FLUID AND THERMAL ANALYSIS OF LN2 FLOWING THROUGH TUBES OF CRYOSTAT SYSTEM

A Major Project Report

Submitted in partial fulfillment of the requirements for the degree

Of

MASTER OF TECHNOLOGY

IN

MECHANICAL ENGINEERING

(CAD/CAM)

By

Kanheyalal K Mehta
Roll no. 04MME020

Under guidance of

Prof N M Bhatt
Mr. N. Raviprakash

Department of Mechanical Engineering

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY OF SCIENCE & TECHNOLOGY,

AHMEDABAD 382 481

May - 2006

C E R T I F I C A T E

This is to certify that the Major Project Report entitled **"FLUID AND THERMAL ANALYSIS OF LIQUID NITROGEN FLOWING TROUGH TUBES OF CRYOSTAT SYSTEM"** submitted by **Mr. Kanheyalal K. Mehta**, towards the partial fulfillment of the requirements for award of Degree of the Master of Technology in Mechanical engineering, specialization of **(CAD/CAM)** of **Nirma University of Science and Technology** is the record of work carried out by him under our supervision and guidance. The work submitted has in our opinion reached a level required for being accepted for examination.

The results embodied in this major project work to the best of our knowledge have not been submitted to any other University or Institute for award of any degree or diploma.

Project Guides:

Prof N M Bhatt	**Mr. N. Ravi Prakash**	**Pro. Y. C. Saxsena**
Institute of Technology	Engineer - "SD"	Senior professor
Nirma University	Institute for plasma Research	Institute for plasma Research
Ahmedabad	Gandhinagar	Gandhinagar

Prof. A. B. Patel	**Dr. H.V.Trivedi**
Head of Mechanical Engineering	Director
Institute of Technology,	Institute of Technology,
Nirma University	Nirma University
Ahmedabad – 382 481	Ahmedabad – 382 481

Examiners: **i**

ii

iii

iv

ACKNOWLEDGEMENT

I wish to acknowledge from bottom of my heart to following dignitaries, whose continue guidance and support is made, successful completion of this project.

I would like to thank to Prof N M Bhatt, who guided to work on the right track without deviating to the wrong path. Also I thanks to Prof. P. I. Jaggad, who regularly help me during this project work.

I am very much thankful to Mr. N. Ravi Prakash (Coordinator, CAE dept., IPR) for his sincere and well defined guidance, without his guidance and co-operation this dissertation work would not be possible to complete successfully. He guides and helps by his best possible way, considering his most busy schedule. I salute him for his behavior to help peoples around up to last extend. I also have filling of fulfillment to learn Ansys at the Institute.

I also wish to extend my thanks to Prof. D.S. Sharma, (Course coordinator, Institute of Technology, Ahmedabad) for his constant help and guidance on many relevant topics.

Also I thank to Prof. A. B. Patel (Hon. HOD – Mechanical & Additional Director) and Dr. H.V. Trivedi (Hon. Director,) for their support and co-operation, extended during the course.

It is not possible to complete this section without salute and tribute to a man with golden heart, which is the possible reason; I am being here in NIT. He is not other than our favorite late Prof. P. B. Poppat sir.

Kanheyalal K Mehta
Roll No. 04MME020
Institute of Technology,
Nirma University, Ahmedabad

ABSTRACT

The steady state superconducting Tokamak (SST-1) is a medium size tokamak with super conducting magnet. The vacuum system is put for isolating superconducting coils from the ambient temperature and pressure. The vacuum system is called Cryostat. The cryostat is made up of SS 304 L materials in sixteen sectors with four panels of each sector. LN_2 at 80 K is used to remove heat from these panels. Three out of four panels are maintained at 80 K temperatures. However, on inner cylinder panel 80 K temperature is not achieved. Design of system is precise for which 80 K temperature is mandatory at all panels for steady state operation.

This report is prepared for analysis of LN_2 at 80 K flowing through tubes of 8 mm diameter. The panels on which tubes are mounted are trapezoidal, rectangle and cylindrical shapes. Heat fluxes are imparted on these panels. In this analysis heat radiated to each panel is estimated. Mass and velocity of flow at each panel is also estimated.

Pressure drop of LN_2 flow on each panel is estimated. Two phase and single phase flow of LN_2 are considered while estimating pressure drop. Pressure drop for two phase flow is calculated by homogeneous model. Along with two phase and single phase flow different layout are also considered for pressure drop estimation. Different layout which are required to consider are horizontal, vertical and bends.

Finite element analysis is done in Ansys CFD. Fluid flow and thermal analysis is done for this project. Initially analysis is carried out for 2.1 bars pressure and 80 K temperature at inlet of each panel. Analysis is done by varying properties of LN_2 with temperatures. Properties of LN_2 which vary with temperatures are density, viscosity, specific heat and conductivity. Inner cylinder panels are analyzed for different pressure starting from 2.1 bars and 80 K temperature to 4.1 bars pressure and temperatures remain same at inlet.

This report is also covered finite element analysis and it’s general features with capabilities of Ansys.

I N D E X

LIST OF FIGURES

LIST OF TABLES

A_1	-	Area of heat radiating surface (m^2)
A_2	-	Area of heat receiving surface (m^2)
BCP	-	Bottom cryostat plate
C_L	-	Bend loss coefficient
d	-	Hydraulic diameter of fluid flow (m)
d_o	-	Outside diameter of tube (m)
f	-	Coefficient of friction
FP	-	Front cryostat panel
g	-	Gravity (m/s^2)
h	-	Latent heat (kJ/kg)
h_L	-	Head loss due to friction (m)
ICY	-	Inner cryostat cylinder
L	-	Length of the flow (m)
L_B	-	length of bend (m)
m	-	Mass flow rate (kg/s)
N	-	No. of bends
P	-	Perimeter of the tube (m)
P_{bend}	-	Pressure drop in bends (N/m^2)
P_{ele}	-	Pressure drop due to elevation (N/m^2)
P_{entry}	-	Pressure drop in entry (N/m^2)
P_{exit}	-	Pressure drop in exit (N/m^2)
P_{fri}	-	Pressure drop due to friction (N/m^2)
q	-	Heat flux (W/m^2)
Q	-	Heat transfer (W)
R	-	Radius of bend (m)
r	-	Radius of tube (m)
R_d^e	-	Reynolds number based on diameter
t	-	Thickness of the tube (m)
T_1	-	Temperature of the radiative surface (K)
T_2	-	Temperature of the receiving surface (K)

TCP	-	Top cryostat plate
TSA	-	Total surface area (m^2)
V	-	Velocity of fluid (m/s)
x	-	Quality of fluid flow

GREEK LETTER

α	-	Void fraction for elevation (m^2/s)
β	-	Bend angle in degree
θ	-	Angle between x – axis and pipe axis (Degree)
ε	-	Emissivity of the surface
μ	-	Dynamic viscosity of the fluid (Ns/m^2)
ρ	-	Density of the fluid (kg/m^3)
ρ_G	-	Density of the vapor (kg/m^3)
ρ_L	-	Density of the liquid (kg/m^3)
σ	-	Stefan boltzman constant ($W/m^2\ T^4$)

1.1 Introduction

1.1.1 Project work

Plasma operation is being carrying in vacuum which requires less than 4 Kelvin temperature. The vacuum system is put for isolating superconducting coils, which are mounted for plasma operation, from the ambient temperature and pressure. The vacuum system is called Cryostat. The cryostat is made up of SS 304 L materials in sixteen sectors with four panels of each sector. LN_2 at 80 K is used to remove heat from these panels. Three out of four panels are maintained at 80 K temperatures. However, on inner cylinder 80 K temperature is not achieved. Design of system is precise for which 80 K temperature is mandatory at all panels for steady state operation. Since 80 K temperature is required to achieve on inner cylinder panel. Therefore, LN_2 flow through tubes is required to study. These works is allotted for the project work to analysis fluid flow and find the solution of problem.

1.1.2 Introduction to SST -1 Tokamak

SST – 1 is a steady state Tokamak device with superconducting toroidal and poloidal magnetic field coils. These coils are required to maintain at temperature less than 4 Kelvin. The surfaces at this temperature are called cold mass surfaces which are to be protected against the radiated and conducted heat flux from the surfaces at temperature more than 300 K. To reduce the heat load from the hot surfaces (conduction / convection) to the cold surfaces, these cold mass are kept in a vacuum system. To reduce the radiative heat load on these surfaces, a radiation shield at temperature lower than hot surface (80 K) is placed in between the cold and hot surfaces. The vacuum system which is used for this purpose is called **Cryostat**. Along with the radiation shield the evacuated cryostat provides the necessary environment for the operation of superconducting TF and Pf coils. The cryostat is made up SS 304 L materials into sixteen sectors, which are assembled and evacuated to a base pressure of 10^{-5} torr. The thickness of cryostat wall is 10 mm. Since the cryostat will be pumped to high vacuum the difference in the pressure will create large stresses on the cryostat which may produce deflection on the surfaces. In order to avoid this, an additional reinforcement is to be provided on the walls. The liquid nitrogen radiation shield is introduced in the inner cellular space between the cold and hot surfaces.

1.1.3 Basic function of cryostat

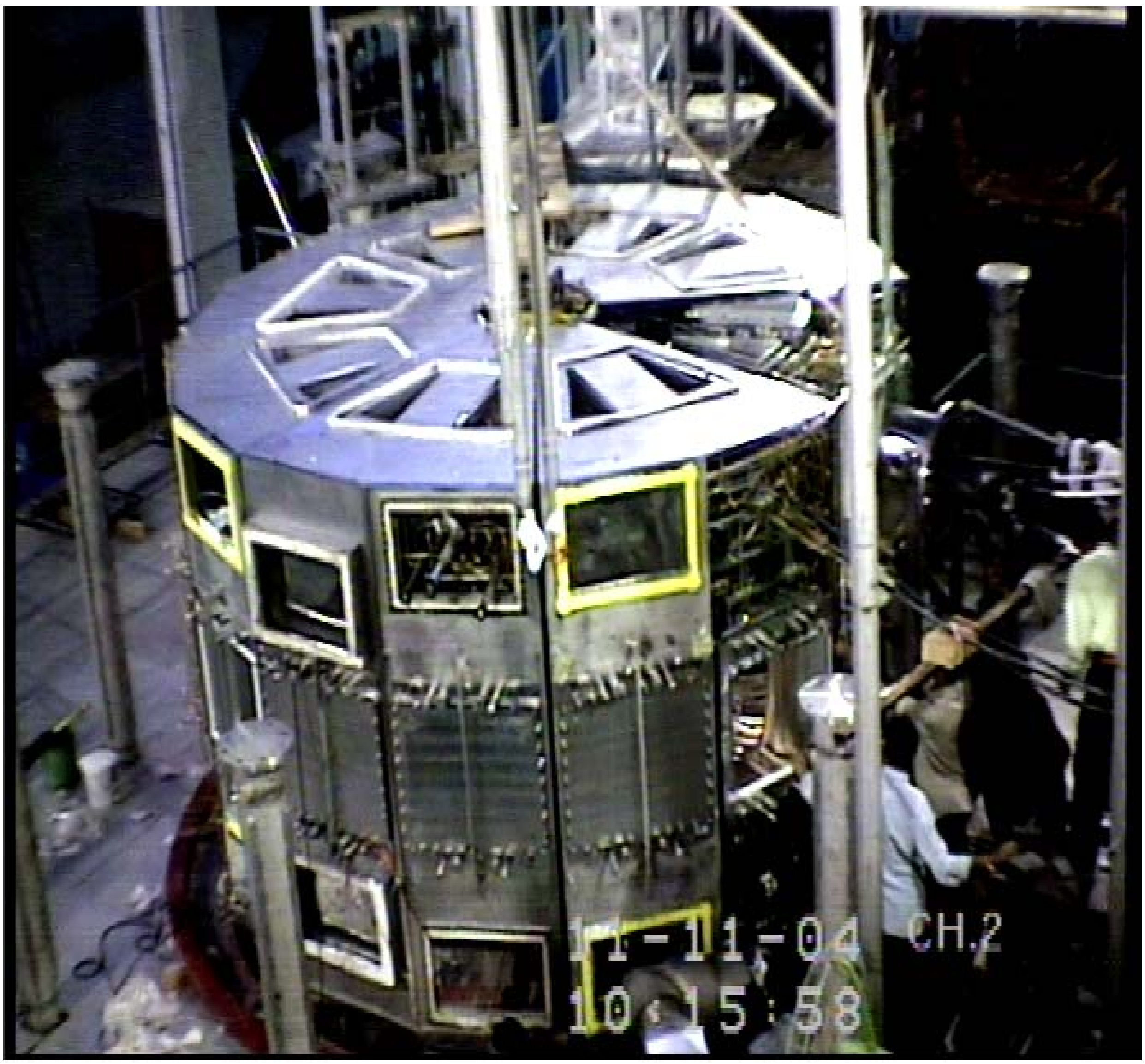
Cryostat which is used as a vacuum system has following function;

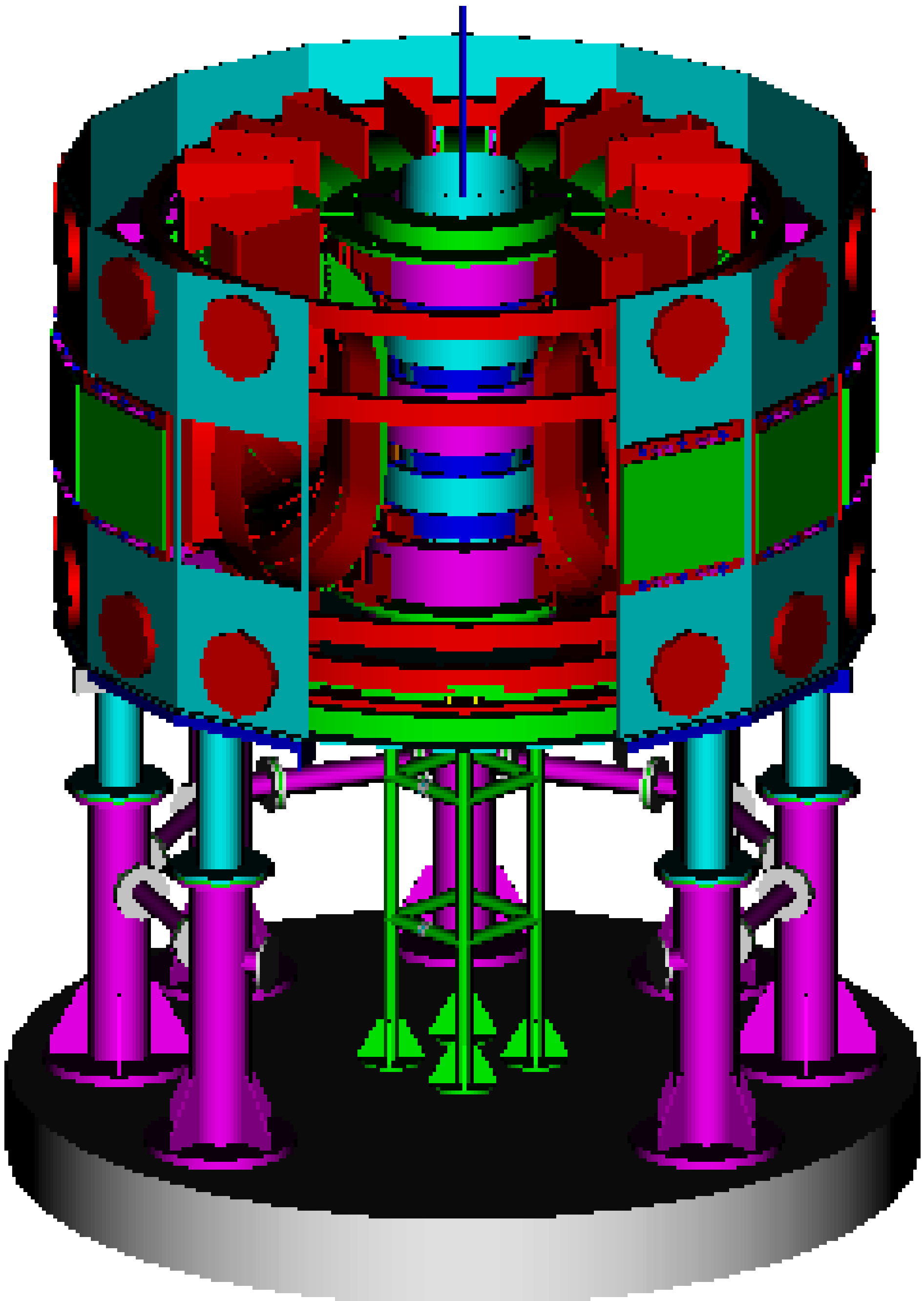
(i) To isolate the superconducting coils from the ambient temperature and pressure.

(ii) To provide rigid support to vessel through the ports extension

(iii) To provide support for the transfer lines of helium, nitrogen, gas feed system and other feed through.

(iv) To transfer the loads of the TF , PF and other components which are supported on it to more rigid and stronger supports.





1.1.4 Components of Cryostat

Cryostat system is made by following components which are provided vacuum. The system is made by sixteen sectors which assembled and evacuated to a base pressure of less than 10^{-5} torr. The leak rate is maintained less than 10^{-5} ml /sec. The designed is such that it is with stand atmospheric load. The structural support provides support to the vacuum vessel, cold mass components and the supporting structures. The cryostat and the vacuum vessel are electrically isolated.

The cryostat system is being cooled by liquid nitrogen having 80 K temperature. Nitrogen is being fed through main feeder from storage tank, located at far way from cryostat. Through feeder than distributes liquid nitrogen to all cooling panels through copper tubes, mounted on these panels.

The panels on which liquid nitrogen is being fed are,

- Top cryostat panel
- Bottom cryostat panel
- Side cryostat panel
- Inner cylinder

1.1.5 Properties of LN_2

LN_2 properties are taken from data book of cryogenic engineering. As temperature varies during flow therefore temperature dependent properties required.[12]

Table 1.1.1 LN_2 properties

Temperature Kelvin	77	78	79	80	81	82	83	84	85	86
Density Kg/m^3	808.5	803.8	799.1	794.4	789.6	784.8	779.9	8.628	8.496	8.371
Specific heat J/kgK	2039	2045	2051	2057	2064	2071	2079	1376	1310	1262
Viscosity (Ns/m^2)*10e-5	15.21	14.65	14.12	13.63	13.15	12.70	12.28	0.5791	0.5858	0.5924
Conductivity (W/mK)*10e-2	13.35	13.21	13.06	12.92	12.77	12.62	12.47	0.849	0.8575	0.8661

1.1.6 Technical details of the system

Technical details of cryostat panels are collected from reports and drawings available as follows.

Table 1.1.2 Technical details

Parts	**Particulars**
Cryostat plates material	SS304L
Tube material	Copper
Thickness	0.001 m
Inner diameter of tube	0.008 m
Feeder diameter	0.0301 m
Recovery	0.0496 m

1.2 Work plan

1.2.1 Introduction

The project work mentioned in this report is focused to study multi-disciplinary subjects. Subjects studied during completion of work are

a. Cryostat system
b. Heat transfer by radiation
c. Fluid mechanics and Thermal engineering
d. Pressure drop in the tubes by two phase flow
e. Convective heat transfer by single and two phase flow
f. Ansys training

1.2.2 Work flow

This report reviews heat load on the cryostat panels, mass of liquid nitrogen required, velocity of fluid, and pressure drop in the panels due to two phase flow. Estimation of these quantities as follows

a. Heat load on each panel is calculated by Stefan Boltzman equation.
b. Heat load on panels use to estimate mass flow of the fluid.
c. Velocity of fluid is estimated by using diameter of the tube.
d. Reynolds numbers is calculated by velocity, density and viscosity of the fluid.
e. Estimation of coefficient of friction is done by Blasius equation.
f. Estimation of pressure drop is done by five components.
    1. Entry losses
    2. Losses in straight tube
    3. Losses in bends
    4. Losses due to change in elevation
    5. Exit losses
g. Estimation of the total pressure drop in each panel by summation of pressure drop components.
h. Benchmark problems are solved by 2-D on Ansys for single phase flow. These problems included analysis of: Pressure, Velocity and Temperature

1.3 Literature Review

1.3.1 Introduction

Literatures are referred for this report has been listed in references. This project is required to refer multi disciplinary subjects for successful completion of assigned task. Subjects which are required to refer like cryogenic, fluid mechanics, thermal engineering, etc. This section of the report is used to describe the literatures are referred. Below, sequentially describe the literature, are referred.

