CFD ANALYSIS ON VARIOUS FINNED TUBE HEAT EXCHANGER

A thesis report submitted in partial fulfillment of the requirement for

The award of the degree of

BACHELOR OF ENGINEERING

IN

MECHANICAL ENGINEERING

Submitted by

T.DURGA PRASAD (314126520157)

K.KIRAN KUMAR (314126520200)

A.SANTOSH (314126520184)

EZRA DAVID (314126520169)

Under the guidance of

K. GOWRI SANKAR

Assistant professor

DEPARTMENT OF MECHANICAL ENGINEERING

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES (Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with 'A' grade)

Sangivalasa - 531162, Bheemunipatnam (Mandal), Visakhapatnam (Dist.), Andhra Pradesh, India. 2018

ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCES

(Permanently Affiliated to Andhra University, Approved by AICTE, Accredited by NBA & NAAC with 'A' grade)

Sangivalasa, Bheemunipatnam, Visakhapatnam, A.P.

PRAGNANAM BRAHMA

ANITS

CERTIFICATE

This is to certify that this project report entitled **"CFD ANALYSIS ON VARIOUS FINNED TUBE HEAT EXCHANGER"** has been carried out **by T.Durgaprasad (314126520157), K.Kiran kumar (314126520200), M.Santhoshkumar (314126520184), EzraDavid (314126520169)** Under the esteemed guidance of **K.Gowrisankar,** in partial fulfilment of the requirement for the "Bachelor of engineering" in mechanical engineering of Andhra University , Visakhapatnam.

APPROVED BY:
Prof.B.NAGARAJU
Head of the Department
Department of Mechanical Engineering
ANITS
Sangivalasa
Visakhapatnam.

PROJECT GUIDE:
K.Gowri sankar
Assistant Professor
Department of Mechanical Engineering
ANITS
Sangivalasa
Visakhapatnam.

Signature of HOD

PROFESSOR & HEAD
Department of Mechanical Engineering
ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY & SCIENCE
Sangivalasa 531 162 VISAKHAPATNAM Dist. A.P.

Signature of Internal Guide

THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINERS

INTERNAL EXAMINER:

Dr. B. Naga Raju
M.Tech,M.E.,Ph.d
Professor & HOD
Dept of Mechanical Engineering
ANITS, Sangivalasa,
Visakhapatnam-531 162.

EXTERNAL EXAMINER:

ACKNOWLEDGEMENT

On the submission of our project report entitled "CFD ANALYSIS ON VARIOUS FINNED TUBE HEAT EXCHANGER" we would like to give our heartiest thanks and gratitude to K.Gowri sankar, Assistant Professor, Department of Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences, for his continuous motivation and constant support and guidance throughout the past year.

We are very thankful to Prof.T.V.HanumanthaRao, Principal and Prof.B.Nagaraju, Head of the Department, Mechanical Engineering, Anil Neerukonda Institute of Technology & Sciences for their valuable support and facilities.

We would like to thank the technical staff of fluid machinery lab for their continuous cooperation and their guidance in helping us to understand the technical details of hydraulic machines in the lab.

Finally, we would like to convey our thanks to everyone, who have contributed directly or indirectly for the completion of this project work.

T.DURGA PRASAD	**(314126520157)**
K.KIRAN KUMAR	**(314126520200)**
A.SANTOSH	**(314126520184)**
EZRA DAVID	**(314126520169)**

ABSTRACT

Improving the radiator efficiency has become a wide research interest in the recent past. Radiator water tube design has also undergone drastic changes for improving the efficiency of the cooling of engine. In this regard radiator water tubes with Multi Lead Riffle have gained more prominence due to its ability to avoid heat loss at walls due to film condensation. So the aim of the present study is to enhance the heat transfer rate of Multi Lead Riffle boiler tubes further by using inserts.

In this study three different tubes models were considered like plain boiler tubes, Multi Lead Rifle boiler tubes, boiler tubes with longitudinal fins. Numerical analysis has been carried out on this tubes using Altair Acusolve. The results are compared with plain tubes. The results showed a rise in temperatures for longitudinal fins and Multi Lead tubes. A highest temperature of xxx^{o}K has obtained at the outlet for full lead tubes . On average it is found that there is a xx% decrease in the temperatures outlet by using full lead tubes than MLR.

Table of Contents

LIST OF FIGURES

LIST OF CONTOURS

LIST OF GRAPHS

CHAPTER 1

INTRODUCTION

1 INTRODUCTION

1.1 Modes of heat transfer

Heat is a form of energy which transfers between bodies which are kept under thermal interactions. When a temperature difference occurs between two bodies or a body with its surroundings, heat transfer occurs.

Heat transfer occurs in three modes. Three modes of heat transfer are described below.

- Conduction
- Convection and
- Radiation

1.1.1 Conduction:

In Conduction, heat transfer takes place due to a temperature difference in a body or between bodies in thermal contact, without mixing of mass. The rate of heat transfer through conduction is governed by the Fourier's law of heat conduction.

Q= -KA(dT/dX)

Where, 'Q' is the heat flow rate by conduction

'K' is the thermal conductivity of bodymaterial

'A' is the cross-sectional area normal to direction of heat flow and

'dT/dx' is the temperature gradient of the section.

1.1.2 Radiation:

In radiation, heat is transferred in the form of radiant energy or wave motion from one body to another body. No medium for radiation to occur. The rate of heat radiation that can be emitted by a surface at a thermodynamic temperature is based on Stefan-Boltzmann law.

$$Q = \sigma T^4$$

Where 'T' is the absolute temperature of surface

'σ' is the Stefan-Boltzmann constant.

1.1.3 Convection:

In convection, heat is transferred to a moving fluid at the surface over which it flows by combined molecular diffusion and bulk flow. Convection involves conduction and fluid flow. The rate of convective heat transfer is governed by the Newton’s law of cooling.

$$\mathbf{Q = hA}(T_s - T_a)$$

Where, Q is the heat transferred per unit time, A is the area of the object, h is the heat transfer coefficient, T_s is the object's surface temperature and T_a is the fluid temperature.

The convective heat transfer coefficient is dependent upon the physical properties of the fluid and the physical situation. Values of h have been measured and tabulated for commonly encountered fluids and flow situations.

1.1.3.1 Different types of convection heat transfer :

Two types of convective heat transfer may be distinguished:

- **Free or natural convection**: when fluid motion is caused by buoyancy forces that result from the density variations due to variations of thermal ±temperature in the fluid. In the absence of an internal source, when the fluid is in contact with a hot surface, its molecules separate and scatter, causing the fluid to be less dense. As a consequence, the fluid is displaced while the cooler fluid gets denser and the fluid sinks. Thus, the hotter volume transfers heat towards the cooler volume of that fluid. Familiar examples are the upward flow of air due to a fire or hot object and the circulation of water in a pot that is heated from below.

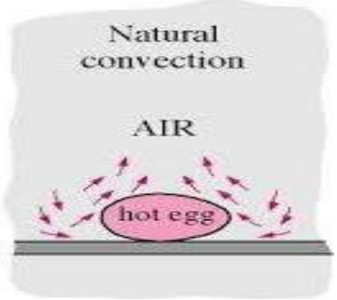


fig 1.1 Natural convection

- **Forced convection**: when a fluid is forced to flow over the surface by an internal source such as fans, by stirring, and pumps, creating an artificially induced convection current.

Internal and external flow can also classify convection. Internal flow occurs when a fluid is enclosed by a solid boundary such when flowing through a pipe. An external flow occurs when a fluid extends indefinitely without encountering a solid surface. Both of these types of convection, either natural or forced, can be internal or external because they are independent of each other. The bulk temperature, or the average fluid temperature, is a convenient reference point for evaluating properties related to convective heat transfer, particularly in applications related to flow in pipes and duct.

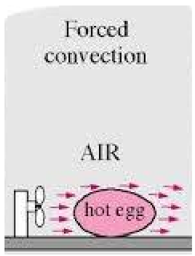


Fig 1.2 Forced convection

Further classification can be made depending on the smoothness and undulations of the solid surfaces. Not all surfaces are smooth, though a bulk of the available information deals with smooth surfaces. Wavy irregular surfaces are commonly encountered in heat transfer devices which include solar collectors, regenerative heat exchangers and underground energy storage systems. They have a significant role to play in the heat transfer processes in these applications. Since they bring in an added complexity due to the undulations in the surfaces, they need to be tackled with mathematical finesse through elegant simplification techniques. Also they do affect the flow and heat transfer characteristics, thereby behaving differently from straight smooth surfaces.

For a visual experience of natural convection, a glass filled with hot water and some red food dye may be placed inside a fish tank with cold, clear water. The convection currents of the red liquid may be seen to rise and fall in different regions, then eventually settle, illustrating the process as heat gradients are dissipated.

Essentially two types of fluid flow are characterized.

a.) Laminar Flow
b.) Turbulent Flow

a.) Laminar Flow:

The fluid particles flow in a flat or curved un-mixing layers and follow a smooth continuous path. The movements of fluid are well defined and the fluid particles retain their relative positions at successive cross section of the flow passage. There is no transverse displacement of particle that remains in an orderly sequence in each layer.

b.) Turbulent Flow:

The movement of fluid particles is irregular, unpredictable and erratic paths. The stream lines are inter winded and their change in position from instant to instant.

The variations in velocity components are superimposed on the main flow, and the velocity of individual fluid elements fluctuate both along the direction of flow and perpendicular to it.

1.2 Enhancement Techniques:

Heat transfer enhancement refers to the development of thermo hydraulic. The techniques are classified into three types and they are as follows:

1.2.1 Active Techniques: which require external power

i. Mechanical aids
ii. Surface vibration
iii. Fluid vibration
iv. Electrostatic fields
v. Injection
vi. Suction
vii. Jet impingement

1.2.2 Passive Techniques: without external power.

i. Treated surfaces
ii. Rough surfaces
iii. Extended surfaces
iv. Displaced enhancement devices
v. Swirl flow devices
vi. Coiled tubes
vii. Surface tension tubes
viii. Additives for liquids
ix. Additives for gases

1.2.1 Active techniques: These techniques are more complex from the use and design point of view , as the method requires some external power input to cause the desired flow modification and improvement in the rate of heat transfer .It finds limited application because of the need of external power in many practical applications. In comparison to the passive techniques, these techniques have not shown much potential as it is difficult to provide external power input in many cases. Various active techniques are as follows:

1. Mechanical aids: Mechanical aids include rotating tube exchangers and scrapped surface heat and mass exchangers. These devices stir the fluid by mechanical means or by rotating the surface.

2. Surface vibrations: These have been used primarily in single phase flows. A low or high frequency is applied to facilitate the vibrations which results in higher convective heat transfer coefficients.

3. Fluid vibration: Instead of applying vibrations to the surface, pulsations are created in the fluid itself. This kind of vibration enhancement technique is employed for single phase flows.

4. Jet impingement: This technique is applicable for both two phase and single phase heat transfer process. In this method, fluid is heated or cooled perpendicularly or obliquely to the heat transfer surface.

1.2.2. Passive techniques:

These techniques generally use surface or geometrically modifications to the flow channel by incorporating inserts or additional devices. They promote higher heat transfer coefficients by disturbing or altering the existing flow behaviour (except for extended surfaces) which also leads to increase in the pressure drop. In case of extended surfaces, effective heat transfer area on the side of the extended surface is increased. Passive techniques hold the advantage over the active techniques as they do not require any direct input of external power. Heat transfer augmentation by these techniques can be achieved by using. The follows

1. Rough surfaces: These surface modifications particularly create the disturbance in the various sub layer region. These techniques are applicable primarily in single phase turbulent flows. Small scale roughness or surface modification promotes turbulence in thr flow field near the wall region by disturbing the viscous laminar sub layer. This disturbance causes higher momentum and heat transfer. This small scale roughness has little effect in laminar flows, but is very effective in turbulent single phase flows. Now a days instead of natural roughness, artificial and structured roughness is used in most applications. Structured roughness can be integral to the surface. Wire coil type inserts can be inserted inside the tube to provide protuberances in the surface. In case of structured roughness almost an infinite number of geometric variations can be produced by machining, casting, or welding. Rough surfaces have been employed to enhance heat transfer in single phase flows both inside tubes and outside tubes. External rough surface can be created by grooving the heat transfer surface and can be used in double pipe and shell and tube bundles to enhance annulus or shell side heat transfer.

2. Extended surfaces: Plain fins are one of the earliest types of extended surfaces used extensively in many heat exchangers. Finned surfaces have become very popular now a days owing to their ability to disturb the flow field apart from increasing heat transfer area. Extended or finned surfaces increase the heat transfer area which could be very effective in case of fluids with low heat transfer coefficients. This technique includes finned tube for shell and tube exchangers, plate fins for compact heat exchanger and finned heat sinks for electronic cooling. Finned surfaces enhance heat transfer in natural or forced convection which can be used for cooling of electrical and electronic devices. The use of extended surfaces for cooling electronic devices is not restricted to the natural convection heat transfer regime but also can be used for forced convective heat transfer . Segmented or interrupted longitudinal fins as shown in fig 1.1 promote boundary layer separation of the fluids and disturb the whole bulk flow field inside circular tubes. Separation and restarting of the boundary layer increases the heat transfer rate. Plate fin or tube and plate fin type of compact heat exchangers, where the finned surfaces provide a very large surface area density, are used increasingly in many automotive, waste heat recovery, refrigeration and air conditioning, cryogenic, propulsion system and other heat recuperative applications . A variety of finned surfaces typically used include offset strip fins, louvered fins , perforated fins and wavy fins.

3. Swirl Flow devices: They produce swirl flow or secondary circulation on the axial flow in channel. Helical twisted tape, twisted ducts and various forms of altered are common examples of swirl flow devices. They can be used for both single phase and two-phase flows.

4. Coiled tubes: In these devices, secondary flows or vortices are generated due to curvature of the coils which promotes higher heat transfer coefficient in single phase flows and in most regions of boiling. This leads to relatively more compact heat exchangers. A coiled or curved tube, as shown in fig causes secondary flows due to continuous change in the bulk velocity vector at the curve surface of the duct. Coiled tubes are used in domestic water heaters, chemical process reactors, solar heating system, industrial and marine boilers, kidney dialysis devices and blood oxygenators. Secondary flows are generated due the centrifugal force on the fluid motion, induced because of the curvature of the coils. This curvature induced flow characteristics of the coiled tubes depends on the geometrical attributes like radius of curvature, helical number etc.

5. Displaced Enhancement Devices: Displaced enhancement devices displace the fluid elements from the core of the channel to heated or cooled surfaces and vice versa. Displaced enhancement devices include inserts like static mixer elements (e.g. kenics, sulzer) , metallic mesh and discs, wire matrix inserts, rings or balls. Different types of conical ring inserts are used in circular tubes. These inserts do not alter heat transfer surface and provide a lot of scope for inter-mixing of the fluid particles. Disks promote higher heat transfer and moderate increase in friction factor whereas friction factor is very high for rings and round balls. Most of the devices are suitable for laminar flow only. The main objective behind the use of static mixers is to increase the fluid mixing, so its application is limited to chemical processes with heat transfer only.

6. Additives for Liquids: Pressure drop in the tube flow is a consequence of the frictional losses with the solid surface. These frictional losses occurs because of the drag force of the fluid. This technique is basically concerned with reducing the drag coefficient using some additives to the fluid in single phase flows. Additives when added to the fluids are found to have operational benefits by lowering the frictional losses. These operational benefits could be fixed pressure or pumping costs. Polymeric additives induce a visco elastic character to the solution which promotes secondary circulation in the bulk flows. These secondary flows have significant effect on the heat transfer coefficient. Some soluble polymeric additives in water have shear thinning effect on the solutions, which lead to a significant reduction in frictional loss as well as a modest increase in the heat transfer coefficient. Some of the additives used are polystyrene spheres suspension in oil and injection of gas bubbles.

1.2.3 Compound techniques:

Any two or more of these techniques (passive and/or active) may be employed simultaneously to obtain enhancement in heat transfer that is greater than that produced by any of those techniques separately. This simultaneous utilization is termed as compound enhancement

Examples of compound techniques,

1) Rough tube wall with twisted pipe
2) Rough cylinder with acoustic vibrations
3) Internally finned tube with twisted tape insert

In the present study divergent nozzles are used as inserts include a circular tube. They are put to enhance heat transfer rate without increasing the surface area and this come under a passive technique.

