ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Performance Improvement of an Automobile Radiator Using CFD Analysis

John Babu Male
MTech Student
Kakinada Institute Of Technology & Science, Divili,
Andhra Pradesh, India.

Mr. S. Rajasekhar
Associate Professor
Kakinada Institute Of Technology & Science, Divili,
Andhra Pradesh, India.

Mr. A.V. Sridhar
Associate Professor
Kakinada Institute Of Technology & Science, Divili,
Andhra Pradesh, India.

ABSTRACT:

Radiators are used to transfer thermal energy from one medium to another for the purpose of cooling. Radiators are used for cooling internal combustion engines, mainly in automobiles but also in piston-engine aircraft, railway locomotives, motorcycles, stationary generating plant. The radiator transfers the heat from the fluid inside to the air outside, thereby cooling the fluid, which in turn cools the engine.

Research is being carried out for several decades now, in improving the performance of the heat exchangers, having high degree of surface compactness and higher heat transfer abilities in automotive industry. These compact heat exchangers have fins, louvers and tubes.

In this project we are designing a radiator without louver fins and with louver fins. The original radiator has no louver fins, we are modifying that by giving louver fins. 3D model is done in Pro/Engineer.

In this project, the computational analysis tool ANSYS is used to perform a CFD analysis on radiator. The radiator considered in this thesis is from the journal paper. The initial parameters are Inlet air velocity, Air Inlet temperature. Heat transfer analysis will also be done to analyze the heat transfer rate by changing the parameters. The material taken is Aluminum alloy 6061 for thermal analysis. The results to be validated are Velocity, Outlet air temperature, Heat carried by air, Heat transfer rate and Pressure drop. Modeling is done in Pro/Engineer / catia and analysis is done in Ansys.

I. INTRODUCTION

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, radiators generally transfer the bulk of their heat via convection, not by thermal radiation, though the term "convector" is used more narrowly; see radiation and convection, below.

The Roman hypocaust, a type of radiator for building space heating, was described in 15 AD. The heating radiator was invented by Franz San Galli, a Polish-born Russian businessman living in St. Petersburg, between 1855 and 1857.

WORKING OF AUTOMOBILE RADIATORS

Almost all automobiles in the market today have a type of heat exchanger called a radiator. The radiator is part of the cooling system of the engine as shown in Figure below. As you can see in the figure, the radiator is just one of the many components of the complex cooling system.

Most modern cars use aluminum radiators. These radiators are made by brazing thin aluminum fins to flattened aluminum tubes. The coolant flows from the inlet to the outlet through many tubes mounted in a parallel arrangement. The fins conduct the heat from the tubes and transfer it to the air flowing through the radiator.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The tubes sometimes have a type of fin inserted into them called a tabulator, which increases the turbulence of the fluid flowing through the tubes. If the fluid flowed very smoothly through the tubes, only the fluid actually touching the tubes would be cooled directly. The amount of heat transferred to the tubes from the fluid running through them depends on the difference in temperature between the tube and the fluid touching it. So if the fluid that is in contact with the tube cools down quickly, less heat will be transferred. By creating turbulence inside the tube, all of the fluid mixes together, keeping the temperature of the fluid touching the tubes up so that more heat can be extracted, and all of the fluid inside the tube is used effectively.

Radiators usually have a tank on each side, and inside the tank is a transmission cooler. In the picture above, you can see the inlet and outlet where the oil from the transmission enters the cooler. The transmission cooler is like a radiator within a radiator, except instead of exchanging heat with the air, the oil exchanges heat with the coolant in the radiator.

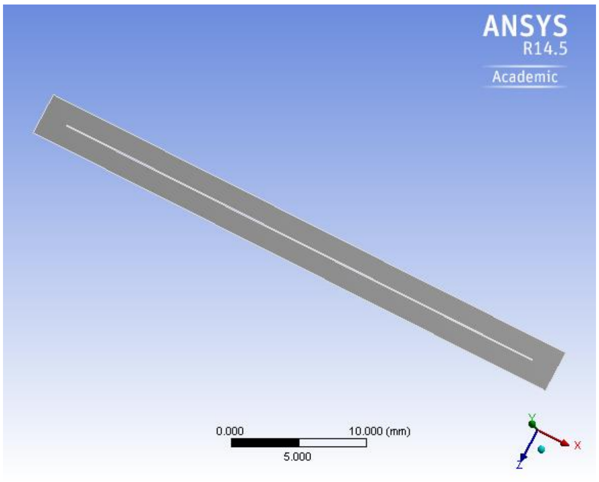
CFD ANALYSIS OF RADIATOR

ORIGINAL MODEL – WITHOUT LOUVERED FINS

Save Pro-E Model as .iges format.

