

بِسْمِ اللَّهِ الرَّحْمَنِ الرَّحِيمِ



Organisation of Islamic Cooperation

“Study of a Convective Heat Transfer for Laminar and Turbulent Flow of Nano-fluid through a Circular Pipe”

A thesis submitted to the department of Mechanical and chemical Engineering (MCE), Islamic University of Technology (IUT), in the partial fulfillment of the requirement for degree of Higher Diploma in Mechanical Engineering.

Prepared By:

AQIB MEHMOOD (121303)

MOTAPON NJIKAM ISMAIL (121304)

BABACAR NDIAYE (121317)

Supervised By: Mr. Monjurul Ehsan

(Assistant professor MCE department)

Department of Mechanical and Chemical Engineering

Islamic University of Technology

CERTIFICATE OF RESEARCH:

The thesis title “**Study of a convective heat transfer for laminar and turbulent flow of Nano-fluid through a circular pipe** ” submitted by AQIB MEHMOOD (121303), MOTAPON NJIKAM ISMAIL (121304), BABACAR NDIAYE (121317) , has been accepted as satisfactory in partial fulfillment of the requirement for the Degree of Higher Diploma in Mechanical and Chemical Engineering on October, 2015.

Signature of the Supervisor

Mr. Monjurul Ehsan

(Assistant Professor MCE Department)

Department of Mechanical and Chemical Engineering.

CANDIDATE'S DECLARATION

It is hereby declared that this thesis or any part of it has not been submitted elsewhere for the award of any degree or diploma

Signatures of the Candidates.

Aqib Mehmood

Student ID:121303

Babacar Ndiaye

Student ID: 121317

Motapon Njikam Ismail

Student id. 121304

Department of Mechanical & Chemical Engineering (MCE)

Islamic University of Technology (IUT), OIC

Board Bazar, Gazipur Bangladesh.

Signature of the Supervisor

Mr. Monjurul Ehsan

(Assistant Professor MCE department)

Department of Mechanical & Chemical Engineering (MCE)

Islamic University of Technology (IUT), OIC

Board Bazar, Gazipur Bangladesh.

ACKNOWLEDGEMENT:

We express our heartiest gratefulness to Almighty Allah for His divine blessings, which made us possible to complete this thesis successfully.

First and foremost, we feel grateful and acknowledge our profound indebtedness to Mr.Monjurul Ehsan, Assistant Professor, Department of Mechanical and Chemical Engineering, IUT. His endless patience, scholarly guidance, continual encouragement, constant and energetic supervision, constructive criticism, valuable advice at all stage has made it possible to complete this project. We would also like to offer thanks to all who contributed in many ways during the project work. We acknowledge our sincere indebtedness and gratitude to our parents for their love.

We seek excuse for any errors that might be in this report despite of our best efforts.

CONTENTS:

Chapter 1 *1-5*

INTRODUCTION

Chapter 2 *6*

LITRATURE REVIEW

Chapter 3 *7-9*

GEOMETRY

Chapter 4 *10-20*

MESHING

Chapter 5 *21-24*

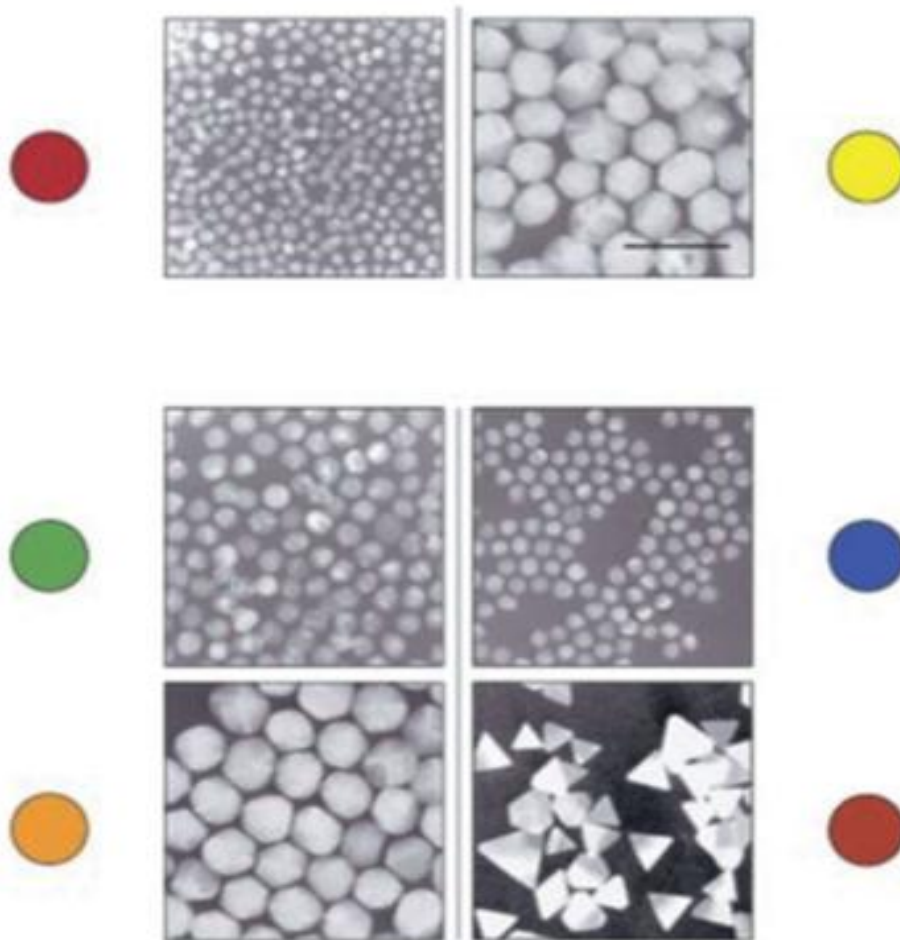
RESULTS AND APPLICATIONS

CHAPTER 1: INTRODUCTION

In recent years, optimum design of compact heat exchangers in engineering and industrial applications is a challenge in the field of thermal engineering science. Nanotechnology is an innovative and inexpensive method of improving the thermal performance of heat exchangers by the stable dispersion of Nano-sized solid metallic particles (usually size less than 100 nm) in traditional base fluid like water, oil, ethylene glycol or any mixture of two liquids. The mixture of the Nano-sized particles with the traditional fluids are called Nano fluids.

Nano fluids is the next frontier in technology. The excitement can be attributed to the sheer brilliance of the idea and the applications of the technology. The properties of Nano fluids need a lot of fine tuning many seemingly contradicting studies need clarity and validation. Nano fluids have potential applications in microelectronics, fuel cells and pharmaceutical industry if we mention a few of the potential applications. The applications of Nano fluid are largely because of the enhanced thermal conductivity. Let us explore the history of nanotechnology a bit and move on to other things. The history of nanotechnology can be traced back to the ninth century A.D. in Mesopotamia (current Iraq) for its use in pottery. Various applications can be seen through the history. The famous paper of Michael Faraday marks the beginning of nanotechnology in the modern world. Faraday talks about the optical properties of nanometer scale meters. The conceptual birth of nanotechnology can be found more in the recent lecture of the legendary scientist Richard P. Feymann. Even though most of the work in the field of nanotechnology came about without the knowledge of this lecture it marked the conceptual origin of modern nanotechnology. The invention of the scanning tunneling telescope triggered the growth of nanotechnology in the 1980's. Nano fluids were a result of the experiments intended to increase the thermal conductivity of liquids. The birth of Nano fluids is attributed to the revolutionary idea of adding solid particles heat transfer fluids to increase the thermal conductivity. This innovative idea was put forth by Maxwell in 1873. Solid particles of micrometer, millimeter magnitudes were added to the based fluids to achieve increase in the thermal conductivity but a posed a range of serious issues like clogging, increase in the pressure drop and the erosion of pipes. These methods couldn't bring about any considerable improvement in the applications of heat transfer fluids. S.U.S Choi and J. Eastman brought about radical changes by introducing Nano scale metallic properties and carbon nanotubes. They worked with a variety of fluids and the result was great. But uncertainty did linger on the practical utility and nature of the Nano fluids. The nanotechnology is still in its early phase and various scientist are working on how to help use Nano fluids to technological riddles of the modern society. Nano fluids are prepared by dispersing nanometer sized particles in heat transfer fluid. They have a large surface area to volume ratio, properties that depend on dispersion, lower kinetic energy and greater stability. Nano fluids are more stable than micro-fluids and milli-fluids. Based fluids behave more or less like pure fluids in the presence of nanoparticles thereby incurring less pressure drop and need for surfactants. The most curious property of Nano fluid is the remarkable enhancement of thermal conductivity even by the addition of very small amounts nanometer sized particles.