Among these techniques, passively enhanced tubes are relatively easy to manufacture, cost effective for many applications where as active techniques, such as vibrating tubes are costly and complex**.**

The heat transfer rate can be improved by introducing a disturbance in the fluid flow there by breaking the viscous and thermal boundary layer. In this process pumping power may increase significantly and ultimately the pumping cost becomes high.

Therefore to achieve a desired heat transfer rate in an existing heat exchanger at economic pumping power, several techniques have been proposed in recent years.

The present experiment is to investigate the enhancement in heat transfer rate in a tube due to inclusion of the inserts.

These tube inserts are enhanced convective heat transfer by establishing one or more combinations of the following conditions which enhance the heat transfer.

1. Continuously development of the boundary layer of the flow and increase the flow degree of turbulence.
2. Continuously increase the heat transfer area.
3. Continuously creating secondary flow.

Heat transfer enhancement is usually accompanied by increased pressure drop and therefore higher pumping power. By forcing the fluids through the tubes at higher velocities, the heat transfer can be increased, but this higher velocity results in larger pressure drop and thus larger pumping costs. So there must be a compromise between enhancement of heat transfer and pumping costs.

1.3 Application of forced convection:

- Computer case cooling
- Cooling/heating system design
- Electric fan simulation
- Fan- or water-cooled central processing unit (CPU) design
- Heat exchanger simulation
- Heat removal
- Heat sensitivity studies
- Heat sink simulation
- Printed Circuit Board (PCB) simulation
- Thermal optimization

1.4 Radiator:

Radiators are heat exchangers used to transfer thermal energy from one medium to another for the purpose of cooling and heating. The majority of radiators are constructed to function in automobiles, buildings, and electronics. The radiator is always a source of heat to its environment, although this may be for either the purpose of heating this environment, or for cooling the fluid or coolant supplied to it, as for engine cooling. Despite the name, most radiators transfer the bulk of their heat via convection instead of thermal radiation. Spacecraft radiators necessarily must use radiation only to reject heat.

FIG 1.3 Water-air convective cooling radiator.

1.4.1 Radiation and convention

Heat transfer from a radiator occurs by all the usual mechanisms: thermal radiation, convection into flowing air or liquid and conduction into the air or liquid. A radiator may even transfer heat by phase change. For example, drying a pair of stocks.In practice, the term “radiator” refers to any of a number of devices in which a liquid circulates through exposed pipes (often with fins or other means of increasing the surface area). The term “convection” refers to a class of devices in which the source of heat is not directly exposed .

Radiators are used for cooling internal combustion engines, mainly in automobiles but also in piston-engined aircraft, railway locomotives, motorcycles, stationary generating plants and other places where such engines are used.

To cool down the engine, a coolant is passed through the engine block, where it absorbs heat from the engine. The hot coolant is then fed into the inlet tank of the radiator (located either on the top of the radiator, or along one side), from which it is distributed across the radiator core through tubes to another tank on the opposite end of the radiator. As the coolant passes through the radiator tubes on its way to the opposite tank, it transfers much of its heat to the tubes which, in turn, transfer the heat to the fins that are lodged between each row of tubes. The fins then release the heat to the ambient air. Fins are used to greatly increase the contact surface of the tubes to the air, thus increasing the exchange efficiency. The cooled coolant is fed back to the engine, and the cycle repeats. Normally, the radiator does not reduce the temperature of the coolant back to ambient air temperature, but it is still sufficiently cooled to keep the engine from overheating.

This coolant is usually water-based, with the addition of glycols to prevent freezing and other additives to limit corrosion, erosion and cavitation. However, the coolant may also be an oil. The first engines used thermosiphons to circulate the coolant; today, however, all but the smallest engines use pumps.

1.5 HEAT EXCHANGER

A **heat exchanger** is a device which transfers heat from one medium to another, a Hydraulic Oil Cooler or example will remove heat from hot oil by using cold water or air. Alternatively a Swimming Pool Heat Exchanger uses hot water from a boiler or solar heated water circuit to heat the pool water. Heat is transferred by conduction through the exchanger materials which separate the mediums being used. A shell and tube heat exchanger passes fluids through and over tubes, where as an air cooled heat exchanger passes cool air through a core of fins to cool a liquid.

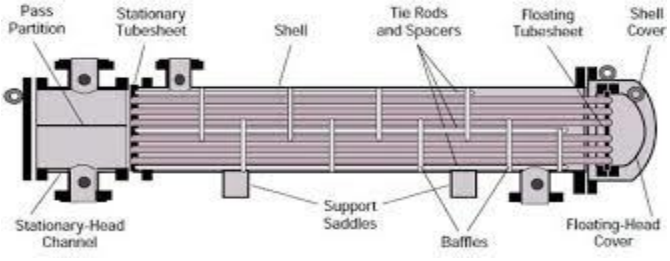


fig 1.4 Heat exchanger.

1.5.1 DIFFERENT TYPES OF HEAT EXCHANGER:

- Shell and tube heat exchanger
- Plate heat exchangers
- Plate and shell heat exchanger
- Adiabatic wheel heat exchanger
- Plate fin heat exchanger
- Pillow plate heat exchanger
- Fluid heat exchangers
- Waste heat recovery units
- Dynamic scraped surface heat exchanger
- Phase-change heat exchangers
- Direct contact heat exchangers
- Microchannel heat exchangers
- Finned tube heat exchangers

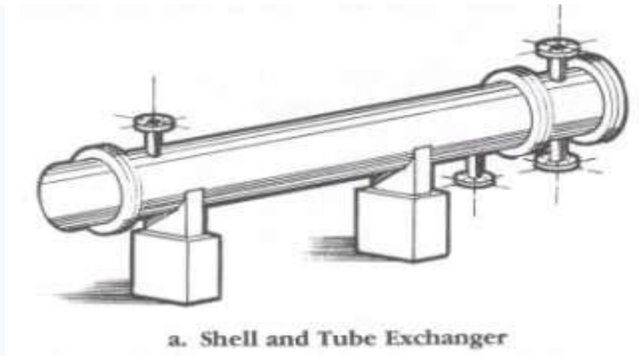
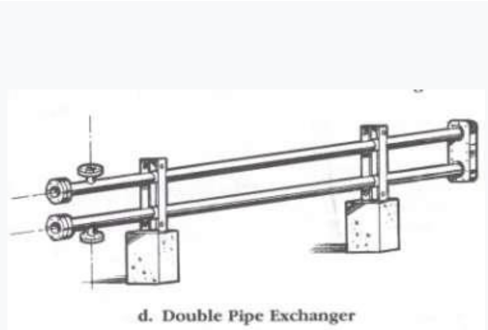


Fig 1.5 shell and tube heat exchanger

Fig1.6 plate exchanger **fig1.7 spiral exchanger**



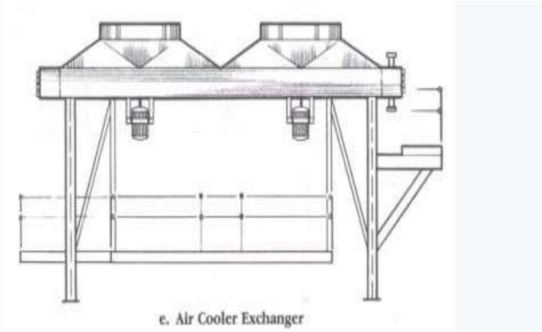


Fig 1.8 double pipe exchanger **fig 1.9 Air cooler exchanger**

1. Finned tube heat exchanger :

Finned tube heat exchanger for heat transfer between air, gas and liquids or steam. Heat exchanger with finned heating surfaces, so-called finned tube heat exchanger, offer the possibility of heat transfer between gases and liquids significantly space-saving and is more efficient to implement than it is possible with straight tubes. Maxxtec finned tube heat exchangers are designed to transfer heat from clean air and gases with high efficiency on liquids or vapors, and vice versa. In this way the media can be heated, cooled or condensed, in a closely space. Finned tube heat exchangers can be used for different applications and in a variety of designs. Maxxtec offers various heat exchangers for economizer, air heater, heaters for gases, air heaters or capacitors in fin-tube design.

FIG 1.10 Finned tube heat exchanger

1.6 Types of Finned Tubes

1. Longitudinal Fins

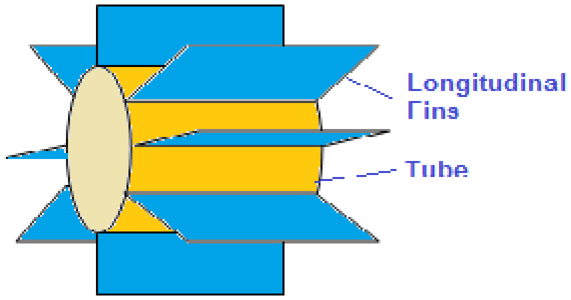


Fig 1.11 Longitudinally finned tube in heat exchanger

Longitudinal fins on a tube are best suited for applications where the flow outside the tubes is expected to be streamlined along the tube length, for example double pipe heat exchangers with highly viscous fluid outside the finned tube.

Longitudinal fins on a tube run along the length of the tubes. The cross sectional shapes of longitudinal fins can be either flat or tapered. For different cross sectional geometries, various correlations are available in the literature to evaluate the heat transfer coefficients on outer side of the tubes.

2. Transverse Fins

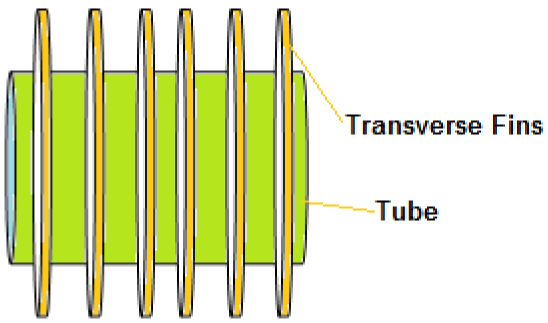


Figure 1.12 - Transversely finned heat exchanger tube.

Transverse fins are normally used for gas flows or turbulent flows and for cross flow type exchangers or shell and tube heat exchangers. For air coolers, tubes with transverse fins are best suited.

Transverse fins are hollow metal discs spaced from each other and fitted along the length of the finned tubes. The transverse fin discs can be flat or tapered. Heat transfer coefficients on the surface of the fin depend on the fin disc geometry and are available in the literature in the form of correlations.

1.7 Application examples for finned tube heat exchanger:

Finned tube heat exchangers are often used in power plants as an exhaust gas heat exchanger to increase the efficiency factor. Further applications in power plants are the preheating of combustion air as well as the condensation of exhaust steam from steam or ORC turbines.

In industrial dryers finned tube heat exchangers will be used for heating of air by hot water, steam or thermal oil in large quantities.

In many industrial production processes, such as for the air conditioning of buildings, finned tube heat exchangers are used as an air cooler for cooling down or re-cooling of liquids. Due to the problems with Legionella, the high consumption of fresh water, as well as the elaborate water treatment, closed cooling circuits with finned tube heat exchangers will be used instead of cooling towers with open water circuit.

Advantages of Maxxtec finned tube heat exchangers

- Robust construction of finned tube heat exchanger that can withstand contrarious operating conditions over a long period.
- Maximum transmission quality
- High condensation rate
- Wide application and temperature spectrum (range)
- Very good value for money
- Ideal for gas-liquid or gas-vapour heat transfer
- Available as stainless steel finned tube heat exchanger
- Highest reliability of operation through extensive quality inspection

1.8 **ACUSOLVE:**

Experienced CFD analysts are all too familiar with the challenges of getting a simulation to run in the majority of commercial solvers. Endless geometry simplification to construct block structured meshes, days spent wrestling with meshing software, and solution divergence due to poor quality meshes are common issues faced in the workflow of many solvers.

Altair's AcuSolve provides you with the tools you need to avoid the CFD pre-processing bottle neck and focus on exploring your CFD simulation results. AcuSolve's powerful solver technology provides you with the most robust solution in the CFD marketplace.

AcuSolve's proprietary numerical methods yield stable simulations and accurate results regardless of the quality and topology of the mesh elements. When combined with automated unstructured meshing, no CFD software gets you to your end goal sooner: analyzing results and exploring the physics of your application.

AcuSolve's single solver architecture and constantly evolving feature set provides a valuable tool to any company tasked with performing CFD analysis.

1.8.1 Benefits:

- **Robustness**

AcuSolve's powerful solver technology provides users with the most robust solution in the CFD marketplace. AcuSolve's proprietary numerical methods yield stable simulations and accurate results regardless of the quality and topology of the mesh elements. Pre-processing and meshing has historically been the bottleneck when performing industrial CFD. However, AcuSolve's solver technology, combined with automated unstructured meshing reduces the pre-processing burden and enables scientists and engineers to reach their end goal sooner.... analyzing results and exploring the physics of their application. AcuSolve achieves all of this without the need for users to perform countless runs to explore different solution procedures.

AcuSolve uses a single solver for all flow regimes with very few parameters to adjust. No tuning required....build your mesh and run it!

- **Speed**

AcuSolve produces results rapidly by solving the fully-coupled pressure/velocity equation system using industry leading, scientifically proven numerical techniques. This yields rapid linear and nonlinear convergence rates for both steady state and transient simulations. In addition to the efficiency of the numerical methods, AcuSolve was architected from the ground up to take full advantage of parallel compute platforms. All algorithms within the solver are designed for multi-core parallel clusters, using a hybrid distributed/shared-memory (MPI/OpenMP) parallel model that is completely automated within the solver's run script.

Message Passing Interface (MPI) is used to communicate between distributed-memory machines, and shared-memory data copy is used between subdomains of a single shared-memory machine. Two level domain decomposition is used to optimize the distribution of elements and nodes based on the shared and distributed memory message passing requirements of a given simulation. These optimizations ensure that message passing overhead is kept to a minimum.

- **Accuracy**

AcuSolve's numerical methods have been customized to deliver both speed and accuracy for the most demanding of CFD applications. One of the keys to its accuracy is the use of equal order interpolation for all variables. This results in second order spatial accuracy for all governing equations. This focus on accuracy further enhances the speed of the solver by enabling you to achieve high levels of accuracy with AcuSolve using far less mesh than what is required to achieve comparable results using other CFD solvers that are on the market. In addition to requiring less mesh to produce a comparable result, AcuSolve is able to retain its second order accuracy on all element topologies. You can confidently simplify your pre-processing by using tetrahedral elements to mesh your domain without sacrificing robustness or accuracy.

CHAPTER 2

LITERATURE REVIEW

2 LITERATURE REVIEW

- The unique characteristics of the boiler to obtain highest efficiency have drawn the attention of many researchers. Apart from active techniques some passive techniques has also been adopted by the researchers to improve the efficiency of the boiler. The passive methods includes usage of additives , inserts etc..,. In this regard T C Mohan kumar, Nice Thomachan 2013 [1] explained about the merits of multi lead rifled [MLR] tubes in vertical water tube boiler using CFD tool. Heat transfer enhancement of MLR tubes was mainly taken into consideration. Performance of multi lead rifled tube was studied by varying its influencing geometrical parameter like number of rifling, height of rifling, length of pitch of rifling for a particular length. The heat transfer analysis was done at operating conditions of an actual coal fired water tube boiler situated at Apollo Tyres LTD, Chalakudy, India for saturated process steam production. The results showed that the heat transfer increased when compared with existing inner plane wall.
- Vincent.H.Wilson, M.Hajee Mohamed [2] studied about the Super Heater Tube Portion which to be affected by High Flue Gas Temperature. The Identified Super Heater Tube Portion to be shielded with SS Plates. The Shield Plate will safe guard the Super Heater Tube where the weak portion has been find out. Providing SS plate Transfer the Heat safe manner to Super Heater Tube. Due to this, the Super Heater Tube Leakage, can be avoided and there is maintenance of Super Heater Tubes, Improve the Power Production without any plant shutdown, subsequently will Improve the Power Plant Income.

