CFD Analysis and Comparison of Air Flow within an Annulus, Airflow over an Aerofoil and Convective Heat Transfer from a Heat Source of a Radiator within an Enclosed Room

Author : Danya Lincy T J
University of Hertfordshire

Table of Contents

ABSTRACT

In this paper the three different simulations has been done and it is analyzed and those simulations has been compared with the literature papers. The first simulation is Air Flow within an Annulus , In this simulation the hollow pipe is simulated using the STAR -CCM and the pressure gradient , velocity parabola are analysed and compared with the literature results. The second simulation is Airflow over an airfoil , In this simulation the subsonic flow over a NACA 0012 is analyzed at different angle of attack with the Mach number of 0.5. The third simulation is Convective Heat Transfer from a Heat Source of a Radiator within an Enclosed Room. In this simulation heat convection is studied as the laminar model is assumed for the steady, Coupled and the ideal gas flow .

CHAPTER ONE

Introduction

Air Flow within an Annulus

　　In this problem a pipe is considered for the simulation. The diameter of the pipe is 0.04 m and the diameter of the solid core is 0.02 m. The inlet velocity is distributed uniformly and the velocity is calculated with the laminar Reynolds number. The fluid taken for the analysis is air and the temperature are given as the 25 degree Celsius. The pipe is circular because pipes with a circular cross section can withstand large pressure differences between the inside and the outside without undergoing significant distortion (cengel-cimbala). The development of velocity is given in the figure 1. The velocity profile is fully developed as parabola near the outlet region. Low Reynolds number is chosen for the analysis as 80 to decrease the entry length of the flow and it will make easier for the short pipe length. The simulation of pipe flow is widely used in all industries because of its flow behaviour.
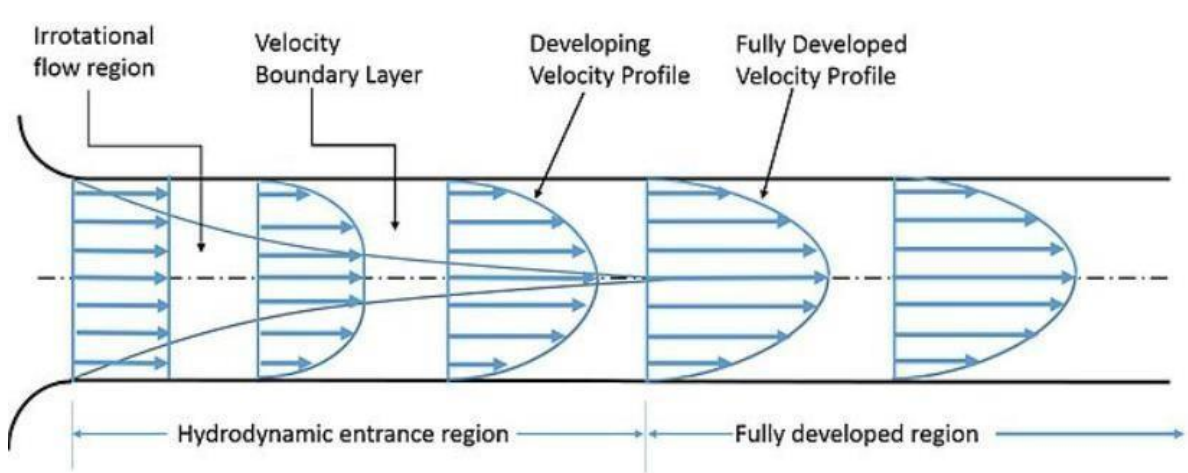
Figure 1: Velocity profile of the Pipe (Ocin)

Expected solution:

　　The simulated physical problem is compared with the analytical solution and it is expected that the velocity profile from the simulation is parabola and it should be biased towards one side because of the logarithmic term.

Simulation Parameters: Inlet Velocity:

STAR-CCM+ is used for the simulation. The inlet velocity is calculated with the Reynolds number selected from the laminar flow. Length is replaced with the hydraulic diameter and it is calculated as
$D_h = 2(a - b)$

Where a is the radius of the pipe b is the radius of the solid core

Velocity for laminar:

$$v = \frac{Re \times \mu}{\rho \times D_H}$$

Re= Reynolds Number

μ=Dynamic Viscosity Pa-s

ρ=Density kg/m^3

DH=Hydraulic Diameter

At 25 degree Celsius the Density and Dynamic viscosity is given by 1.1839 kg/m^3 and 1.84908E-5 Pa-s

The velocity is calculated as
V = (80 * 1.84908E-5)/ (1.1839*0.02)
= 0.062474 m/s

Mesh Quality
The base size of the mesh is taken as 0.0008 m to get the fine mesh surface which provides more accurate results. To make the solution more independent to the mesh we have to carry out the mesh independency check. we have to make sure the convergence of the residual error to 10-4 (Leap Australia). When the convergence criteria are obtained the mesh should be refined globally to get the finer cells of mesh. The mesh should be refined until the error is reduced. The mesh independence was met at around 603090 cells. The mesh is given in figure 2

Mesh size	No of cells
0.004	40254
0.003	49118
0.002	75803
0.001	340336
0.0009	434320
0.0008	603090

Figure 2:Mesh of a pipe

Simulation:

After meshing of model, the simulation iteration is set for 100 steps. The simulation runs for forty minutes. The convergence at 10^-4 is best and in this simulation, it is   converged up to 10^-8 which gives the better results. The residuals for the simulation are given in the figure 3
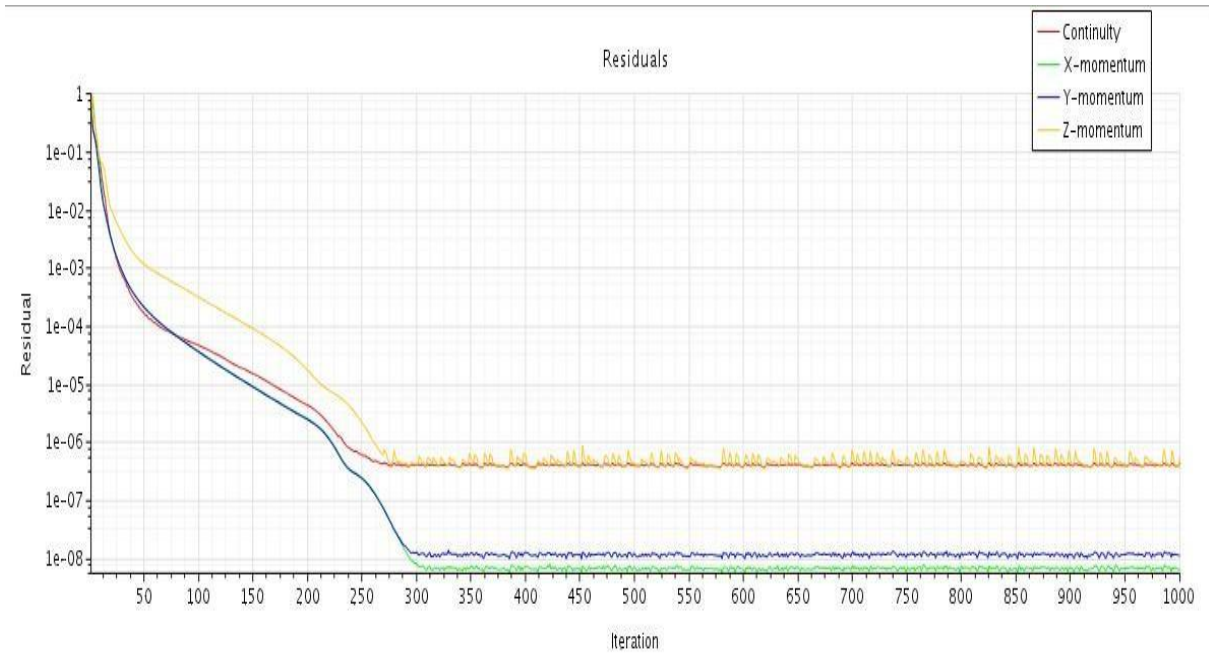

