Finite Element Analysis Of Force Fed Micro Channels For High Flux Cooling Applications

Mr. Bojinkula Rambabu[1], Mr. M.John Babu[2], Mr. V.Satyanarayana[3]
[2, 3]Assistant professor Dept of Mechanical
[1, 2, 3] Kakinada institute of technology and science, Divili

Abstract- *the computational fluid dynamics analysis of mixed flow of hot and cold water in T- pipes and it is mainly concentrated on analytical approach to the areas where Pipes (used for flow) are mostly susceptible to damage. The simulation is done on T-pipe by placing the nozzle at three different places, to know the pressure, temperature and velocity contours throughout the flow and comparison was made. These T- pipes are mostly used in nuclear reactor cooling system to reduce the heat in the nuclear reactors by mixing hot and cold water, where the mixing will takes place efficiently. The 2D model of the pipe is made by GAMBIT and analysis is to be carried out by using K-Epsilon in FLUENT software.*

Keywords- Mixing hot and cold fluid, Different types of T-pipes, Nozzles,Using K-Epsilon in FLUENT software.

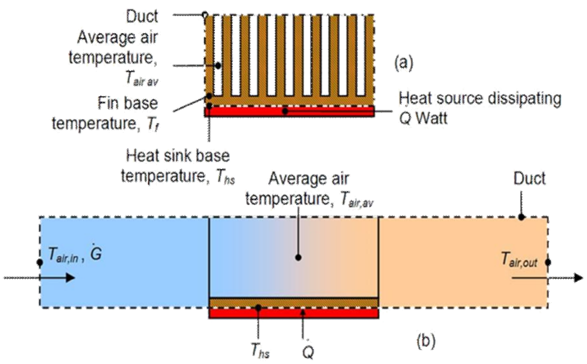
I. INTRODUCTION

HEAT SINK is a passive heat exchanger that cools a device by dissipating heat into thesurrounding medium. In computers, heat sinks are used to cool central processingunits or graphics processors. Heat sinks are used with high-power semiconductor devices such as power transistors and optoelectronics such as lasers and light emitting diodes (LEDs), where the heat dissipation ability of the basic device is insufficient to moderate its temperature.

A heat sink is designed to maximize its surface area in contact with the cooling medium surrounding it, such as the air. Air velocity, choice of material, protrusion design and surface treatment are factors that affect the performance of a heat sink. Heat sink attachment methods and thermal interface materials also affect the die temperature of the integrated circuit. Thermaladhesive or thermal grease improve the heat sink's performance by filling air gaps between the heat sink and the heat spreader on the device.

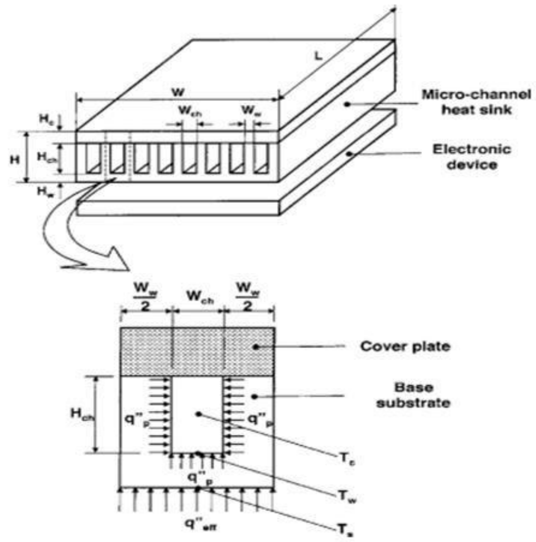
II. HEAT TRANSFER PRINCIPLE

A heat sink transfers thermal energy from a higher temperature device to a lower temperature fluid medium. The fluid medium is frequently air, but can also be water, refrigerants or oil. If the fluid medium is water, the heat sink is frequently called a cold plate. In thermodynamics a heat sink is a heat reservoir that can absorb an arbitrary amount of heat without significantly changing temperature. Practical heat sinks for electronic devices must have a temperature higher than the surroundings to transfer heat by convection, radiation, and conduction. The power supplies of electronics are not 100% efficient, so extra heat is produced that may be detrimental to the function of the device. As such, a heat sink is included in the design to disperse heat to improve efficient energy use



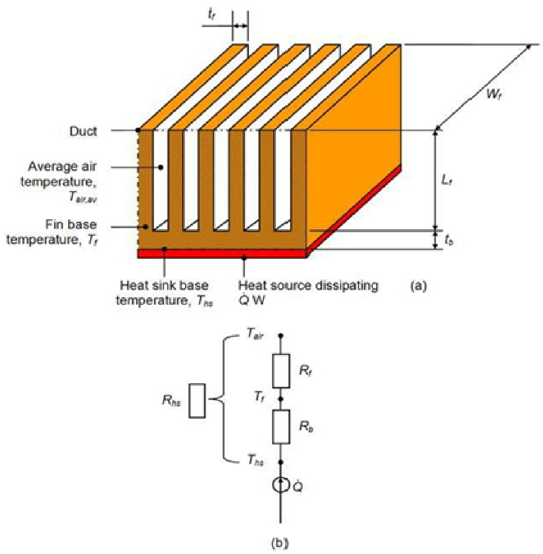
III. STUDIES ANDFINDINGS

The present study was motivated by the lack of a systematic methodology to guide two-phase micro-channel heat sink design. A new methodology is proposed wherein key heat sink parameters and the best available predictive tools are first identified. Different performance characteristics of the heat sink are then examined to optimize micro-channel dimensions in pursuit of acceptable values for the thermal/fluid parameters corresponding to a given heat flux, coolant, and overall dimensions of the heat generating device to which the heat sink is attached.