- Ajay N. Ingale&Vivek C. Pathede [3] analyzed the tube leakage through CFD Modeling considering two temperature, one for steam flowing inside the SH tube and other for temperature of flue gases flowing over the SH Tube. Based on the temperature plots they found that the SH tube bend to be exposed to the high temperature region. This may lead to fatigue and cracks near the weld joints of the tube resulting in change in shape and cracks near the corners.
- Prashant Kumkale [4] has carried out investigation on super heater tubes through CFD analysis with various temperature plots, and found that velocity is high at SH tube bend and pressure drop more at the tube bend.
- Aditya Kumar Pandey [5] conducted flue gas flow distribution on low temperature super heater (LTSH) tubes through CFD analysis and found that flue gas flow changes direction as it flows from extended pass to LTSH tube bundles. Extended pass is tapered. Due to this no uniform velocity distribution can occur at the top of LTSH.
- Saripally et al. [6] conducted a simulation of thermal flow in an industrial boiler using a CFD package. Computer simulation has been employed to understand the thermal flow in the boiler to resolve the operational problem and searchfor optimal solution. The combustion and thermal flow behavior inside the boiler is studied to make the boiler moreefficient, less emissive and less prone to tube rupture. The study performs a detailed simulation of combustion and thermal flow behavior inside the industrial boiler. Due to excessive heating the rupture of super heater tubes may lead to boiler shutdown, increasing the expenses incurred. The CFD analysis provided fluid velocity, pressure, temperature, and species concentration throughout the

solution domain. During the analysis, the geometry of the system and boundary conditions such as inlet velocity and flow rate was changed to view their effect on thermal flow patterns or species concentration distribution.

- The turbulent flow air flow in a tube fitted with louvered inserts was investigated by Fan et al. [7]. The Nusselt number is increased by a 2.75-4.05 times than the smooth tube. The value of PEC range will be varied with 1.60-2.05.
- The heat transfer enhancement for turbulent flow by using double helical tape inserts are experimentally studied by Bhuiya et al. [8]. The tapes are consists of different helix angle ranging from 22000 to 51000 were used. The results indicate that the friction factor and Nusselt number were increased up to 170% to 305% respectively. The thermal performance was 15% for using double helical tape inserts at helix angle 9°. The heat flux was around 60% higher than the plain tube. The helix angle 9° obtained the best performance compared with 15°, 21° and 28°.

SUMMARY OF LITERATURE REVIEW

CHAPTER 3

MODELLING

3 MODELLING

3.1 CATIA HISTORY

CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Davot at that time customer of the CADAMsoftware to develop Dassault's Mirage fighter jet. It was later adopted by the aerospace, automotive, shipbuilding, and other industries.

Initially named CATI (conception assistéetridimensionnelle interactive – French for interactiveaided three-dimensional design), it was renamed CATIA in 1981 when Dassault created a subsidiary to develop and sell the software and signed a non-exclusive distribution agreement with IBM .

3.2 SCOPE OF CATIA

Commonly referred to as a 3DProduct Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM). CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical, fluid and electronic systems design, mechanical engineering and systems engineering.

CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

Case1: PIPE WITH OUT FINS:

1.a) AIR DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre rectangle of 500mm × 120 mm and centre circle of 12mm and then click on exit workbench
- Click on pad definition for the above sketch and given the length as 500mm
- Create a offset plane from xy plane and the distance is 20mm
- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pocket definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction number of objects as 24.
- save the part as 1a_airdomain.

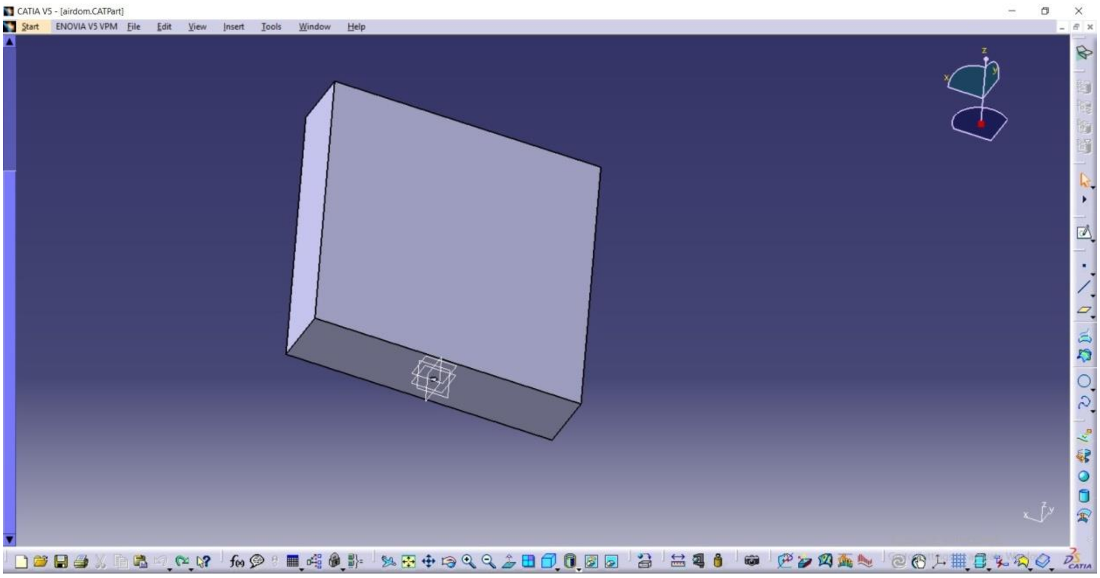


Fig 3.1a Air domain (pipe without fin)

1.b) PIPE DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→XY Plane → sketch
- Create centre circles of 12mm and 10mm, then click on exit workbench
- Click on pad definition for the above sketch and given the length as 510mm as one limit and second limit as 10mm.
- Create a offset plane from xy plane and the distance is 20mm

- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pad definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction number of objects as 24.
- save the part as 1b_pipedomain.

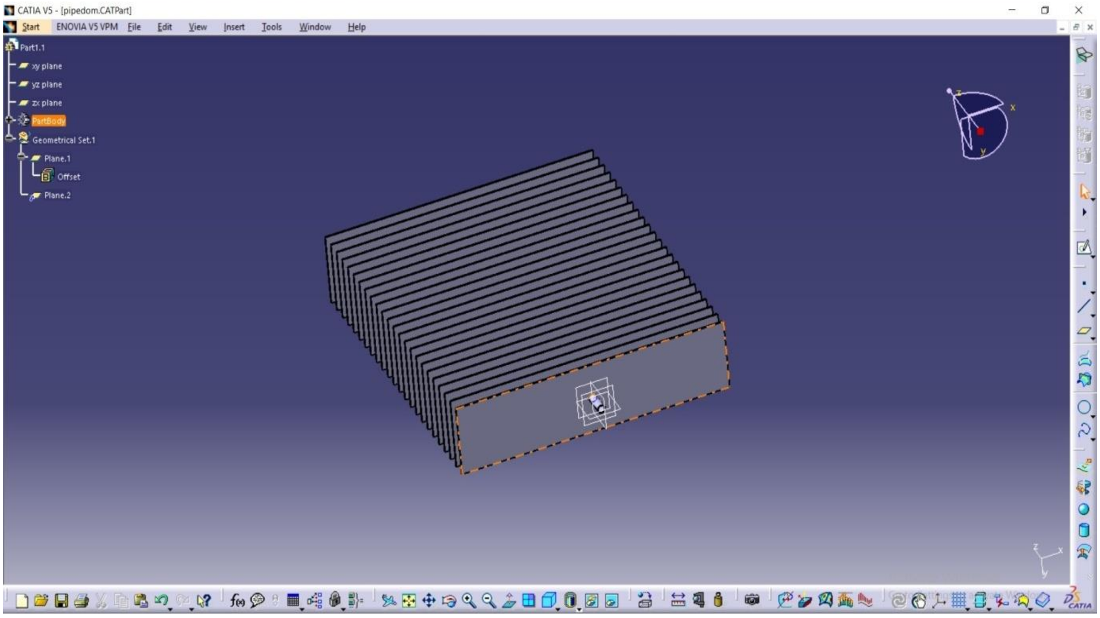


Fig 3.1b pipe domain(pipe without fin)

1.c) WATER DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre circle of 10mm and then click on exit workbench
- Click on pad definition for the above sketch and given the length first limit as 510mm and 10mm.
- save the part as 1c_waterdomain.

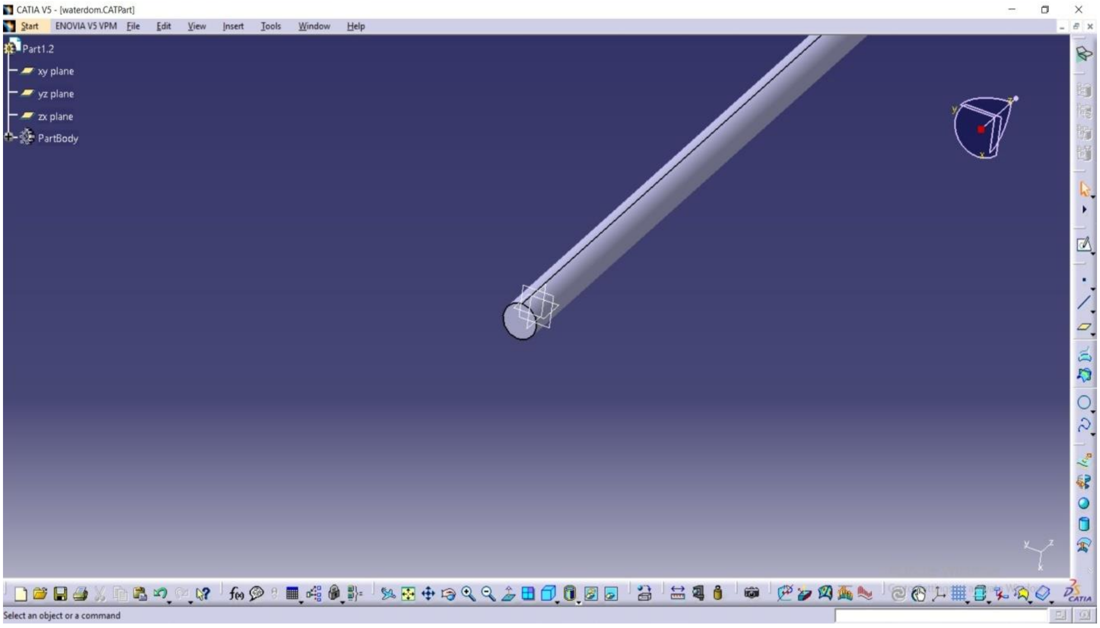


Fig 3.1c water domain(pipe without fin)

1.d) ASSEMBLY:

- Invoke CATIA → Start→mechanical design→Assembly Design
- Click on existing component and click on product in specification tree and browse the 1a_airdomain., 1b_pipedomain., 1c_waterdomain.
- By using axis coincidence and offset assembly constraints assemble final product
- Save the assembly file as 1withoutfins.product.

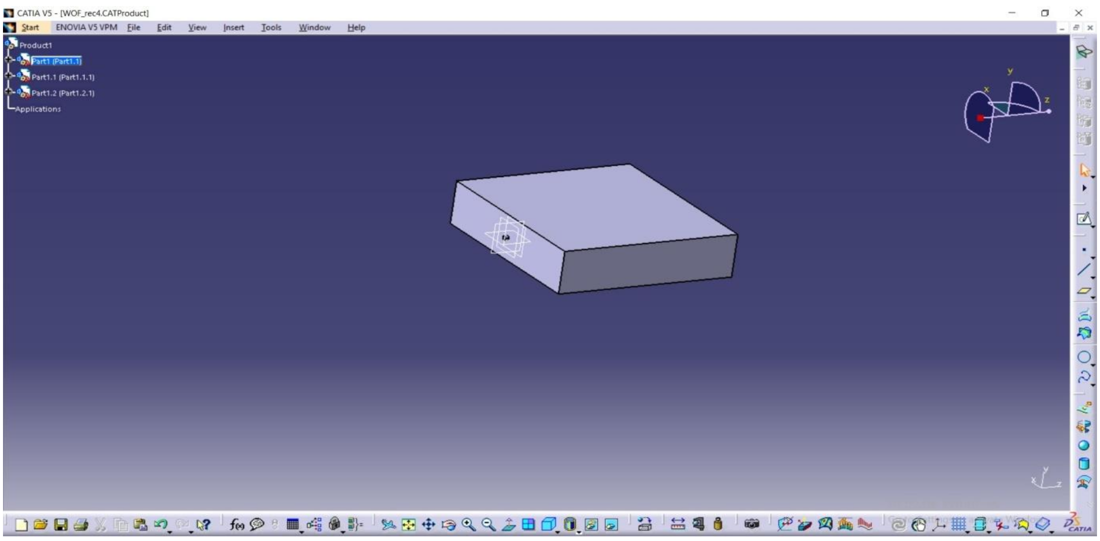


Fig 3.1d Assembly(pipe without fin)

Case2: PIPE WITH LONGITUIDINAL FINS:

2.a) AIR DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre rectangle of 500mm × 120 mm and centre circle of 12mm and then click on exit workbench
- Click on pad definition for the above sketch and given the length as 500mm
- Create a offset plane from xy plane and the distance is 20mm
- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pocket definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction and number of objects as 24.
- Create four rectangles of 1mm x 1mm at outer circle quadrant as base and click on exit workbench
- Click on pocket definition for the above sketch and given the depth as 500mm
- save the part as 2a_airdomain.

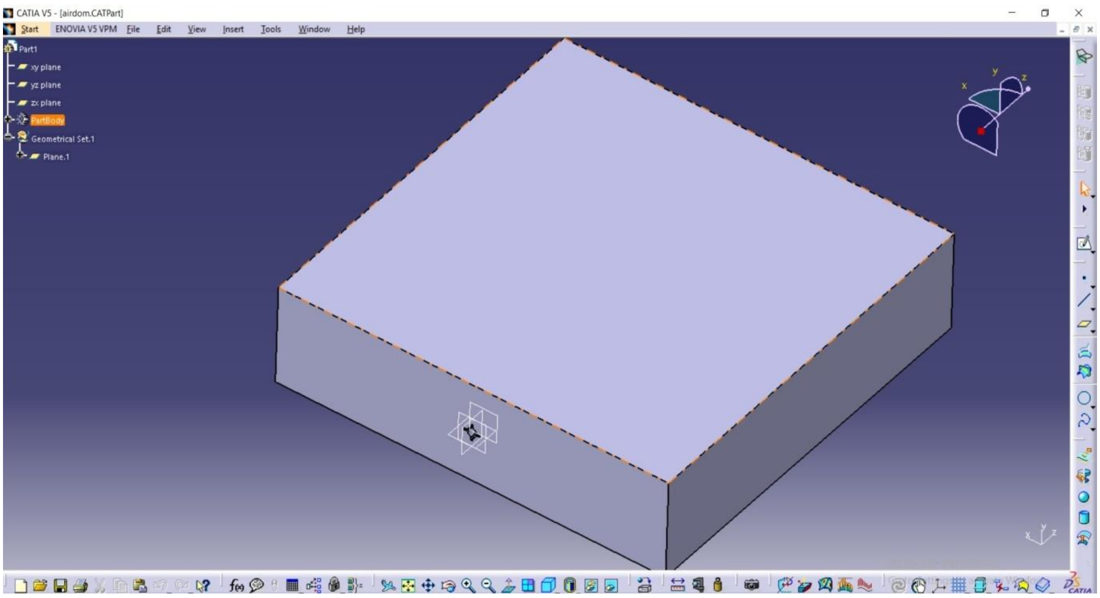


Fig 3.2a air domain(pipe without longitudinal fin)

2.b) PIPE DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→XY Plane → sketch
- Create centre circles of 12mm and 10mm, then click on exit workbench
- Click on pad definition for the above sketch and given the length as 510mm as one limit and second limit as 10mm.
- Create a offset plane from xy plane and the distance is 20mm
- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pad definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction number of objects as 24.
- Create four rectangles of 1mm x 1mm at outer circle quadrant as base and click on exit workbench
- Click on pad definition for the above sketch and given the depth first limit as 510mm and 10mm as second limit.
- save the part as 2b_pipedomain.

