Fluid Mechanics with CFD Exercises

Mario Ibañez Olvera
Irma Hernández Casco
Juan Alfonso Salazar Torres

PUBLICACIONES
UAEM

Fluid Mechanics with CFD Exercises

Fluid Mechanics with CFD Exercises

Mario **Ibañez Olvera**
Irma **Hernández Casco**
Juan Alfonso **Salazar Torres**

PUBLICACIONES
ESTADOS UNIDOS MEXICANOS
UAEM

Toluca, 2020

Fluid Mechanics with CFD Exercises

Mario Ibañez Olvera
Irma Hernández Casco
Juan Alfonso Salazar Torres
Autores

Mario Ibañez Olvera
Traducción

Primera edición: junio 2020

ISBN 978-607-633-160-6 (versión PDF)

Imagen de la portada: https://www.freepik.es

El presente libro cuenta con la revisión y aprobación de dos pares doble ciego externos a la Universidad Autónoma del Estado de México. El arbitraje estuvo a cargo de la Secretaría de Investigación y Estudios Avanzados, según consta en el expediente número 44/2017.

Index

Introduction

This effort is the result of laboratory practices developed with the purpose of furthering abilities in engineering students, by means of industrial applications, solved exercises and series with answers, as well as examples produced via free software.

Over the 2016B semester and thanks to the teaching of fluid mechanics at *Unidad Académica Profesional Tianguistenco*, it was found that students require to make examples as regards the proper use of metric units, metric unit conversion and dimensional analysis, which will allow them to properly use and handle variables and equations often used in engineering.

Most commonly students only analyze examples put forward in the lesson and probably will do some research in a book, however they do not tend to investigate more than the minimum required to pass the course; one way to catch their interest and help them dig deeper into specific subjects and comprehend better the subjects taught in class is to use free software with the purpose of presenting some examples. By using these tools, we can counteract the fact that students use exclusively computer tools presented at school, which tend to be very expensive. Our purpose is to help them explore free alternatives.

After the observations above, we were able detect the importance of practicing examples with industrial applications. This book contains suggested subjects to aid the teaching of fluid mechanics in a classroom, which favors the students' comprehension and gives them useful tools to handle the problems they will have to face in the industry.

In order to accomplish this, we expose the main subjects to study in a basic course in a hierarchic manner: Introduction and basic concepts, statics of fluids, continuity equation, energy general equation, momentum equation, design of pipeline systems and equipment selection.

Answers to every exercise at the end of each unit are included. Surely contributions that complement the exercise can be made simultaneously as the example is solved.

Special thanks to Universidad Autónoma del Estado de México, particularly to *Secretaría de Investigación y Estudios Avanzados* and to *Unidad Académica Profesional Tianguistenco*, and also to the program FORDECYT 273496 for their infinite support in the preparation of this book.

Fluid Mechanics

Fluid mechanics is a branch of mechanic science in charge of studying the behavior of static and moving fluids, in other words, of those substances that can be found in liquid or gas state, whose molecules stay separated and adapt to the shape of the container which holds them.

Figure 1

Studying this discipline enables us to understand the effect of the properties of fluids under specific conditions, this can help us predict their behavior and use this knowledge to select and use the proper equipment for specific systems such as hydraulic installations in houses, mining industry, food industry, oil industry, automobile and aeronautic systems, etc.

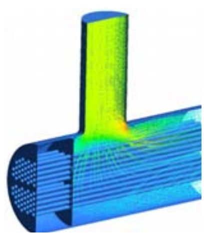


Figure 2

Fluid mechanics is a compulsory subject in many engineering programs and sc iences. Its study requires knowledge of the main properties of matter, essentially those of which the fluid's behavior heavily depends on such as viscosity, superficial tension, density as well as general concepts of physics, statics and dynamics and the proper use of mathematical tools such as integral and differential calculus and software that allows solving the governing equations of this discipline.

Among the abilities that the student of fluid mechanics and of any other engineering field must have is the proper use of metric units in different systems, use of scientific notation and the skill to make dimensional analysis of various mathematical equations.

Formally, fluids are substances deformed under the action of shearing stress, no matter how little this stress is. To properly understand this definition, it is necessary to remember that stress is force applied upon a specific area. There are two types of stress: normal and shear. The first one is applied perpendicularly to the area as pressure. The second is applied tangentially on the surface, as a torsional stress. In the case of fluids, we may think that they comprise successive layers of matter, if we slide a body through the surface of the fluid, the fluid will always deform. On the other hand, if we are dealing with solid objects the shear stress will have to be very big so that it is deformed.

In order to dig deeper into the properties of a fluid, it is necessary to speak of the continuous supposition, which consists in supposing that even though the atoms and molecules of a fluid are separate, it is considered that there are no empty gaps, and so, most of the space is occupied by matter; there is continuity in the matter that constitutes the fluid, this allows us to assume that the average value of the properties of the fluids is constant.

Among fluids' properties there are:

Density

The relation between the mass and volume of a substance

$$\rho = \frac{masa}{volumen} = \frac{m}{v} \qquad \frac{kg}{m^3}$$

It can be expressed in terms of relative density or specific gravity, which is a relation between the density of the substance of interest between the density of water in standard conditions of pressure and temperature.

$$DR = S = \frac{\rho_{\text{int}eres}}{\rho_{water}}$$

Viscosity

Viscosity is the capability of a fluid's molecules that allows them to move according to the molecules around them. If we think of a laminar fluid as the consequence of layers of fluids piled one over another, the relative movement of the layers under the action of a shear stress will depend on viscosity. The relation between shear stress and the movement of the layers is given by the following equation:

$$\tau_{xy} = -\mu \frac{du}{dy}$$

Which shows that the shear stress (τ) applied on a fluid that moves in x direction (u) with respect to the perpendicular distance to the area where such stress is applied (y). The negative sign is due to how heat flows from the areas with the highest temperatures to the ones with lower temperatures, the shear stress flows from the areas with higher velocity to the ones with lower velocity. According to the expression showed before, viscosity in the international unit system is expressed in N s / m^2.

Superficial tension

Superficial tension is the resistance that a fluid shows before breaking its bonds in the surface. In a fluid the molecules at the center are attracted in every direction by the molecules surrounding them, which allows the total force on them to be zero; on the other hand, the molecules on the surface are only attracted by the molecules on the inside of the fluid, and so the total force is towards the inside of the surface, that is why a fluid's surface is more or less curved depending on the intermolecular attractions.

Volumetric expansion coefficient

$$\beta=3\alpha$$

Compressibility coefficient

$$k= \frac{1}{v} \quad \frac{\Delta v}{\Delta p}$$

Flow

In fluid mechanics, flow is the amount of matter that crosses through an area in a specific amount of time. Volumetric flow and mass flow are used very often. Volumetric flow is expressed in m^3/s and mass flow is expressed in kg/s.

$$vol = VA$$

Dimensional Homogeneity

The equations used in engineering must be dimensionally homogenic, which means that the dimensions and units used must be the same. To achieve this, we use the fundamental dimensions and their units in the interest system.

For instance, to ascertain the dimensions of $\dot{W}_b$ in the following equation we use dimensional analysis.

$$\dot{m}\left(\frac{P}{\rho}+\frac{V^2}{2}+gz\right)=\dot{W}_b$$

Since each of the terms inside the parentheses have L^2/s^2 dimensions and the mass flow uses M/s, the dimensions used for W_b are

$$\frac{M}{s}\frac{L^2}{s^2}=\frac{ML}{s^2}\frac{L}{s}=\frac{(Fuerza)(dis\tan cia)}{tiempo}=\frac{trabajo}{tiempo}=potencia$$

And so, its unit in the international system is Watt (W), while in the British system, it is horse power (hp).

$\dot{m}$, kg/s; Mass flow
$\dot{vol}$, m^3/s; Volumetric flow
$\dot{\rho}$, kg/s; Density
V, m/s; Velocity
d, m; Pipeline diameter

Scientific Notation and Engineering Notation

In exact sciences and engineering it is necessary to express quantities in scientific notation and engineering notation, as it corresponds. With the purpose of making the values of different variables more comprehensible, each science and engineering branch has metric scales, depending on the context in which one works; for instance, it is very common to use small quantities in laboratories, so expressing that 0.001 L of a substance has been added in a test tube can be better understood by using scientific notation as 1 mL. On the other hand, the energy supply in big industries such as mining and automobile may be very large, this way it is expressed in MW.

Figure	Scientific notation (x 10^n)	Prefixes
0.01 L	1 x 10^{-2}	1 dL
1234 J		
		23 MW
	23 x 10^5 m	
1234567 m/s		

Table 1

Pressure

Fluid statics is a branch of mechanics that studies nonmoving fluids, this means those for which the sum of all the forces acting on the fluid is zero. This branch of science is very important because of the frequency with which we need to work with static fluids that are artificially or naturally contained and whose effects over what contains them are very important for aspects as safety, design or to choose which material to use. The force applied by a fluid on the surface that contains it is due to the effect of the pressure exercised by the fluid column present in the infinitesimal section of the area.

Force can be ascertained as:
Force= mg=(Vol)g=($A_{section}h_{fluid}$) g
While pressure is given by:
Pressure= Force/$A_{section}$ = ($A_{section}h_{fluid}$) g/$A_{section}$= gh_{fluid}

It is necessary to understand how pressure is expressed, atmospheric pressure (P_{atm}) is that which the air applies on an area, its value is high at sea level and diminishes as altitude increases because the height of the air column decreases, also air density decreases as the altitude increases.

Manometric pressure (P_{man}), it is expressed in reference to atmospheric pressure because the measurement instruments are calibrated to such pressure. This way, a manometer will read zero when exposed to the atmosphere and will begin to give values different from zero when the pressure is higher or lower.

In many cases, it is necessary to specify absolute pressure (P_{abs}), this means the total pressure of the system, which includes the manometric pressure and the atmospheric pressure, which allows us to unify operation conditions and facilitate the reproduction of the ambient conditions for the experiment in any location in the globe.

In cases where the pressure is lower than the atmospheric, this means that a part of the air contained in atmospheric conditions has been evacuated, it is called vacuum pressure, the maximum vacuum pressure that we can get would be if all the gas molecules where evacuated from the system and it would be absolute zero.

An example of the various ways to express pressure is shown: at the sea a scuba diver is exposed to higher pressure as he dives deeper; on the other hand, as he ascends back to the surface the pressure lowers, this is caused because pressure rises as the fluid column that the system supports is larger.

The pressure felt by the scuba diver as he dives to 20 m underwater in salt water (ρ=1030 kg/m^3) is 202.86 kPa manometric and the absolute pressure that he feels is 303.386 kPa, when the atmospheric pressure is 101.3 kPa (P_{atm}).

If a scientist wants to study the properties of a certain material for the scuba diver's watch, such scientist will have to apply an absolute pressure of 303.386 kPa, reproducing the conditions where the watch will be working on, regardless of the atmospheric pressure of the place the scientist is.

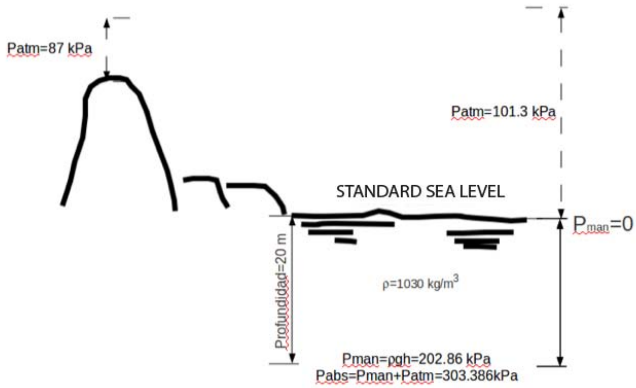


Figure 3

Forces Applied on Flat Submerged Surfaces

On certain occasions, it is required to know the force applied by a fluid over the surface of the container that holds it, for instance to ascertain the necessary force to open a liquid ring vacuum gate in an industrial process or that necessary to maintain a welded section of a tank containing certain mineral pulp, the thickness of a material in certain sections of a container, etc.

To calculate the force applied by the fluid on the surface, it is necessary to divide the area of interest into sections of infinitesimal size, since the pressure applied by the fluid will rise as the areas go deeper, calculating the force in each of them and adding them.

The height of the liquid over the system is expressed in function of the angle that forms the surface of interest with the fluid level (y sin α), which allows us to find out the vertical height of the liquid column on each section of the area, thus force is calculated as:

$$F=\int PdA=\rho gy(sen\alpha)dA=\gamma sen\alpha\int ydA$$

In the equation above the integral is defined as the first moment of area, and so force is expressed as:

$$F = \gamma\bar{y}sen\alpha$$

Where is specific weight, , is the distance from the level of the fluid to the centroid of the area of interest with respect to the inclination of this area.

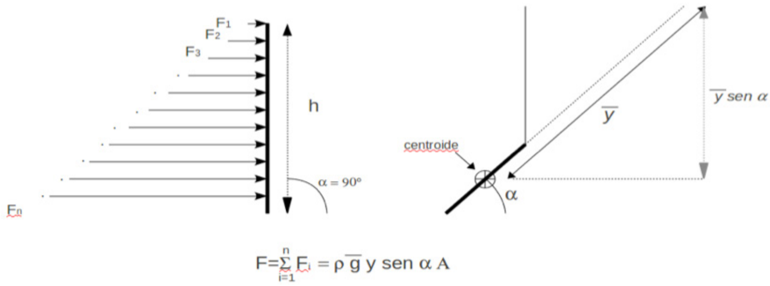


Figure 4

The calculated force represents the sum of the pressure applied on each infinitesimal point of the area of interest, now we must calculate on which point the force mentioned before (is located, and so the calculated force located at this point balances the forces applied by the fluid over and under the calculated force, and so a momentum balance is carried out,

$$y_{cp}F = \int ydF = \int yPdA = \int y(\gamma ysen\alpha)dA = \gamma sen\alpha \int y^2 dA$$

Where the integral on the right is defined as second momentum of inertia of the area which is given by:

$$I_0 = \bar{I} + \bar{y}^2 A$$

Where is the inertia momentum, and so the momentum balance turns to:

$$y_{cp}F = \gamma sen\alpha(\bar{I} + \bar{y}^2 A)$$

$$y_{cp} = \bar{y} + \frac{\bar{I}}{\bar{y}A}$$

Where is the distance (Parallel to the inclination of the area of interest) from the level of liquid to the point where the sum of momentums of the forces applied by the fluid on every infinitesimal section.

One of the most common applications of the calculation of forces on flat or curved surfaces is the design of equipment and structures; for instance, the following exercise: It is required to design a container for a slurry mix used in food industry. Because of the normativity, the container must be made with welded stainless steel. Ascertain the force that the weld must resist, as well as the effort to which a 1-cm thick weld rod is submitted.

Forces Applied on Submerged Curved Surfaces

It is often required to calculate the forces applied by a fluid on curved surfaces, this is because it is better to have curved sections of only one material than flat sections per unit, because in such areas, defects frequently appear in the material, though this can be due to design and aesthetical aspects.

To calculate the force that a curved section must be able to handle, it is important to consider that the fluid applies an impulse in horizontal direction, as calculated for a flat section, considering an inclination angle with respect to the fluid surface bigger than 90°. Besides, there is a vertical force corresponding to the weight of the fluid deposited on the curved section.

In order to make a methodological analysis, the area under analysis is divided into three sections: The section corresponding to the horizontal force (Section A), which is found next to the edge of the curve and toward the bulk of the liquid, a vertical section located above the section that contains the curved surface (Section B), and the section that contains the curved area and that is delimited by it and by the two sections mentioned before (Section C).

The methodology to ascertain the force applied over the curved surface is presented below:

1) The horizontal force at section A is calculated, taking as area of interest the area projected by the curve and following the procedure mentioned before to calculate the force on flat surfaces, locating as well as the pressure center.
2) The vertical force applied on section B (F_{v1}) is determined, which corresponds to the weight of the fluid in the volume of the prismatic figure and is located in the centroid of figure .
3) The vertical force applied by the weight of the fluid contained in section C (F_{v2}) is determined and is located in the centroid of the curve.
4) The total vertical force is calculated (F_{VT}), adding both of the vertical forces calculated earlier, moreover it is calculated the point where the first one is. To achieve it, a sum of momentums of both vertical forces and total vertical force is made.
5) Net force (F_R) is calculated using F_h and the total vertical force $F_{VT,}$ calculating the corresponding inclination angle.
6) The resulting force is located. To achieve this, the horizontal force is located in, the total vertical force is located in its respective pressure center is located at the point where both forces intersect; the resulting force (F_R) is collocated with the calculated inclination angle.

Fluid Flow

We deal more often with moving fluids than with static fluids, whether it is to find out a river's flow or to select the equipment to transport a fluid in industry. For example, if we are dealing with oil, we need to know the characteristics and behavior of the fluid; that is why, in this section, we will explain the particularities of fluids that will allow us to study them properly.

The behavior of a fluid can be studied from two different standpoints: one where it is a fixed reference system selected and it is called Eulerian and a fixed volume control is utilized. From the Lagrangian standpoint, the system is analyzed by means of what passes through the fixed system from a perspective where a particle of the fluid is selected and so the behavior is determined following its trajectory.

