

# **DESIGN AND ANALYSIS OF A TRIANGULAR SHAPE CLOSED LOOP WIND TUNNEL WITH GUIDE VANE RADIATOR USING CFD**

A project report submitted in partial fulfillment of the requirements for the award of the degree of

## **BACHELOR OF TECHNOLOGY IN MECHANICAL ENGINEERING**

By

<b>K.NAVEEN KUMAR</b>	<b>(315126520111)</b>
<b>M.VENKATA SNEHALATHA</b>	<b>(315126520126)</b>
<b>K.NANDA SAI KUMAR</b>	<b>(315126520100)</b>
<b>K.RAHUL SAGAR</b>	<b>(315126520083)</b>
<b>K.MANOJ KUMAR</b>	<b>(315126520096)</b>

Under the guidance of

**Mr. J.MAHESH, M. Tech.**

Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING



**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES**  
(Autonomous)

(Approved by AICTE, permanently affiliated to ANDHRA UNIVERSITY)

Accredited by NBA and NAAC with 'A' Grade

Sangivalasa –531162, Bheemunipatnam, Visakhapatnam.

**ANIL NEERUKONDA INSTITUTE OF TECHNOLOGY AND SCIENCES**

**(Autonomous)**

(Affiliated to Andhra University, Accredited by NBA)

Sangivalasa 531162, Bheemunipatnam Mandal, Visakhapatnam.



**DEPARTMENT OF MECHANICAL ENGINEERING**

**CERTIFICATE**

This is to certify that the project entitled “**DESIGN AND ANALYSIS OF A TRIANGULAR SHAPE CLOSED LOOP WIND TUNNEL WITH GUIDE VANE RADIATOR USING CFD**” describes the bonafide work done by **K.NAVEEN KUMAR(315126520111), M.VENKATA SNEHALATHA (315126520126), K.NANDA SAI KUMAR (315126520100), K.RAHUL SAGAR (315126520083), K.MANOJ (315126520096)**, in partial fulfillment for the award of Degree in **BACHELOR OF TECHNOLOGY** in **MECHANICAL ENGINEERING** under my supervision and guidance during the academic year **2018-2019**.

*Mahesh*

**INTERNAL GUIDE**

**J.MAHESH**

**Assistant Professor**

**Dept. of Mechanical Engineering**

*13.4.19*

**HEAD OF THE DEPARTMENT**

**Dr. B. NAGA RAJU**

**Dept. of Mechanical Engineering**

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

**THIS PROJECT IS APPROVED BY THE BOARD OF EXAMINATIONS**

**EXTERNAL EXAMINER:**

*[Handwritten signature]*  
13/4/19

**INTERNAL EXAMINER:**

*[Handwritten signature]*  
13/6/19

## **ACKNOWLEDGEMENT**

We present this report on “**DESIGN AND ANALYSIS OF A TRIANGULAR SHAPE CLOSED LOOP WIND TUNNEL WITH GUIDE VANE RADIATOR USING CFD**” in the partial fulfillment of the requirement for the award of BACHELOR OF TECHNOLOGY in MECHANICAL ENGINEERING.

We intend to express our thanks with sincere obedience to **Prof. T. Subrahmanyam**, Principal, ANITS and **Prof. B. Naga Raju**, Head of the department, Mechanical Engineering, ANITS for providing this opportunity.

We take this opportunity to express our deep and sincere thanks to **Mr. M. Prem Dheeraj**, Assistant Professor; **Mr. Basa Nagaraju**, Assistant Professor; **Mr. B. RoopSandeep**, Assistant Professor and to our esteemed guide **Mr. J. Mahesh**, Assistant Professor, Mechanical Engineering Department, ANITS, a source of constant motivation and best critic, for his inspiring and infusing ideals in getting our project done successfully.

Lastly we are grateful to one and all who have contributed either directly or indirectly in the completion of the project.

<b>K. NAVEEN KUMAR</b>	<b>(315126520111)</b>
<b>M. VENKATA SNEHALATHA</b>	<b>(315126520126)</b>
<b>K.NANDA SAI KUMAR</b>	<b>(315126520100)</b>
<b>K. RAHUL SAGAR</b>	<b>(315126520083)</b>
<b>K. MANOJ KUMAR</b>	<b>(315126520096)</b>

## LIST OF FIGURES AND TABLES

<b>Figure Number</b>	<b>Figure Description</b>	<b>Page No.</b>
1.1	Pressure storage type wind tunnel	3
1.2	Pressure vacuum wind tunnel	4
1.3	Open loop wind tunnel	4
1.4	Closed loop wind tunnel	5
1.5	Typical lift coefficient versus angle of attack curve	8
3.1	2D geometry of 90° corner	22
3.2	2D geometry of 45° corner	23
3.3	2D geometry of 90° corner with guide vanes	23
3.4	2D geometry of 45° corner with guide vanes	24
3.5	Without guide vanes	25
3.6	With guide vanes	25
3.7	Pressure calculation by function calculator	26
4.1	Effuser	28
4.2	Test chamber	29
4.3	Diffuser	29
4.4	45° corner design step 1	30
4.5	45° corner design step 2	31
4.6	45° corner design step 3	31
4.7	Individual radiator	32
4.8	Assembly 1	32
4.9	Hypotenuse duct	33
4.10	Final assembly	33
5.1	Line diagram	34
6.1	ANSYS Workbench	37
6.2	Air domain	38
6.3	Radiator 1	38
6.4	Water domain 1	39
6.5	Radiator 2 and water domain 2	39
6.6	Mesh of air domain	40
6.7	Boundary condition	40

7.1	Velocity vs inlet temperature of water in first corner	43
7.2	Outlet temperature vs inlet temperature in second corner	44
8.1	Front view of simulation	45
8.2	Bottom view of simulation	46
8.3	Velocity scale	46
8.4	Horizontal view of test section	47
8.5	Vertical view of test section	47
8.6	Air inlet and outlet pressure	48
<b>Table number</b>	<b>Table Description</b>	<b>Page No.</b>
1.1	Mach speeds	2
3.1	Boundary conditions	24
3.2	Overall head loss	26

## **ABSTRACT**

A wind tunnel is an equipment especially used in aerodynamic research to study the effects of air moving past solid objects. A systematic investigation into the design and simulation of flow parameters of a redesigned closed-loop wind tunnel was carried out using Computational Fluid Dynamics (CFD) where the inlet conditions of the flow are obtained by applying Continuity equation at the effuser and test section. As it is a subsonic wind tunnel, a suitable velocity at the test section has to be considered and the corresponding inlet velocity is obtained. The standard model of closed loop wind tunnel-rectangular type is modified to triangular type using Solid works in order to reduce the number of corners thereby reducing overall head loss. The problems encountered on using a closed loop wind tunnel are generation of heat due to repeated circulation of air, head loss due to radiators. These problems are solved by designing a Guide vane radiator which is used for directing the air for circulation, cooling the circulating air and reducing overall head loss. Heat transfer in the guide vane radiators takes place to which necessary equations are applied to get the conditions of inlet water flow

A triangular shaped closed loop wind tunnel with Guide vane radiator is designed and simulated.

# **LIST OF CONTENTS**

<b>CERTIFICATE</b>	<b>ii</b>
<b>APPROVAL</b>	<b>iii</b>
<b>ACKNOWLEDGEMENT</b>	<b>iv</b>
<b>LIST OF FIGURES AND TABLES</b>	<b>v</b>
<b>ABSTRACT</b>	<b>vii</b>
<b>LIST OF CONTENTS</b>	<b>viii</b>
<b>1. INTRODUCTION</b>	<b>1</b>
1.1. Wind Tunnel Introduction	1
1.1.1. Classification of Wind Tunnels	2
1.1.1.1. Based on Speed, Mach No.	2
1.1.1.2. Based on mode of operation	3
1.1.1.3. Based on kind of test section	4
1.1.2. Components of Closed loop Wind Tunnel	6
1.1.2.1. Settling Chamber and Flow Straightener	6
1.1.2.2. Contraction	6
1.1.2.3. Test Section	6
1.1.2.4. Diffuser	7
1.1.2.5. Turning Vanes	7
1.1.3. How it works	7
1.1.4. Measurements made using Wind Tunnels	8
1.1.4.1. Pressure Measurements	8
1.1.4.2. Force and Moment Measurements	8
1.1.5. Applications of Wind Tunnels	8
1.1.5.1. Aerodynamic Applications	9
1.1.5.2. Non-Aerodynamic Applications	9
1.2. Introduction to CFD	



1.2.1. CFD	10
1.2.2. Governing equations	10
1.2.3. Discretization methods of CFD	10
1.2.4. Working of CFD	12
1.2.5. Advantages and Disadvantages	16
1.2.6. Applications of CFD	16
1.2.7. Predicting the turbulent viscosity	16
<b>2. LITERATURE REVIEW</b>	19
<b>3. WHY 45° CORNER WIND TUNNEL?</b>	22
3.1. Geometry of 45° corner and 90° corner	22
3.2. Air flow with and without guide vanes	24
3.3. Head loss at the corners	26
<b>4. DESIGN IN SOLID WORKS</b>	28
4.1. Effuser	28
4.2. Test Section	28
4.3. Diffuser	29
4.4. 45° Corner	30
4.5. Guide Vane Radiator	31
4.6. Hypotenuse Duct	32
4.7. Side Duct	33
4.8. Final Assembly	33
<b>5. a). MODE OF OPERATION</b>	34
<b>b). OBTAINING INPUTS TO THE FLOW</b>	36
<b>6. SIMULATION IN ANSYS WORK BENCH</b>	37
6.1. Geometry	37
6.2. Meshing	39

6.3. Setup	41
6.4. Solution	41
<b>7. HEAT TRANSFER IN THE GUIDE VANE RADIATORS</b>	42
<b>8. RESULTS</b>	45
<b>9. CONCLUSION</b>	49
9.1. Future Scope	49
<b>REFERENCES</b>	50

# 1.INTRODUCTION

## 1.1 INTRODUCTION TO WIND TUNNEL:

Wind tunnels are large tubes with air moving inside. The tunnels are used to copy the actions of an object in flight. Researchers use wind tunnels to learn more about how an aircraft will fly. Some wind tunnels are big enough to hold full-size versions of vehicles. The wind tunnel moves air around an object, making it seem like the object is really flying.

Most of the time, powerful fans move air through the tube. The object to be tested is fastened in the tunnel so that it will not move. The object can be a small model of a vehicle. It can be just a piece of a vehicle. It can be a full-size aircraft or spacecraft. It can even be a simple object like a tennis ball. The air moving around the still object shows what would happen if the object were moving through the air. How the air moves can be studied in different ways. Smoke or dye can be placed in the air and can be seen as it moves. Threads can be attached to the object to show how the air is moving. Special instruments are often used to measure the force of the air on the object.

The development of wind tunnels accompanied the development of the airplane. Large wind tunnels were built during World War II. Wind tunnel testing was considered of strategic importance during the Cold War development of supersonic aircraft and missiles

The advances in computational fluid dynamics (CFD) modelling on high speed digital computers has reduced the demand for wind tunnel testing. However, CFD results are still not completely reliable and wind tunnels are used to verify CFD predictions.

### 1.1.1. CLASSIFICATION OF WIND TUNNELS:

Wind tunnels may be classified based on any of the following:

- Speed , Mach no:

They are classified as low speed or high speed wind tunnels. In wind tunnel parlance, high speed wind tunnels are those operating at speeds where

compressibility effects are important. They are also classified based on Mach no of operation as subsonic, transonic, supersonic or hypersonic wind tunnels.

- Mode of operation (Pressure storage, in-draft or Pressure vacuum type).
- Kind of test section (T.S) – Open, Closed or Semi enclosed.

### 1.1.1.1. Classification based on Speed, Mach no:

Mach Number :

The Mach number is the ratio of flow velocity after a certain limit of the sounds speed. In simple words it is the ratio of the speed of a body to the speed of sound in the surrounding medium.

$$\text{Mach Number (M)} = u/c$$

where, M is Mach number,

u is local flow velocity (m/s)

c is speed of sound in that medium(m/s)

Speed below the speed of sound is termed as subsonic whereas speed above the speed of sound is termed as supersonic, although scientists practicing aerodynamics most of the time use these terms to describe a particular range of Mach values.

Regime	Subsonic	Transonic	Sonic	Supersonic	Hypersonic	High Hypersonic
Mach No.	<0.8	0.8-1.3	1.0	1.3-5	5-10	>10

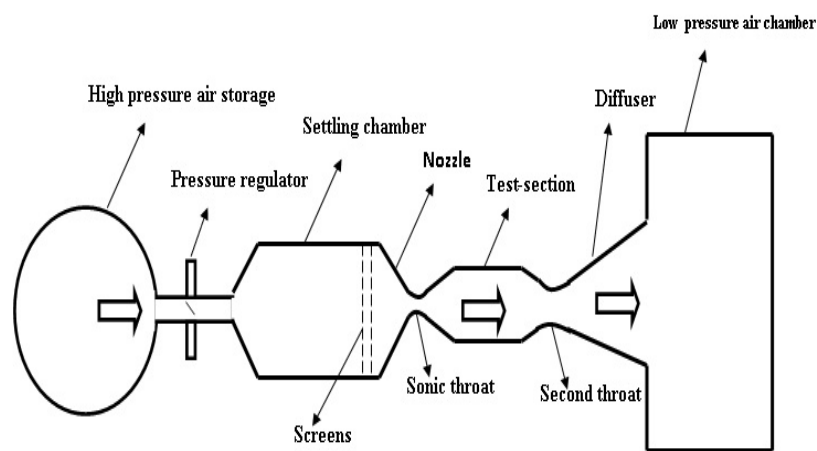
*Table.1.1. Mach speeds*

- Subsonic:  
The Mach is below 0.8.
- Transonic:  
Aircrafts that are built with swept wings. The Mach value is between 0.8-2.1.
- Supersonic:  
The aircrafts created to go supersonic have a definite design it has complete movement of the canards, thin aerofoil sections, and sharp edges. The Mach levels are between 1.2 and 5.0.

- Hypersonic:  
These planes have several distinctive features such as nickel-titanium skin that is cooled and small wings. The Mach values are between 5.0 and 10.0. The U.S. plane X-15 created the world record of flying at Mach 6.72.
- High- Hypersonic:  
The Mach levels are between 10.0 -25.0. When flying at such huge speeds thermal controls becomes an integral portion of the design. The hotness of the surface must be considered beforehand.

### 1.1.1.2 Classification based on Mode of operation:

- Pressure storage type:  
In one such tunnel, air expands from high pressure to the atmospheric pressure where low pressure chamber is excluded (intermittent blow down tunnel). In other cases, the atmospheric air can expand up to a very low vacuum (intermittent in-draft wind tunnel).