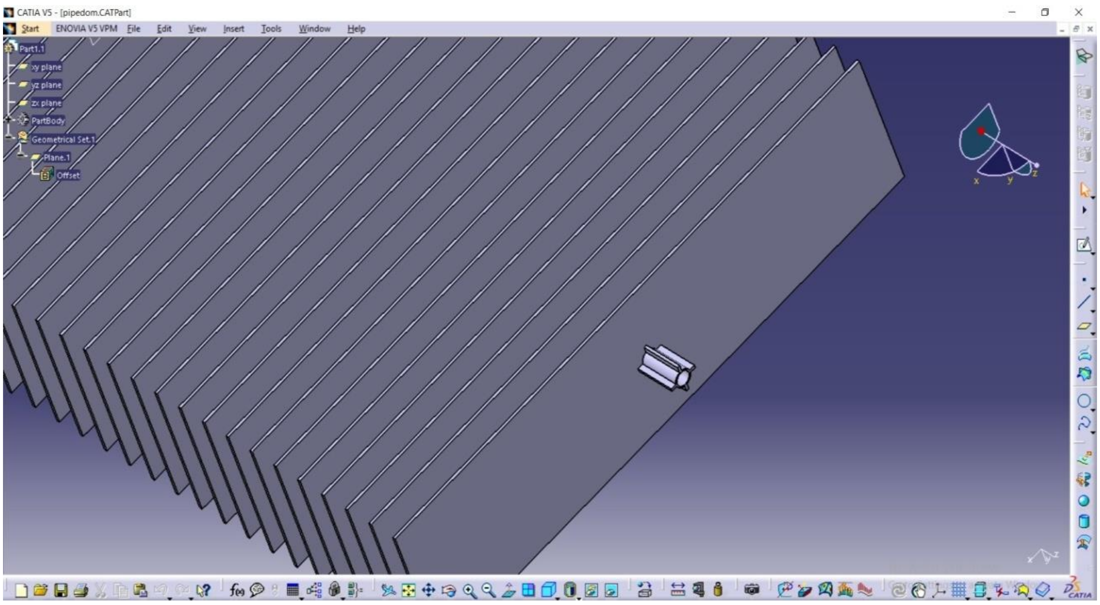


Fig 3.2b water domain(pipe without longitudinal fin)

2.c) WATER DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre circle of 10mm and then click on exit workbench

- Click on pad definition for the above sketch and given the length first limit as 510mm and 10mm.
- save the part as 2c_waterdomain.

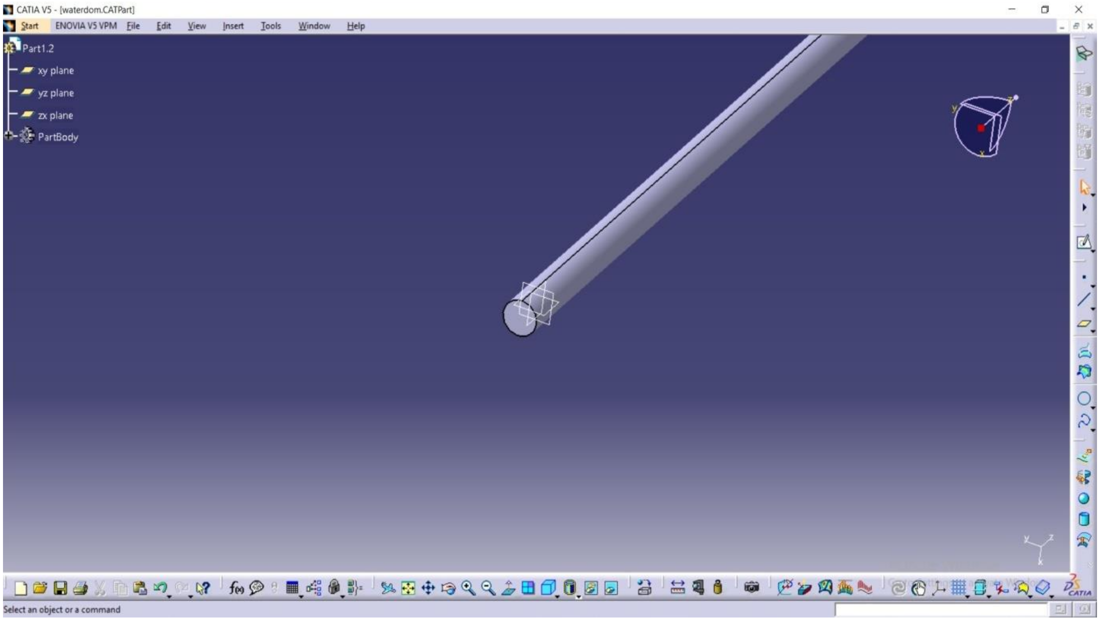


Fig 3.2c water domain(pipe without longitudinal fins)

2.d) ASSEMBLY:

- Invoke CATIA → Start→mechanical design→Assembly Design
- Click on existing component and click on product in specification tree and browse the 2a_airdomain., 2b_pipedomain., 2c_waterdomain.
- By using axis coincidence and offset assembly constraints assemble final product
- Save the assembly file as 2longitudinalfins.product.

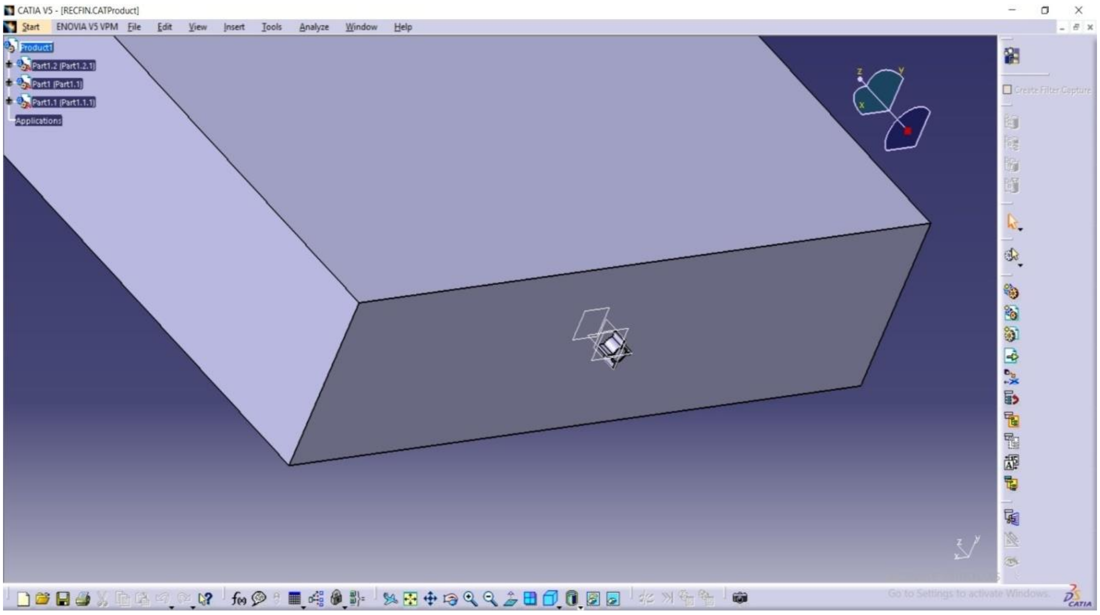


Fig 3.2d Assembly(pipe without longitudinal fin)

Case3: PIPE WITH MULTI LED RIFFLES:

3.a) AIR DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre rectangle of 500mm × 120 mm and centre circle of 12mm and then click on exit workbench
- Click on pad definition for the above sketch and given the length as 500mm
- Create a offset plane from xy plane and the distance is 20mm
- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pocket definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction number of objects as 24.
- save the part as 3a_airdomain.

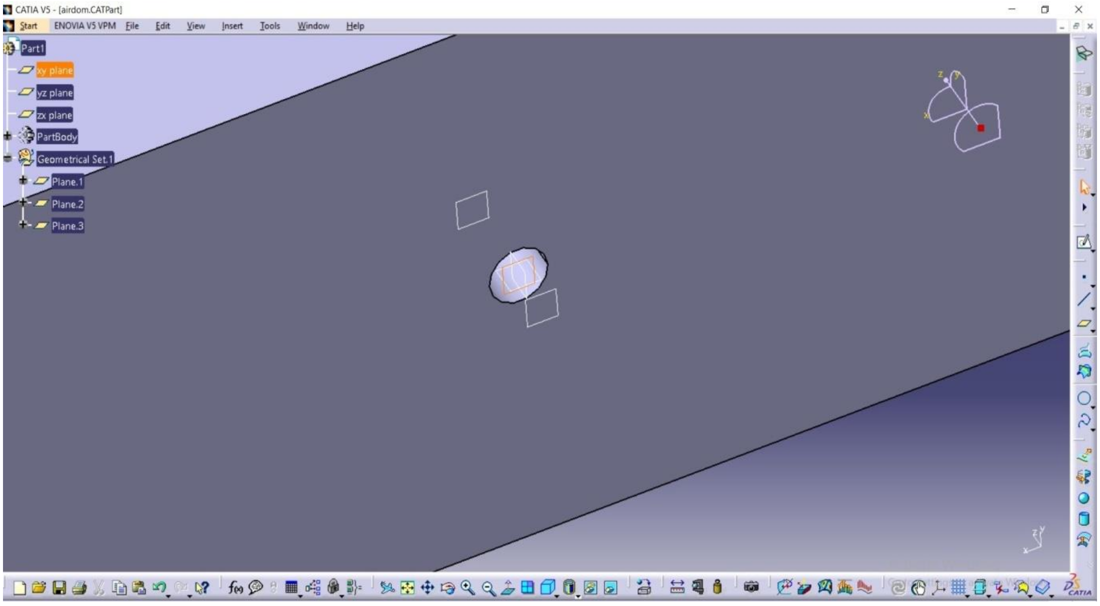


Fig 3.3a Air domain(MLR tube)

3.b) PIPE DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→XY Plane → sketch
- Create centre circles of 12mm and 10mm, then click on exit workbench
- Click on pad definition for the above sketch and given the length as 510mm as one limit and second limit as 10mm.
- Create a offset plane from xy plane and the distance is 20mm
- Create a centre rectangle of 400mm × 100 mm and click on exit workbench
- Click on pad definition for the above sketch and given the depth as 2mm
- Click on rectangular pattern and give this pocket as an object to duplicate the pocket along the z direction number of objects as 24.
- Create four rectangles of 1mm x 1mm at inside of inner circle quadrant as base and click on exit workbench
- Create 3d point using co-ordinates of x=5, y=0 and z=-10.
- Click on start then mechanical design go to wire frame and surface design
- Click on Helix curve definition give the axis as Z-axis and pitch as 10mmand height as 510mm.
- Again go to start then mechanical design and part design, click on rib give the four rectangles as profile and helix as centre curve.

- save the part as 3b_pipedomain.

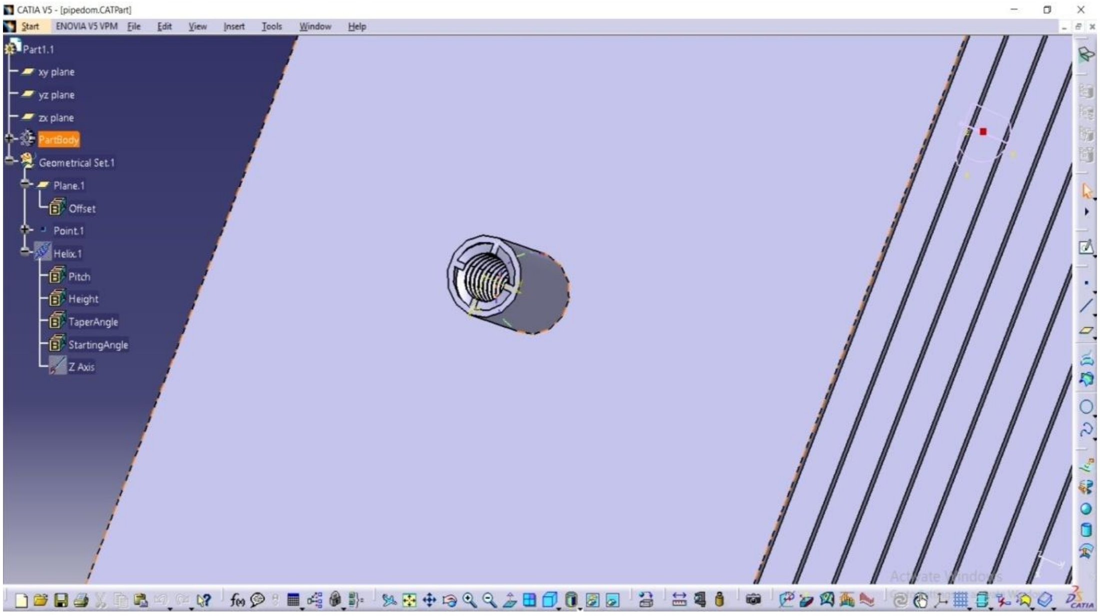
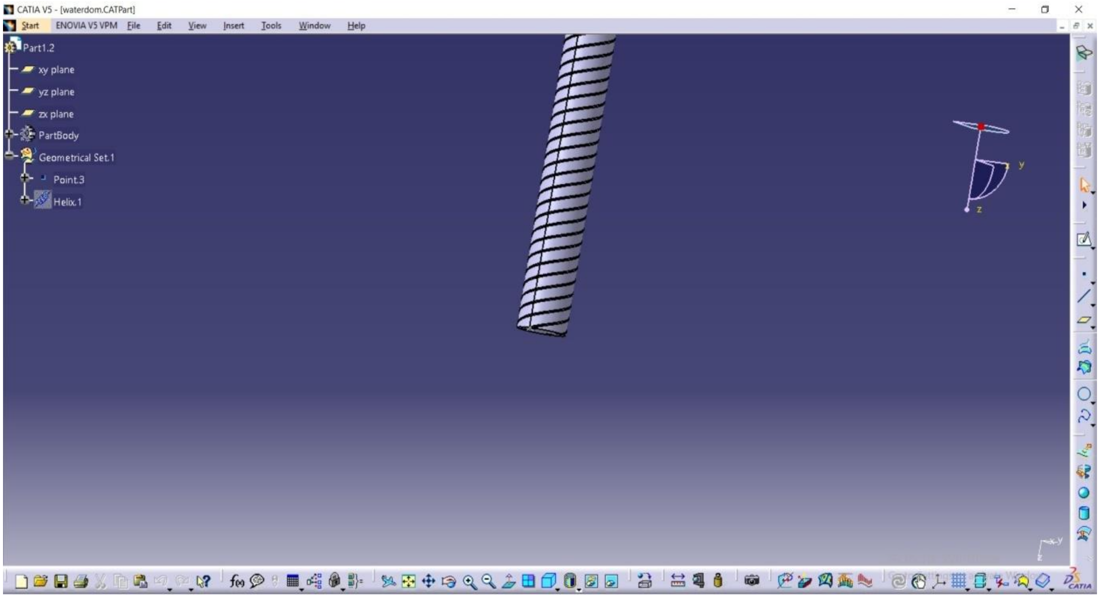


Fig 3.3b pipe domain(MLR tube)

3.c) WATER DOMAIN:

- Invoke CATIA → Start→mechanical design→part design→sketch→
- Create centre circle of 10mm and then click on exit workbench
- Click on pad definition for the above sketch and given the length first limit as 510mm and 10mm.
- Create four rectangles of 1mm x 1mm at inside of inner circle quadrant as base and click on exit workbench
- Create 3d point using co-ordinates of x=5, y=0 and z=-10.
- Click on start then mechanical design go to wire frame and surface design
- Click on Helix curve definition give the axis as Z-axis and pitch as 10mmand height as 510mm.
- Again go to start then mechanical design and part design, click on slot give the four rectangles as profile and helix as centre curve.
- save the part as 3c_waterdomain.



3.3c water domain(MLR tube)

3.d) ASSEMBLY:

- Invoke CATIA → Start→mechanical design→Assembly Design
- Click on existing component and click on product in specification tree and browse the 3a_airdomain., 3b_pipedomain., 3c_waterdomain.
- By using axis coincidence and offset assembly constraints assemble final product
- Save the assembly file as 3MLRfins.product.