Figure 3: Residual of the pipe

The vector and scalar scene have been obtained to analyse the velocity magnitude of the annular pipe from the scalar and vector scene it is clear the velocity will be maximum in the middle and the velocity is lower near the wall of the pipe. The vector scene is shown in the figure 4 & 5.
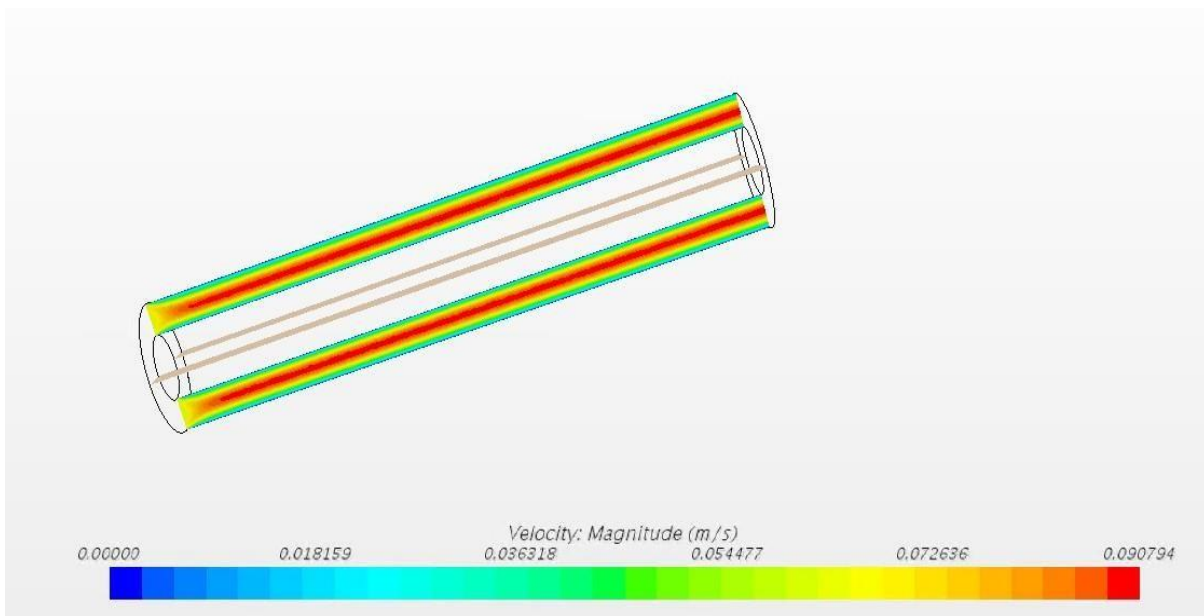

Figure 4: Velocity scalar scene

Analysis and Discussions

For laminar flow the fully developed velocity profile will resembles a parabola. The figure 5 shows that the velocity profile obtained from the simulation is parabola and it biased towards one end. The velocity reaches maximum at 0.090976 m /s.
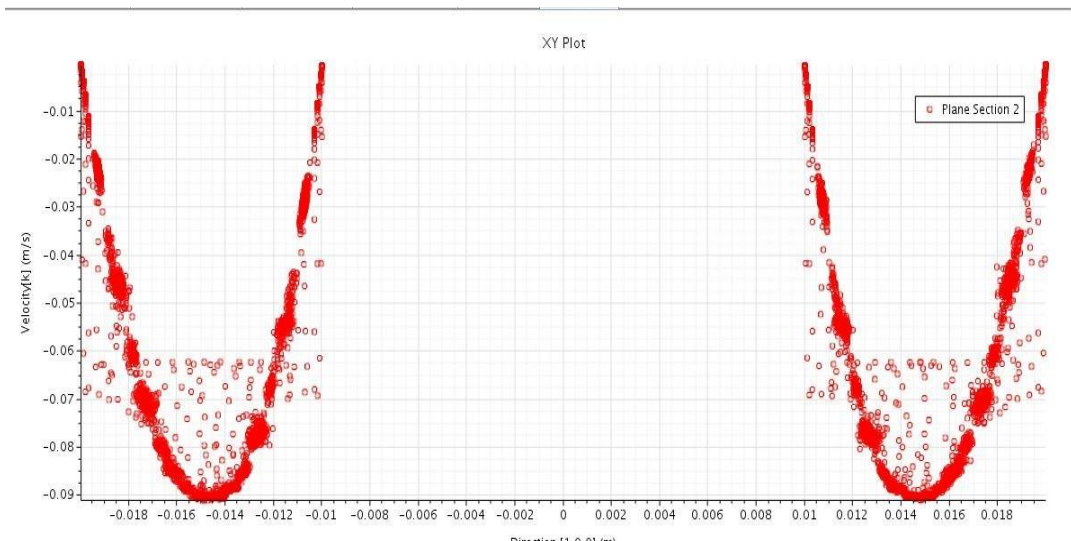

Figure 5: Velocity Profile

Comparison with analytical Results:

The analytical data is obtained from the p*. 362, Fluid Mechanics, 4th edition by F. M. White*. No slip condition is assumed and inner radius is given by b and outer radius is given by a. The governing equation is given by a 1

$$\frac{d}{dr}\left(r\mu\frac{du}{dr}\right) = Kr \qquad K = \frac{d}{dx}(p + \rho g z)$$

1

By integrating above equation twice and the constants are found, the velocity profile is obtained and it is given in the equation 2

$$u = \frac{1}{4\mu}\left[-\frac{d}{dx}(p + \rho g z)\right]\left[a^2 - r^2 + \frac{a^2 - b^2}{\ln (b/a)} \ln \frac{a}{r}\right]$$

2

The velocity parabola obtained will be of parabola. The maximum velocity will be at the radius.

$$r' = \left[\frac{a^2 - b^2}{2 \ln (a/b)} \right]^{1/2} \qquad u = u_{max} \qquad \qquad \text{3}$$