*Fig.1.1 Pressure storage type wind tunnel*

- Pressure vacuum type :  
A wind tunnel that is open on both ends and draws air from the room into text section is called open return tunnel. A blow down wind tunnel as a high pressure vessel upstream of test section and a low pressure reservoir downstream of the test section

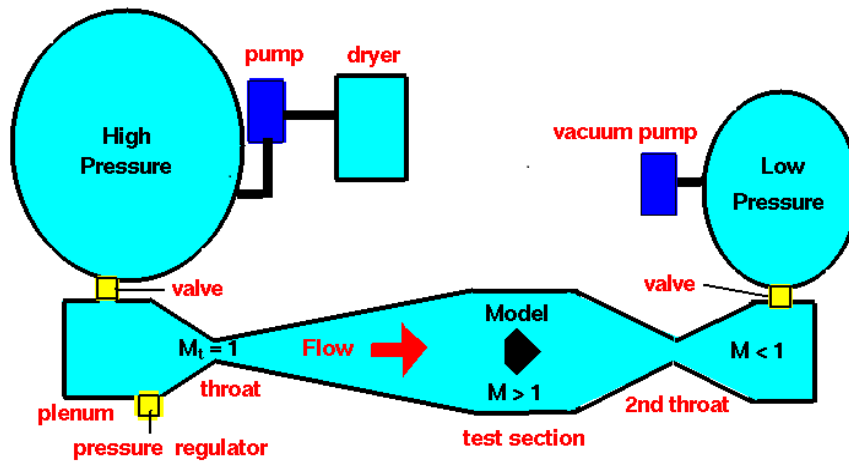


Fig.1.2. pressure vacuum wind tunnel

### 1.1.1.3. Classification based on kind of test section:

- Open loop Wind Tunnel

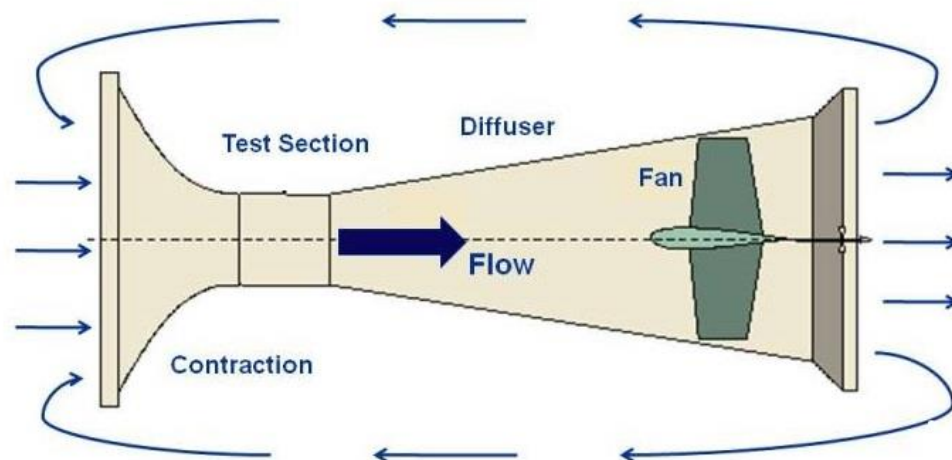
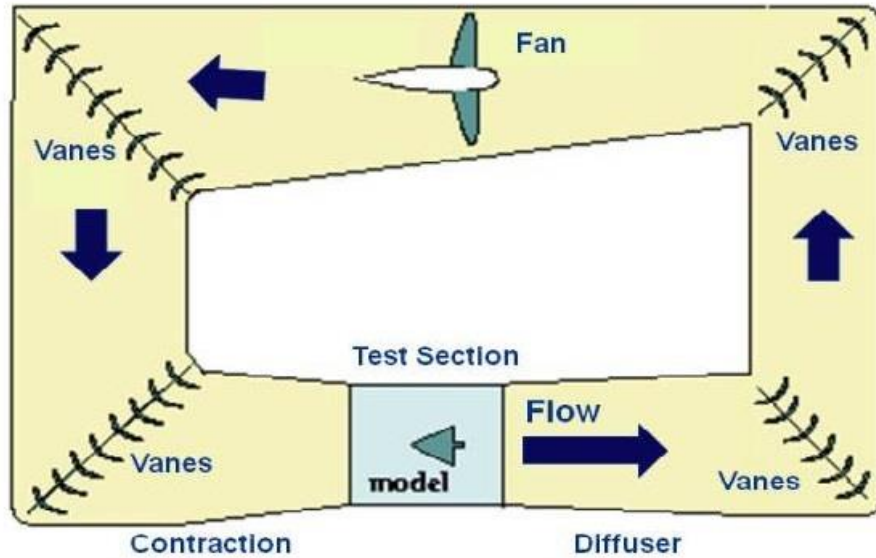


Fig.1.3. Open Loop Wind Tunnel

In the open tunnel, the air that passes through the test section is gathered from the room in which the tunnel is located. The arrows on the figure denote the flow of air through the wind tunnel and around the room.

- Closed Loop Wind Tunnel



*Fig.1.4. Closed loop wind tunnel*

In the closed return tunnel, air is conducted from the exit of the test section back to the fan by a series of turning vanes. Exiting the fan, the air is returned to the contraction section and back through the test section. Air is continuously circulated through the duct work of the closed return tunnel. The arrows on the figure denote the flow of air through the wind tunnel.

Closed return tunnels may operate subsonically or supersonically. Supersonic closed return tunnels are more complex than subsonic tunnels. To prevent condensation in the test section because of low pressure, the air entering the tunnel is often passed over a dryer bed. There is usually an additional throat placed in the tunnel downstream of the test section to shock down the supersonic flow to subsonic.

Advantages of closed loop wind tunnel over open loop wind tunnels:

- Power requirement for a given speed is lower.
- Particulate matter can be contained within the circuit.
- Noise is significantly lower.
- Laboratory air movement (air vents, doors, windows, etc.) does not affect wind tunnel flow.
- Air entering the test section is free of laboratory dust.
- Fan blades are not as vulnerable to damage from model failure.

## **1.1.2. COMPONENTS OF CLOSED LOOP WIND TUNNELS:**

### **1.1.2.1. Settling chamber and flow straightener:**

It mainly comprises of honeycomb and screens as combination. The main function is to reduce the turbulence and straighten the flow only in the axial direction. In this, the air can enter to the tunnel from any directions. But, only the axial flow is desired in the test section. The main purpose of the screen is to reduce the turbulent intensity in the flow and not to allow any unwanted objects to enter the tunnels. The honeycomb can be made with cells or various shapes. These cells are aligned in the stream wise direction in the settling chamber thereby straightens the flow. The honeycomb has a longer length that reduces the transverse velocity components of the flow with minimal pressure drop in the stream wise direction.

### **1.1.2.2. Contraction:**

The prime objective is to accelerate incoming flow from the settling chamber and supplies it to the test section at desired velocity. This section essentially reduces the cross-sectional velocity variation and maintains the flow uniformity. In general, small radius of curvature is used at the entry to this section and curvature of large radius is considered at the exit of the contraction section. However, the boundary-layer separation should be avoided at both the ends of this section. The contraction length is expected to be small so that large contraction area ratios are preferred.

### **1.1.2.3. Test Section:**

It is the basic element of wind tunnel on which all other designs are generally made. All the aerodynamic models are mounted in the test section when the tunnel is operated with desired flow velocity. Various shapes for the test section are considered for constructing the wind tunnel viz. hexagonal, octagonal, rectangle etc. The test section is generally designed on the basis of utility and aerodynamic considerations since cost of construction depends on the test section area. Length of the test section is mostly equal to major



dimension of the cross-section of the same or twice of it. In addition, the test section should also be provided with facilities as per the testing requirement. The test section velocity is generally specified as percentage variation from the average of the cross-section. The ideal test section has steady uniform velocity at the inlet, no cross flow, less or no turbulence and less operating cost.

#### **1.1.2.4. Diffuser:**

It is basically a duct with increase in area attached downstream of the test section. After the test section, it is desired that the air must pass smoothly out of the test section. So, this geometry is made to decrease the flow velocity and increase in pressure. In order to avoid flow reversal, the exit pressure should be higher than the atmospheric in case of open circuit wind tunnel. This is a very critical section in design since the incurred pressure rise reduces the power requirement for the wind tunnel which is proportional to the cube of velocity. Hence maximum pressure recovery to be achieved at least possible distance is the main objective of diffuser design. In general practice, the cone angle of the diffuser is  $7^\circ$  or less so as to avoid boundary layer separation.

#### **1.1.2.5. Turning vanes:**

In a closed-circuit wind tunnel, the air has to circulate in a controlled manner. Typically, the corners of the wind tunnel are of two bends. These corners are provided with turning vanes for smooth passage of the flow. Chambered aerofoils of bent planes are generally accepted as the turning vanes. These vanes are purposely made as adjustable for smooth operation thereby avoiding under/over tuning.

#### **1.1.3. HOW IT WORKS:**

Air is blown or sucked through a duct equipped with a viewing port and instrumentation where models or geometrical shapes are mounted for study. Typically the air is moved through the tunnel using a series of fans. The airflow created by the fans that is entering the tunnel is itself highly turbulent due to the fan blade motion

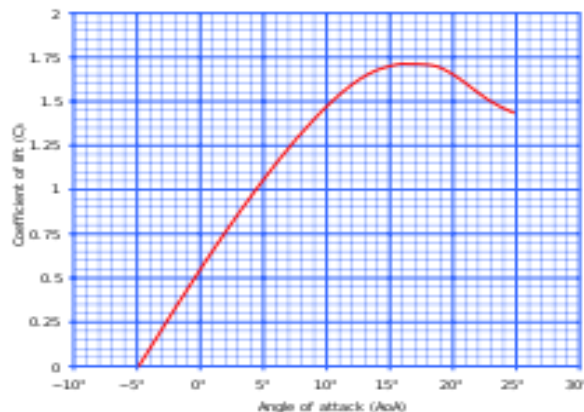
and so is not directly useful for accurate measurements. The air moving through the tunnel needs to be relatively turbulence-free and laminar. To correct this problem, closely spaced vertical and horizontal air vanes are used to smooth out the turbulent airflow before reaching the subject of the testing. In closed loop wind tunnel, as the same air is made to flow throughout the tunnel continuously, temperature increases. In order to reduce the effects caused due to temperature increase, necessary cooling technique has to be followed.

#### 1.1.4. MEASUREMENTS MADE USING WIND TUNNELS:

##### 1.1.4.1. Pressure measurements:

Pressure across the surfaces of the model can be measured if the model includes pressure taps. This can be useful for pressure-dominated phenomena, but this only accounts for normal forces on the body.

##### 1.1.4.2. Force and Moment measurements:



*Fig.1.5. Typical lift coefficient versus angle of attack curve*

With the model mounted on a force balance, one can measure lift, drag, lateral forces, yaw, roll, and pitching moments over a range of angle of attack. This allows one to produce common curves such as lift coefficient versus angle of attack.

Note that the force balance itself creates drag and potential turbulence that will affect the model and introduce errors into the measurements. The supporting structures are therefore typically smoothly shaped to minimize turbulence.

#### 1.1.5. APPLICATIONS OF WIND TUNNELS:

#### **1.1.5.1. Aerodynamics applications:**

Wind tunnels provide a concentrated stream of air in a vertical or horizontal flow. Quite often wind tunnels are used in the testing of products, such as seen in the automotive industry, to develop aerodynamic designs. Wind tunnels are also utilized for recreational applications such as indoor skydiving facilities.

#### **1.1.5.2. Non aerodynamic:**

Drag is often the component that receives the greatest attention as it has dominant effect on fuel consumption at a given speed and on the top speed attainable. The lift force is of extreme importance in determining controllability for performance cars and race cars, becoming more critical as the speed increases.

They are also used in mechanical engineering, civil engineering, automobile engineering and calibration of instruments.

## 1.2. INTRODUCTION TO COMPUTATIONAL FLUID DYNAMICS

### 1.2.1. CFD

CFD is one of the branches of fluid mechanics that numerical methods and algorithms to solve and analyze problem that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluid 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.

### 1.2.2. GOVERNING EQUATIONS

The governing equations of fluid flow represent mathematical statements of the conservation laws of physics. Each individual governing equation represents a conservation principle. The fundamental equations of fluid dynamics are based on the following universal laws of conservation. They are

Unsteady state 3-D equation of continuity:-

$$\frac{\partial \rho}{\partial t} + \text{div} (\rho u) = 0$$

Momentum equation:-

$$\frac{\partial (\rho u)}{\partial t} + \text{div} (\rho u u) = -\frac{\partial p}{\partial x} + \text{div} (\mu \text{grad} u)$$

Energy equation:-

$$\frac{\partial (\rho i)}{\partial t} + \text{div} (\rho I u) = -p \text{div} u + \text{div} (k \text{grad} T) + \phi + S$$

### 1.2.3. DISCRETISATION METHODS OF CFD

There are three discretization methods in CFD:

1. Finite difference method (FDM)
2. Finite volume method (FVM)
3. Finite element method (FEM)

### **1.2.3.1. 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 a discrete representation of the solution.

### **1.2.3.2 Finite volume Method (FVM)**

A finite volume method (FVM) discretization is based upon an integral form of the PDE to be solved for a given finite volume (or cell). The computational domain is discretized into finite volume and then for every volume the governing equations are solved. 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 structural grids, and the effort to convert the given mesh into structural numerical grid internally is completely avoided. As with FDM, the resulting approximate solution is a discrete, but the variables are typically placed at cell canthers rather than at nodal points. This is not always true, as there are also face-cantered finite volume methods. In any case, the valves of field variables at non-storage locations (e.g. Vertices) are obtained using interpolation.

### **1.2.3.3 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 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 restarting 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, 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 continuous (up to a point ) solution. FVM and FDM are generally considered easier to program than FEM, but opinions vary on this point. FVM re generally expected to provide better conservation properties, but opinions vary on this point also.

#### **1.2.4. 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 interface input problem parameters and the results. Hence all codes contain three main elements:

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

##### **1.2.4.1. 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 large 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.

#### **1.2.4.2. Solver:**

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

FLUENT is used in most industries. Flo Wizard is the first general-purpose rapid flow modeling tool for design and process engineers build 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 numeric), similar to those used in mechanical analysis, to discretize the domain. In contrast, the FLUENT solver uses finite volumes (cell centered numeric). 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 modeling 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 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, 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 center; the face values must be interpolated from the cell center 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.

#### **1.2.4.3. 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 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, Particle tracking, View manipulation (translation, rotation, scaling etc.).

### **1.2.5. ADVANTAGES AND DISADVANTAGES**

- No restriction to linearity.
- Complicated physics can be treated.
- Time evaluation of flow can be obtained.
- It has the potential of providing information not available by other means.
- Computational investigation can be performed with remarkable speed. Designer can study the implications of hundreds of different configurations in minimum time and choose the optimum design.
- It gives detailed and complete information. It can provide the values of all the relevant variables (pressure, velocity, temperature, concentration, turbulence) throughout the domain of interest.
- Truncation errors.
- Boundary condition problems.
- Computer costs.
- Computer storage & speed.

### **1.2.6. APPLICATIONS OF CFD**

- Aerodynamics of aircraft & Space vehicles: lift & drag.
- Hydrodynamics of ships.
- Weather prediction.

### **1.2.7. PREDICTING THE TURBULENT VISCOSITY**

The following models can be used to predict the turbulent viscosity:

- Mixing length model.
- Spalart – Allmaras model.
- Standard k- $\epsilon$  model.

- k-ε Realizable model.
- k-ω Model.

### 1.2.7.1. The k-ε model:

- The k-ε model focuses on the mechanisms that affect the turbulent kinetic energy (per unit mass) k.
- The instantaneous kinetic energy k(t) of a turbulent flow is the sum of mean kinetic energy and turbulent kinetic energy k :

$$K = \frac{1}{2}(U^2 + V^2 + W^2)$$

$$k = \frac{1}{2}(\overline{u'^2} + \overline{v'^2} + \overline{w'^2})$$

$$k(t) = K + k$$

ε is the dissipation rate of k.

- If k and ε are known, we can model the turbulent viscosity as:

$$\nu_t \propto 9\lambda \propto k^{1/2} \frac{k^{3/2}}{\varepsilon} = \frac{k^2}{\varepsilon}$$

- We now need equations for k and ε.

### 1.2.7.2. Turbulent kinetic energy k:

The equation for the turbulent kinetic energy k is as follows:

Here  $e_{ij}'$  is fluctuating component of rate of deformation tensor.

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k U) = \text{div}(-\overline{\rho' u'}) + 2\overline{\mu u' e_{ij}'} - \rho \frac{1}{2} \overline{u_i' u_i' u_j'} - 2\overline{\mu e_{ij}' e_{ij}'} + (-\overline{\rho u_i' u_j'} \cdot E_{ij})$$

$$(1) \quad (2) \quad (3) \quad (4) \quad (5) \quad (6) \quad (7)$$

The equation can be read as:

- (1) The rate of change of k, plus
- (2) Transport of k by convection, equals
- (3) Transport of k by pressure, plus
- (4) Transport of k by viscous stresses, plus
- (5) Transport of k by Reynolds stresses, minus
- (6) Rate of dissipation of k, plus
- (7) Turbulence production.

### 1.2.7.3. Turbulent dissipation:

The equations look quite similar.

However, the  $k$  equation mainly contains primed quantities, indicating that changes in  $k$  are mainly governed by turbulent interactions.

Furthermore, term (7) is equal in both equations. But it is actually negative in the  $K$  equation (destruction) and positive in the  $k$  equation: energy transfers from the mean flow to the turbulence.

