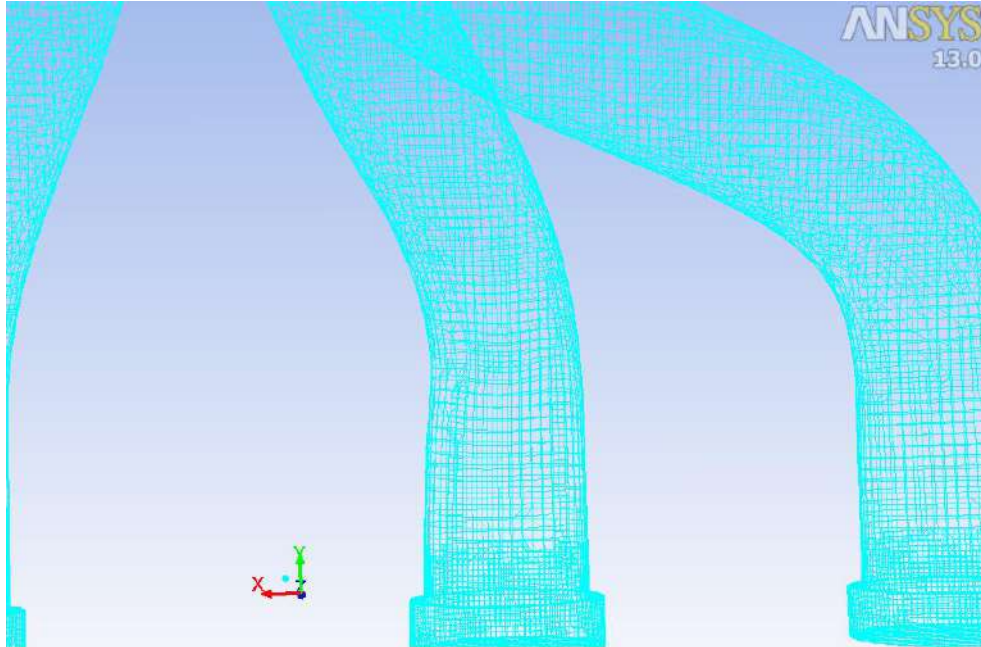
Morphing Preview (A=1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



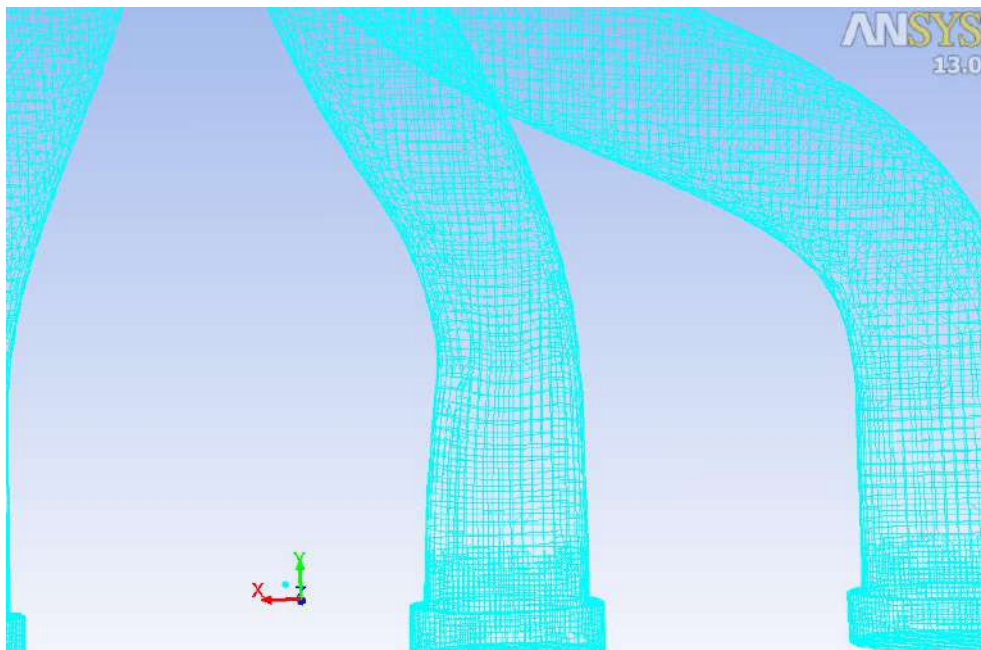
Morphing Preview (A=-1)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-2)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)



Morphing Preview (A=-3)

May 12, 2012
ANSYS FLUENT 13.0 (3d, pbns, ske)

Chapter 5

Results

Final and most important step is to make input parameter from *RBF-Morph* solutions; *Pressure Drops* are our output parameters as seen in chapter 3. In every *Fluent* block must be set input parameters with sequent code:

```
(create-custom-input-parameter "MainCurve" 0 'none)
```

In addition other code must be set in *calculation activities*, to allow *Fluent* initialize every time case, and setting new values for the input parameters.

Obviously Fluid-dynamics settings must be kept absolutely the same to the baseline.

Control of all transactions will be done by main Workbench platform, creating the various combinations of *RBF-Morph* sol, and showing directly on the screen of the *parameter set* the desired output.

In a first approach were applied solutions one at a time, making vary each in range of amplifications which is extended from -3 to 3, and setting the other solutions simultaneously to zero.