The press .... iven by

$$\frac{\partial P}{\partial z} = -\frac{\rho V^2}{2} \frac{f}{D_H} \qquad \qquad \text{4}$$

Where f and Dh is given 'by

$$f = \frac{64\zeta}{Re_{D_h}} \qquad \zeta = \frac{(a - b)^2(a^2 - b^2)}{a^4 - b^4 - (a^2 - b^2)^2/\ln (a/b)}$$

By substituting radius and Reynolds number the friction factor f = 1.165

$$D_h = \frac{4\pi(a^2 - b^2)}{2\pi(a + b)} = 2(a - b) \qquad \qquad \text{6}$$

By substituting f, Dh, Initial velocity the pressure gradient obtained is -0.8912 Pa/m

The pressure gradient obtained from the simulation result is -0.88 Pa /m which are most probably similar to the analytical result. The maximum velocity obtained in the analytical is 0.089 m/s which are in accordance with the simulation result which gives 0.09 m/s. By using the velocity profile, the flow attained in one cross section is calculated and the profile is plotted which gives parabola. Similarly, the velocity profile obtained from the simulation resembles a parabola.

CHAPTER TWO

Introduction

Airflow over an Aerofoil

The subsonic flow over a NACA 0012 is analysed at different angle of attack the Mach number chosen here is 0.5. The model is choosing to be Turbulence. The main aim of this simulation is to predict the behaviour of the air foil in the given condition. The rapid evolution of computational fluid dynamics (CFD) has been driven by the need for faster and more accurate methods for the calculations of flow fields around configurations of technical interest (Eleni, 2012). In the past decade, CFD was the method of choice in the design of many aerospace, automotive and industrial components and processes in which fluid or gas flows play a major role (Eleni, 2012).

Solution Parameter:

The geometry was modelled for NACA 0012 using STAR-CCM+. The geometry was initially modelled as 3D while meshing it was converted to 2D. It is a steady two-dimensional model and the modelling comes under the turbulence modelling. Spalart-Allmaras Turbulence modelling which is based on Navies stoke equation is chosen for the simulation. The velocity is calculated with respect to the Mach number. The velocity Calculation is shown below

$$M = \frac{U}{a}$$

(Anderson, 1999)

$$a = \sqrt{rRT}$$

Where a= speed of sound U=flow velocity
r=adiabatic index R=molecular gas constant
In flow temperature27°C, Speed of Sound is
a=√1.4×287× (273+27) =347.1887m/s
U=0.5×374.1887= 187.07 m/s

Mesh Quality:

The 2-D badge meshing is done for the model. The base size for the finer mesh is taken as 0.01 m. The new surface control mesh is included in the meshing. The base size and the minimum surface size are customized. The wake refinement region is created around the aerofoil to get more accurate results. Number of layers in prism is set to 3. By combining all the meshes, the fine mesh is created. The mesh scene is shown in the figure 6
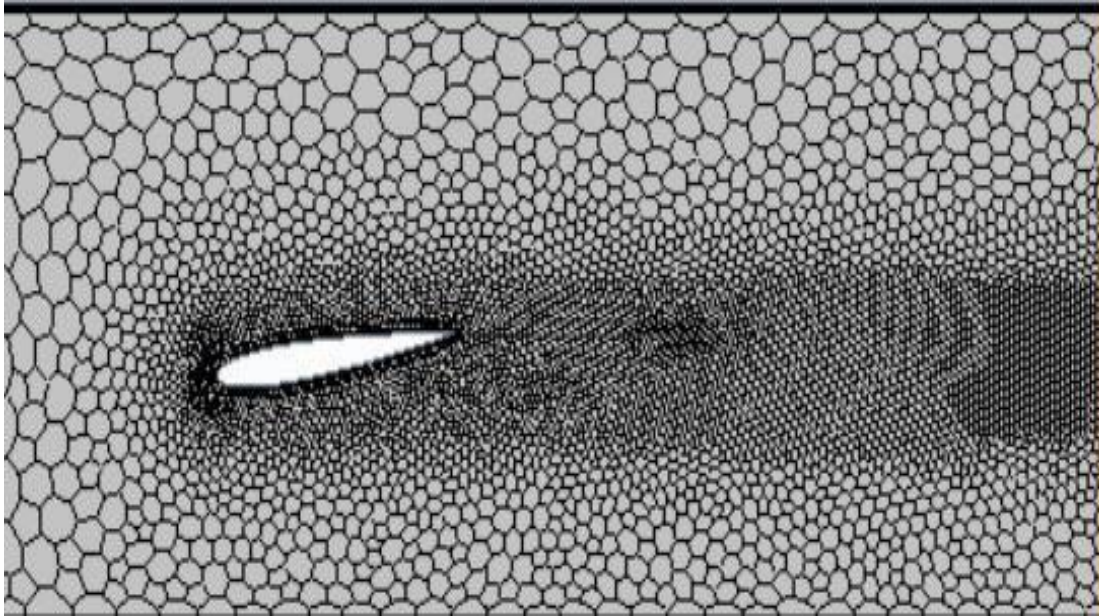
Figure 6: Mesh of an Air foil

Simulation:

The simulation is iterated for 5000 steps. The simulation is done separately for three angle of attack such as 3-degree, 9 degree and 16 degree. The scalar scene for velocity and pressure of three angle of attack are analysed. The drag is also plotted for three different angles of attack. The scalar scenes of velocity of three angle of attack are shown in the figures 7 & 11 &13. The scalar scenes of Pressure are shown in the figures 8 & 10 & 14. The drag monitor plots for three angles of attack are shown in the figure 9 & 12 &15.
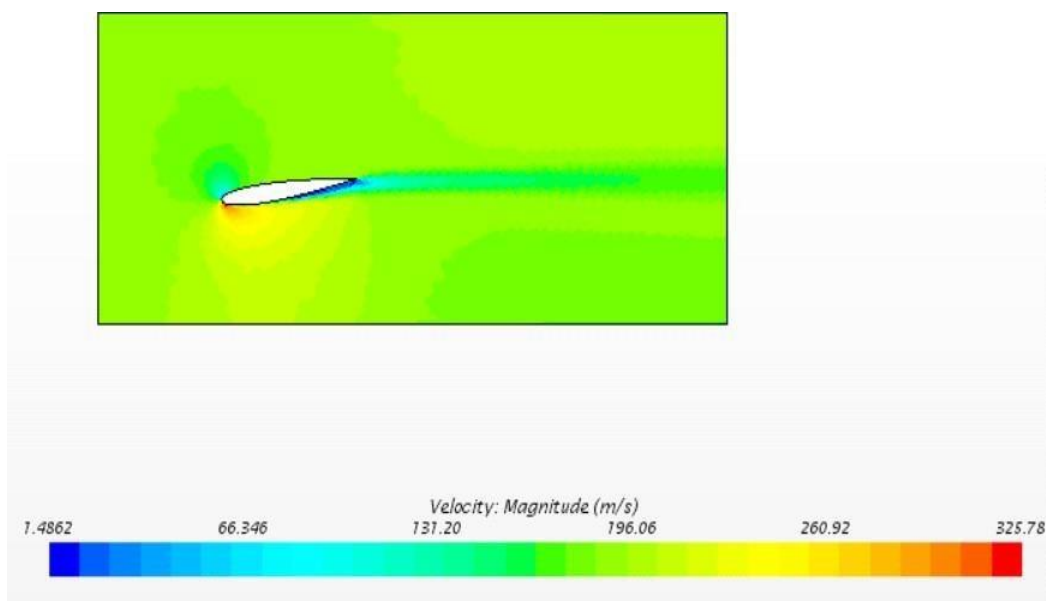

