COMPUTATIONAL METHODS LAB MANUAL

Regulation_MR18 (2018-19 Onwards)

Course: M.Tech

Specialization: Thermal Engineering

Semester: II

LAB Code: 83108

Credits: 1.5

MALLA REDDY ENGINEERING COLLEGE
(Autonomous)

Maisammaguda, Dhulapally (post, Via Kompally),
Secunderabad – 500100

COMPUTATIONAL METHODS LAB

LIST OF EXPERIMENTS

S.No	Name of The Experiment	Page No
1.	Write a program to solve one dimensional steady state heat conduction boundary value problem.	3 - 6
2.	Write a program to solve one dimensional transient heat conduction equation using explicit finite difference method.	7- 9
3.	Write a program to solve one dimensional transient heat conduction equation using explicit finite difference method.	10-13
4.	Write a program to solve one dimensional transient heat conduction equation using implicit finite difference method.	14-16
5.	Write a program to solve two dimensional steady state heat conduction equation using finite difference method.	17-19
6.	Write a program to solve one dimensional linear wave equation using finite difference method and also analyze the dispersion and dissipation error.	20-23
7.	Determine the nodal temperature of any one composite wall using simulation Software.	24-26
8.	Transient thermal analysis of a steel ball using CFD software.	27-29
9.	CFD analysis of flow through a diverging section using CFD software.	30-33
10.	Analysis of flow through converging section using CFD software.	34-36
11.	Analysis of convection heat transfer of a bar in air using CFD software.	37-39
12.	Determine the flow properties after mixing of two fluids in a duct using CFD Software.	40-42

(Signature of the Faculty In Charge) **(Signature of the HOD)**

EXPERIMENT NO-I

NAME OF THE EXPERIMENT: Write a program to solve one dimensional steady state heat conduction boundary value problem.

AIM: Solution for the one dimensional steady state heat conduction equation using finite difference explicit method using **MATLAB.**

The one dimensional steady heat conduction equation is given by

$$\frac{\partial T}{\partial x}=0 \qquad \text{_____ (1)}$$

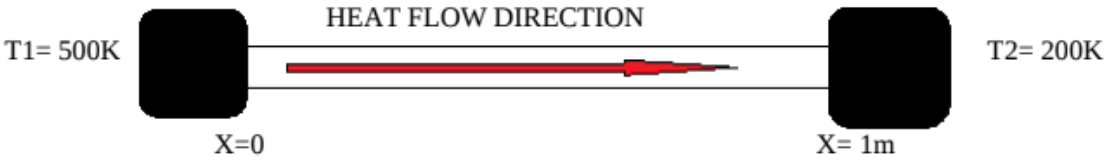
This equation represents the second order derivatives of temperature 'T' with respect to space 'x' for a spacial domain of 0 to L. The boundary condition is given by;

T(x=0) = T1;
T(x=L) = T2;

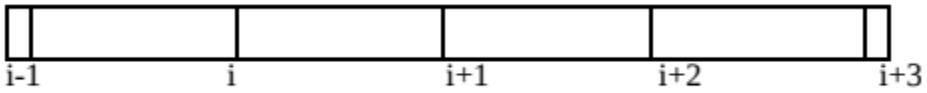
Numerical Solution:-

A. Preprocessor:-

1. **Geometry Creation:-**



2. **Meshing:-**



3. Setup:-

Applying Boundary Condition:

Ti-1 = 500K;
Ti+3 = 200K;

B. Solver :-

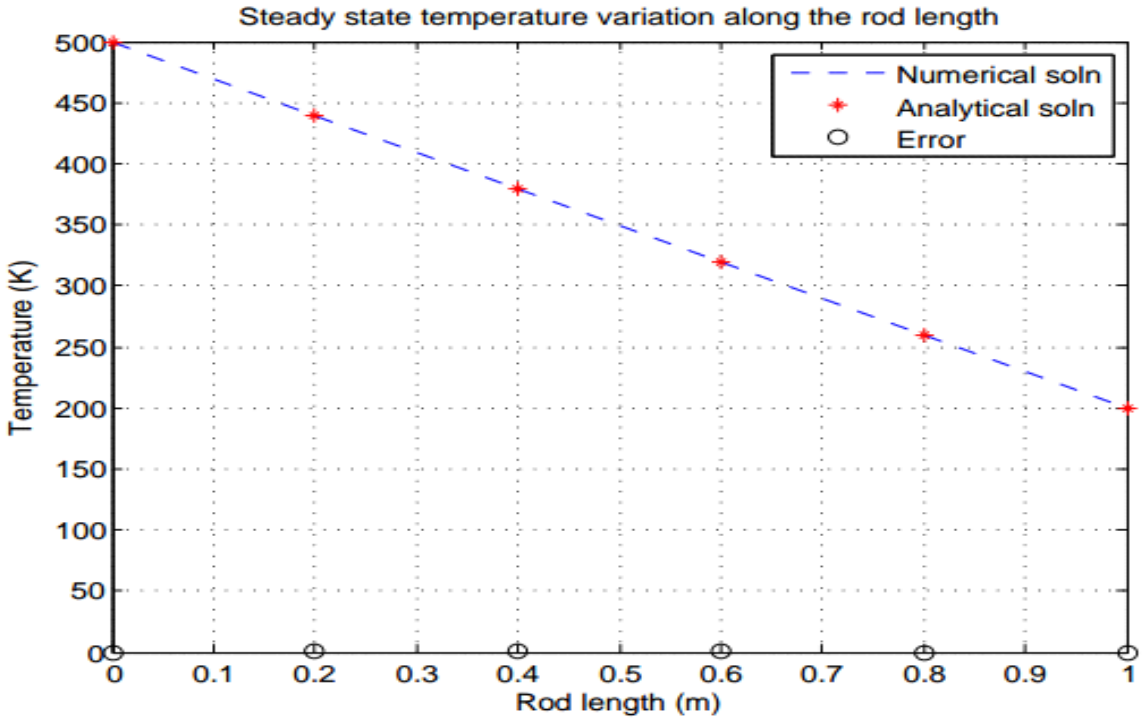
The complate MATLAB code is given below to solve this problem

```
%%% the solution of steady state one dimensional heat conduction eq%%%%
clc
clear all

%%%%%%%%%%%%%%%%%%%%%%%%% PRE-PROCESSOR
%%%%%%%%%%%%%%%%%%%%%%%%% %%%%%%

n=4; % number of cells are(n+1)& number of grids are (n+2). it is number of unknown grids
m=n; % m shows the number of rows
C=zeros(m,n); % the coefficient matrix
L=1; %length of the rod in metre
t0=500; % boundary condition at left end of the rod
t5=200; % % boundary condition at right of the rod
T=zeros(m,1); % the unknown matrix
S= zeros(m,1); % the known matrix
S(1,1)= -500;
S(m,1)= -200;
for i=1:n
   C(i,i)=-2;
end
for i=1:n-1
   C(i,i+1)=1;
   C(i+1,i)=1;
end

 T= inv(C)*S;
T1=zeros(m+2,1) %full temperature solution matrix
T1(1,1)= 500;    %  LEFT BC
T1(m+2,1)= 200;   % RIGHT BC
X=zeros(m+2,1) %rod length descretised matrix
X(1,1)=0;
X(m+2,1)=1;
%%%%%%%%%%%%%%%%%%%%%%%%% SOLVER
%%%%%%%%%%%%%%%%%%%%%%%%%%%
for i=2:(m+1)
   T1(i,1)= T(i-1,1);
   X(i,1)= X(i-1,1)+1/(n+1);
end
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%% POST-PROCESSOR
%%%%%%%%%%%%%%%%
plot(X,T1,'ro')
```

```
grid on
T2= zeros(m+2,1); %analytical solution of the problem
err= zeros(m+2,1); % difference b/w analytical solution and numerical solution
for i= 1:m+2
   T2(i,1)= (T1(m+2,1)-T1(1,1))/L*(X(i,1))+T1(1,1);

      err(i,1)= T2(i,1)- T1(i,1);
end

plot(X,T1,'b--') % variation of temperature along x axis
title(' Steady state temperature variation along the rod length ')
xlabel('Rod length (m)')
ylabel('Temperature (K)')
grid on
 hold on
 plot(X,T2,'r*')
 hold on
 plot(X,err,'Ko')
 hold off
 legend('Numerical soln','Analytical soln','Error')
```