The viscous dissipation term (6) in the  $k$  equation

$$K = -2\mu \overline{e_{ij}' e_{ij}'}$$

describes the dissipation of  $k$  because of the work done by the smallest eddies against the viscous stresses.

We can now define the rate of dissipation per unit mass  $\epsilon$  as:

$$\epsilon = 2\nu \overline{e_{ij}' e_{ij}'}$$

## 2.LITERATURE REVIEW

Various wind tunnels of close loop type have been existed and being developed for applications in aerodynamics and wind engineering. Now-a-days these are being applied for experimentation of PCB cooling.

Recent developments in computational methods have significantly increased the use of numerical models. (Ghani et al., 2001; Launder and Spalding, 1972; Gartmann et al., 2011). Numerical methods are often used jointly with physical experimentation such as wind tunnel studies for validation and assessment of data obtained by simulation. However, the use of CFD to support wind tunnel design has remained very limited and has been situated in other research areas (Moonen et al., 2006; Moonen et al., 2007). Several works on the evaluation of wind tunnel systems using numerical models are highlighted in this section

John Kaiser Calautit (2014) Validated a design methodology for a closed loop subsonic wind tunnel where a systematic investigation into the design and simulation of flow parameters in a closed-loop wind tunnel was carried out using Computational Fluid Dynamics (CFD) where the analytical model for estimating pressure losses were directed as input boundary conditions. Full-scale model of the entire wind tunnel was considered instead of the conventional approach, in which only test section flow is simulated. This allowed for optimization of flow quality not only in the test section but also the flow in the entire circuit. He validated the importance of guide vanes for upstream and downstream flow in a closed loop wind tunnel.

Moonen et al.(2006) established a methodology for numerically modelling the flow conditions in a closed-circuit low speed wind tunnel system. Steady-state and three-dimensional CFD simulations, using standard and realizable k-epsilon equations models, were carried out to determine the total pressure loss and flow rate in the test section. Grid sensitivity analysis was performed to verify the grid independence of the simulation of results for both the cases. The study showed that the accuracy of the results from the full wind tunnel model simulation was 2 to 4 times better than the conventional CFD analysis of only the test section.

Gordon and Imbabi (1998) used CFD to simulate the flow with in critical sections of the closed-loop wind tunnel at the University of Aberdeen. The study aimed to reduce

the overall cost and size of the wind tunnel, facilitate the use of alternative working fluids and at the same time maintain the high quality of the flow in the test section. The results showed that the CFD methods played a significant part in classifying the required modifications to achieve compact and cost effective wind tunnel design.

Moonen et al.(2007) investigated the flow quality in the test section of a closed loop wind tunnel test using numerical analysis. The work established six corresponding indices for the evaluation of the three-dimensional flow quality in the test section. The study highlighted the effect of adding guide vanes and mesh screens in the contraction nozzle to improve the flow quality in the working section. The findings depicted that the vertical guide vanes reduced the skewness and angularity, yielding a higher quality flow while the horizontal guide vanes were found to have a slightly negative impact.

Chad Harris et al.(2000) Where a cross flow micro heat exchanger was designed to maximize heat transfer from a liquid (water-glycol) to a gas (air) for a given frontal area while holding pressure drop across the heat exchanger of each fluid to values characteristic of conventional scale heat exchangers. The predicted performance for these plastic, ceramic, and aluminum micro heat exchangers are compared with each other and to current innovative car radiators. The cross flow micro heat exchanger can transfer more heat/volume or mass than existing heat exchangers within the context of the design constraints specified which can be applied in a wide range of applications (automotive, home heating, and aerospace) where the basic idea for introducing a radiator was visualized.

Muhammad Mahmood Aslam Bhutta et al (2012) Where the applications of Computational Fluid Dynamics (CFD) in the field of heat exchangers. It has been found that CFD has been employed for the following areas of study in various types of heat exchangers: fluid flow maldistribution, fouling, pressure drop and thermal analysis in the design and optimization phase. Different turbulence models available in general purpose commercial CFD tools i.e. standard, realizable and RNG  $k - \epsilon$  RSM, and SST  $k - \epsilon$  in conjunction with velocity-pressure coupling schemes such as SIMPLE, SIMPLEC, PISO etc. have been adopted to carry out the simulations. The quality of the solutions obtained from these simulations are largely within the

acceptable range proving that CFD is an effective tool for predicting the behavior and performance of a wide variety of heat exchangers.

From the previous related work, different types of wind tunnels have been studied using numerical CFD analysis. The numerical simulation and experimental results demonstrated the importance of the use of CFD in assessing the performance of a wind tunnel system.

We took a liberty of designing and simulating a conceptual triangular shaped closed loop wind tunnel with guide vane radiator which have the flexibility in the functioning like a guide vane as well as radiator for cooling so that it overcomes the disadvantage of heat generated in the wind tunnel due to repeated circulating air on the other hand decreasing the overall head loss in the wind tunnel.

### 3.WHY 45° CORNER WIND TUNNEL?

A regular rectangular shaped closed loop wind tunnel has four corners and a radiator to cool down the heat generated due to the continuously flowing fluid. The major head losses are formed at the four corners and due to the radiator.

In order to reduce the head losses formed due to the four corners, the design is modified to two-corner (45° each) wind tunnel of triangular type. Due to this geometry, the 45° corner has more head loss compared to a 90° corner. Since there are less number of corners in a triangular shaped wind tunnel (2) when compared to a rectangular one (4), the overall head loss can be compensated.

Guide vanes play an important role in the uniformity of fluid flow especially at the corners. Non-uniform flow of fluid in the wind tunnel effects the results in the test chamber during experimentation.

#### 3.1. GEOMETRY OF 45° CORNER AND 90° CORNER:

A 45° corner and a 90° corner are developed such that the inlet and outlet of both the corners comprises the same length and the area of the flow is also the same.

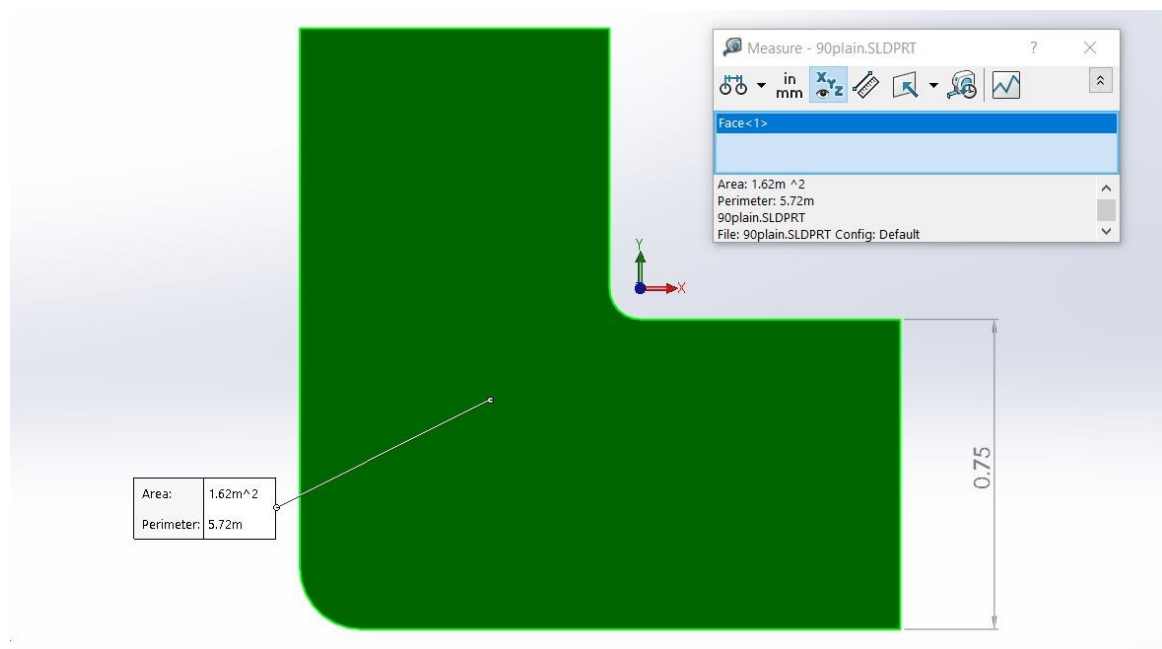
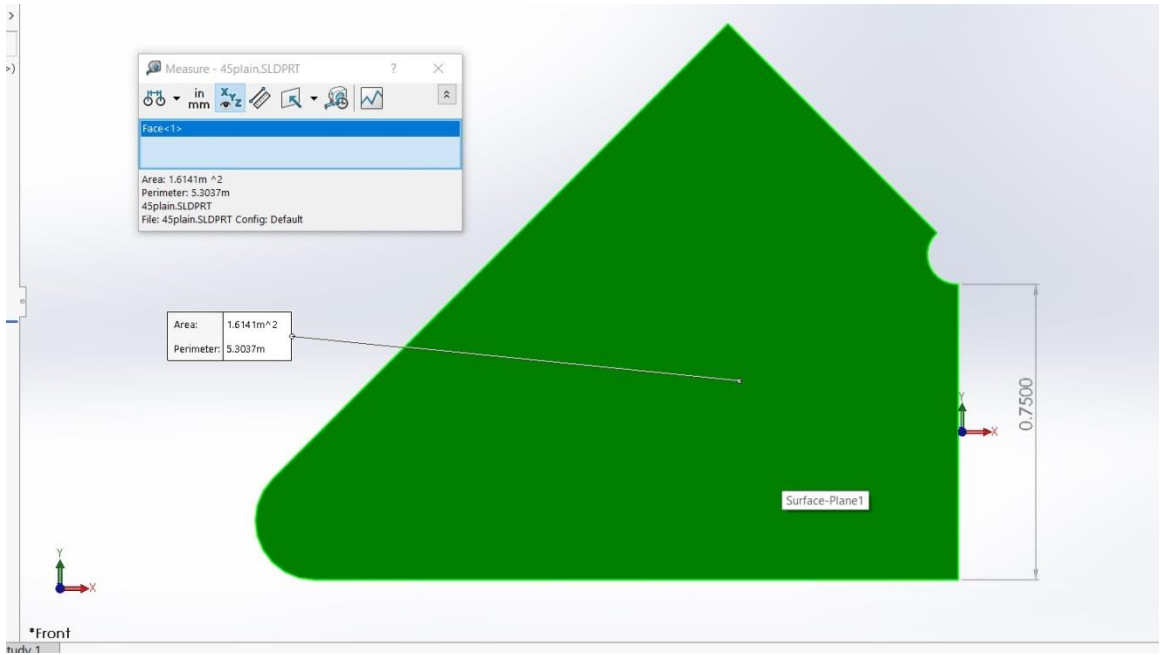
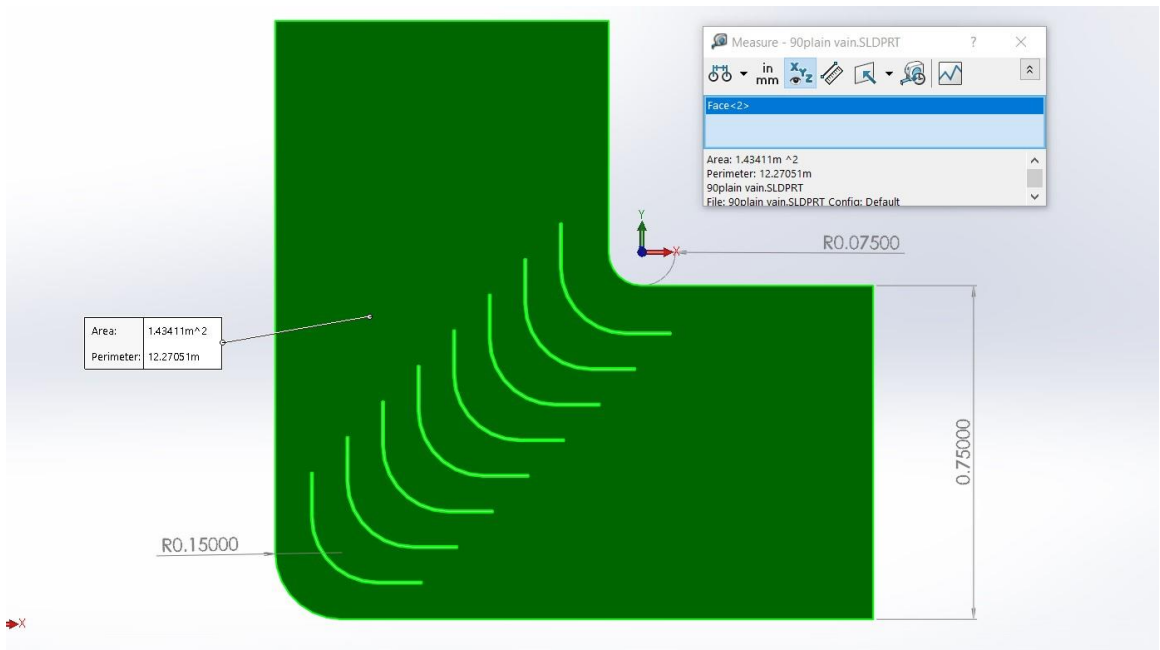


Fig.3.1. 2D geometry of 90° corner





*Fig.3.2. 2D geometry of 45° corner*



*Fig.3.3. 2D geometry of 90° corner with guide vanes*

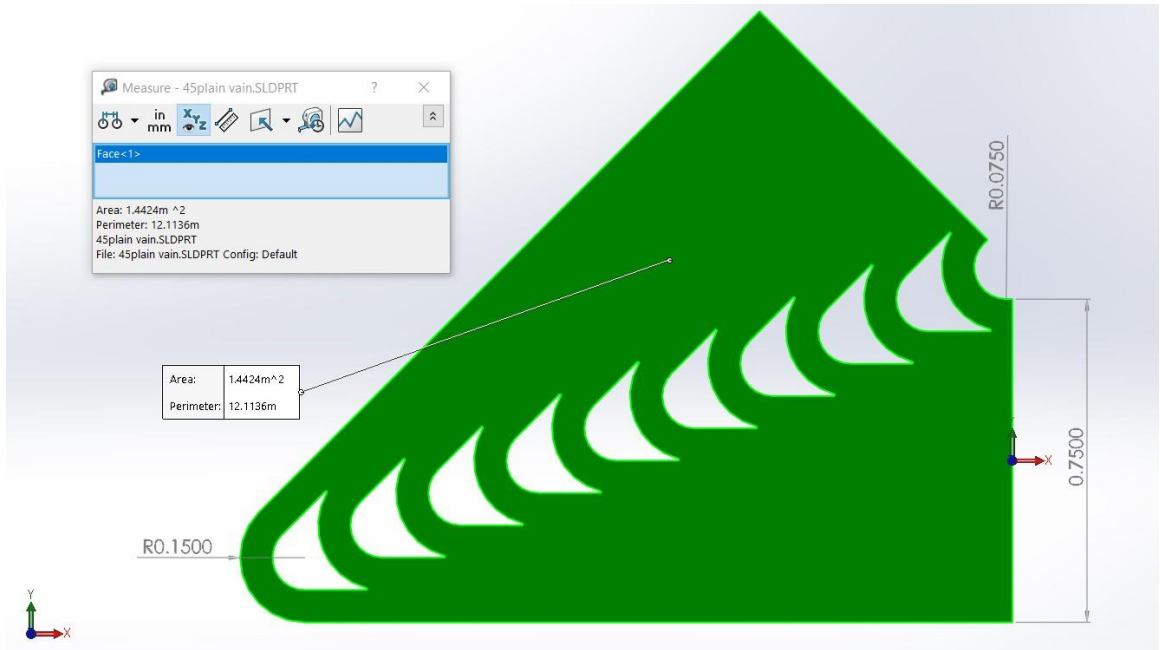


Fig.3.4. 2D geometry of 45° corner with guide vanes

### 3.2. AIR FLOW WITH AND WITHOUT GUIDE VANES:

Taking the same boundary conditions for both the corners, flow simulation has been done to get the flow properties using ANSYS FLUENT with and without guide vanes.

Summary of CFD models boundary conditions.