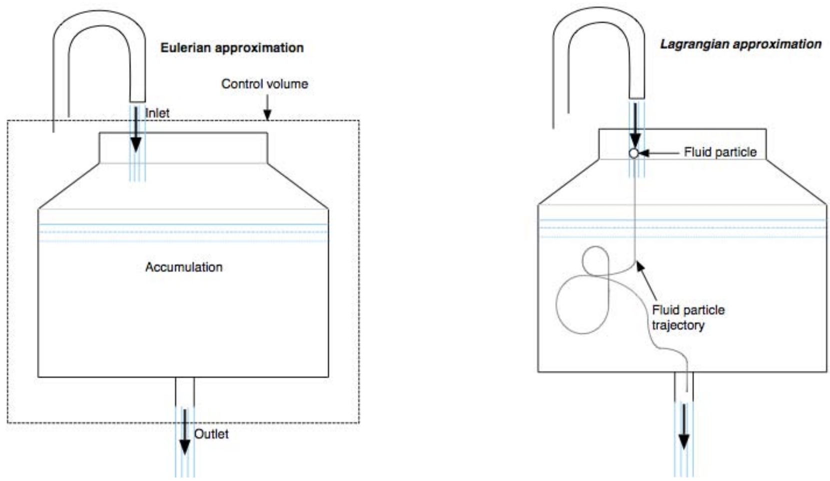


Figure 5

As mentioned above, a fluid is defined as a portion of matter that can be deformed by any slight action of shear stress, which is applied over one or several directions on a specific area. Hence, a fluid can be studied in different directions for each reference system, which will allow us to properly describe the fluid.A laminar fluid can be simplified as unidimensional, whereas a turbulent fluid has to be tridimensional; this behavior is noticed in the trajectory's lines that stablish the path followed by a fluid particle, which means, they are determined using the Lagrangian way.

Systems are needed more often than control volumes. A system is that which is delimitated by real or imaginary frontiers, through which matter and energy can or not pass through. A control volume is a space delimitated by real or imaginary frontiers, through which there is exchange of energy and matter. In physics analysis, we tend to work with systems, while it is not common to work with control volumes, however there is a way to relate a system to a control volume; through the Reynolds Transport Theorem, which tells us that the change in the properties of a system is equal to the net flow of this property through the system's frontiers, plus the sum of this property generated inside the control volume.

For instance, if we fix the change of energy in a water heater that is on as a control volume, the quantity of water that is flowing in and out to concentrate energy must be quantified through the frontiers, as well as considering the energy generated or that escapes from the control volume over the analysis on any other mechanism.

Reynolds Transport Theorem

Reynolds Transport Theorem establishes that if B is a property (mass, energy, force, etc.) and b is the value of this property per mass unit (specific property), the change of property over time in the system is equal to the accumulation of the property in the control volume, plus the net quantity of this property, entering in or out of the control volume. This means,

$$\left.\frac{dB}{dt}\right|_{sistema} = \frac{d}{dt}\int_{Vol\,C} \rho b \; dVol \quad + \int_{Sup\,Vol\,C} \rho \; b \; \bar{V} \cdot \overline{dA}$$

Where the term on the left corresponds to the change of property B respect to the time in the system. The first term after the equal sign corresponds to the generation or consumption of this property respect to time in the control volume. The second term corresponds to the sum of the entrances and exits of this property, through the surfaces of the control volume.

In the water heater example, the Reynolds Transport Theorem would be:

B=E,	energy, J
b=e=E/m,	specific energy, kJ/kg

$$\left.\frac{dE}{dt}\right|_{sistema} = \frac{d}{dt}\int_{Vol\,C} \rho e \; dVol \quad + \int_{Sup\,Vol\,C} \rho \; e \; \bar{V} \cdot \overline{dA}$$

Where the term on the left would correspond to the thermodynamic analysis of the initial and final energy of the system. The first term on the right side of the equation corresponds to the generated or consumed energy inside the control volume; this means, the accumulation that can be either positive or negative. The last term of the equation corresponds to the sum of the energy going through the control volume together with the fluid that flows in and out of the frontier; this term is known as connective term.

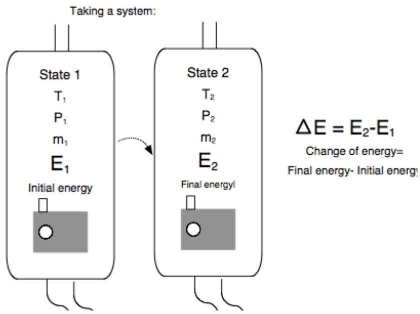


Figure 6

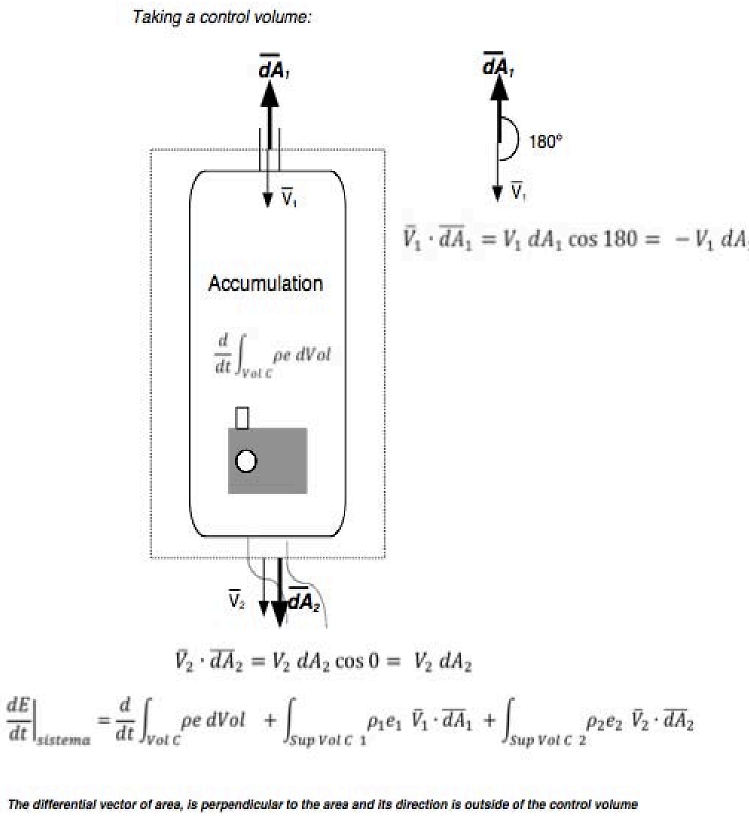


Figure 7

Continuity Equation

The continuity equation is one of the fundamental equations of fluid mechanics that allows analyzing the change of mass with respect to time in a control volume; for this, the Reynolds Transport Theorem is used with mass (M) as the studied property:

B= M, kg
b=M/m=1

$$\frac{dM}{dt}|_{system} = \frac{d}{dt}\int_{Vol\,C} \rho(1)\, dVol + \int_{Sup\,Vol\,C} \rho\,(1)\;\bar{V}\cdot\overline{dA}$$

Owing to this, the continuity equation looks like this:

$$\frac{dM}{dt}|_{system} = \frac{d}{dt}\int_{Vol\,C} \rho\, dVol + \int_{Sup\,Vol\,C} \rho\;\bar{V}\cdot\overline{dA}$$

In many systems of study in engineering, we work with static fluids, that is why the term on the left side of the equation is equal to zero; and so, the continuity equation ends up as:

$$\frac{d}{dt}\int_{Vol\,C} \rho\, dVol \;+\int_{Sup\,Vol\,C} \rho\;\bar{V}\cdot\overline{dA} = 0$$

The first term corresponds to the mass accumulated in the control volume and can be either positive or negative; this means that the control volume can be full or empty. The second term corresponds to the mass that breaks through the control volume frontiers, it may be flowing in or out of it. The entrance or exit of mass is calculated with the second term of the continuity equation, this means, with the point product of the velocity vector, because the point product $\bar{V}\cdot\overline{dA})$ is defined as the magnitude of the velocity, multiplied per the magnitude of the differential area per the cosine of the angle formed by this two vectors; this point defines whether the fluid is entering or exiting the control volume.
In the following figure, a container filled with alimentary syrup is displayed, with S=1.82. When the syrup level inside the tank is equal to 2 m, a 2-inch diameter valve opens, supposing that the exit speed is constant and equal to 8.85 m/s.
Given these facts, solve the following problem: what is the change of height of the liquid's level? and how long does it take for the liquid to be as low as 0.5 m if the diameter of the tank is 3 m?

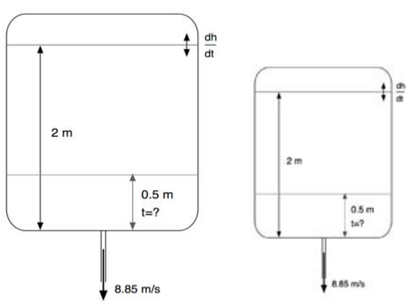


Figure 8

The tank is being drained, for if there is accumulation, this means, that the height *h* of the tank is changing with respect to time, there is only one pipeline of entrance or exit, and so, that is why the second term of the continuity equation turns into one term, turning the equation into the following:

$$\frac{d}{dt}\int_{Vol\ C} \rho\ dVol \ + \int_{Sup\ Vol\ C} \rho\ \bar{V} \cdot \overline{dA} = 0$$

$dVol = A_{tank} dh$

$$\frac{d}{dt}\int_{Vol\ C} \rho\ A_{pipeline} dh\ + \int_{pipeline} \rho\ \bar{V}_{pipeline} \cdot \overline{dA}_{pipeline} \ = 0$$

In the pipeline, the vector of the difference of area is outside, that is why it is parallel to the velocity and its direction is also outside the control volume, and so, $\bar{V}_{tubería} \cdot \overline{dA}_{tubería} = V\ dA \cos 0$, integrating and grouping constants.

$$\rho\ A_{tanque} \frac{dh}{dt} + \rho\ \ VA\cos(0) = \ 0$$

And so, the height change in the liquid's level respect to time is:

$$\frac{dh}{dt} = -\frac{\rho\ V_{pipeline}\ A_{pipeline}}{\rho\ A_{tank}} = -\frac{\left(8.85\frac{m}{s}\right)(\pi)(0.0254m)^2}{(\pi)(1.5m)^2} = -\ 0.0025\ m/s$$

This means that the level of the tank decreases at a rate of 0.0025 m/s, while the time when the height of the liquid inside the tank decreases to 0.5 m will be:

$$\frac{dh}{dt} = -0.0025\ m/s$$

$$\frac{dh}{0.0025\ m/s} = -dt$$

$$-\int dt = \frac{1}{0.0025\frac{m}{s}}\int_{2\ m}^{0.5\ m} dh = 394.07\frac{s}{m}\ (h|_{0.5m} - h|_{2m})$$

$$t = 591.1\ s$$

It is often needed to know the force associated to a control volume, and so a force analysis is carried out, it can be represented in a momentum diagram similar to the process that would be made if it were a free body diagram. To relate the force analysis in a system to the force analysis in a control volume, the Reynolds Transport Theorem is used.

The sum of forces in a direction *i* in the control volume is

$$\Sigma F_i = ma_i$$

As acceleration is dv_i/dt, then,

$$\Sigma F_i = ma_i = m\frac{dv_i}{dt} = \frac{d(mv_i)}{dt}$$

According to Reynold's Transport Theorem:

$$\frac{dB}{dt}\Big|_{system} = \frac{d}{dt}\int_{Vol\,C} \rho b\, dVol + \int_{Sup\,Vol\,C} \rho\, b\, \bar{V}\cdot\overline{dA}$$

And can be equalized to the sum of forces in the control volume for each component:

$$\Sigma F_i = ma_i = m\frac{dv_i}{dt} = \frac{d(mv_i)}{dt}$$

Examples

1. A water nozzle is used to perpendicularly clean grease from a metallic surface submitted to lamination. If the water nozzle flow is equal to 40 Kg/s and its diameter is 2 inches, determine the force components that the support piece that weighs 320 Kg must resist.

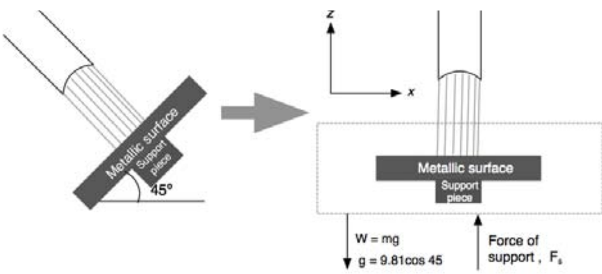


Figure 9

According to the image, the reference axis can rotate and will only be a momentum associated to the impact of the fluid in direction of the z axis, even though the fluid moves in direction x as it impacts against the surface, it is supposed to make it in a symmetric way, and so the forces associated to the fluid are annulated in this direction.

And so, the sum of forces in the control volume will be:

$$\Sigma F_z = F_s - W$$

And the equation of the momentum in static state will be:

$$\frac{d(mv_z)}{dt}\Big|_{system} = \int_{Sup\,Vol\,C} \rho\, v_z\, \bar{V} \cdot \overline{dA} = \rho\, V\, (VA\, cos\, 180)$$

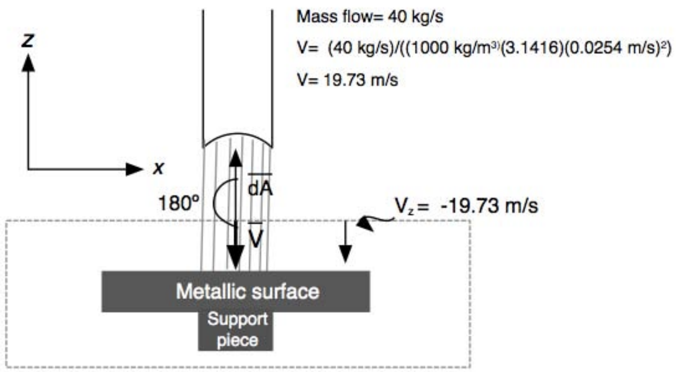


Figure 10

And so, the force that the support piece must resist will be:

$$\Sigma F_z = F_s - W = \rho\, V\, (VA \cos 180) = \rho\ V^2\ A$$

$$F_s = W + \rho\ V^2\ A = (320\,\text{kg})\left(9.81\frac{m}{s^2}\cos 45\right) + \left(1000\frac{kg}{m^3}\right)\left(19.73\frac{m}{s}\right)^2 (\pi)(0.0254m^2) = 33.28\ kN$$

2. A diameter reduction is made to project water to the atmosphere through a nozzle as the one shown below. If the nozzle has to be fixed with a metallic ring to the floor, determine the force the ring must resist.

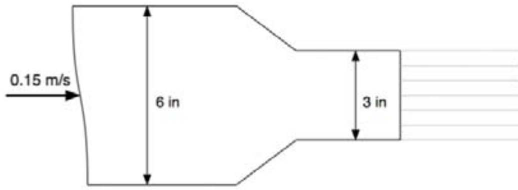


Figure 11

In this case we have a confined fluid, which is why the effect of the pressure will be important, because inside the pipeline the fluid applies a force towards the interior of the control volume, and at the exit the atmosphere applies a force towards the interior of the volume control, the sum of forces will be:

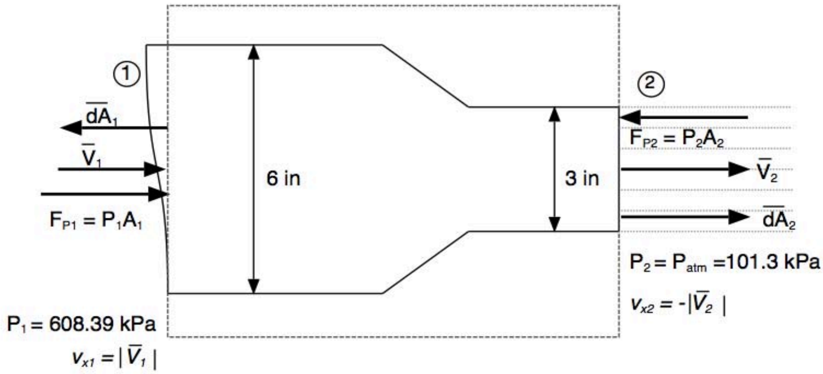


Figure 12

$$\Sigma F_z = -F_s + F_{P1} - F_{p2} = \frac{d}{dt}\int_{Vol\ C} \rho v_i\ dVol \quad + \int_{Sup\ Vol\ C} \rho\ v_i\ \overline{V} \cdot \overline{dA}$$

$$\Sigma F_z = -F_s + F_{P1} - F_{p2} = \int_{Sup\ Vol\ C1} \rho\ v_{x1}\ \overline{V_1} \cdot \overline{dA_1} + \int_{Sup\ Vol\ C2} \rho\ v_{x2}\ \overline{V_2} \cdot \overline{dA_2}$$

$$\Sigma F_z = -F_s + F_{P1} - F_{p2} = \rho v_{x1} V_1 A_1 \cos 180 + \rho v_{x2} V_2 A_2 \cos 0$$

$$-F_s = -F_{P1} + F_{p2} - \rho v_{x1} V_1 A_1 + \rho v_{x2} V_2 A_2$$

And so, the force that the ring will have to resist along the *x* axis is:

$$-F_s = -(608390\ Pa)(\pi)(0.0762m)^2 + (101300\ Pa)(\pi)(0.0381m)^2$$

$$-\left(1000\frac{kg}{m^3}\right)\left(8.22\frac{m}{s}\right)\left(8.22\frac{m}{s}\right)(\pi)(0.0762m)^2$$

$$+\left(1000\frac{kg}{m^3}\right)\left(-32.98\frac{m}{s}\right)\left(32.89\frac{m}{s}\right)(\pi)(0.0381m)^2 = -16.81\ kN$$

There are no data about the weight of the pipeline in the z axis nor of the weight of the fluid in the volume of interest.

Energy Equation

Now the Reynolds Transport Theorem will be used to analyze the energy in a control volume starting from the physical and thermodynamic knowledge of the systems. The energy inside a system is mainly composed of three types of energy: kinetic, potential and internal energy.