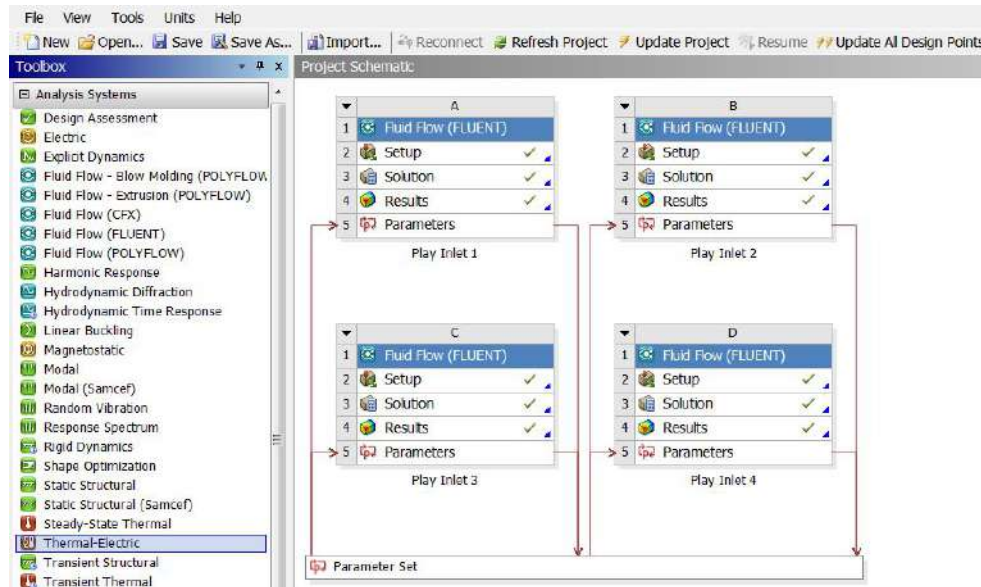


Figure 5.1: Workbench screenshot

5.1 Running and Results

Ended setting phase now must calculating solutions for all cases previously described.

In this study were conducted 49 simulations, with a total calculation time above 1 day. In every design point amplifications of the four fluent block are the same, in addition it was verified that changing geometry of a single pipe, before manifold, doesn't influences results of other pressure drops. It is remembered, also said in third chapter, the object is reducing pressure drop, and more troubling equalizing pressure drop for the different *Inlet*. So it is obtained a table of result for every RBF-Morph modification done.

5.1.1 “Swelling and Necking Curve” Results

With this implementation results are following:

FACET AVERAGE STATIC PRESSURE

	Inlet 1	Inlet 2	Inlet 3	Inlet 4
-3	21181,986	20099,955	19703,66	20236,896
-2.5	21416,078	20156,461	19632,053	20254,684
-2	21522,211	20220,15	19340,525	19922,172
-1.5	21399,785	20297,217	19663,148	20043,74
-1	21482,047	20361,641	19601,996	19936,457
-0.5	21247,559	20318,912	19637,26	20048,48
0	21658,313	20507,096	19550,801	19853,412
0.5	21601,787	20528,754	19704,225	19624,279
1	21734,359	20453,588	19744,115	19951,512
1.5	21513,352	20494,631	19718,316	19969,93
2	21640,076	20542,129	19643,598	19866,746
2.5	21362,135	20670,584	19738,822	19799,861
3	21429,998	20466,588	19742,281	19704,82

Table 5.1: Swelling and Necking Curve Results

It is possible seen that development of pressure function is not linear, or in general following a specific law. It is more or less swinging around baseline values, with differences between pressure drops in different inlets.

An exception is done for pressure drop in Inlet 2, it has a growing trend with increasing of amplification. In Inlet 3 situation is on the opposite with with the addition of an oscillatory pattern.

It is however visible that in the presence of an amplification of 2,5 gap between pressure drop in different inlets is decreased. This happens because of

the pressure drop in the first duct is diminished, and so also in the second, while the third and fourth have a hint of increment. In a hypothetical modification of geometry, given the experimental results, it would lead to opt for this solution. It is to remember that RBF-Morph Solution is set to inflate the curve. Choose a negative amplification means decrease the duct section in conjunction of the curve, creating a necking.

For better clarity, it is possible create a chart, where pressure is tabled as a function of amplification of modification.

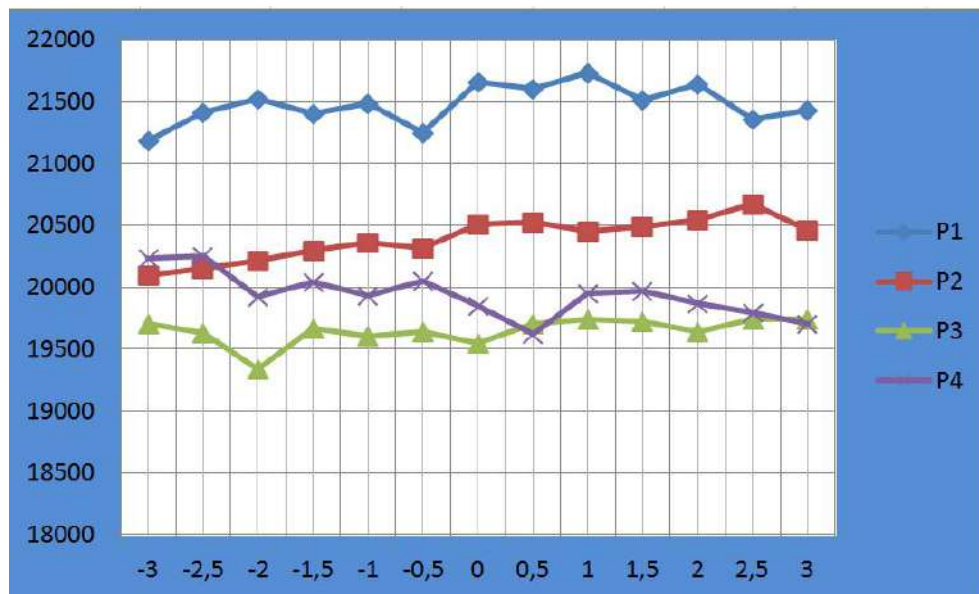


Figure 5.2: Swelling and Necking Curve Results

5.1.2 “Decreasing manifold section” Results

With this implementation results are following:

FACET AVERAGE STATIC PRESSURE

	Inlet 1	Inlet 2	Inlet 3	Inlet 4
-3	22675,854	21817,422	21627,51	21928,65
-2.5	22424,65	21488,656	21013,32	21659,307
-2	22457,973	21123,797	20770,098	21360,689
-1.5	22008,266	20778,598	20267,488	21146,504
-1	21817,666	20908,615	20144,963	20745,227
-0.5	21421,029	20820,332	19757,244	20150,553
0	21658,313	20507,096	19550,801	19853,412
0.5	21240,67	19888,545	19039,334	19598,217
1	20489,99	19010,477	18781,918	19574,324
1.5	19859,664	18961,379	18335,438	18539,213
2	19688,953	18568,533	18602,594	18437,188
2.5	19208,037	18243,666	17999,811	18513,373
3	18989,055	18339,67	18159,107	18762,941

Table 5.2: Decreasing Manifold Section Results

Unlike previous case, this time the trends of pressure drops, are almost framed in a descending trend when amplification is increased; i.e. to reduce the manifold section.

This is because diminishing manifold section, is decreased possibility of creating a turbulence flow.

It has, however, chosen not to decrease overtighten section, otherwise area of section would have been less than inlet and outlet pipes areas, and this would be a disadvantage for the exhaust system, since it would obstruct ”breathing”

of engine.

For better clarity, it is possible create a chart, where pressure is tabled as a function of amplification of modification.

Graph view is very important to identify the best solution and choose relative amplification.

It is clear that we must move towards a positive amplifications, and therefore a decrease of manifold section. For amplification of 2 - 2.5 it is seen that pressure drop decreases, and at the same time also decreases the difference between pressure obtained in different inlets.

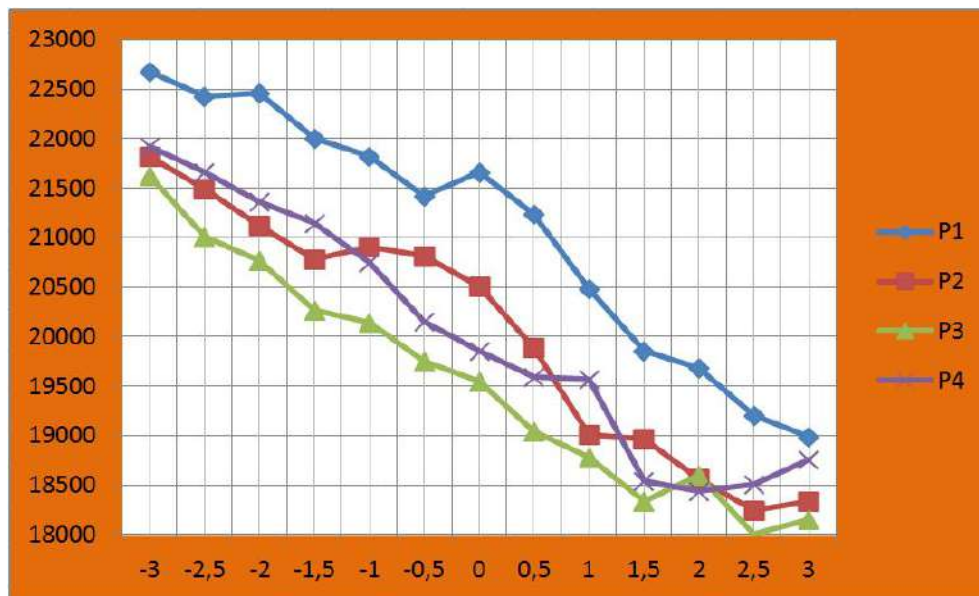


Figure 5.3: Decreasing Manifold Section Results

5.1.3 “Rotating outlet” Results

With this implementation results are following:

FACET AVERAGE STATIC PRESSURE

	Inlet 1	Inlet 2	Inlet 3	Inlet 4
-3	21075,932	19913,521	19661,873	19906,711
-2.5	21149,662	20490,4	19763,092	19996,951
-2	21050,348	20431,311	19759,27	20070,248
-1.5	21497,865	20488,066	19739,203	19873,732
-1	21104,252	20507,07	19512,336	19794,186
-0.5	21196,189	20123,367	19621,842	19791,695
0	21658,313	20507,096	19550,801	19853,412
0.5	21760,834	20143,281	19540,99	19748,414
1	21932,438	20178,998	19663,275	20103,535
1.5	21842,527	20555,648	19622,297	19706,838
2	21459,387	20502,141	19624,947	19775,502
2.5	21999,283	20545,652	19628,539	19588,061
3	22242,803	20568,797	19637,391	19663,24

Table 5.3: Rotating Outlet Results

Last implementation developed for the collector, is the rotation of the output section. Caution to keep, in making this change, should be plenty.

This is because moving the output section does not comply with external constraints.

However, outlet section can be moved for a few degrees, and then rest of the exhaust system is to adapt as designed in this section.

Before proceeding is critical to recollect which: set default rotation is counter-clockwise, i.e. giving a positive amplification, the flow is diverted by a

lesser angle, amplifying a lot in this direction is taken off the curve. On the contrary giving a negative amplification it will tend towards a 180° bend.

Angle set in RBF morph solution is 5°.

As explained above, watching results, the best condition is the amplification of -3, i.e. a closure of the curve about 15°. This solution is not good under the project point of view.

On the opposite it is possible choose a solution of -0.5 amplification, i.e. closure of 2.5°, this involves a minor improvement, but it is closer to the design constraints.