Parameter	Set value
Discretization scheme	Second-order upwind
Algorithm	SIMPLE
Time	Steady state
Velocity inlet	0.5 m/s
Pressure outlet	0 pa
Gravity	-9.81 m/s

Table.3.1. Boundary conditions

The velocity contours obtained are as follows:

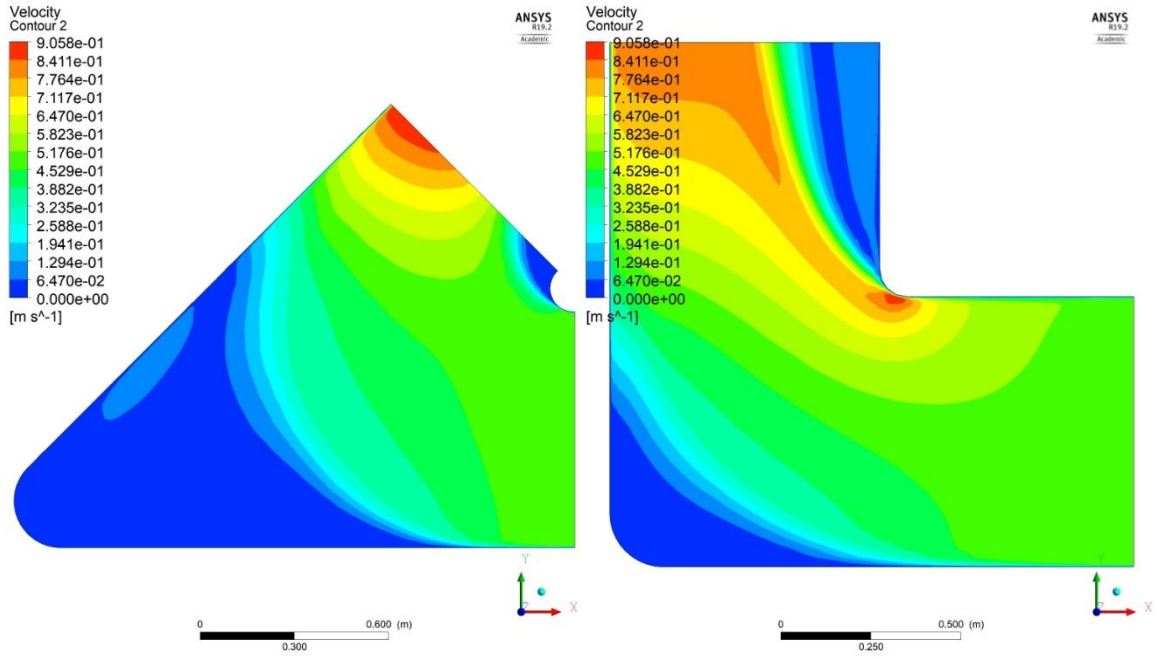


Fig.3.5. Without guide vanes

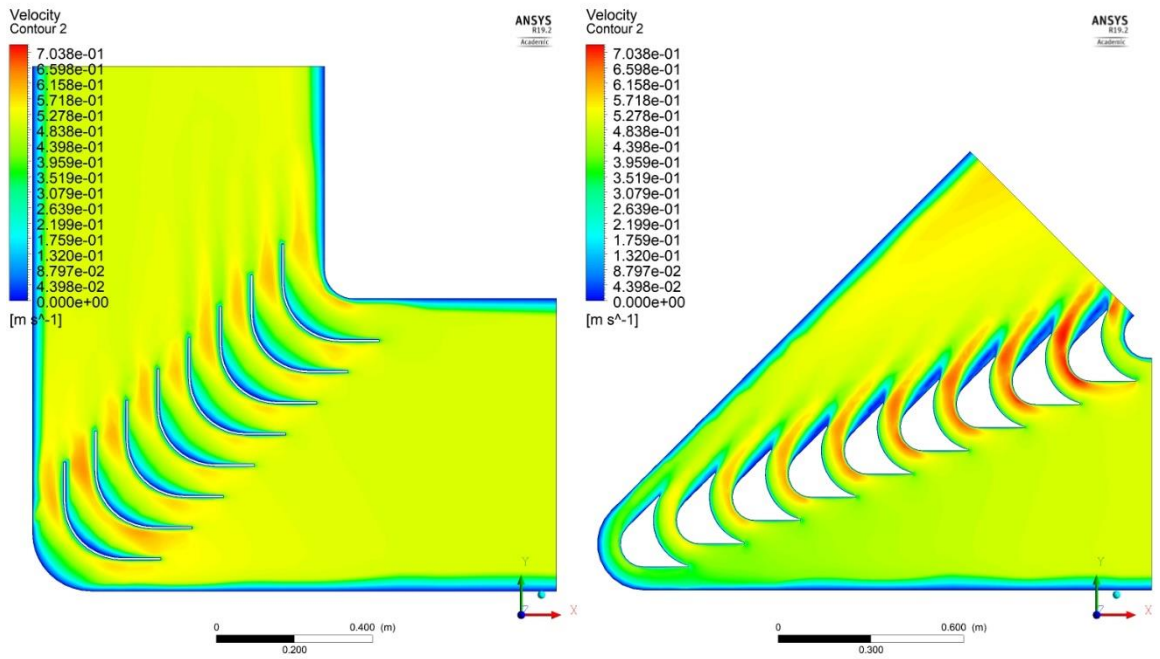


Fig.3.6. With guide vanes

From the results, corners with guide vanes produce uniform flow which is essential for wind tunnel.

### 3.3. HEAD LOSS AT THE CORNERS:

Pressure simulations for both the corners with guide vanes are done in Results-CFD-post and functional calculator is used to find average pressures at inlet and outlet of 2D corners for head losses calculations.

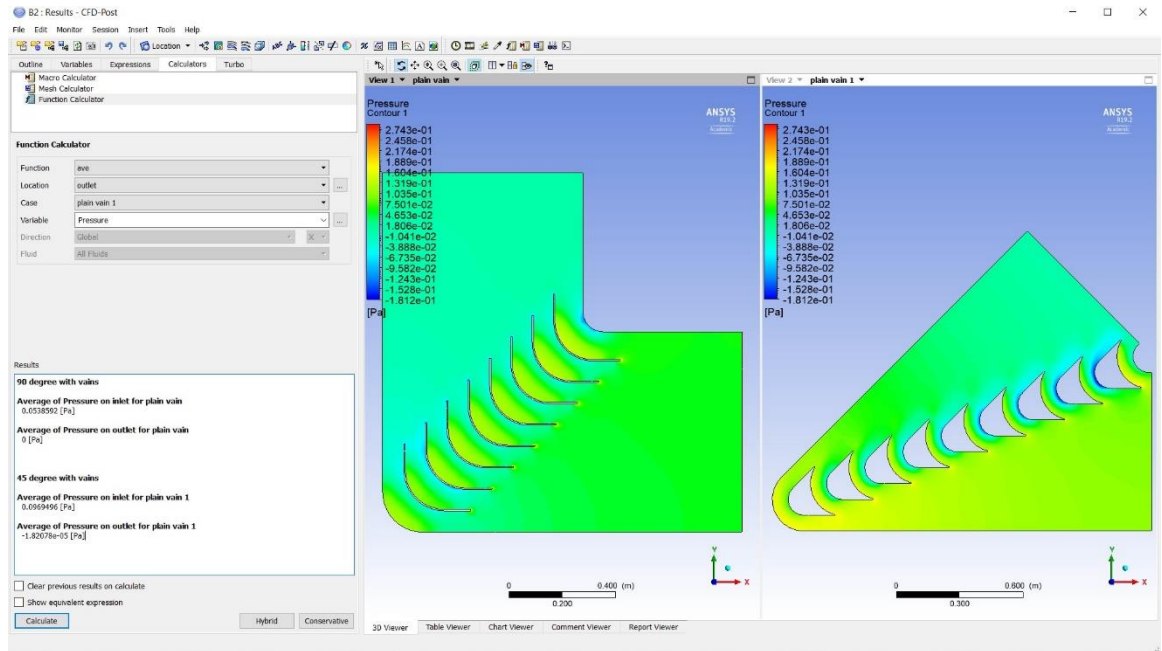


Fig.3.7. Pressure calculation by function calculator

90° simulation	45° simulation
Average of Pressure on inlet	Average of Pressure on inlet
$P_1 = 0.0538592 \text{ pa}$	$P_1 = 0.0969496 \text{ pa}$
Average of Pressure on outlet	Average of Pressure on outlet
$P_2 = 0 \text{ pa}$	$P_2 = -1.82078 \cdot 10^{-5} \text{ pa}$
head loss = $\frac{(P_1 - P_2)}{\rho * g}$	head loss = $\frac{(P_1 - P_2)}{\rho * g}$
$\rho = \text{density of air } 1.225 \text{ kg/m}^3$	$\rho = \text{density of air } 1.225 \text{ kg/m}^3$
$g = \text{gravitational acceleration } 9.81 \text{ m/s}$	$g = \text{gravitational acceleration } 9.81 \text{ m/s}$
head loss = $0.0044818 \text{ m}$	head loss = $0.00806905 \text{ m}$

Table.3.2. Overall head loss

Since there are 4 corners in rectangular wind tunnel the overall head loss due to corners is 4 times the head loss of a 90° bend and it will also have radiator head loss if it is introduced.

$$\begin{aligned}90^\circ \text{ Overall head loss} &= 4 * \text{head loss due to } 90^\circ \text{ corner} + \text{radiator head loss} \\ &= 4 * 0.0044818 + \text{radiator head loss} \\ &= 0.0179272 \text{ m} + \text{radiator head loss}\end{aligned}$$

But in the conceptual triangular shaped wind tunnel will have 3 corners in which two of them were 45° corners the other corner is compensated by a centrifugal blower for circulation of air where there is an advantage of involving radiator within the guide vane itself. So overall head loss due to corners is 2 times the head loss of a 45° bend.

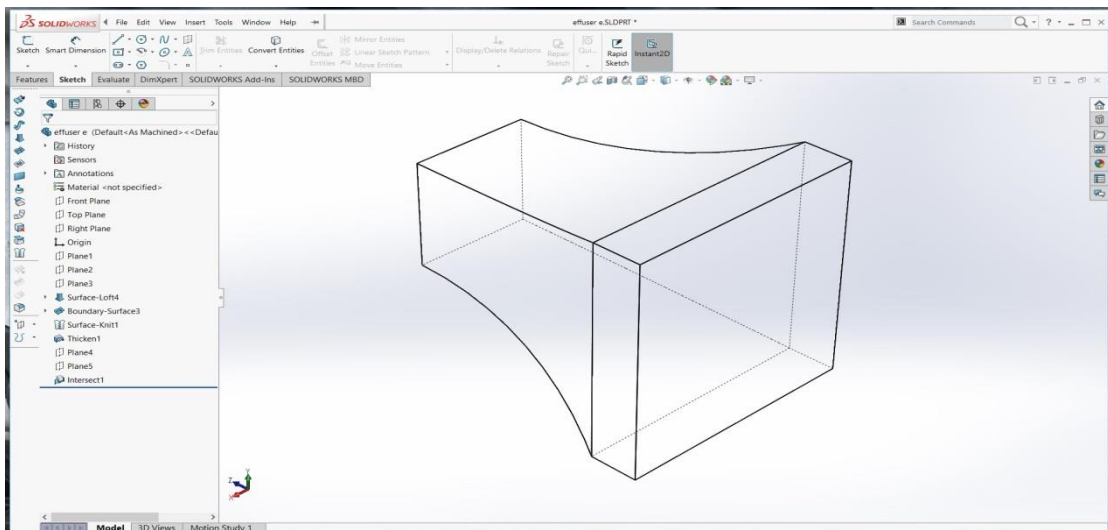
$$\begin{aligned}45^\circ \text{ Overall head loss} &= 2 * \text{head loss due to } 45^\circ \text{ corner} \\ &= 2 * 0.00806905 \\ &= 0.0161381 \text{ m}\end{aligned}$$

Here as the overall head loss in 45° corner is less than the overall head loss in 90° corner, the 45° is preferred to reduce the overall head loss.

## 4.DESIGN PROCEDURE IN SOLID WORKS

### 4.1. DESIGN OF EFFUSER:

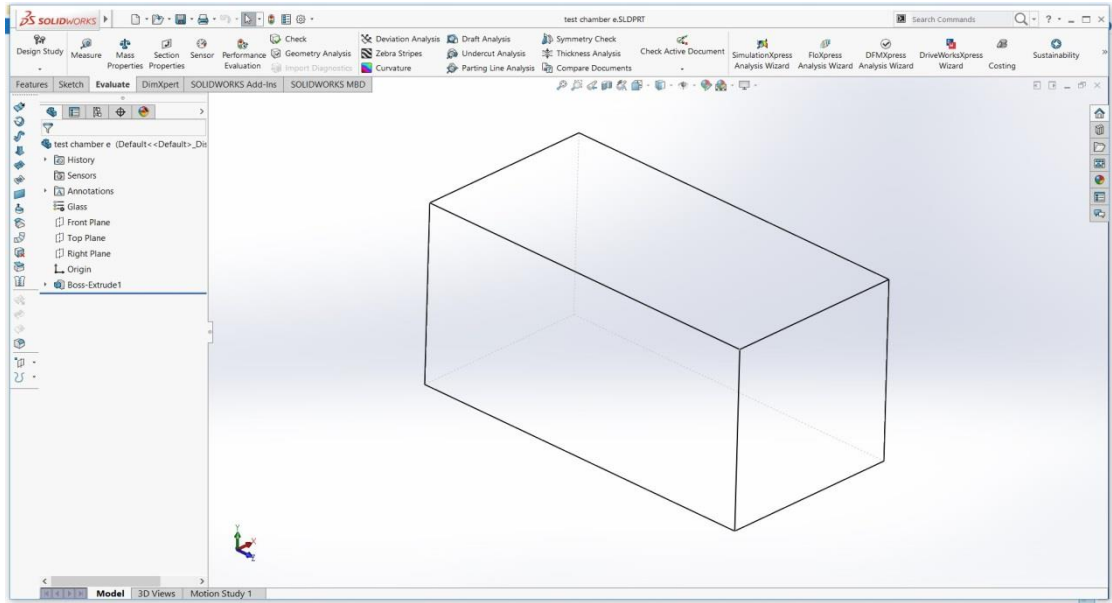
- In SolidWorks, select the right plane and develop a square of  $1 \times 1 \text{ m}^2$  and at a distance of 1.3 m, develop another square of  $0.5 \times 0.5 \text{ m}^2$ .
- Now develop a plane such that it should cut the two squares diagonally. In that plane join the two squares at the corner points by a curve on both sides where the plane cuts the two squares.
- Then the figure has to be extruded to get the Effuser.
- A provision has to be developed by extruding the bigger square to a length of 0.2m.



*Fig.4.1.Effuser*

### 4.2. DESIGN OF TEST SECTION:

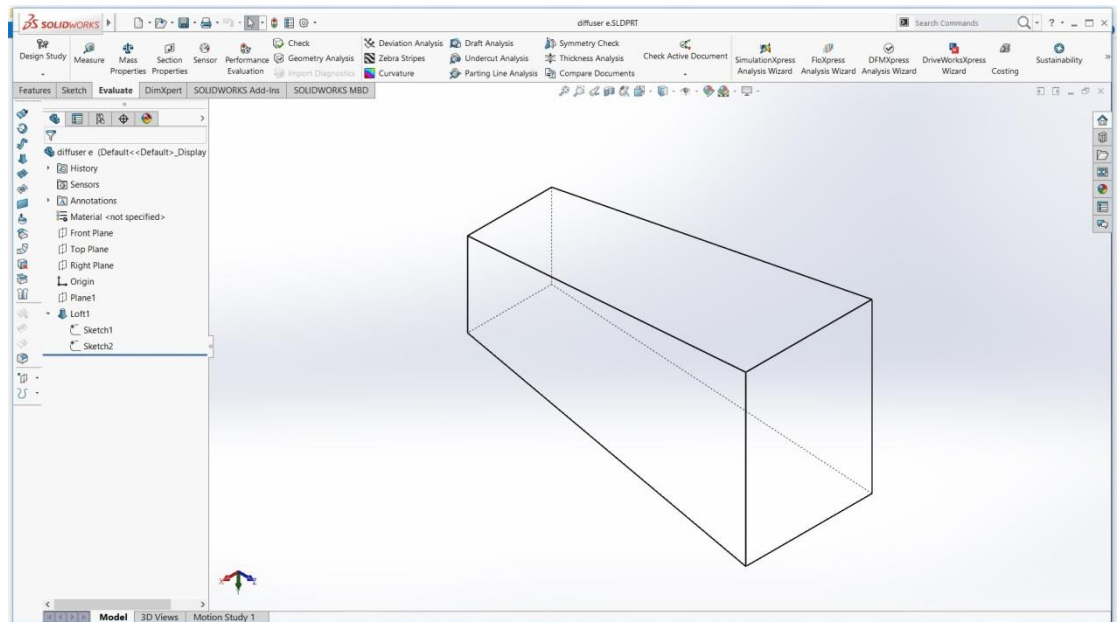
- In the right plane, develop a square of  $0.5 \times 0.5 \text{ m}^2$  and extrude it to 1m.