There are different nanoparticles used in the world of nanotechnology. Each of these nanoparticles have different properties and they are used in different applications. A minute quantity is required to enhance the thermal conductivity of the heat transfer fluid. In general the size of a nanoparticle is between 1 to 100 nm. Metallic nanoparticles have different physical and chemical properties from bulk metals (e.g., lower melting points, higher specific surface areas, specific optical properties, mechanical strengths, and specific magnetizations), properties that might prove attractive in various industrial applications. However, how a nanoparticle is viewed and is defined depends very much on the specific application. Nanotechnology is easily evident in various old churches. A well-known application of early nanotechnology is the ruby red color that was used for stained glass windows during the middle Ages. Beautiful examples of these applications can be found in glass windows of many Gothic European cathedrals, among which the León cathedral, located on the medieval French pilgrimage path to Santiago de Compostela (Spain); its impressive colored windows offer a unique view that certainly warrants a visit. Of course, the medieval artisans were unaware that they were using nanotechnology. They just knew that a particular process produced a beautiful effect. For example, the stained glass of a wonderful rose can be seen at the world heritage Cathédrale Notre-Dame de Chartres in France. The stained glass made in medieval times is displayed in Figure. Later chemistry clarified the reasons behind the generation of the color. These vivid colors were controlled by the size and the form (or shape) of the nanoparticles of gold and silver. The relation between particles and their associated colors has been discussed recently by Jin and coworkers. In an article of 22 February 2005, the New York Times summarized the relationship between the color of stained glass and the size/shape of the nanoparticles. After several decades, the ingredients present in the stained glass (Colored glass) of various churches were clarified subsequent to the development of analytical instruments. People without professional expertise in nanotechnology are also increasingly contributing to the technology.



Source: Dr. Chad A. Mirkin, Institute of Nanotechnology, Northwestern University

The physical and chemical properties of Nano materials depend not only on their composition but also on the particle size and shape. *Accordingly, a high quality synthesis protocol must first of all provide control over particle size and shape.* For example, if the diameter of an Au Nano sphere is made to increase, the surface Plasmon resonance will be gradually shifted from 530 nm to the longer wavelength side. Thus, if nanoparticles differ in size, their optical characteristics will also change significantly. In optical applications of nanoparticles, simplification of the size distribution of the particles becomes a very important factor. *Therefore, it is important to fabricate Nanoparticles with a single target size in mind.* Generally, in order to prepare mono dispersed nanoparticles, it is imperative that the nanoparticles grow very slowly after the rapid generation of the seed particles. If the size of the nanoparticles decreases (i.e., increase in specific surface area), then the increase in the surface energy of such nanoparticles will facilitate their aggregation. Consequently, after their growth to the desired optimal size, it will be necessary to stabilize the particulate surface by addition of a dispersing agent. The historical use of dispersing agents in nanoparticle syntheses is not new; for example, Ag colloids protected by

citrate were reported by Lea way back in 1889. However, where the concentration of nanoparticles is unusually high, the decentralized stabilization will fall, because the protective action of the organic substrate (citrate) is no longer strong enough to prevent aggregation. Thus, several studies of dispersing agents that maintain a high dispersivity of the nanoparticles, and also at various concentrations, have been reported. According to the hard and soft acids and bases (HSAB) rule , Ag + , Au + , Pd 2 + , Pt 2 + are classified as soft acids in the Lewis sense (SA), and substrates possessing the thiol (R – SH) and the phosphine (P – R 3) functional groups, classified as soft bases, have proven to be suitable dispersing agents .

Early research that examined organic thiol molecules as possible dispersing agents was reported by Brust and coworkers. If 1-dodecanethiol is used as the dispersing agent in Au nanoparticle synthesis the 1-dodecanethiol molecule can form a monomolecular layer on the Au nanoparticle surface, and firmly stabilize the dispersed Au nanoparticles.

Regarding our project the nanoparticles will be used in the heat exchangers. Three types of nanoparticles will be used which are Aluminum oxide, titanium oxide and copper.

Aluminum oxide: It is mostly use in the car radiators of cars. Eastman *et al* found that by adding 5% volume fraction of Al₂O₃ (33 nm) nanoparticles in water, the thermal conductivity can be enhanced by 29%. For bigger unfortunately thermal conductivity is not the only property that determines the efficiency of heat transfer in the system. Convective heat transfer coefficient also plays an important role in evaluating heat transfer capability of an automotive radiator. In the forced flow system (such as in engine cooling by radiator) the coolant is pumped through the radiator, introducing convective heat transfer mechanisms and pumping power penalties. Heris investigated convective heat transfer of Al₂O₃/water based Nano fluids under laminar flow conditions through annular copper tube. They observed that the heat transfer coefficient increases with the increase in particle volume fraction. The convective heat transfer coefficient increases with increasing flow velocity, Reynolds number and particle concentration. It is also increases with decreasing particles sizes.

Majority of the experimental studies on the heat transfer of Nano fluids were constrained to the determination of their thermal conductivity, viscosity, convective heat transfer coefficient without considering the performance of the Nano fluids under actual application, especially in automotive cooling. Peyghambarzadeh *et al* conducted heat transfer study in the application of Al₂O₃ Nano fluids as coolant for car radiators. They conclude that by adding as much as 1.0% volume of Al₂O₃ to the base fluid (water or EG), the Nusselt number can be increased by up to 40%. [2]

Copper: Xuan and Li investigated turbulent convective heat transfer of copper/water Nano fluid and observed more than 39% enhancement with 1.5% volume concentration. Xuan and Roetzel developed a heat transfer correlation for Nano fluids to capture the effect of energy transport by particle “dispersion”. Yang et al measured laminar convective heat transfer performance of graphite Nano fluids in horizontal circular tube. Wen and Ding studied Nano fluid laminar flow convective heat transfer and reported 41% enhancement in the entry region. Ding et al observed significant convective heat transfer enhancement of multi-walled carbon

nanotube dispersion in water and the enhancement depended on the flow conditions (Reynolds number), and volume concentration.

Duangthongsuk et al reported 11% enhancement in the convective heat transfer coefficient with 0.2 % volume concentration of TiO₂ nanoparticles flowing in a horizontal double-tube counter flow heat exchanger under turbulent flow conditions. Convective heat transfer of alumina Nano fluid in micro-channels was investigated by Lee et al. Nano fluid application in microelectronics cooling was recently explored by Chein and Chuang . All the Nano fluid studies reported in the literature have concluded that Nano fluids provide higher heat transfer enhancement with respect to the base fluids; and the Nano fluids have higher heat transfer coefficients than those of the base fluids at the same Reynolds number. In the recent past, most of the convective heat transfer studies have been performed in Nano fluids with only oxide nanoparticles. Moreover, Maiga et al reported that, with regard to the Nano fluid thermal properties, the actual amount of experimental data available in the literature remains surprisingly small. However, only very few work has been reported on the hydrodynamic and heat transfer behavior of Nano fluids in the laminar, transition and the turbulent regimes with pure metal nanoparticles such as silver, copper, gold and graphene with different volume concentrations probably less than one volume percentage (< 1 %). Hence, more investigations on the convective heat transfer coefficient and pressure drop characteristics of a Nano fluid suspended with metallic nanoparticles with low volume concentration is essential for developing the new advanced heat transfer fluid.

The use of Nano fluids in a wide variety of applications appears Promising. But the development of the field is hindered by (i) the lack of agreement of the results obtained by different researchers; (ii) poor characterization of suspensions; and (iii) lack of theoretical understanding of the mechanisms responsible for changes in properties.

This article, therefore, concludes by outlining several important issues that should receive greater attention in the near future. Further experimental studies in the convective heat transfer of Nano fluids are needed in the following areas.

Future convective studies must be performed with metallic nanoparticles with different geometries and concentrations to consider heat transfer enhancement in laminar, transition and turbulence regions. 2. The use of Nano fluids in micro heat pipes and looped heat pipes for satellite cooling application could enhance the performance with considerable reduction in thermal resistance. However, recent studies indicate particle aggregation and deposition in micro-channel heat sinks. Further study is required in these areas to identify the reasons for and the effects of particle deposition. 3. There appears to be hardly very few research papers in the use of Nano fluids as refrigerants.

Nanoparticles-refrigerant dispersions in two-phase heat transfer applications can be studied, to explore the possibility of improving the heat transfer characteristics of evaporators and condensers used in refrigeration and air-conditioning appliances. Applied research in Nano fluids which will define their future in the field of heat transfer is expected to grow at a faster pace in the near future.

Titanium oxide: Kavitha et al: Investigated thermal conductivity enhancement of TiO₂-water Nano fluid using transient hot wire device. Thermal conductivity of nanoparticle is increased when using spherical shaped nanoparticle, and thermal conductivity is dependent many parameter such as size, shape, stability, and coating of nanoparticle. Spherical shaped nanoparticle having high heat transfer rate compared with other shaped nanoparticle. Among these three nanoparticles aluminum oxide has the higher heat transfer rate.

Chapter 2: LITERATURE REVIEW.