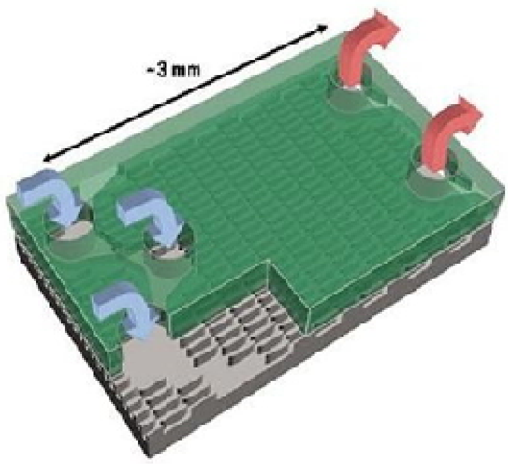


Construction of typical two-phase micro-channel heat sink

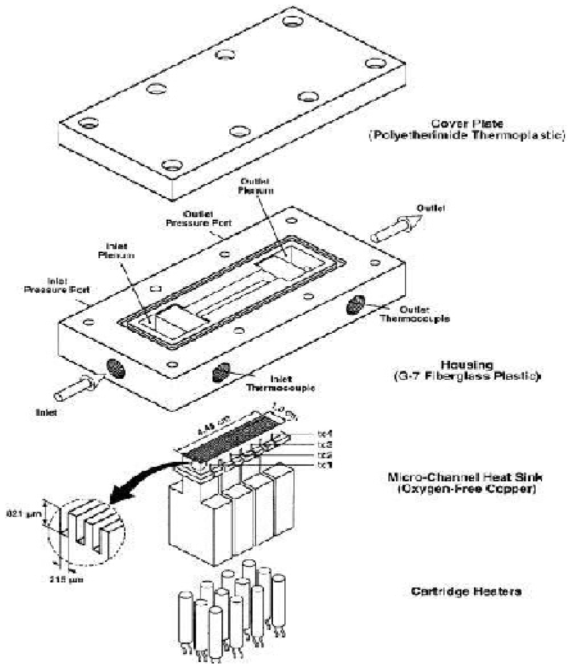
Heat generated by the device is conducted through the base substrate and removed by the coolant flowing through the micro-channels. A unit cell containing a single micro-channel and surrounding solid. Taking advantage of symmetry between micro-channels, the cooling performance of the heat sink can be derived from analysis of the unit cell alone. A variety of system parameters must be examined when designing a two-phase micro-channel heat sink. As shown below, these parameters can be grouped into 1. geometrical parameters, 2. operating parameters, and 3. Thermal/fluid parameters.



Liquid-cooled micro channel heat sinks and coolers have been shown to be a very effective way to remove high heat load. A large heat transfer coefficient can be achieved by reducing the channel hydraulic diameter. In a confined geometry the small flow rate within micro channels produces laminar (smooth) flow, which results in a heat transfer coefficient inversely proportional to the hydraulic diameter. In other words, the narrower the channels in the heat sink, the higher the heat transfer coefficient.



Micro channel Cooler Showing Heat Exchanger Zones

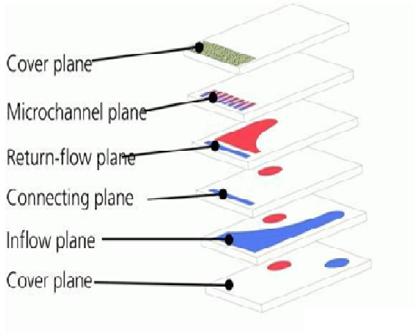


Micro channel Heat Sink Test Module

Micro-channel heat sink technology is a technology which has been standardized for manyyears in the field of high-power diode lasers.

The special feature is that the laser elements are cooled using de-ionized water which flows through the micro-channel heat sinks in the heat sink of each individual laser element. This enables extremely efficient cooling of the laser in case of operation in high power ranges and the resulting great heat generation.

However, when cooling the diode laser using de-ionized water, certain requirements need to be fulfilled with regard to the water, the cooling unit and the individual assemblies. For more information, please refer to our document "Information on Safety and Handling"



IV. INTRODUCTION TO CREO 2.0

Creo 2.0 is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with your industry and company standards. Integrated Creo 2.0 CAD/CAM/CAE solutions allow you to design faster than ever, while maximizing innovation and quality to ultimately create exceptional products.

Customer requirements may change and time pressures may continue to mount, but your product design needs remain the same - regardless of your project's scope, you need the powerful, easy-to-use, affordable solution that Creo 2.0 provides.

Creo 2.0 Benefits:

- Unsurpassed geometry creation capabilities allow superior product differentiation and manufacturability
- Fully integrated applications allow you to develop everything from concept to manufacturing within one application
- Automatic propagation of design changes to all downstream deliverables allows you to design with confidence
- Complete virtual simulation capabilities enable you to improve product performance and exceed product quality goals
- Automated generation of associative tooling design, assembly instructions, and machine code allow for maximum production efficiency

INTRODUCTION TO FEA Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variation calculus to obtain approximate solutions to vibration systems.FEA has become a solution to the task of predicting failure due to unknown stresses by showing problem areas in a material and allowing designers to see all of the theoretical stresses within. This method of product design and testing is far superior to the manufacturing costs which would accrue if each sample was actually built and tested.