C. Post- Propecessor:

The following graph shows the simulation result.


Steady state temperature variation along the rod length
Numerical soln
Analytical soln
Error
Temperature (K)
Rod length (m)
500
450
400
350
300
250
200
150
100
50
0
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1

END*

EXPERIMENT NO-II

NAME OF THE EXPERIMENT: Write a program to solve one dimensional transient heat conduction equation using explicit finite difference method.

AIM: - solution of one dimensional transient heat conduction through a bar using finite difference explicit method.

The one dimensional transient heat conduction equation is given as,

$$\frac{\partial^2 T}{\partial x^2} = \frac{1}{\alpha}\frac{\partial T}{\partial t} \qquad \text{.....(1)}$$

where α is the thermal diffusivity.

Test Case for the Numerical Solution :-

An Aluminium bar has an initial temperature of 600˚C and kept in an oven. To make it cool, it has taken from the oven and kept with it's ends on two heat sources. The temperature of the left end heat source is 200˚C and right end heat source is 400˚C. Find the temperature distribution throughout the bar with an time interval of 0.2s. Take the following condition for computation,

d= 0.5 → Diffusion coefficient

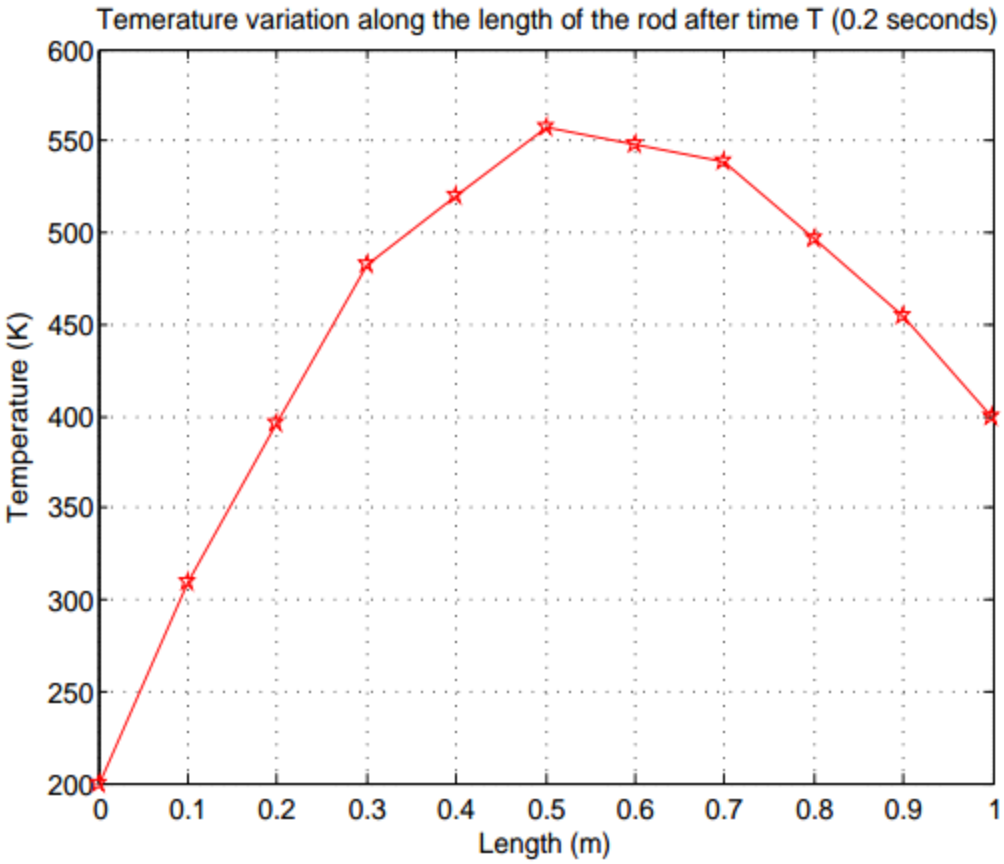
α = 0.0029 m^2/s

The full MATLAB given below gives the solution for this problem :-

```
%%%% Transient heat conduction through a bar using Explicit Scheme %%%%%
clc
clear all

%%%%%%%%%%%%%%%%%%%%%%%%%%%% PRE-PROCESSOR
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

d=0.5; % diffusion coefficient
T=0.2;  % the time in the second
L=1;   % the length of the bar
delX=0.1; % the length of the grid in the spacial direction
alpha=0.2; % the the thermal diffusivity
```

```
n = L/(delX)+1; % total number of grids in the spatial direction
delT = d*(delX)^2/(alpha); % the time interval
m = ceil(T/(delT)+1); % the number of grid points in the time direction
t= zeros(m,n);        % the solution Temperature matrix
X = zeros(1,n);        % the spacial grid matrix
for i=2:n
   X(1,i)= X(1,i-1)+ delX;  % entering the elements of the spacial grid matrix
end
%%%%%%%%%%%%%%%%%%%%%%%% SOLVER
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
for i= 1:m
   t(i,1)=200;  % Specifying the left end boundary condition
      t(i,n)=400; % Specifying the right end boundary condition
  for k = 1:m-1
    for j=2:n-1
      t(1,j)= 600; % specifying the initial condition of the bar
       t(k+1,j) = d*t(k,j-1)+(1-2*d)*t(k,j)+d*t(k,j+1); % Appltying the FTCS Scheme to solve the
tempersture at all time iterval at all the nodal points
    end
  end
end

%%%%%%%%%%%%%%%%%%%
```

Result :-


Temerature variation along the length of the rod after time T (0.2 seconds)
Temperature (K)
600
550
500
450
400
350
300
250
200
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1
Length (m)

END*

EXPERIMENT NO-III

NAME OF THE EXPERIMENT: Write a program to solve one dimensional transient heat conduction equation using explicit finite difference method.