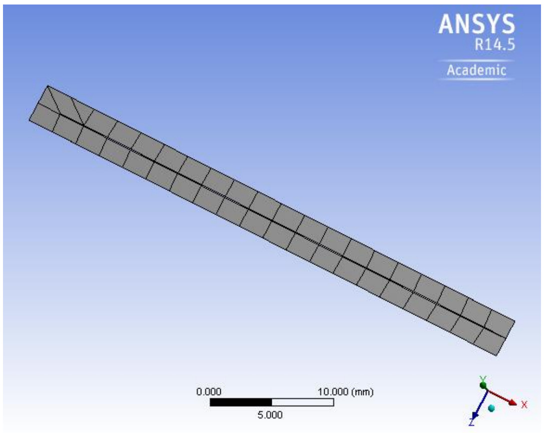
→→Ansys → Workbench→ Select analysis system → Fluid Flow (Fluent) → double click

→→Select geometry → right click → import geometry → select browse →open part → ok



→→ select mesh on work bench → right click →edit

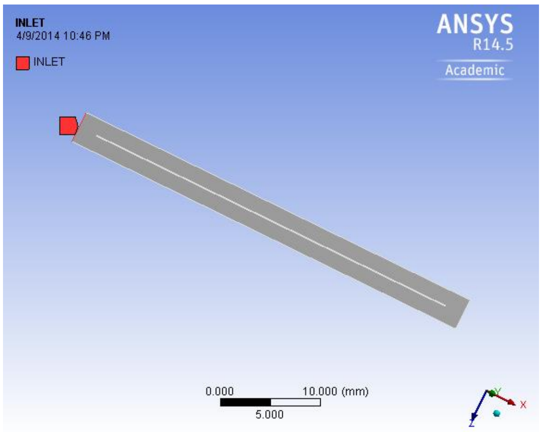
Select mesh on left side part tree → right click → generate mesh →

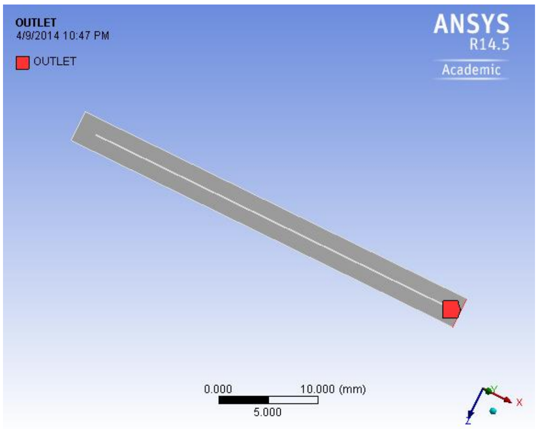


SPECIFYING BOUNDARIES FOR INLET AND OUTLET

Select edge → right click → create named section → enter name → inlet

Select edge → right click → create named section → enter name → outlet





File →export → fluent →input file(mesh) → enter required name → save.

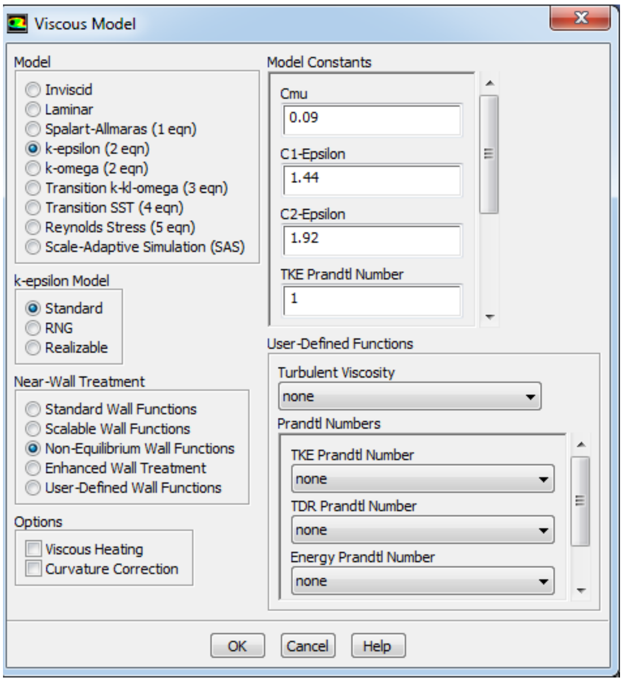
→→ ansys → fluid dynamics → fluent → select 2D or 3D → select working directory → ok

→→file → read → mesh → select file → ok.