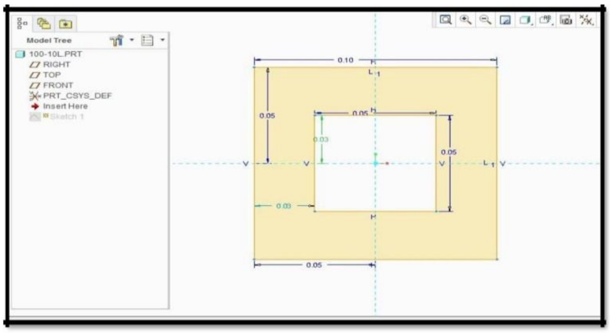
ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions.Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. Such loads include the following:

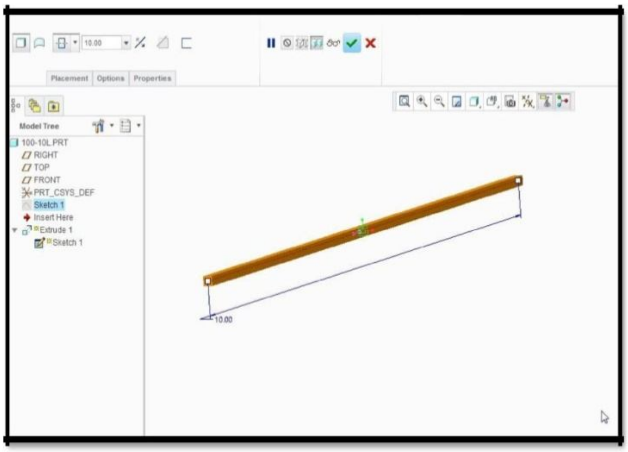
Convection
Radiation
Heat flow rates
Heat fluxes (heat flow per unit area)
Heat generation rates (heat flow/unit vol)
Constant temperature boundaries

INTRODUCTION TO CFD Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.

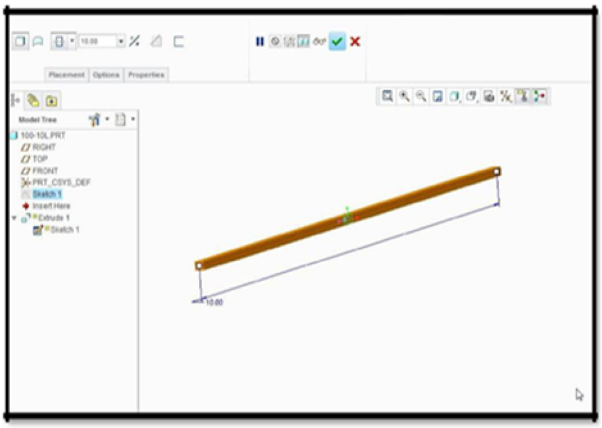
VI 3D MODELING OF MICROCHANNEL HEAT SINK 10mm LENGTH

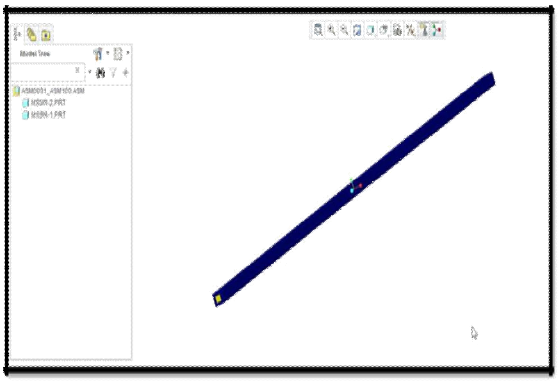
WIDTH 50µm & HEIGHT 100µm





3D model of micro channel heat sink with dimensions 100 μm * 50 μm * 10mm

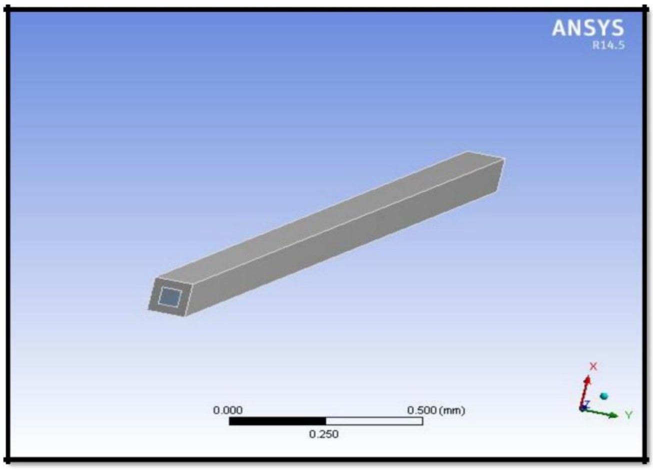




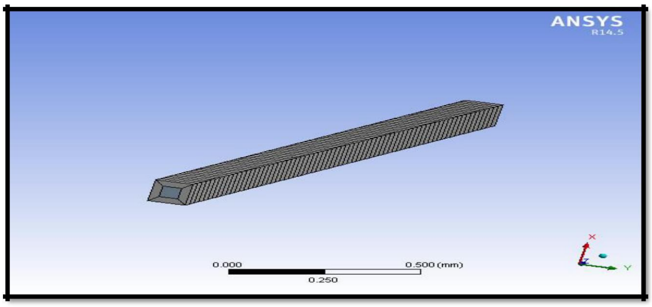
Assembly of micro channel heat sink with fluid area with dimensions 100 μm * 50 μm * 10mm

CFD ANALYSIS OF MICROCHANNEL HEAT SINK 10mm LENGTH WIDTH 50μm & HEIGHT 100μm FLUID –WATER

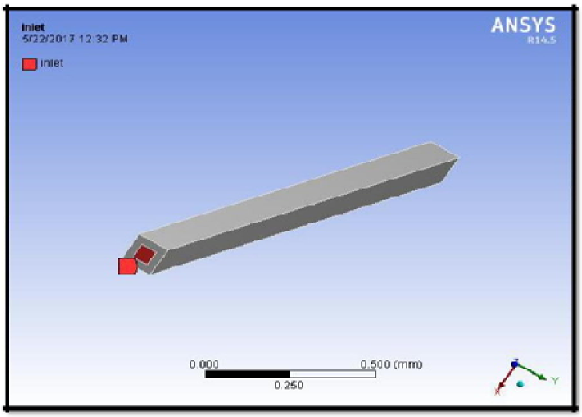
Save Pro/E Model as .iges format
→Ansys → Workbench→ Select analysis system → Fluid Flow (Fluent) → double click →→Select geometry → right click → import geometry → select browse →open part → ok
Import model