If in this case the extensive property (B in Reynolds Transport Theorem) is total energy:

$$B = E = E_C + E_p + U = \frac{mV^2}{2} + mgh + U$$

Therefore, our specific property (b in Reynolds Transport Theorem) is,

$$b = e = e_C + e_p + u = \frac{V^2}{2} + gh + u$$

According to the 1st Law of thermodynamics, which states the conservation of energy,

$$\Delta E = Q + W$$

As we replace it on Reynolds Transport Theorem we obtain the following expression,

$$\frac{dE}{dt}\Big|_{system} = \frac{d}{dt}\int_{Vol\,C} \rho e\, dVol + \int_{Sup\,Vol\,C} \rho\, e\bar{V}\cdot\overline{dA}$$

Where

$$\dot{W}_{flujo} = Pd\dot{V}ol = P\frac{d\dot{m}}{\rho}$$

Considering there is no accumulation,

$$\frac{dE}{dt}\Big|_{system} = \int_{Sup\,Vol\,C} \rho\, e\, V\cdot dA = \dot{Q} + \dot{W}_{arrow} - \dot{W}_{turbine} - \dot{W}_{flow}$$

$$\int_{Sup\ Vol\ C} \rho \left(\frac{V^2}{2} + gz + u\right) V \cdot dA = \dot{Q} + W_{arrow}^{\cdot} - W_{turbine}^{\cdot} - P\frac{d\dot{m}}{\rho}$$

Moving the last term to the integral on the right side, we have,

$$\int_{Sup\ Vol\ C} \rho \left(\frac{V^2}{2} + gz + u + \frac{P}{\rho}\right) V \cdot dA = \dot{Q} + W_{arrow}^{\cdot} - W_{turbine}^{\cdot}$$

With the integration of one same line of current of laminar flow and between 2 points with constant properties, the equation becomes

$$\left(\frac{V_2^2 - V_1^2}{2} + gz_2 - gz_1 + u_2 - u_1 + \frac{P_2 - P_1}{\rho}\right)\dot{m} = \dot{Q} + W_{arrow}^{\cdot} - W_{turbine}^{\cdot}$$

When we divide it between *mg*

$$\frac{V_2^2 - V_1^2}{2g} + z_2 - z_1 + \frac{u_2 - u_1}{g} + \frac{P_2 - P_1}{\rho g} = \frac{\dot{Q}}{\dot{m}g} + \frac{W_{arrow}^{\cdot}}{\dot{m}g} - \frac{W_{turbine}^{\cdot}}{\dot{m}g}$$

As we rearrange it

$$\frac{V_2^2}{2g} + z_2 + \frac{P_2}{\rho g} + \frac{W_{turbine}^{\cdot}}{\dot{m}g} - \frac{\dot{Q}}{\dot{m}g} + \frac{u_2 - u_1}{g} = \frac{V_1^2}{2g} + z_1 + \frac{P_1}{\rho g} + \frac{W_{arrow}^{\cdot}}{\dot{m}g}$$

Where: $h_L = -\frac{\dot{Q}}{\dot{m}g} + \frac{u_2 - u_1}{g}$ is called head losses and they are associated to the energy lost because of friction, rugosity on the surface, accessories of the pipeline such as reducers, amplifiers, etc.

$h_t = \frac{W_{turbine}^{\cdot}}{\dot{m}g}$ is the head turbine loss and is associated with the energy extracted by the turbine in the system.

$h_b = \frac{W_{pump}^{\cdot}}{\dot{m}g}$ head pump is the energy added to the system by the pump.

And so, the general energy equation for a fluid will be:

$$\frac{V_1^2}{2g} + z_1 + \frac{P_1}{\rho g} + h_b = \frac{V_2^2}{2g} + z_2 + \frac{P_2}{\rho g} + h_t + h_L$$

We notice that on the right side of the equation, we have the initial energy of the system added to the energy that can be supplied by a pump; whereas on the left side, we have the final energy of the system added to energy lost because of friction, heating or a device that extracts energy (turbine), This way, point one in the analysis must be at the beginning of the flow, while point two must be the point at which the fluid has already traveled through the pipeline and lost energy.

Simplifying for the cases where there is no pump, assuming there are no head or head pump, the energy general equation would look like Bernoulli's equation

$$\frac{V_1^2}{2g} + z_1 + \frac{P_1}{\rho g} = \frac{V_2^2}{2g} + z_2 + \frac{P_2}{\rho g}$$

To ascertain head losses, $h_{L,}$ the friction effects must be considered because of the characteristics of the pipeline and the fluid (h_f) and the lower head losses associated to accessories such as expansions, contractions, etc (h_k).

To determine h_f, we must consider Re and relative rugosity:

$$Re = \frac{\rho V D}{\mu} = \frac{V D}{\nu} \text{ and } R.r. = \frac{k_s}{D}$$

With these two quantities it is possible to find out the friction factor, *f*, using Moody charts to calculate head losses from friction. These graphics are available in fluid mechanics books and electronic documents.

$$h_f = f \frac{L}{D} \frac{V^2}{2g}$$

These losses shall be calculated on each section of the pipeline where velocity values as well as diameter, rugosity or other factors change.

Speaking about lower head losses, for each accessory of the pipeline, the *k* coefficient must be determined and all the values of h_K must be summed. These values can be found in text books on fluid mechanics and electronic documents.

$$h_k = k \frac{V^2}{2g}$$

Head losses and minor losses are added to calculate head losses in the energy general equation.

As an example, for the system shown below in which 3 Kg/s of water are flowing through a pipeline made of galvanized iron, the required power supply for a pump with 68% of efficiency is calculated as follows:

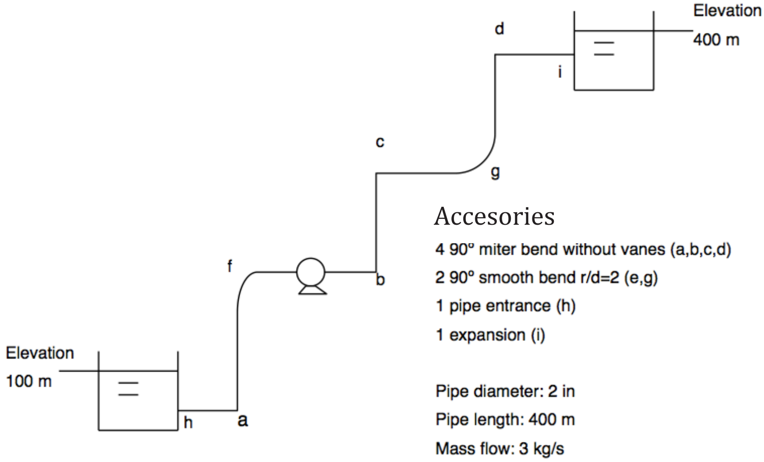


Figure 13

If we need to find out the power of the pump, it is necessary to establish the two points of analysis. The fluid is pumped toward the upper tank; that is why point one must be in the left side of the tank and point two must be on the right side, the known data to solve the general energy equation are:

Point 1 z_1=100 m $P_1 = P_{atm}$ V_1 =0	$V_{pipeline}$=1.53 m/s	**Point 2** z_2=400 m $P_2 = P_{atm}$ V_2 =0
	Re= $\frac{\rho V D}{\mu} = \frac{\left(1000\frac{kg}{m^3}\right)\left(1.53\frac{m}{s}\right)(0.05\,m)}{1.14\,x\,10^{-3} Pa\cdot s}$ $= 6.7\,x\,10^4$	
	k_s^*/D= 0.15 mm/50 mm=0.003	
	f^*=0.028	
	h_f= $f\frac{L}{D}\frac{V^2}{2g} =$ $0.028\left(\frac{400\,m}{0.05m}\right)\left(\frac{(1.53\,m/s)^2}{2\left(9.81\frac{m}{s^2}\right)}\right) =$ $26.73m$	

K_e=0.50* (Entrance of the pipeline, h)	K_b=1.1* (Pipe elbows straight without striations, a, b,c,d)	K_b=0.19* (soft pipeline elbows r/d=2, f y g)	K_E=1.00* (Expansión, D_1/D_2=0.00 y θ=180º)
$h_K = \sum K_i \frac{V^2}{2g} = (0.50 + 4(1.1) + 2(0.19) + 1.00)\frac{\left(1.53\frac{m}{s}\right)^2}{2(9.81\ m/s^2)} = 0.75\ m$			
$h_L=h_f+h_k$= 26.73 m + 0.75 m = 27. 48 m			
$\frac{V_1^2}{2g} + z_1 + \frac{P_1}{\rho g} + h_b = \frac{V_2^2}{2g} + z_2 + \frac{P_2}{\rho g} + h_t + h_L$			
$z_1 + h_b = z_2 + h_L$ $h_b = (z_2 - z_1) + h_L = (400\ m - 100\ m) + 27.48\ m = 327.48\ m$			
$\dot{W}_{pumo}=h_b\,\dot{m}g$= (327.48 m)(3 kg/s)(9.81 m/s²)=9.64 kW $\dot{W}_{real\ pump} = \frac{\dot{W}_{pump}}{\eta} = \frac{9.64\ kW}{0.68} = 14.17\ kW$			

*The values of k_s , K_e, K_B and K_E can be obtained from several tables in various bibliographic references. The value of *f* is obtained from the Moody chart which can also be found in several bibliographic references.

As previously mentioned, a fluid can be characterized by different properties, as well as by the conditions to which it is exposed. Often, we wish to simplify the conditions in which the fluid is at in order to analyze it, predict or manipulate its behavior.

Nevertheless, in other circumstances when a better and more detailed characterization of its behavior is necessary, it is impossible to ignore the variability of its properties, as well as the non-obligated uniform behavior of its molecules. In these situations, we require to use the Navier-Stokes Equations.

The Navier-Stokes equations allow analyzing, at a microscopic environment, the behavior of a fluid in order to find out in an accurate manner its characteristics and these are required when the friction effects or the dependency of the properties on the velocity of the fluid are important. Some examples of the application of the Navier-Stokes equations are found in the lines of current of the flow around an automobile, the flow prediction in the blades of a windmill and even to know the transport of substances through the bloodstream.

In this section it will be shown how the Navier Stoques equations are obtained for rectangular coordinates in the *x* axis. The procedure is similar to obtain the Navier-Stokes equations in the *y* and *x* axes, and also to obtain them in cylindrical and spherical coordinates.

To start with the analysis, a control volume of a fluid will be chosen, keeping in mind that the volume can be smaller and smaller until it is infinitesimal with Δx, Δy and Δz dimensions and on which the force analysis will be made.

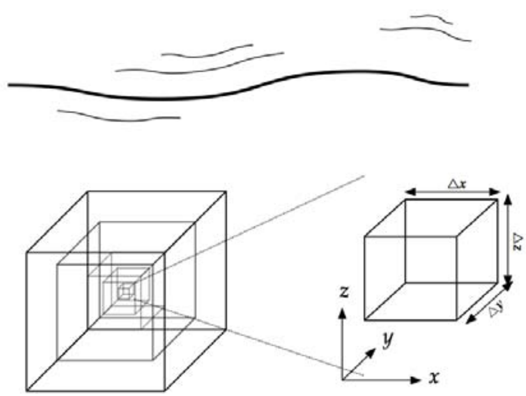


Figure 14

In the control volume of figure 13, four forces are acting: a) shear stress, this means the momentum transfer from a face to the opposite face; b) the transferred momentum by the fluid that goes through control volume; c) the force due to the pressure that the fluid applies on the interior of every face of the control volume; and, d) the weight in the analyzed direction. The effect of the sum of the mentioned forces can be appreciated in the movement of the control volume in the analyzed direction, this means the variation of its velocity with respect to time.

And so, the analysis of the forces in the *x* axis is given by:

a) Shear stress. It is which appears between the successive layers of the fluid and whose effort is affected by the viscosity of the fluid.

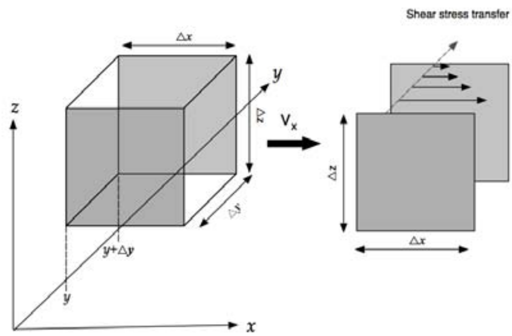


Figure 15

$$\left(\tau_{yx}\big|_{y} - \tau_{yx}\big|_{y+\Delta y}\right)\Delta x \Delta z$$

For a fluid that moves over the *x* axis and whose stress is being transferred in the *z* axis due to the viscosity (τ_{zx}), through the area *ΔxΔy:*

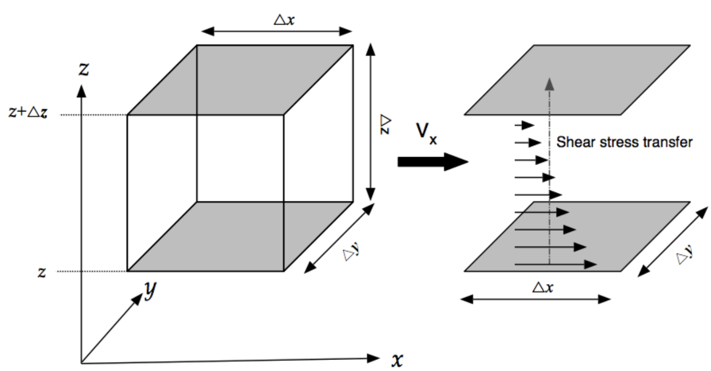


Figure 16

$$\left(\tau_{yx}\big|_{y} - \tau_{yx}\big|_{y+\Delta y}\right)\Delta x \Delta z$$

For a fluid that moves over the *x* axis and whose stress is transferred over the *x* axis due to the viscosity (t_{xx}), through the area *ΔyΔz:*

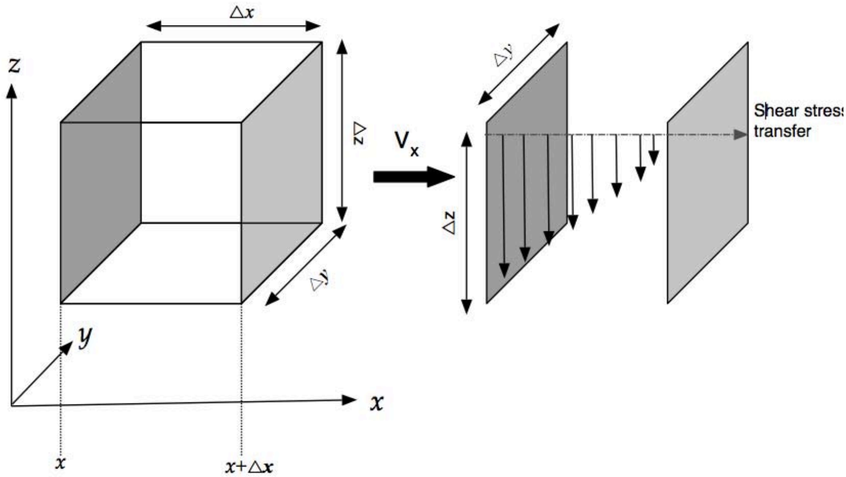


Figure 17

$$(\tau_{xx}|_x - \tau_{xx}|_{x+\Delta x})\Delta y \Delta z$$

b) Momentum balance is the effect of the force of the *x* component of the velocity that passes through the control volume, through any of its faces:

$$F_x = \int \rho\, V_x\, \bar{V} \cdot \overline{dA}\ ,\ \ |F_x| = \ V_x\, \dot{m} = \ \rho\, V_x V A$$

Because of the flow that enters and exits through the faces located in *x* and *x+Δx*, this means the mass flow in *x* that goes through the faces with area *ΔyΔz.*

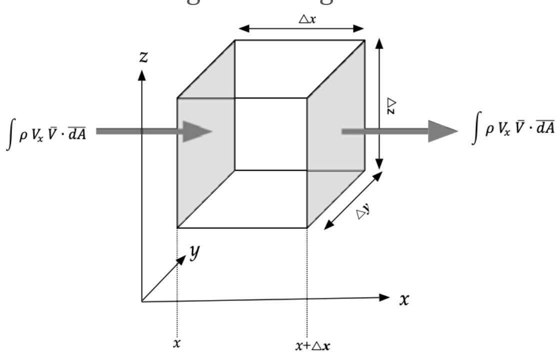


Figure 18

$$(\rho V_x V_x|_x - \rho V_x V_x|_{x+\Delta x})\Delta y \Delta z$$

Figure 18 shows the mass flow that enters and exists through the faces located in *y* and in *y+Δy,* through the area *ΔxΔz.*

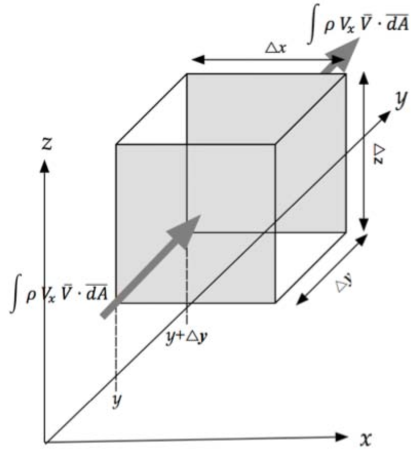


Figure 19

$$\left(\rho V_y V_x\big|_y - \rho V_y V_x\big|_{y+\Delta y}\right)\Delta x \Delta z$$

In the figure 19, we observe the mass flow that enters and exits through the faces located in *z* and *z* and *z+Δz* through the area *ΔxΔy.*

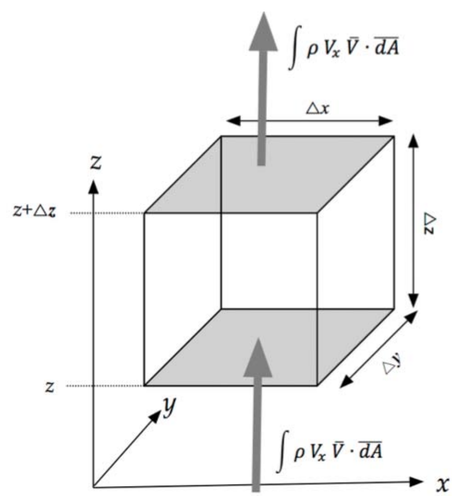


Figure 20

$$(\rho V_z V_x|_z - \rho V_z V_x|_{z+\Delta z})\Delta x \Delta y$$

c) Pressure force applied toward the inside of the control volume in the faces located in x and Δx with an area $\Delta y \Delta z$.