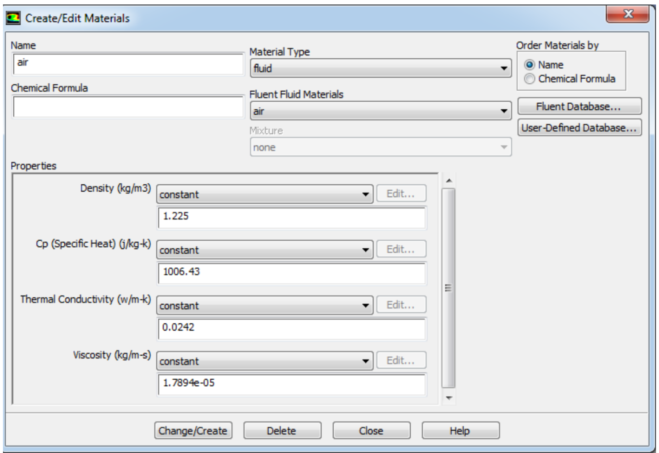
General →Pressure based

Model → energy equation → on

Model → Viscous → Edit

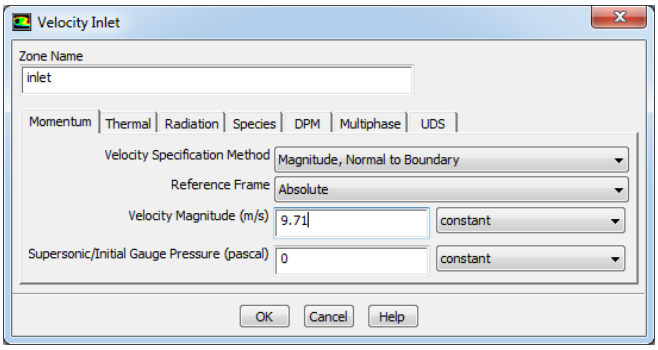


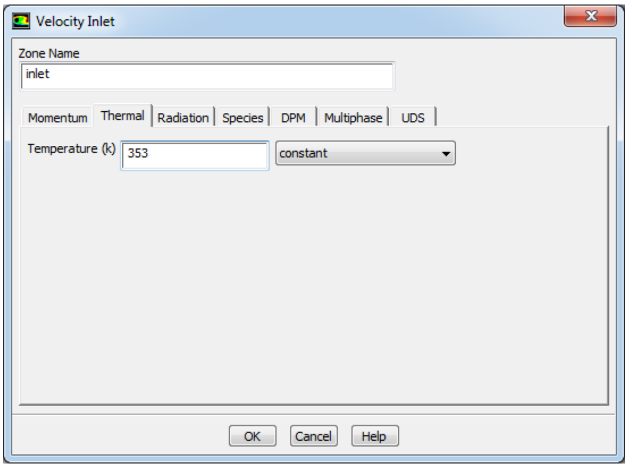
Materials → new → create or edit → specify Fluid material → Air



Boundary conditions → Inlet → Edit

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

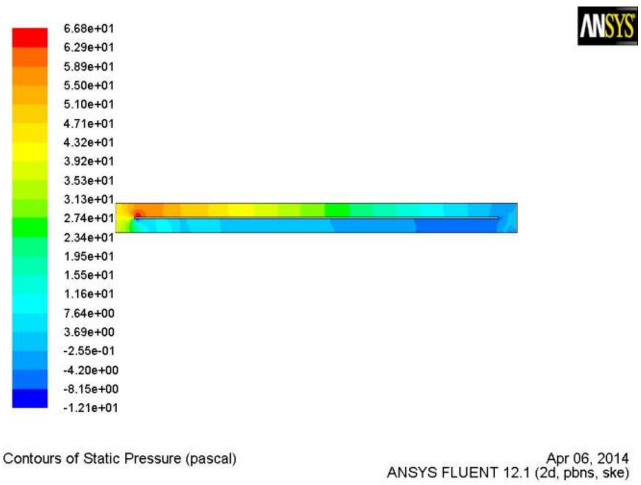




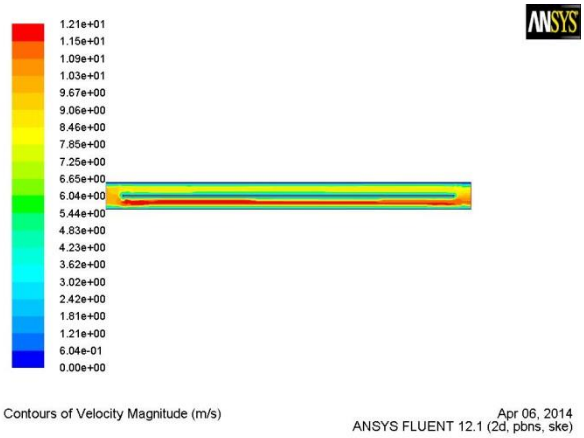
Solution → Solution Initialization→ Hybrid Initialization →done