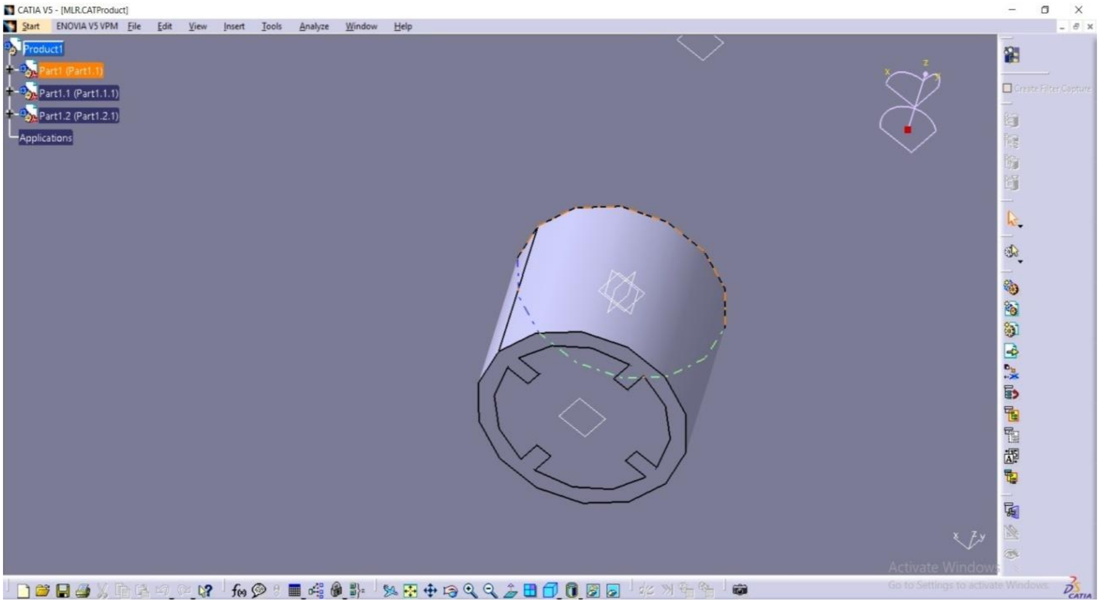


Fig 3.3d Assembly(MLR tube)

CHAPTER 4

COMPUTATIONAL FLUID DYNAMICS

4 COMPUTATIONAL FLUID DYNAMICS

4.1 CFD

CFD is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high speed supercomputers, only approximate solutions can be achieved in many cases. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are an ongoing area of research.

4.2 Discretization Methods in CFD

There are three discretization methods in CFD:

1. Finite difference method (FDM)

2. Finite volume method (FVM)

3. Finite element method (FEM)

4.2.1 Finite Difference Method (FDM)

A finite difference method (FDM) discretization is based upon the differential form of the PDE to be solved. Each derivative is replaced with an approximate difference formula (that can generally be derived from a Taylor series expansion). The computational domain is usually divided into hexahedral cells (the grid), and the solution will be obtained at each nodal point. The FDM is easiest to understand when the physical grid is Cartesian, but through the use of curvilinear transforms the method can be extended to domains that are not easily represented by brick-shaped elements. The Discretization results in a system of equation of the variable at nodal points, and once a solution is found, then we have a discrete representation of the solution.

4.2.2 Finite Volume Method (FVM)

A finite volume method (FVM) discretization is based upon an integral form of the PDE to be solved (e.g. conservation of mass, momentum, or energy). The PDE is written in a form which can be solved for a given finite volume (or cell). The computational domain is discretized into finite volumes and then for every volume the governing equations are solved. The resulting system of equations usually involves fluxes of the conserved variable, and thus the calculation of fluxes is very important in FVM. The basic advantage of this method over FDM is it does not require the use of structured grids, and the effort to convert the given mesh in to structured numerical grid internally is completely avoided. As with FDM, the resulting approximate solution is a discrete, but the variables are typically placed at cell canters rather than at nodal points. This is not always true, as there are also face-cantered finite volume methods. In any case, the values of field variables at non-storage locations (e.g. vertices) are obtained using interpolation.

4.2.3 Finite Element Method (FEM)

A finite element method (FEM) discretization is based upon a piecewise representation of the solution in terms of specified basis functions. The computational domain is divided up into smaller domains (finite elements) and the solution in each element is constructed from the basic functions. The actual equations that are solved are typically obtained by restating the conservation equation in weak form: the field variables are written in terms of the basic functions; the equation is multiplied by appropriate test functions, and then integrated over an element. Since the FEM solution is in terms of specific basis functions, a great deal more is known about the solution than for either FDM or FVM. This can be a double-edged sword, as the choice of basic functions is very important and boundary conditions may be more difficult to formulate. Again, a system of equations is obtained (usually for nodal values) that must be solved to obtain a solution.

Comparison of the three methods is difficult, primarily due to the many variations of all three methods. FVM and FDM provide discrete solutions, while FEM provides a continuous (up to a point) solution. FVM and FDM are generally considered easier to program than FEM, but opinions vary on this point. FVM are generally expected to provide better conservation properties, but opinions vary on this point also.

4.3 WORKING OF CFD

CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

1. Pre-processing.
2. Solver
3. Post-processing.

4.3.1 Pre-Processing

This is the first step in building and analyzing a flow model. Pre-processor consist of input of a flow problem by means of an operator –friendly interface and subsequent transformation of this input into form of suitable for the use by the solver. The user activities at the Pre-processing stage involve:

- Definition of the geometry of the region: The computational domain.
- Grid generation the subdivision of the domain into a number of smaller, non overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modelled).
- Definition of fluid properties
- Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary. The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

4.3.2 Solver

The CFD solver does the flow calculations and produces the results. FLUENT, FloWizard, FIDAP, CFX and POLYFLOW are some of the types of solvers.

FLUENT is used in most industries. FloWizard is the first general-purpose rapid flow modelling tool for design and process engineers built by Fluent. POLYFLOW (and FIDAP) are also used in a wide range of fields, with emphasis on the materials processing industries. FLUENT and CFX two solvers were developed independently by ANSYS and have a number of things in common, but they also have some significant differences. Both are control-volume based for high accuracy and rely heavily on a pressure-based solution technique for broad applicability. They differ mainly in the way they integrate the fluid flow equations and in their equation solution strategies. The CFX solver uses finite elements (cell vertex numerics), similar to those used in mechanical analysis, to discretize the domain. In contrast, the FLUENT solver uses finite volumes (cell centred numerics). CFX software focuses on one approach to solve the governing equations of motion (coupled algebraic multi grid), while the FLUENT product offers several solution approaches (density, segregated- and coupled-pressure-based methods)

The FLUENT CFD code has extensive interactivity, so we can make changes to the analysis at any time during the process. This saves time and enables to refine designs more efficiently. Graphical user interface (GUI) is intuitive, which helps to shorten the learning curve and make the modelling process faster. In addition, FLUENT's adaptive and dynamic mesh capability is unique and works with a wide range of physical models. This capability makes it possible and simple to model complex moving objects in relation to flow. This solver provides the broadest range of rigorous physical models that have been validated against industrial scale applications, so we can accurately simulate real-world conditions, including multiphase flows, reacting flows, rotating equipment, moving and deforming objects, turbulence, radiation, acoustics and dynamic meshing. The FLUENT solver has repeatedly proven to be fast and reliable for a wide range of CFD applications. The speed to solution is faster because suite of software enables us to stay within one interface from geometry building through the solution process, to post-processing and final output.

The numerical solution of Navier–Stokes equations in CFD codes usually implies a discretization method: it means that derivatives in partial differential equations are approximated by algebraic expressions which can be alternatively obtained by means of the finite-difference or the finite-element method. Otherwise, in a way that is completely different from the previous one, the discretization equations can be derived from the integral form of the conservation equations: this approach, known as the finite volume method, is implemented in FLUENT, because of its adaptability to a wide variety of grid structures. The result is a set of algebraic equations through which mass, momentum, and energy transport are predicted at discrete points in the domain. In the freeboard model that is being described, the segregated solver has been chosen so the governing equations are solved sequentially. Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained and each of the iteration is carried out as follows:

(1) Fluid properties are updated in relation to the current solution; if the calculation is at the first iteration, the fluid properties are updated consistent with the initialized solution.

(2) The three momentum equations are solved consecutively using the current value for pressure so as to update the velocity field.

(3) Since the velocities obtained in the previous step may not satisfy the continuity equation, one more equation for the pressure correction is derived from the continuity equation and the linearized momentum equations: once solved, it gives the correct pressure so that continuity is satisfied. The pressure–velocity coupling is made by the SIMPLE algorithm, as in FLUENT default options.

(4) Other equations for scalar quantities such as turbulence, chemical species and radiation are solved using the previously updated value of the other variables; when inter-phase coupling is to be considered, the source terms in the appropriate continuous phase equations have to be updated with a discrete phase trajectory calculation.

(5) Finally, the convergence of the equations set is checked and all the procedure is repeated until convergence criteria are met.

The conservation equations are linearized according to the implicit scheme with respect to the dependent variable: the result is a system of linear equations (with one equation for each cell in the domain) that can be solved simultaneously. Briefly, the segregated implicit method calculates every single variable field considering all the cells at the same time. The code stores discrete values of each scalar quantity at the cell centre; the face values must be interpolated from the cell centre values. For all the scalar quantities, the interpolation is carried out by the second order upwind scheme with the purpose of achieving high order accuracy. The only exception is represented by pressure interpolation, for which the standard method has been chosen.

4.3.3 Post-Processing:

This is the final step in CFD analysis, and it involves the organization and interpretation of the predicted flow data and the production of CFD images and animations. Fluent's software includes full post processing capabilities. FLUENT exports CFD's data to third-party post-processors and visualization tools such as Ensight, Fieldview and TechPlot as well as to VRML formats. In addition, FLUENT

CFD solutions are easily coupled with structural codes such as ABAQUS, MSC and ANSYS, as well as to other engineering process simulation tools.

Thus FLUENT is general-purpose computational fluid dynamics (CFD) software ideally suited for incompressible and mildly compressible flows. Utilizing a pressure-based segregated finite-volume method solver, FLUENT contains physical models for a wide range of applications including turbulent flows, heat transfer, reacting flows, chemical mixing, combustion, and multiphase flows. FLUENT provides physical models on unstructured meshes, bringing you the benefits of easier problem setup and greater accuracy using solution-adaptation of the mesh. FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc.

Geometry and grid generation is done using GAMBIT which is the pre-processor bundled with FLUENT. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools. These include : Domain geometry & Grid display ,Vector plots, Line & shaded contour plots, 2D & 3D surface plots, Particle tracking, View manipulation (translation, rotation, scaling etc.)

4.4 **Advantages of CFD**

Major advancements in the area of gas-solid multiphase flow modeling offer substantial process improvements that have the potential to significantly improve process plant operations. Prediction of gas solid flow fields, in processes such as pneumatic transport lines, risers, fluidized bed reactors, hoppers and precipitators are crucial to the operation of most process plants. Up to now, the inability to accurately model these interactions has limited the role that simulation could play in improving operations. In recent years, computational fluid dynamics (CFD) software developers have focused on this area to develop new modeling methods that can simulate gas-liquid-solid flows to a much higher level of reliability. As a result, process industry engineers are beginning to utilize these methods to make major improvements by evaluating alternatives that would be, if not impossible, too expensive or time-consuming to trial on the plant floor. Over the past few decades, CFD has been used to improve process design by allowing engineers to simulate the performance of alternative configurations, eliminating guesswork that would normally be used to establish equipment geometry and process conditions. The use of CFD enables engineers to obtain solutions for problems with complex geometry and boundary conditions. A CFD analysis yields values for pressure, fluid velocity, temperature, and species or phase concentration on a computational grid throughout the solution domain. Advantages of CFD can be summarized as:

1. It provides the flexibility to change design parameters without the expense of hardware changes. It therefore costs less than laboratory or field experiments, allowing engineers to try more alternative designs than would be feasible otherwise.

2. It has a faster turnaround time than experiments.

3. It guides the engineer to the root of problems, and is therefore well suited for trouble-shooting.

4. It provides comprehensive information about a flow field, especially in regions where measure elments are either difficult or impossible to obtain.

CHAPTER 5

ACUCONSOLE & MESHING

5 AcuConsole&Mesh

AcuConsole is a GUI based pre-processor for AcuSolve, Altair Engineering's CFD analysis package.

AcuConsole provides the following functionality:

•Read CAD geometry from CATProduct, Parasolid, ACIS, Granite and STL formats.

•Generate a mesh on the geometry. Mesh generation in AcuConsole is carried out by a stand-alone executable called AcuMeshSim.

•Read an existing mesh from various formats (such as AcuSolve .inp, ICEMCFD and FLUENT)

•Set analysis attributes and boundary conditions for CFD analysis with AcuSolve.

•Launch AcuSolve

•Monitor the CFD analysis using AcuProbe

•Basic post-processing using AcuOut

•Launch third-party visualization software such as AcuFieldView, EnSight and ParaView.

This section focuses on the mesh generation capability in AcuConsole. Mesh generation consists of specifying mesh attributes and launching the mesh generator from AcuConsole. For our model, a pipe with water and air domain CATIA assembly file is loaded into AcuConsole. After loading the geometry, three volume groups called fluid-Water and air, solid-aluminium is created and three surface groups (inflow, outflow and wall for pipe) and (inflow, outflow and walls for air) are created.

Once the appropriate geometric faces are attached to the inflow, outflow, wall and interface surface groups, the geometric volumes region is attached to the fluid and solid volumes group. A geometric face is a two dimensional entity bounded by edges. Each Surface group can have one or more geometric faces.

A geometric region is a three dimensional entity bounded by geometric faces. Each Volume group can have one more geometric regions.

There are many levels of mesh controls that are provided in AcuConsole to get the desired element sizing. Mesh controls can be set at the model level, at the individual volume level, or at the surface level. In addition, AcuConsole has the capability to set the mesh sizes as a function of space through **Zone Mesh Attributes**. All of the mesh attributes are hierarchical in AcuConsole. That is, a mesh size applied to the entire model will be inherited by each of the geometric faces and regions in the model. A mesh size applied to a volume region will be inherited by all of the faces attached to the volume region. All of the geometric entities that have an overlap with any of the zone mesh attributes will inherit mesh sizes from the zone mesh attributes. AcuConsole uses the smallest of all the sizes that are applied to a geometric entity to determine the final mesh size setting.

Once regions and groups are complete, click on Tools then Generate Mesh to open the Launch AcuMeshSim dialog.

The Launch AcuMeshSim dialog has some basic and advanced parameters that control the meshing process. In this click on global mesh attributes mesh size type as Relative, relative mesh size as 0.01curvature refinement parameters on curvature angle is 25, curvature mesh size factori s 0.5, mesh growth rate as 1, maximum sweep angle as 45 and then click Ok.