*Fig.4.2. Test chamber*

#### **4.3. DESIGN OF DIFFUSER:**

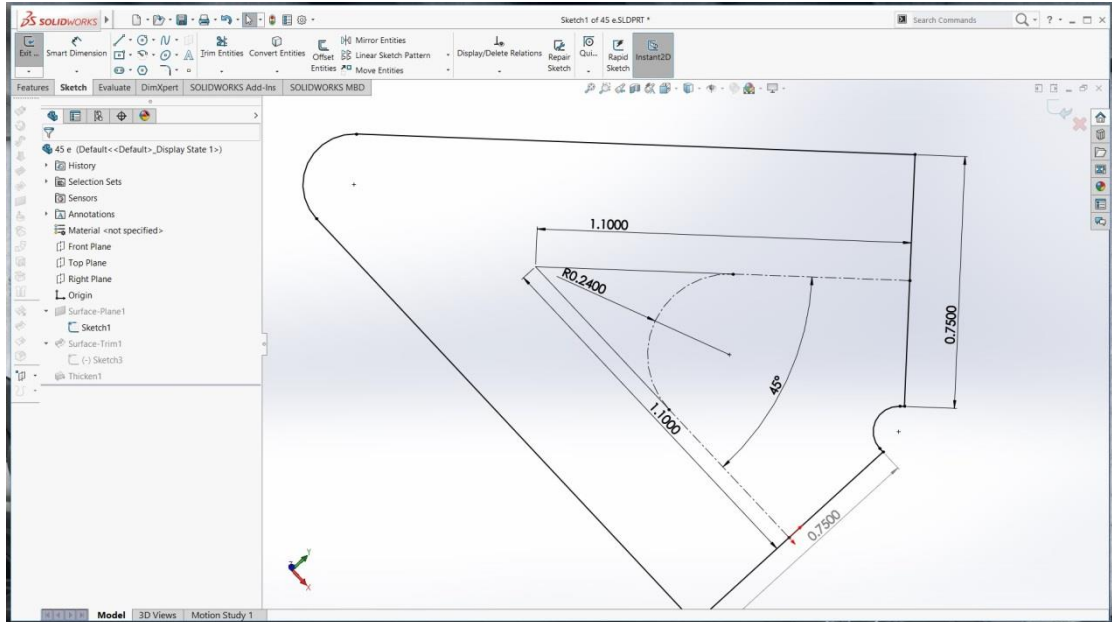
- In the right plane, develop a square of  $0.5 \times 0.5 \text{ m}^2$  and another rectangle of  $1 \times 0.75 \text{ m}^2$  and extrude it such that it has an area ratio of 1:3 and an included angle of  $8^\circ$  and  $4^\circ$ .



*Fig.4.3. Diffuser*

#### **4.4. DESIGN OF 45° CORNER:**

- Develop the outline of the vane in such a way that the inlet and outlet are of lengths 0.75m, their perpendiculars coincided at 45° to each other and the inlet and outlet are joined by arcs of radii 0.075m and 0.150m as shown.



*Fig.4.4. 45°corner design step 1*

- For the guide vane radiators, the available length of inlet and outlet (0.75m) is divided into 10 equal parts each of length 0.075m.
- Develop a provision for guide vane radiator taking 0.15m as lower circle radius and center as that point which is used for developing an arc of 0.075m radius (used for joining the inlet and outlet) .The radius of the upper arc of the guide vane radiator should be taken as 0.075m and a suitable center such that all the nine vanes are equally spaced. The two obtained arcs are then joined by lines. A chamfer of radius 0.005 is given at the corners to obtain the guide vane radiator.
- Similarly provisions for all the nine guide vanes are developed.



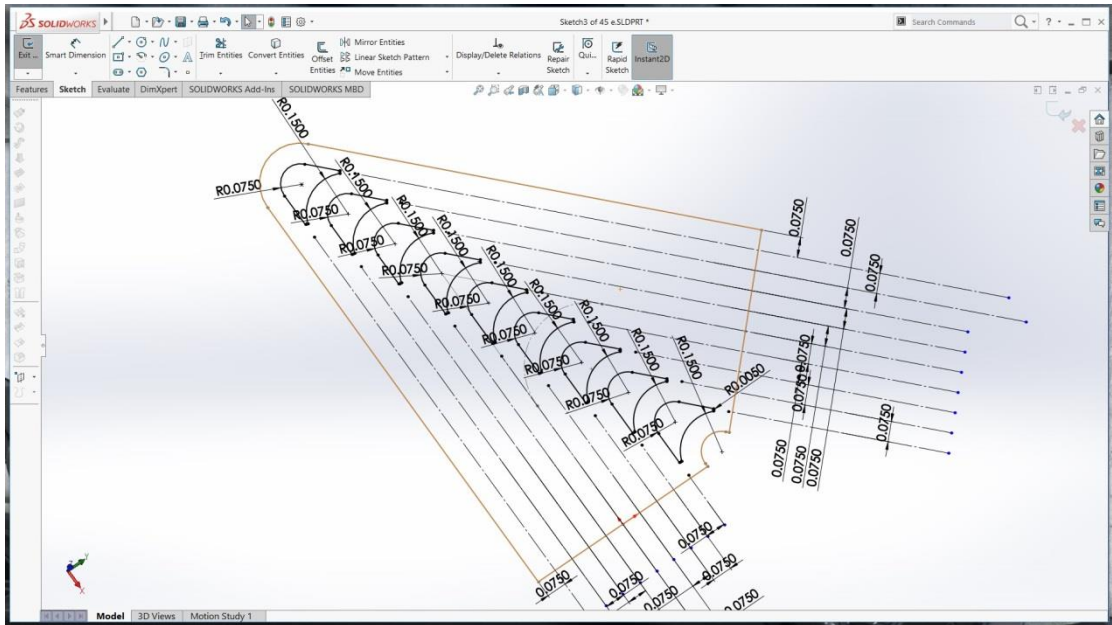


Fig.4.5. 45°corner design step 2

- The obtained figure is extruded to 1m to get the 45° vane and extrude cut to get the provisions for guide vane radiators.

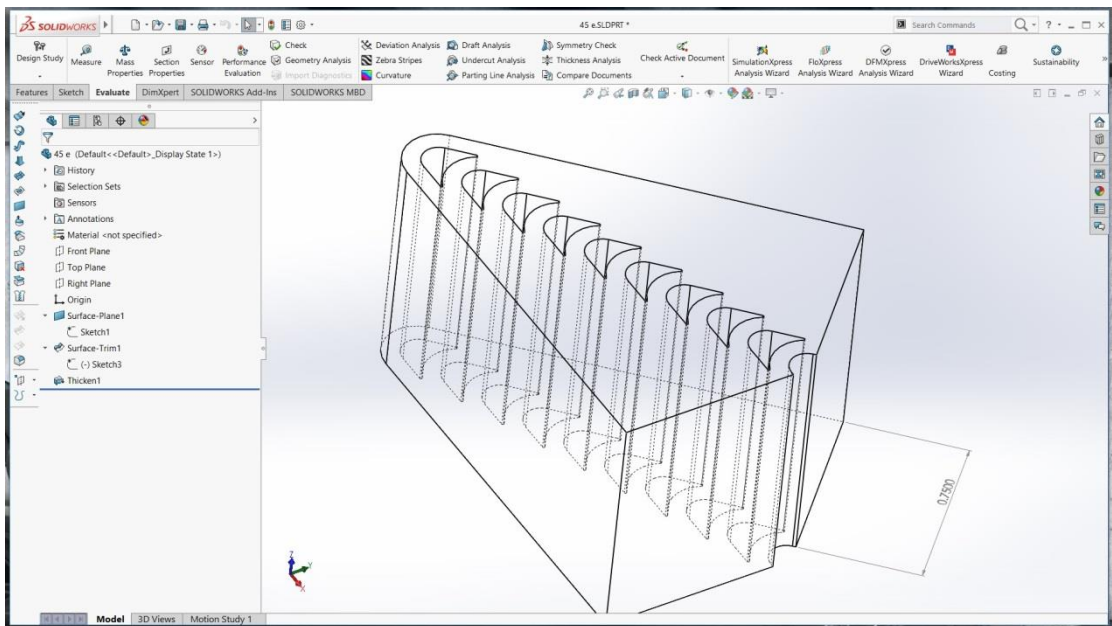


Fig.4.6. 45°corner design step 3

#### 4.5. GUIDE VANE RADIATOR:

- As mentioned earlier, a guide vane radiator is developed with the given dimensions.

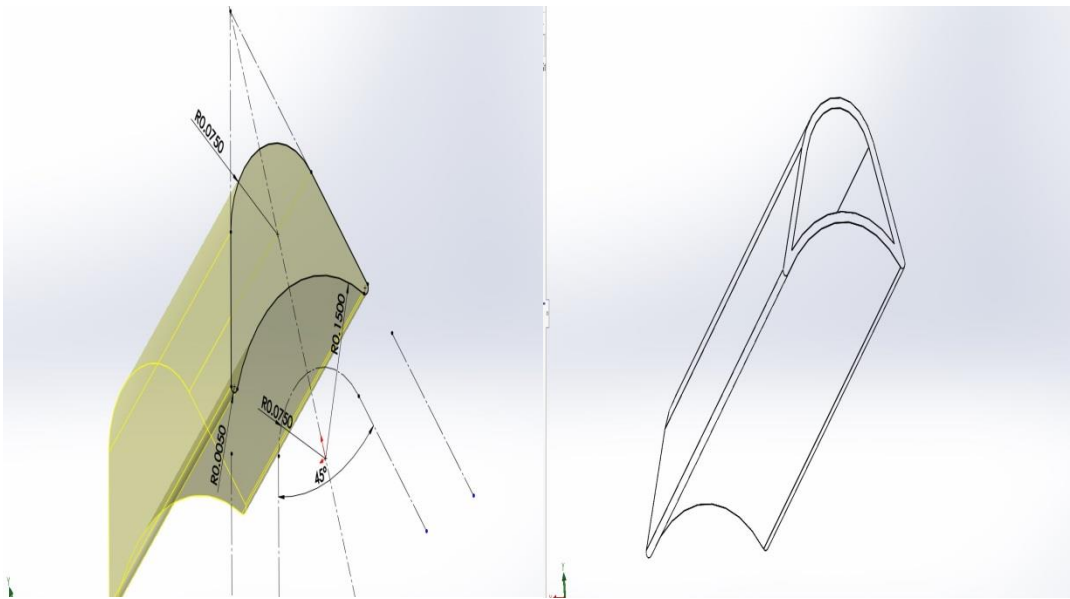


Fig.4.7. Individual radiator

- The guide vane radiators are mated with the 45° vane to get the final Guide Vane Radiator assembly.

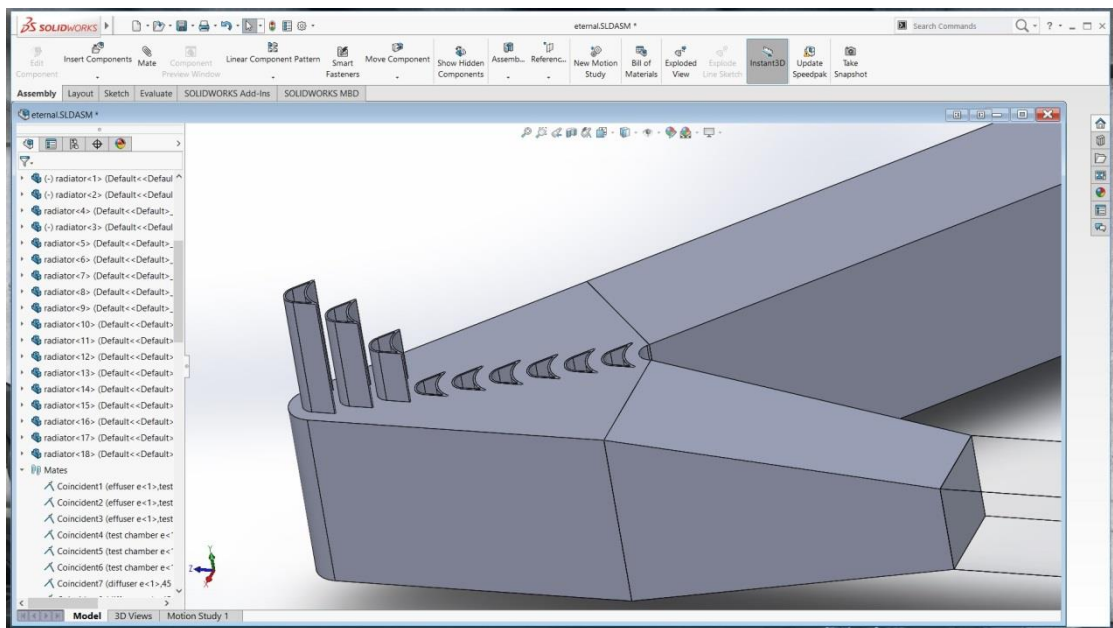
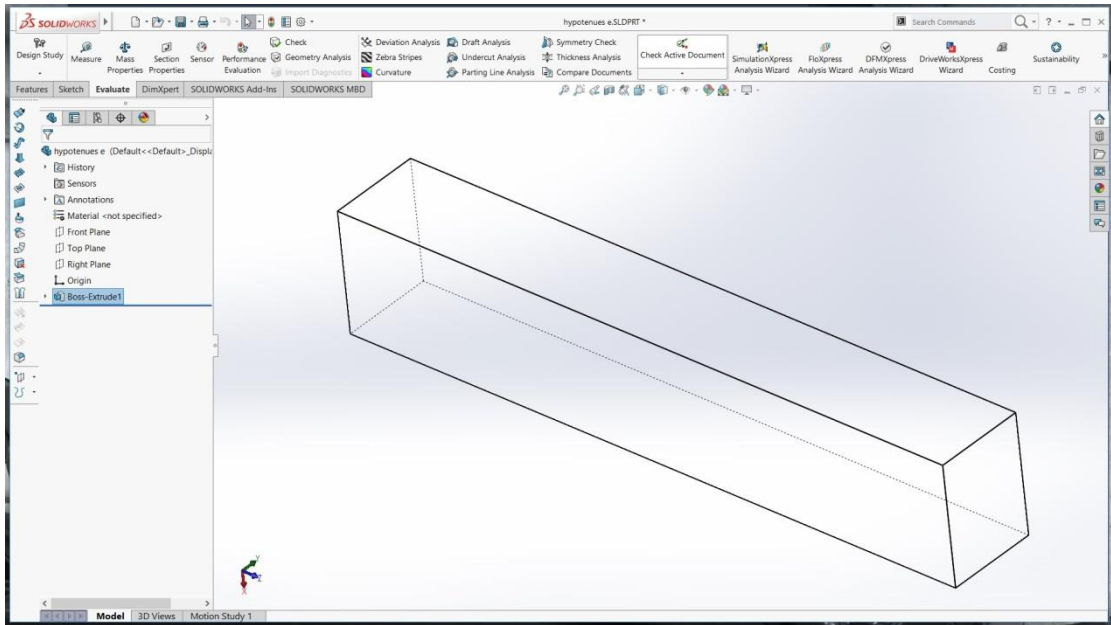


Fig.4.8. Assembly 1

#### 4.6. HYPOTENUSE DUCT:

- A rectangle of  $0.75 \times 1 \text{ m}^2$  is drawn in the top plane and extruded to 5.4m to obtain the hypotenuse duct.