Run calculations → No of iterations = 100 → calculate → calculation complete

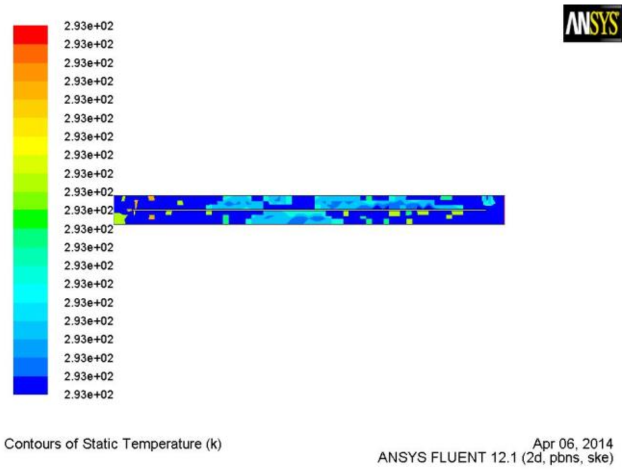
Pressure



VELOCITY



TEMPERTAURE



"Flux Report"

Total Heat Transfer Rate	**(w)**
air_inlet	-185.60265
air_outlet	185.3676
wall-part_1	0
Net	-0.23504639

Mass Flow Rate	**(kg/s)**
air_inlet	0.038023997
air_outlet	-0.037975796
interior-part_1	-0.75629127
wall-part_1	0
Net	4.8201531e-05

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

WITH LOUVERED FINS

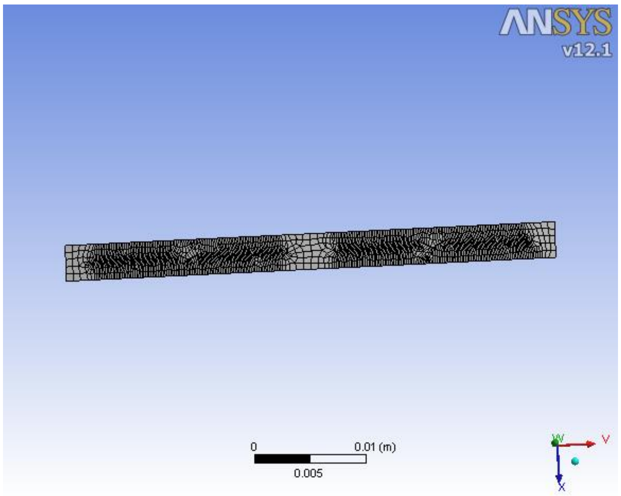
Save Pro-E Model as .iges format.

→→Ansys → Workbench→ Select analysis system → Fluid Flow (Fluent) → double click

→→Select geometry → right click → import geometry → select browse →open part → ok

→→ select mesh on work bench → right click →edit

Select mesh on left side part tree → right click → generate mesh →



SPECIFYING BOUNDARIES FOR INLET AND OUTLET

Select edge → right click → create named section → enter name → inlet

Select edge → right click → create named section → enter name → outlet

File →export → fluent →input file(mesh) → enter required name → save.

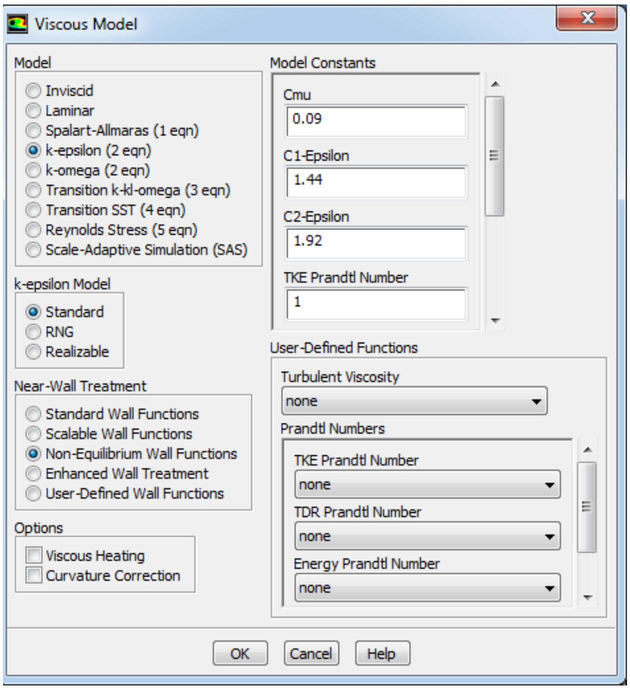
→→ ansys → fluid dynamics → fluent → select 2D or 3D → select working directory → ok

→→file → read → mesh → select file → ok.