AIM: - Solution of one dimensional transient heat conduction through a bar using finite element method.

The one dimensional transient heat conduction equation is given as,

$$\frac{\partial^2 T}{\partial x^2} = \frac{1}{\alpha}\frac{\partial T}{\partial t} \qquad \text{……..(1)}$$

where α is the thermal diffusivity.

Test Case for the Numerical Solution :-

An Aluminium bar has an initial temperature of 600˚C and kept in an oven. To make it cool, it has taken from the oven and kept with it's ends on two heat sources. The temperature of the left end heat source is 200˚C and right end heat source is 400˚C. Find the temperature distribution throughout the bar with an time interval of 0.01s. Take the following condition for computation,

$$d = 0.5 \quad \rightarrow \text{ Diffusion coeffient}$$

$$\alpha = 0.0029$$

The full MATLAB given below gives the solution for this problem :-

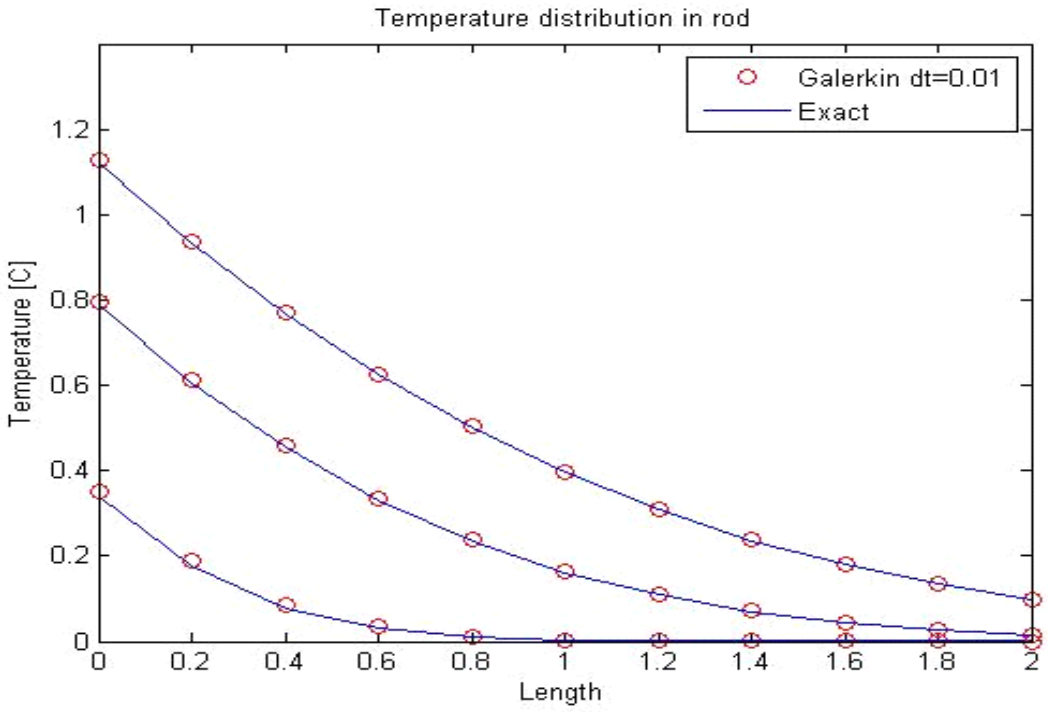
```
%%% 1D Heat conduction
%%% Raushan Kumar
%%%[1] Strang G. and Fix G. (2008): An analysis of the Finite Element Method,Second Edition,
Wellesley-Cambridge Press, Wellesley USA

%%%--- Problem ---%%%
% Solving 1-D Heat diffusion in a unit rod
%
clear all
clc

%%%--- Mesh generation ---%%%
h=0.2;                  % Distance between nodes [m]
units=20;                % Length of rod
```

```
p=[0:h:units]';            % Nr of nodes
x = [0:h:units];           % Spatial discretisation
nr_nodes = length(p);      % Nr of nodes

 for i = 2:nr_nodes        % Creates the element matrix
    e(i-1,:) = [i-1 i];
 end

%%%--- Creating space ---%%%
nr_elements =length(e);    % Nr of elements

K=zeros(nr_nodes,nr_nodes); % Empty stiffness matrix
M=zeros(nr_nodes,nr_nodes); % Empty mass matrix
F=zeros(nr_nodes,1);       % Empty forcing vector
Q=zeros(nr_nodes,1);       % Initial forcing vector,

%%%--- Parameters ---%%%
P=1;                       % Heat conductivity
pc=1;                      % Heat capacity
A=1;                       % Element cross section

%%%--- Assembling matrices ---%%%
for k = 1:nr_elements                  % Loop over all elements

      nodes=e(k,:);                    % Identify nodes
      h=p(k+1)-p(k);                   % Calculate distance between nodes

      Ke = ((A*P)/h)*[1 -1;-1 1];      % Stiffness entry
      Me = ((pc*A*h)/6)*[2 1;1 2];     % Mass entry
      Fe = ((A*h)/6)*[2 1;1 2]*Q(nodes,1); % Forcing entry

      K(nodes,nodes)=K(nodes,nodes)+Ke;    % Position stiffness entry at identified nodes
      M(nodes,nodes)=M(nodes,nodes)+Me;    % Position mass entry at identified nodes
      F(nodes,1)=F(nodes,1)+Fe;            % Position forcing entry at identified nodes
end


%%%--- Initial conditions ---%%%
Told=zeros(nr_nodes,1);    % Initial temperature, 0 everywhere
F(1)=1;                    % With unit heat production at node one

%%%--- Time parameters ---%%%
dt = 0.01;                 % Size of time step
t=1;                       % Time

%%%--- Preparing matrices ---%%%
% See [2] page 108

A=((1/2)*K+(1/(dt))*M);
B=(-(1/2)*K+M/(dt));

C=(M+0.5*dt*K);
D=(M-0.5*dt*K);
```

```
R=((1/3)*K+(1/(2*dt))*M);
S=((-1/6)*K+(M/(2*dt)));

R(nr_nodes,:)=0;
R(:,nr_nodes)=0;
R(nr_nodes,nr_nodes)=1;

S(nr_nodes,:)=0;
S(:,nr_nodes)=0;
S(nr_nodes,nr_nodes)=1;

Tnew = Told;
Fold = F;
Fnew = Fold;

%%%--- Time stepping ---%%%
for i = 1:t/dt

    thermometer(i)=Tnew(1);                  % Vector to store temperature data at node 1

  %  Tnew = C\(D*Told+dt*F) ;                 % Crank Nicolson
   Tnew=R\((S*Told+((1/6)*(Fold+2*Fnew))));    % Galerking

   Told=Tnew;

end


%%%--- Analytical solution ---%%%
tt=[0:dt:t-dt];         % Time discretisation

for nt = 1:length(tt)   % Loop over time

% Analytical solution
Tana = 2*(tt(nt)/pi)^(1/2)*((exp((-x.^2./(4*tt(nt))))-0.5.*x.*(sqrt(pi/tt(nt)).*erfc((x./(2*sqrt(tt(nt))))))));

thermometerTana(nt)=Tana(1);    % Store temperature at node 1
end

  figure(1) % [2] page 109
  plot(x,Tnew,'ro',x,Tana,'b')
  title('Temperature distribution in rod')
  xlabel('Length')
  ylabel('Temperature [C]')
  legend(['Galerkin dt=',num2str(dt)],'Exact')
  axis([0 2 0 1.4])
%
  figure(2) % [2] page 110
  plot(tt,thermometerTana,tt,thermometer,'r')
  title('Temperature at point x=0 (Heating Curve)')
  xlabel('Time [s]')
  ylabel('Temperature [C]')
```

```
  legend('Exact',['Galerkin dt=', num2str(dt)])
%

%  Tnew1=Tnew;
%  Tana1=Tana;

%   Tnew05=Tnew;
%   Tana05=Tana;
% %
%   Tnew01=Tnew;
%   Tana01=Tana;
%
%
%   figure(1)
%   plot(x,Tnew,'ro',x,Tana,'b',x,Tnew1,'ro',x,Tana1,'b',x,Tnew05,'ro',x,Tana05,'b')
%   title('Temperature distribution in rod')
%   xlabel('Length')
%   ylabel('Temperature [C]')
%   legend('Galerkin dt=0.01','Exact')
%   axis([0 2 0 1.4])
```