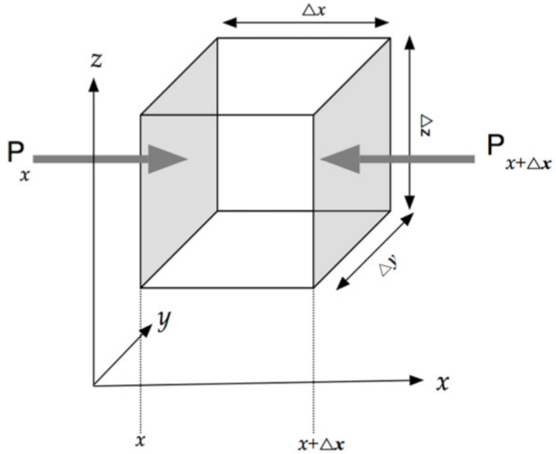


Figure 21

$$(P|_x - P|_{x+\Delta x})\Delta y \Delta z$$

d) Force due to the component x of gravity $F_g = mg_{x..}$

For the chosen system there is no gravity component in the x axis, albeit, the axis could rotate in a way where the effect of gravity in this direction is different from zero, and so this term would have to be included, because it may be required to analyze some study cases.

$$\rho g_x \Delta x \Delta y \Delta z$$

e) The sum of forces mentioned before is equalized to the accumulation, this means the variation of velocity over the x axis of the control volume:

$$\frac{d}{dt}\rho V_x \Delta x \Delta y \Delta z$$

As we add all the terms equal to the accumulation, and then divide it by the volume, we have:

$$\frac{d}{dt}\rho V_x = \frac{\tau_{yx}\big|_y - \tau_{yx}\big|_{y+\Delta y}}{\Delta y} + \frac{\tau_{zx}|_z - \tau_{zx}|_{z+\Delta z}}{\Delta z} + \frac{\tau_{xx}|_x - \tau_{xx}|_{x+\Delta x}}{\Delta x}$$
$$+ \frac{\rho V_x V_x|_x - \rho V_x V_x|_{x+\Delta x}}{\Delta x} + \frac{\rho V_y V_x\big|_y - \rho V_y V_x\big|_{y+\Delta y}}{\Delta y} + \frac{\rho V_z V_x|_z - \rho V_z V_x|_{z+\Delta z}}{\Delta z}$$
$$+ \frac{P|_x - P|_{x+\Delta x}}{\Delta x} + \rho g_x$$

As mentioned before, since we start the force analysis, the control volume is infinitesimal, which means its volume tends to zero and so the majority of the terms of this equation, we would have the negative definition of the derivate:

$$\lim_{\Delta x \to o} \frac{P|_x - P|_{x+\Delta x}}{\Delta x} = -\frac{dP}{dx}$$

As it is a multivariable analysis, the Navier-Stokes equation in the *x* axis would be expressed like this:

$$\frac{\partial}{\partial t}\rho V_x = -\left[\frac{\partial}{\partial y}\tau_{yx} + \frac{\partial}{dz}\tau_{zx} + \frac{\partial}{\partial x}\tau_{xx}\right] - \left[\frac{\partial}{\partial x}\rho V_x V_x + \frac{\partial}{\partial y}\rho V_y V_x + \frac{\partial}{dz}\rho V_z V_x\right] - \frac{\partial P}{\partial x} + \rho g_x$$

And so, an analysis can be made for axes *y* and *z*:

$$\frac{\partial}{\partial t}\rho V_y = -\left[\frac{\partial}{\partial x}\tau_{xy} + \frac{\partial}{dy}\tau_{yy} + \frac{\partial}{\partial z}\tau_{zy}\right] - \left[\frac{\partial}{\partial x}\rho V_x V_y + \frac{\partial}{\partial y}\rho V_y V_y + \frac{\partial}{dz}\rho V_z V_y\right] - \frac{\partial P}{\partial y} + \rho g_y$$

$$\frac{\partial}{\partial t}\rho V_z = -\left[\frac{\partial}{\partial x}\tau_{xz} + \frac{\partial}{dy}\tau_{yz} + \frac{\partial}{\partial z}\tau_{zz}\right] - \left[\frac{\partial}{\partial x}\rho V_x V_z + \frac{\partial}{\partial y}\rho V_y V_z + \frac{\partial}{dz}\rho V_z V_z\right] - \frac{\partial P}{\partial z} + \rho g_z$$

These equations are generally solved simultaneously with the continuity equation. As we are dealing with multiple variables and non-constant properties, the exact solution of these equations has been developed only for some specific cases, while its application to complex systems must be made with the help of specialized software, using numerical methods that will enable us to have a simultaneous solution to all the equations.

Next is presented the proper use of Navier-Stokes equations for two cases where there is an exact solution.

Parallel Plates Moving in Opposite Directions

Lubricant is placed between two large plates that are moving in opposite directions at a constant speed of 2 m/s. Solve the following case: find out the speed profile of the lubricant between the two plates.

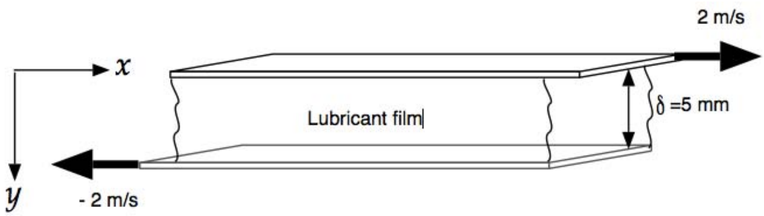


Figure 22

Initially the conditions of the system are stablished; the plates move in a stationary state, due to its constant speed, besides it is assumed that the thermodynamic property of the fluid is constant as well and uniform in the system, and the fluid is moving only along the direction over which the plates move, this means that the flow is unidimensional.

According to the diagram, the flow moves over the *y* axis, while the shear stress is transmitted over the *x* axis, this is why Navier-Stokes equation must be used in rectangular coordinates for the *y* axis:

$$\frac{\partial}{\partial t}\rho V_y = -\left[\frac{\partial}{\partial x}\tau_{xy} + \frac{\partial}{dy}\tau_{yy} + \frac{\partial}{\partial z}\tau_{zy}\right] - \left[\frac{\partial}{\partial x}\rho V_x V_y + \frac{\partial}{\partial y}\rho V_y V_y + \frac{\partial}{dz}\rho V_z V_y\right] - \frac{\partial P}{\partial y} + \rho g_y$$

The term on the left side of the equation mentioned before has a value equal to zero, because the system is stationary. The first term on the right side is different from zero, because the fluid is moving along the *x* axis, and the shear stress is being transferred over the *y* axis, the second and third terms on the right are equal to zero, as there is no transference, at the moment, over the *y* or *z* axes since the flow is unidimensional, and so $V_y=V_z=0$ and $d/dy(\Delta V_y V_y)=0$, that is why all the terms in the second parenthesis on the right side are zero, the term $dP/dy=0$ because the thermodynamic properties of the fluid are constant and the last term $r\Delta g_y=0$, as for the chosen system there is no gravity in the *y* axis and so the equation will be:

$$\frac{d}{dy}\tau_{yx} = 0$$

Solving by variable separation

$$\tau_{yx} = C_1$$

By definition $\tau_{yx} = -\mu\frac{dV_x}{dy}$, and so $-\mu\frac{dV_x}{dy} = C_1$, solving once again by variable separation

$$V_x = C_1 y + C_2$$

To determine the value of constants C_1 and C_2:

CF1 $y=0$ $V_y=2$ m/s
CF2 $y=d$ $V_y= - 2$ m/s

Assessing the conditions at the border, the speed profile equation is:

$$V_x = -\frac{4\frac{m}{s}}{\delta}y + 2\,m/s$$

And so, the distribution of velocity between the plates is linear and does not depend on the properties of the fluid.

y, mm	V_x, m/s
0	2
0.5	1.6
1	1.2
1.5	0.8
2	0.4
2.5	0
3	-0.4
3.5	-0.8
4	-1.2
4.5	-1.6
5	-2

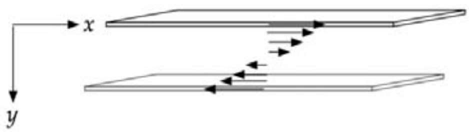


Figure 23

Natural Flow in an Inclined Wall

A fluid film slides by gravity along an inclined wall, as fluids in industrial processes do, as in mineral mud transporting, painting processes, etc.

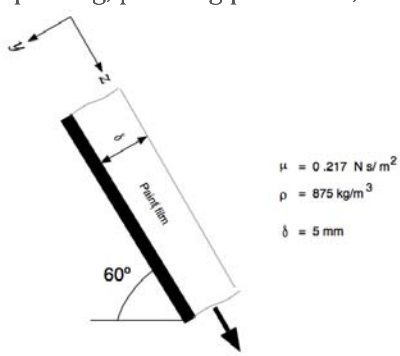


Figure 24

As it is shown in the figure above, *y* and *z* coordinates where chosen, rotating them to align them to the wall. The fluid moves over the *x* axis, while the effort is transmitted over the *y* axis. The same as in the last example, some considerations will have to be made, namely: unidimensional flow (over the *z* axis), stationary state and constant thermodynamic properties.

And so, the Navier-Stokes equation in the *z* axis will be

$$\frac{\partial}{\partial t}\rho V_z = -\left[\frac{\partial}{\partial x}\tau_{xz} + \frac{\partial}{dy}\tau_{yz} + \frac{\partial}{\partial z}\tau_{zz}\right] - \left[\frac{\partial}{\partial x}\rho V_x V_z + \frac{\partial}{\partial y}\rho V_y V_z + \frac{\partial}{dz}\rho V_z V_z\right] - \frac{\partial P}{\partial z} + \rho g_z$$

$$\frac{d}{dy}\tau_{yz} = \rho g_z$$

Solving by variable separation:

$$\tau_{yz} = \rho g_z y + C_1$$

Replacing $\tau_{yz} = -\mu \frac{dV_z}{dy}$:

$$V_z = -\frac{\rho g_z y^2}{2\mu} + C_1 y + C_2$$

And stablishing the equations in the frontier:

CF1 $y=0$, $\Delta_{yz}=0$

This condition in the frontier is stablished between fluids in contact with air, because the shear stress between the air and the fluid is negligible compared with the shear stress between the layers and the fluid. Besides, this condition in the frontier is stablished in symmetric flows as well, over the symmetric axis.

CF2 $y=\Delta$, $V_z=0$, because of the non-sliding condition.

As we assess the conditions at the border, the velocity profile in the oil film is given by:

$$V_z = \frac{\rho g_z}{2\mu}(\delta^2 - y^2)$$

In this case we obtain a parabolic profile, which now depends on the properties of the fluid.

y, mm	V_x, m/s
0	0.428
0.5	0.423
1	0.411
1.5	0.389
2	0.359
2.5	0.321
3	0.274
3.5	0.218
4	0.154
4.5	0.081
5	0.0

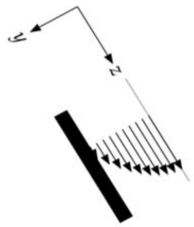


Figure 25

Analysis of the Reduction of the Pipeline in a Turbulent Regime (Numeric Simulation)

In this section we will develop a numeric simulation with a basic exercise in which we make use of the theory explained in previous chapters. It is important to keep in mind that the equations will not be exactly the same as in previous chapters, this is because the software we will be using for the development of this exercise already contains the programs of the discretized equations.

The term discretize refers to converting a mathematical equation into a computer language with the purpose of solving it by computer numerical methods.

For this exercise a simulation tool called ANSYS Fluent was utilized; it is very powerful and extensively used in the industry. The purpose is that the reader obtains basic information to be able to develop an exercise of fluids with ANSYS; another purpose is that the user acquires experience that will possibly be useful in their professional career.

Example to be Simulated

In this example we will see the necessary considerations to run a simulation of the behavior of the flow inside a pipeline, with characteristics that will be explained later, as well as the procedure previous to the simulation. Basically, the problem to be solved will be the simulation of a copper tube, which suffers a reduction in its geometry with a 25^{o} angle. This reduction to the geometry or flow will generate a phenomenon in the flow that will be observed in the simulation with the ANSYS tool.

Modeling

Following some specifications will be presented in order to delimit the simulation and avoid losing the main idea, which is simulating the effect of a flow.

Considerations:

- The development will be made for a steady state.
- A 3D model will be produced.
- The characteristics of the pipeline are: Copper with an absolute rugosity of 0015 mm.
- The angular reduction of the pipeline is 25 degrees.
- The material, whose behavior will be studied, is water.

The geometry will be similar to that in figure 26, which has with the dimensions of the pipeline, as observed.

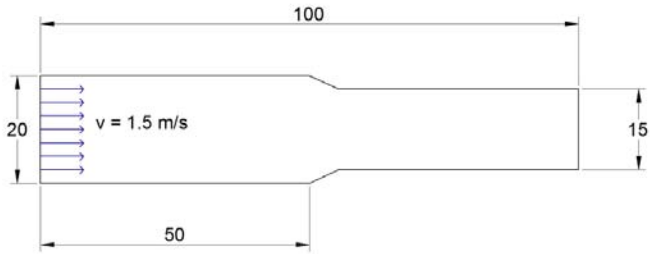


Figure 26

We will start with the development of the 3D object with ANSYS Fluent; for this, we will be basing on the development of figure 26. Using the geometry module with the application DesignModeler, as shown in figure 27.

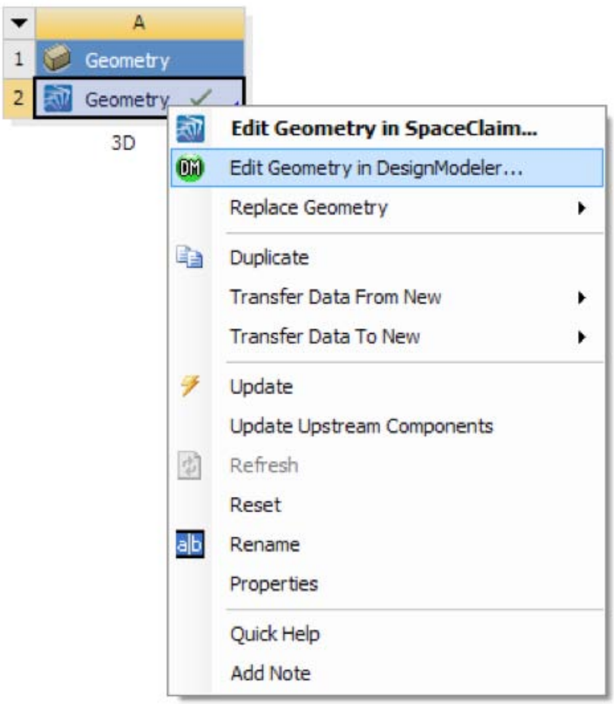


Figure 27

We selected mm as working unit, as shown in figure 28.

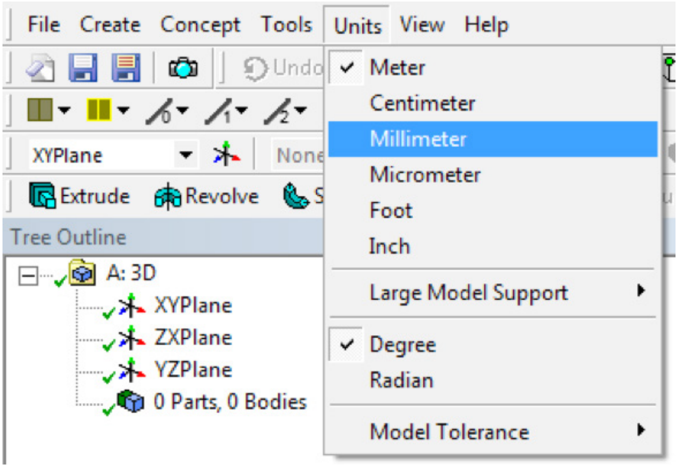


Figure 28

Now we make a new sketch over the XY axis (figure 29).

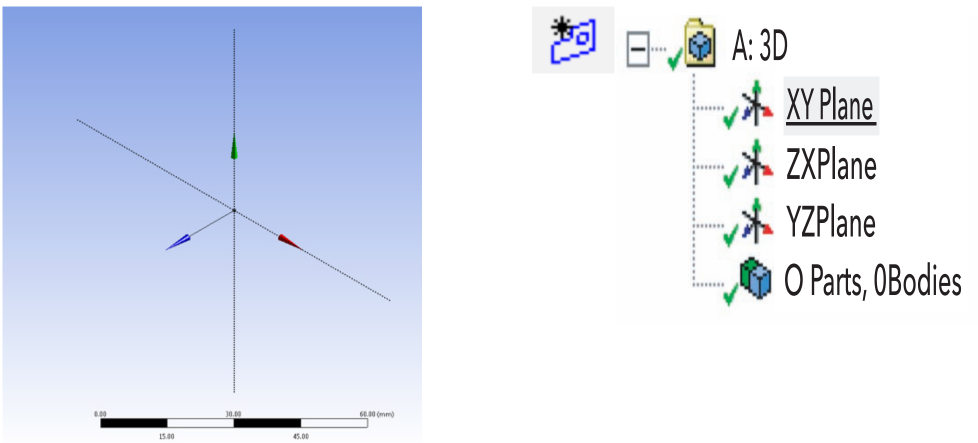


Figure 29

Look at face: Allows us to look at the different faces of the sketch (Figure 30).