For better clarity, it is possible create a chart, where pressure is tabled as a function of amplification of modification.

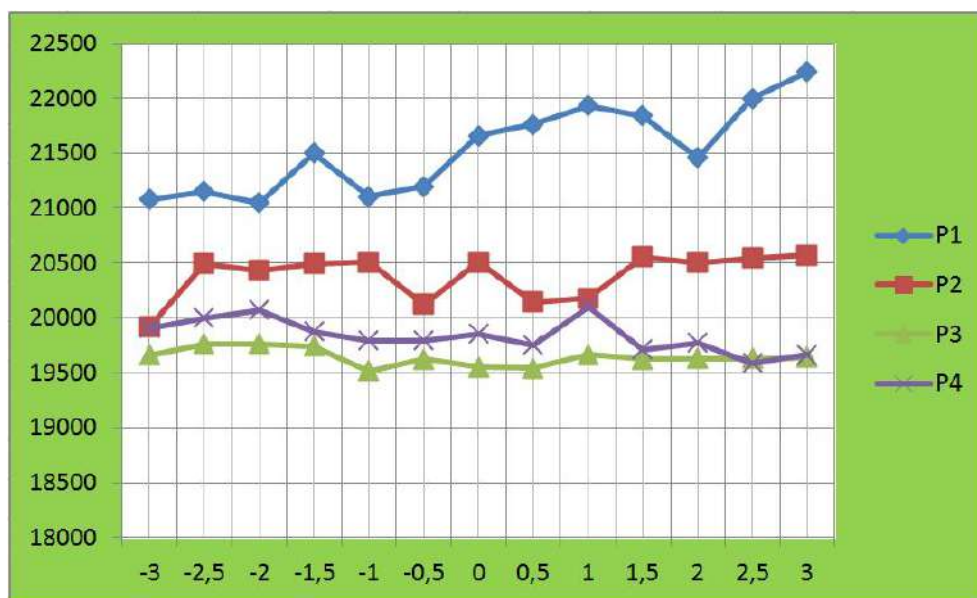


Figure 5.4: Rotating Outlet Results

5.1.4 “Single Pipe Modifications” Results

With this implementation results are following:

FACET AVERAGE STATIC PRESSURE

	Inlet 1	Inlet 2	Inlet 3	Inlet 4
-3	21658,869	20765,428	19755,666	21311,596
-2.5	21373,506	20536,775	20200,541	20536,762
-2	21756,879	20647,072	20010,305	20423,156
-1.5	21294,008	20168,25	19920,916	20214,262
-1	21505,918	20488,504	19800,578	19924,018
-0.5	21746,457	20524,285	19727,781	19997,17
0	21658,313	20507,096	19550,801	19853,412
0.5	21794,229	20427,873	19422,254	19665,563
1	21713,443	20441,707	19163,199	19501,955
1.5	21611,348	20083,307	19552,766	19393,088
2	21798,25	20412,896	19517,379	19366,756
2.5	21259,525	20393,67	18889,25	19318,734
3	21771,303	20473,969	19014,197	19148,684

Table 5.4: Single Pipe Modifications Results

In the end, chosen manifold modifications, it is time to apply morph solutions at every single pipe before conjunction in manifold.

Positive amplifications will produce swelling of the pipes, negative amplifications will produce necking of the pipes.

Trend of pressure is very different, depending on the pipes.

In first and second pipes trend is swinging around the baseline values. While in pipes three and four it can be seen that, unless some singularities, the pressure drops decreasing with enlargement of duct, and growing with increasing necking

of the curve.

This behavior leads us to choose a necking, as otherwise the gap between the pressure increases too much.

As mentioned earlier, can be chosen for different ducts, different amplifications of solutions.

However it should be noted as a good result is achieved by modifying all conducted the same way,choosing a-2.5 amplification. In this way the pressures are more uniforms, and the geometry is symmetrical, in favour of technological production strategies.

For better clarity, it is possible create a chart, where pressure is tabled as a function of amplification of modification.

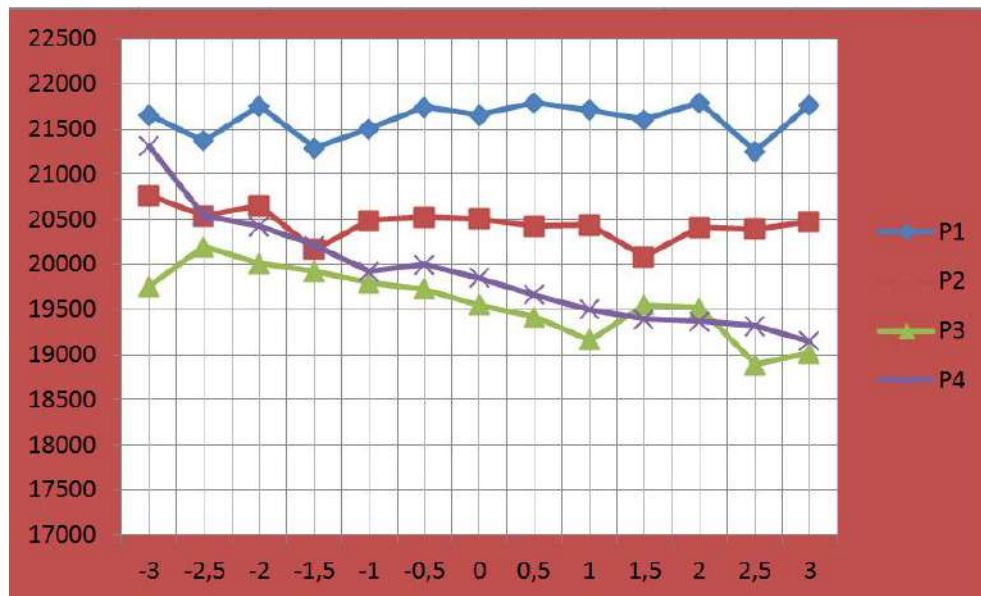


Figure 5.5: Single Pipe Modifications Results

5.2 Coupled Results

Applying a decoupled implementations, , results obtained were seen in the previous section. Considering these results, it can be created a series of combination and then completed the geometry optimization. With amplification chosen, resulting combinations are following, accompanied by relatives results.