Knowing the notion of nanotechnology and nanoparticles from the introduction above we are going to examine one of its application which deals with heat exchangers. Here we will study the convective heat transfer of the study of convective heat transfer of laminar and turbulent flow of Nano-fluid through a circular pipe. We all know heat exchangers are meant for transferring heat between two fluid bodies at different temperatures without mixing the two fluid bodies together. These are done with the help of tubes. For better. The heat transfer in the exchangers depend on certain factors like wall thickness, thermal conductivity, area just to name a few. When using a base fluid like water, ethylene glycol and oil in heat exchanger transfer of heat between the fluid will on go on. But with the addition of nanoparticles to these base fluids will enhance the heat transfer characteristics. This is because these nanoparticles have superior thermo-physical properties. Moreover we are also going to calculating the Nusselt number for the convective laminar and turbulent flow. We will also verify if the Nusselt number increases with the increase in the nanoparticle fraction in the base fluid. Furthermore we also calculate the heat transfer coefficient for both turbulent and laminar flow. Note in this project we will use water as the base fluid and copper, aluminum oxide and titanium oxide as nanoparticles at different fraction and determine which of these nanoparticles have a better heat transfer enhancement.

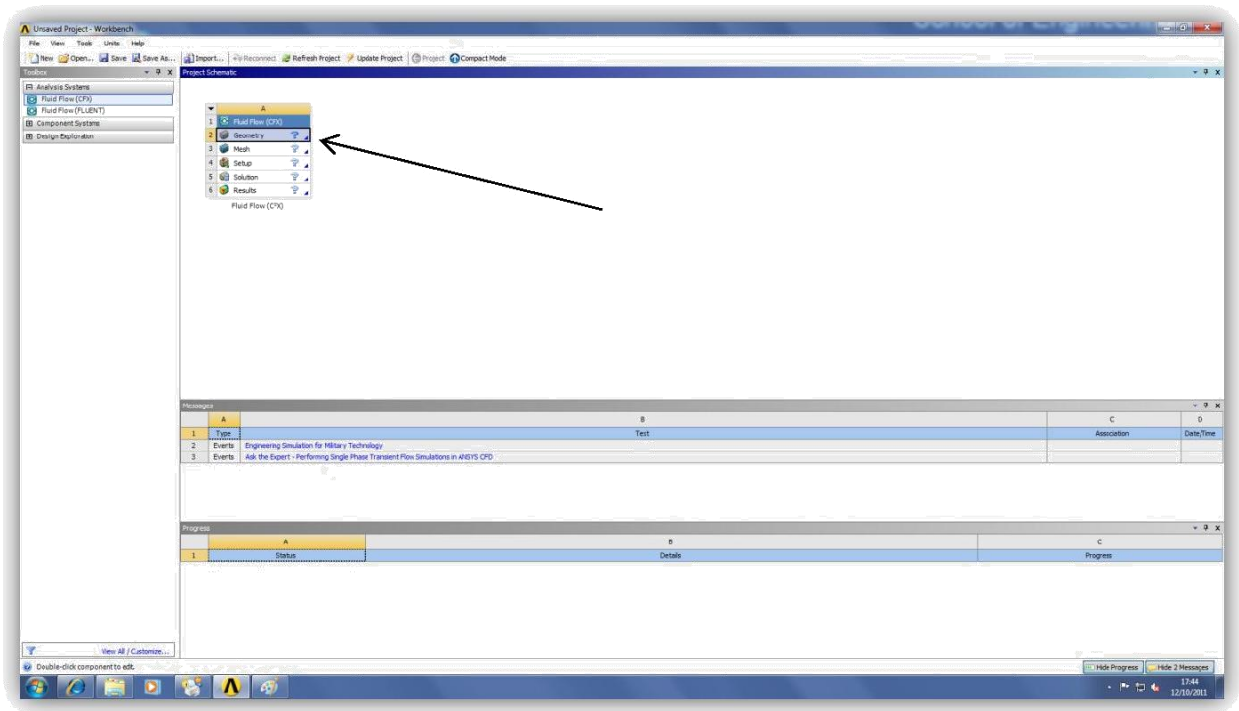
In this project we are going to a circular pipe having a diameter of 5mm and length 1m. The simulation of this project will be done using the ANSYS software more specifically we are going to use the ANSYS fluent. The simulation is controlled by governing equations which includes momentum equation, continuity equation and energy equation. For different Reynolds number we will obtain Nusselt number for both laminar and turbulent flow. Note that for a laminar flow in a circular pipe the Nusselt number equal 4.36. But for turbulent flow it varies depending on the correlation being used. In this project the Dittus-Boetler correlation was used. The Nusselt number was using the CFD (Computational Fluid Dynamics) post. CFD is a branch of fluid mechanics which uses algorithm to solve and analyze problem that involves fluid flows. Here computers are used to perform the calculations required to simulate. [1]

Chapter 3: Geometry

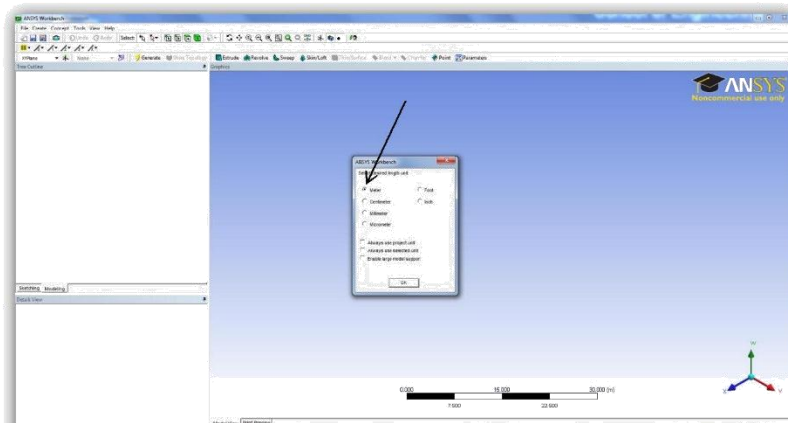
Geometry:

In this project, we have used a straight Nano pipe having specified length and diameter. The steps of modelling the pipe within Ansys DesignModeler will be discussed farther:

Step1:Launch ANSYS ,by going to the start-up menu and double clicking on workbench file in the ANSYS 13.0 folder, Go to Analysis Systems (CFX) and double click, then double click on Geometry.

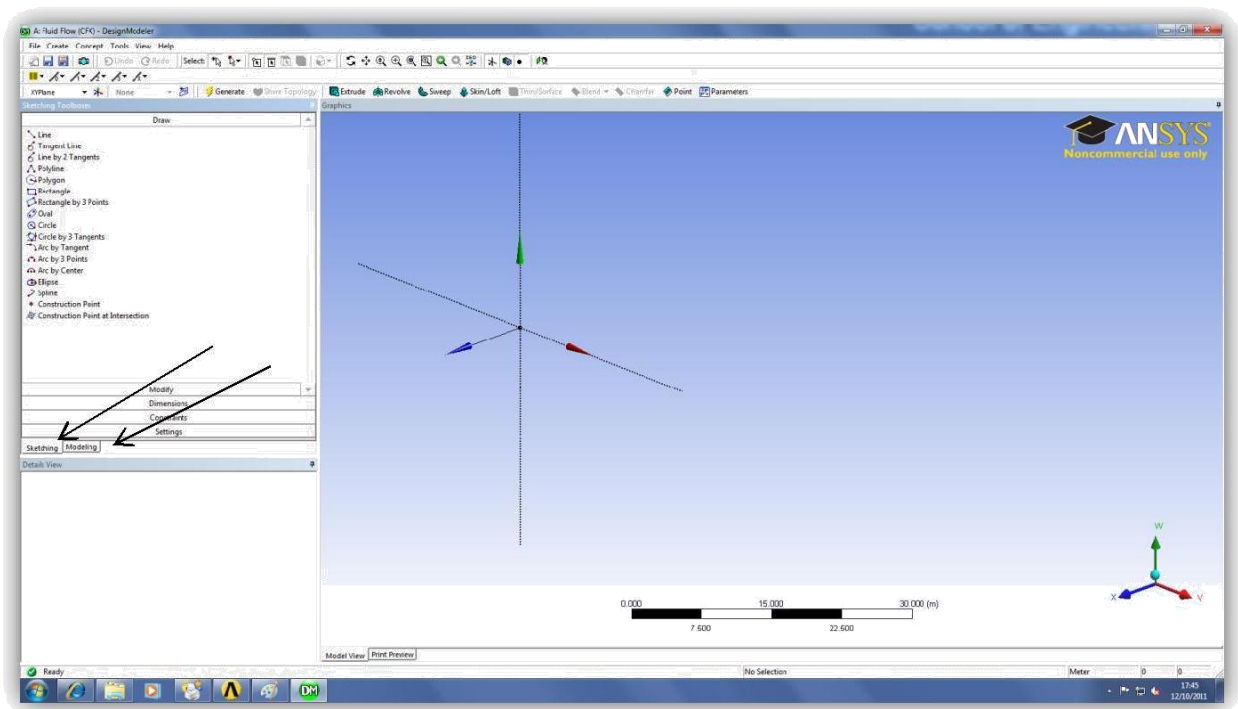


Step2: Select the unit, in here the units selection depends on what kind of units will fit you job. In our project we used meters.



Step3: Click on the icon that has the Cartesian coordinate's icon. Once it's clicked you can see where the plotting plane is.