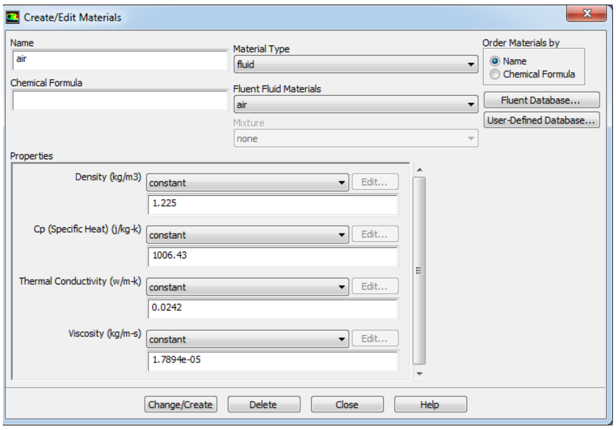
General →Pressure based

Model → energy equation → on

Model → Viscous → Edit

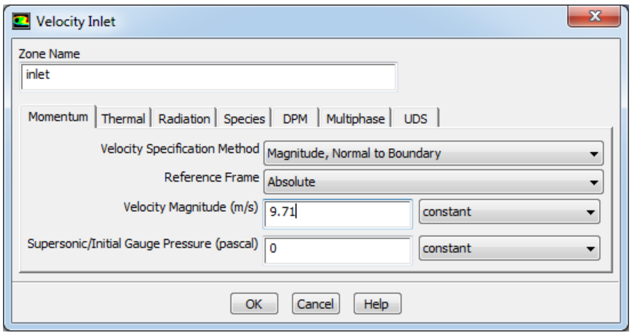


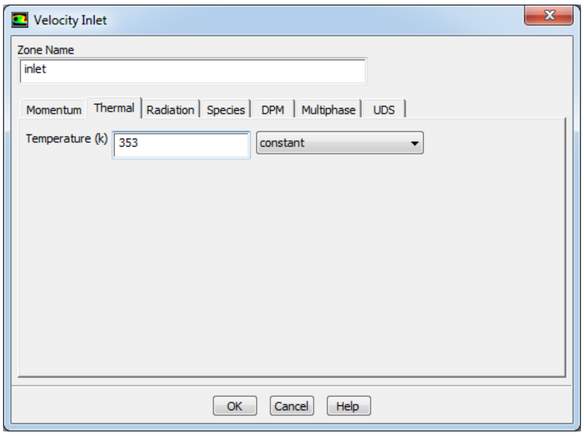
Materials → new → create or edit → specify Fluid material → Air



Boundary conditions → Inlet → Edit

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal



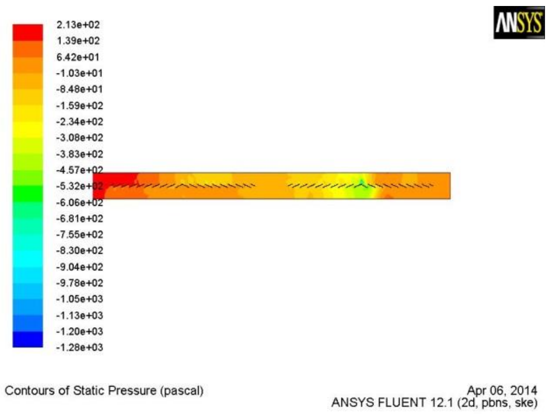


Solution → Solution Initialization→ Hybrid Initialization →done

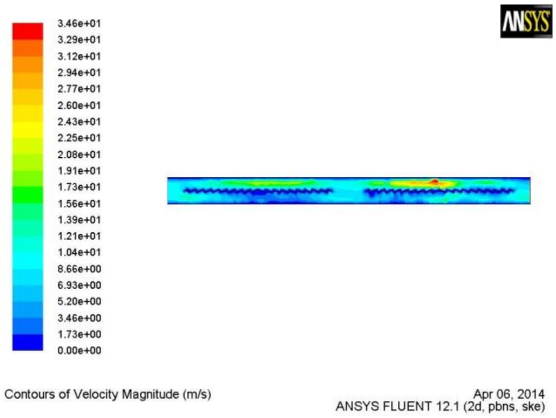
Run calculations → No of iterations = 100 → calculate → calculation complete

→→ Results → graphics and animations → contours → setup

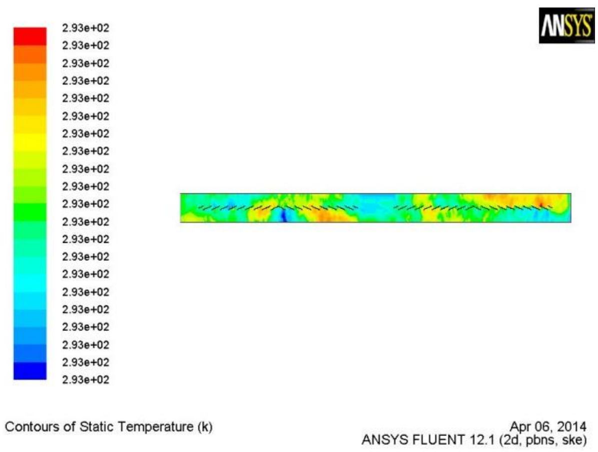
PRESSURE



VELOCITY MAGNITUDE



TEMPERTAURE



"Flux Report"