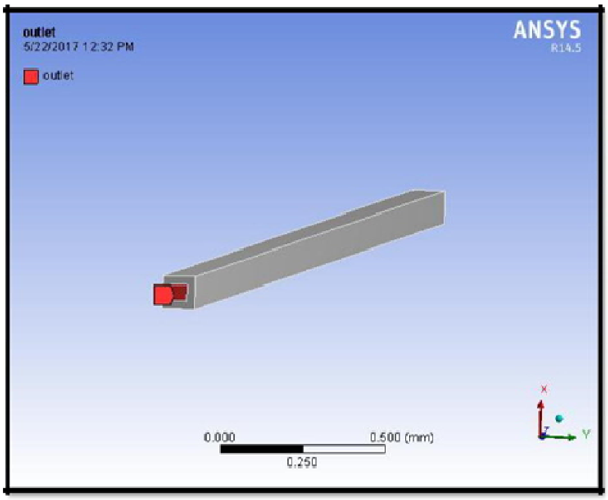
Meshed model



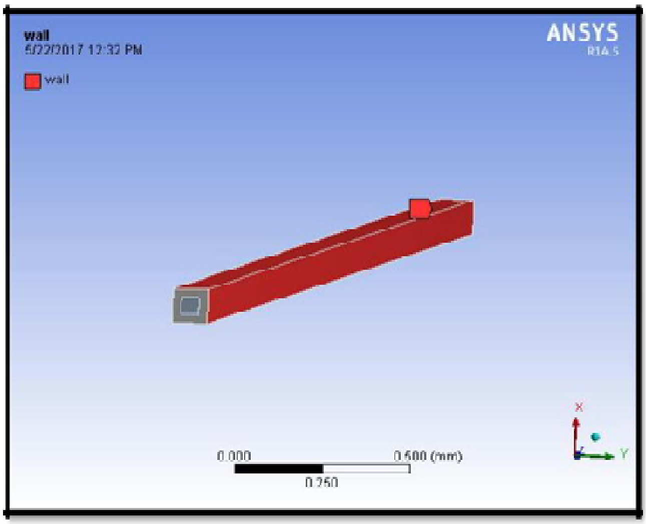
Inlet



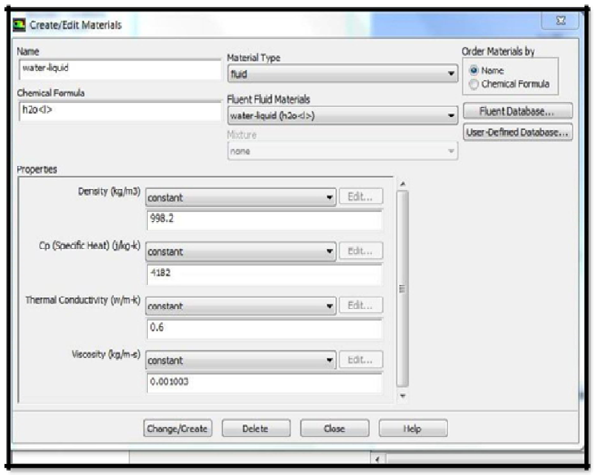
OUTLET

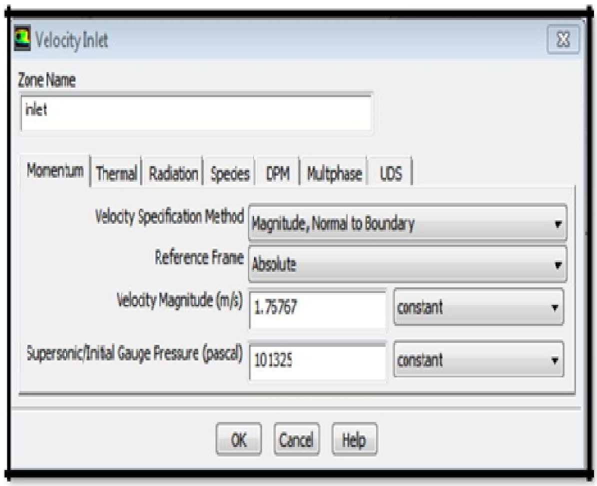


WALL

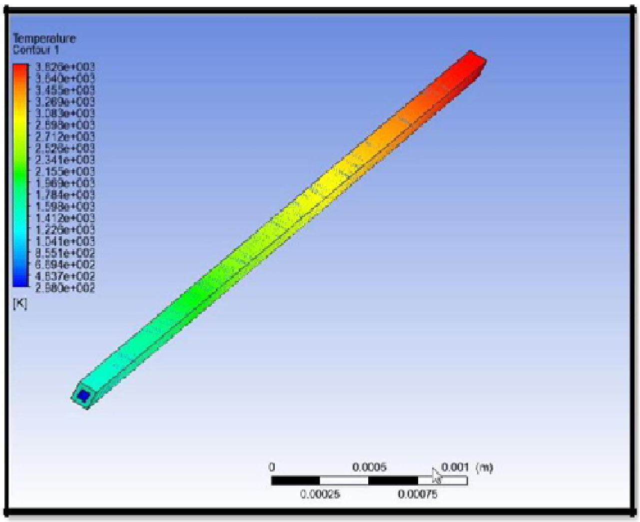


Select fluid Water

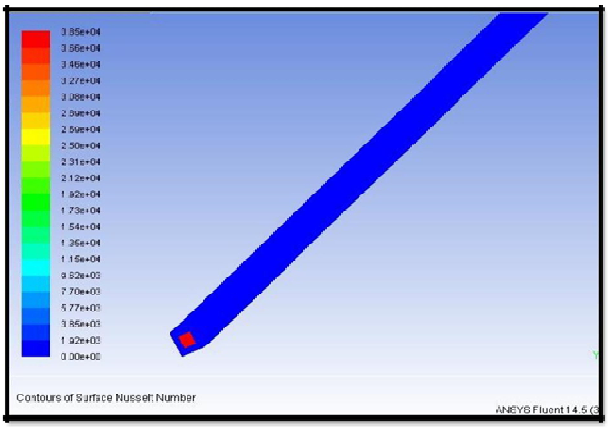




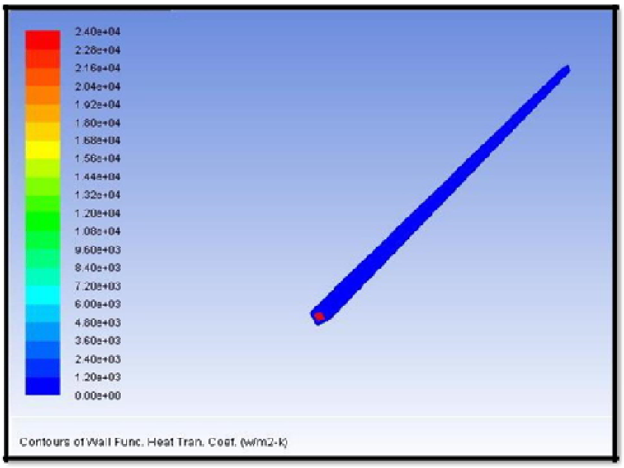