Step4: Looking at the window you will see the Cartesian coordinates system is shown on the screen with a blue arrow referring to the x axis, green arrow referring to the z axis, red arrow referring to the y axis. A note to the user that 2d sketching is conducted on the specified xy plane as long as the user specifies some other plotting plane (xz plane ,yz plane)



The user can access the modelling tools by pressing on the Sketching tab or he can access the modelling planes by going to Modelling.

Step5: Due to the need to conduct a 2d sketch, as desired in our specific project, it's advisable to project the view in the direction to the xy plane, this is done by pressing on the right hand side button of the mouse and selecting look at. However you can choose other viewing options provided under views.

Step6: Select the Line and start sketching the pipe profile, the line command is done by pressing twice, once you have drawn the first line, press again on the last plotted point and

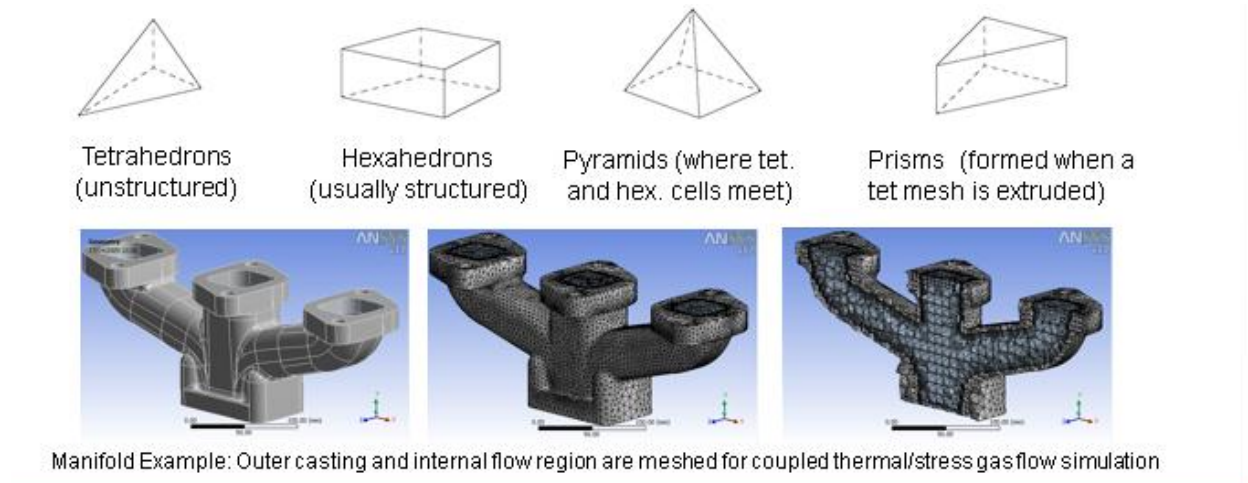
continue till you plot all the profile, remember to make sure that the profile contour is closed. There is a second alternative which consist to click-on on the rectangle icon and draw your rectangle of the desired length and the radius of the pipe. Note that it's is advisable to always start in 0,0 coordinates. In other words, the origin of your Cartesian coordinates system.

Step7: Finally select the revolving command to revolve the obtained figure to finally end up with your pipe with desired length and diameter.It is also to note that the model can be imported to ANSYS Workbench from other modeler like SolidWorks and some other Cad program.

CHAPTER 4: Meshing

For both CFD (fluid) and FEA (solid) modelling, the software performs the computations at a range of discrete locations within the domain. The purpose of meshing is to decompose the solution domain into an appropriate number of locations for an accurate result.

The basic building-blocks for a 3D mesh are illustrated in the following image:



Types of mesh:

There are basically two types of meshing. Free and Mapped Meshing.

Before meshing the model, and even before building the model, it is important to think about whether a free mesh or mapped mesh is appropriate for the analysis. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it. A mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. A mapped area mesh contains either only quadrilateral or only triangular elements, while a mapped volume mesh contains only hexahedron elements. In addition, a mapped mesh typically has a regular pattern, with obvious rows of elements. If you want this type of mesh you must build the geometry as a series of fairly regular volumes and/or areas that can accept a mapped mesh.

Mesh control:

The default mesh controls that the ANSYS program uses may produce a mesh that is adequate for the model you are analyzing. In this case, you will not need to specify any mesh controls. However, if you do use mesh controls, you must set them before meshing solid model.

Mesh controls allow you to establish such factors as the element shape, midsize node placement, and element size to be used in meshing the solid model. This step is one of the most important of your entire analysis, for decisions you make at this stage in your model development will profoundly affect the accuracy and economy of your analysis.

For further Mesh details, consult ANSYS website or any book about the software.

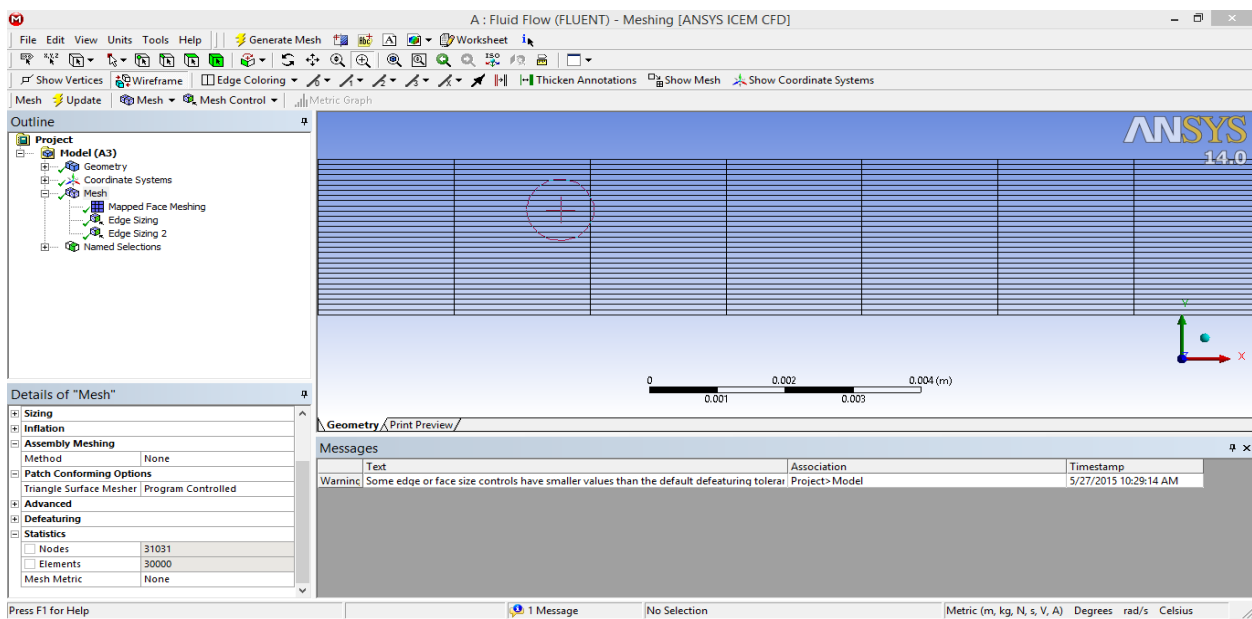
Now we would like to give a brief discussion about how the meshing has been done on our particular project.

Step1: Launch your workbench as discussed in previous parts.

Step2: Check on the toolbox and double click on “fluent flow”

Step3: After completing the step2, a window normally pops up on the project schematic and then double click on “Mesh”.

Step4: Define your element sizing, depending on the number of element desired from the window at the down part of the toolbox. We had 30000 elements on our projects.



Step5: This final step consist to just generate your mesh after defining your mesh elements and others, by just clicking on “generate Mesh” on the top left corner of your workbench.

However, some setting has got to be mad for a better mesh, as we used mesh control, we allocated every edge the desired number of element: for the Wall and the Axis we allocated 10000 elements and for inlet and outlet we got 30 elements. And never forget to click on “hard” for the behavior and “number of division” on the type, as shown on the image below.

Note that the image was not taken from our project meshing.

Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	12
Behavior	Hard
Bias Type	_____ - - - -
Bias Factor	1.5

Setup:

This part is basically the most important one. This is where the simulation process takes its information from. It is quite different from one project to another, e.g.; one system or material to another. However we will discuss according to our project.

Step1: Adding constraints to ANSYS Design Modeler Parameters in ANSYS Workbench.

In this step, you will start ANSYS Workbench, open the projectile file, review existing parameters, create new parameters, and add constraints to existing ANSYS Design Modeler parameters.

1. Start ANSYS Workbench by clicking the windows start menu, then selecting the workbench option in the ANSYS14.0 program group.

Start → All Programs → ANSYS14.0 → Workbench14.0

This will displays the ANSYS workbench application window, which has the **toolbox** on the left and the **project schematic** to its right. Various supported applications are listed in the toolbox, and the components of the analysis system are in the Project schematic.

2. Open the project file for your defined project. Parameter already has a FLUENT-based fluid flow analysis system that includes the geometry and mesh, as well as some predefined parameters. You will first examine and edit parameters within Workbench, then later proceed to define the fluid flow model in ANSYS FLUENT.