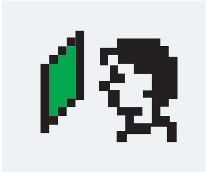


Figure 30

In the sketching window we can select the *Line* in order to draw something similar at the middle of our model without having to worry about measurements (Figure 31).

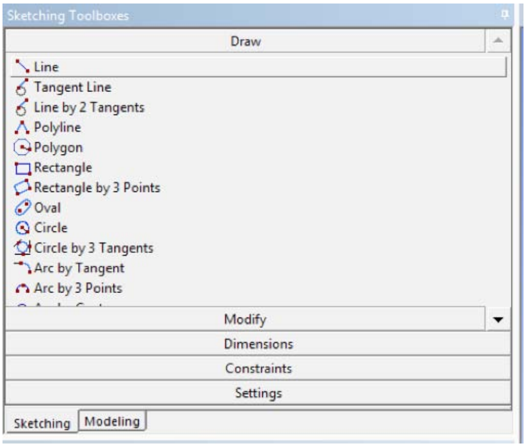


Figure 31

The middle of the model must match with the *x* axis, which will be used next as the turning axis (Figure 32).

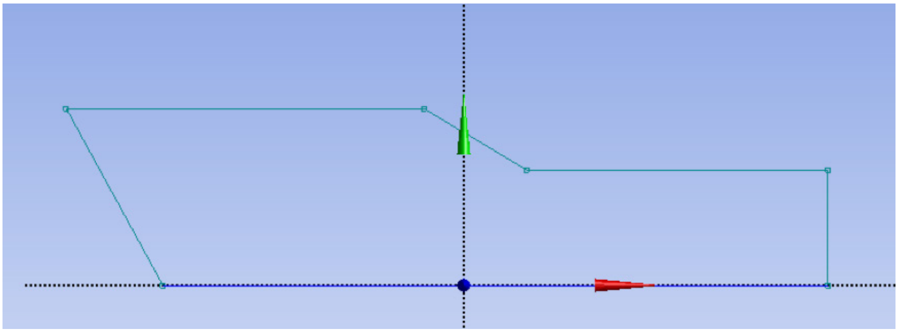


Figure 32

As we draw, some constrains might appear regarding the geometry; the most common are:

V: Vertical constrain
H: Horizontal constrain
C: Line coincidences
P: Point coincidences

We will apply the necessary restrictions, if they were not generated as we drew the lines, obtaining a similar model as the one shown in Figure 33.

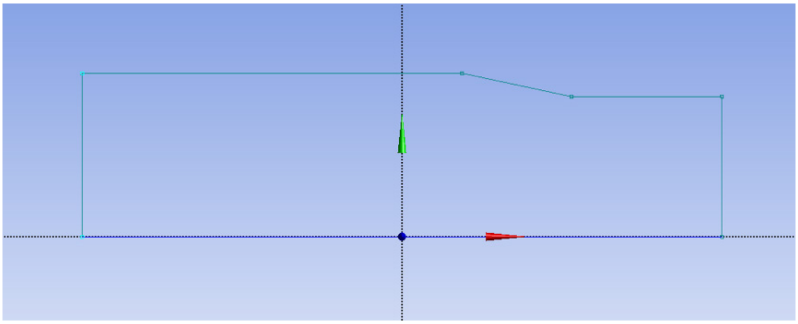
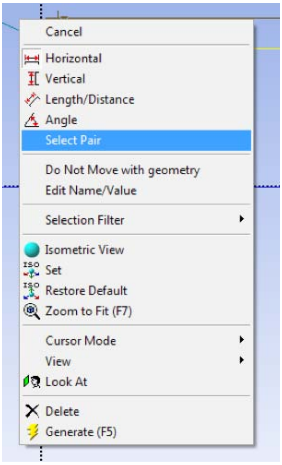


Figure 33

Afterwards we will scale the model, considering symmetry. Making use of the specific scale tools, or selecting the option *general*, and using the right click to see the available options for the chosen element.



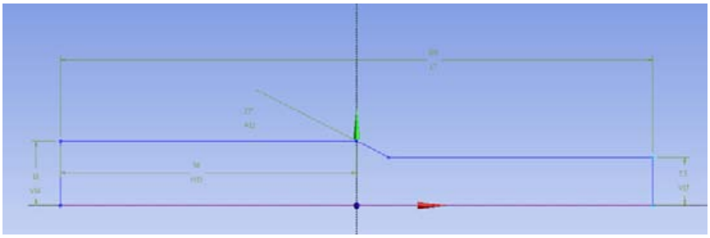
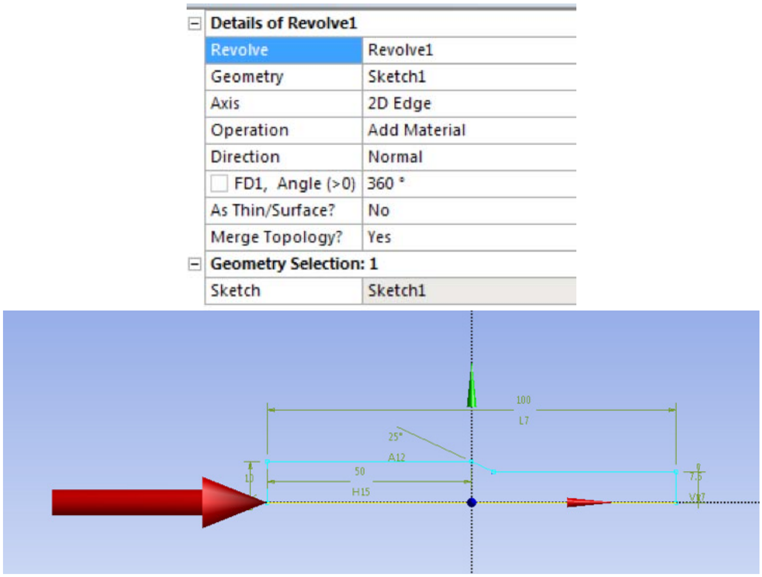


Figure 34

As we have correctly scaled the sketch, we can hit exit and generate a Revolve on the axis (Figure 35).



Details of Revolve1	
Revolve	Revolve1
Geometry	Sketch1
Axis	2D Edge
Operation	Add Material
Direction	Normal
FD1, Angle (>0)	360 °
As Thin/Surface?	No
Merge Topology?	Yes
Geometry Selection: 1	
Sketch	Sketch1

Figure 35

For the Revolve operation, we must select the sketch that we have just generated as geometry, and we will use the *x* axis as the turning axis.

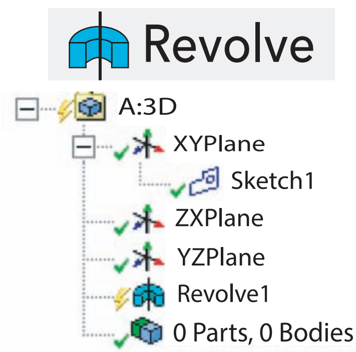
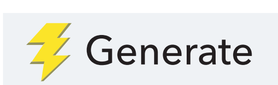


Figure 36

This operation will not be performed until we hit the button *Generate* or F5 (Figure 37).



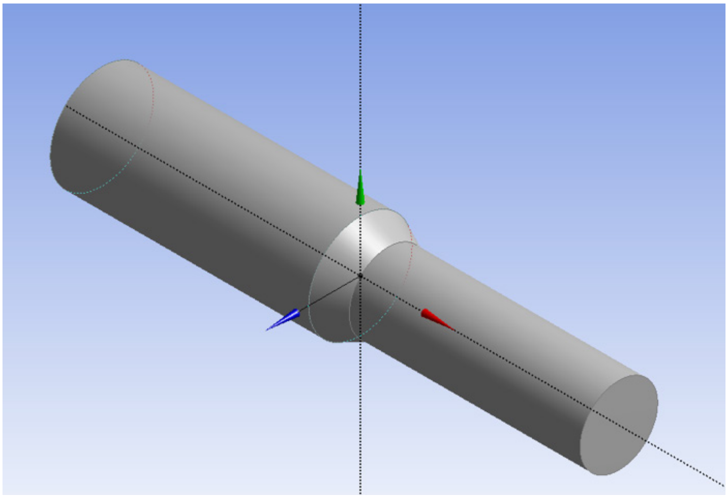


Figure 37

And so, the making of the geometric model is finished.

Model Discretization

Once we have the geometry of the model to simulate, the next step is the generation of a mesh on the geometry. In order to do this, we will add an ICEM FCD module, located in the window Workbench. We can do this by right clicking: Geometry>Transfer to new>ICEMFCD (Figure 38).

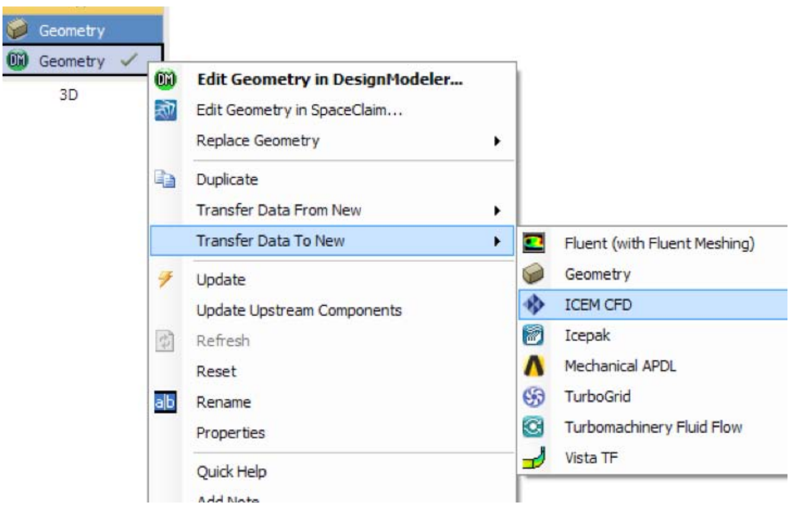


Figure 38

Another way to do this would be dragging the component from the section Component Systems and laying it over Geometry (Figure 39).

Figure 39

We double click on Model to initiate ICEM (Figure 40).

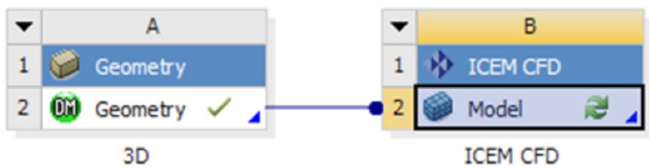


Figure 40

Y+ Model Discretization

Before generating the mesh, we will calculate Y+, which is critical as we use the realizable κ-ε model made, together to wall treatments.

Wall treatments consist in the elaboration of equations that solve the viscose sub-coat by average values, instead of solving it by small elements near to the wall.

The viscose sub-coat plays a very important role in turbulent fluids. In this zone near to the wall some effects such as rugosity or friction appear, as well as a high rise of the velocity. This might be very difficult to solve for elements near to the wall where we search for a y+=1, in the sub-viscous coat.

As we use wall treatments, we obtain average values for this coat. Depending on the type, we will need to position in certain range. As we use realizable κ-ε, we have to position in the overlap coat.

Y+ indicates us in which part of the fluid we are. Parting from the turbulence model and the treatment type we will have to position within certain range.

The wall treatment requires a range from 30 to 300 y+, locating them in the overlap coat, which produces a smaller number of elements (Figures 41 -A and 41 -B).

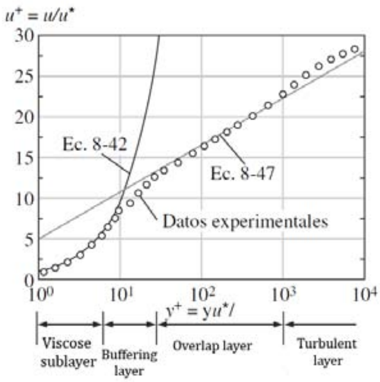


Figure 41 - A

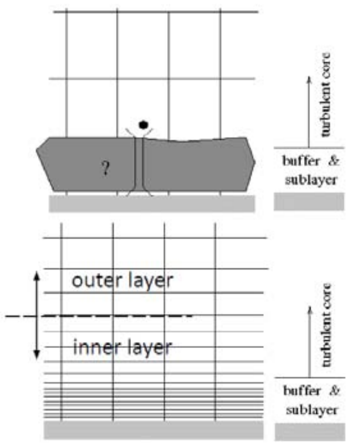


Figure 41 - B

In order to calculate Y+, we must be aware of the model's operation conditions and some of its characteristics:

D The diameter of the pipeline or the characteristic length.
ε Absolute rugosity of the pipeline.
V Average velocity in the pipeline.
ρ Flow density.
μ Dynamic viscosity of the fluid.

Y+ is known as the dimensionless distance, because through it the position of a point respect to the wall *y* is located. And so, *y=0* indicates there is contact with the pipeline.

Y+ depends on the characteristics of the fluid, such as dynamic viscosity, and density, as well as the operation conditions of the system, which is described by the dimensionless velocity *u*.*

$$u* = \sqrt{\frac{\tau_w}{\rho}}$$

The dimensionless velocity *u** depends on the density of the fluid and the shear stress in the wall of the fluid:

$$\tau_w = \frac{1}{2} \cdot \rho \cdot V^2 * Cf$$

The shear stress depends on the velocity of the fluid and the shear stress in the wall of the fluid:

$$\tau_w = \frac{1}{2} \cdot \rho \cdot V^2 * Cf$$

The shear stress depends on the velocity of the fluid *V*, the density and the friction coefficient *Cf*, which should not be confused with Darcy's friction factor.

Next, we will present a way to calculate it. In case of wishing not to do the calculations, there are some webpages that with the conditions and characteristics shown above will do the necessary operations and will give the value of *Y*:

http://www.pointwise.com/yplus/
https://www.cfd-online.com/Tools/yplus.php

Next, we will present the Reynolds equation that describes the inertial and viscous forces that are present in a fluid.

$$\text{Re} = \frac{V \cdot D}{\nu} = \frac{V \cdot D \cdot \rho}{\mu}$$

Based on the previous equation we will indicate with the calculation of Reynolds number the type of fluid we are working with, whether it is turbulent (Re>4000), in transition (4000>Re>2300), or laminar (Re<2300). It is very important to say that in a laminar flow turbulence models do not apply, besides the mesh form will not play a relevant role:

$$\frac{1}{\sqrt{\lambda}} = -2 \cdot \log_{10}\left(\frac{R_r}{3.7} + \frac{2.51}{\mathrm{Re} \cdot \sqrt{\lambda}}\right) \qquad\qquad Cf = \frac{\lambda}{4}$$

Then, using Colebrook equation, Darcy's friction coefficient, , the friction factor *Cf* will be calculated. The friction factor *Cf* corresponds to a fourth of Darcy's coefficient.

A Moody diagram can be used to obtain the same results.

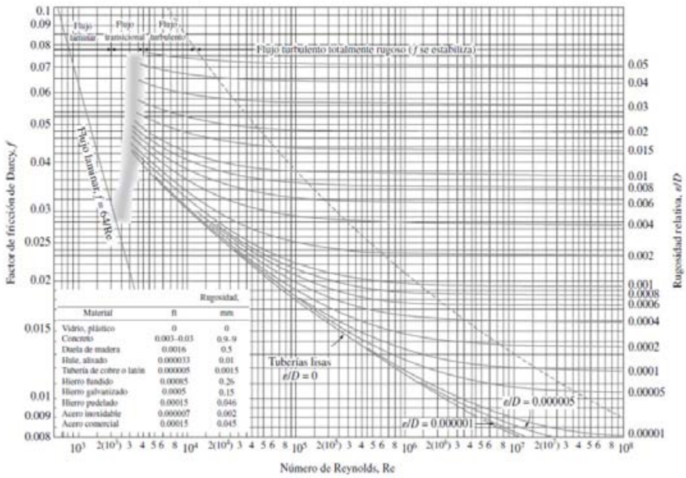


Figure 42

Source: Çengel, Y. A., Cimbala, J. M., & Sknarina, S. F. (2006).

Mecánica de fluidos: fundamentos y aplicaciones (Vol. 1). McGraw-Hill.

For our model we have two pipelines, we will analyze both of them to verify the most adequate value for Y+. Our model comprises water:

$$\hat{V} = V * A$$

D1 = 20mm	D2 = 15mm	Pipeline diameter.
ε = 0.0015mm		Absolute rugosity of copper.
V1 = 1.5 m/s	V2 = 2.667 m/s	Average velocity in the pipeline.
= 998.6 Kg/m^3		Water density.
μ = 0.000103	Kg/m*s	Dynamic viscosity of water.

We will try to obtain a y+ equal to 40, which is still within the recommended range. A y+ equal to 30 might not be accomplished, because of variations due to geometry, pressure, velocities, et cetera.

Two different calculations will be made, one for each type of pipeline (20 and 15mm), where the diameter and velocity values will change in each case.
Due to the pipeline reduction, we will obtain a more turbulent fluid in the reduced section, this can be observed in Reynolds numbers.

Re1 = 29085 Re2 = 51714

The larger the Reynolds number, the finer the mesh will be.

y1 = 1.37e-3 y2 = 7.74e-5

We will use the largest value, in this case y1. We will use the largest value, because if we use the smallest one, this will not be valid for less turbulent sections.

Once the distance of the first cell *y* is defined, we will proceed to create the mesh in ICEM-CFD.

Naming the entities where we will apply the conditions at the border. For this we click right on Parts and select Create Part (Figure 43).

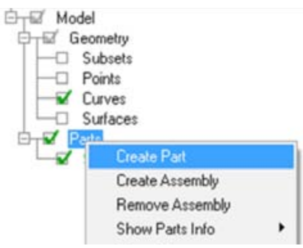


Figure 43

In order to see the surfaces, we must select the Surfaces box, inside the Fluent tree in the geometry section (figure 44).

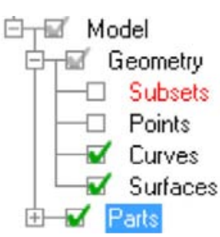


Figure 44

We select the lateral surfaces naming them: WALL (Figure 45).