RESULT:-



*****END*****

EXPERIMENT NO-IV

NAME OF THE EXPERIMENT: Write a program to solve one dimensional transient heat conduction equation using implicit finite difference method.

AIM:- solution of one dimensional transient heat conduction through a bar using finite difference implicit method.

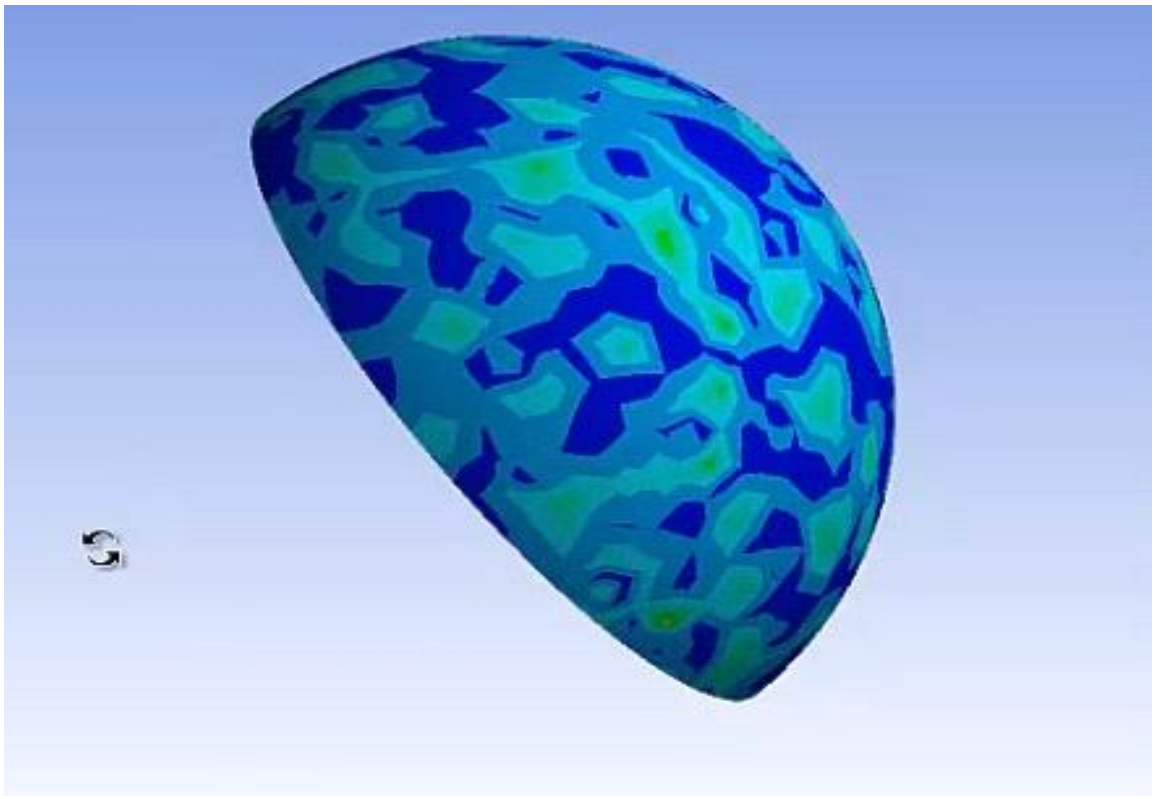
The one dimensional transient heat conduction equation is given as,

…….(1)

where α is the thermal diffusivity.

Test Case for the Numerical Solution :-

An Aluminium bar has an initial temperature of 600˚C and kept in an oven. To make it cool, it has taken from the oven and kept with it's ends on two heat sources. The temperature of the left end heat source is 200˚C and right end heat source is 400˚C. Find the temperature distribution throughout the bar after an time inte



END

EXPERIMENT NO-IX

NAME OF THE EXPERIMENT: CFD analysis of flow through a diverging section using CFD software.

Aim: To simulate flow through diverging channel.

Problem Description: consider water is flowing through a symmetrical diverging section having inlet area 314.15926 mm□, outlet area 706.858347 mm□, length of channel is 50 mm and inlet velocity 5m/s and pressure 3 X 10^5 gauge discharging into the atmospheric condition. Assuming steady state condition & calculate the velocity variation inside the function numerically using software package Ansys CFX. The other conditions are given below:

Atmosphere pressure =101.525 Pa

Water density =1000 kg/m^3

Atmospheric temp = 288.16K

Governing equation required:

To solve this problem the following equation required: Continuity equation

Steps involved in fluid flow (CFX):

1. Creation of geometry:

a) fluid flow (CFX) → Design model

b) Geometry → modeling → x-y plane → sketching →draw→polyline →draw diverging chaneel → specify dimensions.