1.3.2 Cryogenic system

This literature is referred to understand cryogenic and its applications [1].It is referred that cryostat is used where desired environment preferred. Cryostat provides atmosphere to the system where experiments is to be required. Author has elaborated details about cryogenic. Major part of this literature is devoted to application and purpose of cryogenic. By using this literature very well understanding is created about cryogenic system. There are many applications are discussed with their advantage and disadvantage compare with other cryogenic. Liquid nitrogen has properties which changes with pressure and temperature. The assigned work has been related to liquid nitrogen therefore this literature help to understand operation of cryostat system and how nitrogen behavior with change of temperature and pressure especially liquid nitrogen. The report is meant for cryostat system therefore the reference is relevant.

1.3.3 Radiation shields for SST-1

The literature has been referred to understand radiation heat transfer in cryostat [10]. The superconducting magnetic filed coils such as Toroidal and poloidal filed coils are maintained at temperatures $\subseteq$ 4.2 K. The plasma chamber is baked at 525 K after commissioning to achieve vacuum $\subseteq 10^{-9}$ torr. The wall temperature of the plasma chamber is maintained at 425 K during plasma operation. The liquid nitrogen shield in installed between the surface at temperatures $\subseteq$4.2 K and the surfaces at the temperatures $\supseteq$300 K to reduce the heat load on the surface at liquid helium temperatures. The cryostat is pumped using turbomolecular pumps for clean vacuum. The pressure $\subseteq 10^{-5}$ torr is maintaining in the cryostat to reduce the heat load due to conduction from hot surfaces to the surfaces at liquid nitrogen and liquid helium temperatures.

The heat load on these cold surfaces due to the conduction of heat from hot surfaces by residual gas in the cryostat (mainly helium and hydrogen) and due to the radiation of heat from hot surfaces should be estimated. The heat load due to residual gas conduction from hot surface to liquid nitrogen surface is negligible compare to the heat load due to radiation from hot surface. The heat load on liquid nitrogen surface due to radiation from hot surface is calculated using following equation.

$$q = \frac{\sigma A_1 (T_2^4 - T_1^4)}{\frac{1}{\varepsilon_1} + (\frac{A_1}{A_2})(\frac{1}{\varepsilon_2} - 1)}$$

Where ε_1 and ε_2 are emissivity of the hot and cold surfaces respectively.

1.3.4 Mass of fluid flow

This literature provides guide lines for estimating LN2 flow in the tubes. The assumption for calculation mass of fluid flow in the tubes is heat transmitted to cryostat panels will be carried by liquid nitrogen. Following relation has been given to find mass flow of fluid [11].

$q = m \times h$ Where m = Mass flow in kg/sec

h = Latent heat of liquid nitrogen = 200 kJ/kg.

q = Heat radiated to the panel, kW

1.3.5 Viscous flow in pipe

The topic describes flow nature of fluid in the tube [3]. Flow nature is important to find pressure drop due to friction in tube. Flow nature is depended on geometry of fluid flow, velocity of fluid, viscosity and density of the fluid. Flow nature is defined by non dimensional Reynolds numbers. Different co-relation has been given in the literature to find coefficient of friction, f. The magnitude of Reynolds number will decide which relation is to be used for finding coefficient of friction. Following co-relation has been given for finding f.

Red < 2500	$f = \frac{64}{R_d^{e(-0.25)}}$
2500< Red <50000	$f = 0.316 * R_d^{e(-0.25)}$
Red > 50000	$f = 0.0054 + (0.396 * R_d^{e(0.3)})$

1.3.6 Velocity of fluid flow

Velocity of fluid is required for estimation of pressure drop in the tubes. The flow of fluid is estimated considering uniform velocity through out tubes. However, it is not true as density will be varying due to heat flux on the tubes which varies velocity of the fluid. Following relation is used for estimation of fluid velocity in each panel [11].

$$m = A\ V\ \rho_L$$

where,

m	=	Mass flow rate, kg/s
d	=	Hydraulic diameter of the fluid $= \dfrac{4A}{P}$
d_o	=	Outer diameter of the tube, m
t	=	thickness of the tube, feeder & recovery, 0.001 m
ρ_L	=	density of the LN_2; Kg/m^3
A	=	Cross section area of the flow $= \dfrac{\pi d^2}{4}$
P	=	Perimeter of the tube $= \dfrac{\pi(d_o + d_i)}{2}$

1.3.7 Pressure drop calculation for two and single phase flow

This problem is defined as a two phase flow. The presence of two-phase flow complicates the problem of predicting the pressure drop of the flowing fluid in several ways. First, the flow pattern is often different for vertical, horizontal and inclined flow. Second, there are several different flow patterns that may exist. Third, the flow may be laminar in the liquid and turbulent in the vapor phase or any of four different combinations may exit.

Finally, the flow pattern changes along the length of the pipe if the quality of the fluid changes because of heat transfer or pressure drop. Two fundamental models have been developed to predict the pressure drop in two-phase flow [6] & [7]:

1. The homogeneous model, in which the two phases are treated as a single phase possessing suitably averaged fluid properties.

2. The separated-flow model, in which the two phases are considered to be artificially segregated into two streams.

Considering simplicity of correlation homogeneous model is used to find pressure drop in the two flows.

1.3.8 Pressure drop at inlet of tube

When there is sudden change in pipe section on flow path, a rapid retardation or acceleration takes place as the kinetic energy is converted into the pressure energy. In our case there is sudden change of section to contraction. The energy loss in a sudden contraction depends on the coefficient of contraction which varies with the area of contraction. Following relation has been given in the literature for estimation of pressure drop due to straight abrupt change [3]

$$P_{ent} = \frac{\rho_L V^2}{2}$$

Where,

ρ_L = density of the LN_2 ; Kg/m^3

V = velocity of the fluid; m/s

1.3.9 Losses in straight tube due to friction

A pressure loss in the tube is dependent on properties of the flowing fluid and nature of contact surface on which fluid flows. When flow is laminar the pressure drop will independent on contact surface because development of the sub-layer on surface. But in case of turbulent flow pressure drop dependent on surface of contact as absence of sub-layer.

Homogeneous model has been used to calculate pressure drop. Therefore, mean properties of the fluid has been taken in the calculation. Following Hagen-Poiseuille´s equation has been given in the literature for estimating pressure drop in straight tube due to friction [11]

$$h_L = \frac{fLV^2}{2gd}$$

Where,

f	=	coefficient of friction
L	=	Length of the tube (m)
g	=	Acceleration into gravity (m/s^2)

1.3.10 Losses in bends due to friction

Pressure drop in bends is due to friction and the change of the direction of flow. Pressure drop in the bends is estimated by following correlation which has been given in the literature [11].

$$P_{bend} = \left(\frac{C_L V^2 \rho}{2} + \frac{8 f m^2 L_B}{\pi^2 d^5 \rho} \right) * N$$

Where,

C_L	=	bend loss coefficient,
	=	$0.13 + 1.85 \left(\frac{r}{R} \right)^{0.35} * \left(\frac{\beta}{180} \right)^{0.5}$
β	=	Bend angle in degree
L_B	=	Bend length (m)
R	=	Bend radius (m)
r	=	Radius of tube (m)
N	=	No. of bends

1.3.11 Losses due to change in elevation

Pressure drop due to change in elevation is estimated differently for single and two phase flow. In two phase flow following correlation has been given in the literature considering homogeneous flow [7].

$$p_{ele} = (\alpha * \rho_G + (1-\alpha)\rho_L)g \sin\theta$$

Where,

α	=	Void fraction, $\dfrac{1}{\left(1+\left(\dfrac{1-x}{x}\right)\left(\dfrac{\rho_G}{\rho_L}\right)\right)}$
x	=	quality of the fluid
ρ_G	=	density of the vapor fluid (Kg/m^3)
ρ_L	=	density of the liquid fluid (Kg/m^3)
g	=	gravity (m/s^2)

1.3.12 Exit losses

When there is sudden change in pipe section on flow path, a rapid retardation or acceleration takes place as the kinetic energy is converted into the pressure energy. In our case there is sudden change of section to expansion. The energy loss in a sudden expansion depends on the coefficient of expansion which varies with the area of expansion. In the referred literatures following correlation is used to estimate pressure drop in the tube at exit [3].

$$P_{exit} = \frac{V^2 \rho}{4}$$

Where,

ρ	=	density of the LN_2 ; Kg/m^3
V	=	velocity of the fluid; m/s

CHAPTER 2

Finite element analysis and Ansys

2.1 Finite element analysis

2.1.1 Introduction

The basic idea in the finite element method is to find the solution of complicated problem by replacing it by a simpler one. Since actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution. The existing mathematical tools will not be sufficient to find the exact solution and some times even an approximate solution of the most of the practical problem. Thus in the absence of any other convenient method to find even the approximate solution of given problem, we have to prefer to finite element method. Moreover, in the finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort.

Finite element method has been developed to high level of refinement in structural mechanics. Such methods are also applicable to steady and transient heat transfer problems. Finite element method provides piecewise, or regional, approximation to partial differential equation. Finite difference method provide are relatively easy to implement, except when irregular geometry's or unusual boundary conditions are present Under such condition, it may be desirable to use more general approach, such as a finite element method, even at the expense of programming complexity.

A finite element method is a mathematical procedure for satisfying the partial differential in an average sense over a finite element. Various methods exist. All of them require that an irregular integral representation of a partial differential equation be construction. The approaches include variational calculus, method of weighted residual, and moment of energy balance. Classical finite element methods for structural mechanics are based on variational principal. Variational principal is also applicable to steady state diffusion and conduction processes. However, for transient diffusion and conduction processes and for convective heat transfer processes, it is necessary to use more general procedure, such as the weighted residual.

2.1.2 General description of the finite element method

In the finite element method, the actual continuum or body of matter like solid, liquid or gas is represented as an assemblage of subdivisions called finite elements. The selected elements are considered to be interconnected at specified joints called nodes or nodal points.

The nodes usually lie on the element boundaries where adjacent elements are considered to be connected. Since the actual variation of the field variable displacement, temperature, velocity, inside the continuum is not known, we assume that the field variable inside a finite element can be approximated by the simple function. These approximating functions are defined in term of the value of the field variable at nodes. When field equations for the whole continuum are written, the new unknown will be nodal values of the field variable. By solving the field equations, which are generally in the form of the matrix equation, the nodal values of the field variable will be known. Once these are known the approximating functions are defined the field variable throughout the assemblage of element.

2.1.3 Discretization of the solution region

The first step in the finite element is to divide the solution region into subdivisions or elements. Hence the region is to be modeled with suitable finite element. The number, type, size and arrangement of the element are to be decided.

2.1.4 Selection of the proper interpolation function

Since the solution of the region for the field variable for the complex shape of the region can not predicted exactly, we assume some suitable solution within an element to approximate the unknown solution. The assumed solution must be simple form a computational point of view, but it should satisfy certain convergence requirement. In general, the solution or the interpolation model is taken in the form of a polynomial.

2.1.5 Derivation of the element matrix and load vector

From the assumed proper interpolation function the element matrix [K] and the load vector P of the element "e" are to be derived by using either equilibrium conditions or a suitable variational principal.

2.1.6 Assemblage of element equation to obtain the overall equilibrium equation

Since the region is composed of several finite elements, the individual element matrices and load vector are to be assembled in a suitable manner and the overall equilibrium equation has to be formulated as

$[K]\,\Phi = P$ Where, K = Element matrix

Φ = Vector of primary field variable at nodes values

P = vector load

2.1.7 Solution for the unknown nodal values

The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. After the interpolation of boundary condition, the equilibrium equation can be expressed as
$[K]\ \Phi = P$

2.1.8 Computation of element results

For the unknown nodal values of the primary field variable derived properties or variable are computed using necessary equation. The nodal results are likely to be displacement in structural problem, temperature in case of thermal problem, pressure and velocity in the fluid problem. The results or derived quantities are stresses, energy strain in structural, heat flux, heat flow, temperature gradient, film coefficient, etc. in thermal and stream function, floe rate, wall shear stress, pressure coefficient in the fluid problem.

2.1.9 Discretization process

In most of the engineering problems, we need to find the values of a field variable such as displacement, stress, temperature, pressure, velocity as a function of the spatial co-ordinate system. In the case of the transient or unsteady state problem, the field variable has to be found as a function of not only the spatial co-ordinate (x ,y ,z) but also time (t). The geometry of the solution region is often the irregular. The first step in finite element analysis is to discretization of the irregular shape into smaller and regular regions or elements.
The various considerations have been taken in the discretization process are given below.

2.1.10 Types of the elements

Often the type of element to be used will be evident from the physical problem itself. Here the number of the degree of freedom needed, the expected accuracy, the ease with which the necessary equation can be derived and the degree to which the physical structure can be modeled without approximation will dictate the choice of element type to be used for the idealization. In certain problems, the given body can not be represented as an assemblage of only one type of element. In such cases, we may have to use two or more type of the elements.

2.1.11 Size of the elements.

The size of the elements influences the convergence of the solution directly and hence it has to be chosen with care. If the size of the elements is small, the final solution is expected to be more accurate. However, we have to remember that the use of the element of smaller size will also mean more computational time. Sometimes, we may have to use element of different size in same body. In general, whenever steep gradient of the field variable are expected, we have to use a finer mesh in those regions. Another characteristic related to the size of the element which affects the finite element solution is the aspect ratio of the elements. Aspect ratio of the nearly of unity generally yield good results.

2.1.12 Location of the node

If the body has no abrupt changes in the geometry, material properties and external conditions like load, temperatures etc. body can be divided into equal subdivision and hence the spacing of the nodes can be uniform. On other hand, if there are any discontinuities in the problem, nodes have to be introduced obviously at the discontinuities.

2.1.13 Number of the elements

The number of element to be chosen for the idealization is related to the accuracy desired, size of the elements, and the number of degree of freedom involved. Although an increase in the number of elements generally means more accurate results, for any given problem, there will be a certain number of elements beyond which the accuracy can not be improved by any significant amount.

2.1.14 Simplification afforded by the physical configuration of the body

If the configuration of the body and loading conditions are symmetric, we may consider only sector of the body for finite element analysis. The symmetry conditions however have to be incorporated in the solution procedures.

2.1.15 Finite representation of the physical configuration of the body

In most of the problem, like in the case of the analysis of beam, plates and shells, the boundaries of the body or continuum are clearly defined. Hence entire body can be considered for the element idealization. However, in some cases, like in the case of analysis of the dams, foundations and infinite bodies, the boundaries are not clearly defined. Once the significant extent of the infinite body is identified, the boundary conditions for these semi infinite bodies have to be incorporated in the solution.

2.2 Ansys

2.2.1 Introduction

ANSYS finite element analysis software enables to perform the following tasks:

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both.

2.2.2 General information

2.2.2.1 Interactive Mode

In interactive mode, you work with menus and dialog boxes to drive the ANSYS program. You have easy access to ANSYS graphics capabilities, online help, and other tools, such as wizards.

The standard ANSYS GUI is the default. This layout shows the Utility Menu, Standard Toolbar, Input Window, ANSYS Toolbar, Main Menu, Graphics Window, Status Area, and Output Window. You can resize the toolbars, the overall size of the GUI, the font, and the color

You can also run ANSYS through the ANSYS Workbench Products. The ANSYS Workbench provides a framework for integrating the various ANSYS computer-aided engineering tools into a single working environment, combining the user interface strengths of the Design Space product with the solution capabilities of ANSYS.

2.2.2.2 Choosing an ANSYS Product

ANSYS builds a variety of products. Your site may have licenses for one, several, or all ANSYS products and product combinations. By invoking ANSYS with the appropriate product variable, you can run a more efficient ANSYS session. You can specify the appropriate product via the launcher or via command line using the -p option. The default product (if you are licensed for it) is ANSYS Mechanical.

2.2.2.3 Getting Started with ANSYS

The ANSYS program has many finite-element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guides in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

2.2.2.4 Building the Model

Building a finite element model requires more of your time than any other part of the analysis. First, you specify a jobname and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

Specifying a Jobname and Analysis Title

This task is not required for an analysis, but is *recommended*.

2.2.2.5 *Defining the Jobname*

The *jobname* is a name that identifies the ANSYS job. When you define a jobname for an analysis, the jobname becomes the first part of the name of all files the analysis creates. (The extension or suffix for these files' names is a file identifier such as .DB.) By using a jobname for each analysis, you ensure that no files are overwritten.

If you do not specify a jobname, all files receive the name FILE or file, depending on the operating system.

2.2.2.6 Defining an Analysis Title

The /TITLE command (Utility Menu> File> Change Title), defines a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output. You can issue the /STITLE command to add subtitles; these will appear in the output, but not in graphics displays.

2.2.2.7 Defining Units

The ANSYS program does not assume a system of units for your analysis. Except in magnetic field analyses, you can use any system of units so long as you make sure that you use that system for all the data you enter. (Units must be consistent for all input data.)

2.2.3 Elements

2.2.3.1 Element Name

An element type is identified by a name (8 characters maximum), such as BEAM3, consisting of a group label (BEAM) and a unique, identifying number (3). The element is selected from the library for use in the analysis by inputting its name on the element type command [**ET**].

2.2.3.2 Nodes

The nodes associated with the element are listed as I, J, K, etc. Elements are connected to the nodes in the sequence and orientation shown on the input figure for each element type. This connectivity can be defined by automatic meshing, or may be input directly by the user with the **E** command. The node numbers must correspond to the order indicated in the "Nodes" list. I node is the first node of the element. The node order determines the element coordinate system orientation for some element types.

2.2.3.3 Degrees of Freedom

Each element type has a degree of freedom set, which constitute the primary nodal unknowns to be determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltages, etc. Derived results, such as stresses, heat flows, etc., are computed from these degree of freedom results. Degrees of freedom are not defined on the nodes explicitly by the user, but rather are implied by the element types attached to them. The choice of element types is therefore, an important one in any ANSYS analysis.

2.2.3.4 Real Constants

Data which are required for the calculation of the element matrix, but which cannot be determined from the node locations or material properties, are input as "real constants." Typical real constants include area, thickness, inner diameter, outer diameter, etc. A basic description of the real constants is given with each element type. The real constants are input with the **R** command. The real constant values input on the command must correspond to the order indicated in the "Real Constants" list.

2.2.3.5 Material Properties

Various material properties are used for each element type. Typical material properties include Young's modulus (of elasticity), density, coefficient of thermal expansion, thermal conductivity, etc. Each property is referenced by an ANSYS label - EX, EY, and EZ for the directional components of Young's modulus, DENS for density, and so on. All material properties can be input as functions of temperature.

Some properties for non-thermal analyses are called *linear* properties because typical solutions with these properties require only a single iteration. Properties such as stress-strain data are called *nonlinear* because an analysis with these properties requires an iterative solution. Linear material properties are input with the **MP** family of commands while nonlinear properties are input with the **TB** family of commands. Some elements require other special data which need to be input in tabular form.

2.2.3.6 Surface Loads

Various element types allow surface loads. Surface loads are typically pressures for structural element types, convections or heat fluxes for thermal element types, etc.

2.2.3.7 Body Loads

Various element types allow body loads. Body loads are typically temperatures for structural element types, heat generation rates for thermal element types, etc. Body loads are designated in the "Input Summary" table of each element by a label and a list of load values at various locations within the element. For example, for element type PLANE42, the body load list of "Temperatures: T(I), T(J), T(K), T(L)" indicates that temperature body loads are allowed at the I, J, K, and L node locations of the element. Body loads are input with the **BF** or **BFE** commands. The load values input on the **BFE** command must correspond to the order indicated in the "Body Load" list.

2.2.3.8 Special Features

The keywords in the "Special Features" list indicate that certain additional capabilities are available for the element. Most often these features make the element nonlinear and require that an iterative solution be done. For a description of the special feature "Plasticity”.

2.2.3.9 KEYOPTS

KEYOPTS (or key options) are switches, used to turn various element options on or off. KEYOPT options include stiffness formulation choices, printout controls, element coordinate system choices, etc. A basic description of the KEYOPTS is given with each element type. Values for the first six KEYOPTS (KEYOPT(1) through KEYOPT(6)) may be input with the ET or KEYOPT commands. Values for KEYOPT(7) or greater on any element are input with the KEYOPT command.