3. Review the parameters that have already been defined in ANSYS Design Modeler.

a. Click the **Parameter Set** bar in the ANSYS Workbench Project schematic.

b. Select the **outline** option under the View menu.

View → Outline

This displays the Outline of all Parameters view in ANSYS Workbench, where you can view an outline of all parameters associated with the current project.

c. Select Properties option under the View menu

View → **Properties**

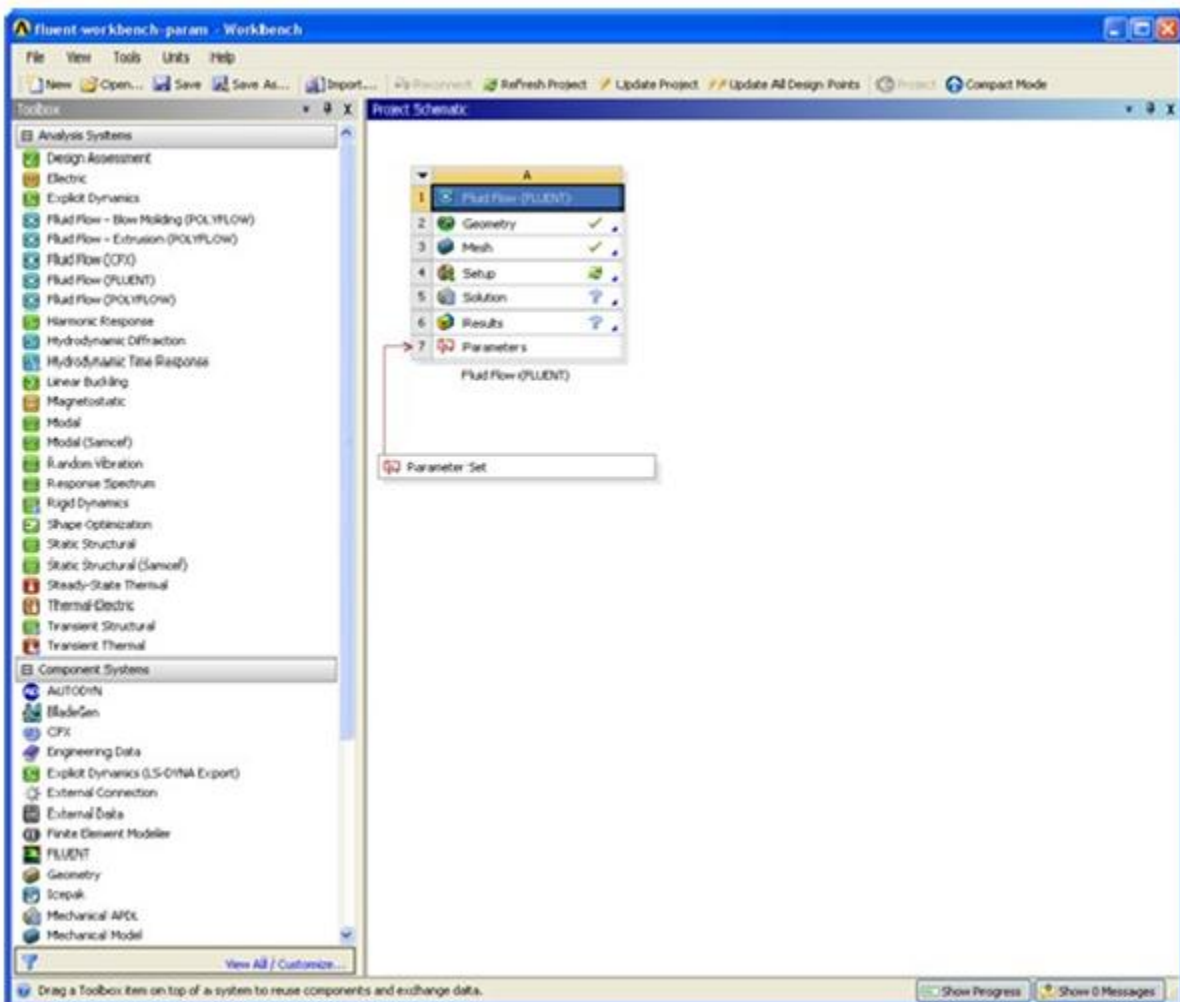
This displays the Properties view in ANSYS Workbench

d. Select the Files option under the View menu (optional)

View → **Files**

This displays the Files view in ANSYS Workbench, where you can view the files associated with the current project.

The Project Loaded into ANSYS Workbench Displaying Parameter Outline, Properties, and Files View



Note the types of files that have been created for this project. Also note the states of the cells for Fluid Flow analysis system. Since the geometry and mesh have already been defined, the status of the Geometry and Mesh is up to date. Also since the ANSYS FLUENT setup is not complete, the setup cell's state is refresh required and since the simulation has yet to be performed, with no corresponding results the state for the solution and results cells is unfulfilled, with no corresponding results, the state for the solution and results cells is unfulfilled.

Step2: Setting up the CFD Simulation in ANSYS fluent.

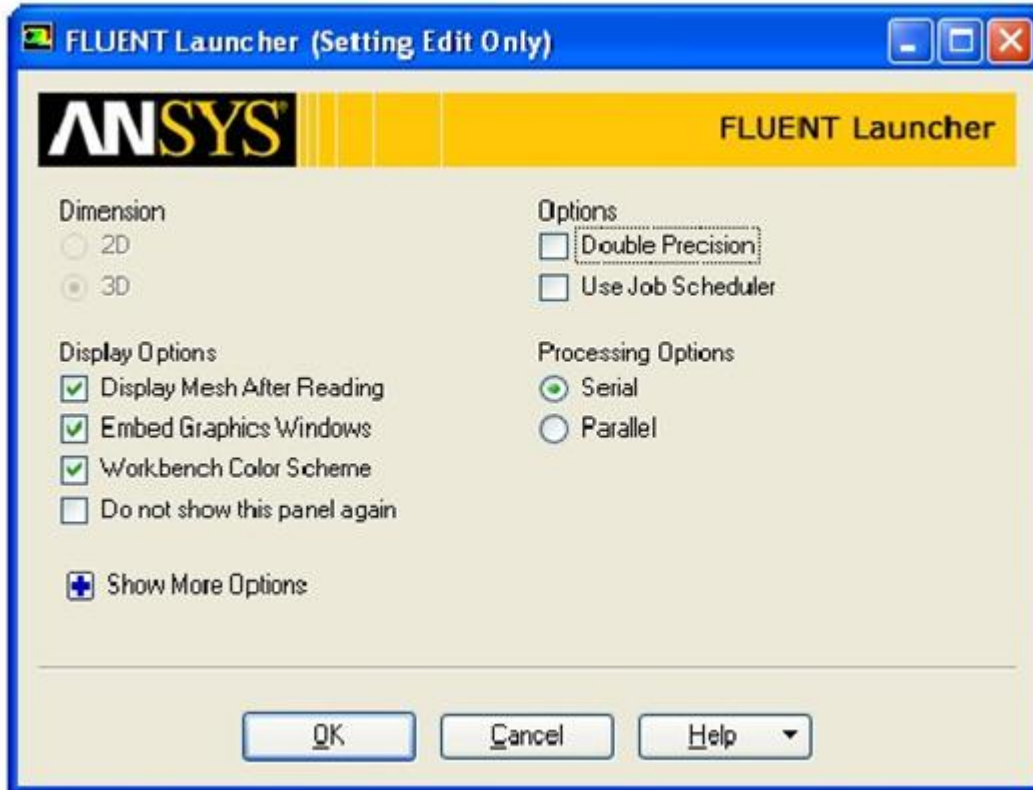
Now that you have edited the parameters for the project, you will set up a CFD analysis using ANSYS Fluent. In this step, you will start ANSYS Fluent, and begin setting up the CFD simulation.

1. Start the ANSYS FLUENT.

In the ANSYS Workbench project schematic, double click the Setup cell in the ANSYS FLUENT fluid flow analysis system. You can also right click the setup cell to display the context menu where you can select the edit option.

When ANSYS Fluent is first started, fluent launcher is displayed, allowing you to view and/or set certain ANSYS Fluent start-up option.

Fluent launcher allows you to decide which version of ANSYS Fluent you will use, based on your geometry and on your processing capabilities.



This figure shows the ANSYS FLUENT Launcher.

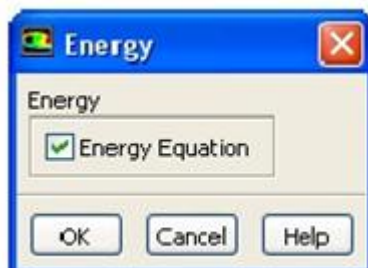
Note that the dimension 2D or 3D is automatically set by the software according to your geometry, and it is unchangeable.

Make sure to enable these options on the launcher based on your project.

2. Set up your models for the CFD simulation.

a. Enable heat transfer by activating the energy equation.