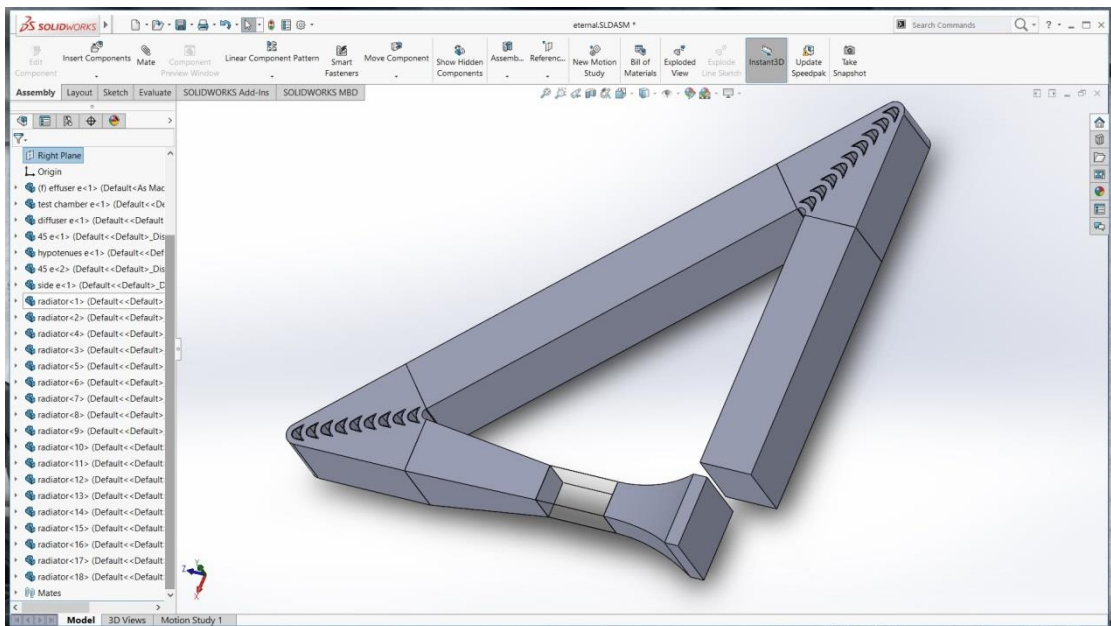
*Fig.4.9. Hypotenuse duct*

#### 4.7. SIDE DUCT:

- Another duct of dimensions  $0.75 \times 1 \text{ m}^2$  is drawn and extruded to 3.6m to obtain the duct.

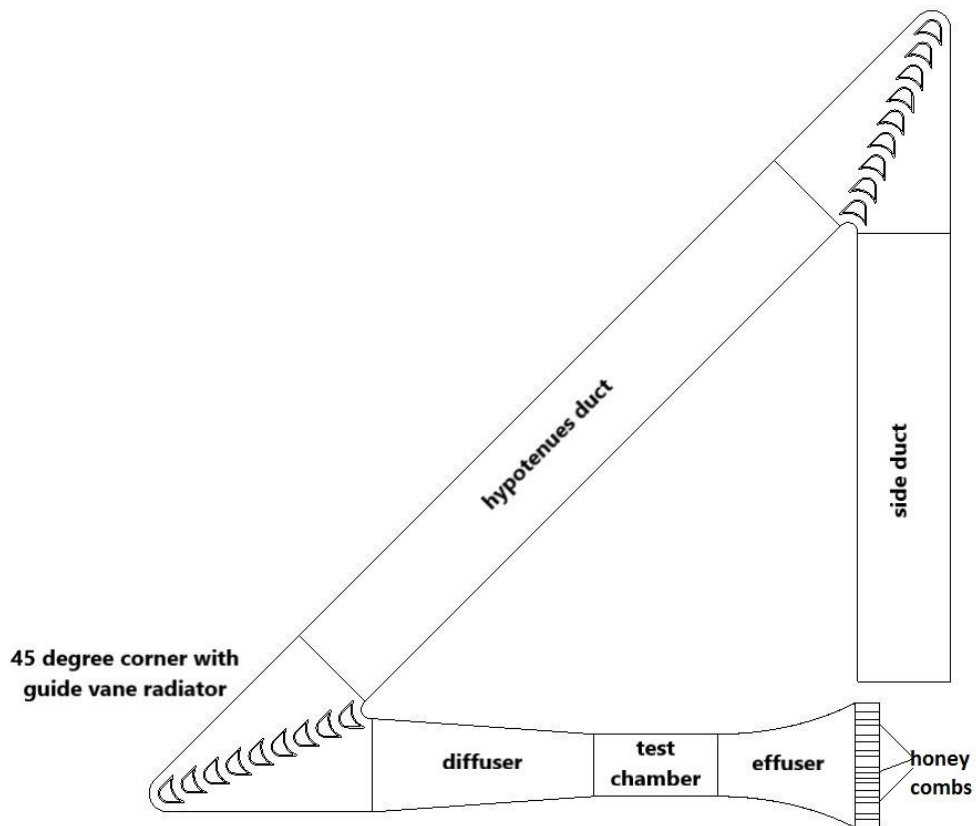
#### 4.8. FINAL ASSEMBLY:

- All the individually developed parts are assembled using face mates to get the final assembly.



*Fig.4.10. Final assembly*

## 5. a. MODE OF OPERATION



*Fig.5.1. Line diagram*

The wind is made to flow through the tunnel through the effuser section of 1m x 1m area. In order to get a smooth flow, the effuser is equipped with a series of honey combs having narrow way to drive the wind blowing from all directions into a horizontal path. The wind flows through the honey combs and enters the contraction. Here the area of the section goes on decreasing up to 0.5m x 0.5m to have an increase in velocity by decreasing pressure.

From there, the wind enters the test section of 0.5m x 0.5m, where the body to be tested is placed for experimentation. In the test section, to view the results, sensors are generally employed. The air after passing through the test section diffuses in the diffuser having inlet area of 0.5m x 0.5m and outlet area of 0.75m x 1m with included angles of 4° and 8°.

The air enters the 45° corner and changes its direction. Guide vanes direct the air to a specific direction without compromising on the uniformity. The guide vanes are so

designed that they also have a provision for the flow of coolant through it so that cooling of the air takes place without use of the radiator. The inner walls of the guide vanes are made of copper due to high thermal conductivity. Hence the guide vanes can be used both for directing the air and also for the cooling purpose. The air is then made to flow in a straight path of 0.75m x 1m area and length 5.4m called hypotenuse to reach another 45° corner. The air again follows the same procedure as done in the previous 45° corner and decreases the temperature of air to a much lower value. Then finally the air is made to flow through a duct of 0.75m x 1m area and 3.6m length to reach the contraction section.

The same air is made to circulate in the same manner and cooling is also done simultaneously to get the results in test section at a specific point based on the requirement.

## 5.b. OBTAINING INPUTS TO THE FLOW

As the wind tunnel is designed for subsonic flow where there will not be any compressibility air in the wind tunnel. The continuity equation holds good, from previous journals we had assumed that the velocity at the inlet of test chamber and outlet of effuser is 16 m/s

Considering effuser

Area at the outlet of the effuser  $A_2 = 0.25m^2$

Area at the inlet of the effuser  $A_1 = 1m^2$

Velocity at the outlet of effuser, inlet of test chamber  $V_2 = 16 m/s$

Velocity at the inlet of effuser  $V_1 = ?$

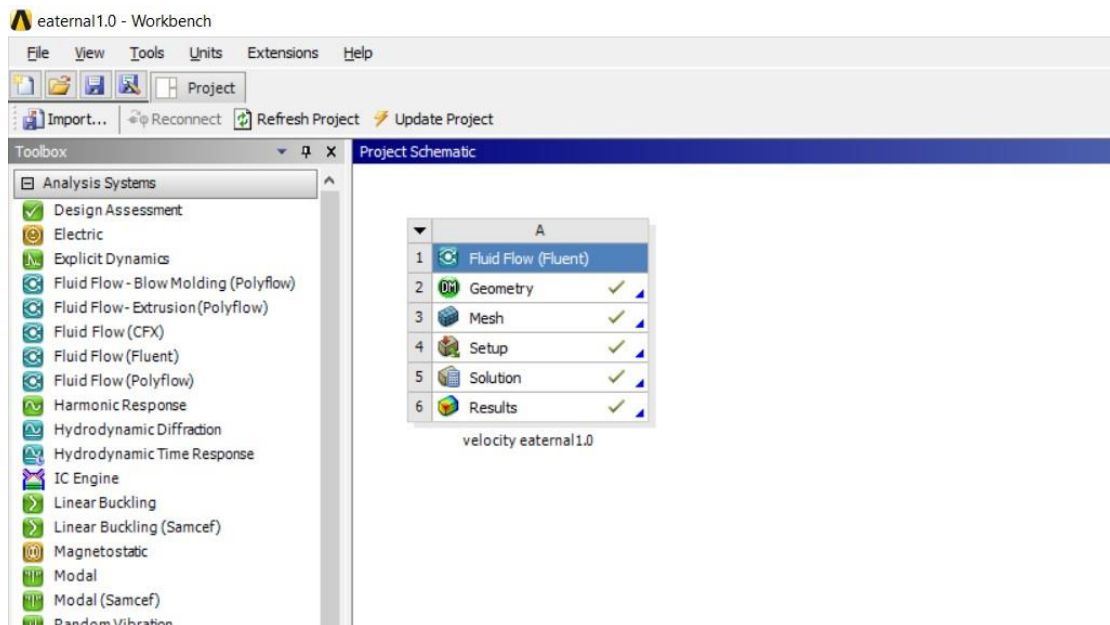
Continuity equation:  $A_1V_1 = A_2V_2$

$$\begin{aligned} V_1 &= \frac{A_2 * V_2}{A_1} \\ &= \frac{0.25 * 16}{1} \\ &= 4 m/s \end{aligned}$$

From the above equation input condition is obtained which is used in ANSYS simulation for velocity input boundary condition.

## 6. SIMULATION IN ANSYS WORKBENCH

- ANSYS R 15.0 is used to do the simulations in Analysis System → Fluid Flow (Fluent)
- Then a dialogue box appears in Project Schematic as shown below



*Fig.6.1. ANSYS WORKBENCH*

- It was named as velocity external 1.0

### 6.1. GEOMETRY:

- Select geometry (bourse) → Input, to input IGS file of external which was exported in solid works assembly.
- Click import → Generate, all the solid objects are generated as they were created in solid works.
- Then the objects are differentiated and united using Boolean operation by, Create → Boolean → Named selection
- Using fill operation, the gaps between the radiator are filled, Tools → Fill → Named selection
- Some of the faces which are default created are merged using merge operation Tools → Merge → Face selection