List of Elements by Classification

Beam Element	**CIRCU Element**	**COMBINE Element**	**CONTACT Element**
FLUID	FOLLOW	HF	INFIN
INTER	LINK	MASS	MATRIX
MESH	MPC	PIPE	PLANE
PRETS	ROM	SHELL	SOLID
SOLSH	SOURCE	SURF	TARGE
TRANS	VISCO		

2.3 Element FLUID141

2.3.1 Element Description

You can use FLUID141 to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. Use this FLOTRAN CFD element to solve for flow and temperature distributions within a region, as opposed to elements that model a network of one-dimensional regions hooked together (such as FLUID116). You can also use FLUID141 in a fluid-solid interaction analysis.

For the FLOTRAN CFD elements, the velocities are obtained from the conservation of momentum principle, and the pressure is obtained from the conservation of mass principle. (The temperature, if required, is obtained from the law of conservation of energy.) A segregated sequential solver algorithm is used; that is, the matrix system derived from the finite element discretization of the governing equation for each degree of freedom is solved separately. The flow problem is nonlinear and the governing equations are coupled together. The sequential solution of all the governing equations, combined with the update of any temperature- or pressure-dependent properties, constitutes a *global iteration*. The number of global iterations required to achieve a converged solution may vary considerably, depending on the size and stability of the problem. Transport equations are solved for the mass fractions of up to six species.

You may solve the system of equations in a constant angular velocity rotating coordinate system. The degrees of freedom are velocities, pressure, and temperature. Two turbulence quantities, the turbulent kinetic energy and the turbulent kinetic energy dissipation rate, are calculated if you invoke an optional *turbulence* model. For axisymmetric models, you can calculate an optional *swirl* - velocity VZ normal to the plane. You also can specify swirl at the inlet or a boundary (moving wall).

2.3.2 FLUID141 Input Data

The element is defined by three nodes (triangle) or four nodes (quadrilateral) and by isotropic material properties. The coordinate system is selected according to the value of KEYOPT(3), and may be either Cartesian, axisymmetric, or polar.

Node and Element Loads describes element loads. For a fluid-solid interaction analysis, you can apply a fluid-solid interaction flag using the SF family of commands (SF, SFA, SFE, or SFL) and the FSIN surface load label. You must also apply the same interface number to the solid interface where load transfer takes place. FLUID141 Fluid Elements

If the material number [MAT] of a FLUID141 element is 1, it is assumed to be a fluid element. Its properties density, viscosity, thermal conductivity and specific heat - are defined with a series of FLDATA commands. You can analyze only one fluid, and it must be in a single phase. Thermal conductivity and specific heat are relevant (and necessary) only if the problem is thermal in nature. The properties can be a function of temperature through relationships specified by the FLDATA7,PROT command or through a property database. In addition, the density may vary with pressure (per the ideal gas law) if the fluid is specified to be air or a gas.

Six turbulence models are available. You can activate turbulence modeling with the FLDATA1,SOLU,TURB,T command. The Standard k-ε Model and the Zero Equation Turbulence Model are available along with four extensions of the Standard k-ε Model. KEYOPT(1) activates multiple species transport, which allows you to track the transport of up to six different fluids (species) in the main fluid. KEYOPT(4) allows you to use displacement DOFs to specify motion of boundaries when using the Arbitrary Lagrangian-Eulerian (ALE) formulation.

2.3.3 FLUID141 Distributed Resistance

A *distributed resistance* provides a convenient way to approximate the effect of porous media (such as a filter) or other such flow domain features without actually modeling the geometry of those features. It is an artificially imposed, unrecoverable loss associated with geometry not explicitly modeled. Any fluid element with a distributed resistance will have a real constant set number [REAL] greater than 1 assigned to it.

The resistance to flow, modeled as a distributed resistance, may be due to one or a combination of these factors: a localized head loss (K), a friction factor (f), or a permeability (C). The total pressure gradient is the sum of these three terms, as shown below for the X direction.

If large gradients exist in the velocity field within a distributed resistance region, you should deactivate the turbulence model by setting ENKE to 0 and ENDS to 1.0 in this region.

Non-Newtonian viscosity models also are available for this element. Currently, ANSYS provides a Power Law model, a Bingham model, and a Carreau model.

2.3.4 FLUID141 Fan Model

The *fan model* provides a convenient way to approximate the effect of a fan or pump in the flow domain. It is an artificially imposed momentum source that provides momentum source terms associated with a fan or a pump not explicitly modeled.

The pressure rise associated with a fan model is given by the pressure gradient times the flow length through the elements with the fan model real constants. For a one-directional fan model, (real constant TYPE = 4), three coefficients are input. The pressure gradient can be treated as a quadratic function of velocity, as shown below for the X direction.

V is the fluid velocity and C_1, C_2, and C_3 are the coefficients specified as real constants. For an arbitrary direction fan model (real constant TYPE = 5), the three coefficients are the components of the actual coefficients along a coordinate direction.

2.3.5 FLUID141 Non-Fluid Elements

If the material number [MAT] of the element is greater than 1, it is assumed to be a non-fluid element. Only the energy equation is solved in the non-fluid elements. You can define up to 100 different non-fluid materials. To specify density, specific heat, and thermal conductivity for the non-fluid elements, use the MP command. Temperature variation of the non-fluid properties is permitted, and you specify it via the MP or MPDATA commands. Orthotropic variation also is permitted, with the restriction that the spatial variation is always with respect to the global coordinate system. Note that element real constants have no meaning for non-fluid FLUID141 elements.

2.3.6 FLUID141 Input Summary

Nodes	:	I, J, K, L
Degrees of Freedom	:	VX, VY, VZ, PRES, TEMP, ENKE, ENDS
Material Properties	:	Non-fluid: KXX, KYY, C, DENS
Fluid	:	Density, viscosity, thermal conductivity, specific heat (use FLDATA commands) or MPTEMP and MPDATA.

Surface Loads	:	HFLUX, CONV, RAD, RDSF, FSIN
Body Loads	:	HGEN, FORC
Special Features	:	Nonlinear

Six turbulence models

Incompressible or compressible algorithm

Transient or steady state algorithm

Rotating or stationary coordinate system

Algebraic solvers particular to FLOTRAN

Optional distributed resistance and fan models

Multiple species transport

2.4 Analyzing Thermal Phenomena

2.4.1 Introduction

A *thermal analysis* calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate *thermal stresses* (that is, stresses caused by thermal expansions or contractions).

2.4.2 Types of Thermal Analysis

ANSYS supports two types of thermal analysis:

1. A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.

2. A transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

2.4.3 Meshing Requirements

The ANSYS program has no formal criteria for evaluating the finite element mesh. However, thermal gradients are often extremely high near thermal boundaries, especially heat flux boundaries. Therefore, the mesh should usually be denser near thermal boundaries.

2.5 FLOTRAN CFD Analyses

2.5.1 Introduction

The ANSYS FLOTRAN derived product and the FLOTRAN CFD (Computational Fluid Dynamics) option to the other ANSYS products offer you comprehensive tools for analyzing 2-D and 3-D fluid flow fields. Using either product and the FLOTRAN CFD elements FLUID141 and FLUID142, you can achieve solutions for the following:

- Lift and drag on an airfoil
- The flow in supersonic nozzles
- Complex, 3-D flow patterns in a pipe bend

In addition, you can use the features of ANSYS and ANSYS FLOTRAN to perform tasks including:

Calculating the gas pressure and temperature distributions in an engine exhaust manifold

- Studying the thermal stratification and breakup in piping systems
- Using flow mixing studies to evaluate potential for thermal shock
- Doing natural convection analyses to evaluate the thermal performance of chips in electronic enclosures
- Conducting heat exchanger studies involving different fluids separated by solid regions

2.5.2 Types of FLOTRAN Analyses

You can perform these types of FLOTRAN analyses:

- Laminar or turbulent
- Thermal or adiabatic
- Free surface
- Compressible or incompressible
- Newtonian or Non-Newtonian
- Multiple species transport

These types of analyses are *not* mutually exclusive. For example, a laminar analysis can be thermal or adiabatic. A turbulent analysis can be compressible or incompressible.

2.5.3 Laminar Flow Analysis

In these analyses, the velocity field is very ordered and smooth, as it is in highly viscous, slow-moving flows. The flow of some oils also can be laminar.

2.5.4 Turbulent Flow Analysis

Turbulent flow analyses deal with problems where velocities are high enough and the viscosity is low enough to cause turbulent fluctuations. The two-equation turbulence model in ANSYS enables you to account for the effect of the turbulent velocity fluctuations on the mean flow.

Laminar and turbulent flows are considered to be incompressible if density is constant or if the fluid expends little energy in compressing the flow. The temperature equation for incompressible flow neglects kinetic energy changes and viscous dissipation.

2.5.5 Thermal Analysis

Often, the solution for the temperature distribution throughout the flow field is of interest. If fluid properties do not vary with temperature, you can converge the flow field without solving the temperature equation. In a conjugate heat transfer problem, the temperature equation is solved in a domain with both fluid and non-fluid (that is, solid material) regions. In a natural convection problem, the flow results mainly or solely from density gradients brought about by temperature variations. Most natural convection problems, unlike forced convection problems, have no externally applied flow sources.

2.5.6 Compressible Flow Analysis

For high velocity gas flows, changes in density due to strong pressure gradients significantly influence the nature of the flow field. ANSYS uses a different solution algorithm for compressible flow.

2.5.7 Non-Newtonian Fluid Flow Analysis

A linear relationship between the stress and rate-of-strain cannot describe many fluid flows adequately. For such non-Newtonian flows, the ANSYS program provides three viscosity models and a user-programmable subroutine.

2.5.8 Multiple Species Transport Analysis

This type of analysis is useful in studying the dispersion of dilute contaminants or pollutants in the bulk fluid flow. In addition, you can use multiple species transport analysis for heat exchanger studies where two or more fluids (separated by walls) may be involved.

2.5.9 Free Surface Analysis

Free surface analyses deal with problems involving a unconstrained gas-liquid surface. You can use this type of analysis to solve two dimensional planar and axisymmetric problems such as flow over a dam and tank sloshing.

2.5.10 Considerations and Restrictions for Using the FLOTRAN Elements

The FLOTRAN elements have some limitations:

- You cannot change the problem domain during a single analysis.
- Certain features of the ANSYS program do not work with the FLOTRAN elements.
- You cannot use certain commands or menu paths with the FLOTRAN elements.
- If you use the ANSYS GUI, only the features and options called for in the FLOTRAN SetUp portion of the menus and dialog boxes will appear.

2.6 FLOTRAN Analysis procedure

2.6.1 Determining the Problem Domain

You need to determine the proper domain for each problem you analyze. Locate the boundaries of the problem where conditions are known. If you do not know precise conditions and must make assumptions about them, do not locate boundaries too close to the regions of greatest interest or near regions that have steep gradients in the solution variables.

Sometimes, you may not realize that steep gradients occur too near the outlet or in some other region until you see the analysis results. Should this happen, you can re-analyze the problem with a different problem domain.

For specific recommendations on determining problem domain, see the sections discussing the various flow phenomena.

2.6.2 Determining the Flow Regime

You need to estimate the character of the flow. The character is a function of the fluid properties, geometry, and the approximate magnitude of the velocity field.

Fluid flow problems that FLOTRAN solves will include gases and liquids, the properties of which can vary significantly with temperature. The flow of gases is restricted to ideal gases. You must determine whether the effect of temperature on fluid density, viscosity, and thermal conductivity is important. In many cases, you can get adequate results with constant properties.

To assess whether you need the FLOTRAN turbulence model, use an estimate of the Reynolds number, which measures the relative strengths of the inertial and viscous forces.

To determine whether you need to use the compressible option, estimate the Mach number. The Mach number at any point in the flow field is the ratio of the fluid speed and the speed of sound. At Mach numbers above approximately 0.3, consider using the compressible solution algorithm. At Mach numbers above approximately 0.7, you can expect significant differences between incompressible and compressible results. You may want to compare results from each algorithm for a representative problem.

2.6.3 Creating the Finite Element Mesh

You will need to make assumptions about where the gradients are expected to be the highest, and you must adjust the mesh accordingly. For example, if you are using the turbulence model, then the region near the walls must have a much denser mesh than would be needed for a laminar problem. If it is too coarse, the original mesh may not capture significant effects brought about through steep gradients in the solution. Conversely, elements may have very large aspect ratios with the long sides along directions with very low gradients.

For the most accurate results, use mapped meshing. It more effectively maintains a consistent mesh pattern along the boundary. You can do this by issuing the command MSHKEY,1 (Main Menu> Preprocessor> Meshing> Mesh> *entity*> Mapped).

In some cases, you may wish to use hexahedral elements to capture detail in high-gradient regions and tetrahedral elements in less critical regions. For flow analysis, especially turbulent, you should not use pyramid elements in the region near the walls because it may lead to inaccuracies in the solution.

Wedge elements can be useful when a complex area can be easily meshed with triangles that are then extruded. For a quick solution, you can use wedge elements in the region near the walls. However, for accurate results, you should use hexahedral elements in those regions.

Wedge elements are considered to be degenerate hexahedral elements. When using the ANSYS Mesh Tool (Main Menu> Preprocessor> Meshing> MeshTool) to sweep triangles into wedges, you must select Hex elements.

2.6.4 Applying Boundary Conditions

You can apply boundary conditions before or after you mesh the domain. Consider every model boundary. If a condition is not specified for a dependent variable, a zero gradient of that value normal to the surface is assumed.

You can change boundary conditions between restarts. If you need to change a boundary condition or accidentally omit it, you do not need to restart your analysis unless the change causes instabilities in the analysis solution.

2.6.5 Setting FLOTRAN Analysis Parameters

In order to use options such as the turbulence model or solution of the temperature equation, you must activate them. Specific items to be set, such as fluid properties, are a function of the type of flow problem at hand. Other sections in this document recommend parameter settings for various types of flow.

2.6.6 Solving the Problem

You can monitor solution convergence and stability of the analysis by observing the rate of change of the solution and the behavior of relevant dependent variables. These variables include velocity, pressure, temperature, and (if necessary) turbulence quantities such as kinetic energy (degree of freedom ENKE), kinetic energy dissipation rate (ENDS), and effective viscosity (EVIS).

2.6.7 Convergence and Stability Tools

The ANSYS program offers several tools to help with convergence and solution stability.

2.6.7.1 Relaxation Factors

The relaxation factor is the fraction of the change between the old solution and the newly calculated solution that is added to the old solution, giving the results for the new global iteration. The relaxation factors for every component must be between 0.0 (resulting in no update to the degree of freedom or property) and 1.0 inclusive.

2.6.7.2 Inertial Relaxation

Inertial relaxation of the equation set for a DOF provides diagonal dominance to make a solution stable. Hypothetically, when a solution is converged in the absence of round off-error, the inertial relaxation applied does not affect the value of the answer. However, in real situations, some round off-error always occurs, so the inertial relaxation may affect your solution.

You can apply inertial relaxation to the momentum equations (MOME), turbulence equations (TURB), the pressure equation (PRES), and the temperature equation (TEMP). To do so, use either of these methods:

The inertial relaxation factor is in the denominator of the term added, so smaller values have a greater effect. Typical useful values range between 1.0 (mild) and 1.0 x 10^{-7} (severe).

2.6.7.3 Modified Inertial Relaxation

Modified inertial relaxation adds a local positive value to the diagonal term to guarantee a positive diagonal. You can apply modified inertial relaxation to the momentum equations (MOME), turbulence equations (TURB), and the temperature equation (TEMP). To do so, use either of these methods:

A larger modified inertial relaxation factor gives a more robust scheme, but it may yield a slower convergence. The recommended range is 0.1 to 1.0.

You should consider inertial relaxation and modified inertial relaxation to be mutually exclusive.

2.6.7.4 Artificial Viscosity

Artificial viscosity smooths the velocity solution in regions of steep gradients. It has proven useful in aiding convergence of compressible problems and in smoothing velocity solutions in incompressible problems with distributed resistances. For incompressible analyses, you should keep the artificial viscosity within an order of magnitude of the effective viscosity.

2.6.7.5 DOF Capping

DOF capping allows you to prevent variables from going out of boundaries you specify. You can limit the velocities, pressure, and temperature degrees of freedom (VY, VY, VZ, PRES, TEMP). To do this, use either of these methods:

Velocity capping eliminates the effects of velocity spikes on properties, which may occur in the early stages of convergence. Capping is especially suited to compressible analyses, where velocity spikes can cause kinetic energy terms great enough to produce negative static temperatures. When a degree of freedom is capped, ANSYS prints a message along with the convergence monitor printout.

The pressure value calculated by the solution of the pressure equation is capped, not the relaxed value. Therefore, if you introduce pressure capping upon restarting an analysis, pressure values may still be outside the caps. Capping applies to relative values of pressure and absolute values of temperature. You should cap the total temperature when performing compressible thermal analyses. It will help ensure negative properties do not enter the calculations..

2.6.7.6. The Quadrature Order

You have control over the quadrature order. In axisymmetric problems, the quadrature order automatically is set to 2 upon solution. This is because quadrature orders of 2 produce more accurate results for problems with irregularly shaped elements. The quadrature order is also automatically set to 2 upon solution for 3-D problems using cylindrical coordinates.

If you notice anomalous results near a region of skewed hexahedral elements, reset the quadrature order to 2. There might be a problem with the results when the included angles of the hexahedra exceed 120 degrees.

2.6.7.7 Evaluating a FLOTRAN Analysis

The two basic questions an analyst must answer are:

- When is the analysis completed?
- Has it been done correctly?

These questions are interrelated, since convergence may not be achieved it you have not set up and executed the analysis correctly.

If you have set the input parameters and boundary conditions correctly, the analysis is complete when the convergence monitors for all variables stop decreasing and the average, maximum, and minimum values of the solution variables no longer increase or decrease. There is no guarantee, however, that you will achieve a single exact answer because nature does not guarantee that a single exact answer exists. Oscillatory problems (for example, vortex shedding behind a cylinder) may not yield stationary results from a steady-state or a transient solution algorithm. You may wish to continue executing the analysis to verify whether a solution has a stable or a fluctuating nature.

2.7 Coupled-Field Analyses

2.7.1 Introduction

A coupled-field analysis is a combination of analyses from different engineering disciplines (physics fields) that interact to solve a global engineering problem, hence, we often refer to a coupled-field analysis as a multiphysics analysis. When the input of one field analysis depends on the results from another analysis, the analyses are coupled.

Some analyses can have one-way coupling. For example, in a thermal stress problem, the temperature field introduces thermal strains in the structural field, but the structural strains generally do not affect the temperature distribution. Thus, there is no need to iterate between the two field solutions. More complicated cases involve two-way coupling. A piezoelectric analysis, for example, handles the interaction between the structural and electric fields: it solves for the voltage distribution due to applied displacements, or vice versa. In a fluid-structure interaction problem, the fluid pressure causes the structure to deform, which in turn causes the fluid solution to change. This problem requires iterations between the two physics fields for convergence.

The coupling between the fields can be accomplished by either direct coupling (matrix coupling) or sequential coupling (load vector coupling) . Load transfer can take place across surfaces or volumes. Coupling across fields can be complicated because different fields may be solving for different types of analyses during a simulation. For example, in an induction heating problem, a harmonic electromagnetic analysis calculates Joule heating, which is used in a transient thermal analysis to predict a time-dependent temperature solution. The induction heating problem is complicated further by the fact that the material properties in both physics simulations depend highly on temperature.

Some of the applications in which coupled-field analysis may be required are pressure vessels (thermal-stress analysis), fluid flow constrictions (fluid-structure analysis), induction heating (magnetic-thermal analysis), ultrasonic transducers (piezoelectric analysis), magnetic forming (magneto-structural analysis), and micro-electromechanical systems (MEMS).

2.7.2 Sequentially Coupled Physics Analysis

A sequentially coupled physics analysis is the combination of analyses from different engineering disciplines which interact to solve a global engineering problem. For convenience, this chapter refers to the solutions and procedures associated with a particular engineering discipline as a *physics analysis*. When the input of one physics analysis depends on the results from another analysis, the analyses are coupled.

Some cases use only one-way coupling. For example, the calculation of the flow field over a cement wall provides pressure loads which you can use in the structural analysis of the wall. The pressure loadings result in a deflection of the wall. This in principle changes the geometry of the flow field around the wall, but in practice, the change is small enough to be negligible. Thus, there is no need to iterate. Of course, in this problem fluid elements are used for the flow solution and structural elements, for the stress and deflection calculations.