Static Temperature



NUSSELT NUMBER



Heat transfer coefficient

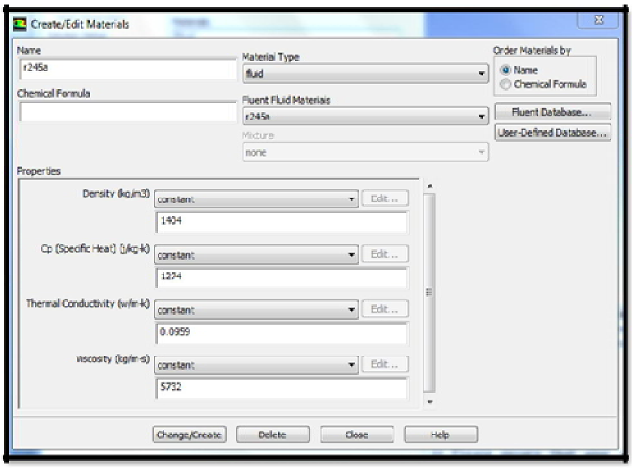


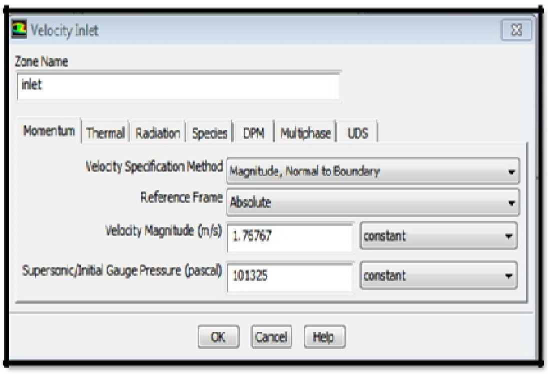
"Flux Report"

Total Heat Transfer Rate	(w)
contact_region-src	0
contact_region-trg	0
inlet	-0.0034991279
outlet	-49.196571
wall	49.2
wall-13	0
wall-14	0
wall-7	-49.2
wall-7-shadow	49.2
wall-____msbr	0
Net	-7.0063439e-05