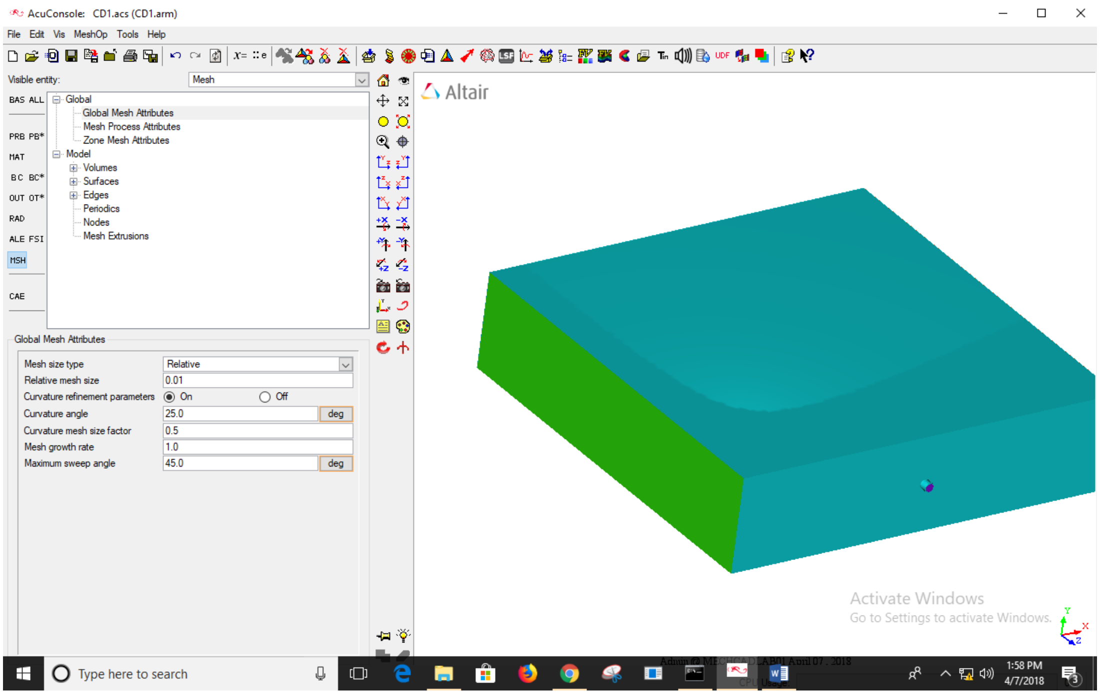
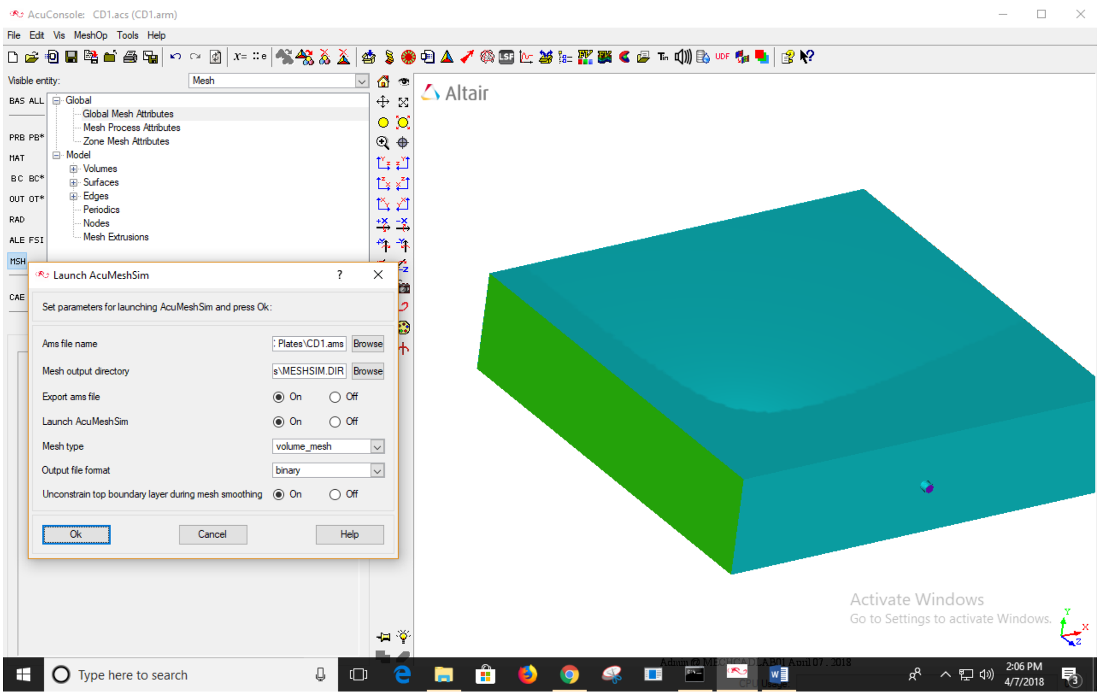


Fig 5.1 final assembly before meshing



Once the mesh generation is complete, the mesh data is written on the disk and AcuConsole notifies you that the mesh generation was successful.

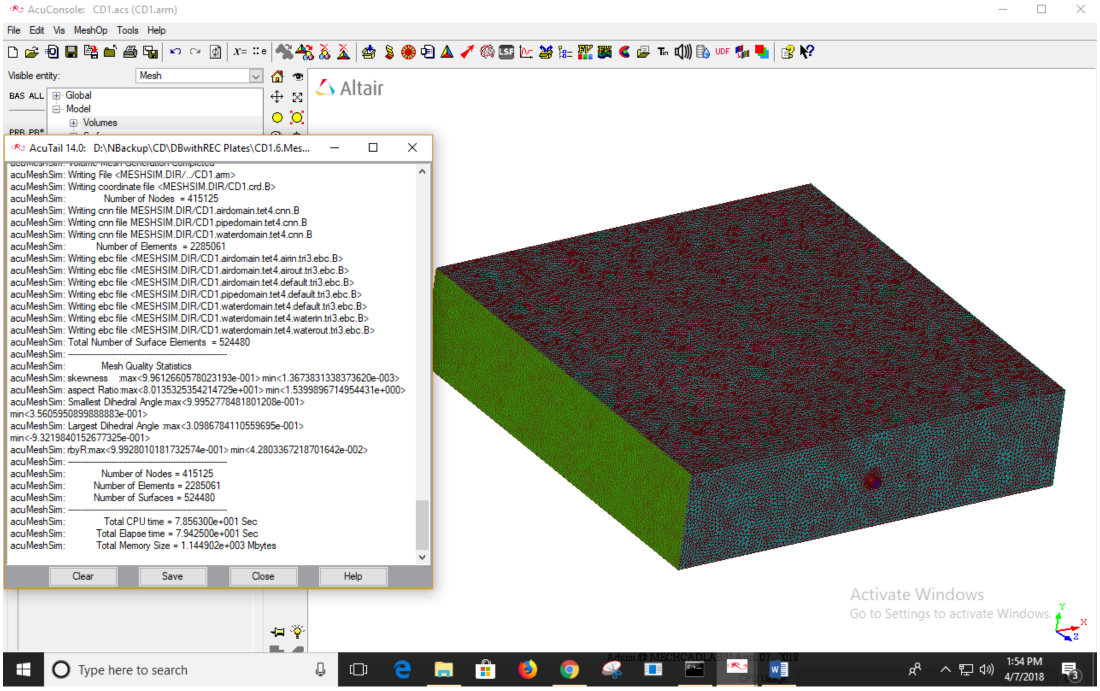


Fig 5.2 final assembly after meshing

Boundary conditions and other simulation attributes can be set with or without mesh or geometry loaded in AcuConsole. Once the simulation attributes are set, AcuSolve can be launched to perform the CFD analysis.

5.1 **AcuSolve-Procedure**

- Go to Start and All programs click on Acu console.
- Go to File then click on new and type file name as CD.acs and save.
- Go to File Import type as CATIA Product File. Browse the CAD model from CATIAProduct file.
- Go to Global then go to problem description, double click. Temperature equation select as Advective Diffusive.
- Go to Material model and right click and new rename the material model as solid and given the properties of aluminum. Again Go to Material model and right click and new rename the material model as fluid and given the properties of air. Go to Material model and right click and new rename the material model as fluid and given the properties of water.
- Go to model, click on volume manager. Click on new and name as pipedom add to select the geometry and set medium as solid, set material model aluminium. Go to model, click on volume manager. Click on new and name as waterdom add to select the geometry and set medium as fluid, set material model water. Go to model, click on volume manager. Click on new and name as airdom add to select the geometry and set medium as fluid, set material model air and close.
- Go to model and right click on surface manager. generate six new surfaces by clicking new tab. Name the surface1 as airin, surface2 as airout, surface3 as adiabatic walls, surface4 as waterin, surface5 as waterout, surface6 as interface walls. Right click on airin goto add to select the airinlet surface, right click on airout goto add to select the airoutlet surface, right click on waterin goto add to select the water inlet surface, right click on waterout goto add to select the water outlet surface, right click on adbwalls goto add to select the adiabatic surfaces. select all the remaining surfaces as interfaces.
- Once the surface addition and volume addition is over, right click on the volumes and surfaces in model click on purge.
- Go to BC and select airout surface and click on simple boundary condition type of boundary condition as ouflow, pressure outlet as 0pa.

- Go to BC and select airin surface and click on simple boundary condition type of boundary condition as inflow, normal velocity as 1m/s, change temperature as $300^{o}K$
- Go to BC and select waterin surface and click on simple boundary condition type of boundary condition as inflow, normal velocity as 0.2m/s, change temperature as $360^{o}K$.

Go to BC and select waterout surface and click on simple AcuSolve-Procedure

- Go to Start and All programs click on Acu console.
- Go to File then click on new and type file name as CD.acs and save.
- Go to File Import type as CATIA Product File. Browse the CAD model from CATIAProduct file.
- Go to Global then go to problem description, double click. Temperature equation select as Advective Diffusive.
- Go to Material model and right click and new rename the material model as solid and given the properties of aluminum. Again Go to Material model and right click and new rename the material model as fluid and given the properties of air. Go to Material model and right click and new rename the material model as fluid and given the properties of water.
- Go to model, click on volume manager. Click on new and name as pipedom add to select the geometry and set medium as solid, set material model aluminium. Go to model, click on volume manager. Click on new and name as waterdom add to select the geometry and set medium as fluid, set material model water. Go to model, click on volume manager. Click on new and name as airdom add to select the geometry and set medium as fluid, set material model air and close.
- Go to model and right click on surface manager. generate six new surfaces by clicking new tab. Name the surface1 as airin, surface2 as airout, surface3 as adiabatic walls, surface4 as waterin, surface5 as waterout, surface6 as interface walls. Right click on airin goto add to select the airinlet surface, right click on airout goto add to select the airoutlet surface, right click on waterin goto add to select the water inlet surface, right click on waterout goto add to select the water outlet surface, right click on adbwalls goto add to select the adiabatic surfaces. select all the remaining surfaces as interfaces.

- Once the surface addition and volume addition is over, right click on the volumes and surfaces in model click on purge.
- Go to BC and select airout surface and click on simple boundary condition type of boundary condition as ouflow, pressure outlet as 0pa.
- Go to BC and select airin surface and click on simple boundary condition type of boundary condition as inflow, normal velocity as 1m/s, change temperature as 300^{o}K
- Go to BC and select waterin surface and click on simple boundary condition type of boundary condition as inflow, normal velocity as 0.2m/s, change temperature as 360^{o}K.
- boundary condition type of boundary condition as ouflow, pressure outlet as 0pa.
- Go to Mesh then got to global mesh attributes and give the relative mesh size as 0.01 ckick on Generate Mesh icon to generate mesh.
- Click on LaunchAcusolve icon and press ok tab to run the solver.

5.2 **Materials**

Plate fin heat exchangers can be made in a variety of materials. Aluminium ispreferred in cryogenic and aerospace applications because of its low density, high thermalconductivity and high strength at low temperature. The maximum design pressure for brazedaluminium plate fin heat exchangers is around 90 bar. At temperatures above ambient,most aluminium alloys lose mechanical strength. Stainless steels, nickel and copper alloyshave been used at temperatures up to 500^0C. The brazing material in case of aluminiumexchangers is an aluminium alloy of lower melting point, while that used in stainless steelexchangers is a nickel based alloy with appropriate melting and welding characteristics.

5.3 **Applications**

Plate-fin and tube-fin heat exchangers have found application in a wide variety of industries. Among them are air separation (production of oxygen, nitrogen and argon by low temperature distillation of air), petro-chemical and syn-gas production, helium and hydrogen liquefiers, oil and gas processing, automobile radiators and air conditioners, and environment control and secondary power systems of aircrafts.

These applications cover a wide variety of heat exchange scenarios, such as:

(1) exchange of heat between gases, liquids or both,

(2) condensation, including partial and reflux condensation,

(3) boiling,

(4) sublimation, and

(5) heat or cold storage

5.4 **Flow Arrangement**

A plate fin heat exchanger accepts two or more streams, which may flow in directions parallel or perpendicular to one another. When the flow directions are parallel, the streams may flow in the same or in opposite sense. Thus we can think of three primary flow arrangements –

(i) parallel flow,

(ii) counterflow and

(iii) cross flow.

(iv) cross-counter flow.

Thermodynamically, thecounterflow arrangement provides the highest heat (or cold) recovery, while the parallel flow geometry gives the lowest. The cross flow arrangement, while giving intermediate thermodynamic performance, offers superior heat transfer properties and easier mechanical layout. Under certain circumstances, a hybrid cross – counterflow geometry provides greater heat (or cold) recovery with superior heat transfer performance. Thus in general engineering practice, plate fin heat exchangers are used in three configurations:

(a) Cross flow (Fig. (a))

In a cross flow heat exchanger, usually only two streams are handled, thus eliminating the need for distributors. The header tanks are located on all four sides of the heat exchanger core, making this arrangement simple and cheap. If high effectiveness is not necessary, if the two streams have widely differing volume flow rates, or if either one or both streams are 5 nearly isothermal (as in single component condensing or boiling), the cross flow arrangement is preferred. Typical applications include automobile radiators and some aircraft heat exchangers.

(b) Counter flow (Fig. (b))

The counterflow heat exchanger provides the most thermally effective arrangement for recovery of heat or cold from process streams. Cryogenic refrigeration and liquefaction equipment use this geometry almost exclusively. The geometry of the headers and the distributor channels is complex and demands proper design.

(c) Cross-Counter flow (Fig. (c))

The cross-counterflow geometry is a hybrid of counterflow and cross flow arrangements, delivering the thermal effectiveness of counterflow heat exchanger with the maximum effectiveness.

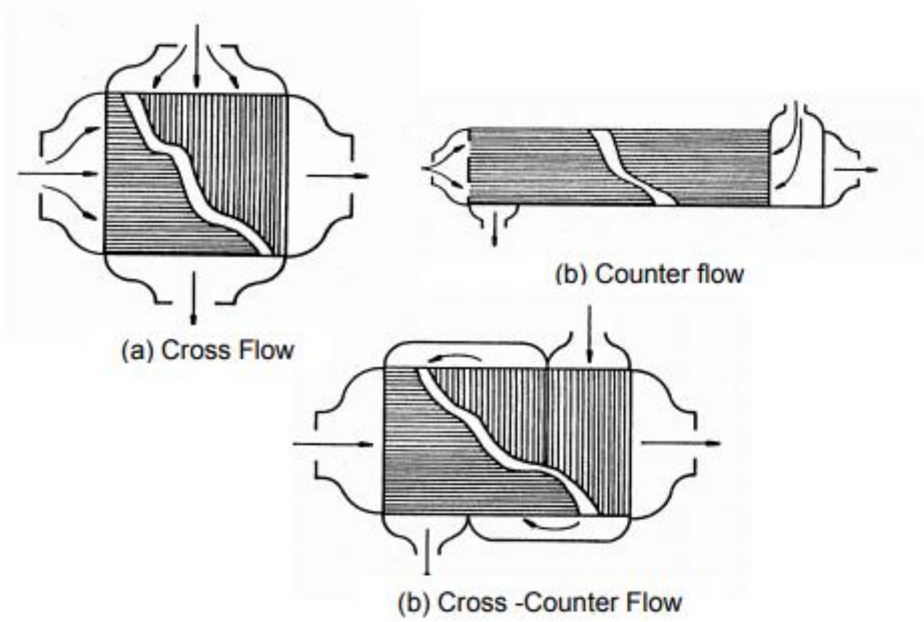


Fig 5.3 Flow arrangements

(a) Cross Flow(b) Counter flow(c) Cross -Counter Flow

Figure : Heat exchanger flow arrangements

The residual ratios used by AcuSolve are a non-dimensional measure of how "out-of-balance" the governing equations are. The residual ratios are recomputed at each timestep to give an indication of how well the solution is converging.

The solution ratio that is computed by AcuSolve represents the ratio of the change in the solution between timesteps to the solution at the previous timestep. This value indicates the level of unsteadiness present in the solution.

AcuSolve Residual Computation

Convergence checks are performed on the residual and solution increments. For the residual, the norm of the residual is normalized with respect to the norm of the forces making up the residual. This ratio is the relevant measure, because it measures the ratio of the out-of-balance forces to the value of the forces. This ratio is computed separately for each solution field.