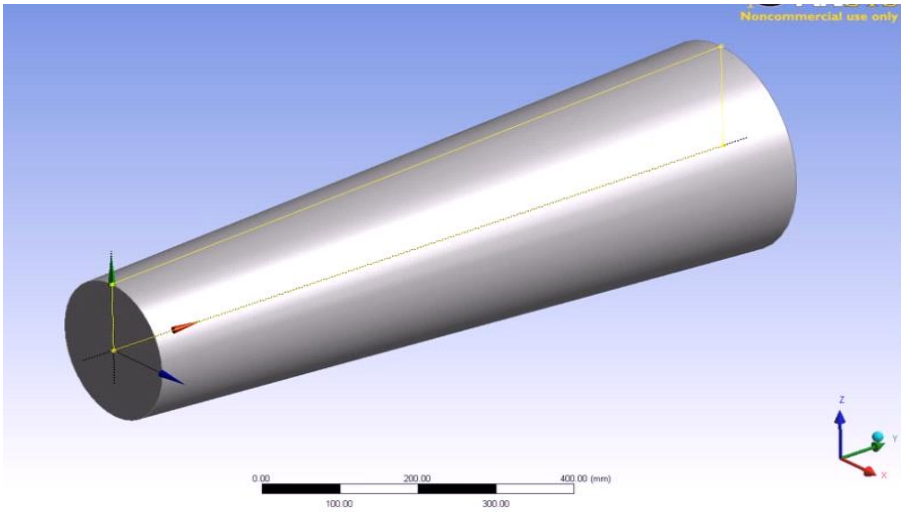
c) specify the correct dimension and completer 2D-geometry.

d) Go to revolve → select geometry → select axis → generate.

e) 3-D geometry completed.

f) Create a folder with your name→save the file in the new folder in your named folder→named it as exp name.

g) print review and save the picture in the project folder.

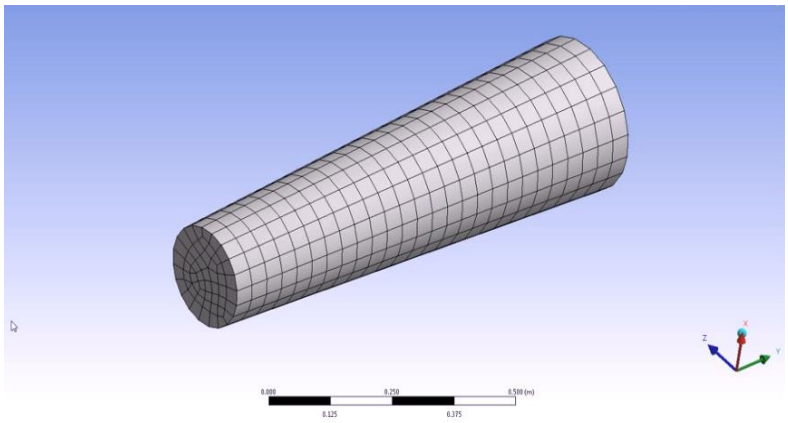


2. Creation of mesh:

- Fluid flow (CFX) → meshing

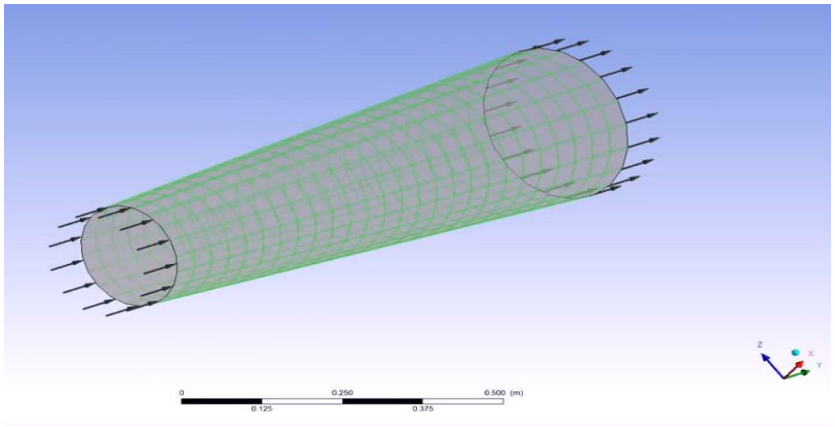
- Mesh → Detail of mesh → Sizing→ Use advance size function → select off → create mesh→ now the software will create mesh→Sizing →use advanced size function→one proximity and curvature→ Update mesh

- Go to print preview and save the mesh geometry.



3. Creation of setup:

- Fluid flow (CFX) → CFX –Pre.
- Zoom your Geometry
- Default domain → (right click) rename →pipe
- left click on pipe → fluid model →material library → under material choose water→Ok
- Select pipe→(right click) Insert →Insert boundary → named it as Inlet →boundary inlet→ (Basic setting) location → Choose appropriate inlet boundary in your geometry→ boundary detail →normal speed → specify normal speed (as per given in the problem) →Ok
- Select pipe→(right click) Insert →Insert boundary → named it as outlet →boundary outlet→ (Basic setting) location → Choose appropriate inlet boundary in your geometry→ boundary detail →relative pressure→ specify relative pressure (as per given in the problem it is zero) →Ok
- Select pipe→pipe default→named it as pipe wall (right click and rename) →(left click) boundary pipe wall→basic setting →specify wall location→boundary detail→specify no slip boundary condition (applicable for viscous flow)→Ok
- Save the file in the same folder.
- Save the picture in the same folder.

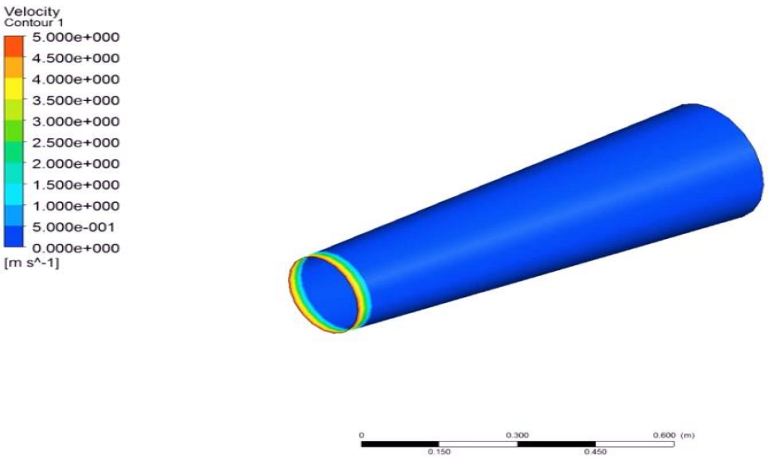


Solution:-

- Fluid flow (CFX) → CFX- solver manager.
- Define run →run difinition → Using double precession run the simulation →Start run.
- Check the workspace if solution diverged then go for result & if not move to back & look your grid → if required changed it →repeat the above steps from meshing onwards.

5. Result:-

- Fluid flow (CFX) → cfd-post.
- Select the contour and analyze the result for full domains.
- Outline →pipe→tick mark on inlet, outlet and pipe wall →user location and plots→wire frame→ default legend view→ click contour →details of contour 1→ domains → pipe→location inlet →variables →pressure, velocity →apply → change other locations → see the flow variable variations →analyze your result.
- Save file in the folder.
- Save the picture in the same folder.
- Take print of report views.
- Discuss your result.



END

EXPERIMENT NO-X

NAME OF THE EXPERIMENT: Analysis of flow through converging section using CFD software.