FACET AVERAGE STATIC PRESSURE

N°	Rotate Outlet	Decrease section	S.&N. Curve	Single Pipe	Inlet1	Inlet2	Inlet3	Inlet4
1	-0.5	2	-3	-3	19920,42	19104,3	19212,25	20028,63
2	-0.5	2	-3	-2.5	19998,85	19074,73	19036,84	19761,09
3	-0.5	2	-2.5	-3	20038,68	19109,92	19092,92	19893,78
4	-0.5	2	-2.5	-2.5	19615,22	19036,25	19025,96	19630,27
5	-0.5	2.5	-3	-3	19793,76	19063,39	19295,61	20049,85
6	-0.5	2.5	-3	-2.5	19758,52	18988,08	19180,93	20038,72
7	-0.5	2.5	-2.5	-3	19809,91	18976,53	19049,88	20105,44
8	-0.5	2.5	-2.5	-2.5	19758,16	18881,91	19042,8	19958,3

In this case it is senseless creating a function chart, representing Pressure Drop in Function of Combination could be done through an histogram, like following.

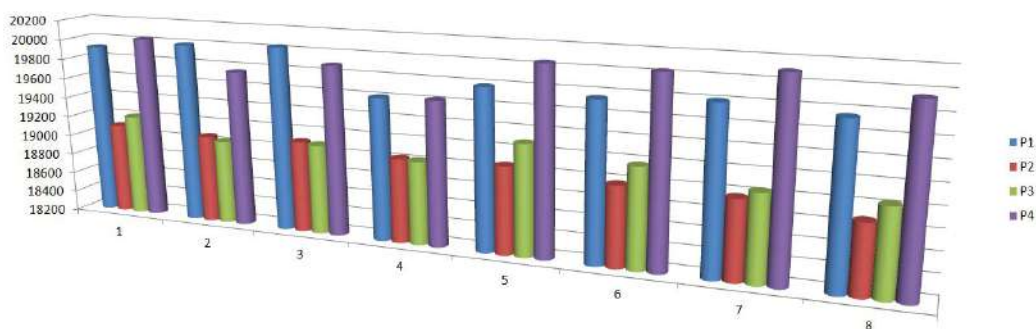


Figure 5.6: Coupled Results Graphic

Seen that histogram, as it could be seen in the table, combination N°4 brings out desired results, as displayed in the third chapter. Pressure drop is decreased in sequent percentages:

Pipe 1	Pipe 2	Pipe 3	Pipe 4
9.4%	7.2%	2.7%	1.1%

Table 5.5: Percentage of pressure decreases

It is clear that pipe 1 is more optimized by the rest.

Decrease of pressure isn't too much, cause of "in single pipe modification" it was chosen a negative amplification, i.e. necking of pipes.

This brings a more pressure drops in a side. On the other side this brings that differences between pressure drops eventually are deleted, and this is more important desired goal.

In fact differences of pressure drops in symmetrical pipes are totally deleted.

In addition differences of pressure drops in non symmetrical pipes, are reduced by 70%.