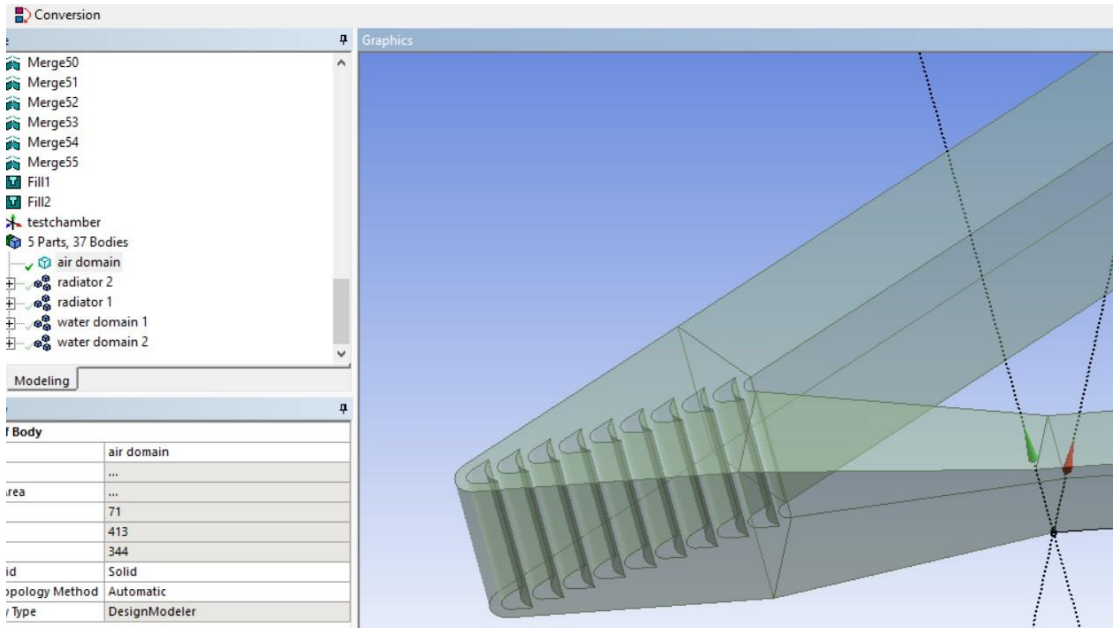


Fig.6.2. Air domain

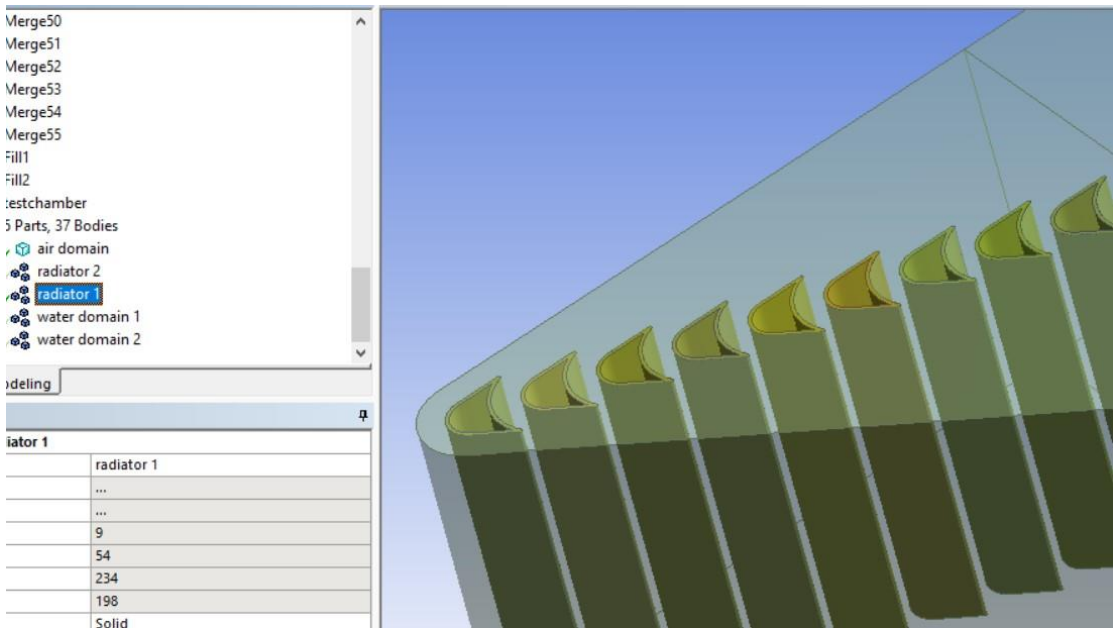


Fig.6.3. Radiator 1



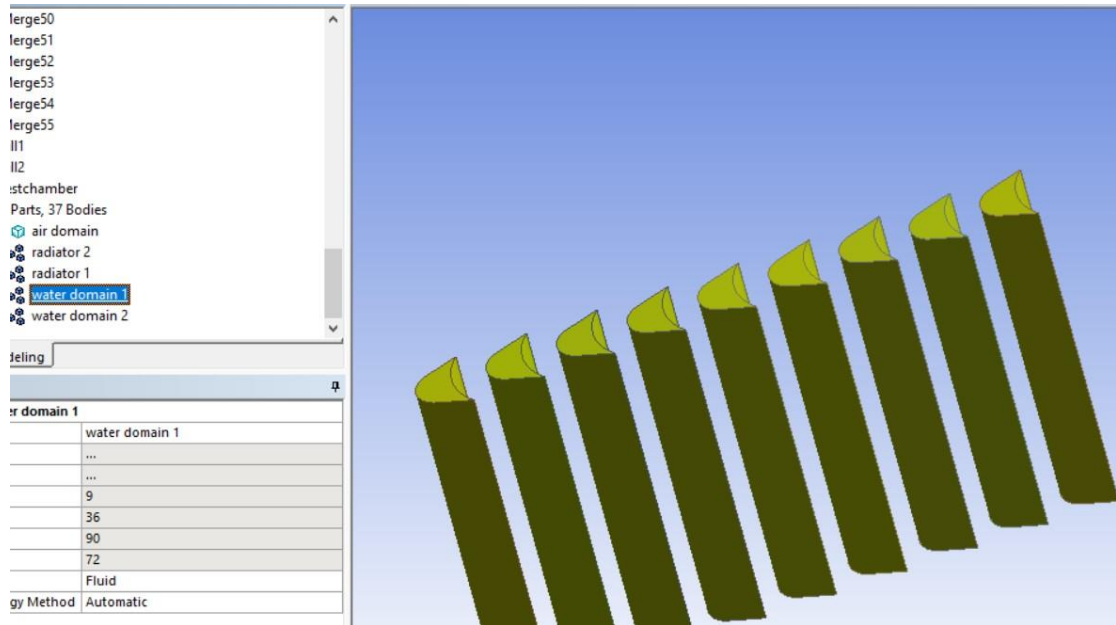


Fig.6.4. Water domain 1

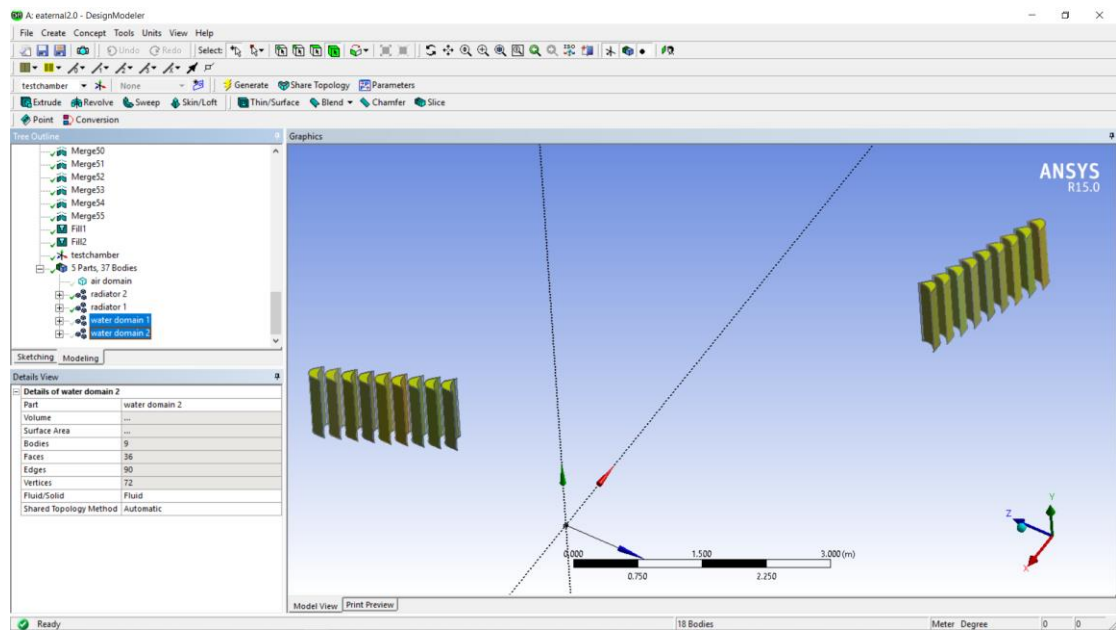


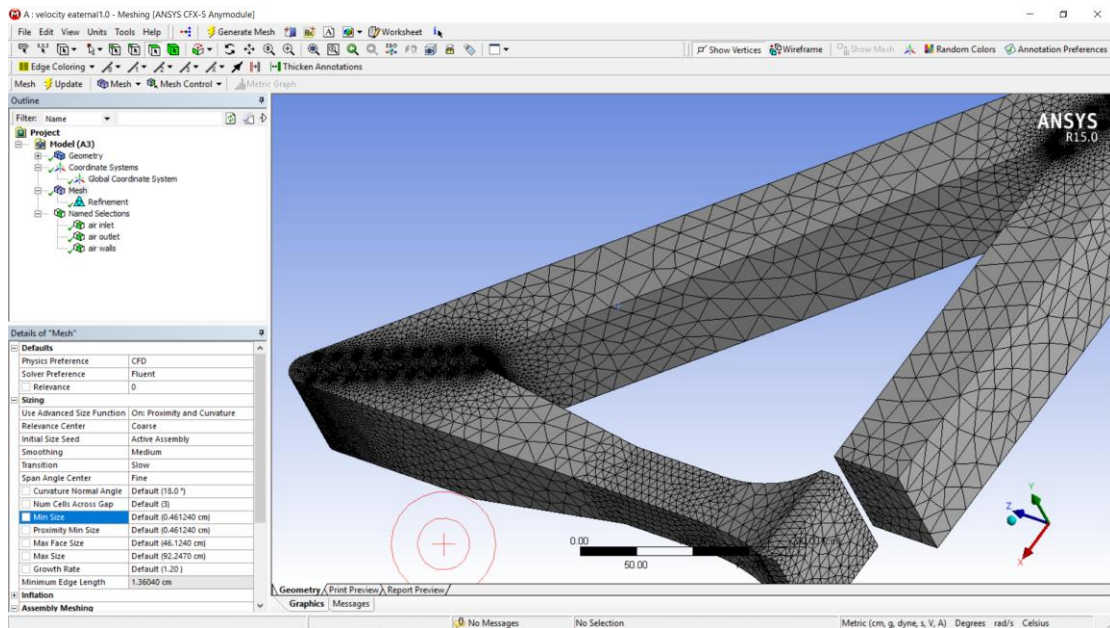
Fig.6.5. Radiator 2 and water domain 2

In order to know the velocities for heat transfer calculations; radiator 1, radiator 2, water domain 1 and water domain 2 were suppressed and only air domain is considered.

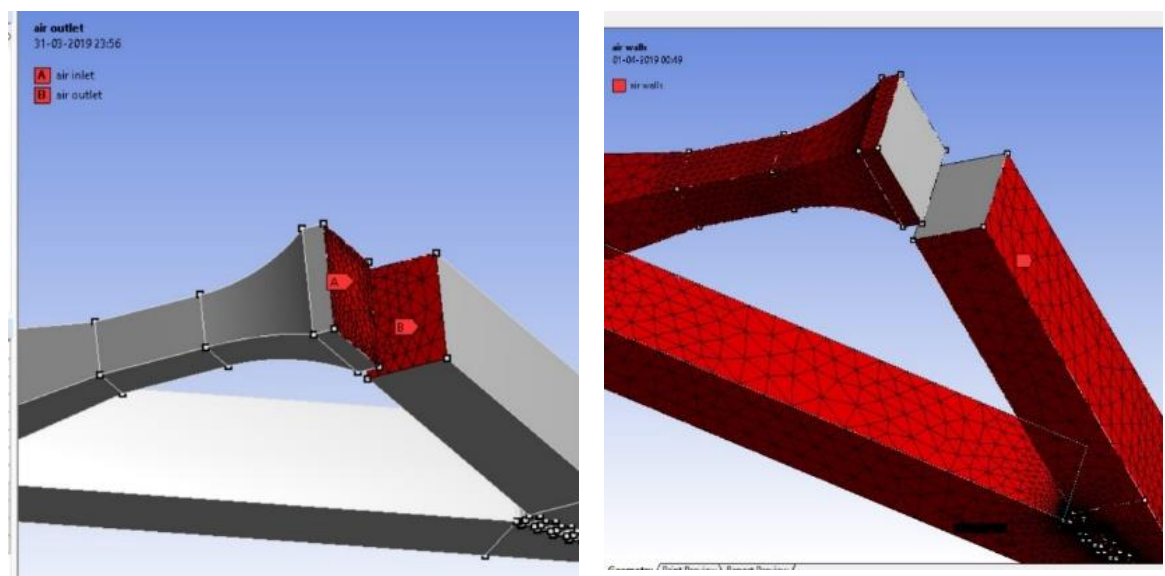
## 6.2. MESHING:

- Right click to the Mesh → Edit, to the body

- Sizing: on the proximity and curvature for better results, due to computational limits of Relevance: coarse, Smoothing: medium
- Proximity min size:0.4 cm, Max face size:46 cm, Max size:92 cm
- For better accuracy of flow refinement is done on faces of test chamber, Mesh → Insert → Refinement and rest were set to default
- Mesh is generated, Mesh → Generate mesh, Inlet and outlet conditions are defined using named selections



*Fig.6.6. Mesh of air domain*



*Fig.6.7. Boundary condition*

### 6.3. SETUP:

- Setup → Edit → Double precision on → Parallel 4 processors is set → OK
- General  
Type → Pressure based, Velocity formation → Absolute, Time → Steady  
Gravity =  $9.81\text{m/s}$
- Modes  
k-epsilon → Realizable → Enhance wall treatment
- Materials  
Fluent database → air → copy
- Cell zone condition  
Add material → air → OK
- Boundary conditions  
Air inlet → velocity inlet =  $4\text{ m/s}$   
Air outlet → pressure outlet =  $0\text{ pa}$

### 6.4. SOLUTION:

- Solution methods  
Scheme → SIMPLE, Gradient → Least square cell method, Pressure → second order, Momentum → second order upwind
- Solution initialization  
Hybrid initialization is done
- Run calculations due to hard ware limit 5 iterations were done
- All the above conditions are very much feasible in increases the accuracy of velocity and pressure contours

## 7. HEAT TRANSFER IN THE GUIDE VANE RADIATORS

When two fluids having different temperatures come in contact either directly or indirectly, heat transfer takes place between the two fluids. This heat transfer depends on the inlet temperatures of the fluids, heat transfer coefficient of the medium and the velocities with which the fluids come in contact.

As the air flows continuously in the tunnel, the temperature of air increases and it has to be cooled down.

If the temperature of air is increased to 60°C, it has to be brought back to the ambient temperature before the start of next cycle. As two 45° corners are available for the cooling purpose, the cooling procedure is divided into two phases, where the first phase includes cooling down the temperature from 60°C to 45°C and in the second phase, cooling is done to reduce the temperature from 45°C to ambient temperature i.e. 30°C.

### 7.1. HEAT TRANSFER IN THE FIRST 45° CORNER:

Inlet temperature of the air = 60°C

Outlet temperature of the air = 45°C

Outlet temperature of the water = 40°C

Velocity of air = 4 m/s

Let, the inlet temperature of water be 'T<sub>in</sub>'

Velocity of water be 'V'

Here,

Heat loss by the air = Heat gain by the water

$(m \cdot C_p \cdot \Delta T)_{\text{air}} = \text{No. of vanes} \cdot (m \cdot C_p \cdot \Delta T)_{\text{water}}$

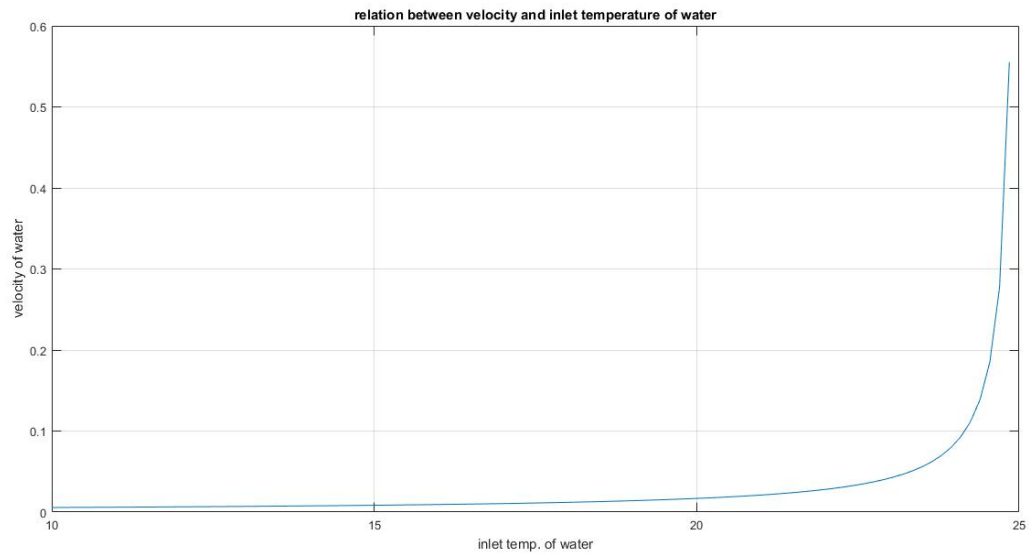
As  $m = \rho \cdot A \cdot V$ ,

$(\rho \cdot A \cdot V \cdot C_p \cdot \Delta T)_{\text{air}} = 9 \cdot (\rho \cdot A \cdot V \cdot C_p \cdot \Delta T)_{\text{water}}$

$1.06 \cdot 0.7 \cdot 4 \cdot 1 \cdot (60 - 45) = 9 \cdot 1000 \cdot 0.0140 \cdot V \cdot 4.2 \cdot (40 - T_{\text{in}})$

$V \cdot (40 - T_{\text{in}}) = 0.084127$

A graph is plotted between V and T<sub>i</sub> in MATLAB to find the inlet velocity of water.



*Fig.7.1. Velocity vs inlet temp. of water in first corner*

From the graph, the required inlet temperature of water and the velocity of water are determined.

## 7.2. HEAT TRANSFER IN THE SECOND 45° CORNER:

Inlet temperature of the air = 45°C

Outlet temperature of the air = 30°C

Velocity of water = 0.0084127 m/s (from first corner)

Let, the outlet temperature of water be 'T<sub>out</sub>'

Inlet temperature of the water be 'T<sub>in</sub>'

Here,

Heat loss by the air = Heat gain by the water

$(m \cdot C_p \cdot \Delta T)_{\text{air}} = \text{No. of vanes} \cdot (m \cdot C_p \cdot \Delta T)_{\text{water}}$

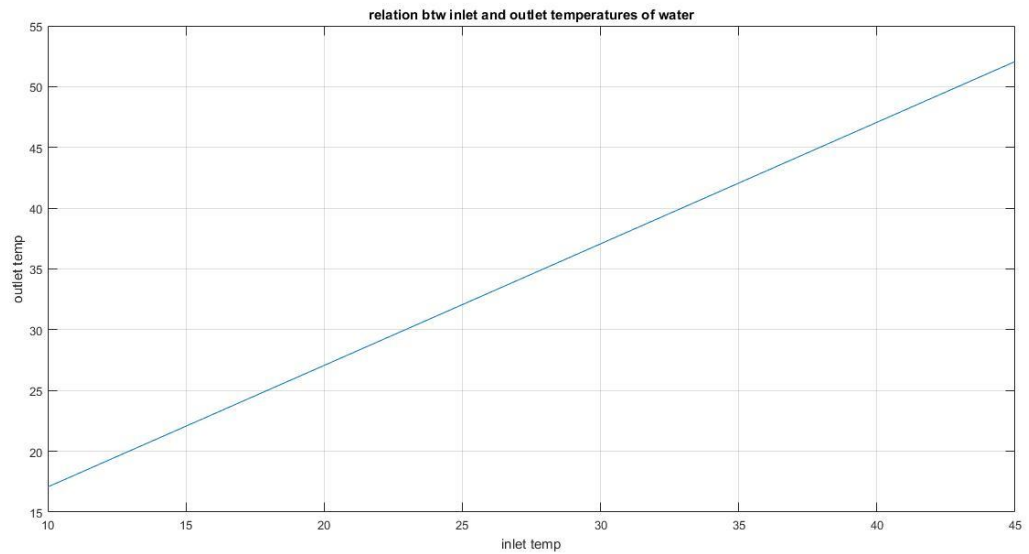
As  $m = \rho \cdot A \cdot V$ ,

$(\rho \cdot A \cdot V \cdot C_p \cdot \Delta T)_{\text{air}} = 9 \cdot (\rho \cdot A \cdot V \cdot C_p \cdot \Delta T)_{\text{water}}$

$1.06 \cdot 0.7 \cdot 4 \cdot 1 \cdot (45 - 30) = 9 \cdot 1000 \cdot 0.0140 \cdot 0.0084127 \cdot 4.2 \cdot (T_{\text{out}} - T_{\text{in}})$

$(T_{\text{out}} - T_{\text{in}}) = 10^\circ\text{C}$

A graph is plotted between  $T_{out}$  and  $T_{in}$  in MATLAB to find the corresponding inlet and outlet temperature of water.



