PROJECT REPORT

EML 5526 FINITE ELEMENT ANALYSIS AND APPLICATIONS

INSTRUCTOR: Dr. KUMAR

BHARGAV SRIRAM SIDDANI

UF ID 95191948

PROBLEM STATEMENT

The objective of this project was to develop a Finite Element Solver for 2D Linear Steady Heat Transfer problems. The solver was designed for 3 Node Triangular elements. It can handle boundary conditions like Fixed Temperature, Heat Flux coming in or going out of the system, and Convection Heat Transfer. The solver can also take Heat Generation occurring in a system into account.

It reads the mesh data from a text file (Mesh.txt), and boundary conditions and material properties from another text file (BoundaryConditions.txt). The solver was written using MATLAB and it can be used to plot Temperature, Heat Flux along x and Heat Flux along y in the entire domain. The Heat Flux at a node is averaged based on the number of triangular elements that share this node. This averaging process results in smoothing of the values and it is also responsible for gradually varying Heat Flux contours. The solver also creates an output file called 'Result.txt' which contains Node Number, position of the node in x and y, Temperature, Heat Flux along x, and Heat Flux along y for all the nodes in the domain.

DESCRIPTION

2D LINEAR STEADY HEAT TRANSFER

The governing equation for a 2D Steady State Heat Transfer isotropic system with heat generation Q_g is given by

$$\nabla.(k\nabla T) + Q_g = 0$$

Where

k is Thermal Conductivity

Applying Galerkin's Method, the Weak Form of the above equation is

$$\left(\int k\nabla\delta T\nabla T dA\right)_A + \left(\int h\delta T T dS\right)_{S_3} = \left(\int -q_n\delta T dS\right)_{S_1} + \left(\int q_0\delta T dS\right)_{S_2} + \left(\int h\delta T T_\infty dS\right)_{S_3} + \left(\int Q_g\delta T dA\right)_A$$

Where

δT Weighting Function

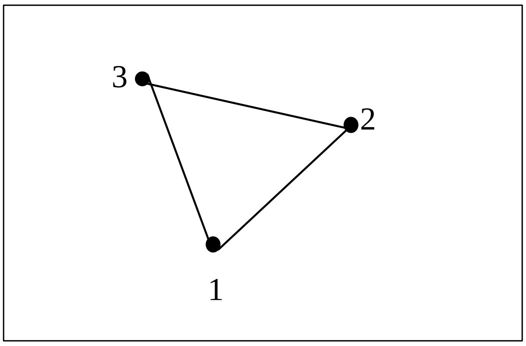
A represents the entire domain

S_1 is a boundary where the temperature is fixed and q_n is the normal heat flux leaving the system at that boundary.

S_2 is a boundary where a heat flux of q_0 is entering the system through this boundary.

S_3 is a boundary where Convective Heat Transfer is occurring. Here, T_∞ represents the Ambient Surrounding Temperature and h is the Convective Heat Transfer Coefficient at the boundary.

3 NODE TRIANGULAR ELEMENT[1]



$$T(x,y) = \{N_1\ N_2\ N_3\}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix}$$

Where

$N_1(x,y) = \dfrac{1}{2A_e}(f_1 + b_1 x + c_1 y)$

$N_2(x,y) = \dfrac{1}{2A_e}(f_2 + b_2 x + c_2 y)$

$N_3(x,y) = \dfrac{1}{2A_e}(f_3 + b_3 x + c_3 y)$

$f_1 = x_2 y_3 - x_3 y_2$

$f_2 = x_3 y_1 - x_1 y_3$

$f_3 = x_1 y_2 - x_2 y_1$

$b_1 = y_2 - y_3$

$b_2 = y_3 - y_1$

$b_3 = y_1 - y_2$

$c_1 = x_3 - x_2$

$c_2 = x_1 - x_3$

$c_3 = x_2 - x_1$

$A_e = \dfrac{1}{2} det \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$

$\{\nabla T\} = \dfrac{1}{2A_e}\begin{bmatrix} b_1 & b_2 & b_3 \\ c_1 & c_2 & c_3 \end{bmatrix}\begin{Bmatrix} T_1 \\ T_2 \\ T_3 \end{Bmatrix}$

The terms of the Weak Form can be simplified into the following for a 3 Node Triangular Element:

$$\left(\int k\nabla\delta T\nabla T dA\right)_{A_e} = \left\{\delta T^{(e)}\right\}^T \left[k_T^{(e)}\right]\left\{T^{(e)}\right\}$$