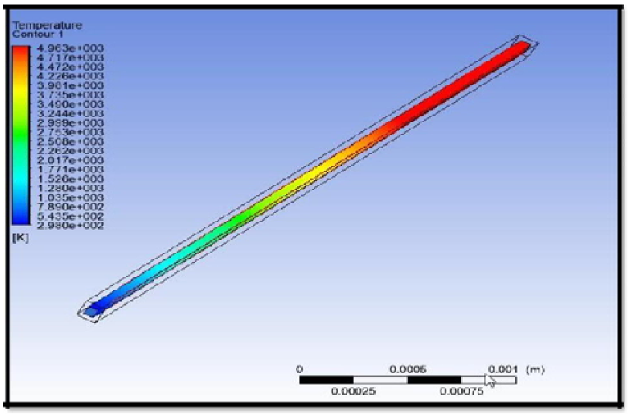
FLUID –R245A

Select fluid R245A

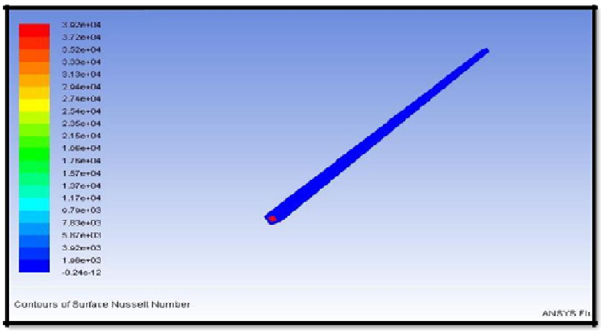




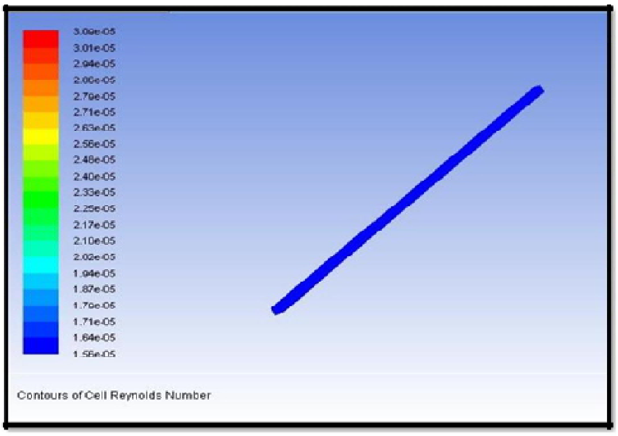
setup Static Temperature



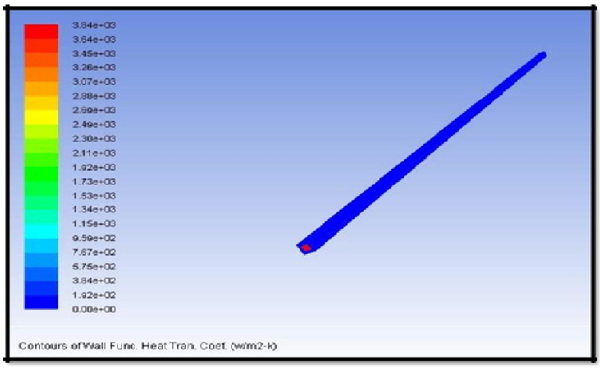
NUSSELT NUMBER



Reynolds number



Heat transfer coefficient

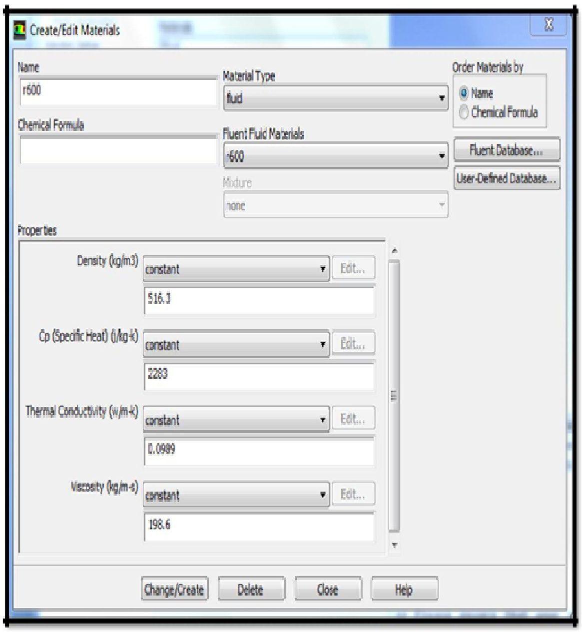


"Flux Report"

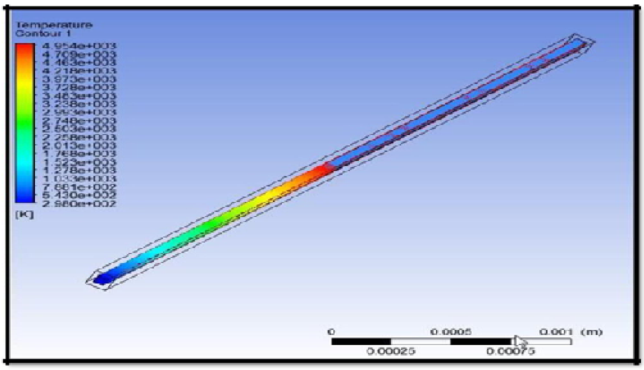
Total Heat Transfer Rate	(w)
contact_region-src	0
contact_region-trg	0
inlet	-0.0014832973
outlet	-37.166642
wall	49.2
wall-13	0
wall-14	0
wall-7	-12.31542
wall-7-shadow	12.31542
wall-____msbr	0
Net	12.031874