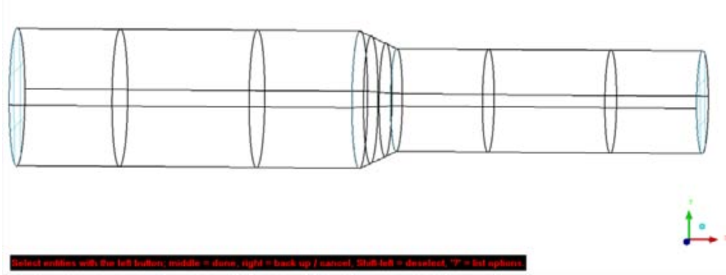


Figure 45

As we make the operations on ICEM, we can middle click to apply the option in process, or by clicking on the Apply button (Figure 46).

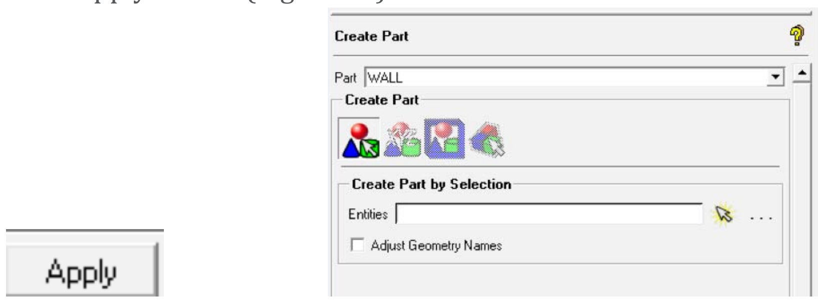


Figure 46

Then we create the entity INLET with the surface shown in figure 47.

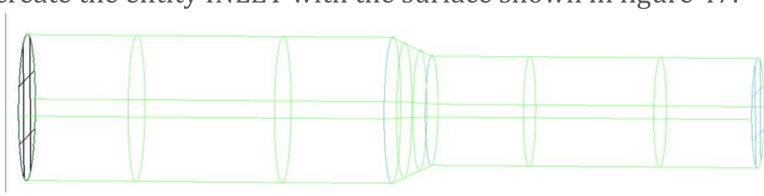


Figure 47

And we finalize with OUTLET (Figure 48).

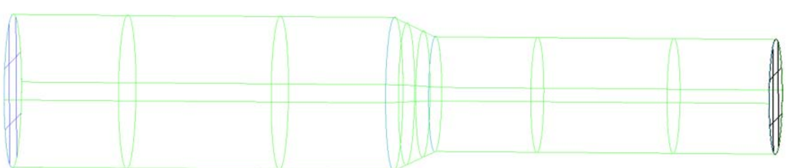


Figure 48

Our tree will be very similar to the one in figure 49.

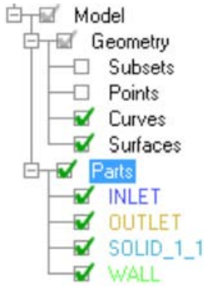


Figure 49

If we wish to change the color of a specific part, we right click on Change Color (Figure 50).

Figure 50

Initiating with the creation of the block of our model (Figure 51).

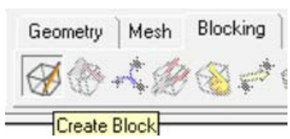


Figure 51

Defining the name of the domain as FLUID, by a 3D block projecting the vertexes (Figure 52).

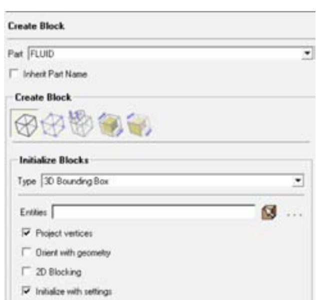


Figure 52

Selecting everything visible on the screen (Figure 53).

Figure 53

The initial block must be very similar to the one in figure 54.

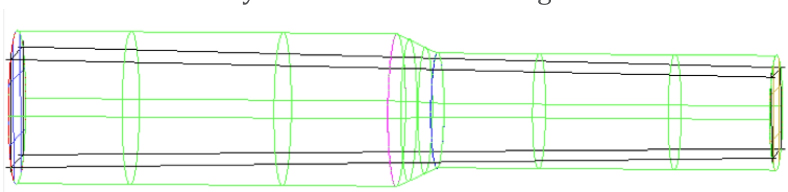


Figure 54

We will show the points of our geometry in order to make cuts through them as seen in figure 55.

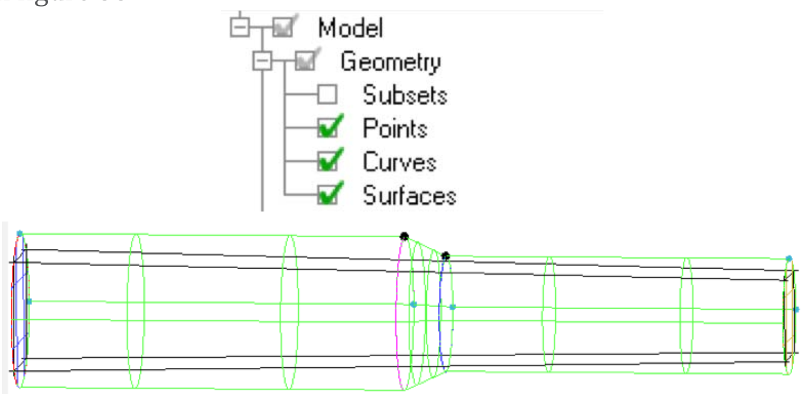


Figure 55

We will make the next two cuts through a point in our block (Figure 56).

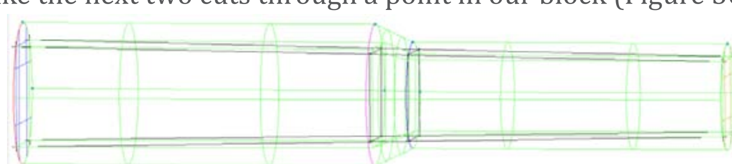


Figure 56

The cuts will be made through a point, selecting first a line that is perpendicular to the cut to be made and then fixing the cutting point (Figure 57).

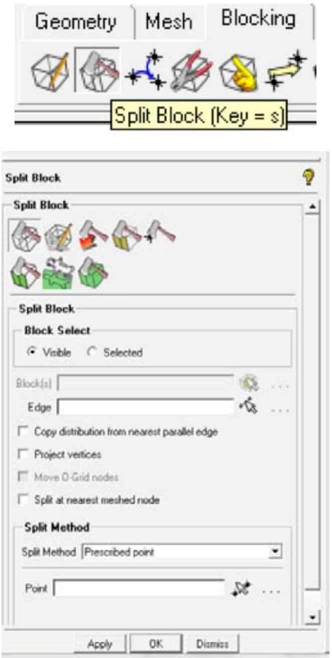


Figure 57

We associate the borders to its closest curve, activating the projecting vertexes option (Figure 58).

Figure 58

In order to associate, we do the following selections Blocking Associations > Associate Edge -> Curve, activating the projecting vertexes option (Figure 59).

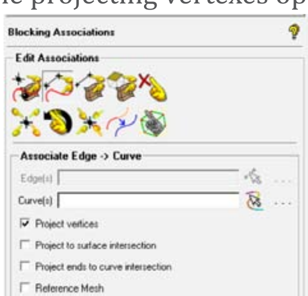


Figure 59

As we do the associations projecting vertexes, they move the closest to the curve they have been associated with, besides blocking movements outside of it (Figure 60).

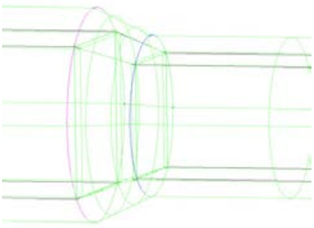


Figure 60

Next, we will create an O-Grid in our block. It will adapt the block to a curved surface, by a series of cuts in the block, as it forms in the shape of an O, hence the name.

This formation prevents obtaining deformed elements in the geometry, as seen in figure 61.

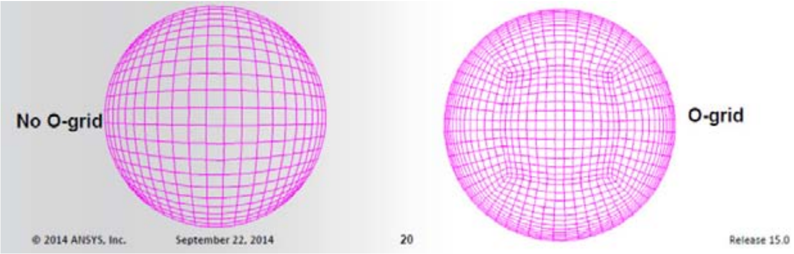


Figure 61

In order to be able to apply an O-Grid we go to the section Split Block (Figure 62).

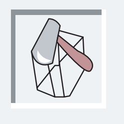


Figure 62

Blocks, faces, borders and vertexes can be selected in 3D mode (Figure 63).

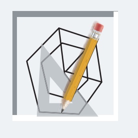


Figure 63

We select all the blocks of our model, as well as the faces corresponding to INLET and OUTLET (Figure 64).

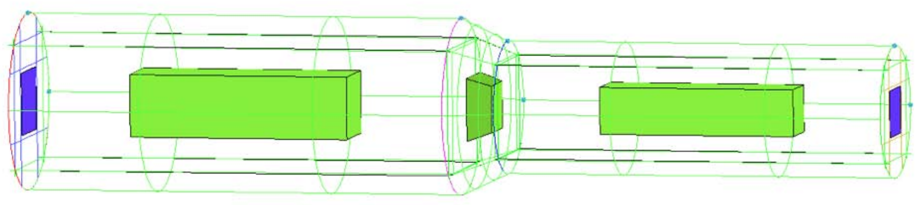


Figure 64

Before clicking on Apply, we activate the Absolute option, which will convert the units into meters from the gap of the O-Grid and not from the proportions of the block.

In this step we will apply the value of *y* obtained through a *y+* equal to 40: The value will be 1.4 e-3; afterwards we click on Apply so we can see the applied changes (Figure 65).

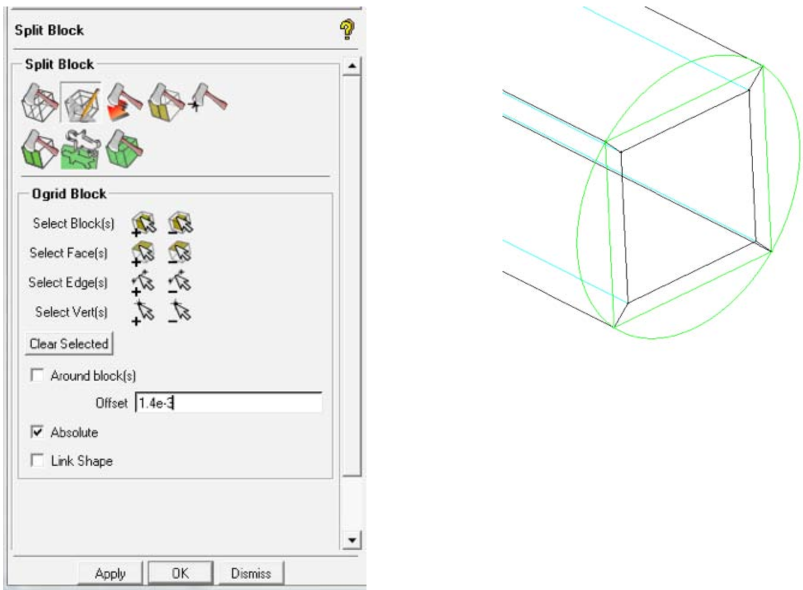
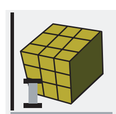


Figure 65

Following, we will define the parameters or divisions in our model (Figure 66).



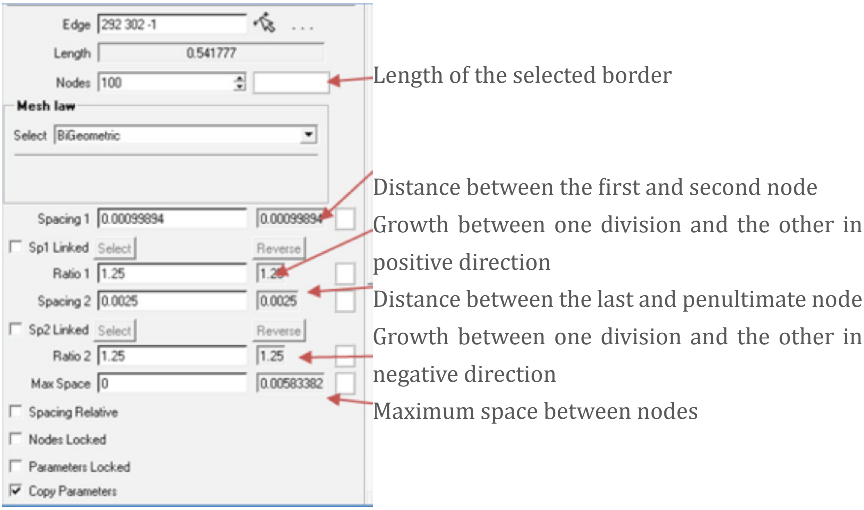


Figure 66

As we select a border, an arrow appears pointing out at the indicated direction of the border. As we apply defined growths and spacing, the first one corresponds to the beginning of the border and the second to the end (Figure 67).

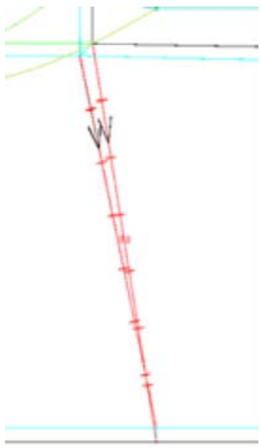


Figure 67

Edge Discretization

We must be sure to activate the option that allows us to copy the parameters to the parallel borders; we will begin with the lengthwise dimensions on the pipeline (Figure 68).

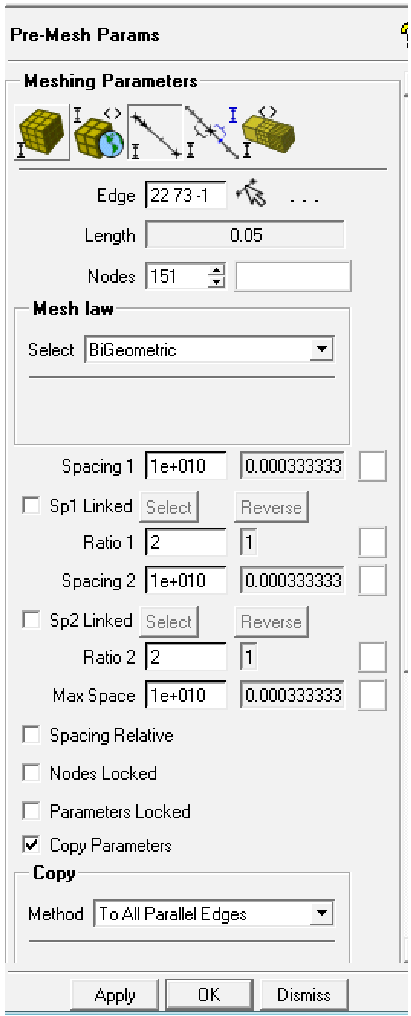


Figure 68

Applying 50 nodes (generating 49 divisions), through the 20 mm and 15 mm constant sections. As we apply the Mesh law Bigeometric divisions with the same longitude throughout the border are applied (Figure 69).

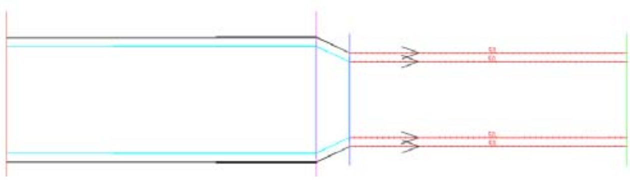


Figure 69

In the reduction, we will apply 15 divisions (figure 70).

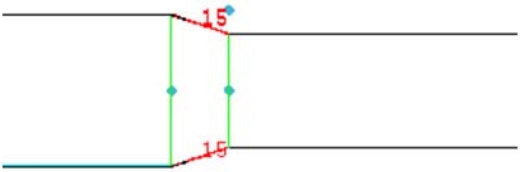


Figure 70

In the sections that form the circumference an initial size of 1.5e-3 will be applied in both sides; this will help use fewer nodes and prevent rough changes between the first layer (based on y+=40) and the other layers that do not need values as small as the first layer does (Figure 71).

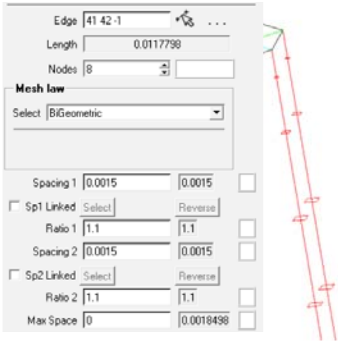


Figure 71

To verify the quality of the mesh, click on the icon shown in figure 72.

Figure 72

Fluente will ask if we wish to update the mesh; it will not be possible to see the quality if we do not click yes (Figure 73).

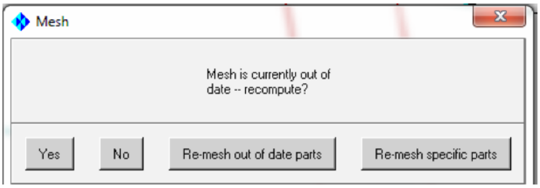


Figure 73

Afterwards, we will apply the 3x3x3 Determinant criteria (Figure 74).