A more complicated case is the induction heating problem, where an AC electromagnetic analysis calculates Joule heat generation data which a transient thermal analysis uses to predict a time-dependent temperature solution. The induction heating problem is complicated further by the fact that the material properties in both physics simulations depend highly on temperature. This requires iteration between the two simulations.

The term *sequentially coupled physics* refers to solving one physics simulation after another. Results from one analysis become loads for the next analysis. If the analyses are fully coupled, results of the second analysis will change some input to the first analysis. The complete set of boundary conditions and loads consists of the following:

- Base physics loads, which are not a function of other physics analyses. Such loads also are called nominal boundary conditions.
- Coupled loads, which are results of the other physics simulation.
- Typical applications you can solve with ANSYS include the following:
- Thermal stress
- Induction heating
- Induction stirring

- Steady-state fluid-structure interaction
- Magneto-structural interaction
- Electrostatic-structural interaction
- Current conduction-magnetostatics

The ANSYS program can perform multiphysics analyses with a single ANSYS database. A single set of nodes and elements will exist for the entire model. What these elements represent are changes from one physics analysis to another, based on the use of the physics environment concept.

2.7.3 What Is a Physics Environment?

The ANSYS program performs sequentially coupled physics analyses using the concept of a *physics environment*. The term physics environment applies to both a file you create which contains all operating parameters and characteristics for a particular physics analysis and to the file's contents. A physics environment file is an ASCII file you create using either of the following:

Chapter 3

Analytical solution

3.1 Heat radiation

3.1.1 Introduction

Heat load on cryostat panels due to the conduction of heat load from hot surfaces by residual gas in the cryostat and due to the radiation of heat from hot surfaces are required to estimate. The heat load due to residual gas conduction from hot surfaces to liquid nitrogen surface is negligible compare to the heat load due to radiation from the hot surfaces.

Stefan Boltzman equation is applied to find heat radiated to the cryostat panels [2]

$$Q = \frac{\sigma A_1 \left(T_2^4 - T_1^4\right)}{\frac{1}{\varepsilon_1} + \left(\frac{A_1}{A_2}\right)\left(\frac{1}{\varepsilon_2} - 1\right)}$$

Where,

Stefan Boltzman constant, σ	=	$5.676 * 10^{-8}$ W/m²k
T_1	=	LN2 temperature, 80 K
T_2	=	Cryostat panel temp., 300 K
A_1	=	TSA of receiving panel
A_2	=	TSA of radiative body
ε_1	=	Emissivity of receiving panel
	=	Range from 0.03 to 0.3
ε_2	=	Emissivity of radiative body
	=	Range from 0.03 to 0.3

Using above relation, the heat load on each sector is tabulated as below.

3.1.2 Heat load calculation on panels (per sector)

The heat load on each panel is calculated. Temperature of the radiative surface is taken 300 K and receiving surface temperature is taken 80 K.

Table 3.1.1 Heat load on each panels

Sr. No.	**Panel Name**	**Area m²**		**Temp**		**Heat load**	**Heat flux**
		A1	**A2**	**T1**	**T2**	**W**	**W/m²**
01	TCP	0.750	0.705	300	80	37.07	49.43
02	FP	0.740	0.696	300	80	36.56	49.41
03	BCP	0.750	0.705	300	80	37.07	49.43
04	ICY	0.109	0.198	300	80	010.00	05.00

3.1.3 Summary

Radiated heat is maximum on top cryostat and bottom cryostat panels; these panels are in horizontally located. Heat flux on the inner cylinder is only 5 W/m², which is located in vertically. This heat is less compare to heat load on other panels.

3.2 Mass flow of LN_2

3.2.1 Introduction

The heat which is transferred to the cryostat plates is required to be carried by LN2 passing through copper tubes attached to the panels. The heat radiated to each panel is already calculated. Assuming the total heat radiated to the cryostat plate, will be carried away by the liquid nitrogen. Therefore, maximum amount of heat is considered in the calculation.

3.2.2 Flow parameter

Following equation is used to calculate mass flow rate [10]

$$q = m \times h,$$

Where,

h = is the latent heat of liquid nitrogen.
= 200 kJ/kg.
q = heat radiated to the panel, kW

Using above equation, quantity of the flow is calculated. The result is tabulated as below.

Table 3.2.1 Flow requirement

Sr. no.	Panel name	Avg. heat (w)	No. of path	Mass per Path (kg/s)	Total mass (kg/s)
1	TCP	37.40	4	$0.186*10^{-3}$	$0.744*10^{-3}$
2	BCP	37.40	4	$0.186*10^{-3}$	$0.744*10^{-3}$
3	FP	37.00	4	$0.185*10^{-3}$	$0.740*10^{-3}$
4	ICY	10.00	4	$0.050*10^{-3}$	$0.200*10^{-3}$

Total mass flow through the cryostat system per path is = 0.607×10^{-3} kg/s

3.2.3 Flow module of the Cryostat system

The system has flow module as shown below. The liquid nitrogen flows from main storage tank. Liquid nitrogen enters into main feeder line. The feeder has been attached to all panels. The nitrogen than distributed to all panels. When fluid enters into system, it is in 100% liquid form. The fluid leaves from system it becomes vapor due to gain of heat from surrounding.

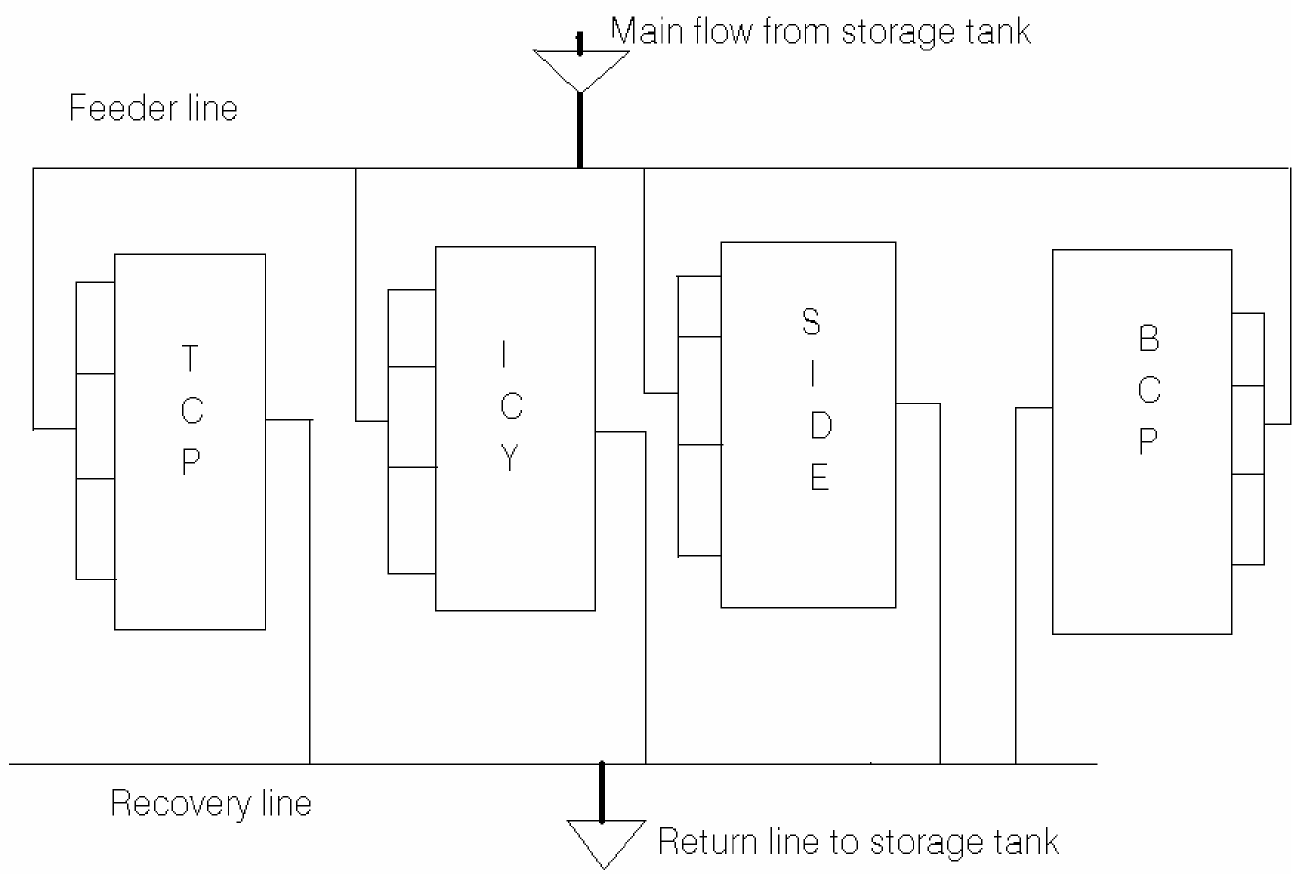


Fig. 3.2.1 Flow module

3.2.4 Parameter of flow for velocity

Estimate the mass flow rate at each sector of the cryostat plate. The velocity of the flow will be calculated from these flow rates.

Following mass continuity equation is used to calculate velocity [10, b]

$$m = A\ V\ \rho_L$$

where,

m	=	Mass flow rate, kg/s
d	=	Hydraulic diameter of the fluid $= \dfrac{4A}{P}$
d_o	=	Outer diameter of the tube, m
t	=	thickness of the tube, feeder & recovery, 0.001 m
ρ_L	=	density of the LN_2; Kg/m^3
A	=	Cross section area of the flow $= \dfrac{\pi d^2}{4}$
P	=	Perimeter of the tube $= \dfrac{\pi(d_o + d_i)}{2}$

3.2.5 Fluid velocity on the cryostat panels

LN_2 enters into main feeder line with 100% liquid form. When fluid leaves the panels and enters into recovery line it will be in 100% vapor form. Liquid density is used to find the velocity in feeder and vapor density is used to find velocity in recovery line. Velocity of the fluid is found on panels by mean density of the fluid.

Table 3.2.2 Velocity of flow

Panel	Mass (kg/s)	Hydraulic dia. (m)	Density kg/m^3	Velocity m/s
Feeder	$0.607*10^{-3}$	$2.10*10^{-3}$	800	0.2190
TCP	$0.186*10^{-3}$	$2.00*10^{-3}$	411	0.1440
ICY	$0.050*10^{-3}$	$2.00*10^{-3}$	411	0.0387
FP	$0.185*10^{-3}$	$2.00*10^{-3}$	411	0.1430
BCP	$0.186*10^{-3}$	$2.00*10^{-3}$	411	0.1440
Recovery	$0.607*10^{-3}$	$2.10*10^{-3}$	12.264	14.290

3.2.6 Summary

Estimate mass flow as per heat load on each panel. Using dimension of the supply system, the velocity of the fluid flow is calculated. As max load on top, bottom and front cryostat plate, more flow is required on these panels. The exit of the fluid is vapor therefore at recovery very high velocity of the fluid is required due to sharp reduction of density.

3.3 Losses in the tubes

3.3.1 Introduction

The presence of two-phase flow complicates the problem of predicting the pressure drop of the flowing fluid in several ways. First, the flow pattern is often different for vertical, horizontal and inclined flow. Second, there are several different flow patterns that may exist. Third, the flow may be laminar in the liquid and turbulent in the vapor phase or any of four different combinations may exit.

Finally, the flow pattern changes along the length of the pipe if the quality of the fluid changes because of heat transfer or pressure drop. Two fundamental models have been developed to predict the pressure drop in two-phase flow [7]:

1. The homogeneous model, in which the two phases are treated as a single phase possessing suitably averaged fluid properties.

2. The separated-flow model, in which the two phases are considered to be artificially segregated into two streams.

We are using homogeneous model to find pressure drop in the flow.

3.3.2 Pressure drop components

There are following five components of pressure drop

a. Entry losses
b. Losses in straight tubes due to friction.
c. Losses in bends due to friction.
d. Pressure drop due to change in elevation
e. Exit losses

3.3.3 Entry losses

When there is sudden change in pipe section on flow path, a rapid retardation or acceleration takes place as the kinetic energy is converted into the pressure energy. In our case there is sudden change of section to contraction. The energy loss in a sudden contraction depends on the coefficient of contraction which varies with the area of contraction.

Following relation has been used to calculate pressure drop due to straight abrupt change [3]

$$P_{ent} = \frac{\rho_L V^2}{2}$$

Where,

ρ_L = density of the LN_2 ; Kg/m^3

V = velocity of the fluid; m/s

When nitrogen enters into feeder of the system, it will be 100% liquid. When nitrogen leaves into recovery line of the system will be 100% vapor.

Assuming 10% vaporization of liquid nitrogen and it will be 90% liquid when enters into system. Using above equation pressure drop due to entry are tabulated as follows

Table 3.3.1 Entry pressure losses

Panels	**Pressure losses N/m²**
Feeder	19.18
TCP/BCP	7.465
FP	7.360
ICY	0.539

3.3.4 Losses in straight tube due to friction

A pressure loss in the tube is dependent on properties of the flowing fluid and nature of contact surface on which fluid flows. When flow is laminar the pressure drop will independent on contact surface because development of the sub-layer on surface. But in case of turbulent flow pressure drop dependent on surface of contact as absence of sub-layer.

Homogeneous model has been used to calculate pressure drop. Therefore, took mean properties of the fluid in the calculation. Following Hagen-Poiseuille´s equation has been used for calculating pressure drop [11]

$$h_L = \frac{fLV^2}{2gd}$$

Where,

f = coefficient of friction

L = Length of the tube (m)

g = Acceleration into gravity (m/s²)

Coefficient of friction is depended on Reynolds number, which is the ratio of the inertia force to viscous force. Which decides the major role in the deciding the flow regime [3].

Reynolds number, $R_d^e = \frac{Vd\rho}{\mu}$

Where,

μ = Dynamic viscosity of nitrogen,

= $8.015 * 10^{-5}$ Ns/m²

ρ = Density of fluid , 411 Kg/m³

Table – 3.3.2 f & R_d^e relation

Red < 2500	$f = \frac{64}{R_d^{e(-0.25)}}$
2500< Red <50000	$f = 0.316 * R_d^{e(-0.25)}$
Red > 50000	$f = 0.0054 + (0.396 * R_d^{e(0.3)})$

Initially, calculated coefficient of friction and tabulated result as follows;

Table 3.3.3 Value of "f" for panels

Sr. no	Rel	f
BCP	4738	0.0381
TCP	4738	0.0381
FP	4738	0.0381
ICY	1588	0.0400

The above pressure drop equation is modified by replacing velocity in term of mass flow, as follows [11]

Frictional pressure drop, $P_{fri} = \dfrac{8fm^2L}{\pi^2 d^5 \rho}$

This equation gives following results;

Table 3.3.4 Friction pressure losses

Panels	Pressure losses N/m²
TCP	0.315
BCP	0.315
FP	0.736
ICY	0.210

3.3.5 Losses in bends due to friction

Pressure drop in bends is due to friction and the change of the direction of flow, is calculated by following equation [11]

$$P_{bend} = \left(\frac{C_L V^2 \rho}{2} + \frac{8fm^2 L_B}{\pi^2 d^5 \rho} \right) * N$$

Where,

C_L = bend loss coefficient,

$= 0.13 + 1.85\left(\frac{r}{R}\right)^{0.35} * \left(\frac{\beta}{180}\right)^{0.5}$

β = Bend angle in degree

L_B = Bend length (m)

R = Bend radius (m)

r = Radius of tube (m)

N = No. of bends

Using this equation, the results are tabulated as follows;

Table 3.3.5 Pressure losses in bends

Panels	N	R	r	B	CL	Pbend (N/m²)
TCP/BCP 1	1	4	4	180º	1.98	39.06
2	1	4	4	180º	1.98	39.06
3	1	8	4	70º	1.0352	36.07
4	1	6	4	60º	1.057	42.07
Sum	--	--	--	--	--	156.26

FP	13	4	4	90°	1.98	186.14
ICY	14	4	4	180°	1.98	32.78

3.3.6 Losses due to change in elevation

Pressure drop due to change in elevation. Inner cylinder has cooper tubes which are vertical in position.

Applied homogeneous correlation [7]

$$p_{ele} = (\alpha * \rho_G + (1-\alpha)\rho_L)g \sin\theta$$

Where,

α = Void fraction, $\dfrac{1}{\left(1+\left(\dfrac{1-x}{x}\right)\left(\dfrac{\rho_G}{\rho_L}\right)\right)}$

x = quality of the fluid

ρ_G = density of the vapor fluid (Kg/m³)

ρ_L = density of the liquid fluid (Kg/m³)

g = gravity (m/s²)

In given cryostat system, the inner cylinder has vertical configuration. Therefore, the pressure lose due to elevation is occurred in inner cylinder, which is calculated by using above relation. In reference paper, Idsinga et. al has emphasized that during upward flow fluid, pressure gradient will be more as compare to downward flow. And he assume percent of error by above equation would be ± 20%.

Therefore, we here multiplied 0.8 to down ward flow pressure increased and 1.2 to upward flow pressure drop to get approximate pressure gradient.

Table 3.3.6 Pressure gradient at inner cylinder

Sr. No.	**Flow pattern**	**Void fraction**	**Pr. Change per unit length (N/m²)**	**Total pr. changed (N/m²)**
01	Downward	0.998	+6362.00	+55477
02	Upward	0.989	- 9457.00	- 92654
03	Downward	0.977	+6228.00	+54432
04	Upward	0.942	- 9007.00	- 78721

Pressure drop due to elevation at inner cylinder=61466 N/m²

3.3.7 Exit losses

As already calculated pressure drop at entry, similarly found the pressure drop at exit. Following correlation used to find pressure drop at exit [3]

$$P_{exit} = \frac{V^2 \rho}{4}$$

The density of the flow is taken for 100% vapor.
Using above relation, we tabulated results as follows;

Table 3.3.7 Exit pressure losses

Panel	**Velocity (m/s)**	**Pressure (N/m²)**
TCP	15.12	700
FP	15.11	700
BCP	15.12	700
ICY	1.29	5.56

3.3.8 Total pressure drop

The pressure drop form each panel has been considered. The causes of the pressure drop have been defined. Each component of pressure drop has been calculated. Finally, summation all the component of pressure drop to have total pressure drop on each panel. The results have been tabulated as follows.

Table 3.3.8 Total pressure drop

Panel	**Entry N/m²**	**Friction N/m²**	**Bend N/m²**	**Elevation N/m²**	**Exit N/m²**	**Total N/m²**
TCP	7.465	0.315	156.26	--	700	864.04
BCP	7.465	0.315	156.26	--	700	864.04
FP	7.360	0.736	186.14	--	700	894.24
ICY	0.539	0.210	32.78	61465	5.56	61504

3.3.9 Summary

The pressure drop has been calculated by homogeneous approach. Pressure drop due to entry is minor as compared to other pressure drop. Pressure drop due to bend is dominant. Pressure drop due to bend is very high as compare to friction and exit losses. These losses are maximum in top and bottom cryostat plates. Pressure drop is high in exit because of vapor state. Most pressure drop of the flow of the fluid is in inner cylinder due to vertical flow i.e. against gravity.

Pressure drop is maximum on flow path of fluid in inner cylinder. As maximum pressure drop is on inner cylinder, therefore, the flow of the fluid may not be adequate as per requirement of the heat load. This will be verified during analysis on Ansys.

CHAPTER 4

Analysis with Ansys

4.1 Analysis of TCP & BCP by Ansys

4.1.1 Configuration

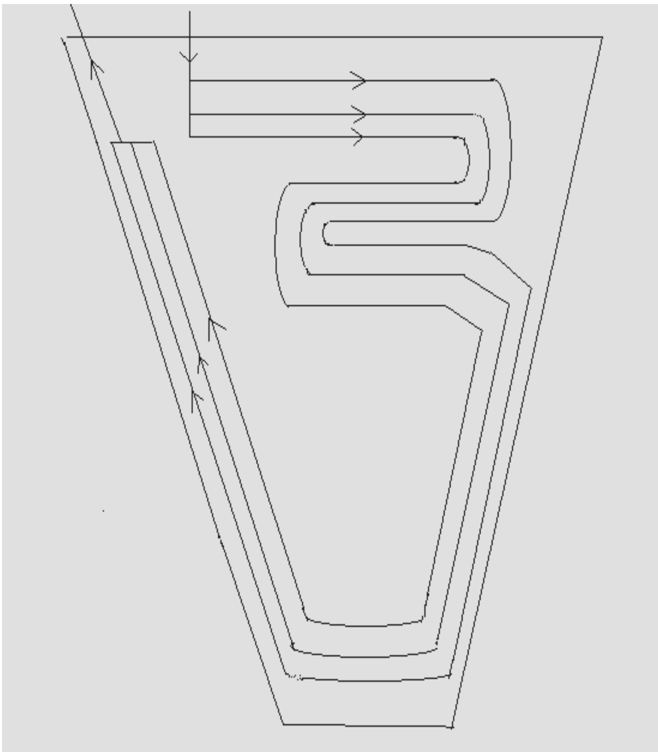


Fig. 4.1.1 Top/bottom cryostat plate

The top and bottom cryostat plate are trapezoidal in shape.