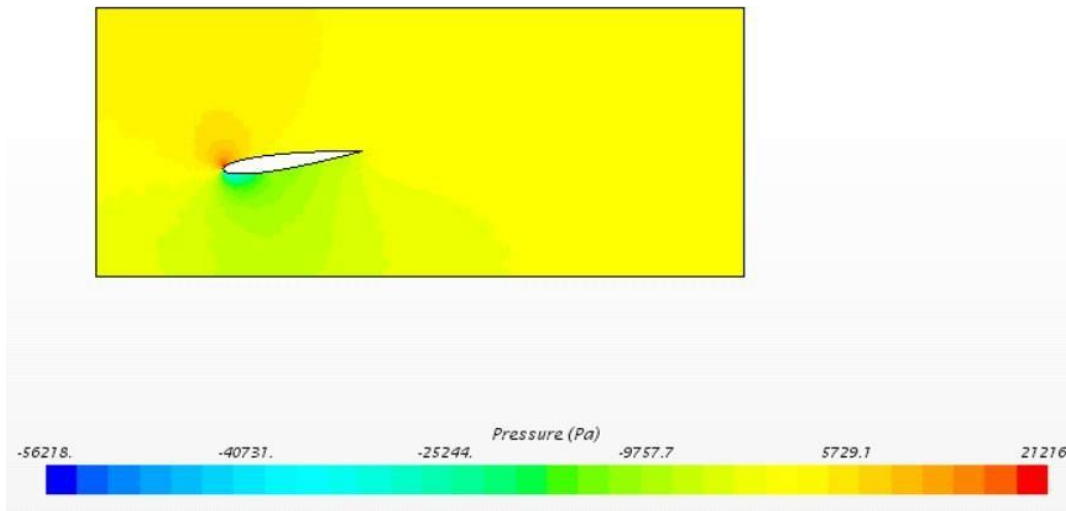*Figure 7: Velocity scalar scene Angle of Attack 9*

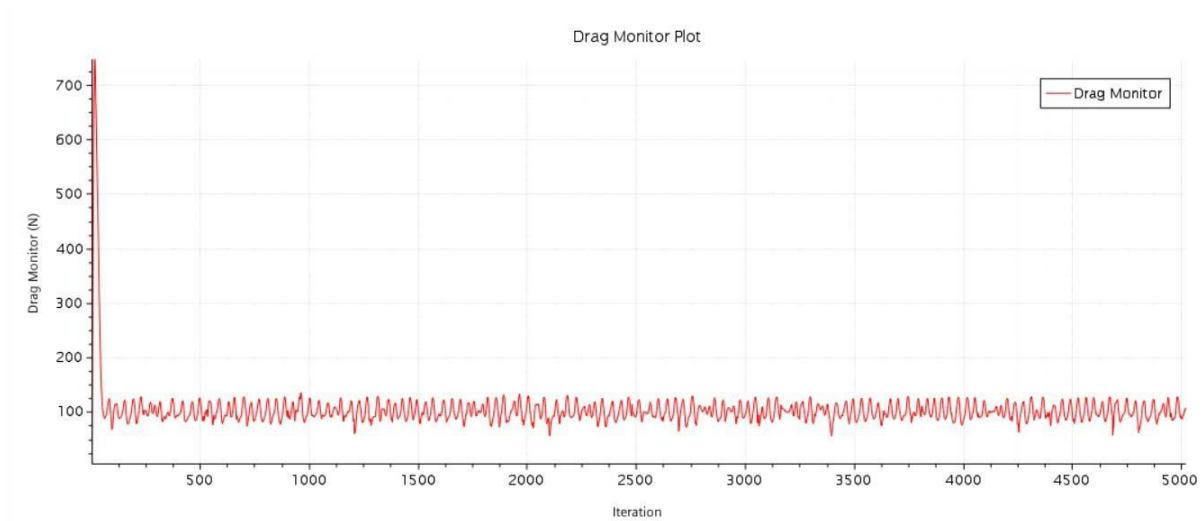*Figure 8: Pressure scalar scene Angle of Attack 9*
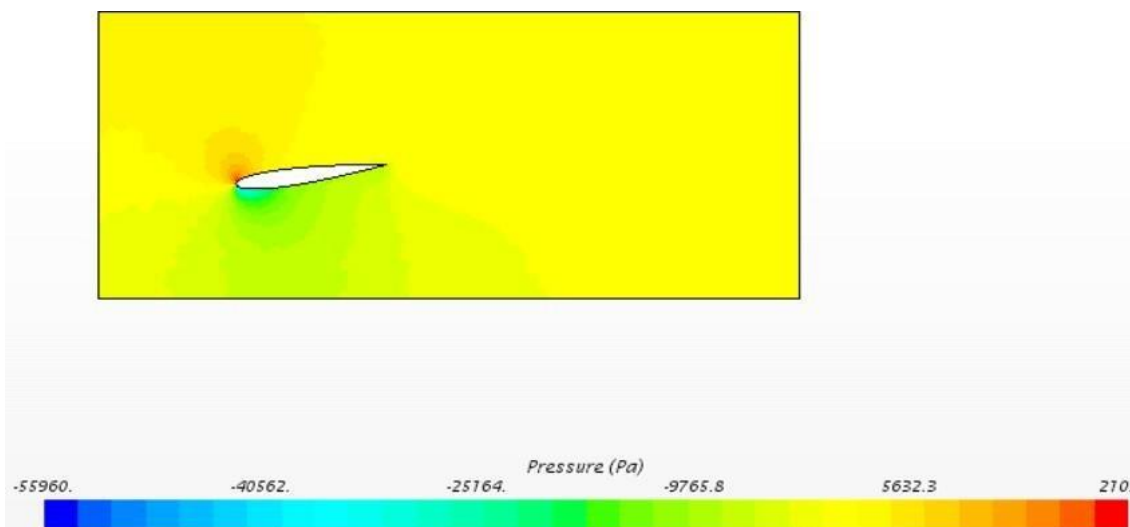