Aim: To simulate flow through Converging channel.

Problem Description: consider water is flowing through a symmetrical diverging section having inlet area 706.858347 mm^2, outlet area 314.15926 mm^2, length of channel is 50 mm and inlet velocity 5m/s and pressure 3 X 10^5 gauge discharging into the atmospheric condition. Assuming steady state condition & calculate the velocity variation inside the function numerically using software package Ansys CFX. The other conditions are given below:

Atmosphere pressure =101.525 Pa

Water density =1000 kg/m^3

Atmospheric temp = 288.16K

Governing equation required:

To solve this problem the following equation required : Continuity equation

Steps involved in fluid flow(CFX):

1.Creation of geometry:

a) fluid flow (CFX) → Design model

b) Geometry → modeling → x-y plane → sketching →draw →polyline→ draw converging geometry→specify correct dimensions.

c) specify the correct dimension and completer 2D-geometry.

d) Go to revolve → select geometry → select axis → generate.

e) 3-D geometry completed.

f) Create a folder with your name→save the file in the new folder in your named folder→named it as exp name.

g) print review and save the picture in the project folder.

2. Creation of mesh:

- Fluid flow (CFX) → meshing
- Mesh → Detail of mesh → Sizing→ Use advance size function → select off → create mesh→ now the software will create mesh→Sizing →use advanced size function→one proximity and curvature→ Update mesh
- Go to print preview and save the mesh geometry.

3.creation of setup:

- Fluid flow (CFX) → CFX –Pre.
- Zoom your Geometry
- Default domain → (right click) rename →pipe
- left click on pipe → fluid model →material library → under material choose water→Ok
- Select pipe→(right click) Insert →Insert boundary → named it as Inlet →boundary inlet→ (Basic setting) location → Choose appropriate inlet boundary in your geometry→ boundary detail →normal speed → specify normal speed (as per given in the problem) →Ok
- Select pipe→(right click) Insert →Insert boundary → named it as outlet →boundary outlet→ (Basic setting) location → Choose appropriate inlet boundary in your geometry→ boundary detail →relative pressure→ specify relative pressure (as per given in the problem it is zero) →Ok
- Save the file in the same folder.
- Save the picture in the same folder.

4. Solution:-

a) Fluid flow (CFX) → CFX- solver manager.

b) Define run →run difinition → Using double precession run the simulation →Start run.

Check the workspace if solution diverged then go for result & if not move to back & look your grid → if required changed it →repeat the above steps from meshing onwards.

5. Result

a) Fluid flow (CFX) → cfd-post.

b) Select the contour and analyze the result for full domains.

c) Outline →pipe→tick mark on inlet, outlet and pipe wall →user location and plots→wire frame→ default legend view→ click contour →details of contour 1→ domains → pipe→location inlet →variables →pressure, velocity →apply → change other locations → see the flow variable variations →analyze your result.

d) Save file in the folder.

e) Save the picture in the same folder.

f) Take print of report views.

g) Discuss your result.

END*

EXPERIMENT NO-XI

NAME OF THE EXPERIMENT: Analysis of convection heat transfer of a bar in air using CFD software.

Aim:- Compute the transient temperature distribution throughout the bar.

Problem Statement:-

A cylindrical steel bar having an initial temeperature of -50˚C, surrounded by atmospheric air at 20˚C. Find the time required to attain the temperature of the bar equal to the surrounding temperature, due to natural convection.

Govering Equation:-

1. Energy Equation (Heat convection equation)

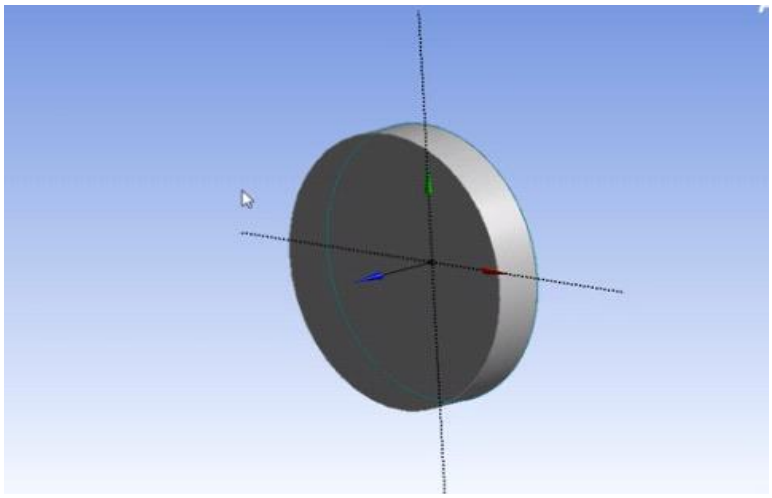
Solution Using ANSYS WORKBENCH Transient Thermal Analysis System

1. **First put the appropriate engineering data for the prolem statement**

Go to the workbench →Analysis System → Transient Thermal → Engineering data → (Click over engineering data) → Engineering data source →select structural steel → check the property of structural steel → close it and return to the project schematic →start with geometry.

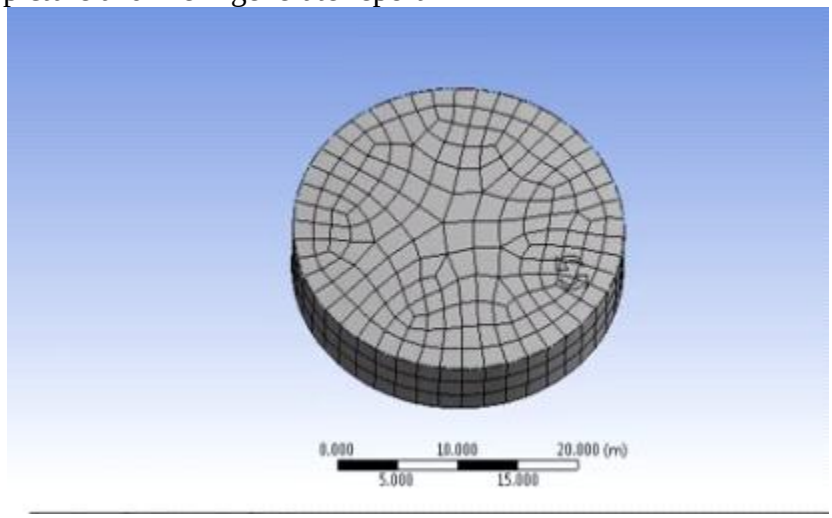
2. **Geometry:-**

Design Modular →choose Xyplane→view→sketching→Draw→circle→specify it's dimension(1m)→ extrude →geomerty →apply FD1, depth(>0)→ 5 m→
Generate → save the file in the folder → save the picture in the folder