Where

$\delta T^{(e)}$ is δT of the nodes in the element,

$T^{(e)}$ is T of the nodes in the element,

[1] Theory of this section was taken from ‘INTRODUCTION TO FINITE ELEMENT ANALYSIS AND DESIGN’ by ‘NAM H.KIM,BHAVANI V.SANKAR,ASHOK V. KUMAR’

$$\left[k_T^{(e)}\right] = \frac{k}{4A_e}\begin{bmatrix} b_1^2 + c_1^2 & b_1 b_2 + c_1 c_2 & b_1 b_3 + c_1 c_3 \\ b_1 b_2 + c_1 c_2 & b_2^2 + c_2^2 & b_3 b_2 + c_3 c_2 \\ b_1 b_3 + c_1 c_3 & b_3 b_2 + c_3 c_2 & b_3^2 + c_3^2 \end{bmatrix}$$

$$\left(\int Q_g \delta T dA\right)_{A_e} = \left\{\delta T^{(e)}\right\}^T \frac{1}{3} Q_g A_e \begin{Bmatrix} 1 \\ 1 \\ 1 \end{Bmatrix}$$

$$\left(\int q_0 \delta T dS\right)_{S_2} = \left\{\delta T^{(e)\prime}\right\}^T \frac{q_0 L_e}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

Where

$\left\{\delta T^{(e)\prime}\right\}$ is δT for the nodes on the boundary S_2

L_e is the length of the edge of the element on S_2

$$\left(\int h \delta T T dS\right)_{S_3} = \left\{\delta T^{(e)\prime}\right\}^T \frac{h L_e}{6} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}$$

$$\left(\int h \delta T T_\infty dS\right)_{S_3} = \left\{\delta T^{(e)\prime}\right\}^T \frac{h T_\infty L_e}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

Where

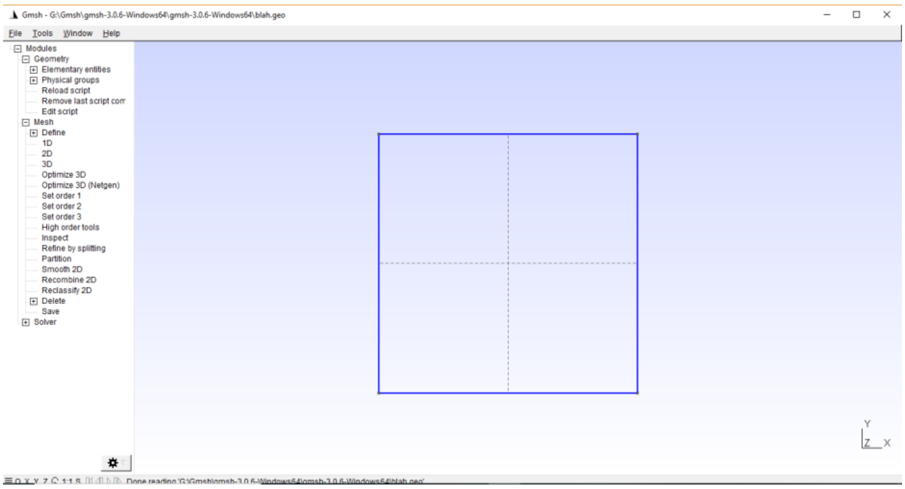
$\left\{\delta T^{(e)\prime}\right\}$ is δT for the nodes on the boundary S_3

L_e is the length of the edge of the element on S_3

STEPS TO USE THIS FEM SOLVER

GMSH[2]:Mesh generator used for this solver

The developed solver reads 2d mesh data generated from GMSH-3.0.6-Windows64. Gmsh is a free 3D finite element mesh generator with a built-in CAD engine and post-processor[3]. Generate a 2D 3 node triangular mesh for any arbitrary shaped system using GMSH. ***The dimensions of your geometry have to be in metres to get proper results from the solver.***



---

[2] C. Geuzaine and J.-F. Remacle. *Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities*. International Journal for Numerical Methods in Engineering 79(11), pp. 1309-1331, 2009.

[3] http://gmsh.info/

Figure 1 A square geometry created using GMSH

Define your boundary conditions (edges) and interior region (surface) using Physical Groups.