Figure 9: Drag monitors plot angle of attack 9

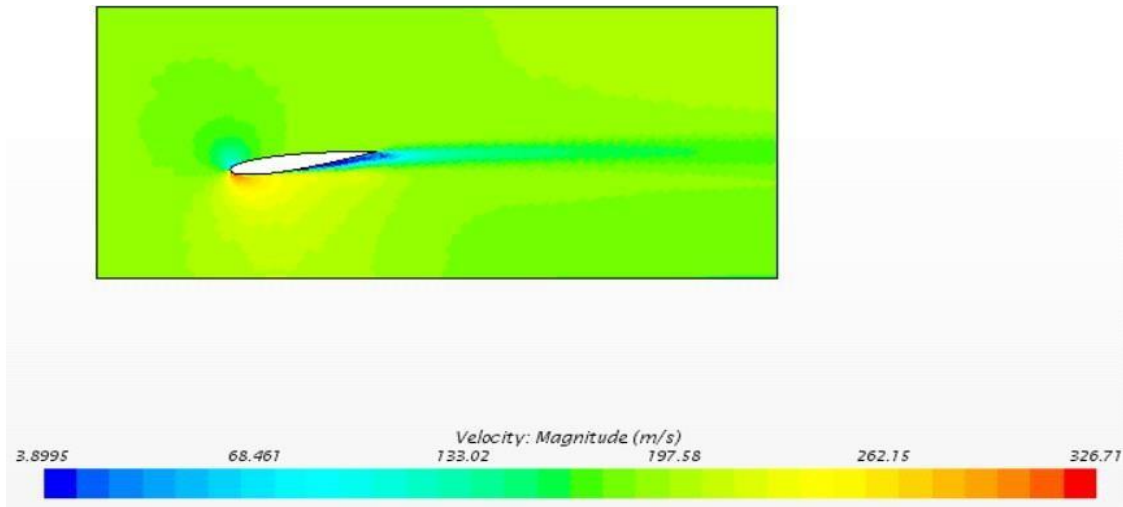*Figure 10: Pressure scalar scene Angle of Attack 3*

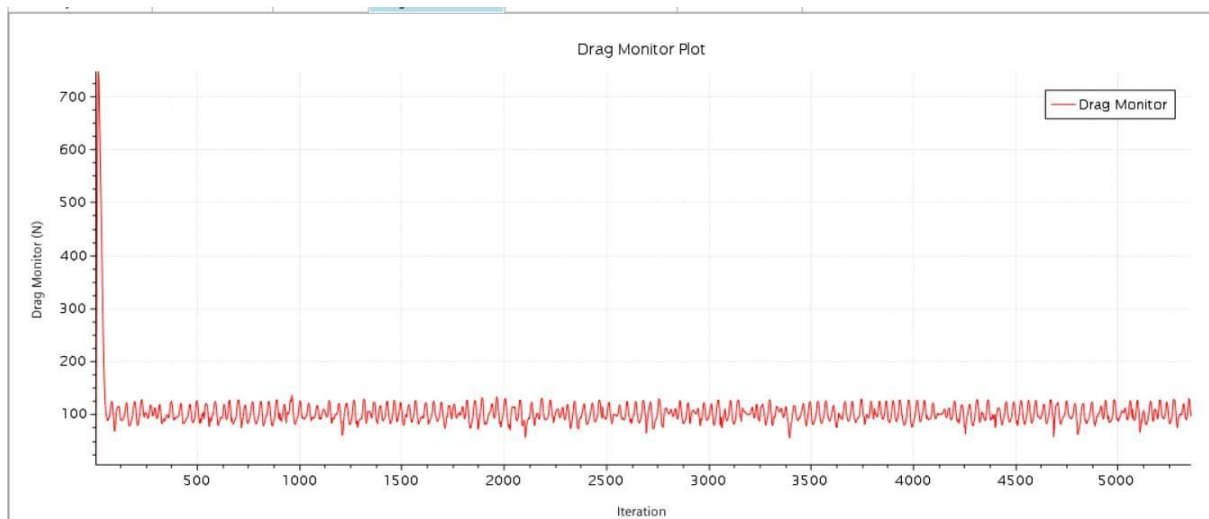*Figure 11: Velocity scalar scene Angle of Attack 3*

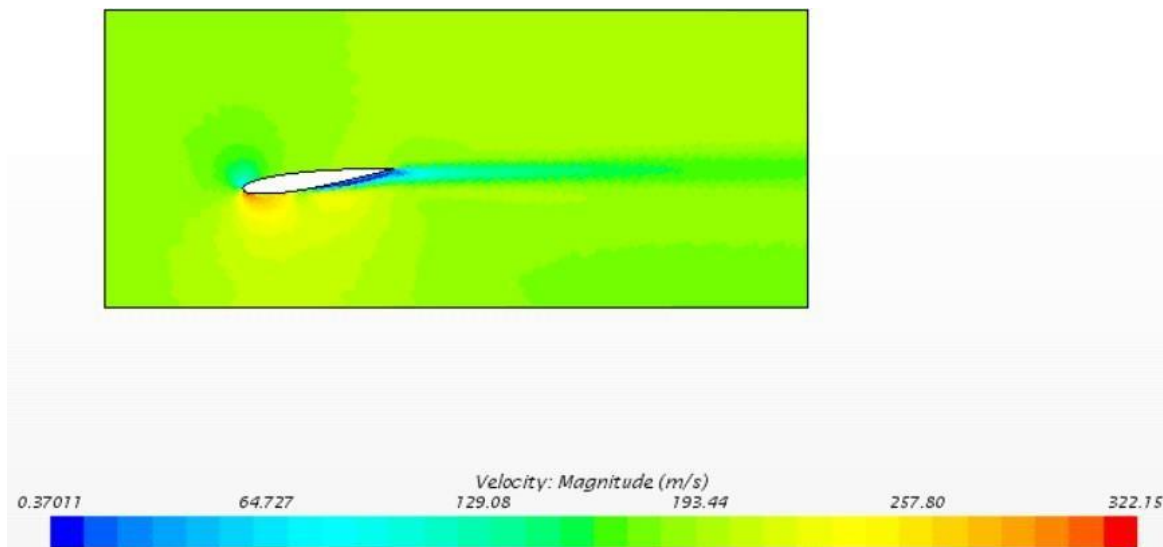*Figure 12Drag Monitor plot Angle of attack: 3*