Models → Energy → Edit



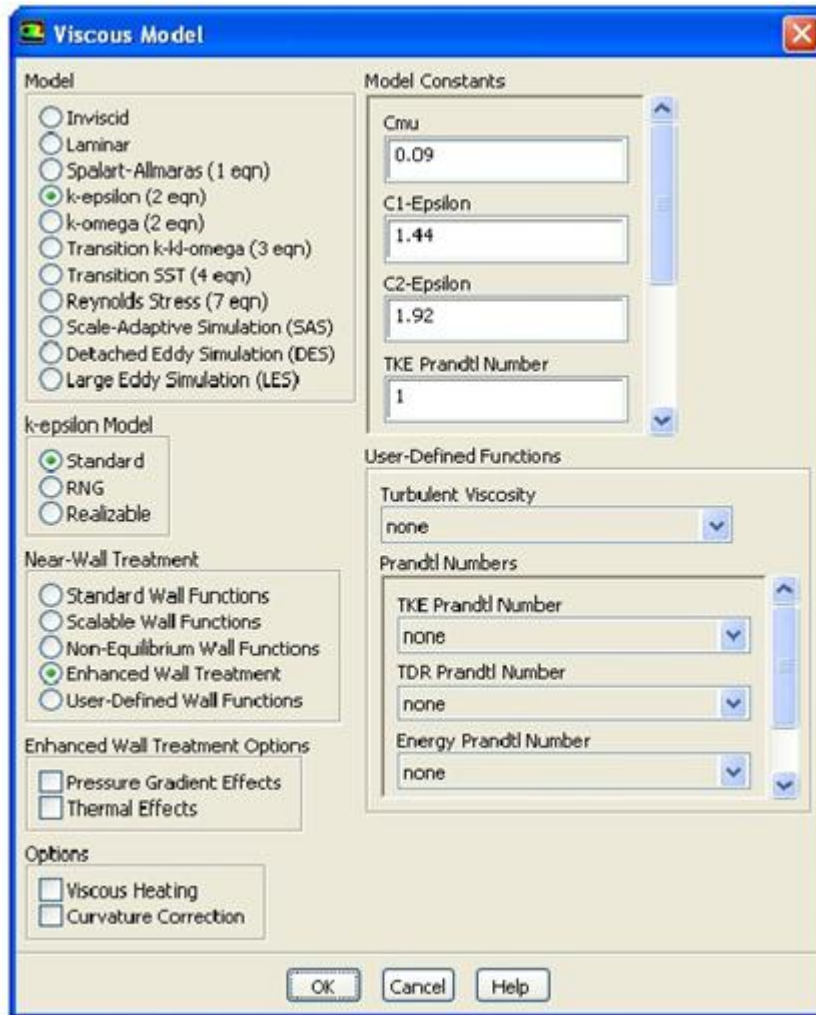
i. Enable the Energy equation option.

ii. Click Ok to close the Energy dialog box.

b. Enable the k-epsilon turbulence model.

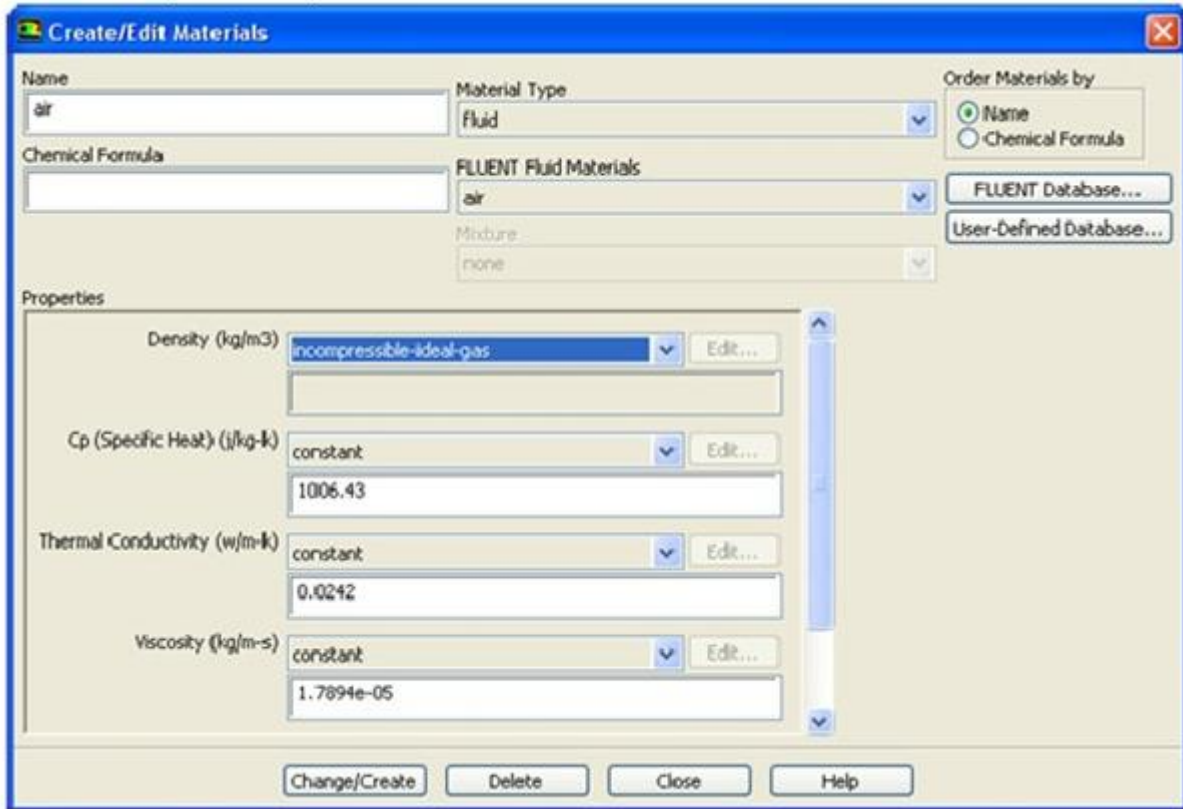
Models → viscous → edit

Again. Make sure to enable or disable any option according to your project. These selections has been done according to our project.



4. Set up your materials for the CFD simulation.

Material → water → create/edit



Note that in this figure Air is used instead of water. Unlike ours where we used water.

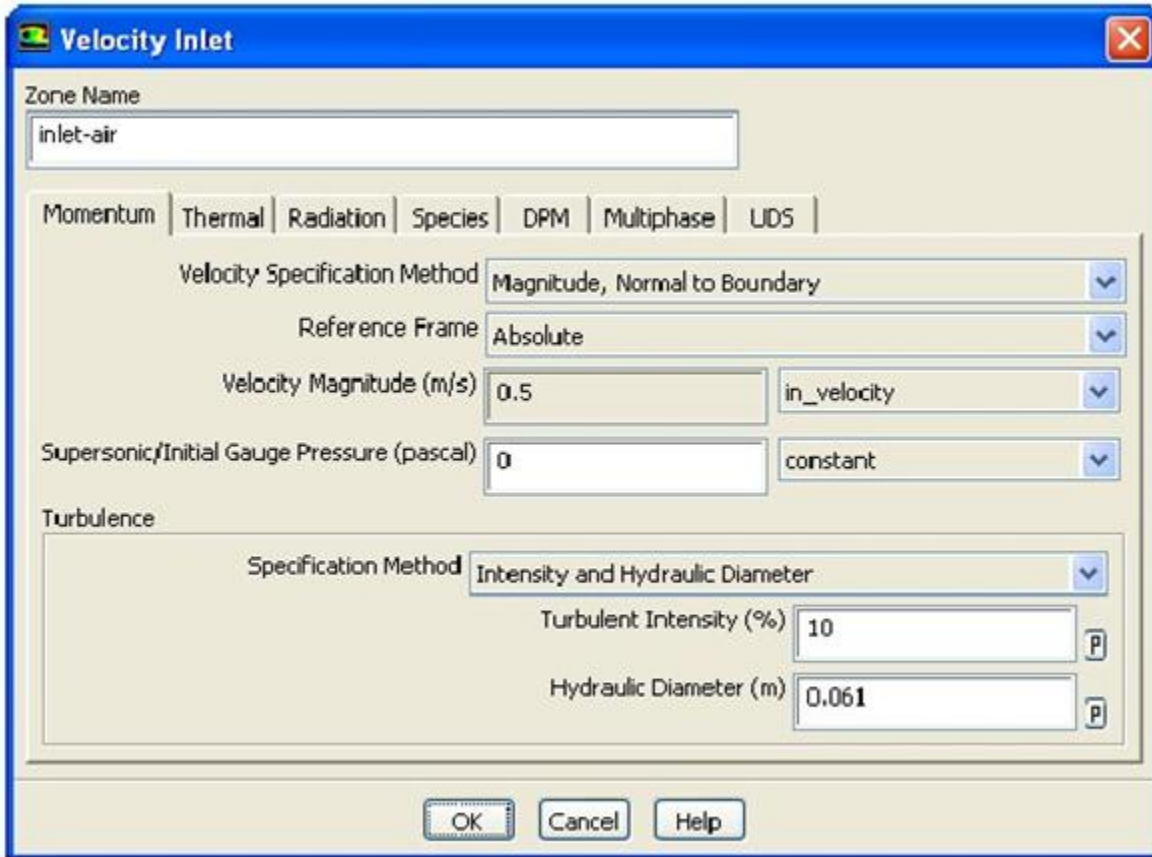
Step3: Define Input and Output Parameters in ANSYS Fluent and Running the Simulation

You have now started setting up the CFD analysis using ANSYS Fluent. In this step, you will define input parameters for the velocity inlet, set output parameters for the heat exchanger volumes, define heat source boundary conditions for the evaporator and heat exchanger volumes, define heat source boundary conditions for heat exchanger, then calculate a solution.