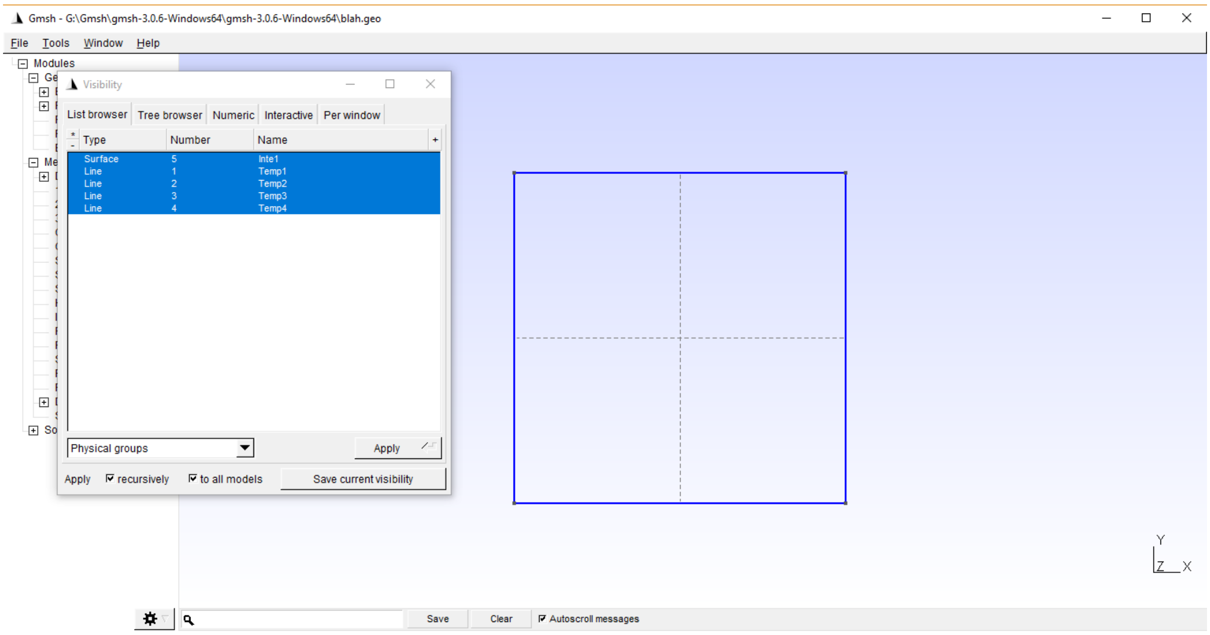


Figure 2 Physical Groups defined for a square shaped geometry

The following rules have to be followed while naming the Physical Names.

- ***Any Interior Region (surface) which is defined should have "Inte" as its first 4 letters.***
- ***Any Temperature Fixed Boundary (edge) which is defined should have "Temp" as its first 4 letters.***
- ***Any Known Heat Flux Boundary (edge) which is defined should have "Flux" as its first 4 letters.***
- ***Any Convection Boundary (edge) which is defined should have "Conv" as its first 4 letters***.

After creating the Physical Names, generate your mesh with desired level of refinement and save the mesh using the File menu.