Tubes mounted on SS304L plate.

Straight length = 3877 mm

Bend length = 1507 mm

No. of bends = 04

ID of tube = 08 mm

4.1.2 Parameters

- Mass flow rate = $0.186 * 10^{-3}$ Kg/s
- Velocity of flow at inlet = 0.144 m/s
- Temperature at inlet = 80 Kelvin
- Inlet fluid pressure = 210000 N/m²
- Heat flux on surface = 49.43 W/m²

4.1.3 Element selection

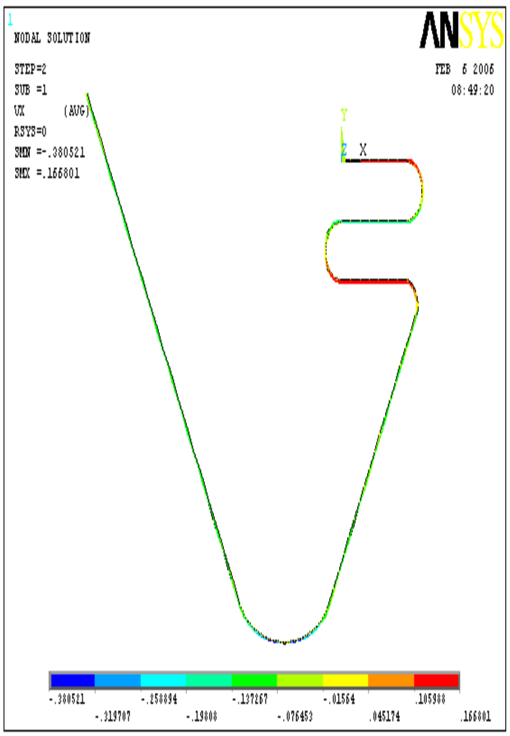
- FLUID 141 element is selected
- Densities, viscosities, specific heat & conductivities are input with temperature variation.

4.1.4 Boundary condition

- Velocity on the walls is zero
- Inlet velocity applied at inlet
- Heat flux applied
- Reference pressure at outlet is zero
- Inlet temperature is applied
- Reference pressure is set

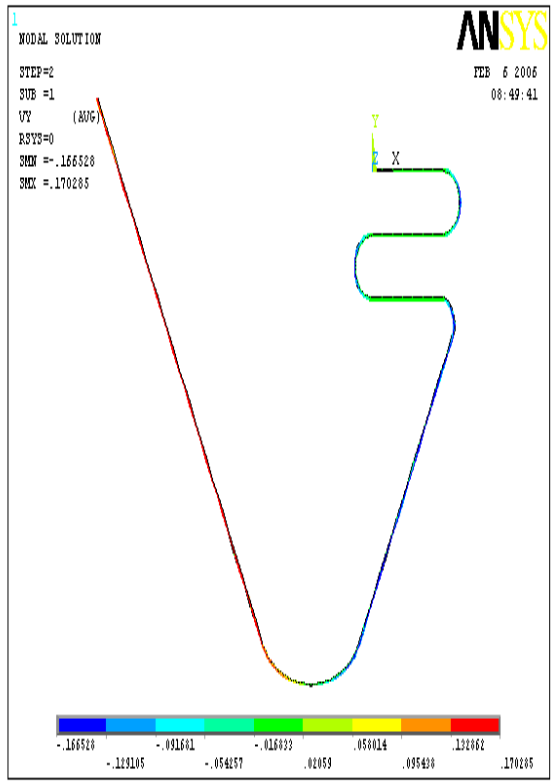
4.1.5 Results of Ansys analysis

Following result are produced from Ansys analysis. The figures are shown velocities of fluid flow in the tube. Initially inlet velocity is 0.144 m/s. But due to heat flux on the panel, density of the fluid is decreased which cause increase in velocity to 0.3805 m/s.



Velocity image x direction

4.1.2 velocity x- direction

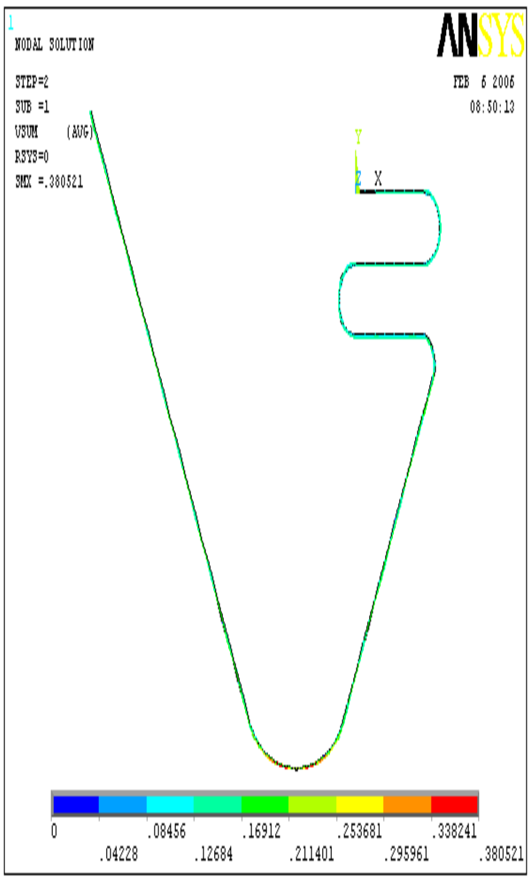


Velocity distribution y direction

4.1.3 velocity y-direction

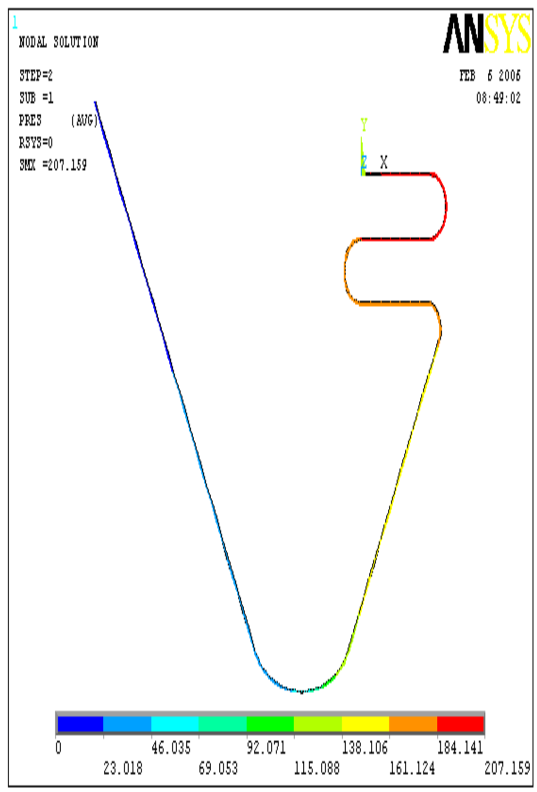
Results of Ansys analysis

Pressure loss during flow is shown in following figure. Analytically pressure drop for two phase flow is 864.4 N/m^2. While for single phase flow it is 140 N/m^2. Ansys analysis has given 207 N/m^2 pressure drop. Analytically estimated pressure drop is on basis of inlet density for single phase and mean density for two phase flow. However, due to heat flux density and velocity is changing which account more pressure drop as compare to analytical solution.



Velocity image

4.1.4. velocity



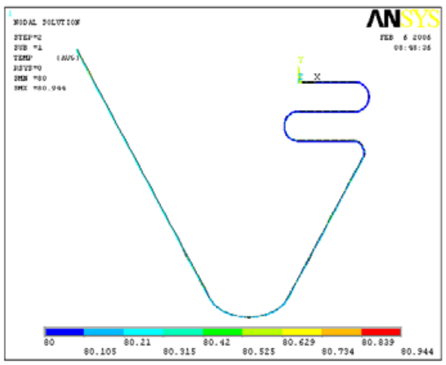
Pressure distribution

4.1.5 pressure distribution

Images of velocity & pressure distribution

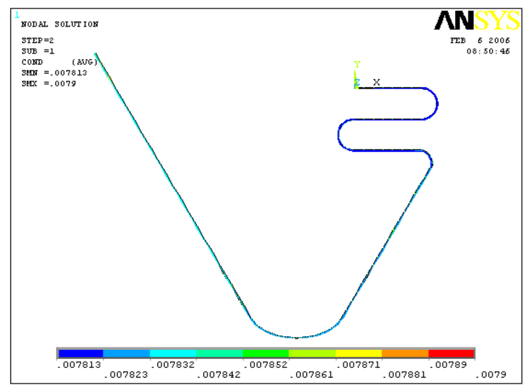
Following figures of analysis are shown for temperature and conductivity variation during flow. Rise in temperature is 0.944 K from inlet to out let. This will affect density, conductivity, specific heat and viscosity of the fluid.

As temperature rise is very little therefore the fluid is not converted in to vapor form. This means that flow remain single phase only.



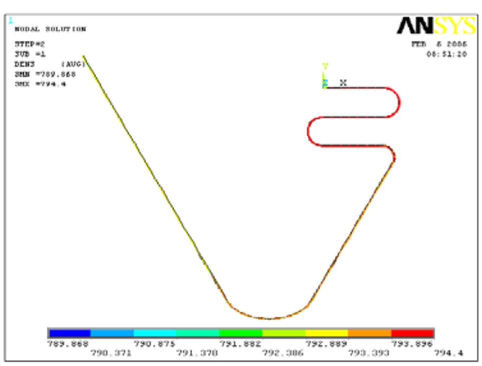
Temperature distribution

4.1.6. Temperature variation



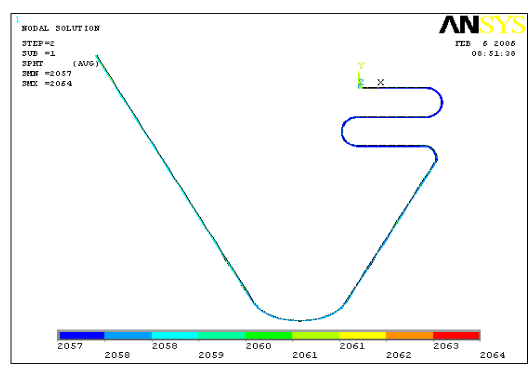
conductivity variation

4.1.7 conductivity variation



Density variation

4.1.8 density variation



Specific heat variation

4.1.9. specific heat variation

Images of properties changed

4.1.6 Results review of Ansys analysis of TCP/BCP

The results of analysis of Top and Bottom cryostat panel have been reviewed. This section of report is used to discussion analysis results. The following are summarized the

1. Estimated inlet velocity of the flow is 0.144 m/s. From analysis we understood that velocity is increased to 0.3805 m/s. This change in velocity is due to change in density of the flow. Density is varying because of temperature rises.

2. The analyses also predict that temperature is reached to 80.994 K from 80 K at inlet. This shows that temperature variation is little bit. That is 0.994 Kelvin only. If we see the properties of nitrogen w.r.t. temperature, we understand that flow remain single phase unlike two phase assumed initially.

As flow remains single phase therefore we require calculating pressure drop for considering single phase only.

4.2 Pressure drop calculation for single phase flow

4.2.1 Entry losses

Same equations are being used for calculating pressure drop due to single phase flow. To estimate entry pressure drop we used following relation, [3]

$$P_{ent} = \frac{\rho_L V^2}{2}$$

Where,

ρ_L = density of the LN2 ; Kg/m³

V = velocity of the fluid; m/s

Table 4.2.1 Entry pressure losses

Panels	Pressure losses N/m²
Feeder	19.18
TCP/BCP	8.29
FP	8.18
ICY	0.599

4.2.2 Losses in straight tube due to friction

As flow is single phase therefore we use general equation, unlike homogeneous model had been used to calculate pressure drop for two phase. Therefore, inlet properties at 80 Kelvin are used to calculate the pressure drop. Following Hagen-Poiseuille´s equation is used [11]

$$h_L = \frac{fLV^2}{2gd}$$

Where,

f = coefficient of friction

L = Length of the tube (m)

g = Acceleration into gravity (m/s²)

Coefficient of friction is depended on Reynolds number, which is the ratio of the inertia force to viscous force. Which decides the major role in the deciding the flow regime [3]. At this stage properties are used at inlet conditions, rather homogeneous properties for two phase flow.

Reynolds number, $R_{ed} = \frac{Vd\rho}{\mu}$

Where,

μ = Dynamic viscosity of nitrogen,

= $8.015 * 10^{-5}$ Ns/m²

ρ_L = Density of fluid , 794.5 Kg/m³

Following correlation is valid for estimation of coefficient of friction as this is independent of phase of flow [11]:

Table – 4.2.2 f & Red relation

Red < 2500	$f = \frac{64}{R_{ed}}$
2500< Red <50000	$f = 0.316 * R_{ed}^{(-0.25)}$
Red > 50000	$f = 0.0054 + (0.396 * R_{ed}^{(0.3)})$

Estimated of coefficient of friction from above equation is tabulated as follows;

Table 4.2.3 Value of "f" for panels

Sr. No.	R_{ed}	f
BCP	855.8	0.0748
TCP	855.8	0.0748
FP	844	0.0758
ICY	610.81	0.0400

The above pressure drop equation is modified by replacing velocity in term of mass flow, as follows [11]

Frictional pressure drop, $P_{fri} = \frac{8fm^2L}{\pi^2 d^5 \rho}$

This equation gives following results;

Table 4.2.4 Friction pressure losses

Panels	Pressure losses N/m²
TCP	0.162
BCP	0.162
FP	0.378
ICY	0.108

4.2.3 Losses in bends due to friction

Pressure drop in bends is due to friction and the change of the direction of flow, is calculated by following equation [11]

$$P_{bend} = \left(\frac{C_L V^2 \rho}{2} + \frac{8 fm^2 L_B}{\pi^2 d^5 \rho_L} \right) * N$$

Where,

C_L = bend loss coefficient,

$= 0.13 + 1.85 \left(\frac{r}{R} \right)^{0.35} * \left(\frac{\beta}{180} \right)^{0.5}$

β = Bend angle in degree

L_B = Bend length (m)

R = Bend radius (m)

r = Radius of tube (m)

N = No. of bends

Using this equation, the results are tabulated as follows;

Table 4.2.5 Pressure losses in bends

Panels	**N**	**R**	**r**	**β**	**CL**	**Pbend (N/m²)**
TCP/BCP 1	1	4	4	180º	1.98	32.20
2	1	4	4	180º	1.98	32.20
3	1	8	4	70º	1.0352	28.25
4	1	6	4	60º	1.057	36.03
Sum	--	--	--	--	--	128.68
FP	13	4	4	90º	1.98	250.64
ICY	14	4	4	180º	1.98	29.064

4.2.4 Exit losses

Estimate pressure drop at exit by similar equation of two phase flow but here used liquid density of the fluid unlike vapor density of two phase flows.

Following correlation used to find pressure drop at exit [3]

$$P_{exit} = \frac{V^2 \rho_L}{4}$$

Using above relation, we tabulated results as follows;

Table 4.2.6 Exit pressure losses

Panel	Velocity (m/s)	Pressure (N/m²)
TCP	0.1440	3.733
FP	0.1440	3.733
BCP	0.1430	3.680
ICY	0.0387	0.2695

4.2.5 Total pressure drop

The pressure drop form each panel has been considered. The causes of the pressure drop have been defined. Each component of pressure drop has been calculated. Finally, summation all the component of pressure drop to have total pressure drop on each panel. The results have been tabulated as follows.

Table 4.2.7 Total pressure drop

Panel	Entry N/m²	Friction N/m²	Bend N/m²	Exit N/m²	Total N/m²
TCP	8.29	0.162	128.68	3.733	140.91
BCP	8.29	0.162	128.68	3.733	140.91
FP	8.18	0.378	250.64	3.680	262.88
ICY	0.599	0.108	29.064	0.269	30.04

- Estimated pressure drop due to single phase flow is 140.91 N/m²
- Ansys analysis pressure drop is 207.159 N/m²
- During estimation we used density of the fluid is 794.5 kg/m³ but in actual density varies across flow due to change in temperature therefore velocity increases, which influence pressure drop.

4.3 Analysis of the Inner cylinder by Ansys

4.3.1 Configuration of Inner cylinder

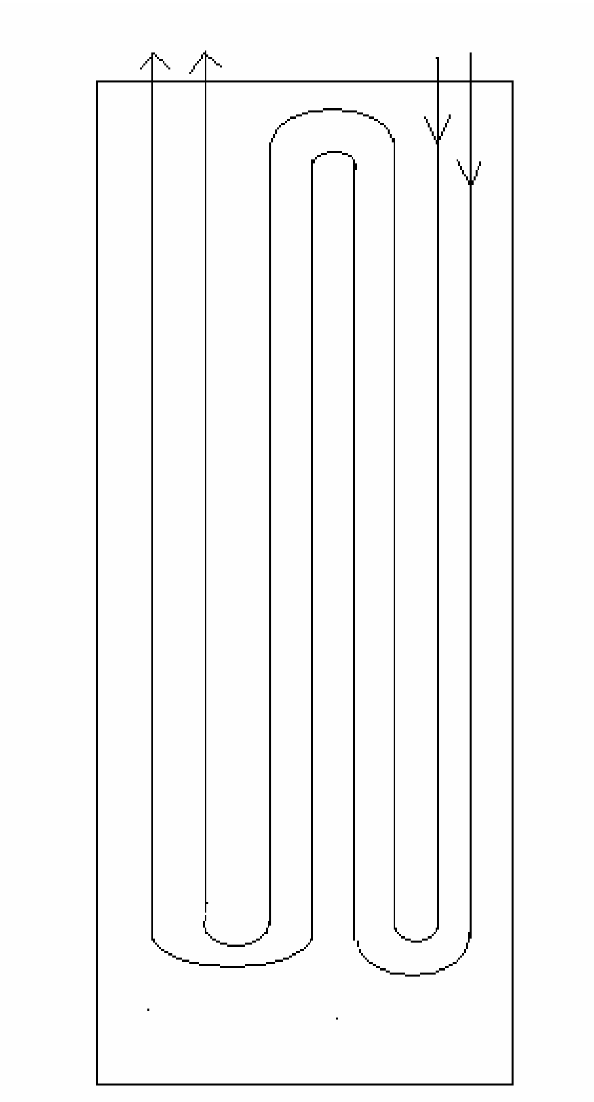


Fig.4.3.1 Inner cylinder

Inner cylinder has in cylindrical shape.
The cylinder had been made from SS304L.
Tubes are mounted on the surface.
Straight length =34960mm
Bends length =3934 mm
No. of bends = 14
ID of tube = 8 mm