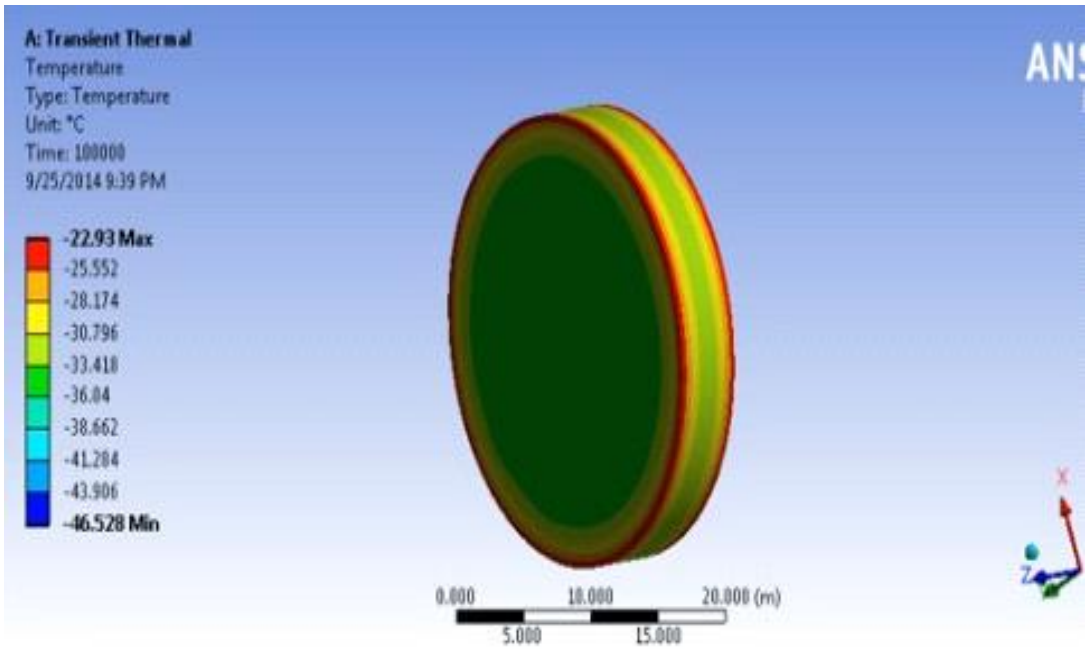


3. **Model**

mesh→ update → transient thermal→ insert →connection →geomerty →seect geometry all face→apply → geometry→ apply→ details of connection→definition →film coefficient →left click→import temperature dependent→ sta gnent air (simplified case)→ ok→ change film coefficient →10W/m²k (step applied)→ ambient temp→ 295.15K →outline →initial tamp→ initial temp value -50°C →analysis setting→ step controls→ number of steps→ 30 →solution (A6)→ insert →thermal →temperature →select hole volume of the bar →geomerty →apply→ solution(A6)→ temperature→ after solver finishes it's run temperature plot you will get. →save the
picture and file →generate report



4. **Result** :-
Analyse your result.


A: Transient Thermal
Temperature
Type: Temperature
Unit: °C
Time: 100000
9/25/2014 9:39 PM
-22.93 Max
-25.552
-28.174
-30.796
-33.418
-36.04
-38.662
-41.284
-43.906
-46.528 Min
0.000
10.000
20.000 (m)
5.000
15.000

END*

EXPERIMENT NO-XII

NAME OF THE EXPERIMENT: Determine the flow properties after mixing of two fluids in a duct using CFD Software.
AIM:- comparison of cfd simulation of hot and cold fluid mixing in t-pipe by placing nozzle at different places.

As the hot water and cold water enters at different velocities of mixed flow will be increased as pressure decreases. Pressure for mixed flow at the outlet will be reduced because of the cross section variation at the joining point of hot and cold water and also high velocities will cause more pressure drop. Turbulence is created at the inter section point of the T-pipe because the cross section at that point has changed suddenly and also the velocity at that point has increased suddenly due to the mixing of hot and cold water at that point.

Test Case for the Numerical Solution :-

The geometry was done in the GAMBIT with measurements; pipe diameter is 50mm, radius of the pipe 25mm and length of the pipe 500mm. Defining required boundaries like inlet, outlet and wall of the geometry and mesh under tetrahedron.

Defining the boundary conditions for the water. The velocity at inlet of the cold fluid is 4m/sec and at outlet is 2m/sec and the gravitational acceleration of 9.81 m/s2 in downward flow direction was used.

Governing Equation Required:-

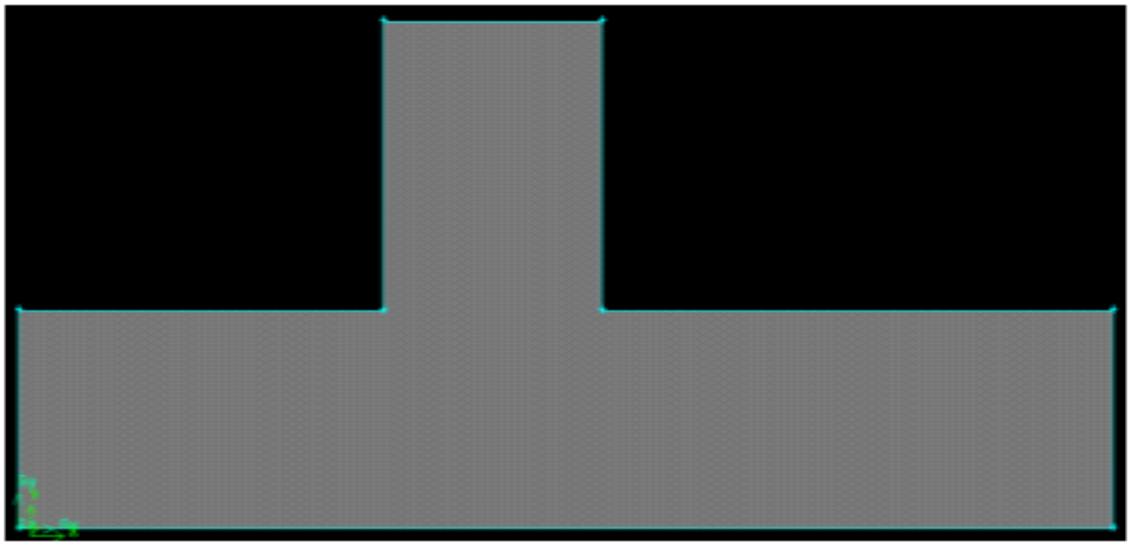
- Continuity Equations
- Momentum Equations
- Bernoullis Equations