Mass Flow Rate	(kg/s)
air_inlet	0.038063198
air_outlet	-0.036695994
interior-part_1	2.3720436
wall-part_1	0
Net	0.0013672039

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Total Heat Transfer Rate	(w)
air_inlet	-185.79402
air_outlet	179.12578
wall-part_1	0
Net	-6.6682434

THERMAL ANALYSIS

WITHOUT LOUVER FINS

Set Units - /units,si,mm,kg,sec,k

File- change Directory-select working folder

File-Change job name-Enter job name

Select element-Solid-20node 90

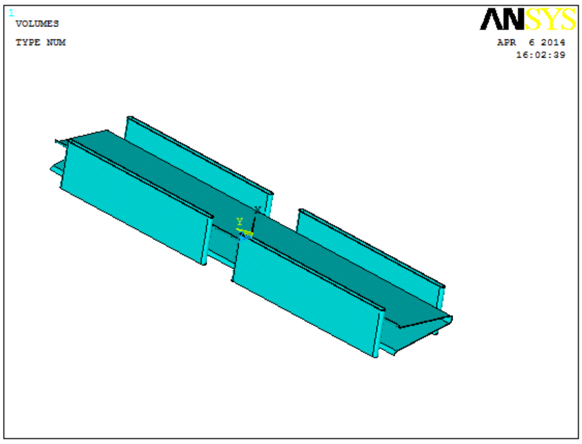
Material Properties – Aluminum Alloy 6061