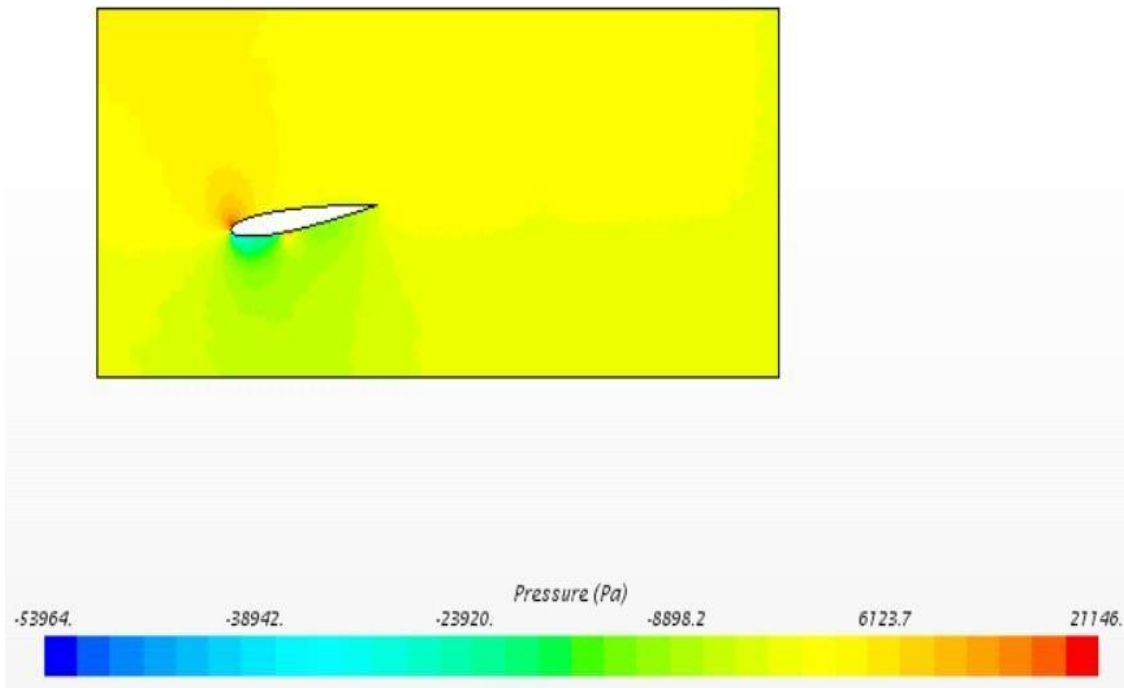*Figure 13: Velocity scalar scene Angle of Attack 12*

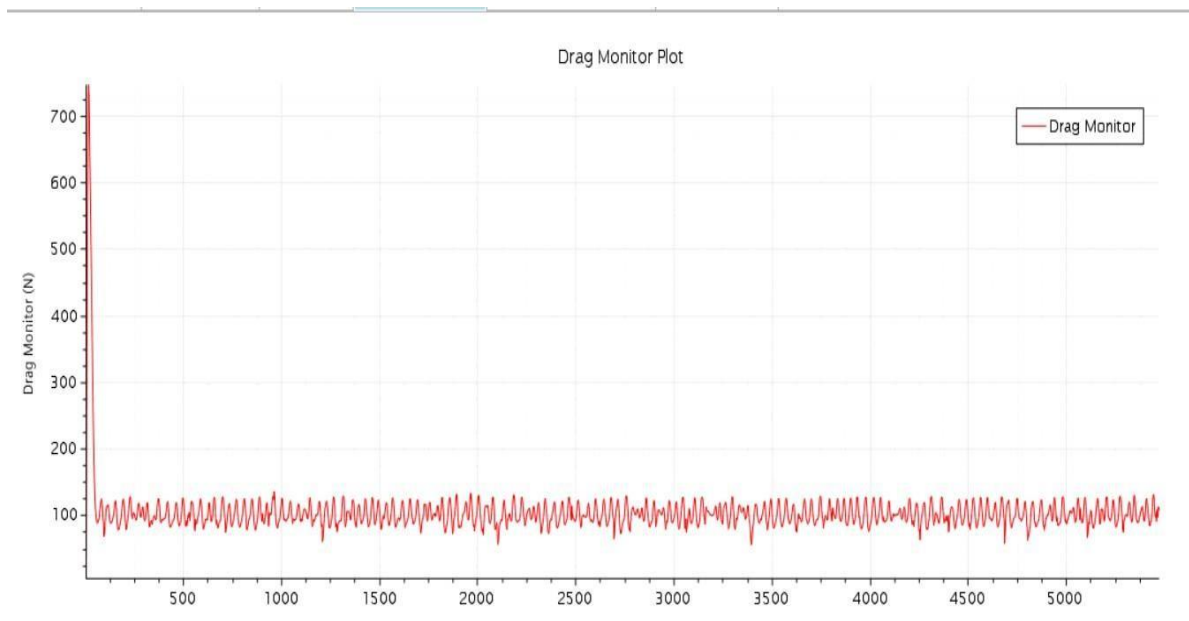*Figure 14: Pressure scalar scene Angle of Attack 12*

Figure 15: Drag monitor plot of angle of attack 12

The velocity magnitude at 3-degree angle of attack is the highest which is 326.17. The pressure is maximum at 9-degree angle of attach which is 21216 Pa.

Analysis and Discussion

At the angle of attack 16 degree upper surface flow on the airfoil began to separate and Stall begins to happen . The appropriate static pressure is obtained from the angle of three and nine as mentioned in the literature paper .In Pressure distribution , compared to incoming flow stream the pressure on the llower surface is greater therefor the airfoil is pushed upwards which is normal to the incoming flow stream.The incoming flow velocity with respect to the airfoil is slowed when the components of the pressure distribution is parallel to the incoming flow velocity .The velocity of the airfoil is more in the upper surface as compare to the lower surface .when the angle of attack increases the velocity in the upper surface is much higher than the velocity in the lower surface.

Comparison from Literature Results:

The literature paper CFD *Douvi, E. (2010). CFD CALCULATIONS OF THE FLOW OVER A NACA 0012 AIRFOIL. Research gate , 8.* Is considered for the comparison with the simulation results . In this paper they have used spalart – Allmaras model and the same model is used for this simulation as this model give more accurate result and it is most appropriate for this simulation . Spalart-Allmaras is an one equation model of Navier-stoke Equation and it is solved for turblent flow . It was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients (Douvi, 2010). The three angle of degree(3 , 9 ,12) is chosen in the literature paper and the velocity , pressure contours are plotted .likewise the scalar scenes of three angle of degree is plotted in this simulation

CHAPTER THREE

Introduction :
Convective Heat Transfer from a Heat Source of a Radiator within anEnclosed Room

The transfer of thermal energy from medium to medium for heating and cooling is done by radiators. The radiators are mainly used for constructions, electronics and automobiles. Whether it is for cooling or heating radiator can be used. It acts as a purpose of heating in case of heating and it acts as coolant in case of heating. Instead of thermal radiation most of the heat by radiators is transferred via convection. The important aspects in designing is setting of temperature in a building or a room. This design, whether to the cost of construction and maintenance of the ventilation system or in terms of the best performance, requires extensive research. (Ref) therefore simulation and analysis are done each structure. In this work radiator in an enclosed room is simulated.

The geometry is model by creating a small room of dimension 3m*3m and it is two dimensional. The walls of the temperature set to the temperature and it is denoted as T1. Inside the room a small radiator is designed and the temperature is set to T2. The dimension of the radiator is 0.3 m * 0.75 m. The heat is transferred as convective heat in the room. The temperature is too chosen with at least ten degrees between T1 and T2.