Solution Strategy:-

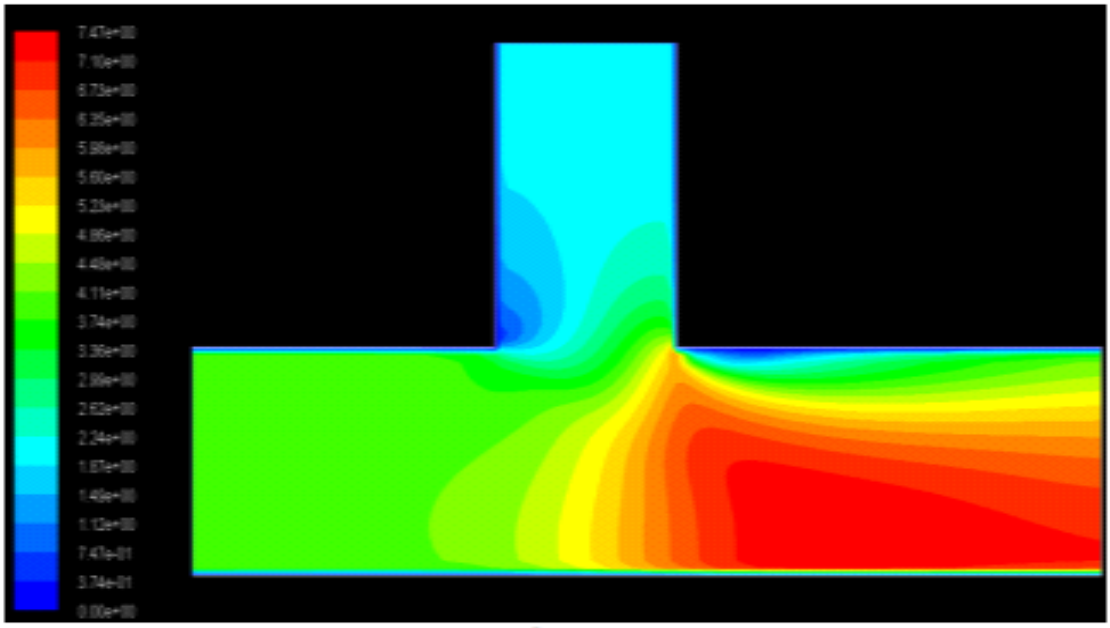
The simulation is done in the FLUENT based upon the governing equations.

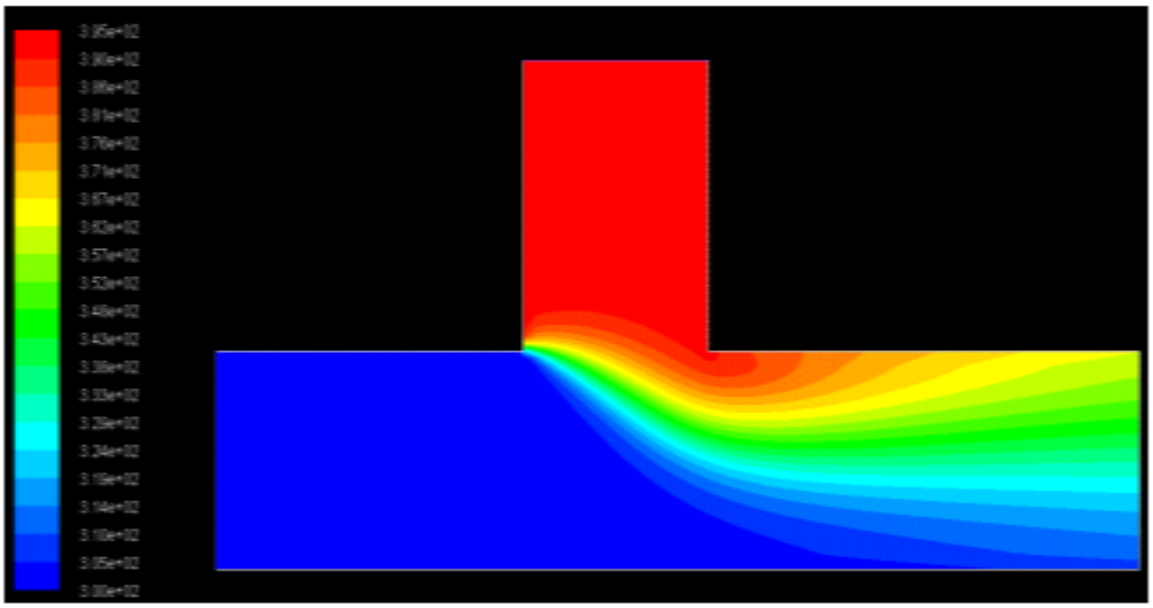
The steps followed in the fluent are define Model, define Material, define cell zone, boundary condition, solve, iterate, and analyze results.

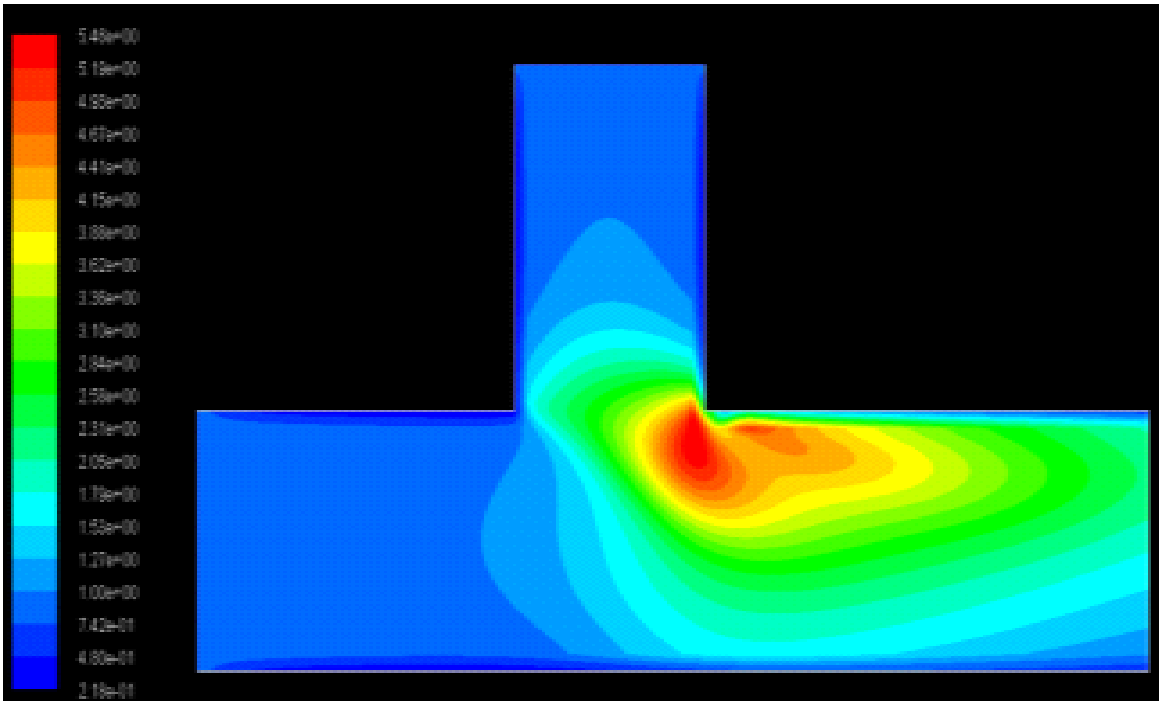
The model and result are shown in figure:-



The following figures shows that velocity, turbulence and temperature plot during mixing.







THE END