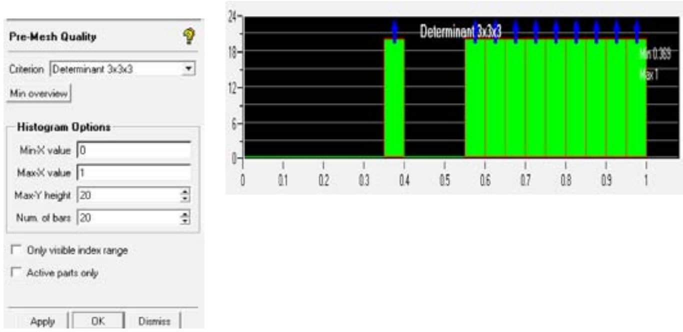


Figure 74

With the blocking and parameters shown, the quality must be similar. In order to generate the mesh we right click on Pre-mesh in Fluent tree; afterwards we will go to Convert to Unstruct Mesh (Figure 75).

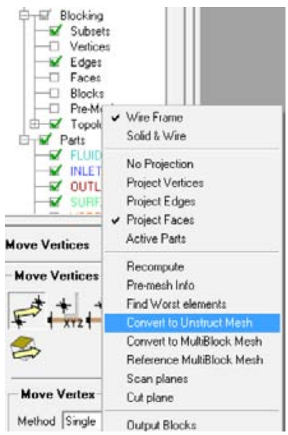


Figure 75

If we have already generated a mesh, a window will appear, and so, we must click on Replace (Figure 76).

Figure 76

Model Configuration

In this section, we will proceed to place the physics into the system inside of the geometry of the system.

We will add a Fluent component in our Workbench session, dragging the Fluent component and dropping it over the mesh.

Another option is to locate over the mesh component and right click on transfer Data To New>Fluent (Figure 77).

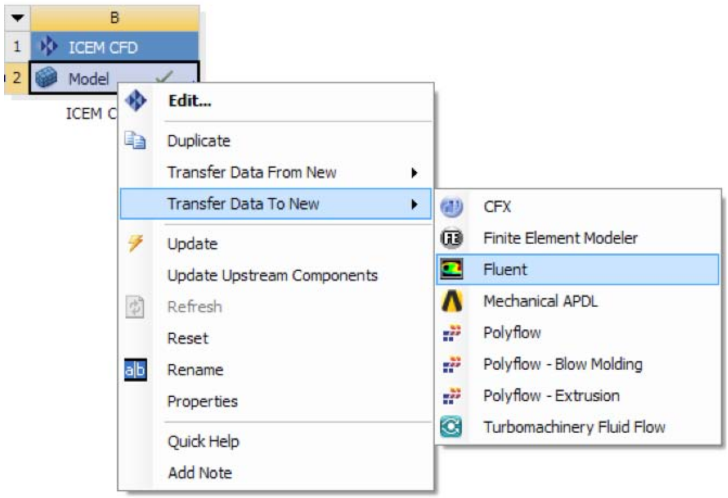


Figure 77

As we add it, we must right click on the ICEM component and then hit update, so it turns the mesh into Fluent format (Figure 78).

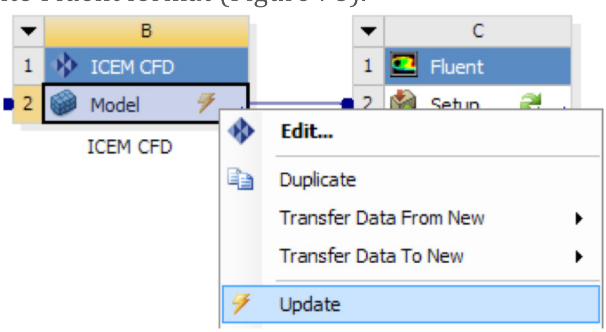


Figura 78

We will activate double precision in the Fluent configuration and then hit Ok (Figure 79).

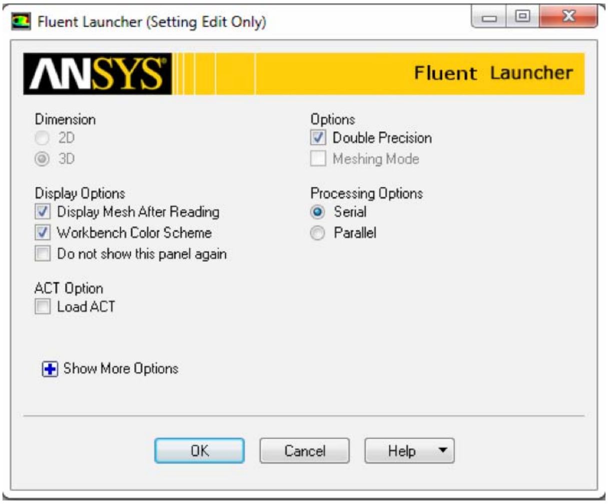


Figure 79

As the mesh is loaded in Fluent, we must proceed to verify if there are any errors as well as the quality. These options can be found in the "General" tab of the Fluent tree (Figure 80).

Figure 80

We will click on "Check" and then on "Report Quality" to verify that there are no errors (Figure 81).

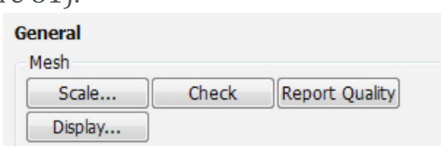


Figure 81

In Model section, we will select the turbulence κ-ε model, in Viscous section. We will not modify any of the options of the κ-ε model (Figure 82).

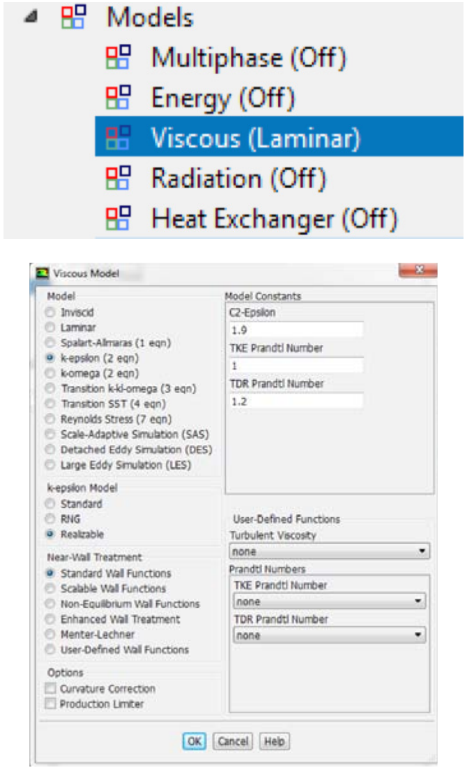


Figure 82

Afterwards, we will add the work fluid from the Fluent library; we will double click on the materials section and then click on air.

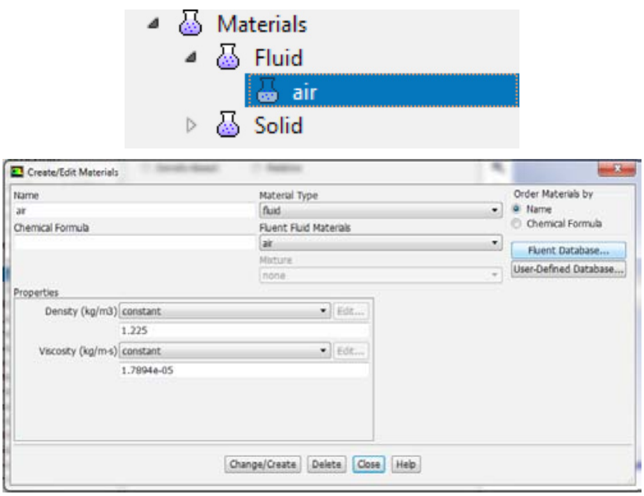


Figure 83

In the materials tab we will go straight to the Fluent data base, and we will search for water-liquid (h2o<l>), then we will copy this material (Figure 84).

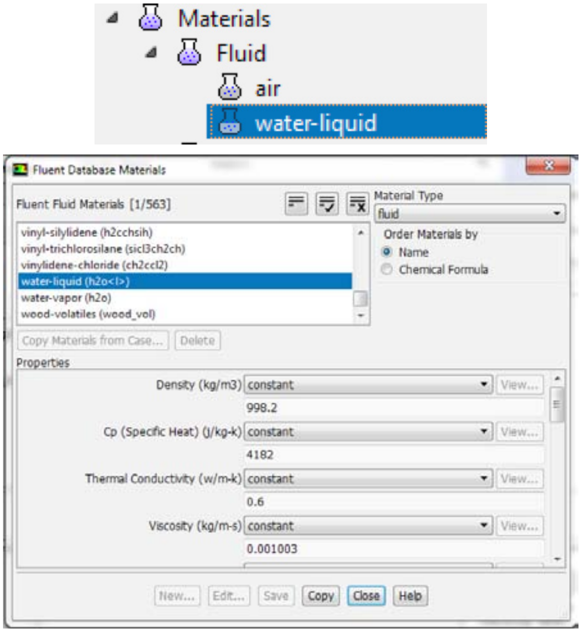


Figure 84

Just click once to add the material to the list of materials in use.

If any material was added by accident, it is possible to delete it by right clicking on the material and then selecting Delete. (Figure 85).

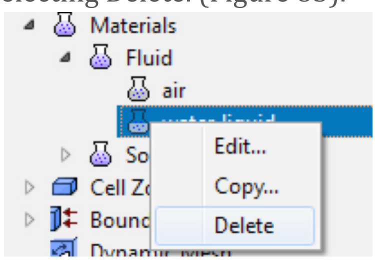


Figure 85

Conditions at the Border

In this section the conditions at the border will be established in the system, referring to the entrances and exits of flow inside the geometry.

We will change the fluid material, because Fluent uses air as a pre-established option.

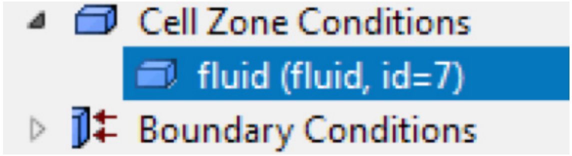


Figure 86

We must verify that the conditions are set as it is shown next:
Inlet = velocity-inlet
Outlet = Pressure-outlet
Wall_1 Wall_2 = wall

The conditions at the border appear as shown in Figure 87.

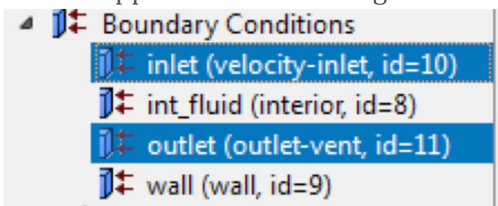


Figure 87

In case they do not concur, it is necessary to right click on the entity. In our case, the outlet is defined as an outlet-vent. We will select pressure-outlet as condition type (Figure 88).

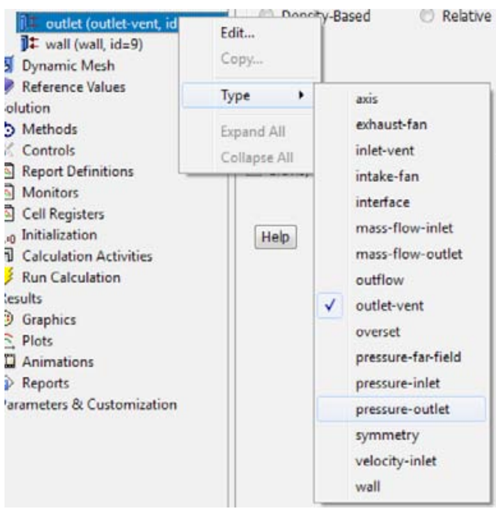


Figure 88

As we change the type of condition a window will appear, where we can indicate the operation conditions of the entity. We will keep the pressure at 0 Pa, but we will change the turbulence method to intensity and the hydraulic diameter to 0.015m (Figure 89).

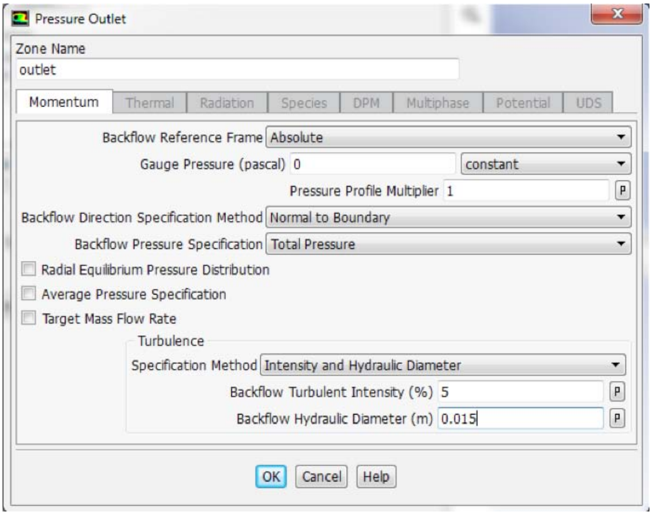


Figure 89

At the input we will apply a 1.5 m/s velocity, as well we will apply a turbulence method through intensity and a hydraulic diameter of 0.020m (Figure 90).

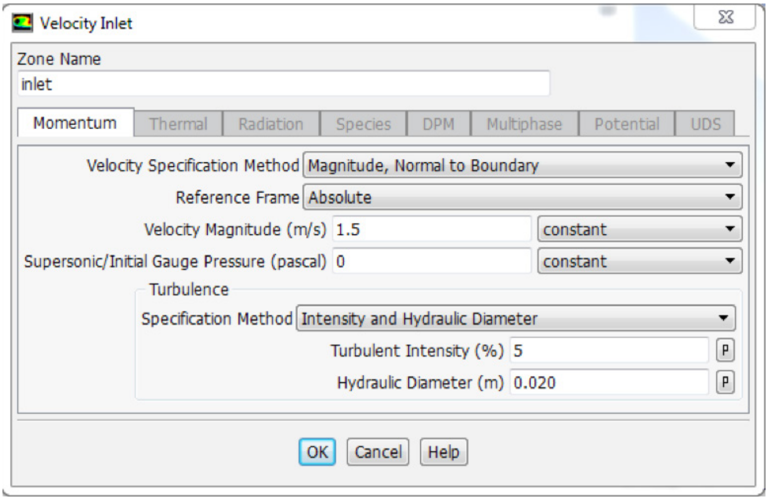


Figure 90

Now applying rugosity on the wall of the pipeline (Figure 91).

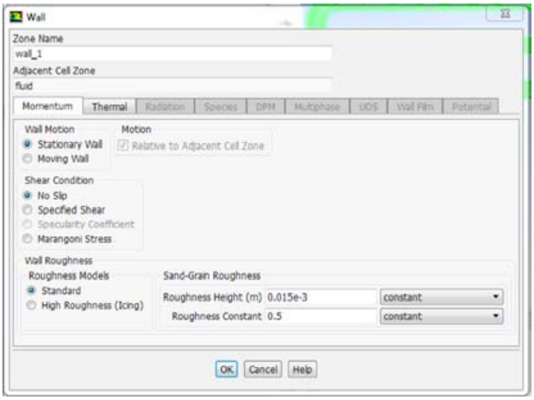


Figure 91

Model Solution

This section will configure the possible results we may obtain thanks to the physics placed inside the system. We will begin with the configuration to solve the model.

We will choose a Coupled method checking the box Pseudo Transient (figure 92).

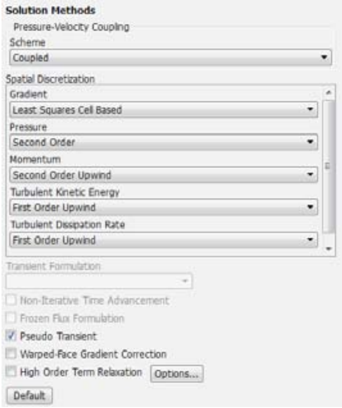
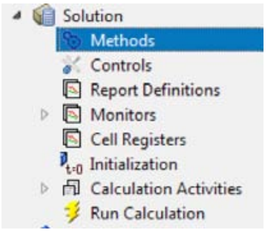


Figure 92

We will start the simulation in a hybrid form, we will only hit click in Initialize; then, we will calculate with 100 iterations. The solution should converge in fewer than 50, as we observe in figure 93.



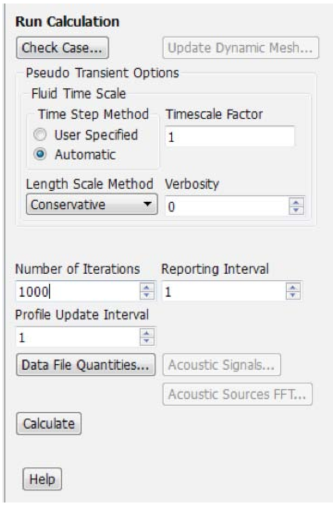


Figure 93

Processing

In this section we will analyze the results of the programmed calculations for this problem and we will establish the ways in which we want the results to be shown.

Our objective is to obtain results in a graphic form through contours. Before starting the pressure and velocity contours we will add two windows in Fluent (figure 94).

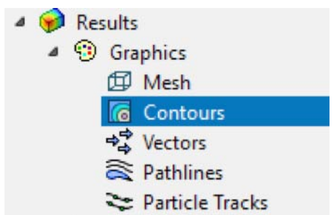


Figure 94

In order to add a new window, we will place ourselves on the current window of Fluent (right click) and we will annex a new window (figure 95).

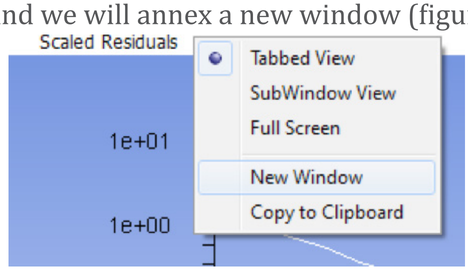


Figure 95

We add a pressure contour on the first window with the displayed options. Once it is configured, click the Save/Display option and then click on Close (figure 96).