Solution Parameters:
In this simulation heat convection is studied. For studying the heat convection laminar model is assumed as it will be more accurate for the heat convection. The gravity effect is added in addition to the laminar model. The flow is steady, Coupled and the ideal gas. The temperature of T1 is set as 285 K which is 11.85 degrees and the Temperature T2 is set as 305 degrees which is 31.85 degrees. With respect to initial condition the temperature difference is more than ten degree Celsius. The static temperature is given as 300 K.

Mesh Quality
The geometry is modelled in STAR-CCM+. The 2-D badge meshing is done for the model. The base size for the finer mesh is taken as 0.01 m. The new surface control mesh is included in the meshing. The base size and the minimum surface size are customized. The wake refinement region is created around the aerofoil to get more accurate results. Number of layers in prism is set to 3. By combining all the meshes, the fine mesh is created with 108215 cells, 277423 inner faces and 181008 vertices. The mesh scene is shown in the figure 16
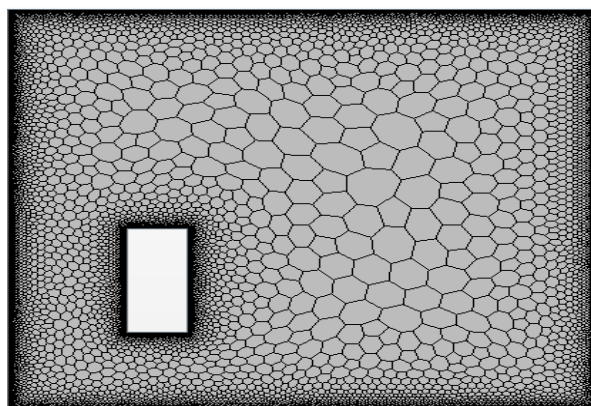
Figure 16: Mesh of a Radiator in a room

Simulation:

The model is simulated to 10000 iterations. The residuals should attain minimum of 10^-4 to get more accurate results. The residuals are shown in the figure 17
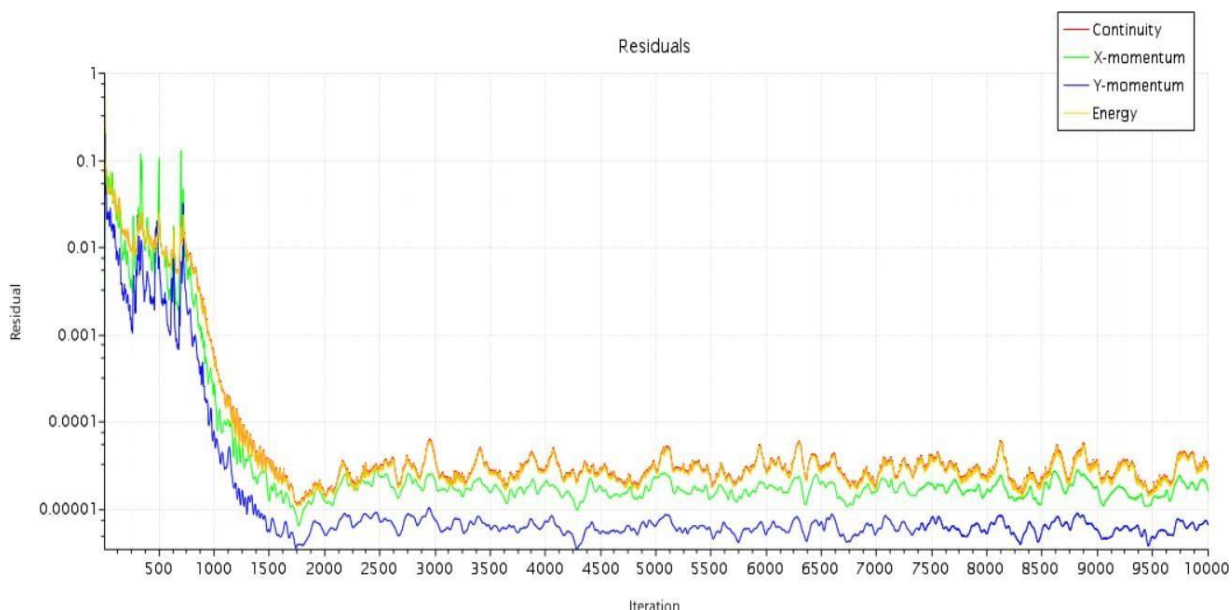

Figure 17: Residuals

Due to convective heat and gravity the cold air will be at the bottom and the hot air will move upwards. The scalar scene for the temperature and the vector scene for the pressure is shown in the figure 18& 19. The scalar scene of velocity is given in the figure 20.
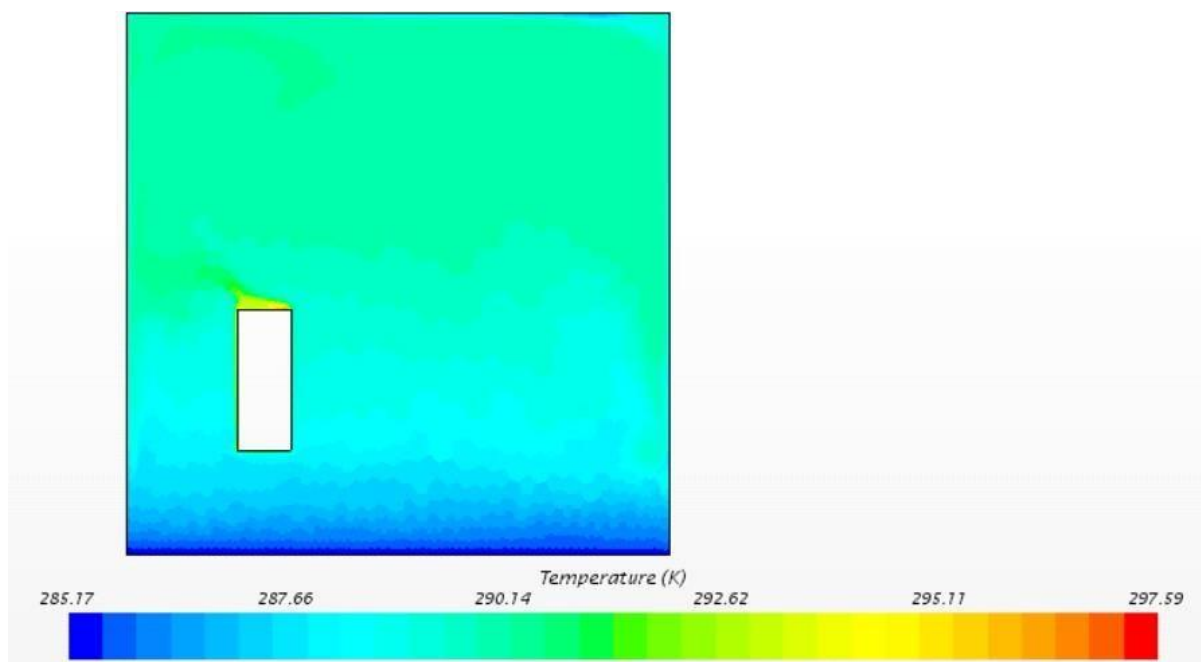