FLUID –R600A

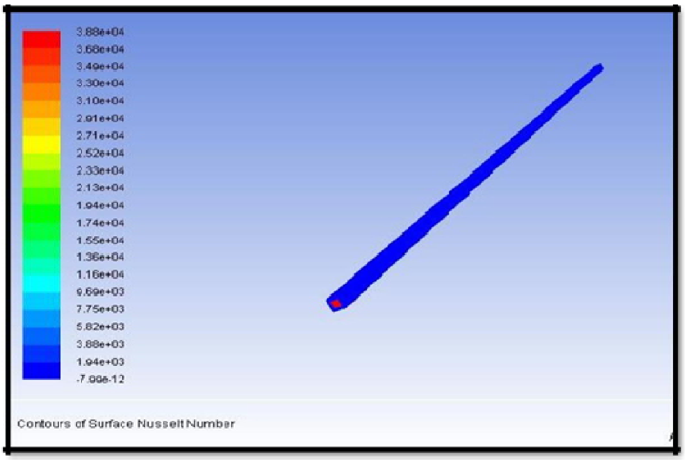
Select fluid R600A



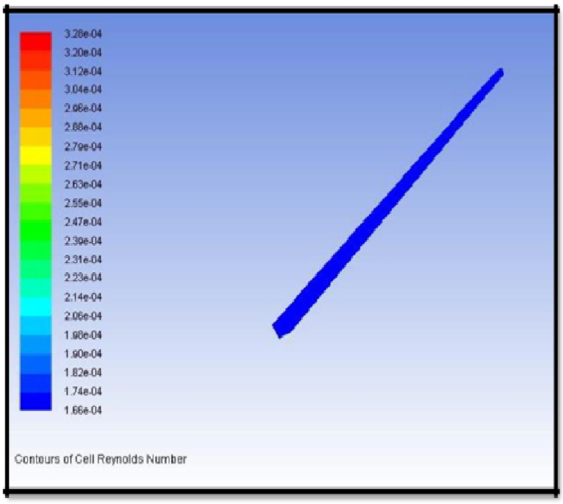
Static Temperature



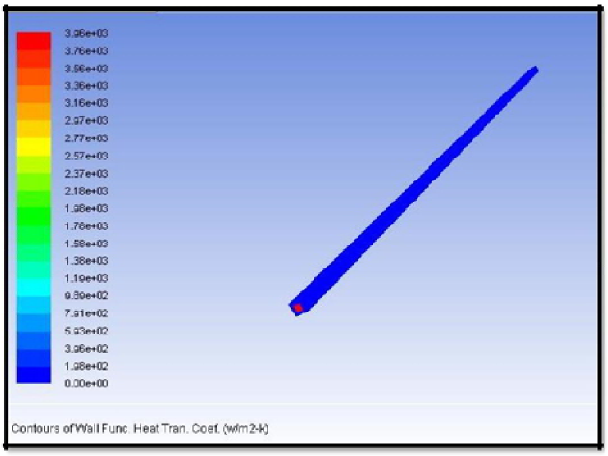
NUSSELT NUMBER



Reynolds number



Heat transfer coefficient




```
                    "Flux Report"

      Total Heat Transfer Rate                         (w)
---------------------------- --------------------
          contact_region-src                             0
          contact_region-trg                             0
                       inlet                 -0.0012421905
                      outlet                    -24.492032
                        wall                          49.2
                     wall 13                             0
                     wall-14                             0
                      wall-7                    -8.4322679
               wall-7-shadow                     8.4322679
               wall-____msbr                             0
         ----------------- --------------------
                         Net                     24.706726
```

CFD ANALYSIS

Table 6.1: Length 10 mm
Fluid – Water

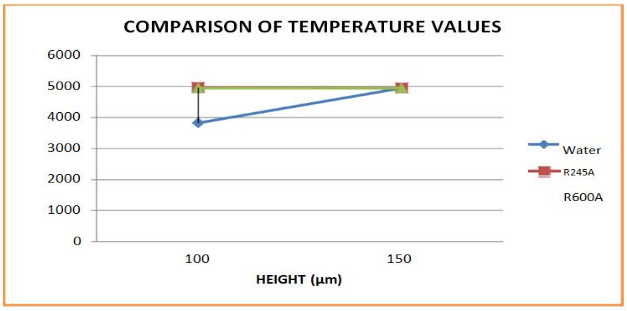
(µm)	Temperature (K)	NUSSELT NUMBER	REYNOLDS NUM	heat transfer coeff	HEAT TRANSFER RATE
height 100 µm	3.826 e+03	3.85e+04	1.26e+02	2.40e+04	7.00634 e-05
Height 150 µm	4.954e +03	3.86e+04	1.26e+02	2.40e+04	0.115 2642

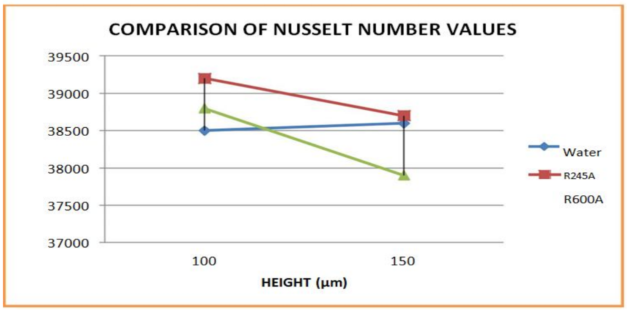
FLUID-R245A

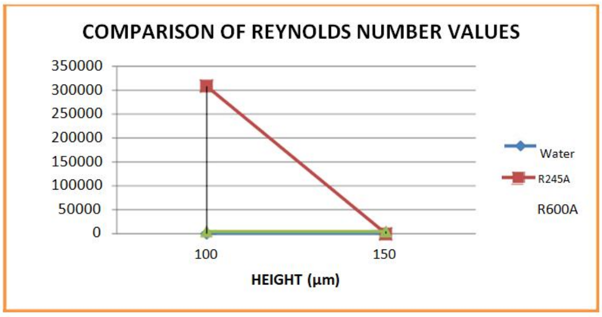
(µm)	Temperature (K)	NUSSELT NUMBER	REYNOLDS NUM	heat transfer coeff	HEAT TRANSFER RATE
height 100 µm	4.963e +03	3.92e+04	3.09e-05	3.84e+03	12.0318 74
Height 150 µm	4.954e +03	3.87e+04	3.09e-5	3.84e+03	36.6317 42

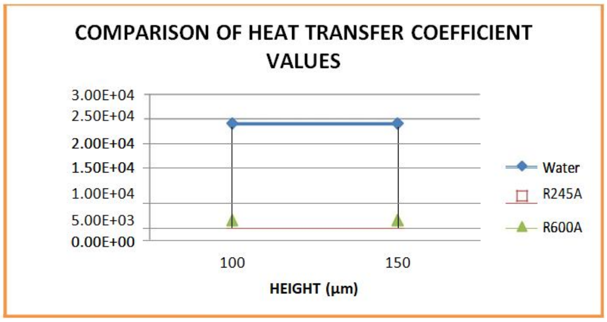
FLUID-R600A

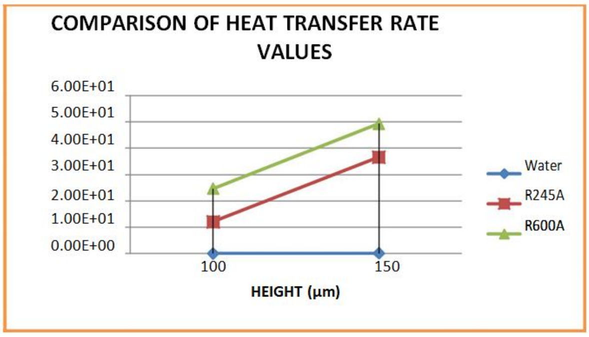
(µm)	Temperature (K)	NUSSELT NUMBER	REYNOLDS NUM	heat transfer coeff	HEAT TRANSFER RATE
height 100 µm	4.954e +03	3.88e+04	3.28e-04	3.96e+03	24.7067 26
Height 150 µm	4.954e +03	3.79e+04	3.28e-04	3.96e+03	49.3064 75