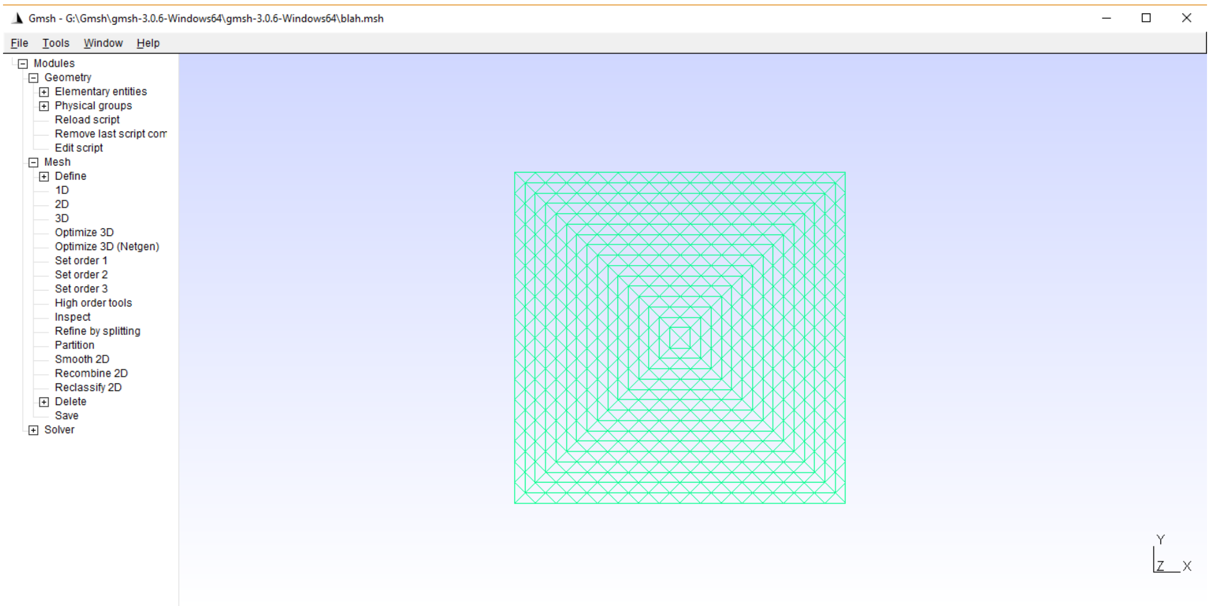
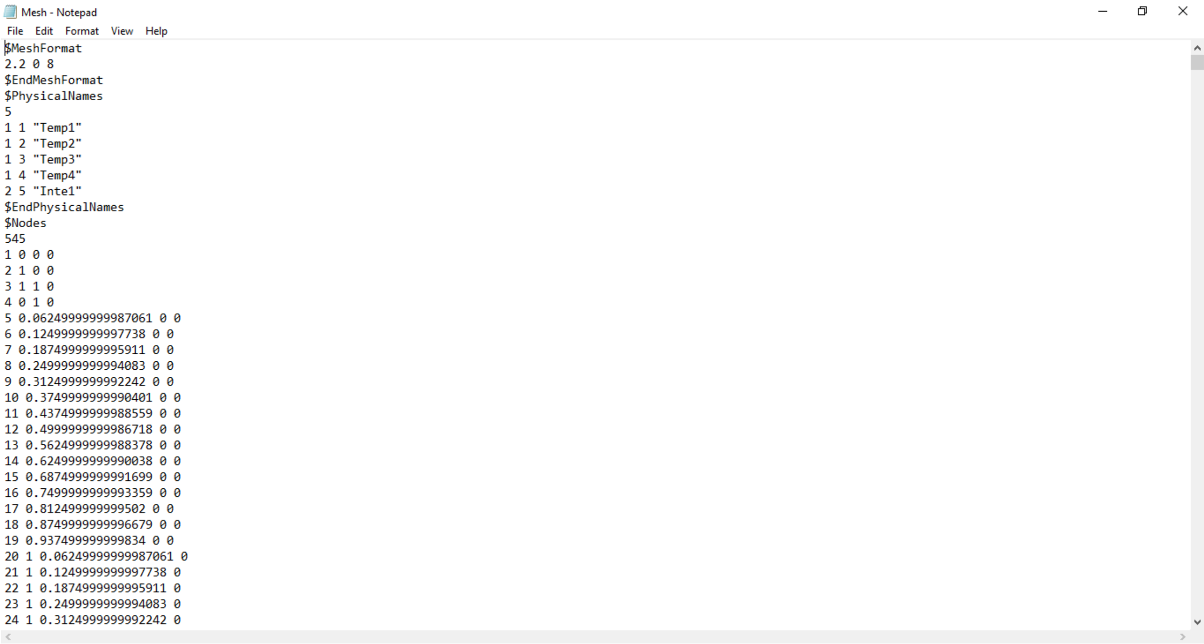


Figure 3 Square geometry meshed using 3 node triangles

This will create a “.msh” file. The contents of this file have to be saved as “Mesh.txt” (text file) without making any changes to it.



Mesh - Notepad

File Edit Format View Help

```
$MeshFormat
2.2 0 8
$EndMeshFormat
$PhysicalNames
5
1 1 "Temp1"
1 2 "Temp2"
1 3 "Temp3"
1 4 "Temp4"
2 5 "Inte1"
$EndPhysicalNames
$Nodes
545
1 0 0 0
2 1 0 0
3 1 1 0
4 0 1 0
5 0.06249999999987061 0 0
6 0.1249999999997738 0 0
7 0.1874999999995911 0 0
8 0.2499999999994083 0 0
9 0.3124999999992242 0 0
10 0.3749999999990401 0 0
11 0.4374999999988559 0 0
12 0.4999999999986718 0 0
13 0.5624999999988378 0 0
14 0.6249999999990038 0 0
15 0.6874999999991699 0 0
16 0.7499999999993359 0 0
17 0.812499999999502 0 0
18 0.8749999999996679 0 0
19 0.937499999999834 0 0
20 1 0.06249999999987061 0
21 1 0.1249999999997738 0
22 1 0.1874999999995911 0
23 1 0.2499999999994083 0
24 1 0.3124999999992242 0
```