For the solution increment, the norm of the solution increment is normalized with respect to the norm of the solution field prior to updating the solution. This ratio is also computed separately for each solution field. These ratios are used to check for convergence

Convergence occurs when all residual and solution increment ratios fall below the specified tolerance. The CONVERGENCE_CHECK_PARAMETERS command may be used to control the relative importance of each ratio. Convergence checks are performed on the residual and solution increments.

To be more precise, consider the vector of out-of-balance residual for a given equation, such as continuity equation:

R = {Ri} i = 1, ...,nNodes

where "nNodes" is the number of nodes in the system. The ideal goal is to find a solution (velocity, pressure, etc.) such that "R" (i.e. every single "Ri") is zero. This, of course, is not practical, so instead we require that "R" be small. Since "R" is dimensional and highly dependent on the particular problem, we need to normalize it first.

Consider the continuity equation (which corresponds to the pressure degree of freedom). " Ri " is the imbalance in the mass fluxes associated with the node "i". This "Ri " is the sum of the element contributions to the node "i". For example consider the case where we have four element contributing to the node "i":

```
o---------o    o---------o
|         |    |         |
| 1       |    | 2       |
```

```
|         |         |         |
o---------o         o---------o
          i
o---------o         o---------o
|         |         |         |
| 3       |         | 4       |
|         |         |         |
o---------o         o---------o
```

Then Ri = Ri 1 + Ri 2 + Ri 3 + Ri 4

where " Ri 1" is the contribution of element 1 to node "i".

or, " Ri 1" is the integrated mass flux of element 1 towards node "i".

Given this definition, we can define: M(Ri) = |Ri1| + |Ri2| + |Ri3| + |Ri4| where "M(Ri)" is a measure of the mass flux for node "i"; and "|Ri1|" is the absolute value of "Ri1". With the help of "M(Ri)" we can now normalize the residual to get a, so called "residual ratio":

residual ratio = ||R|| / ||M(R)||

where "||R||" is a norm of the vector "R" and "||M(R)||" is a norm of the vector "M(R)={M(Ri)}". For computational efficiency, we use a 1-norm to compute the above norms. Recall that 1-norm is defined as the sum of absolute values:

||R||_1 = Sum_i |Ri |

For the solution increment, the norm of the solution increment is normalized with respect to the norm of the solution field prior to updating the solution. This ratio is also computed separately for each solution field.

To be more precise, once the residual vector, "R", and the left-hand-side (LHS) matrix, "A", is computed, we solve the following linear equation system for the solution increment "ΔP":

$A \Delta P = -R$

and the solution is updated as

$P \longleftarrow P + \Delta P$

The solution increment ration is then defined as:

solution ratio = $||\Delta P|| / ||P||$

In this case, we use a 2-norm

$||P||\_2 = Sqrt(Sum\_i\ Pi2\ )$

These ratios are used to check for convergence. Convergence occurs when all residual and solution increment ratios fall below the specified tolerance. Recall that the CONVERGENCE_CHECK_PARAMETERS command may be used to control the relative importance of each ratio.

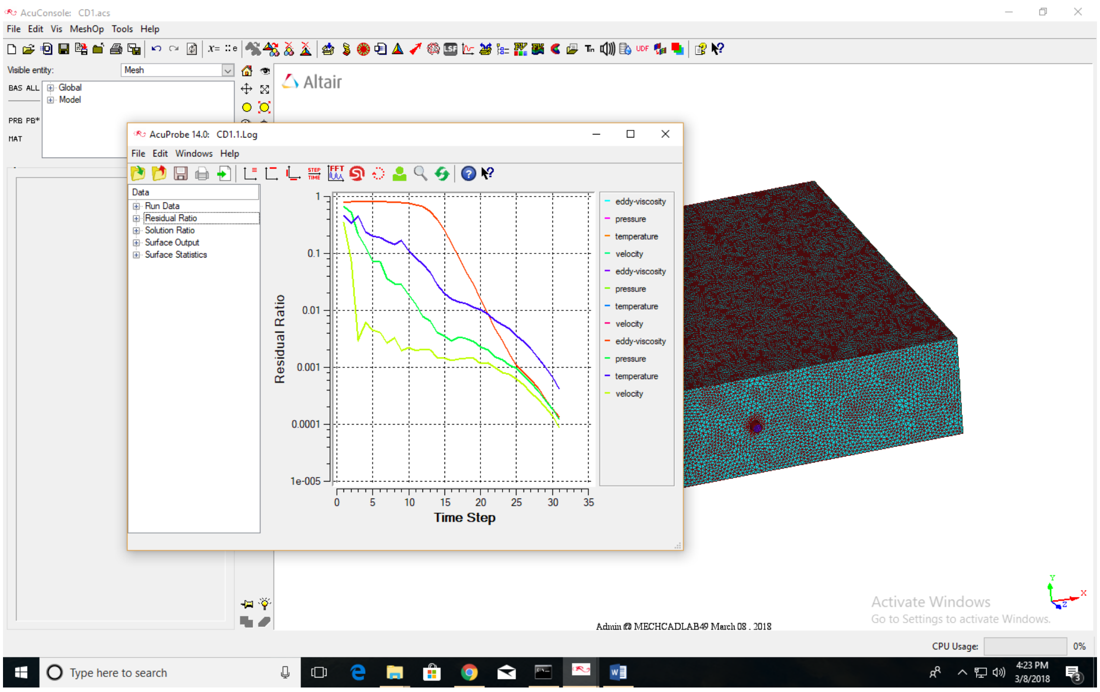


Fig 5.4 residual plot

Chapter 6

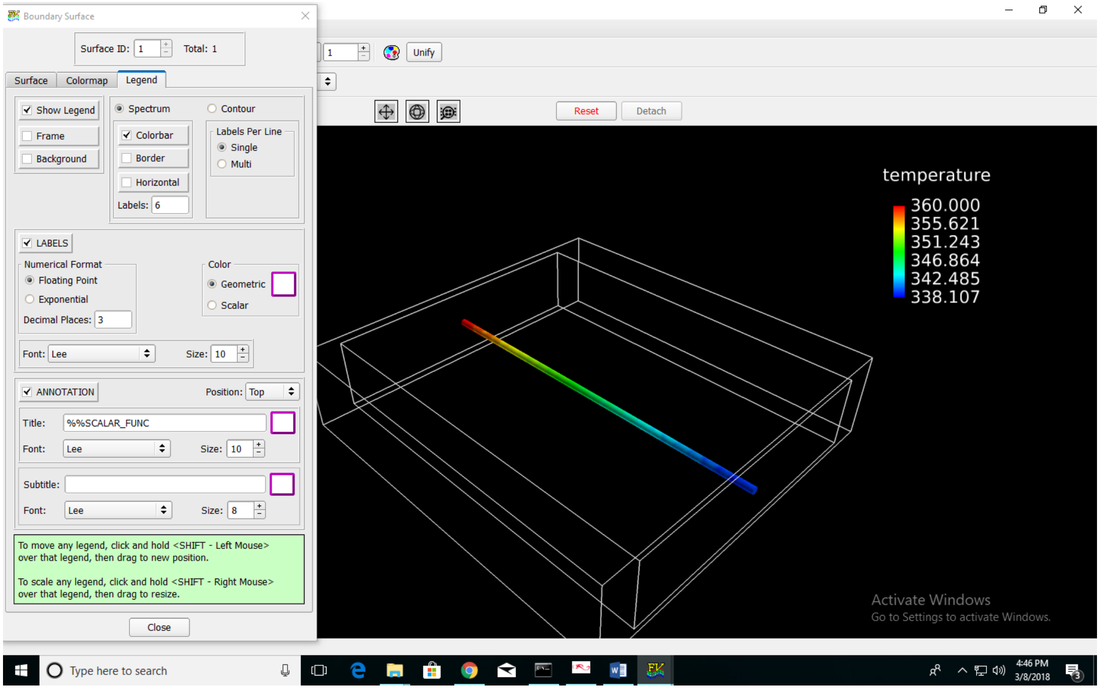
Results and discussions

6 Results and Discussions:

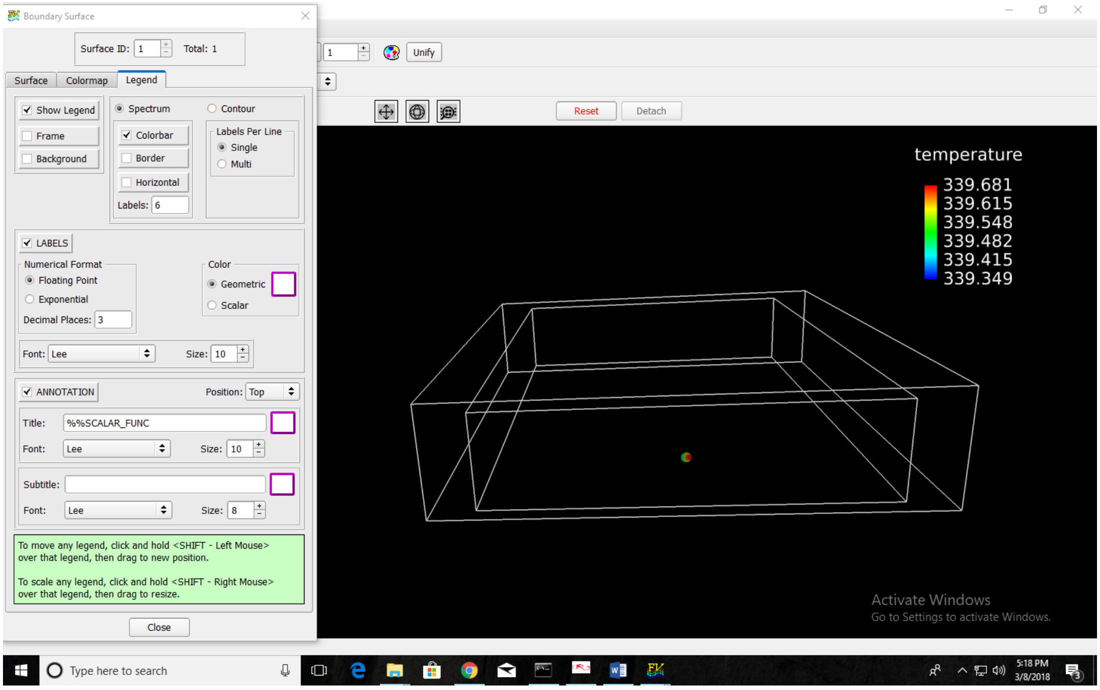
6.1 Tube without Inserts:

Isothermal contours shown in Fig. _ reveals about the temperature distribution at the outlet of water tube without any kind of insert. Lowest temperature is recorded near to the outlet walls which is due to the exposure of air. A temperature of 338.1 K has been recorded at the outlet for the water tube without inserts. Temperature distribution in the form of concentric circles can be sited in the figure shown which is due to lack of swirl or high turbulence inside the tubes.

6.



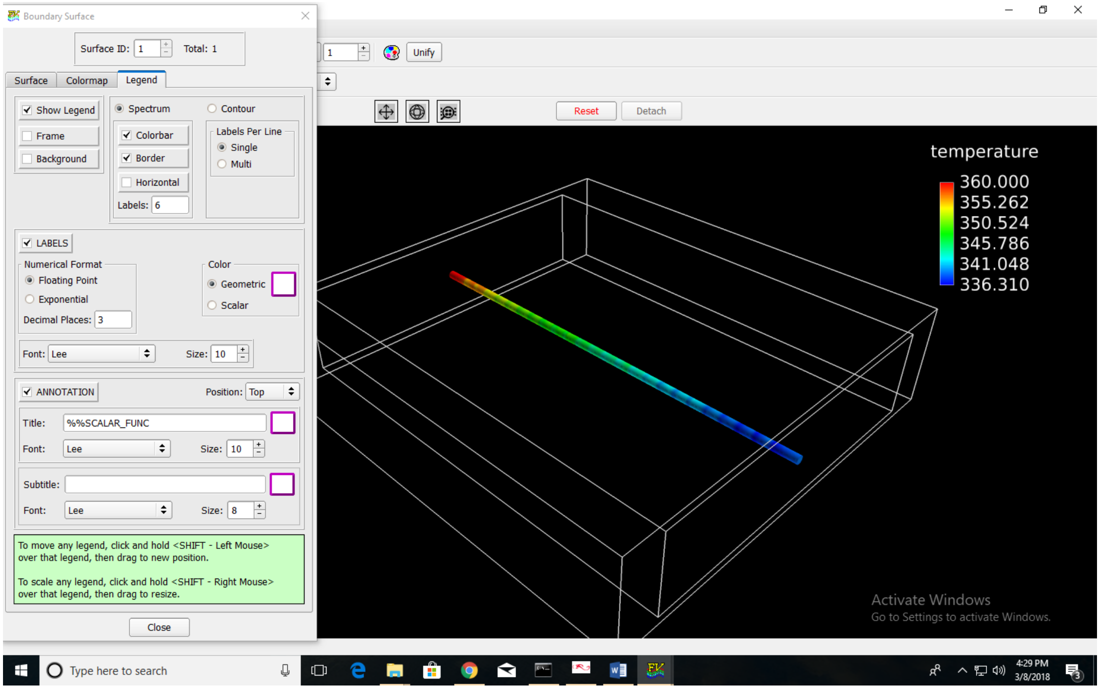
6.1 temperature contour of pipe without inserts



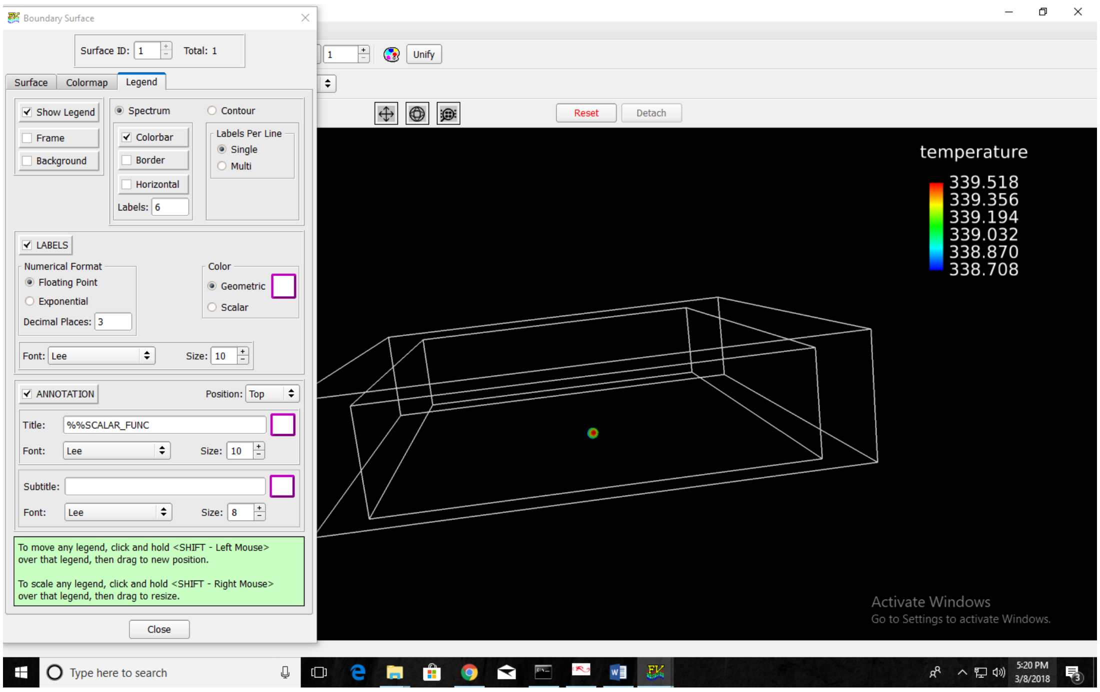
6.2 temperature contour at outlet of pipe without inserts

6.2 **Tube with longitudinal fins:**

Fig shows the isothermal contours for radiator tube with longitudinal fins. From the fig it can be observed that there is a non uniform distribution of thermal contours depicting a strong swirl motion outside the tube. A lowest temperature of 336.3 K has been recorded..