1. Define an input called in_velocity for the velocity at the inlet boundary

Boundary conditions **inlet-water** **Edit**

This displays the velocity inlet dialog box



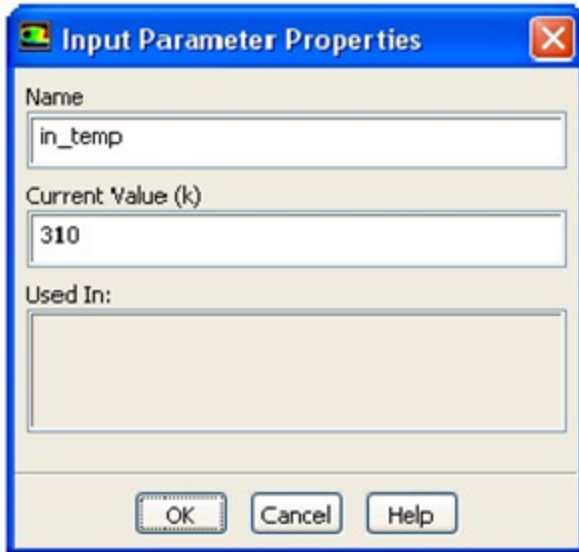
Note that this figure has been captured while performing a project that used air instead of water.

In the velocity inlet dialog box, select new input parameter from the drop down list for the velocity magnitude. This displays the input parameter properties dialog box. And from there, enter in_velocity for the name, and enter your current value, based on your project.

Click OK to close the input Parameter properties dialog box.

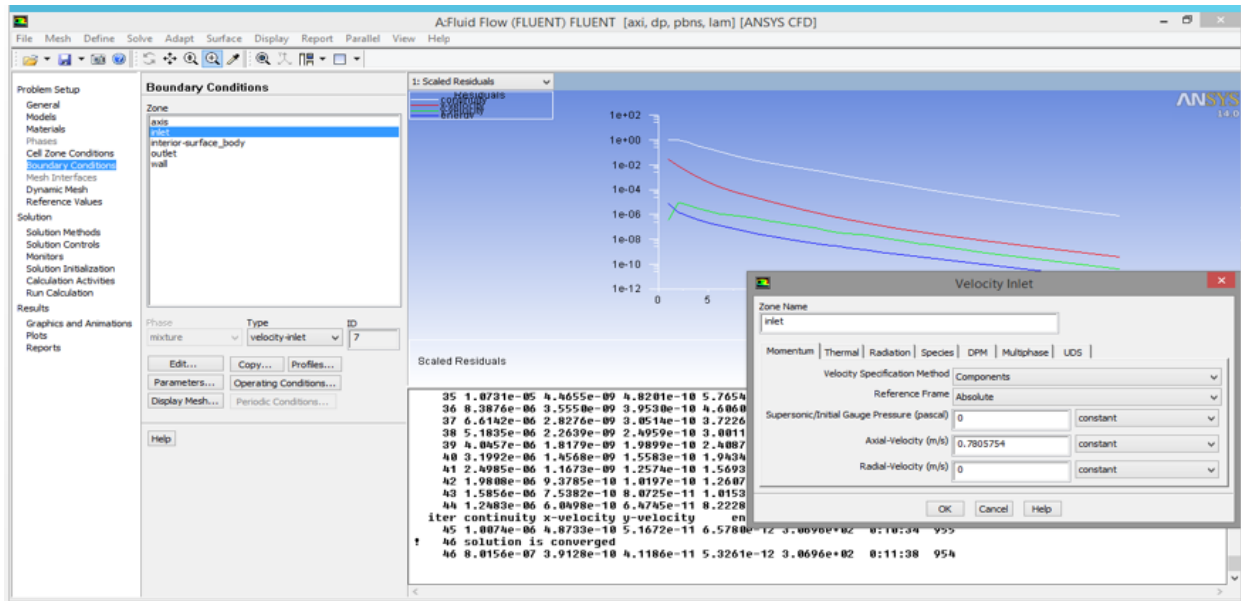
In the inlet velocity inlet dialog box, select intensity and hydraulic diameter from the specification method drop down list in the turbulence group box. Retain the value of the turbulence intensity according to your project and enter the hydraulic diameter.

In the thermal tab of the velocity inlet dialog box, select New input Parameter... from the drop-down list for the temperature.



You know can assign your desired inlet temp. Keep the name in_temp and then click OK to close the input parameter properties dialog box. Again click OK to close the velocity inlet dialog box.

The following figure gives a glance about our boundary and inlet condition according to our project, where we used water.

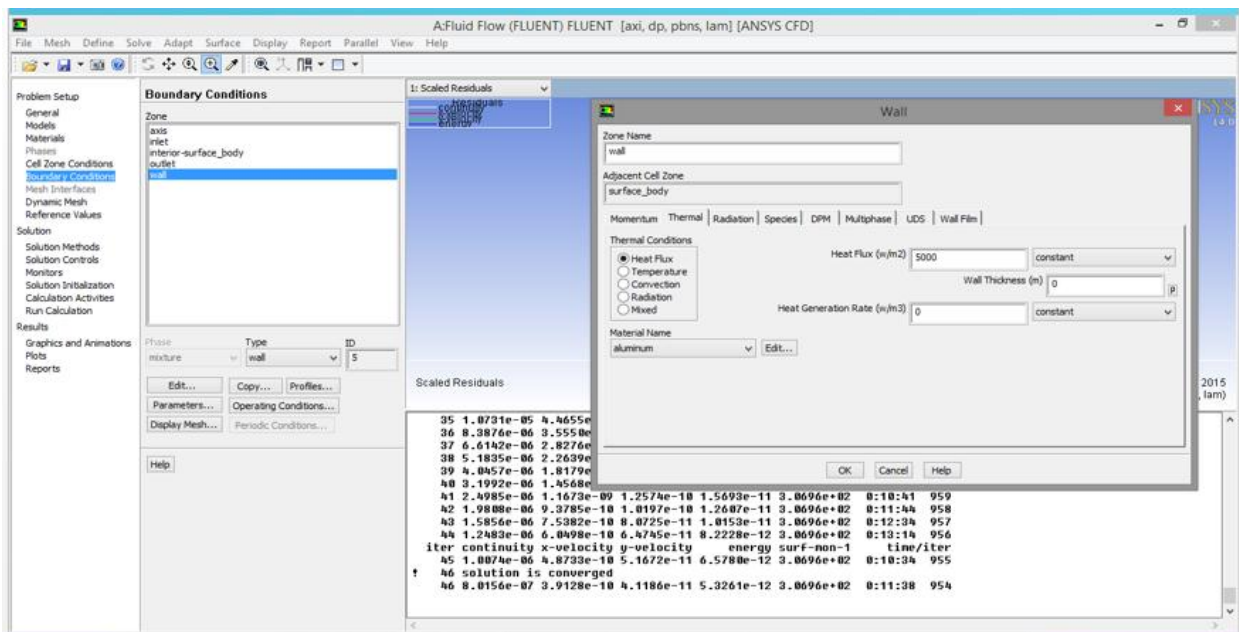


5. Solution Initialization.



Retain the default selection of Hybrid initialization and click the initialize button.

After the initialization, you can directly set the number of iterations, run the simulation and wait for the result to converge. The following figure shows the wall conditions according to our project and the number of iteration at which the solution has converged.



Note that, in our project, it converged at 47th iteration. However it depends from one project to another. It can take fewer or more time to have a converged solution.

CHAPTER 5: RESULTS AND APPLICATIONS.