Density – 0.0000027 Kg/mm^3

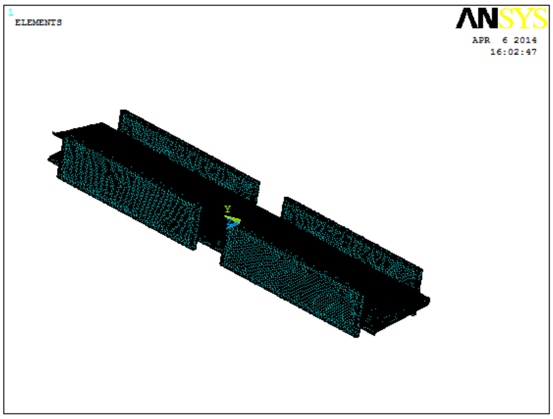
Thermal Conductivity – 180W/mK

Specific Heat – 896 J/Kg K

Imported Model



Meshed model



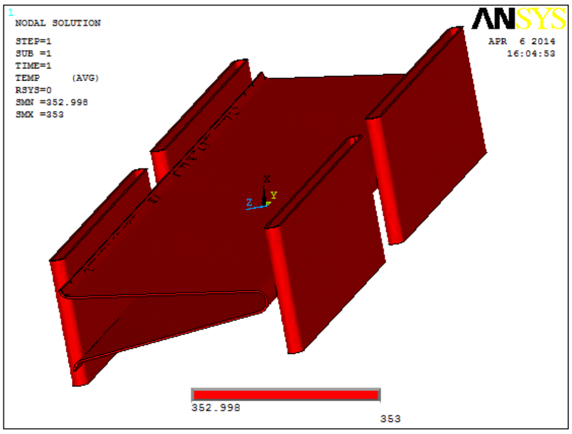
Apply Thermal-Temperature- on Area=353K

Convections – on Area-Film Co-efficient – 0.0000131 W/mm^2 K

Bulk Temperature – 303 K

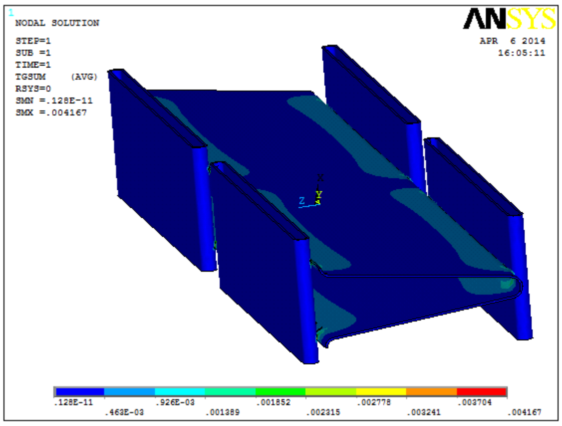
Results

Temperature

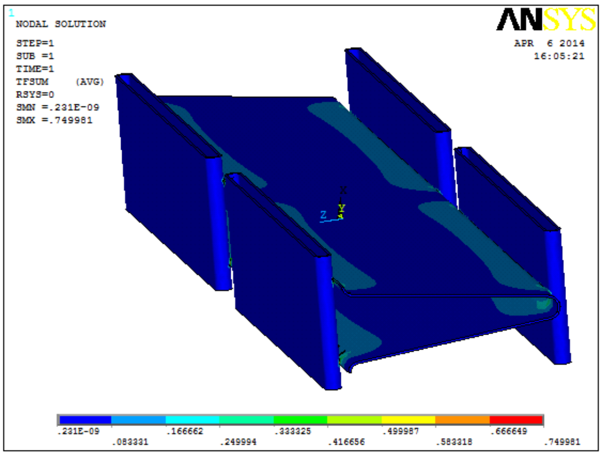


ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Thermal gradient



Heat flux



WITH LOUVER FINS

Set Units - /units,si,mm,kg,sec,k

File- change Directory-select working folder

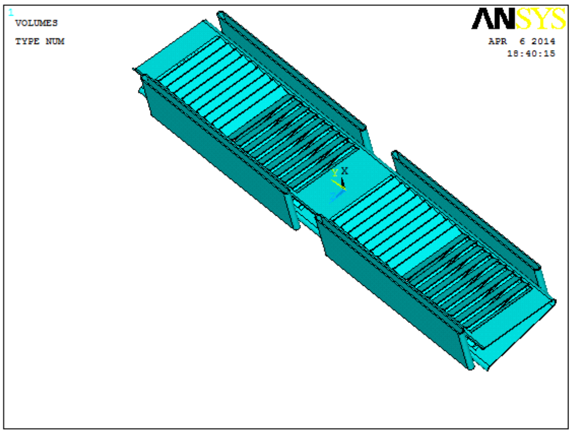
File-Change job name-Enter job name

Select element-Solid-20node 90

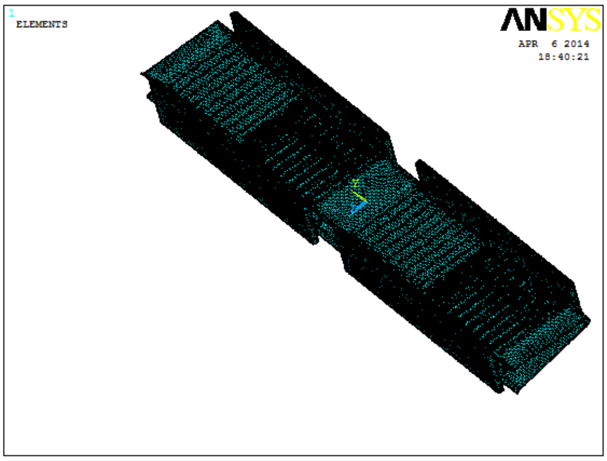
Material Properties

Youngs Modulus =

Imported



Meshed model



Loads

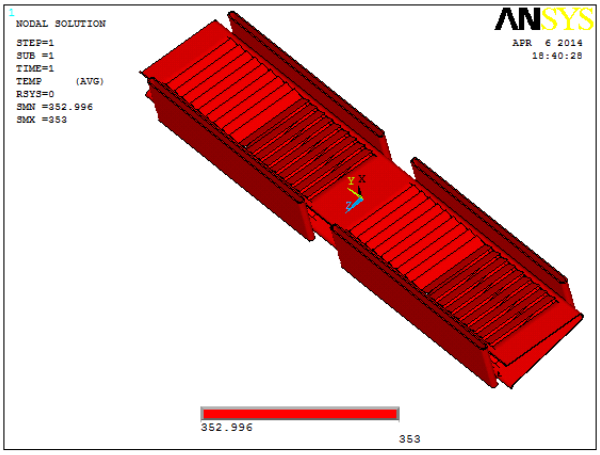
Apply Thermal-Temperature- on Area=353K

Convections – on Area-Film Co-efficient – 0.0000131 W/mm^2 K

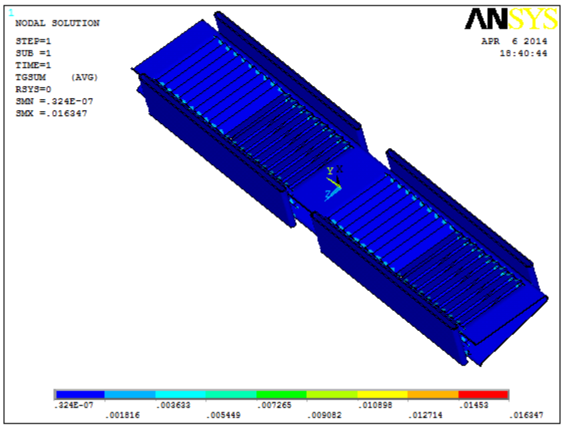
Bulk Temperature – 303 K

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

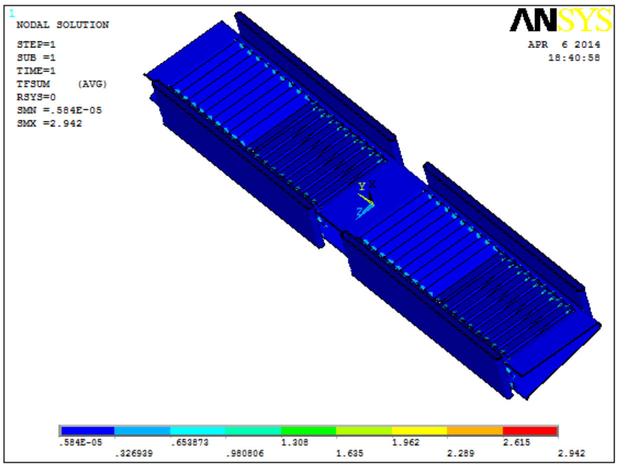
Temperature



Thermal gradient



Heat flux



RESULTS TABLE

CFD ANALYSIS

	With louvers	Without louvers
Pressure (Pa)	2.13e+02	1.08e+02
Velocity (m/s)	3.46e+01	1.21e+01
Temperature (K)	2.93e+02	2.93e+02
Mass Flow Rate (Kg/S)	0.00136	0.235
Total Heat Transfer rate (W)	6.668	4.8

Thermal Results

	With louvers	Without louvers
Temperature (K)	353	353
Thermal Gradient (K/mm)	0.16347	0.004167
Heat Flux (W/mm^2)	2.942	0.749981

CONCLUSION

In this project we are designing a radiator without louver fins and with louver fins. The original radiator has no louver fins, we have modified the design by specifying louver fins. 3D model is done in Pro/Engineer.

In this project, the computational analysis tool ANSYS is used to perform a CFD analysis on radiator. The radiator considered in this thesis is from the journal paper. The initial parameters are Inlet air velocity, Air Inlet temperature.