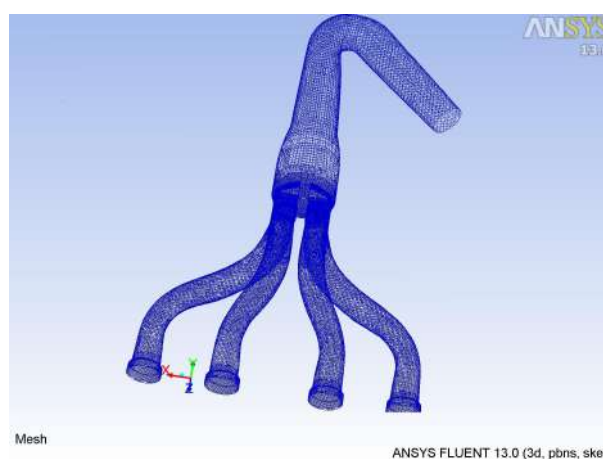


Figure 5.7: Optimized Geometry

Differences between Baseline and Morph Geometry

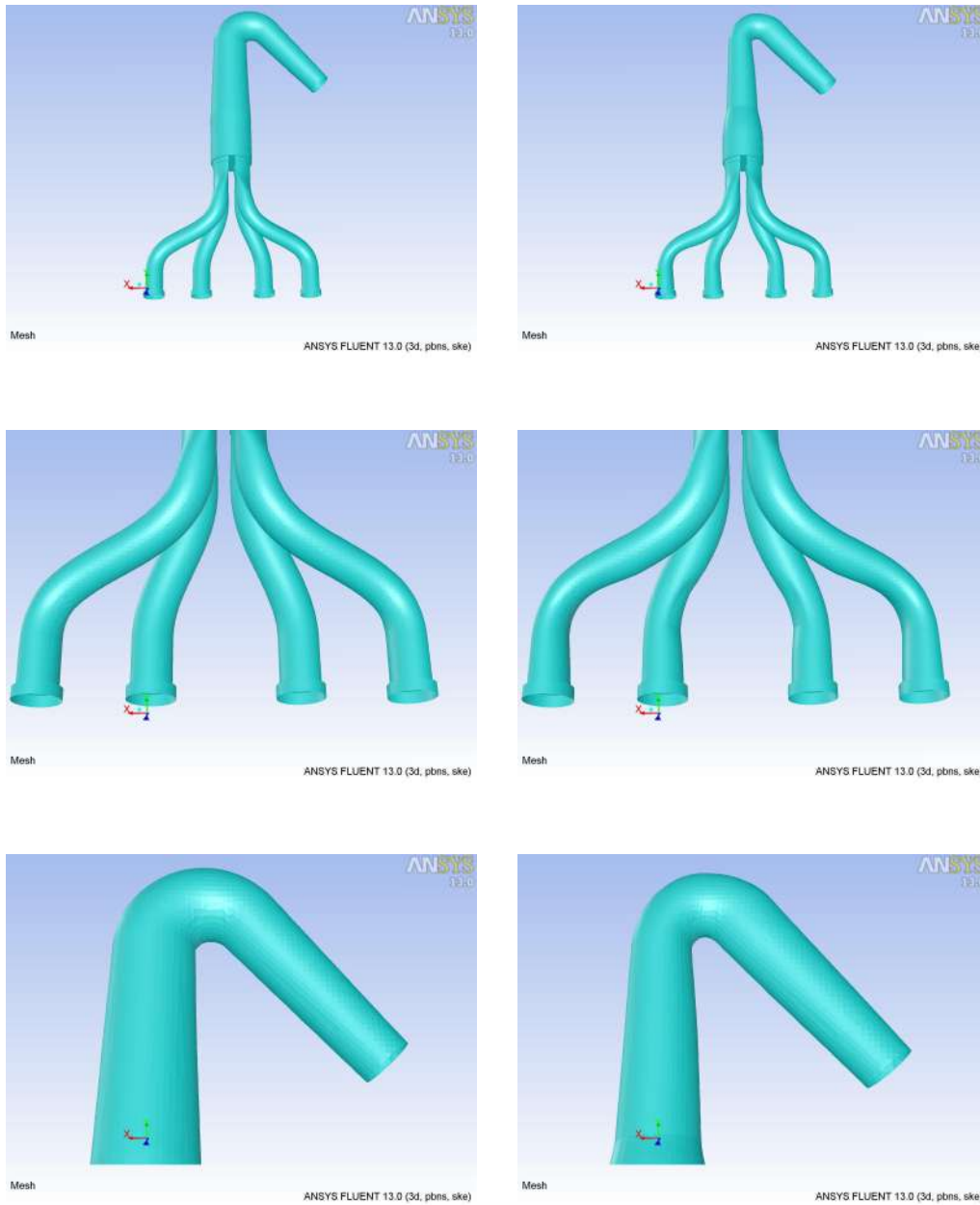


Figure 5.8: Morph and baseline geometry comparison

Streamline

With shape optimization is it clear that streamline are better than in baseline. In fact as seen in figure turbulence zone immediately after entering the manifold. In addition vorticity present immediately after the curve, was also eliminated.

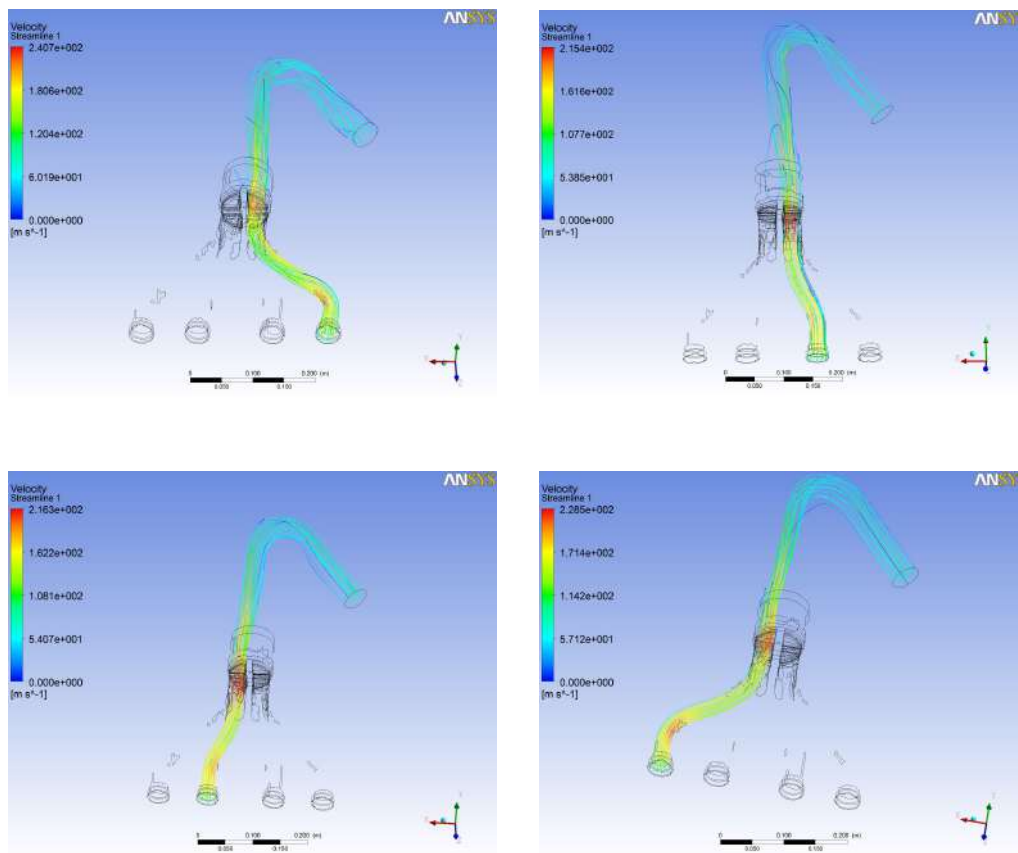


Figure 5.9: Morph geometry Streamline

Pressure Contours

It is also noteworthy a comment about pressure contours. Pressure drop is no longer concentrated in certain areas of the duct, but it is distributed, and more uniform.