Figure 4 Sample Mesh text file

Boundary Conditions parameters and Interior Region parameters and material properties have to be defined in another text file named as “BoundaryConditions”. ***The parameters values have to be inputted in SI units to get meaningful results from the solver.***

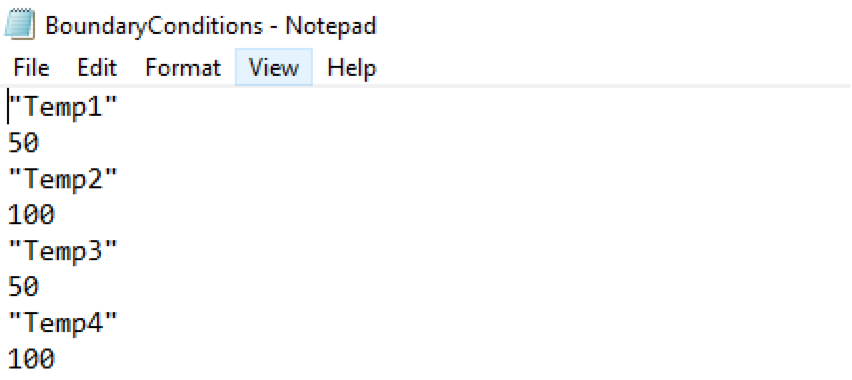
The Physical Names (Names of edges (Boundary Conditions) and surfaces (Interior Region)) can be defined in any order ***but should start from the first line of the text file.***
The format that has to be followed when defining the Physical Names and their parameters in the BoundaryConditions.txt file has been explained using an example.

Example:
Let’s assume that a system has a single interior region (Surface) defined as Inte1. The system has 4 Boundary Conditions (Edges) namely Temp1, Flux1, Conv1, and Temp2. Then the BoundaryConditions.txt can be defined as follows:

```
“Inte1”
200 10
“Temp1”
25
“Temp2”
30
“Flux1”
10
“Conv1”
2 15
```

Every Physical Name has to be defined inside double quotes (“”) followed by its parameters in the next line.

- ***Every Physical Name of type ‘Inte’ will require 2 parameters. The first parameter is Thermal Conductivity and the second parameter is Heat Generation. Both the parameters should only be separated by a single space.***
- ***Every Physical Name of type ‘Temp’ requires only 1 parameter, which is the Fixed Temperature on that edge.***
- ***Every Physical Name of type ‘Conv’ would need 2 parameters. The first parameter should be its Convection Coefficient and the second one has to be Ambient Surrounding Temperature.***
- ***Every Physical Name of type ‘Flux’ needs only a single parameter, which is normal heat flux at the edge. Convention which has to be followed is to assign positive value to flux entering the system and negative value to flux leaving the system.***


BoundaryConditions - Notepad
File Edit Format View Help

```
"Temp1"
50
"Temp2"
100
"Temp3"
50
"Temp4"
100
```

Figure 5 Sample BoundaryConditions.txt file

These text files have to be in the same folder as the solver (.m file).

Solver opens ‘Mesh.txt’ file to read the Physical Names and their corresponding numbers assigned by GMSH and then closes the ‘Mesh.txt’ file

Solver opens ‘BoundaryConditions.txt’ file to read parameters of the Physical Names of the system and closes the ‘BoundaryConditions.txt’ file.

Solver reopens ‘Mesh.txt’ file to read the data (Node number, x and y positions of the node) of the nodes and then closes it.
The Solver again reopens ‘Mesh.txt’ file to read element data (element number, Physical Name Number and nodes in the element) and closes it.

Solver creates ‘Global Stiffness Matrix’ and ‘Equivalent Force Column Matrix’ ([K]{X}={F}) using the element and node data.

The Solver solves the ‘System of Equations’ using Gauss Seidel Iterative Method and uses a convergence of 10^{-6}, which can be changed by changing the value of the variable “converg”. The solver then evaluates the heat flux in each element and assigns values to all nodes. These values are obtained by averaging the heat flux a node receives from all the elements that contain the node.

The solver then prints Node number, x and y position of the node, Temperature, Heat flux along x, and Heat Flux along y for all the nodes in the system into a text file named as ‘Result.txt’ file.

Flowchart of the Solver

The Solver also contains commands to display contours of Temperature, Heat Flux along x, and Heat Flux along y at the end and these can be commented out if not required.

RESULTS

Validation of the solver was done by comparing the results obtained by using the solver with those of using SolidWorks for a simple square geometry with a side of 1 metre.