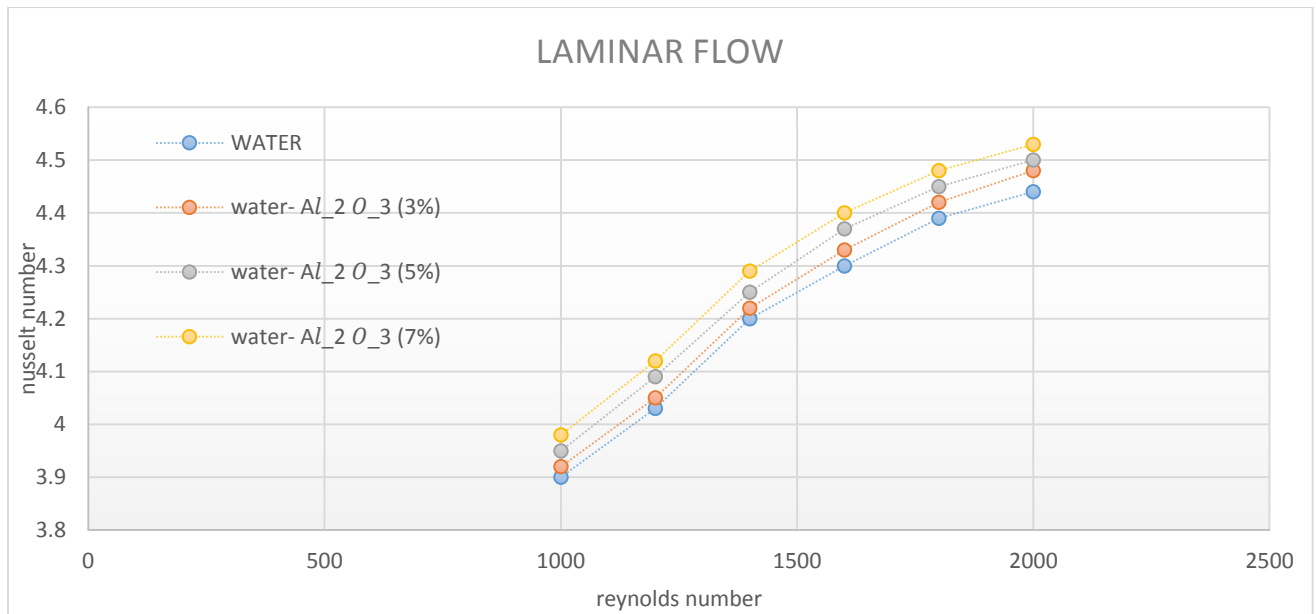
1) CFD RESULTS FOR LAMINAR FLOW

Reynold's Number (Re)	Heat transfer coefficient (h)	Nusselt Number (Nu)	Theoretical Nusselt number	Percentage error(%)
1000	587.682	3.91	4.36	10
1200	608.568	4.05	4.36	7
1400	628.614	4.18	4.36	4
1600	647.836	4.31	4.36	1
1800	666.311	4.43	4.36	1.6
2000	684.088	4.55	4.36	4

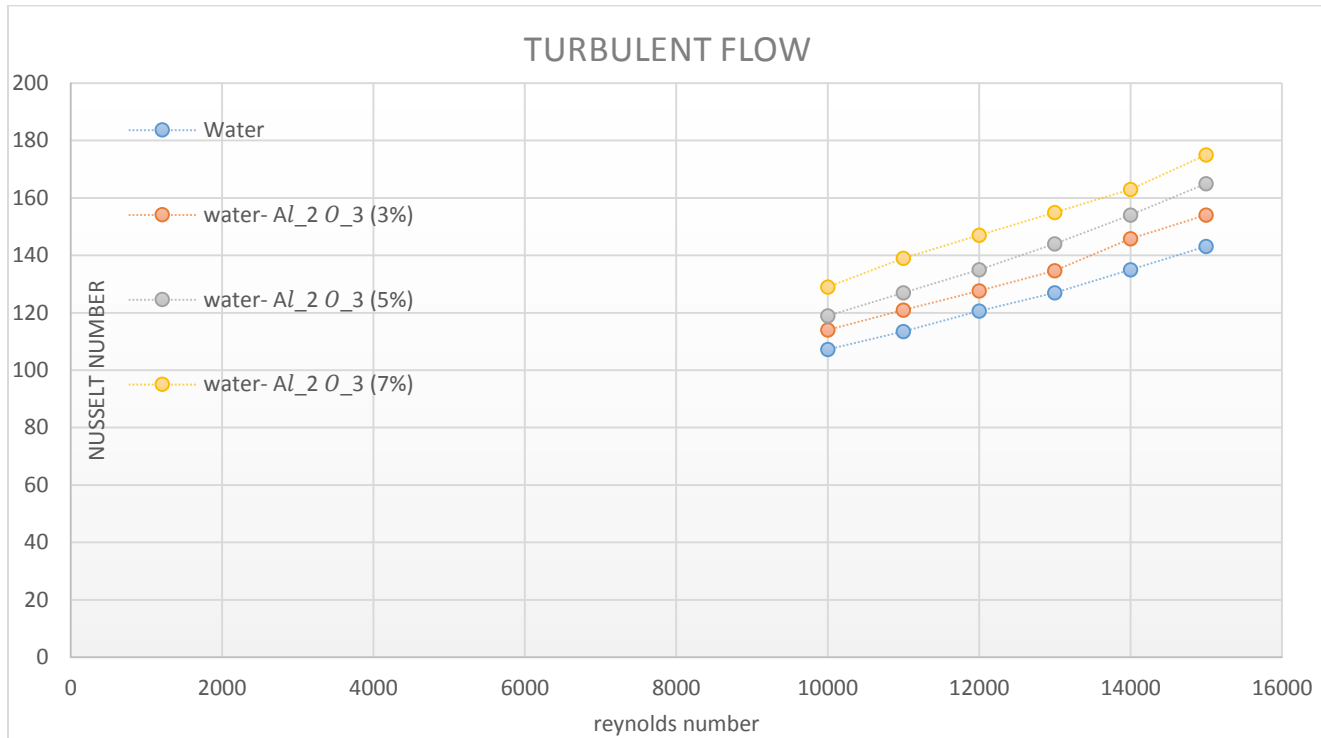
2) CFD RESULTS FOR TURBULENT FLOW

Reynold's Number (Re)	Heat transfer coefficient (h)	Nusselt Number (Nu)	Heat transfer coefficient from ditus boelter correlation.	Nusselt number from ditus boelter correlation.	Percentage error for the Nusselt number(%).
10000	17543.85	116.71	14537.67	96.66	20
11000	18587.36	123.6	15671.68	104.2	18
12000	19531.25	129.93	16799.68	111.7	16
13000	21008.4	139.75	17914.144	119.11	17
14000	22321.4	148.9	19010.46	126.4	17.8
15000	23696.7	157.64	20093.44	133.6	17.9
16000	25000	166.3	21146.24	140.6	18.2
17000	26315.8	175.03	22199.04	147.6	18.5
18000	27624.3	183.77	23236.8	154.5	18.7

3) GRAPH FOR LAMINAR FLOW



4) GRAPH FOR TURBULENT FLOW.



APPLICATIONS:

Nano fluids can be used to improve heat transfer and energy efficiency in various fields of thermal systems. Recently, the number of companies that see the potential of Nano fluid technology and are in active development work for specific industrial applications is increasing project.

Transportation:

An ethylene glycol and water mixture, used as automotive coolant, is a relatively poor heat transfer fluid compared to water alone. Engine oils perform even worse as a heat transfer medium. The addition of nanoparticles to the standard engine coolant has the potential to improve automotive and heavy duty engine cooling rates. Such improvement can be used to remove engine heat with a reduced -size coolant. Smaller coolant systems result in smaller and lighter radiators, which in turn benefit almost every aspect of car and truck performance and lead to increased fuel economy.

MECHANICAL APPLICATIONS:

Nanoparticles in Nano fluids form a protective film with low hardness and elastic modulus on the worn surface can be considered as the main reason that some Nano fluids exhibit excellent lubricating properties. Magnetic fluids are kinds of special Nano fluids. Magnetic liquid rotary

seals operate with no maintenance and extremely low leakage in a very wide range of applications, and it utilizing the property magnetic properties of the magnetic nanoparticles in liquid.

OTHER APPLICATIONS:

Nano fluids are primarily used as coolant in heat transfer equipment such as heat exchangers, electronic cooling system (such as flat plate) and radiators. Heat transfer over flat plate has been analyzed by many researchers. Graphene based Nano fluid has been found to enhance Polymerase chain reaction efficiency. Nano fluids in solar collectors is another application where Nano fluids are employed for their tunable optical properties.

CONCLUSIONS:

We notice that from the laminar flow results as the Reynolds number increases our Nusselt number also increases. This also goes through for the turbulent flow. The reason is Reynolds number increases as fluid velocity increases and/or as fluid viscosity decreases. These two factors (velocity increase and viscosity decrease) both enhance heat transfer as they promote higher rates contacts between “portions” of fluid, which are at different temperatures, in different sites (locations).

The Nano fluid with higher volume fraction yields better enhancement in heat transfer rate as we used the water- Al_2O_3 Nano-fluid. The results shows that the convective heat transfer coefficient of Nano fluid is slightly higher than that of the base liquid at same mass flow rate and at same inlet temperature. The Nusselt number was determined for different Reynolds number using the CFD-Post from the Ansys software.

REFERENCES:

- <https://en.wikipedia.org/wiki/Nanofluid#Applications>
- “STUDY OF CONVECTIVE HEAT TRANSFER FOR TURBULENT FLOW OF NANOFLUIDS THROUGH CORRUGATED CHANNELS” (ASME International Mechanical Engineering Congress and Exposition.)[1]
- Study of turbulent convective heat transfer enhancement by Al₂O₃-water nanofluid through a rough circular tube by M. Monjurul Ehsan. (IUT MCE)[2]
- <https://confluence.cornell.edu/display/SIMULATION/FLUENT+-+Laminar+Pipe+Flow>
- http://www.researchgate.net/publication/236406851_Review_and_assessment_of_nanofluid_technology_for_transportation_and_other_applications