There is a disadvantage in everything, and is that in the curves of pipes were created concentrated pressure loss. However, as explained previously, that modification was needed to achieve the desired purpose.

Moreover in engineering anything is obtained for free!

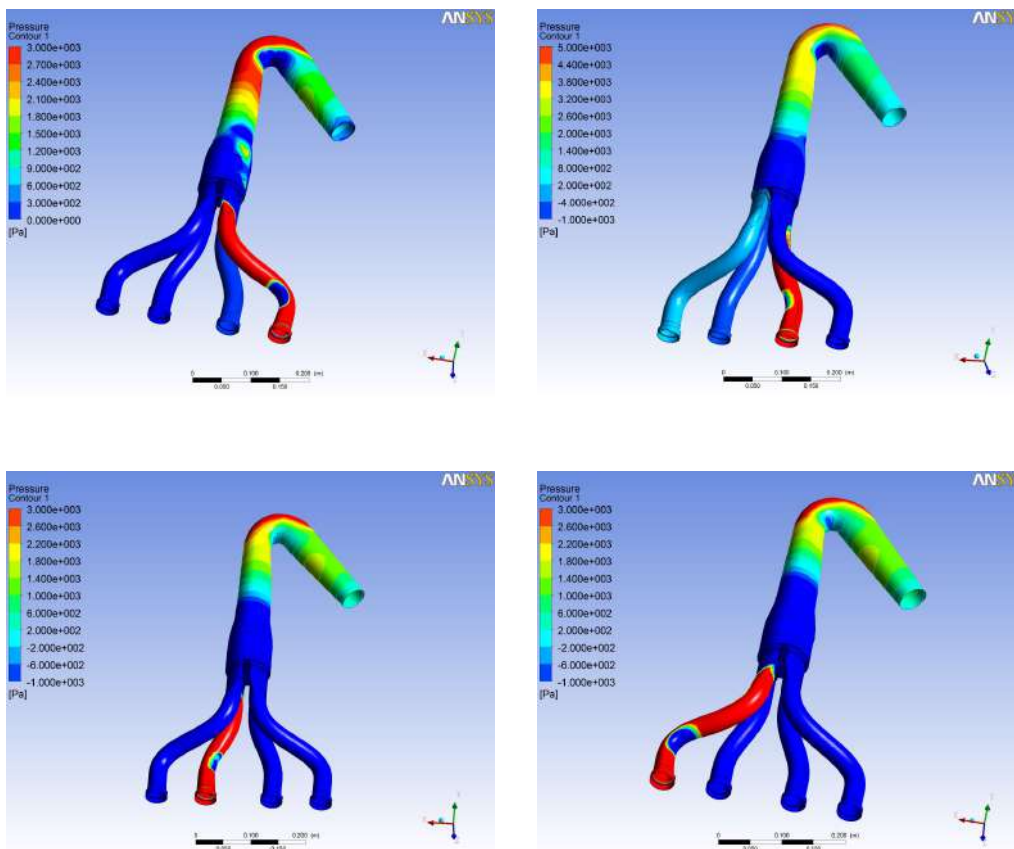


Figure 5.10: Morph Geometry Pressure Contours

Conclusions

In this work a model provided by Ansys Italy and describing a manifold exhaust system for four cylinders of an internal combustion engine was parametrically analyzed. Parameter to be optimized was the pressure drops between inlet and outlet. Through a detailed baseline analysis a significant pressure loss, resulting different in the four pipes, was found especially among the symmetrical lines, where the path that follows the stream was practically the same. In fact, difference between the first pipe and the third pipe, was about $2108 Pa$, i.e. 20%. Even more troubling was the difference between the first and the fourth pipe, i.e. $2075 Pa$ and between the second and the third pipe, about $1000 Pa$.

By analyzing the baseline streamlines, a turbulent zone clearly appears after the flow enters the main pipe, as well as an hint of vorticity occurs when the stream met the main curve. Regarding such a turbulence production as a problem, it was solved by proposing and analyzing three different geometric changes:

- Swelling and necking of the main curve;
- Rotating of the outlet cross-section;
- Decreasing of the manifold section.

In addition there were developed RBF-Morph solutions, one for each pipe.

RBF-Morph solutions were integrated within the Fluent framework, setting them as input parameters and keeping pressure drops as output parameters. Numerical simulations have been performed and controlled during the simulation phase.

In this way it is possible to optimize geometry in a much more rapid and efficient way: without leaving Fluent.

Parametrization allowed to run, firstly consisting in 48 decoupled simulations, and then in 8 coupled simulations, one for each pipe, in a total of 224 fluid-dynamics simulations.

A large part of the activity needed to obtain significant and interesting results, consisted in generating suitable and effective Morph-based solutions, as well as in setting them as input data for CFD analyses.

After decoupled results were obtained and synthesized, intervals of changes for parameter values were chosen based on the best results of output parameters in decoupled analyses and used for coupled simulations.

The following significant results were then obtained:

- *differences of pressure drops in symmetrical pipes are totally deleted;*
- *differences of pressure drops in non symmetrical pipes, are reduced by 70%.*

This optimized geometry has closed its cycle back to Ansys Italy, and it will be presented in the *Webinar*:

”Learn How Mesh Morphing Accelerates CFD-Driven Shape Modifications”

on Thursday, July 26, 2012.