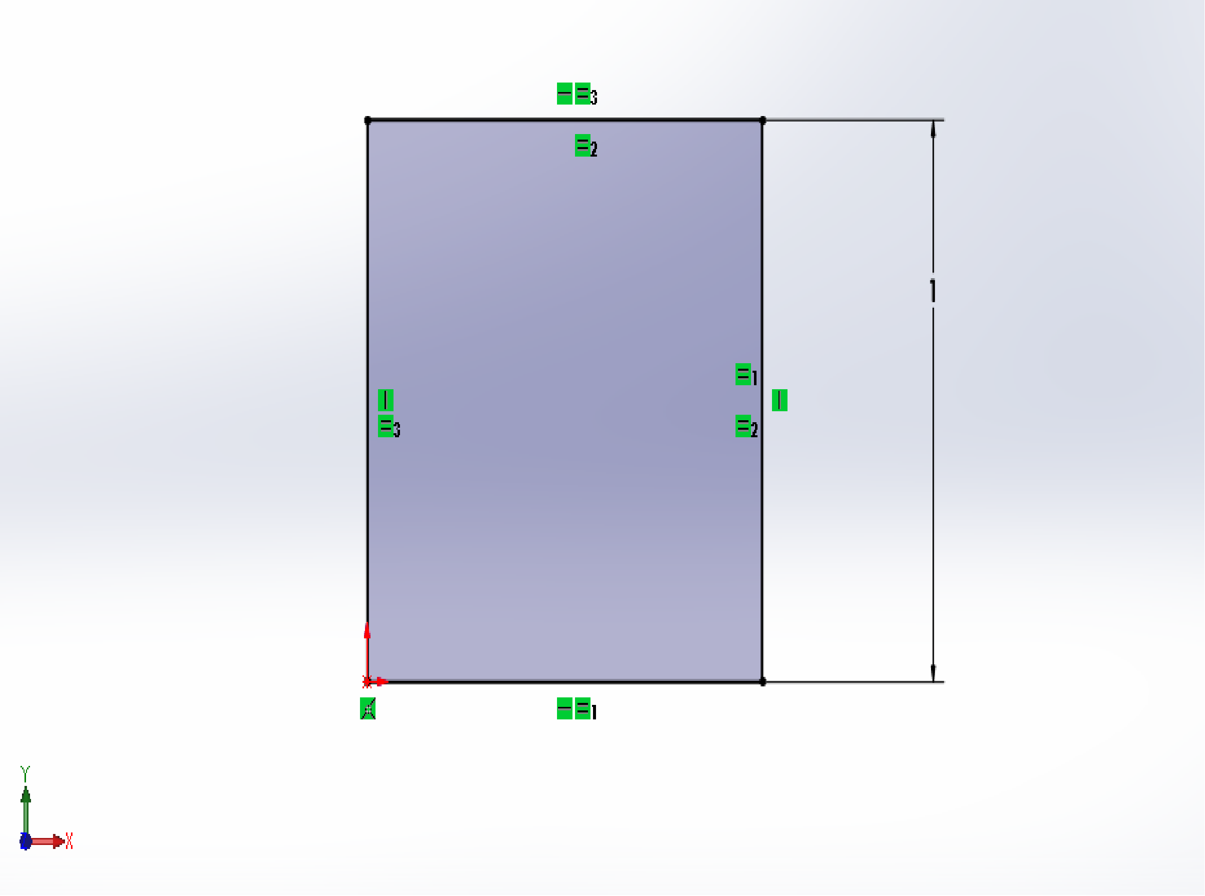


Figure 6 Geometry of square used for validation

Thermal conductivity of the system is 200 W/mK and there is no heat generation in it. Left edge has 0 W/m^2 heat flux boundary condition. Bottom and Right edges have Convection Heat transfer with a Convection coefficient of 50 W/m^2K and an Ambient Temperature of 10 K. The Top edge has a fixed temperature of 100 K.