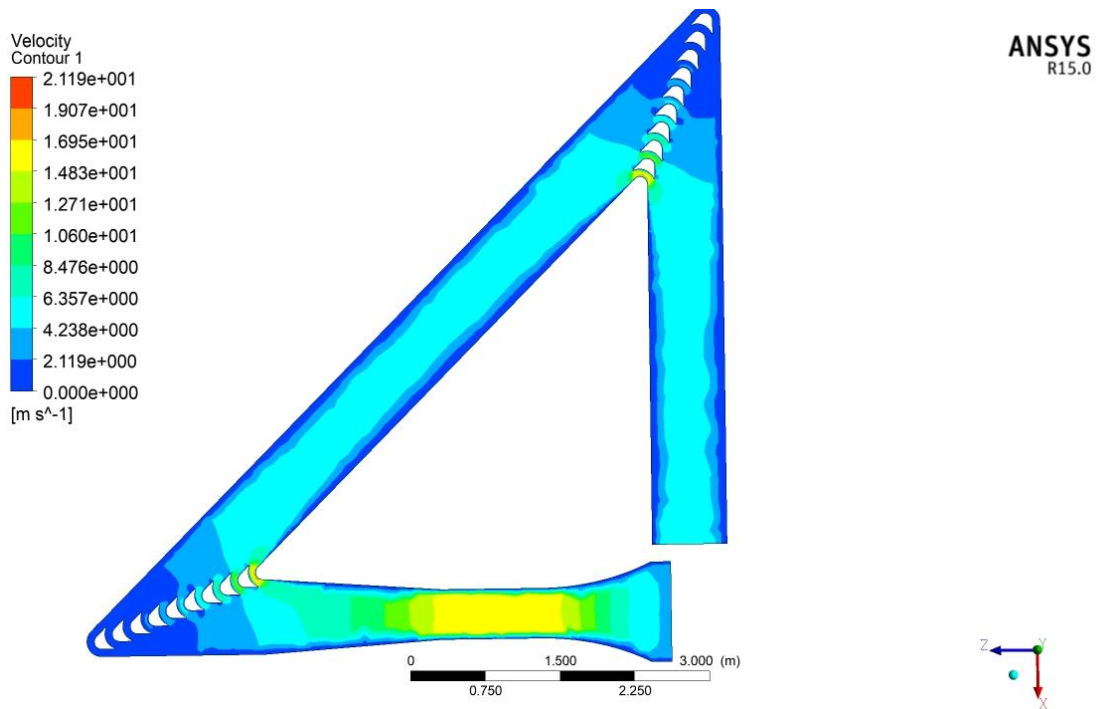
*Fig.7.2. Outlet temp vs inlet temp in second corner*

From the graph, the inlet and outlet temperatures of the water differ by  $10^{\circ}\text{C}$ . So based on the graph, suitable temperatures are taken and given as the conditions for the flow of water.

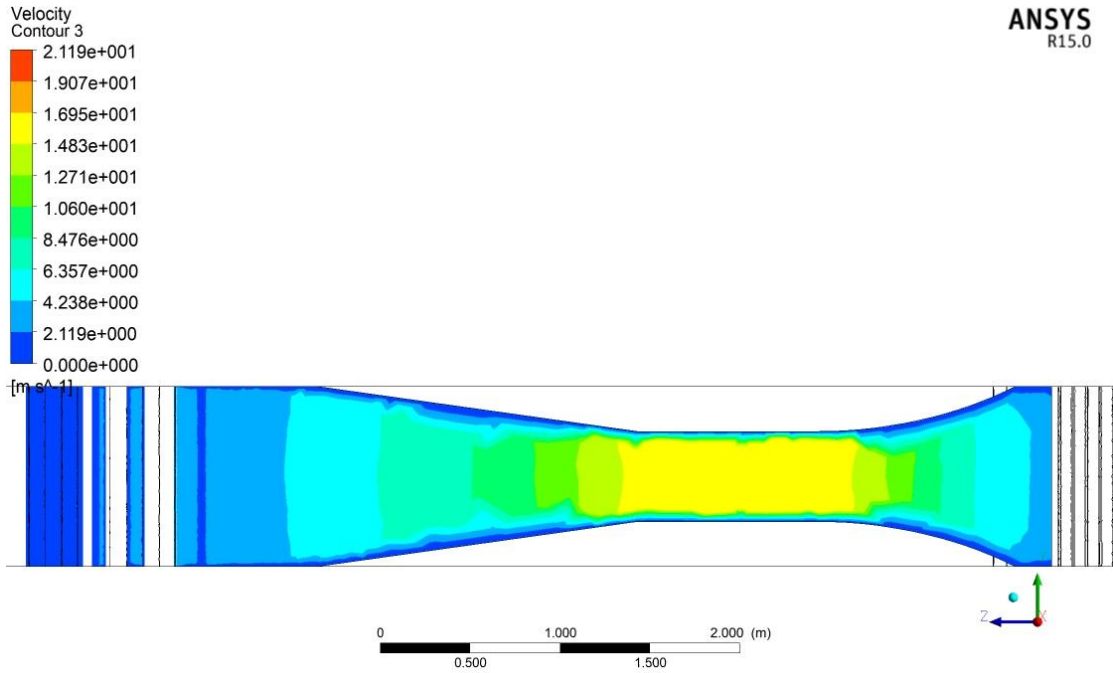
Hence the heat transfer takes place between the air and water to bring down the temperature of the air to the required temperature by passing the water through the guide vane radiators. Thus, the cooling process can be achieved without the use of radiators.

## 8. RESULTS

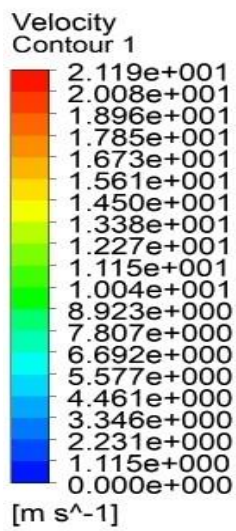
The flow profile inside the wind tunnel which was obtained is shown below. The velocity variation within the test section is significantly low which is very much important for experimentation purposes. The average velocity in the test chamber is  $16.73 \text{ m/s}$



*Fig.8.1. Front view of simulation*

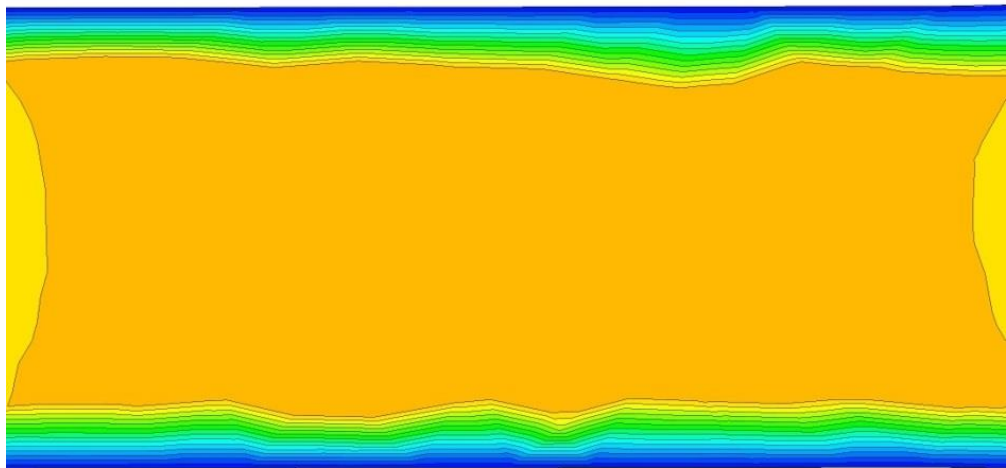


*Fig.8.2. Bottom view of simulation*

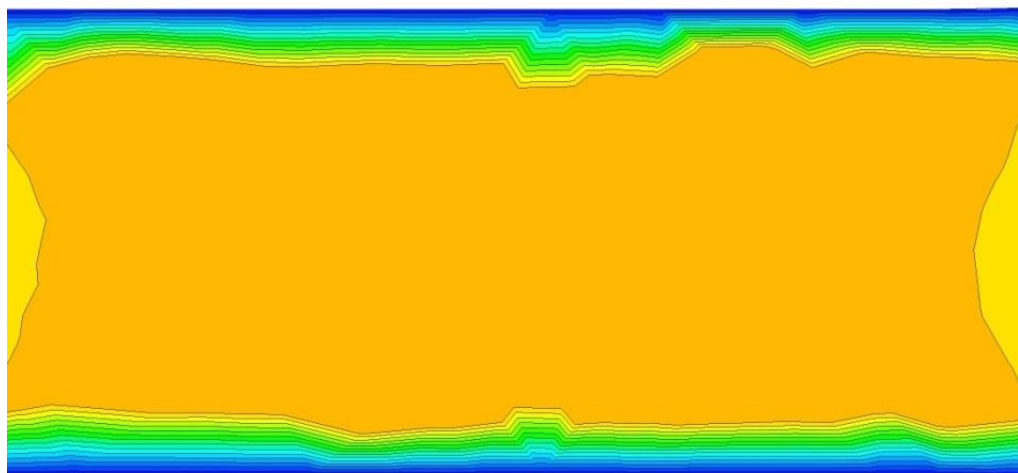


*Fig.8.3. velocity scale*



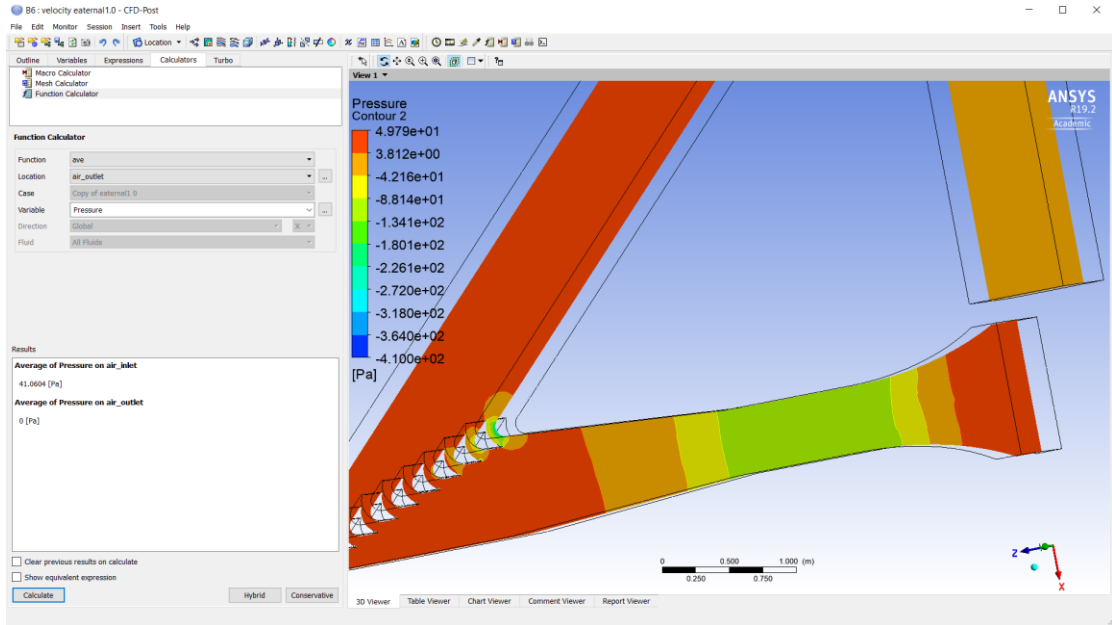


*Fig8.4. Horizontal view of test section*



*Fig8.5. Vertical view of test section*

By using functional calculator in CFD post- results total pressure loss was calculated



*Fig.8.6. Air inlet and outlet pressure*

Average of Pressure on air inlet  $P_1 = 41.0604 \text{ pa}$

Average of Pressure on air outlet  $P_2 = 0 \text{ pa}$

Total pressure loss  $P = P_1 - P_2$

$$= 41.0604 \text{ pa}$$

By this the total pressure loss throughout the wind tunnel is 41.0604. It should be provided by the centrifugal blower to have average velocity 16  $m/s$  at the test chamber.

## 9. CONCLUSION

The wind tunnel is designed and developed in SolidWorks. Various components of the wind tunnel are created and then all the components are assembled based on the required dimensions. Guide vane radiators are introduced into the 45° corners to get the final wind tunnel assembly with guide vanes. The regular closed loop wind tunnel is of rectangular type having four 90° corners. It is replaced by two 45° corners so that the total head loss is decreased. The guide vanes are also provided with some provisions in order to pass the coolant to decrease the temperature of the continuously flowing air. Hence it can act as a radiator and moreover the head loss due to radiator is eliminated. The fluid simulation is done in the designed wind tunnel and a uniform flow of wind in the tunnel is achieved which can be shown as a proof that the designed model can be presented. For the heat transfer, the necessary equations are taken and applied at the air-water boundary and the inlet conditions of the water are derived which can be used for achieving the required heat transfer.

Therefore, with the designed wind tunnel, head loss due to corners and radiator is decreased and a new mode of cooling is introduced without any loss of other factors.

### 9.1. Future Scope:

- A Centrifugal blower which is suitable for inlet boundary condition is to be created.
- Proper circulation system for cooling fluid can be created.
- Analysis of a test specimen in this wind tunnel can be done.
- Concept of guide vane radiator can be applied in various other situations.

## REFERENCES

- [1] John Kaiser Calautit, Hassam Nasarullah Chaudhry, “A validated design methodology for a closed-loop subsonic wind tunnel” *J. Wind Eng. Ind. Aerodyn.* 125 (2014) 180-194.
- [2] John Kaiser Calautit, Hassam Nasarullah Chaudhry, “Comparison between evaporative cooling and a heat pipe assisted thermal loop for a commercial wind tower in hot and dry climatic conditions” *Applied Energy* 101 (2013) 740-755.
- [3] Peter Moonen, Bert Blocken, “Numerical modeling of the flow conditions in a closed-circuit low-speed wind tunnel” ” *Journal of Wind Engineering and Industrial Aerodynamics* 94 (2006) 699-723.
- [4] Peter Moonen, Bert Blocken, “Indicators for the evaluation of wind tunnel test section flow quality and application to a numerical closed-circuit wind tunnel” ” *Journal of Wind Engineering and Industrial Aerodynamics* 95 (2007) 1289-1314.
- [5] Adrian R. Wittwer, Sergio V. Moller, “Characteristics of the low speed wind tunnel of the UNNE” *Journal of Wind Engineering and Industrial Aerodynamics* 84 (2000) 307-320.
- [6] Morimasa Watakabe, Masamiki Ohashi, “Comparison of wind pressure measurements on tower-like structure obtained from full-scale observation, wind tunnel test, and the CFD technology” *Journal of Wind Engineering and Industrial Aerodynamics* 90 (2002) 1817-1829.
- [7] B.Sahin, A.J. Ward-Smith, “The pressure drop and flow characteristics of wide angle screened diffusers of large area ratio” *Journal of Wind Engineering and Industrial Aerodynamics* 58 (1995) 33-50.
- [8] F. Kevin Owen, Andrew K.Owen, “Measurement and assessment of wind tunnel flow quality” *Progress in Aerospace Sciences* 44 (2008) 315-348.
- [9] Aslam Bhutta, Nasir Hayat, “CFD applications in various heat exchangers design” *Applied Thermal Engineering* 32 (2012) 1-12.
- [10] Chad Harris, Mircea Despa, “Design and Fabrication of a Cross Flow Micro Heat Exchanger” *Journal of Microelectromechanical systems*, Vol. 9, No.4, December 2000.