4.3.2 Parameters of Inner cylinder

- Mass flow rate = 0.05e-3 kg/s
- Velocity of the flow at inlet = 0.0387 m/s

- Temperature at inlet = 80 Kelvin

- Inlet fluid pressure = 210000 N/m²
- Heat flux on surface = 5.00 W/m²

4.3.3 Element selection & properties

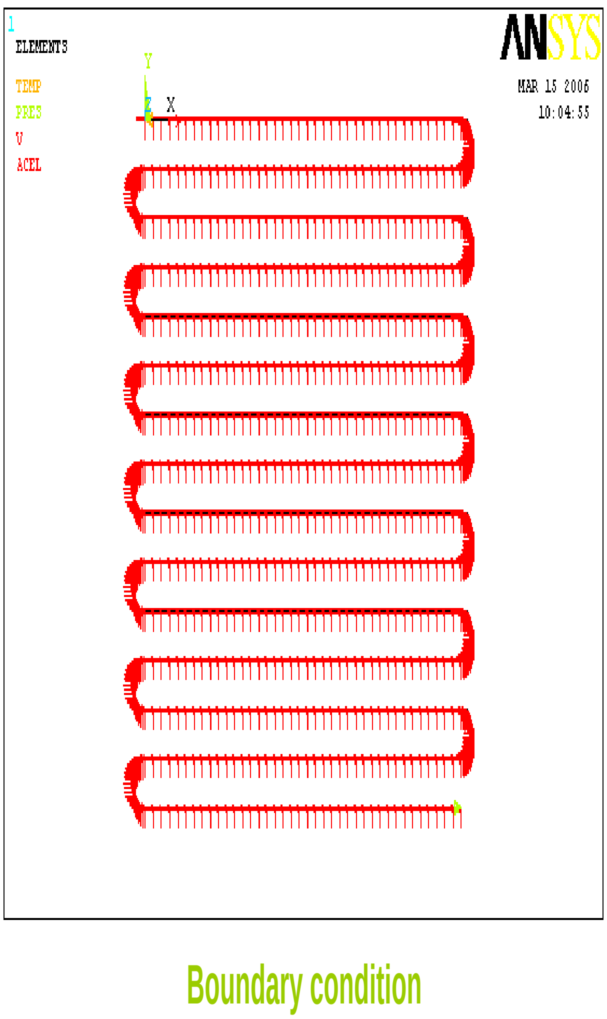
- FLUID 141 element is selected.
- Densities, Viscosities, Specific heat and Conductivities are input with temperature variation

4.3.4 Boundary conditions applied

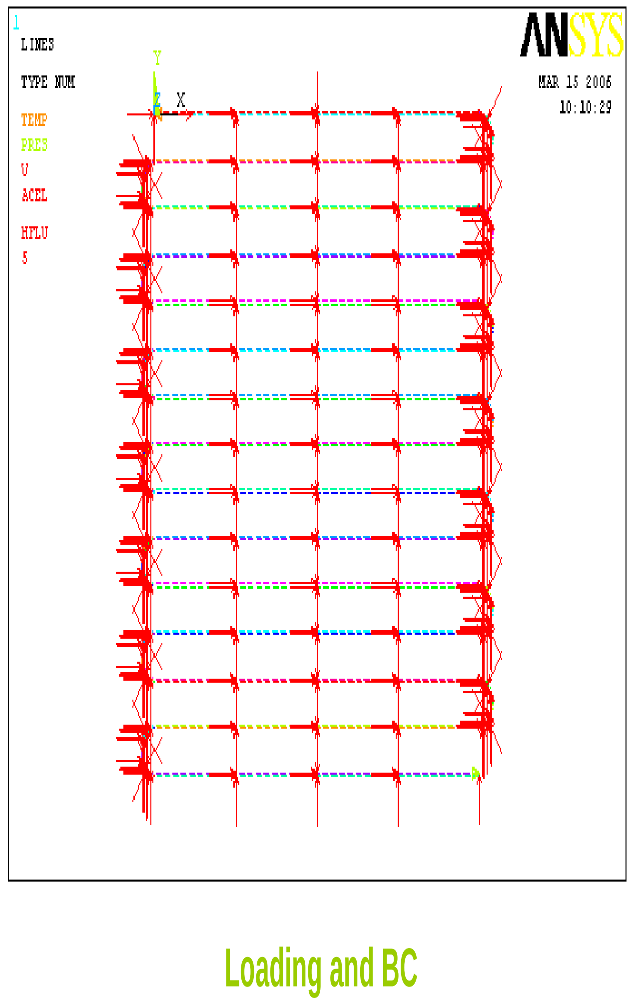
- Heat flux applied on lines.
- Velocity on the walls is zero.
- Inlet velocity is given to the fluid.
- Out let pressure is zero.
- Inlet temperature is given to fluid.
- Set pressure of the flow

4.3.5 Results of Ansys analysis

Inner cylinder panel have total length of tubes are 34.96 meters. The tubes are located vertically. The aspect ratio of elements along length is 600 while at the bend it is 800. The meshing is dense inner wall while coarse towards centre of the flow. Heat flux of 5 W/m² is imparted on tube wall. Velocity at inlet is applied 0.0384 m/s with temperature of 80 K. Setting inlet reference pressure 2.1 bars. Outlet relative pressure is set to zero. Following figures are shown boundary and loading condition of the system.

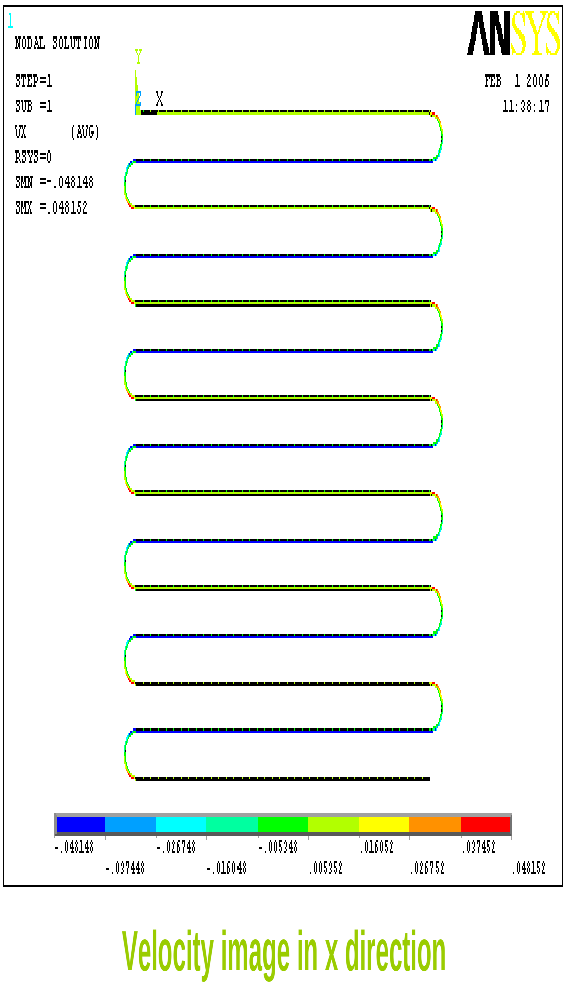


4.3.2 Boundary condition

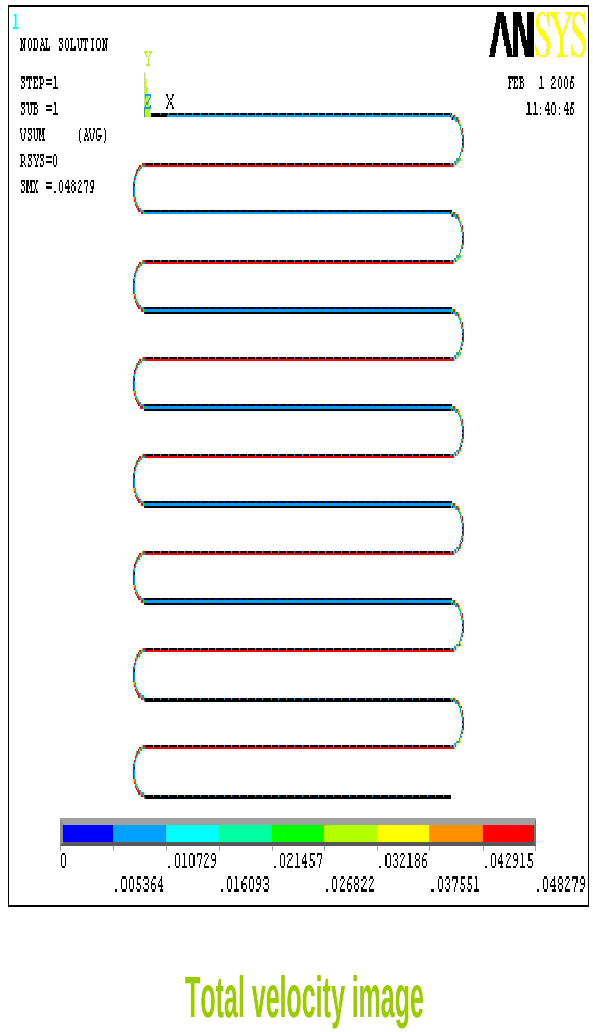


4.3.3 Lading condition

Following figures are shown velocity distribution in the tubes. Inlet velocity is 0.0384 m/s. From Ansys analysis results it is shown that velocity along x direction is increased to 0.04815 m/s because of decreased in density of the fluid. Actual velocity is increased to 0.048279 m/s.

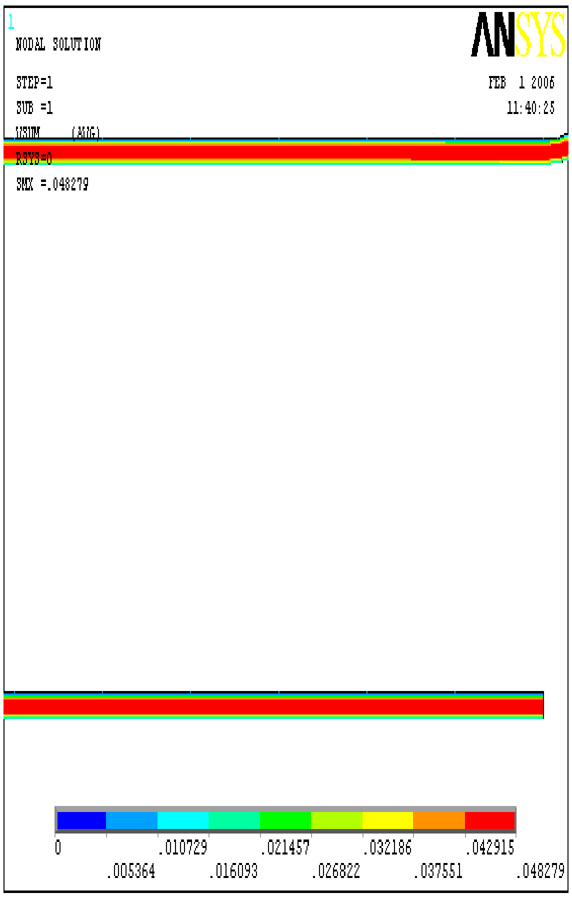


4.3.4 X – direction velocity



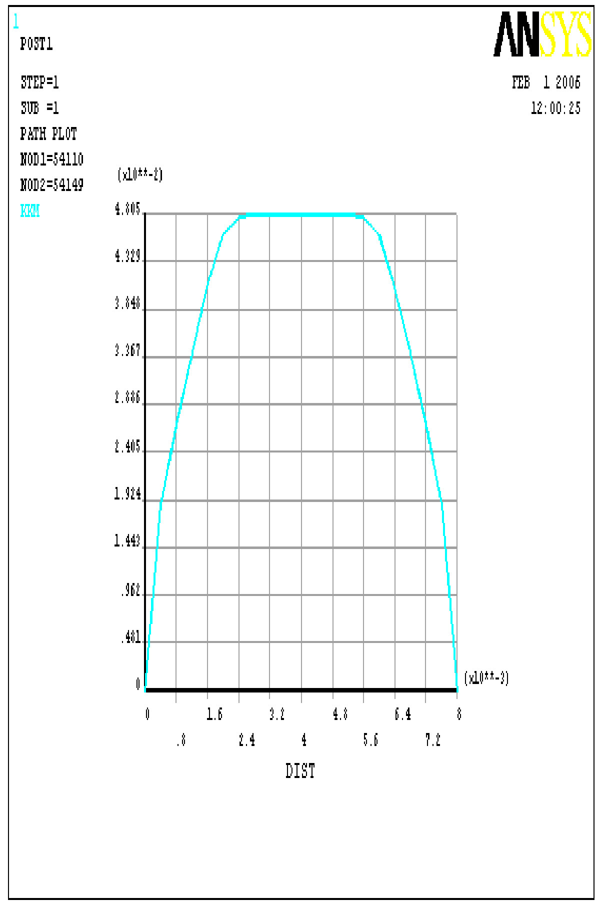
4.3.5 velocity

Following figures are shown closed view of velocity and velocity profile inside the tube. The maximum velocity is along longitudinal axis of the tube. In velocity profile, it is shown that velocity at the tube wall is zero and it is increased towards centre of the tubes by parabolic path.



Closed view of velocity image

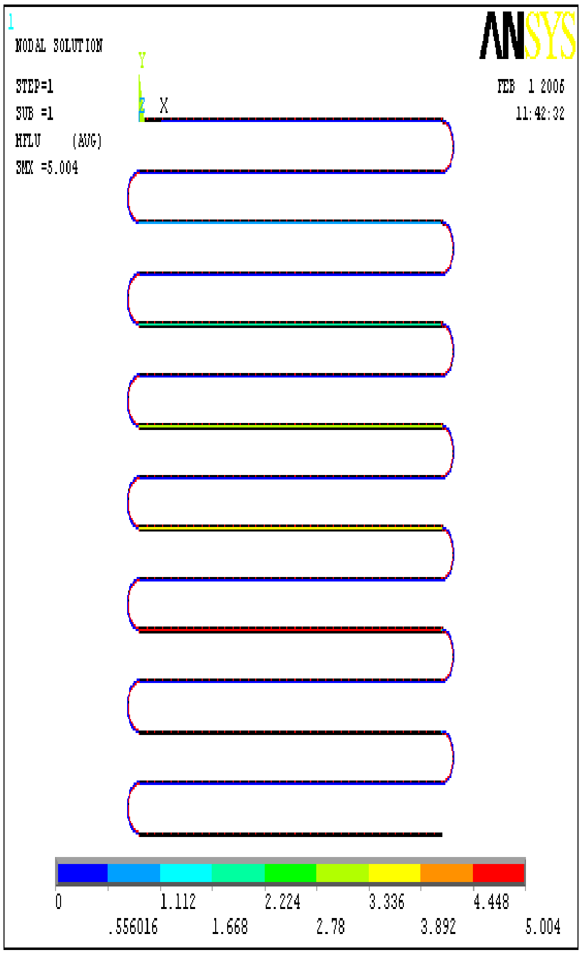
4.3.6 close view velocity



Velocity profile

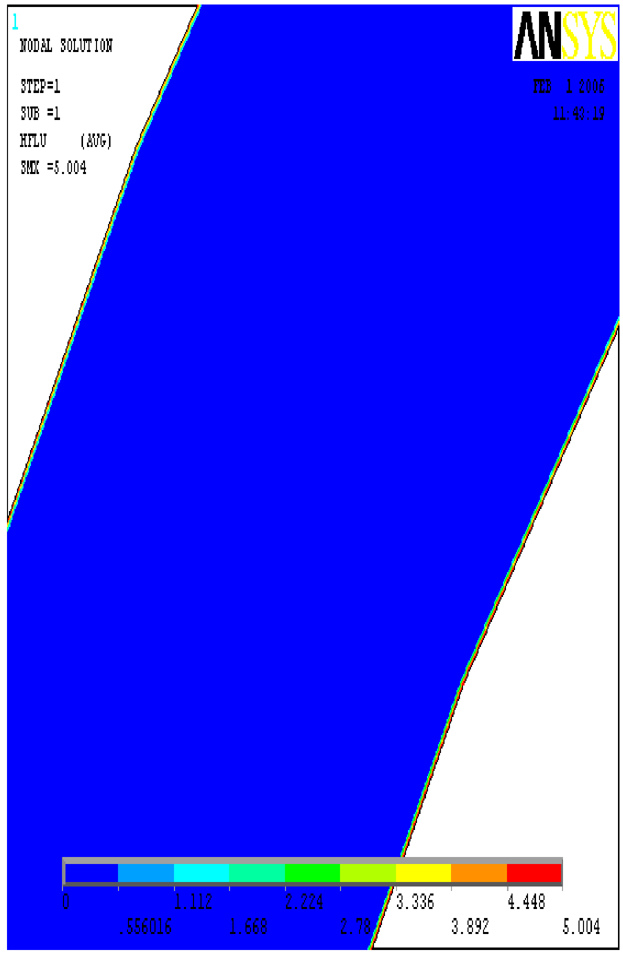
4.3.7 velocity profile

Figure below are shown heat flux on the tubes and closed view of the applied heat flux. Heat applied on wall is maximum while towards the centre it is decreasing.



Heat flux distribution image

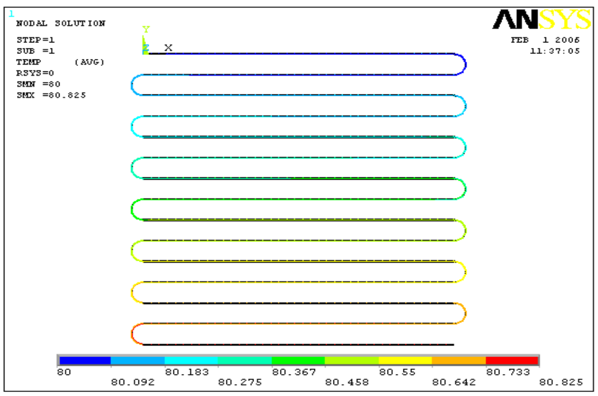
4.3.8 heat flux



Closed view of heat flux distribution

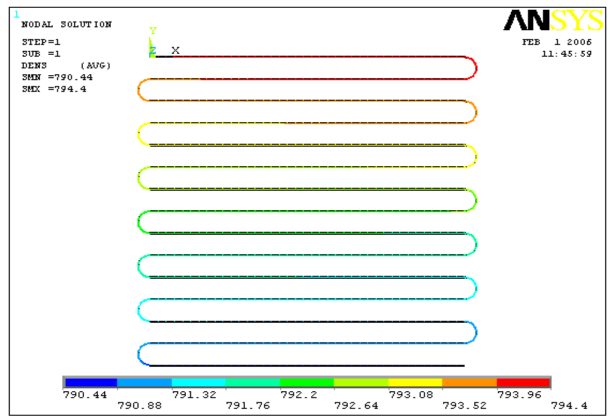
4.3.9 close view heat flux

Figure 4.3.10 is shown temperature variation in the flow. Inlet temperature is 80 K while rise in temperature due to heat flux is 80.826 K. Maximum temperatures at outlet of the flow. Figure 4.3.11 is shown density variation. Density is varies from inlet 794.40 kg/m³ to 790.44 kg/m³. Maximum density is at inlet while lowest density is at outlet. Similarly, viscosity variation are shown in 4.3.12 and 4.3.13.



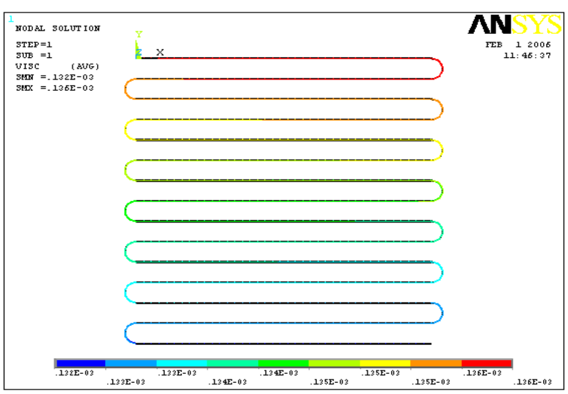
Nodal temperature distribution

4.3.10 temperature



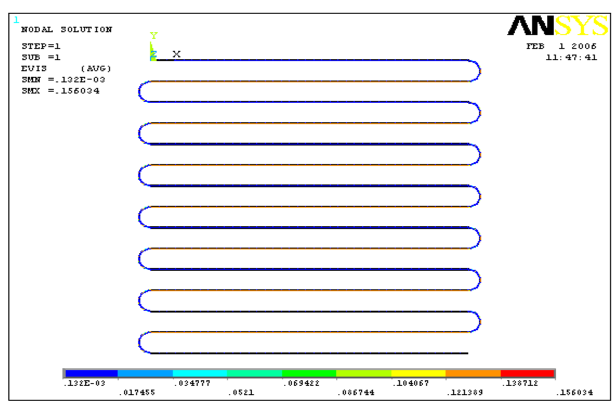
Density variation

4.3.11 density



Viscosity variation

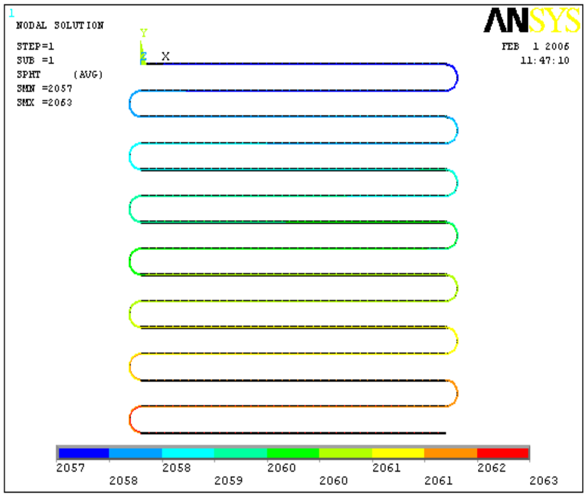
4.3.12 viscosity



Effective viscosity

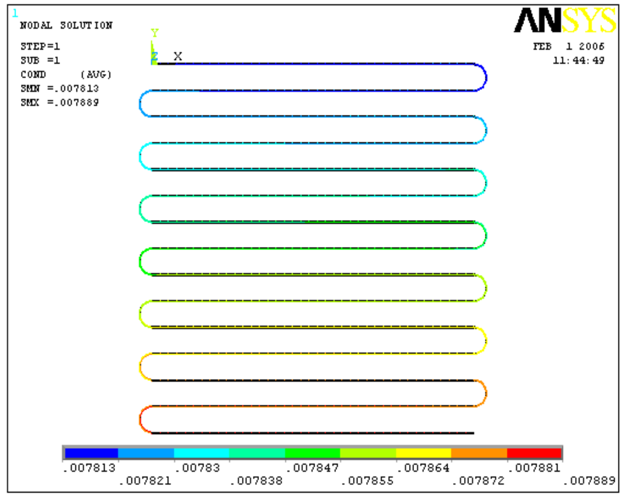
4.3.13 effective viscosity

Following figures are shown specific heat, conductivity, and pressure distribution inside the tube. Specific heat is changed from 2057 J/kg K to 2043 J/kg K. While conductivity of fluid is changed due to temperature variation is shown in figure 4.3.15.