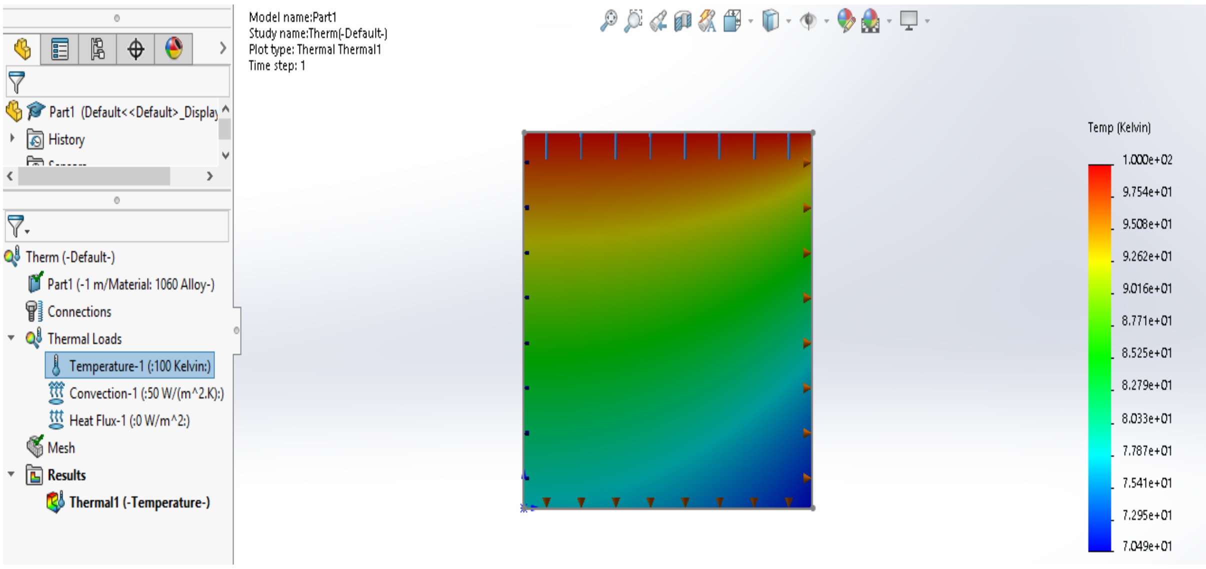


Figure 7 Temperature Contour obtained by using SolidWorks

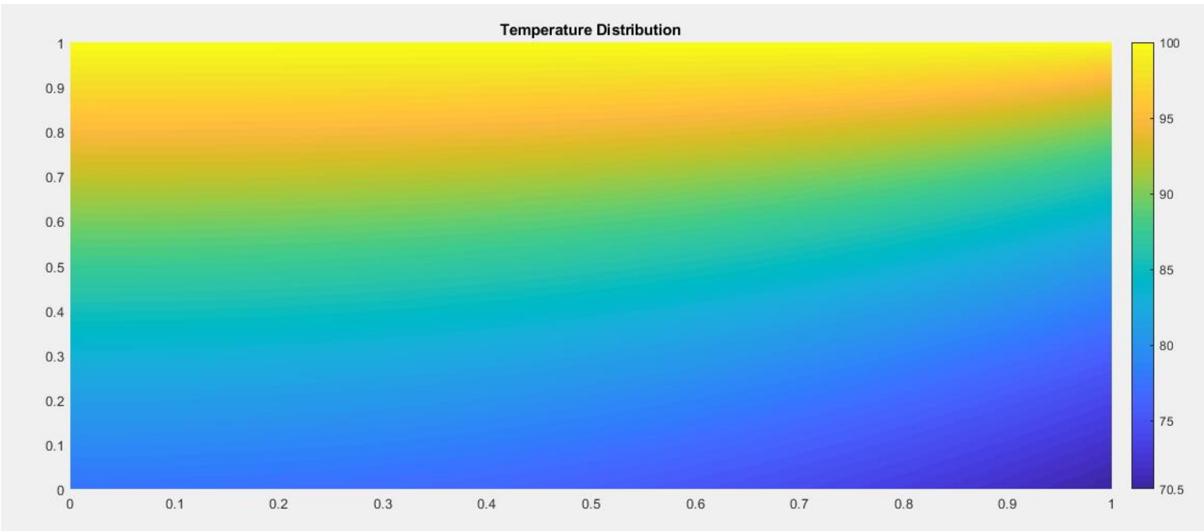


Figure 8 Temperature Contour obtained by using Solver Code

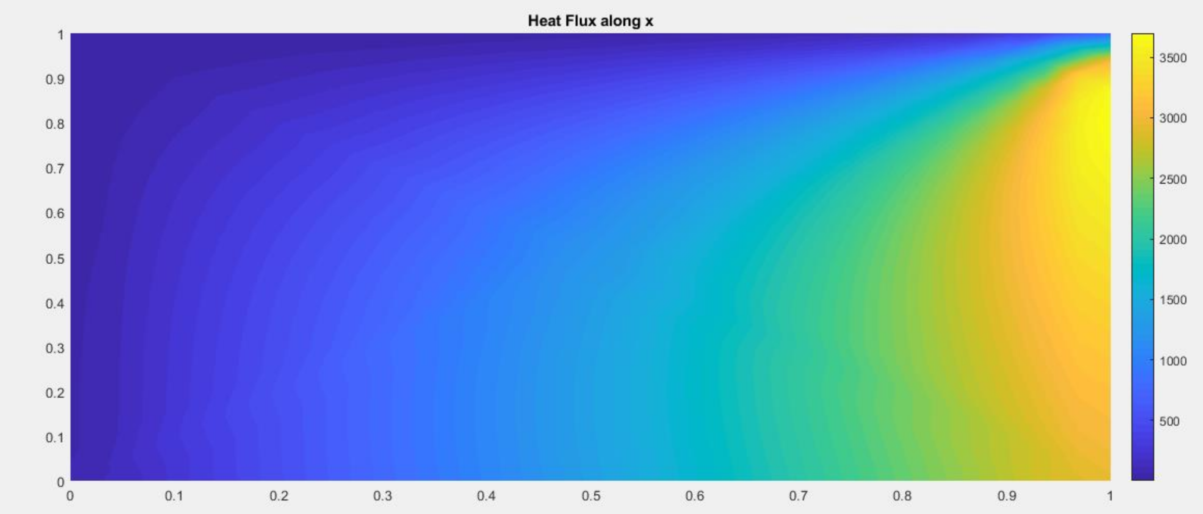


Figure 9 Heat Flux along x contour obtained by using Solver Code

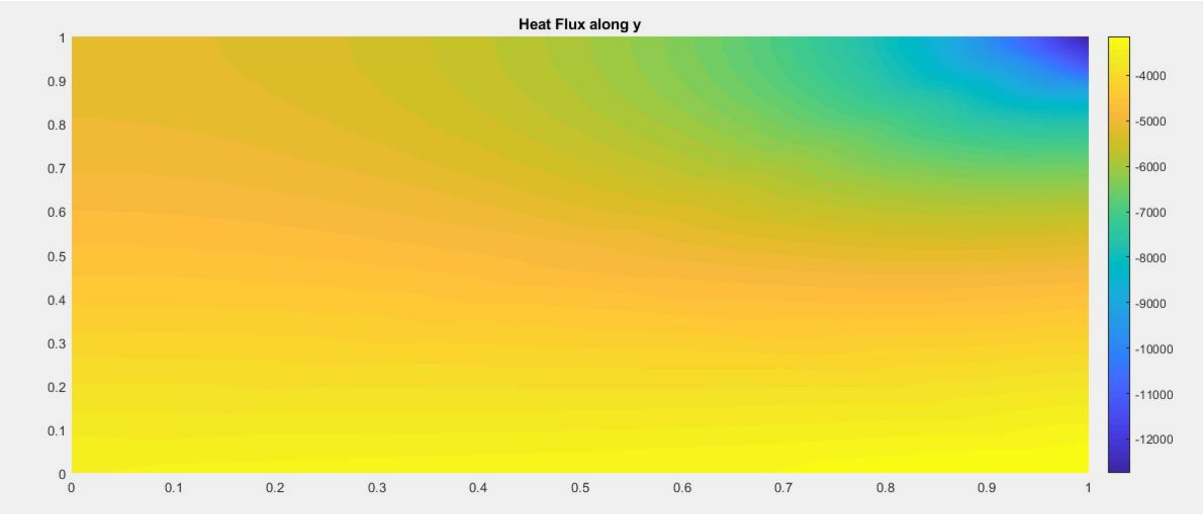


Figure 10 Heat Flux along y contour obtained by using Solver Code

The results obtained using Solver Code are in very good agreement with those from SolidWorks.

CONCLUSIONS AND DISCUSSION

The Solver Code written in MATLAB can be used to solve any 2D system which is meshed using 3 node triangular elements in GMSH. It generates reliable results and also provides smooth contour plots for Temperature, heat Flux along x, Heat Flux along y when a decently refined mesh is used for analysis.

This project has given me a good insight into how mesh data is read and stored by a solver and how it accesses that data. Though the system of equations here have been solved using Gauss Seidel Iterative Method, they can also be solved using Direct Methods like Gaussian Elimination, LU Decomposition.

Heat Flux in an element is constant as a 3 node triangular element is used in this solver. More accurate results can be obtained on a much coarser mesh using elements with varying heat flux in them.

The solver can be further extended to solve Non Linear Heat Transfer equations by including radiation. Then the system of equations have to be solved iteratively using Under Relaxation.

REFERENCES

[1] https://www.mathworks.com/help/index.html