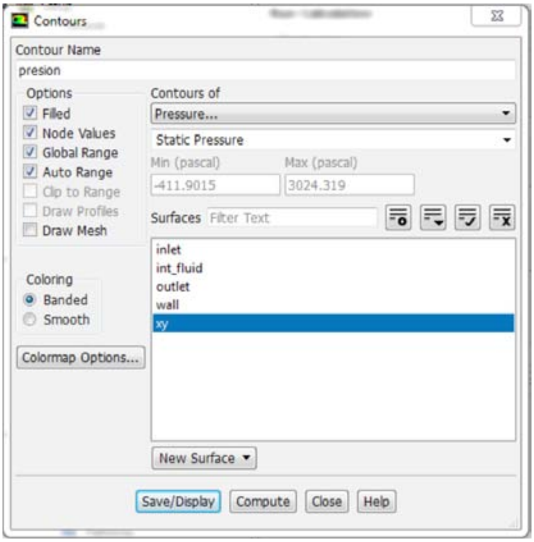


Figure 96

We will add a new plane to show the inside of our new model for the plane XY, as shown in figure 97.

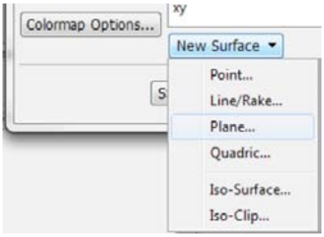


Figure 97

We will apply the values shown in figure 98, click the Create option and then click on the Close option.

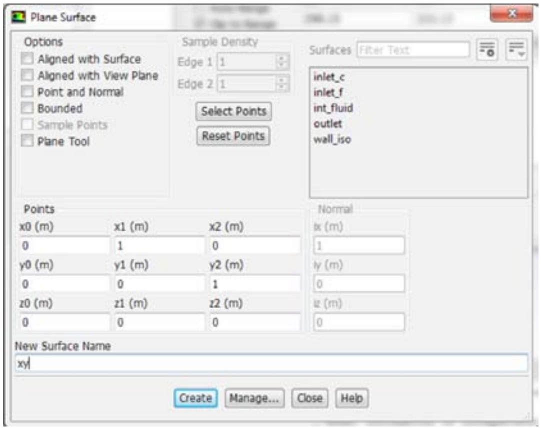


Figure 98

We observe the pressure contours in our model from the following image.

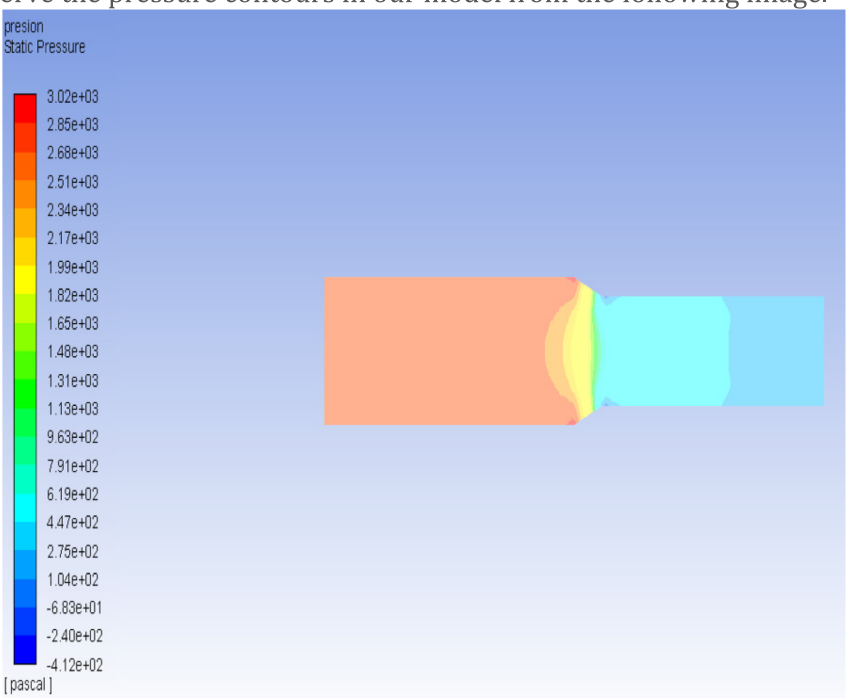


Figure 99

We will implement the same process, but now with velocity added (figure 100).

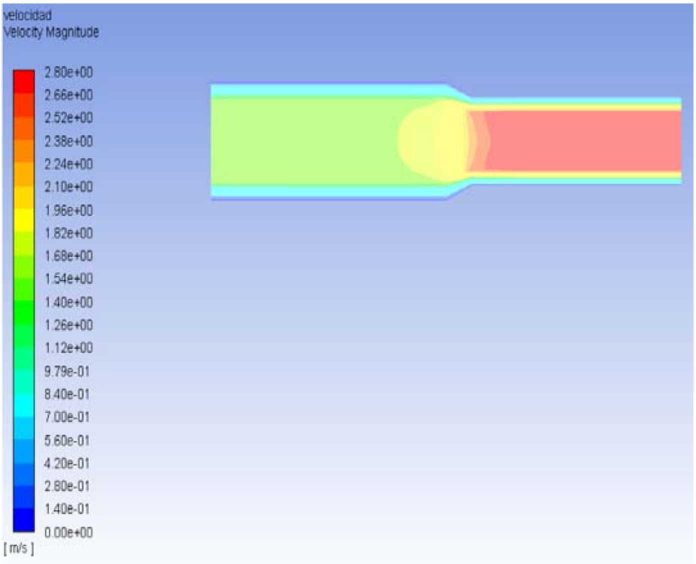


Figure 100

Finally, we will show the velocity vectors on the plane XY. In order to do this, we must add a new window, we will select the shown options, click on the Save/Display option and then we close the window in figure 101.

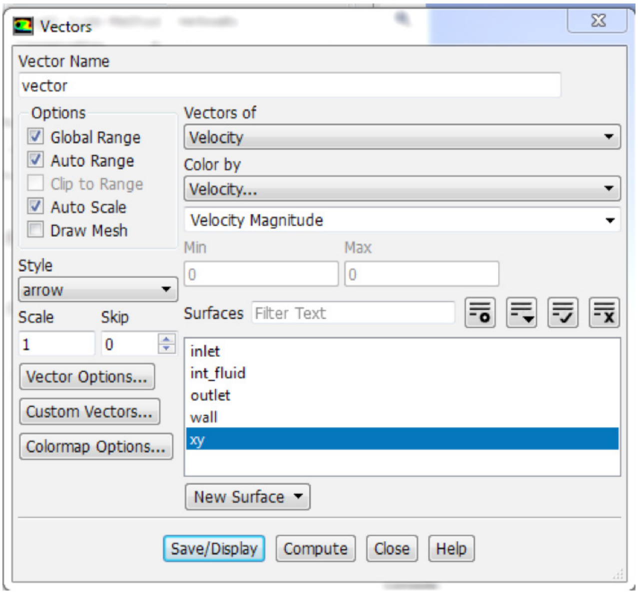


Figure 101

In the last figure we show the result of the velocity vector (figure 102).

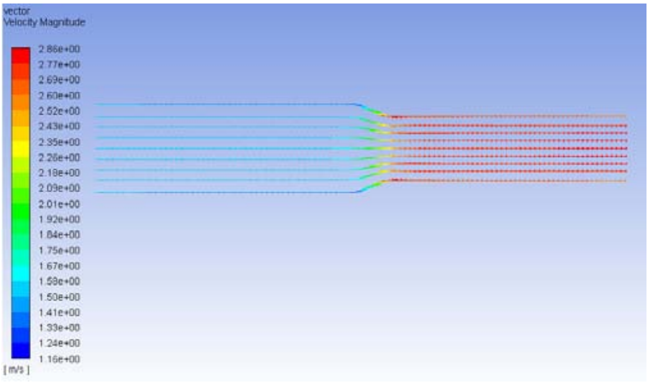


Figure 102

Summary

In this exercise, different points were analyzed to carry on a simulation in Fluent:

Meshing of the model in ICEM-CFD
Y+: considerations and calculations
Tools and basic processes
Configuration of the model κ-ε Feasible Creation of planes
Result contours
Result vectors

Conclusions

Fluid mechanics is a very extensive field of study where we have the opportunity to explain different phenomena related to fluids, behaviors that could be generating inside the different systems that may be mechatronic, electronic, mechanic and above all multiphysics.

For example, in the mechatronic area we could develop a system applied in the medical field to rehabilitate certain parts of the body (hands, feet, head, etc.) and at the same time to have these devices to carry a hydraulic system used to rehabilitate a determined part of the body; in order to achieve this, it would be necessary to develop both the electronic control and the hydraulic system of such device.

In hydraulic design, it is fundamental to carry on the study of the behavior of the working fluid because the hydraulic system applied has to adapt to the desired mechatronic system. For this reason, previous studies on visualization are carried out and it is even possible to carry out a numeric simulation of the behavior of the hydraulic system in the various devices that interact directly with the hydraulic system for a proper performance.

It is very common to have non-desired flow behaviors in the different developments in which hydraulic systems are involved; therefore, without a doubt, it is essential to move forward to the correction of such non-desired behaviors. The indicated phenomena can be calculated and be numerically analyzed as it has been shown in this book. Likewise, numerical simulation can be performed from a specialized software such as Ansys, Comsol or Code Saturne, in order to simulate the different behaviors that could provide an answer and solution to potential errors in the reality.

So far, we have provided the basic tools to analyze different flow behaviors as well as the option to assess the result of such behaviors. In the exercises it is possible to observe that the phenomena in fluid mechanics can be very complex but in its right dimension, they turn out to be simple, as long as the basic principles of the fluid mechanic are well grounded.

Finally, it is important to stress that the applications of fluid mechanics encompass the industries and their different activities. Here, we have limited ourselves to a couple of exercises. It is very important to mention this as the different products of industrial use and consumption are related in one way or another to fluid mechanics. To name some brands: Mabe, Samsung, Whirpool, among others. We hope to be able to include other alternatives in a subsequent volume.

This way, it is also important to mention that this book's main idea was to offer students analytic examples and developments, as well as the numeric simulation of real cases for them to have the necessary knowledge to develop any study involving fluid mechanics.

No doubt, one of the book's main inputs for students is numeric simulation. There are different applications for this academically as well as industrially and in research centers, where it is used even in structural mechanics and electromagnetic fields, to mention some. Therefore, it is important for students to start working with numeric simulation so that at the end of their studies they master the basic knowledge of numeric simulation with fluid mechanics so that they have job opportunities.

Brief Bibliographic Note

The idea of writing this book was to offer the readers a tool, to help those interested, to be able to work in the analysis of fluid mechanics, in a complete way, though at once summarized. Based on their personal experience, the authors provide the necessary information to start working in the analysis of the mechanics of fluids. This book includes the fundamental equations of the field in a summarized way and these are explained to the reader in an easy way to understand.

The equations were a compilation from the most common books on mechanic of fluids, namely: Y. A. Cengel, J. M. Cimbala y S. F. Skanarina, *Mecánica de fluidos: fundamentos y aplicaciones*, (Vol. 1), McGraw Hill (2016). This book describes the fundamental equations of mechanics of fluids with an extensive vision, and for this reason we gathered the most important points in the fundamental equations here presented from this book. We have the same situation with the book by A. Crespo, *Mecánica de fluidos,* Ed. Thomson (2018), where the equations are outlined in a less demonstrative way, but we consider that the terminology is more explicit than in the first book. Finally, the book by A. Barrero Ripoll and others, *Fundamentos y aplicaciones de la mecánica de fluidos*, Ed. McGraw Hill (2016), we consider it to be one of the most complete as regards the useful descriptions to produce this book.

Regarding numeric simulation, there is vast bibliography with analysis of the method of finite volume that is applied by ANSYS software to develop the exercise exposed in this book, but the aim of numeric simulation is not only to know the study and development of the method of finite volume, but also to show different ways to simulate a phenomenon of fluid, which is the main topic of this book.

For the use of ANSYS, the above mentioned does not occur. On the contrary, consultation references are almost nonexistent for the knowledge and operation of ANSYS software. The reason for this is that as it is specialized software, distributors of the simulation tool do not share any bibliography regarding its use. This shows the importance of this material that is created from the experience of those who wrote *Fluid Mechanics with CFD exercises*.

Distributors of the tool for the knowledge of the software ANSYS offer specialized courses in various fields (mechanics, fluids, electromagnetism) for which ANSYS works. In the field of fluids that belongs to ANSYS Fluent, for example, to be able to perform what is shown in this book, it would be necessary to take the basic course on ANSYS Fluent. This course has a duration of 24 hours and is taught over three days and, as a pre-requirement, experience in the use of ANSYS Mechanical is demanded. The course comprises twelve topics, such as: Introduction to CFD methodology, cellular zones, border conditions, post-process, resolution configurations, turbulence modeling, heat transference modeling, mobile zones, transitory simulations, and user defined functions (UDF).

As it may be observed, there are numerous requirements for the use of ANSYS Fluent. Therefore, we trust that the numeric simulation developed in this book with ANSYS Fluent, has been helpful and has turned out to be practical and simple for users who begin to deepen in the use of ANSYS Fluent, fundamental in modern industry.

Glossary

ANSYS. Numeric simulation tool, based on finite element and volume.
ANSYS Fluent. Numeric simulation software, specialized for modeling of fluids, turbulences, heath transferences and reactions for industrial applications.
Tree of the model. Display of the functions generated in the model in ANSYS.
CAD. Use of computing systems to help in the creation, modification, analysis or the optimization of a design.
CAE. In the use of the software, it simulates performance with the objective of improving product designs or contributing to the solution of engineering problems for very diverse sectors.
Mesh quality. It is the fineness with which the mesh has been discretized in geometry.
CAM. Software to generate a CNC program. It can be used to program any machine controlled by CNC.
Code Saturne. Free license software of computational numeric simulation, based on the finite volume method.
Comsol. Private license software of numeric simulation, based on the finite element method.
Border conditions. They are the conditions presented in the model or differential equations that explain certain physics problems.
Convergence in the simulation. It means that the result is about to have a possible numeric solution.
Moody diagram. It is the graphic representation in double logarithmic scale of the friction factor in function of the number of Reynolds and the relative roughness of a pipeline.
Mathematical discretization. In the computing field, it refers to a mathematical equation, which means interpreting it in the form of algorithms to give it a solution through computational methods.
Discretization of the model. Discretization of the equations to be implemented in the system.
Discretization of the model Y+. It consists in implementing equations to solve the viscose sub-cape through average values.
Divergence in the simulation. It means that the result has no solution, so that it is considered there is an error in the simulation.
Energy equation. It can be considered as a proper declaration of the principle of energy conservation for the flow of fluids.
Continuity equation. Equation used to analyze mass change in regard of time.
Navier-Stokes equation. Equation that allows us to analyze in a microscopic way the behavior of a fluid in order to accurately ascertain its characteristics and they

are needed when the effects of friction or the dependence of the properties on fluid velocity are important.

Geometry. Graphic representation of the model to be simulated, normally generated by a CAD program.

Dimensional homogeneity. In every physics equation, each term should have the same dimensions.

Inlet. Term used in ANSYS to identify an entry of the system.

Iterations. Repetition of a process with the intention to accomplish a desired goal, objective or result.

Velocity magnitude. It links the change of position (or displacement) to time.

Meshing. Discretization of a geometry.

Mechanic of fluids. It studies the laws of fluids' movement and its processes of interaction with solid bodies.

Method of finite element. It is a discretization of a geometry in a finite quantity of elements for which differential equations, which describe the displacement of the nodes or points of intersection of elements, are simultaneously solved.

Method of finite volume. A method that allows discretizing and numerically solving differential equations. The discretization of the meshing represents a point that is no more than a volume inside the system.

Model. Physical, mathematical or any other logical form of representation of a system, entity, phenomenon or process.

Node. Point of intersection.

Scientific Notation. It is a specific way of writing very large or very small numbers.

Engineering Notation. It is as Scientific Notation, excepting the fact that it only uses powers of 10 that are multiples of 3.

Reynolds number. It is a dimensionless number used in mechanic of fluids, reactor design and transportation phenomena to characterize the movement of a fluid.

Outlet. Term used in ANSYS to identify an exit of the system.

Pre-mesh. Tool used to visualize the possible geometric discretization, before the final discretization of the geometry.

Processing. Calculations made by a computer in order to generate possible results.

Data processing. Accumulation and manipulation of certain elements of the data, to produce significant information.

Post processing. The interpretation and visualization of the results.

Turbulent regime. It is the chaotic movement of a fluid, in which its particles move in disorder and their trajectories produce small periodic whirlpools.

Revolve. ANSYS tool, which generates solids of revolution.

Absolute rugosity. It is defined as the average variation of the inner radius of the pipeline.

Relative rugosity. It is defined as the quotient between absolute rugosity and the diameter of the pipeline.
Numeric simulation. It is the execution of the program in one or several computers in order to find out the behavior of a specific system.
Sketch. Selects where we will be working on.
Revolved solids. It is a solid figure obtained as a consequence of rotating a certain flat region around.
Revolved surface. Exterior surface of a solid of revolution.
Reynolds Transport Theorem. It studies the density variation inside an infinitesimal control volume.
Water-liquid (h2o<l>). Term used by ANSYS to describe water.

Fluid Mechanics with CFD Exercises

de la autoría de Mario Ibañez Olvera, Irma Hernández Casco y Juan Alfonso Salazar Torres, se terminó de editar el 5 de junio de 2020.

Corrección
Luis Cejudo Espinosa
Teresa Romero Reynoso

Formación y diseño
Hugo Iván González Ortega

Coordinación editorial
Patricia Vega Villavicencio

Por disposición del Reglamento de Acceso Abierto de la Universidad Autónoma del Estado de México se publica la versión PDF de este libro en el Repositorio Institucional de la UAEM.