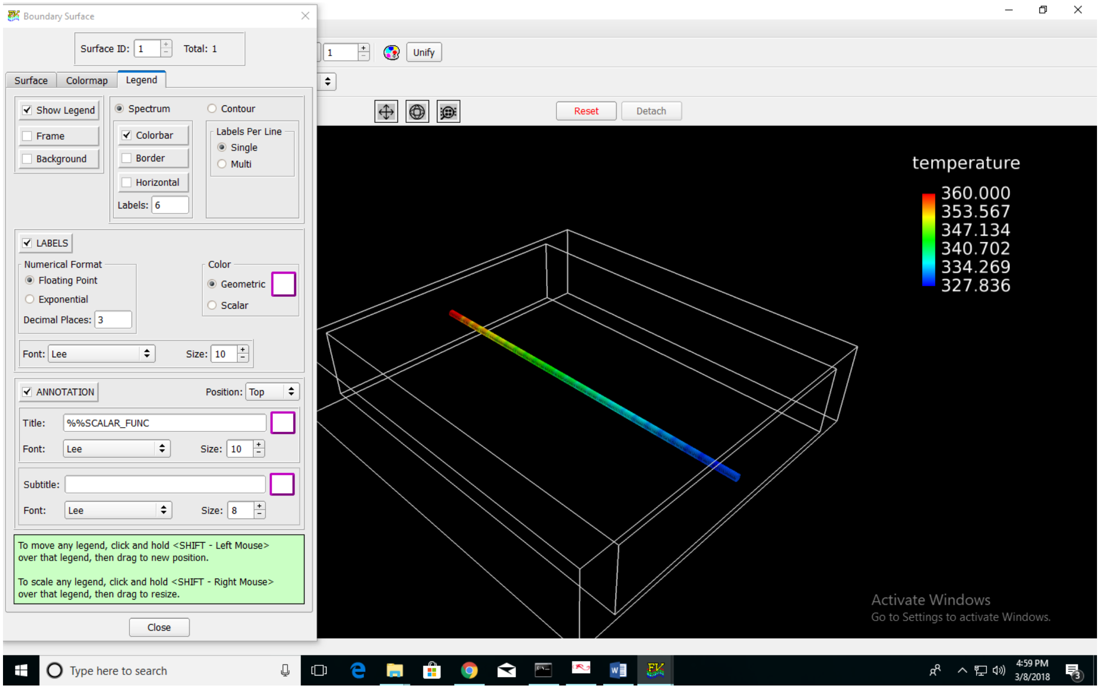
6.3 temperature contour of tube with longitudinal fin



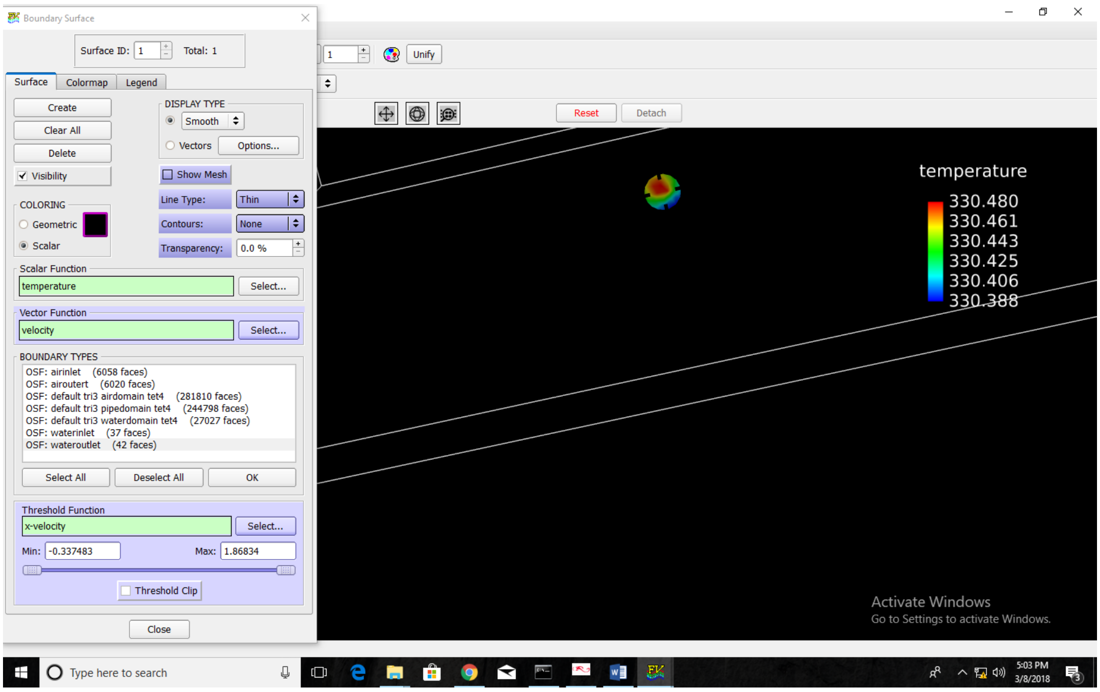
6.4 temperature contour of pipe at outlet with longitudinhal fin

6.3 **MLR-Tube:**

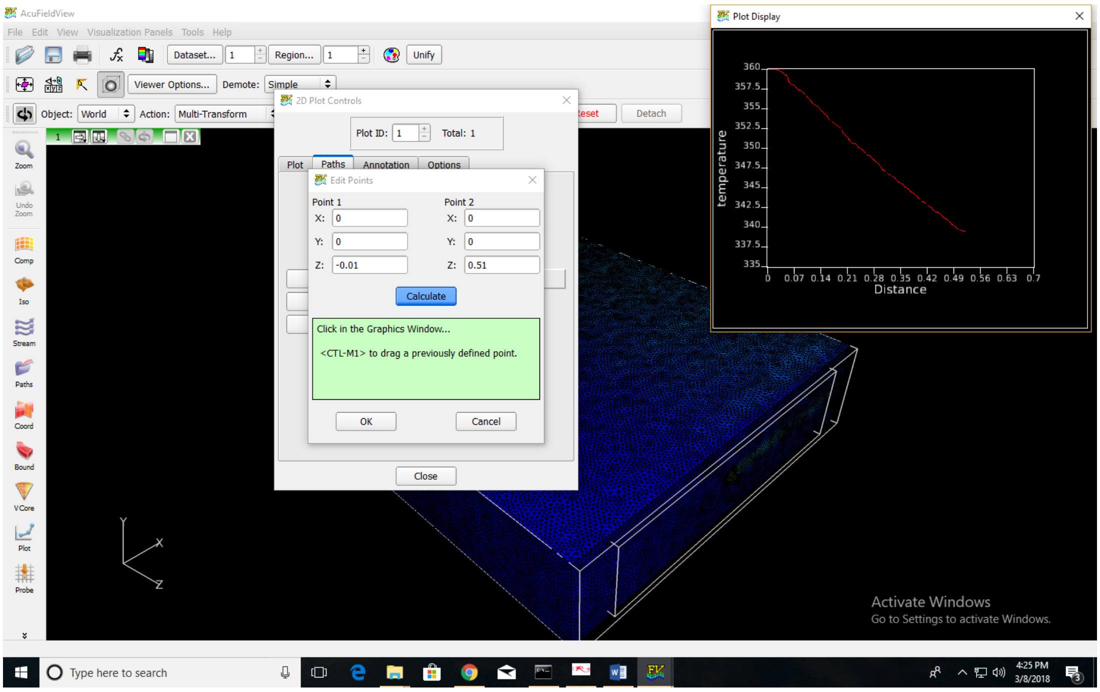
The contours in fig shows the temperature distribution at the outlet of the tube with leads. A careful observation of the Isothermal contours depicts decrease in temperature at the leads. This is due to the more conduction and decrease of film condensation at the inner tube walls. Temperature of 327.8 K has been obtained at outlet of the tube, which shows enhanced heat transfer with the use of leads in boiler tubes.



6.5 temperature contour of MLR tube

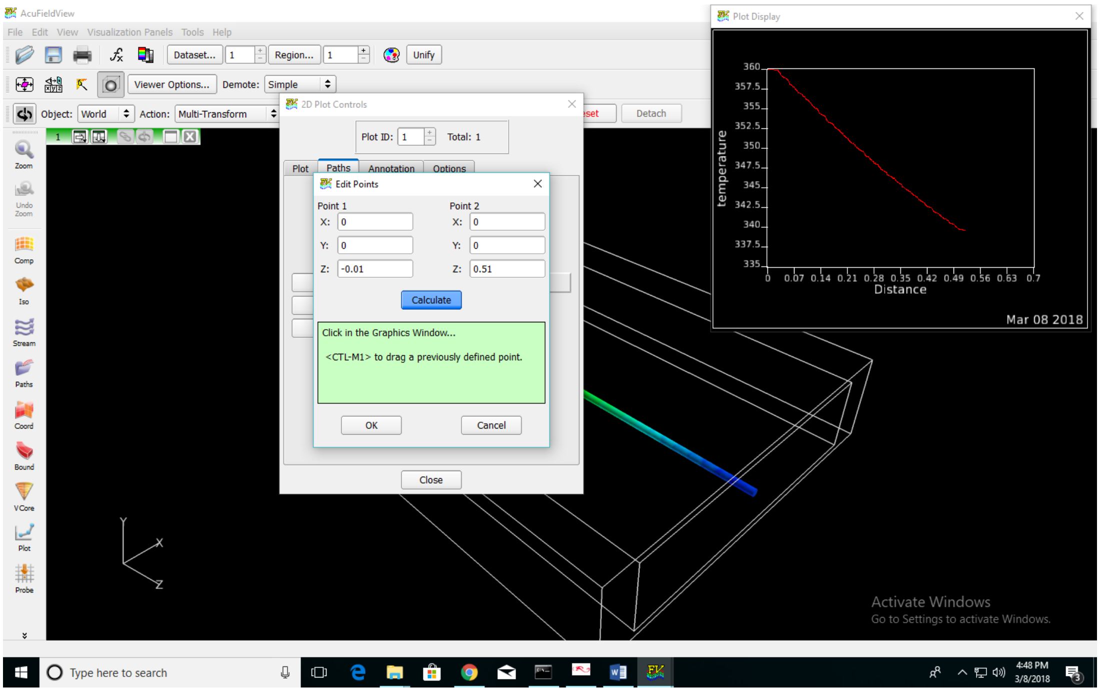


6.6 temperature contour of MLR tube at outlet



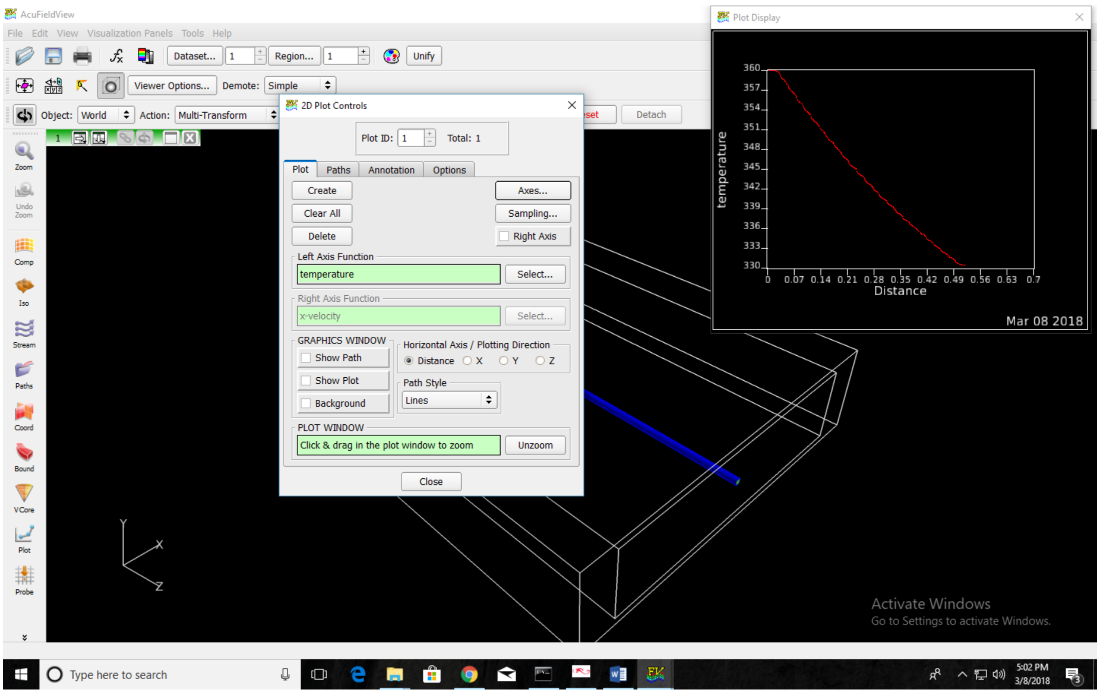
GRAPH 6.1 temperature plot of pipe without fin

The variation of temperature of water with the direction of pipe inlet center to pipe outlet center (Z-direction), at different values of pipe length vs temperature is shown in Fig. From the plot there is drop in temperature from 360 k to 338 k in Z direction uniformly



GRAPH 6.2 temperature plot of pipe with longitudinal fin

The variation of temperature of water with the direction of pipe inlet center to pipe outlet center (Z-direction), at different values of pipe length vs temperature is shown in Fig. From the plot there is drop in temperature from 360 k to 336 k in Z direction uniformly.



GRAPH 6.3 temperature plot of MLR tube

The variation of temperature of water with the direction of pipe inlet center to pipe outlet center (Z-direction), at different values of pipe length vs temperature is shown in Fig. From the plot there is drop in temperature from 360 k to 327 k in Z direction uniformly.

Chapter 7

CONCLUSION

7 Conclusions

From the numerical study carried out the following conclusions can be drawn from the results obtained.

1. The results showed that conventional radiator tubes have high temperatures and more losses.
2. It is also observed from the numerical results that multi lead riffled tubes are effective in loss in heat transfer tube to film condensation at walls. Andecrease in the temperatures is also cited.
3. Radiator tubes with and multi lead riffled as shown in fig. showed improved performance when compared with conventional and longitudinal finned tubes. A low temperature of 327.8 K has been obtained.

Finally it can be conclude that when compared with conventional and longitudinal finned radiator tubes with Multi lead rifled tubes has high heat transfer rates and temperatures which is highly desirable in radiators as it improve its efficiency.

Future scope:

- We can analyze the MLR tube by varying the pitch of helix .
- We can use helical ribbons instead of MLR to analyze the heat transfer rate.
- Analyze the heat transfer for MLR tube with longitudinal fins.

References:

[1] Nice Thomachan et al. Int. Journal of Engineering Research and Applications Vol. 3, Issue 5, Sep-Oct 2013, pp.24-26.

[2]Vincent.H.Wilson; M.Hajee Mohamed International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-216 Volume 1 Issue 12 (December 2014).

[3] Ajay N. Ingale, VivekC.Pathade – *"CFD Analysis of Super Heater in View of Boiler Tube Leakage"* –International Journal of Engineering and Innovative Technology (IJEIT) Volume 1, Issue 3, March 2012.

[4] PrashantKumkale - *"COMPUTATIONAL STUDY OF FLOW THROUGH A SUPER HEATER FOR STUDY THEVARIOUS HEAT TRANSFER CHARACTERISTICS"* - International Journal of Advance Research In Science AndEngineering - IJARSE, Vol. No.3, Issue No.9, September 2014 - ISSN-2319-8354(E)

[5] Aditya KumarPanday& L.A. Kumaraswamidhas-*"*Analysis of Flow Induced Vibration In Super Heater Tube Bundles in Utility Boilers Using Computational *Method"* – International Journal of Computational Engineering Research/ISSN:2250-3005.IJCER\Mar-Apr 2012\Vol.2\Issue No.2\481-486.

[6] Raja Saripally, Ting Wang and Benjamin Day. (2005)Simulation of combustion and thermal flow in an Industrial boiler, Proceedings of 27th Industrial Energy Technology Conference, New Orleans, Louisiana.

[7] Fan, A.W., Deng, J.J., Nakayama, A., Liu.W, "Parametric study on turbulent heat transfer and flow characteristics in a circular tube fitted with Louvered strip inserts", International Journal of Heat Transfer and Mass Transfer, 2012, vol. 55, pp. 5205-5213.