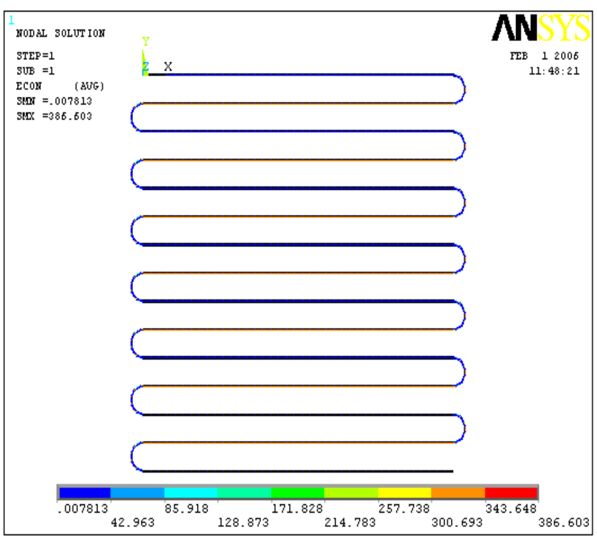
Specific heat variation

4.3.14 specific heat variation



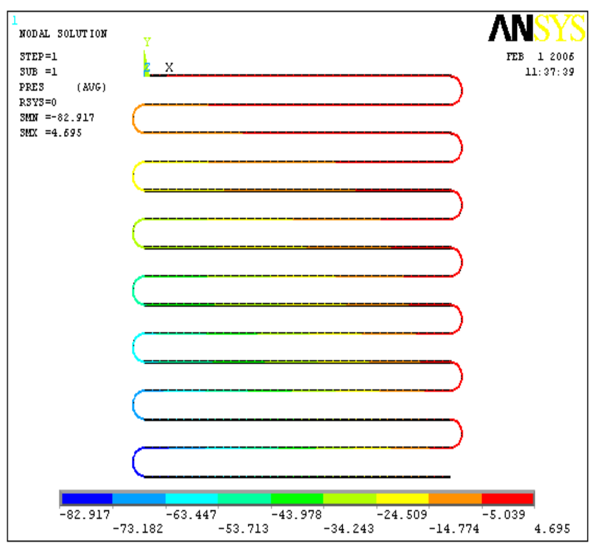
Conductivity variation

4.3.15 conductivity variation



Effective conductivity

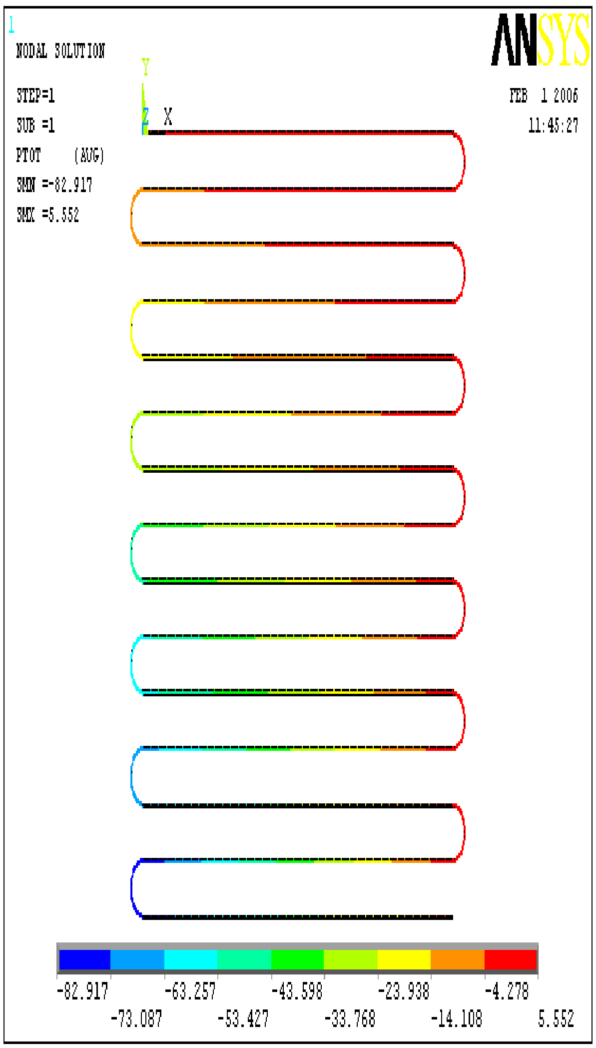
4.3.16 effective conductivity



Pressure distribution

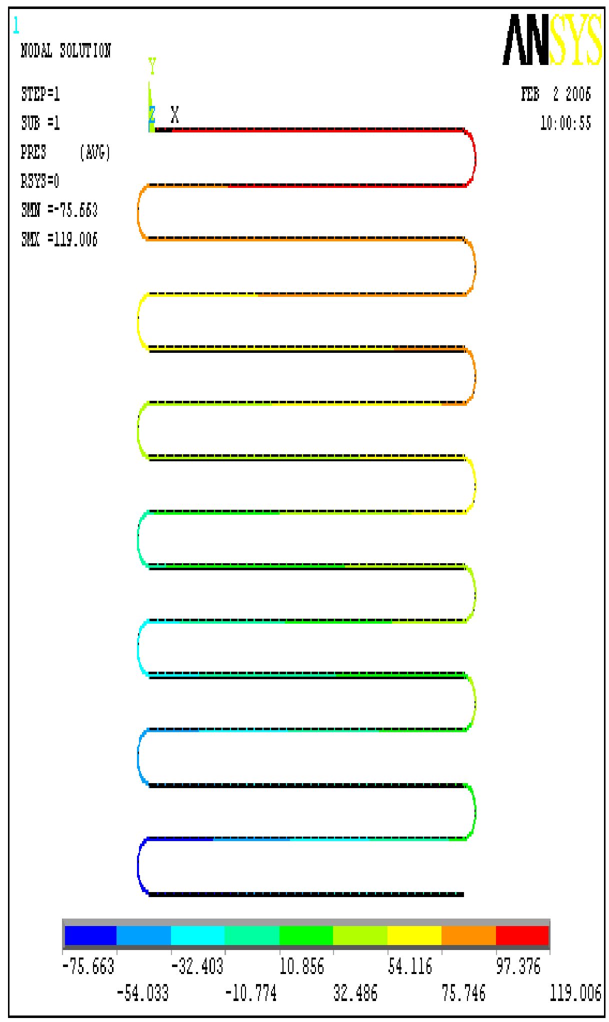
4.3.17 pressure

Figure shown below is for pressure distribution of the flow inside the tubes. Figure 4.3.18 is shown pressure distribution for inlet pressure of 2.1 bars. The results are indicated that negative pressure is developed inside tube. Negative pressure developed in tubes is -82.917 N/m². Figure 4.3.19 is indicated pressure distribution in tubes for increased inlet pressure 3.1 bars. This figure is shown that negative pressure developed is reduced to -75.663 N/m².



Total pressure distribution

4.3.18 pressure



Inlet pressure increased to 3.1 bar

4.3.19 pressure

3.4.6 Results review of Ansys analysis of Inner cylinder

The results of analysis of inner cylinder panel have been reviewed. This section of report is used to discussion analysis results. The following are summarized the

1. Temperature is increased little i.e. 0.825 Kelvin. Therefore, density variation is small which indicates that fluid remains single phase in the inner cylinder also.
2. Estimated pressure drop due to two phase flow in inner cylinder is 61504 N/m² but Ansys analysis show that flow remains single phase due to little heat flux which increase temperature by 0.825 Kelvin only.
3. Therefore, we require to estimate pressure drop considering single phase for this panel.
4. Already estimated pressure drop by single phase in inner cylinder is 30.04 N/m²
5. As negative pressure in the flow which indicates that flow is chocked due to static pressure of the fluid
6. Now, we require to find solution for problem to normalize the flow.

4.4 Option available to solve problem

4.4.1 Introduction

We understood that flow is get chocked due static pressure. Now we require to solve the problem considering constraints and economy of the system. There are two options by which the solution can be had. These are as follows.

1. Re-design the complete system.
2. Increase inlet pressure

4.4.2 Relevance of the solution

The system is already been laid therefore it is costly to go for redesign the system. This will creates new technical problem for other part of system. Also it will increase cost of the system. At this stage, the solution which will easy to get with lower effort is second option we have. That is to increase inlet pressure at inner cylinder from 210000 N/m^2.

4.4.3 Iteration for finding suitable inlet pressure

Initially inlet pressure was 210000 N/m^2. But from this analysis, it is understood that inlet pressure is not suitable for normalizing flow through inner cylinder. However, this pressure is suitable for normalize flow through Top, Bottom and Side cryostat panel. Therefore, it is required to estimate inlet pressure through inner for normalizing flow. This is done by increasing inlet pressure from 2.1 bar and find out the pressure at inlet which gives normalize flow.

Chapter 4

Conclusion and solution

4.1 Conclusion

The work has been assigned to find the problem of cryostat panel in which desired temperature is not being achieved. Initially analytical solution is estimated. In analytical solution, heat load on panels due to radiation is calculated. Through this load estimation, it is understood that maximum heat load I son the TCP and BCP. On inner cylinder very low heat fluxes are available that is 5 W/m^2.

In analytical solution, mass flow and velocity on each panel is being estimated. The diameter of all the tubes through which LN_2 is flowing is 8 mm. But velocity and mass flow rate is different on each panel.

Next estimation was for pressure drop calculation. Initially pressure drop for each panel is estimated considering two phase flow. However, when results are reviewed from Ansys analysis, it is understood that temperature rise due to heat flux on panel is just less than one Kelvin. Therefore, all panels requires to analysis for single phase flow only. Than again pressure drop calculation is done for single phase flow.

Following conclusions are drawn from analysis:

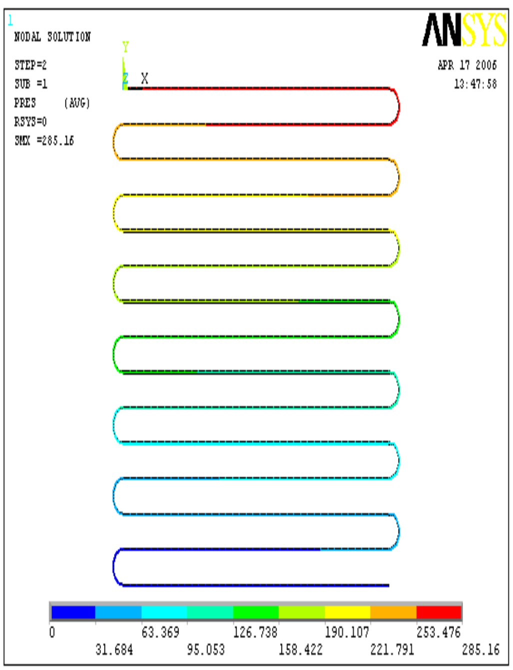
- The flow remain single phase, unlike it is assumed for two phase flow.
- Temperature rise of LN_2 flow is less than one Kelvin
- As LN_2 sensitive to temperature and pressure for its properties. Therefore it's properties like density, conductivity, specific heat and viscosity are changed.
- The flow through TCP, BCP and side cryostat panel is normal for given pressure. As TCP & BCP are in horizontal location therefore no gravity effects on this panels. But side cryostat panels has a few meters of length in vertical direction, however applied pressure is able to flow through normally.
- At inner cylinder, flow is not normal for applied pressure. As inner cylinder has flow towards and against gravity all the times. Therefore gravity has great effect on flow due to number of bends and total length of tube is large as compare to other panels.

- Considering above parameters in consideration, it is conclude that flow get chocked due to it's own static pressure.
- As flow gets chocked therefore flow is not normal. This is happened due very low heat load on inner cylinder panel which does not turn liquid nitrogen in to vapor.
- As comparing pressure drop by analytically calculated and Ansys analysis, it is not matched as density of the fluid is changing when it is flowing. However, for analytical calculation constant density and velocity is used but this is not true for actual practice.
- In this case, if initially density and velocity are taken for pressure drop the analytical pressure drop is less than Ansys drop. Reversely, if final density and velocity are taken for analytical solution than pressure drop is more than Ansys analysis.

4.2 Solution

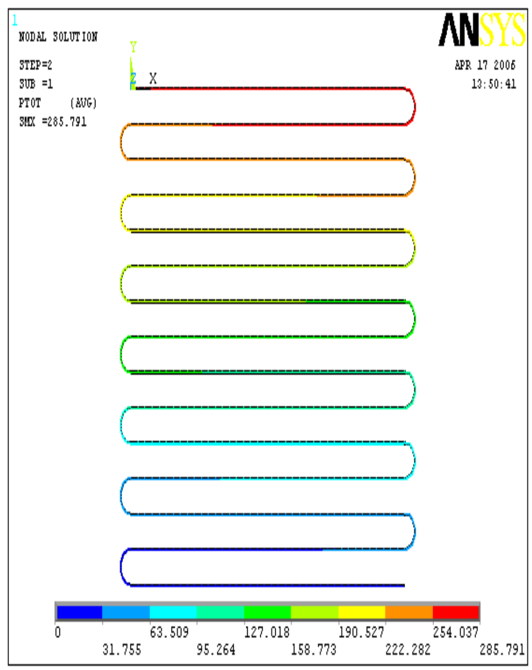
At this stage, analytical and Ansys results are available. Also problem and it's cause is defined. Now it is required to find solution for the problem. As it has already been discussed about solution via 3.5 articles. The solution will have by increasing inlet pressure from 210000 N/m² which is initial inlet pressure.

By iterative process, final inlet pressure at which flow gets normalized in inner cylinder is achieved at 410000 N/m².



Inlet pressure increased to 4.1 bar

4.2.1 Pressure distribution



Total pressure

4.2.2 Total pressure distribution

4.3 Future scope of work

The assigned work is started for analysis by assuming two phase fluid flow. The two phase flow in the system is like that, at entrance nitrogen is in liquid form and at exit it is in vapor form. As this assumption is on base that when liquid nitrogen flowing through tubes it gains heat due to heat flux on tube which turn liquid into vapor.

However, it is conclude from analysis that liquid nitrogen remains liquid as heat flux is less which is not capable for rising temperature up to required vapor form. Therefore, unlike stated at initial assumption for two phase flow, it is single phase flow only.

Therefore, study of two phase flow behavior will be the future scope of work for this project. This will be achieved by increasing heat flux on tubes which in turn make two phases flow.

References

[1] Bose,P.K. “ Cryogenic system”, “Introduction of cryogenic Engineering and gas application” pp.15 -18, (2001)

[2] Welty, R. James, Wicks, E. Charles and Wilson, E. Robert “Convective heat transfer ” “Fundamental of moment, heat and mass transfer”, pp. 312--326, (2000)

[3] Munson, R. Bruch, Young, F. Donald and Okiishi, H. Theodore,“ viscous flow in pipe”,“ Fundamental of fluid mechanics”, pp.460 – 488,(1998)

[4] Chu, H.H.S. , Churchil,S.W.“ Lamina & turbulent flow”, “Correlating equations for laminar & turbulent flow free convection from a vertical plate”, pp. 253-269,(2003) .

[5] Sukhatme,S.P.“ Laminar & turbulent flow”, “Correlation in single phase convection heat transfer”, pp.99-101,(2002)

[6] Tong,L.S. and Tang, Y.S.,“Hydrodynamics of two phase flow”,“Boiling heat transfer and Two phase flow”,pp.168-217,(1997)

[7] Jensen, K. Michael,” correlation for two phase flow”,“ Boiling heat transfer and two phase flow in tubes & tubes bundles”,pp.413-421,(2001)

[8] Yanhui Yuan, Minghan Han, Yi Cheng, Dezheng Wang and Yong Jin,“ Experimental and CFD analysis of two phase cross/countercurrent flow in the packed column with a novel internal”, (2005)

[9] Takao Sano,“ Introduction”,“Transient natural convection between horizontal concentric cylinder”,pp.1-2,(1985)

[10] Kumar, E.Rajendra, Venugopalachary,B and Pathak,H.A.,“ Radiation shields for SST-1” ,“Design of cryostat and radiation shields”,“ Institute for plasma research library reference”,pp.6-8,(1995)

[11] “ Design calculations”,“ LN2 panel cooling scheme – vacuum group”,“Institute for plasma research library reference”, pp.5-9

Appendix

Bench mark problem - A

A.1 Description of the problem

Pressure-driven flow in a straight duct of circular cross-section with radius R. Inlet and outlet boundaries have uniform, dissimilar pressure boundary conditions. The mean fluid temperature at the inlet is $T_{m,i}$. The pipe is subjected to a uniform wall heat flux, q", throughout its length, L.

Fluid Properties	Geometric Properties
Mercury (Hg) at 300K $\rho = 13{,}529\ kg/m^3$ $\mu = 1.523 \times 10^{-3}\ kg/m\text{-}sec$ $k = 8.54\ W/m\text{-}K$ $C_p = 139.3\ J/kg\text{-}K$	$R = 2.5\ mm$ $L = 0.1\ m$ **Loading** $\Delta P = 1.0\ Pa$ $T_{m,i} = 300\ K$ $q'' = 5000\ W/m^2$

A.2 Solution

This problem will be solved by assuming

1. Steady-state flow

2. Incompressible fluid

3. Pipe wall has negligible thermal resistance

4. No body forces

5. Fluid properties are constant

The is solved by following steps.

Step 1. Modeling is done in ansys with given dimensions, as shown in fig.A.1

Step 2. Meshing

Mapped meshing has been applied to the flow. Dense near boundary and coarse at the centre. Also dense at transition section and coarse at outlet section to capture pressure and velocity in those sections where change prominent.

Step 3 Loading

Entry pressure 1 Pa. has been applied on the line at the entry. As flow is laminar the velocity at the boundaries will be zero. Reference pressure at the exit is zero. Temperature and heat flux is applied on the boundary lines.

Step 4 Flotran CFD set up

All properties of the fluid flow have been added. The flow environment has been set.

Step 5 Solution

Finally, Flotran Run has been activated. The solution is produced during flotran Run with calculation of velocities and pressures.

Step 6 Results read and plot.

Through results read, the results have been read as these were available to general post Processors via solution.

Step 7 Plot results

As results have been already read, next step is to see the results by plotting.