By observing the analysis results, the velocity, pressure and heat transfer rate is more for the radiator with louver fins that of the original model.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Heat transfer analysis is done to analyze the heat transfer rate to determine the thermal flux. The material taken is Aluminum alloy 6061 for thermal analysis.

By observing the thermal analysis results, thermal flux is more for the radiator with louver fins that of the original model, so heat transfer rate is more.

So we can conclude that modifying the radiator model with louver fins yields better results.

REFRENCES

[1] R. Saidur, K.Y. Leong and H.A. Mohammad, A Review on Applications and Challenges of Nanofluids. Renewable and Sustainable Energy Reviews, 15, 3 (2011), 1646–1668.

[2] Pelaez, R.B., Ortega, J.C., Cejudo-Lopez, J.M., A three-dimensional numerical study and comparison between the air side model and the air/water side model of a plain fin and tube heat exchanger, Applied Thermal Engineering, 30 (2010), pp.1608-1615.

[3] Sahin, H.M., Dal, A.R., Baysal, E., 3-D Numerical study on correlation between variable inclined fin angles and thermal behavior in plate fin-tube heat exchanger, Applied Thermal Engineering, 27 (2007), pp.1806-1816.

[4] Wen, M.Y. Ho, C.Y., Heat transfer enhancement in fin and tube heat exchanger with improved fin design, Applied Thermal Engineering, 29(2009), pp.1050-1057.

[5] Yan, W.M., Sheen, P.J., Heat transfer and friction characteristics of fin and tube heat exchangers, International Journal of Heat and Mass Transfer, 43 (2000), pp.1651-1659.

[6] Wolf, I., Frankovic, B., Vilicic, I., A numerical and experimental analysis of neat transfer in a wavy fin and tube heat exchanger, Energy and the Environment (2006) pp.91-101.

AUTHOR DETAILS

1. **STUDENT**

JOHN BABU MALE received the BTech degree in mechanical engineering from DR SAMUEL GEORGE INSTITUTE OF ENGINEERING AND TECHNOLOGY, JNTUK, MARKAPUR, Andhra Pradesh, India, in 2010 year, and pursuing MTech in Thermal Engineering from Kakinada institute of technology & science, Divili, Andhra Pradesh, India.

2. **GUIDE 1**

Mr. S. Rajasekhar, Associate professor, Kakinada institute of technology & science, Divili, Andhra Pradesh, India.

3. **GUIDE 2**

Mr. A.V. Sridhar, Associate professor, Kakinada institute of technology & science, Divili, Andhra Pradesh, India.