This webinar will inform you how you can drastically reduce the simulation time to complete a set of design changes, without the burden of regenerating a geometry or mesh.

RBF Morph is a unique mesh morpher that combines the very accurate control of geometric parameters with extremely fast mesh deformation, all fully integrated with the ANSYS Fluent solving process and Workbench Platform.

Industrial application examples regarding External Aerodynamics, Multi-Phase and Internal Flows will be shown.

List of Figures

1.1	Mini Cooper exhaust manifold	2
1.2	Mazda mx5 racing beat exhaust manifold	4
1.3	Peugeot 206 16V GTI Stainless steel exhaust manifold	6
1.4	Backpressure	10
1.5	phases of pressure waves	11
1.6	Pressure Waves	13
1.7	Concentrated Pressure Loss	17
2.1	Fluent window: “Grid” menù	27
2.2	Fluent window: Solver Panel	28
2.3	Fluent window: Viscous model	29
2.4	Fluent Database Materials	30
2.5	Operating Condition & Boundary Condition	31
2.6	Solution Controls	32
2.7	Solution Methods	33
2.8	Monitors	34
2.9	Solution Inizialization	35
2.10	Run Calculation	36

3.1	Cad Design with CATIA	45
3.2	Manifold cutcell mesh	46
3.3	Manifold cutcell detail	47
3.4	Manifold cutcell mesh 2	48
3.5	Turbolence Model	50
3.6	Velocity-Inlet BC	51
3.7	Pressure-Outlet BC	51
3.8	Wall solid BC	52
3.9	Output Parameter	52
3.10	Solution Method	53
3.11	CFD Post Window	56
3.12	Contour of Pressure Inlet1	57
3.13	Stream Line, Inlet1	58
3.14	Contour of Pressure Inlet2	59
3.15	Stream Line, Inlet2	59
3.16	Contour of Pressure Inlet3	60
3.17	Stream Line, Inlet3	60
3.18	Contour of Pressure Inlet4	61
3.19	Stream Line, Inlet4	61
4.1	Domain Set Necking	65
4.2	Domain Display Necking	66
4.3	Moving Set Necking	67
4.4	Moving Display Necking	67

4.5	Display Points Necking	68
4.6	Preview Points Necking	68
4.7	Solve Panel	69
4.8	Preview Panel	69
4.9	Morphing Panel	70
4.10	Domain Set	74
4.11	Domain Disp	74
4.12	Surface Point Display	75
4.13	Surface Point Preview	75
4.14	Surface Motion Set	76
4.15	Surface Selection Set	76
4.16	Set Moving	77
4.17	Display Moving	77
4.18	Display Points	77
4.19	Set Moving 2	81
4.20	Set Moving 3	81
4.21	Display Moving 2	82
4.22	Display Moving 3	82
4.23	Set Moving 1	82
4.24	Display Moving 1	82
4.25	Display Points	83
4.26	Preview Points	83
4.27	Set Domain	88

4.28	Display Domain	88
4.29	Set Moving	88
4.30	Display Moving	88
4.31	Display points	88
4.32	Preview Points	88
4.33	Surf Set	89
4.34	Set Domain	93
4.35	Display Domain	93
4.36	Set Moving	93
4.37	Display Moving	93
4.38	Display Points	94
4.39	Preview Points	94
4.40	Surf Set	94
5.1	Workbench screenshot	99
5.2	Swelling and Necking Curve Results	101
5.3	Decreasing Manifold Section Results	103
5.4	Rotating Outlet Results	105
5.5	Single Pipe Modifications Results	107
5.6	Coupled Results Graphic	108
5.7	Optimized Geometry	109
5.8	Morph and baseline geometry comparison	110
5.9	Morph geometry Streamline	111
5.10	Morph Geometry Pressure Contours	112

List of Tables

3.1	Baseline Results	54
5.1	Swelling and Necking Curve Results	100
5.2	Decreasing Manifold Section Results	102
5.3	Rotating Outlet Results	104
5.4	Single Pipe Modifications Results	106
5.5	Percentage of pressure decreases	109

Bibliography

- [1] Giuseppe Vairo, “*A numerical model for wind loads simulation on long-span bridges*”, ELSEVIER, 2003
- [2] Marco Evangelos Biancolini, “*Mesh morphing and smoothing by means of Radial Basis Functions (RBF): a practical example using Fluent and RBF Morph* ”, 2011
- [3] E.Simiu, H.Scanlan “*Wind effects on structures*”, Wiley-Interscience, 1996
- [4] J. Anderson Jr, “*Fundamentals of aerodynamic*”, McGraw Hill, 2001
- [5] M. Olesen, “*Automatic topography optimization of automotive exhaust system with a continuous adjoint cfd-method in open foam*”, Open source cfd, 2009
- [6] M. Cross, D. Smith, “*Generic Automotive Engine Intake Port Optimisation* ”, Advantage CFD, 2005
- [7] P. Canhoto,A. Reis, “*Optimization of fluid flow and internal geometric structure of volumes cooled by forced convection in an array of parallel tubes*”, ELSEVIER, 2011

-
- [8] J. Galindo, J.M. Lujran, J.R. Serrano, V. Dolz, S. Guilain, “*Design of an exhaust manifold to improve transient performance of a high-speed turbocharged diesel engine*”, ELSEVIER, 2004
- [9] M. Landon, “*Shape Optimization of Hamilton-Sundstrand’s 4507419 Fan Duct Using Sculptor & Fluent*”, President Optimal Solutions Software, 2006
- [10] C. Hinterberger, M. Olesen, “*Industrial application of continuous adjoint solvers for the optimization of automotive exhaust system*”, CFD & Optimization, 2011
- [11] P. Rudolf, M. Desova, “*Flow characteristics in curved ducts*”, University of west bohemia, 2007