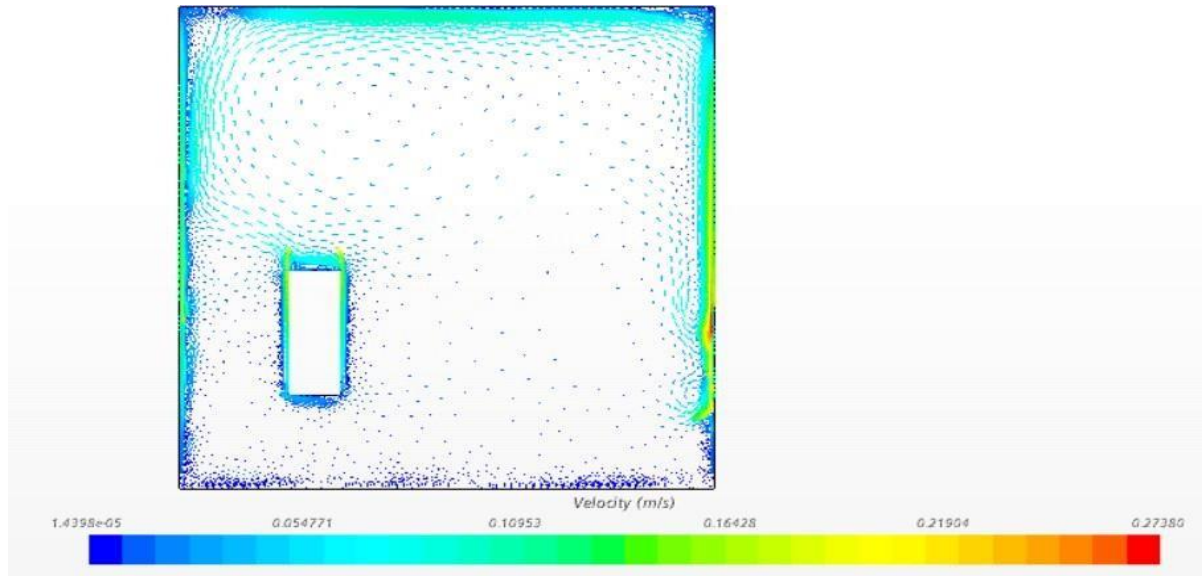*Figure 18: Temperature Scalar Scene*
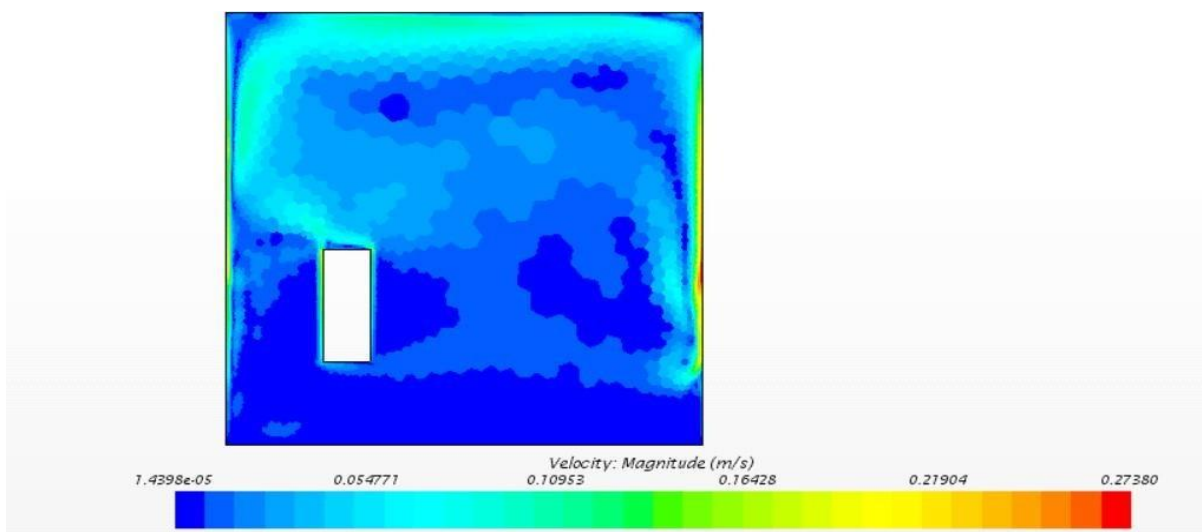
Figure 19: Velocity vector scene

Figure 20: Velocity Scalar Scene

Analysis and Discussion:

The simulation ran with zero initial velocity but when the temperature changes in the room the velocity of the flow is obtained. The increasing velocity denotes the increase in the temperature of the room. Due to gravitational effect and low density the hot air will circulate on the upper region of the room. Near the radiator there will be thinner cold region as shown in scalar scene of temperature which is due to the heating effect of the radiator. The maximum temperature is shown as 297.66 in the simulation.

Comparison With theoretical Results:

The literature paper Valero, A. d. (2015). CFD Analysis of Convective Heat Transfer Coefficient on External Surfaces of Buildings. Sustainability, 12 is considered for the comparison. In this paper convective heat transfer between the building surface and the ground is studied. The convective heat transfer coefficient will always be higher on the external side (upper) and it is lower on the internal (lower) and the simulation result obtained in the performed simulation is in accordance with the theoretical results.

CONCLUSION

The simulation in annular pipe is simulated Using STAR-CCM+ and the velocity parabola is obtained which is in accordance with the analytical results.

The simulation of NACA 0012 Air foil is done with the S-A navies stoke equation model. The result is analysed and it is compared with the Literature Result.

The Radiator in an enclosed room is simulated using STAR-CCM+ and convective heat transfer is studied. The results are analysed and compared with the literature Results.

REFERENCES

[1]. Anderson. (1999). *Introduction to flight* . McFraw-Hill. cengel-cimbala. *Fluid mechanics.* LPS.

[2]. Douvi, E. (2010). CFD CALCULATIONS OF THE FLOW OVER A NACA 0012

[3]. AIRFOIL. *Research gate* , 8.

[4]. Eleni, D. C. (2012). Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil. *Journal of Mechanical Engineering Research* , 12.

[5]. *Leap Australia*. (n.d.). Retrieved from https://www.computationalfluiddynamics.com.au/convergence-and-mesh-independent-study/

[6]. Ocin. (n.d.). *CFD Online*. Retrieved from https://www.cfd- online.com/Forums/paraview/187839-how-display-multiple-velocity-profiles.html

[7]. Pakdee, W. (2013). STAR CCM+ CFD Simulations of Enhanced Heat Transfer in High- PowerDensity Electronics Using Forced Air Heat Exchanger and Pumped Fluid LoopCold Plate Fabricated from High Thermal. *Research Gate* , 12.

[8]. Vollaro, A. d. (2015). CFD Analysis of Convective Heat Transfer Coefficient on External Surfaces of Buildings. *sustainability* , 12.