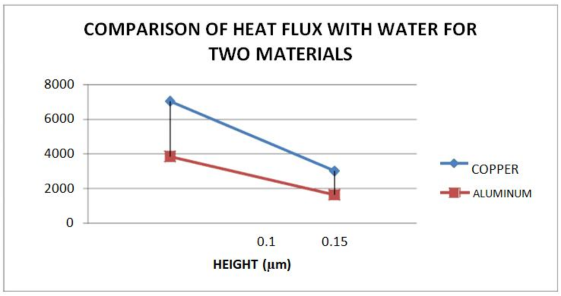


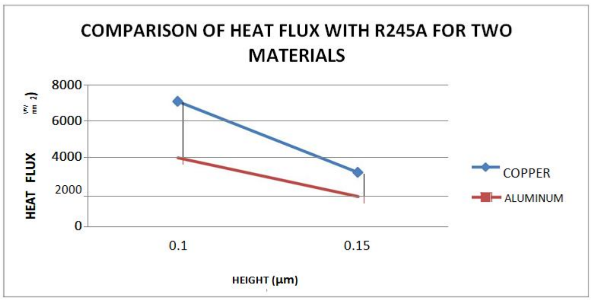


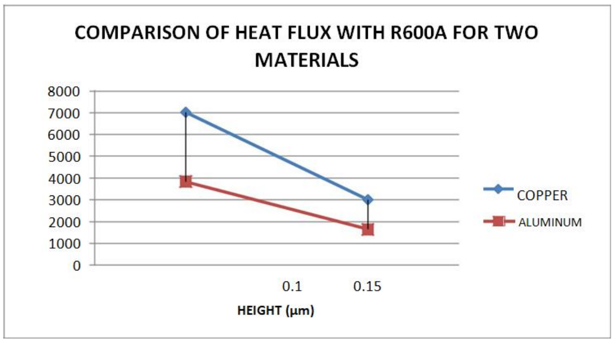


THERMAL ANALYSIS:10mm Length

			Temperature (K)	Heat flux (W/mm^2)
COPPER	Width 50µm & height 100µm	Water	365	7041.6
		R245a	365	7043.6
		R600	365	7019
	Width 50µm & Height 150µm	Water	365	3013.8
		R245a	365	3014.1
		R600	365	3009.7
ALUMINUM	Width 50µm & height 100µm	water	365	3842
		R245a	365	3842.5
		R600	365	3835.2
	Width 50µm & Height 150µm	Water	365	1644.1
		R245a	365	1644.2
		R600	365	1642.9







V. CONCLUSION

Different refrigerants Water, R245A, R600A are analyzed for thermal performance in micro channel heat exchangers using Ansys. Models are done in Creo 2.0.Different models are modeled by varying micro channel heat sink height and compared by analysis. The channel length taken is 10mm. The width is kept constant at 50µm and the height is varied by 100µm and 150µm.

CFD and Thermal analysis are done in Ansys. By observing the CFD analysis results, for all fluids, Nusselt number, Reynolds number are decreasing and heat transfer rate are increasing by increase of height and heat transfer coefficient is same by increase of height. Nusselt number is more when water is used, Reynolds number is more when R245A is used, Heat Transfer Coefficient is more when water is used and Heat Transfer rate is more when R600A is used.

By observing the thermal analysis results, the heat flux is more for Copper than Aluminum. Heat flux is decreasing with increase of height and the value is more when R245A is used since its heat transfer coefficient is more.

REFERENCES

[1] A 3D numerical study of heat transfer in a single-phase micro-channel heat sink using graphene, aluminum and silicon as substrates by Ahmed JassimShkarah, MohdYusoff Bin Sulaiman, MdRazali Bin HjAyob, Hussein Togun, International Communications in Heat and Mass Transfer 48 (2013) 108–115

[2] Fluid flow and heat transfer investigations on enhanced microchannel heat sink using oblique fins with parametric study by Yong Jiun Lee, Pawan K. Singh, PohSeng Lee, International Journal of Heat and Mass Transfer 81 (2015) 325–336

[3] Analysis of three-dimensional heat transfer in micro-channel heat sinks by WeilinQu, IssamMudawar, International Journal of Heat and Mass Transfer 45 (2002) 3973–3985

[4] Experimental and numerical study of pressure drop and heat transfer in a single-phase micro-channel heat sink by WeilinQu, IssamMudawar, International Journal of Heat and Mass Transfer 45 (2002)

[5] Numerical Investigation of Flow Dynamics and Heat Transfer Characteristics in a Microchannel Heat Sink by Md. Emran, Mohammad Ariful Islam, International Conference on Mechanical Engineering, Volume 90, 2014, Pages 563–568

[6] Experimental investigation of heat transfer performance for a novel microchannel heat sink by Y Wang and G-F Ding, Journal of Micromechanics and Microengineering, 2008 J. Micromech. Microeng. 18 035021 doi:10.1088/0960-1317/18/3/035021

[7] Flow Boiling Heat Transfer in Micro-Channel Heat Sinks under Sub-Atmospheric Pressures by X. Hu, G. Lin, X. Bu, Y. Caia& D. Wen, A Journal of Thermal Energy Generation, Transport, Storage, and Conversion, pages 85-113

[8] Theoretical and Experimental Investigation of Heat Transfer Characteristics through a Rectangular Microchannel Heat Sink by Dr. B. S. Gawali , V. B. Swami , S. D. Thakre, International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 3, Issue 8, August 2014