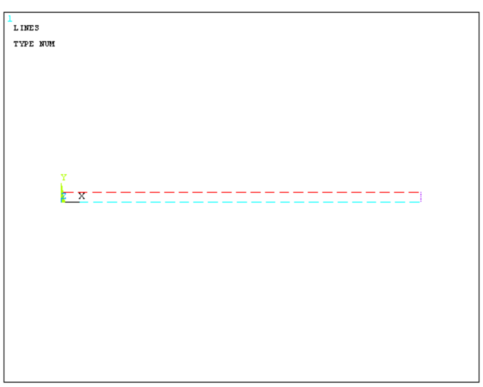


Fig. A.1 Modeling

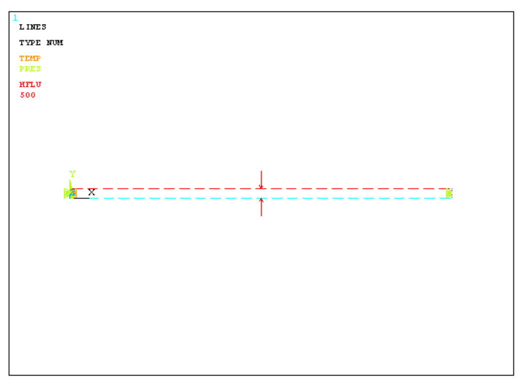


Fig. A.2 Loading

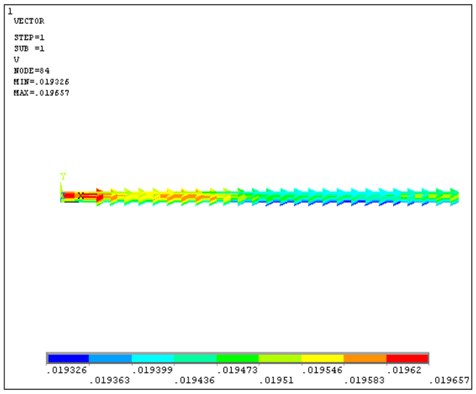


Fig. A.3 Velocity image

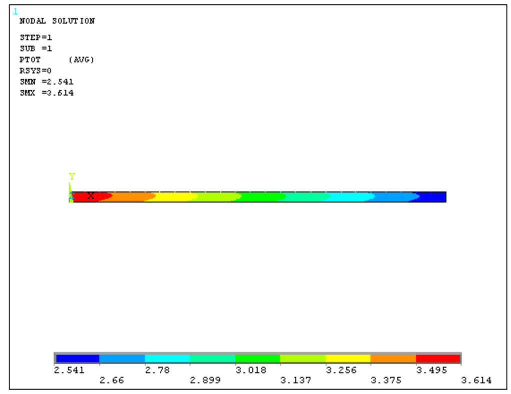


Fig. Pressure image

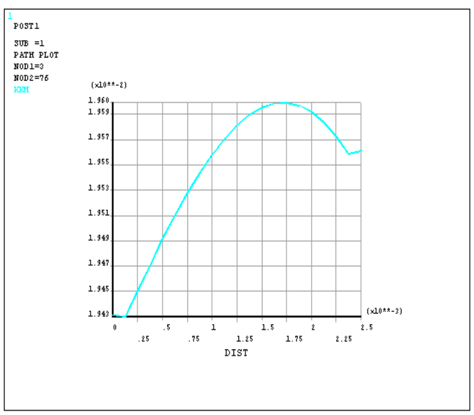


Fig.A.5 velocity profile (entry)

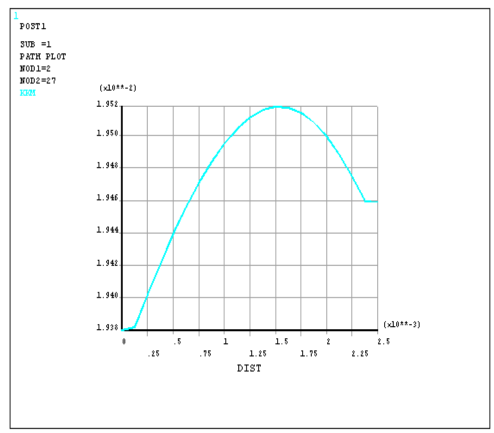


Fig.A.6 velocity profile (exit)

Bench mark problem - B

B.1 Description of the problem

Air is following through a pipe having inlet diameter of 1 cm and length 4 cm and having outlet diameter of 2.5 cm with lengths of 4 & 34 cm. Velocity of air at the inlet would be 1 & 50 m/s. Properties of the air are standard for atmospheric pressure. Determine the pressure variation and velocity profile for both velocity and dimensions.

B.2 Solution: - In first move, outlet region will be taken 4 cm length. After completing this analysis, length will be increased to 34 cm.

First stage

The problem is solved for two dimensional flows.

Element type for the analysis FLUID 141

Step 1. Modeling is done in ansys with given dimensions, as shown in fig.A.1

Step 2. Meshing

Mapped meshing has been applied to the flow. Dense near boundary and coarse at the centre. Also dense at transition section and coarse at outlet section to capture pressure and velocity in those sections where change prominent.

Step 3 Loading

Entry velocity of 1 & 50 m/s in two cases have been applied on the line at the entry. As flow is laminar the velocity at the boundaries will be zero. Reference pressure at the exit is zero.

Step 4 Flotran CFD set up

All properties of the fluid flow have been added. The flow environment has been set.

Step 5 Solution

Finally, Flotran Run has been activated. The solution is produced during Flotran Run with calculation of velocities and pressures.

Step 6 Results read and plot.

Through results read, the results have been read as these were available to general post processors via solution.

Step 7 Plot results

As results have been already read, next step is to see the results by plotting.

a. Initially, velocity image of the flow is produced, as follows for velocities of 1 & 50 m/s. respectively.

b. Pressure image of the flow has been taken after that.

c. Lastly, velocity profile of both velocities has been taken.

d. It is also possible to see animation of flow to clearly understand flow profile

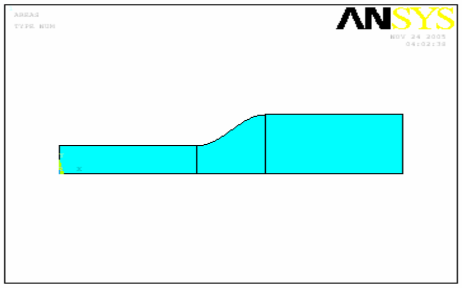


Fig. B.1 Modeling

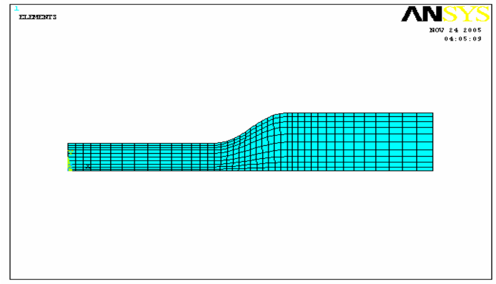


Fig. B.2 Meshing

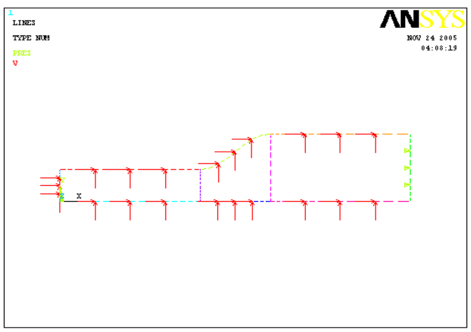


Fig. B.3 Loading

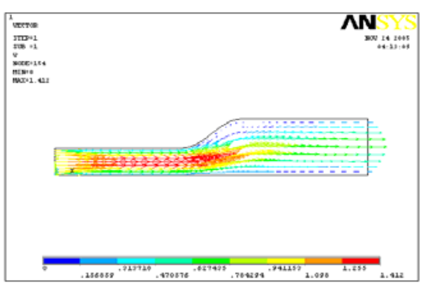


Fig.B.5 velocity image (1 m/s)

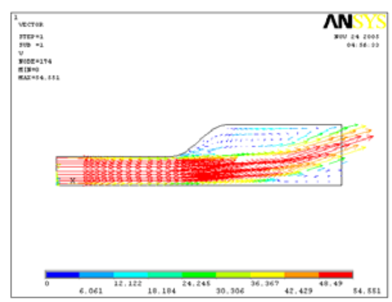


Fig.B.6 velocity image(50 m/s)

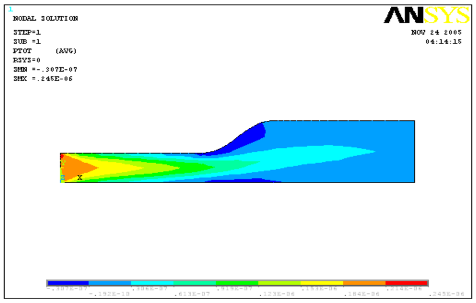


Fig.B.6 Pressure image (1 m/s)

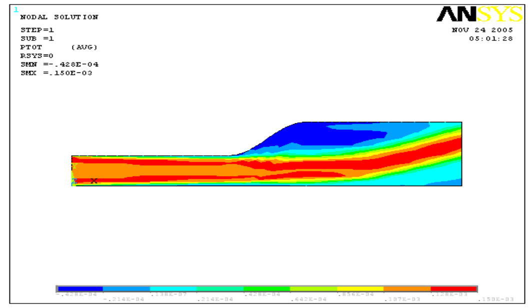


Fig. B.7 Pressure image (50 m/s)

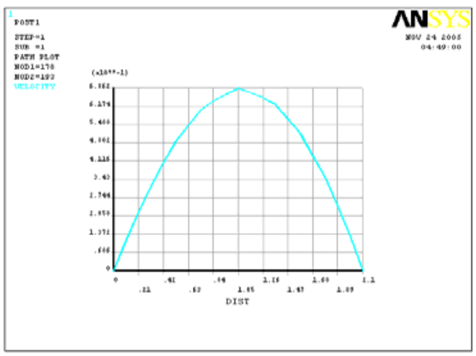


Fig. B.8 velocity profile (1m/s)

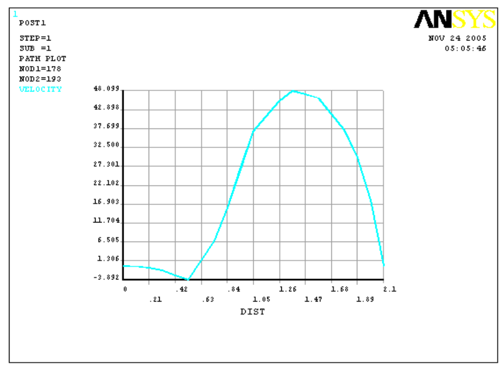


Fig. B.8 velocity profile (1m/s)

Second stage

Modified modeling

Increased outlet length of 30 in is added. Meshed the new pattern is made with dense mesh at outer boundary and coarse at outlet of the flow.

Similarly, above procedure will be followed by adding new length at outlet.

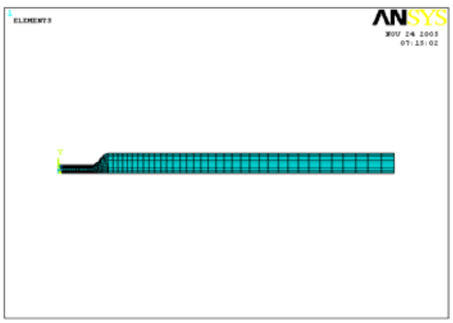


Fig. B.9 Modified meshing

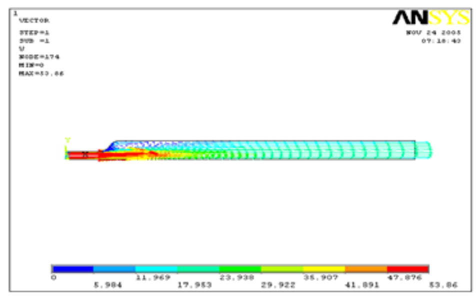


Fig. B.10 velocity image

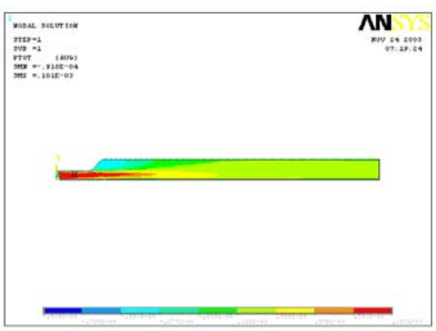


Fig. B.11 pressure image

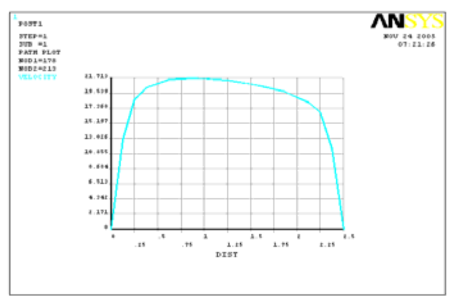


Fig. B.12 velocity profile

Bench mark problem - C

C.1 Description:

Problem B.1 (stage -2) has been applied with heat flux of 35.

C.2 Solution

Similar procedure has been used for modeling, meshing, loading and solution.

C.3 Plot results

As results have been already read, next step is to see the results by plotting.

a. Initially, velocity image of the flow is produced, as follows for velocity 50 m/s.
b. Pressure image of the flow has been taken after that.
c. Lastly, velocity profile velocity has been taken.

It is also possible to see animation of flow to clearly understand flow profile

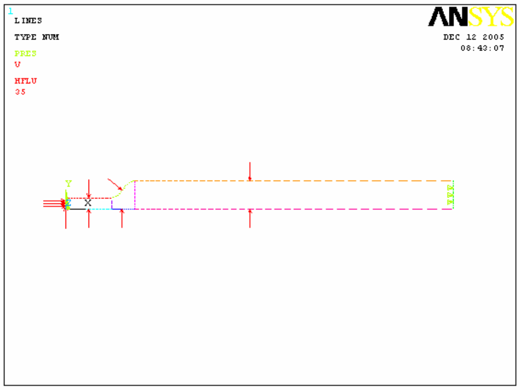


Fig. C.1 loading

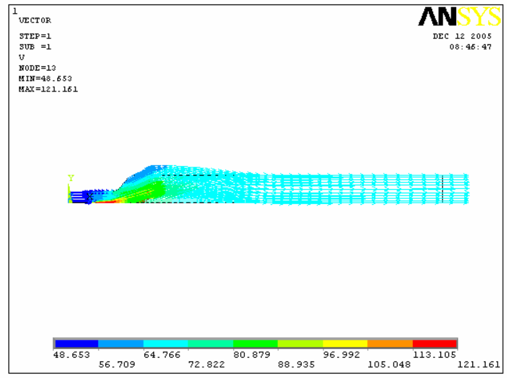


Fig. C.2 velocity image

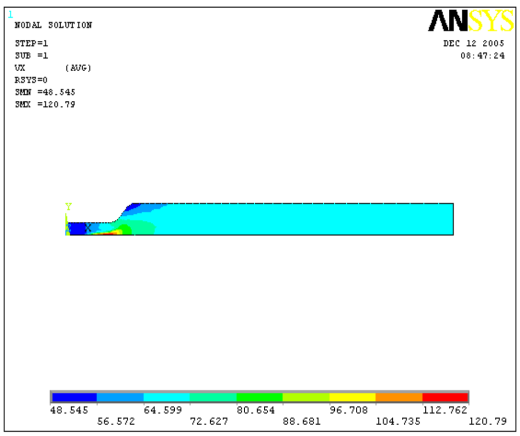


Fig. C.3 nodal plot (velocity)

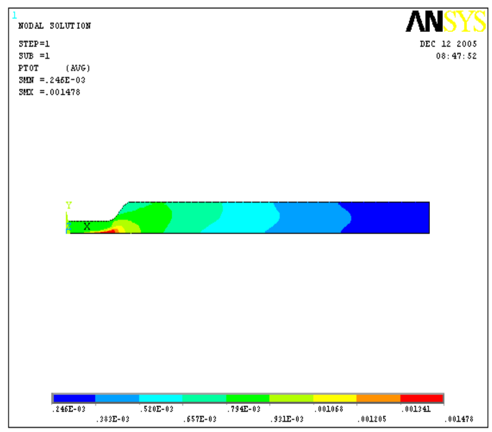


Fig. C.4 pressure image

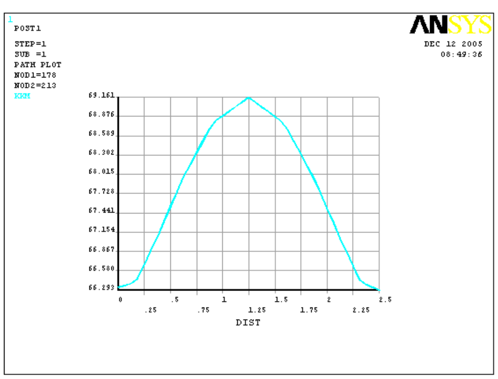


Fig. C.5 velocity profile outlet

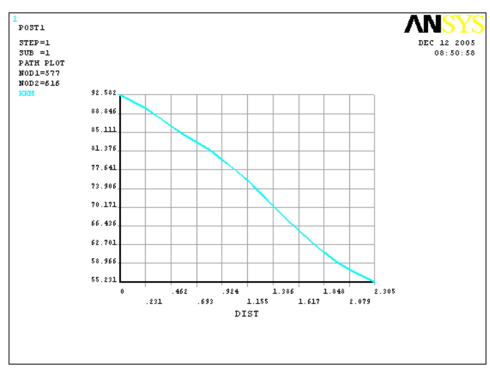


Fig.C.6 velocity profile ventury

Annexure – D

Classification		Elements
Structural Point		MASS21
Structural Line	**2-D**	LINK1
	3-D	LINK8 , LINK10, LINK11, LINK180
Structural Beam	**2-D**	BEAM3, BEAM23, BEAM54
	3-D	BEAM4, BEAM24, BEAM44, BEAM188, BEAM189
Structural Solid	**2-D**	PLANE2, PLANE25, PLANE42, PLANE82, PLANE83, PLANE145, PLANE146, PLANE182, PLANE183
	3-D	SOLID45, SOLID64, SOLID65, SOLID92, SOLID95, SOLID147, SOLID148, SOLID185, SOLID186, SOLID187
Structural Shell	**2-D**	SHELL51, SHELL61, SHELL208, SHELL209
	3-D	SHELL28, SHELL41, SHELL43, SHELL63, SHELL93, SHELL143, SHELL150, SHELL181
Structural Solid Shell	**3-D**	SOLSH190
Structural Pipe		PIPE16, PIPE17, PIPE18, PIPE20, PIPE59, PIPE60
Structural Interface		INTER192, INTER193, INTER194, INTER195, INTER202, INTER203, INTER204, INTER205
Structural Multipoint Constraint Elements		MPC184
Structural Layered Composite		SOLID46, SHELL91, SHELL99, SOLID186 Layered Solid, SOLSH190, SOLID191
Explicit Dynamics		LINK160, BEAM161, PLANE162, SHELL163, SOLID164, COMBI165, MASS166, LINK167, SOLID168
Visco Solid		VISCO88, VISCO89, VISCO106, VISCO107, VISCO108
Thermal Point		MASS71
Thermal Line		LINK31, LINK32, LINK33, LINK34
Thermal Solid	**2-D**	PLANE35, PLANE55, PLANE75, PLANE77, PLANE78
	3-D	SOLID70, SOLID87, SOLID90
Thermal Shell		SHELL57, SHELL131, SHELL132
Thermal Electric		PLANE67, LINK68, SOLID69, SHELL157
Fluid		FLUID29, FLUID30, FLUID38, FLUID79, FLUID80, FLUID81, FLUID116, FLUID129, FLUID130, FLUID136, FLUID138, FLUID139, FLUID141, FLUID142

Classification		Elements
Electric Circuit		SOURC36, CIRCU94, CIRCU124, CIRCU125
Electromechanical		TRANS109, TRANS126
Coupled-Field		SOLID5, PLANE13, SOLID62, SOLID98, ROM144, PLANE223, SOLID226, SOLID227
Contact		CONTAC12, CONTAC52, TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175, CONTA176, CONTA178
Combination		COMBIN7, COMBIN14, COMBIN37, COMBIN39, COMBIN40, PRETS179
Matrix		MATRIX27, MATRIX50
Infinite		INFIN9, INFIN47, INFIN110, INFIN111
Surface		SURF151, SURF152, SURF153, SURF154, SURF156, SURF251, SURF252
Follower Load		FOLLW201
Meshing		MESH200
Magnetic Electric		PLANE53, SOLID96, SOLID97, INTER115, SOLID117, HF118, HF119, HF120, PLANE121, SOLID122, SOLID123, SOLID127, SOLID128, PLANE230, SOLID231. SOLID232