Conduction
&
LaplacianFoam

Mr. Omkar Bhandari
Mr. Biraj Khadka
Project Research Assistant @ FOSSEE, IIT Bombay

Outline

- ❑ laplacianFoam
- ❑ Setting up a 1D case
- ❑ Running the Simulation
- ❑ Results and Post-Processing

Conduction-Diffusion Equation

Physical Equation in 1D form without source

$$\frac{\partial T}{\partial t} - \frac{\partial}{\partial x}\alpha\frac{\partial T}{\partial x} = 0$$

$$\frac{\partial T}{\partial t} - \alpha\frac{\partial^2 T}{\partial x^2} = 0$$

$$\frac{\partial T}{\partial t} - \alpha\nabla^2 T = 0$$

Where, T is Temperature
DT & α is Diffusivity
t is Time
x is position

Equation written in OpenFOAM

cd $FOAM_SOLVERS/basic/laplacianFoam

```
while (simple.loop(runTime))
{
    Info<< "Time = " << runTime.timeName() << nl << endl;

    while (simple.correctNonOrthogonal())
    {
        fvScalarMatrix TEqn
        (
            fvm::ddt(T) - fvm::laplacian(DT, T)
         ==
            fvOptions(T)
        );

        fvOptions.constrain(TEqn);
        TEqn.solve();
        fvOptions.correct(T);
    }
```

Problem Statement-1

Q.1 Solve the 1-D unsteady heat conduction equation using OpenFOAM with the given conditions?
Given the 1-D unsteady heat conduction equation:

$$\frac{\partial T}{\partial t} = \alpha \frac{\partial^2 T}{\partial x^2} \tag{1}$$

With initial condition at $t = 0$ sec: $T = 0$, and boundary conditions for $t > 0$:

1. at $x = 0$: $T = 0$
2. at $x = 1$ m: $T = 1$

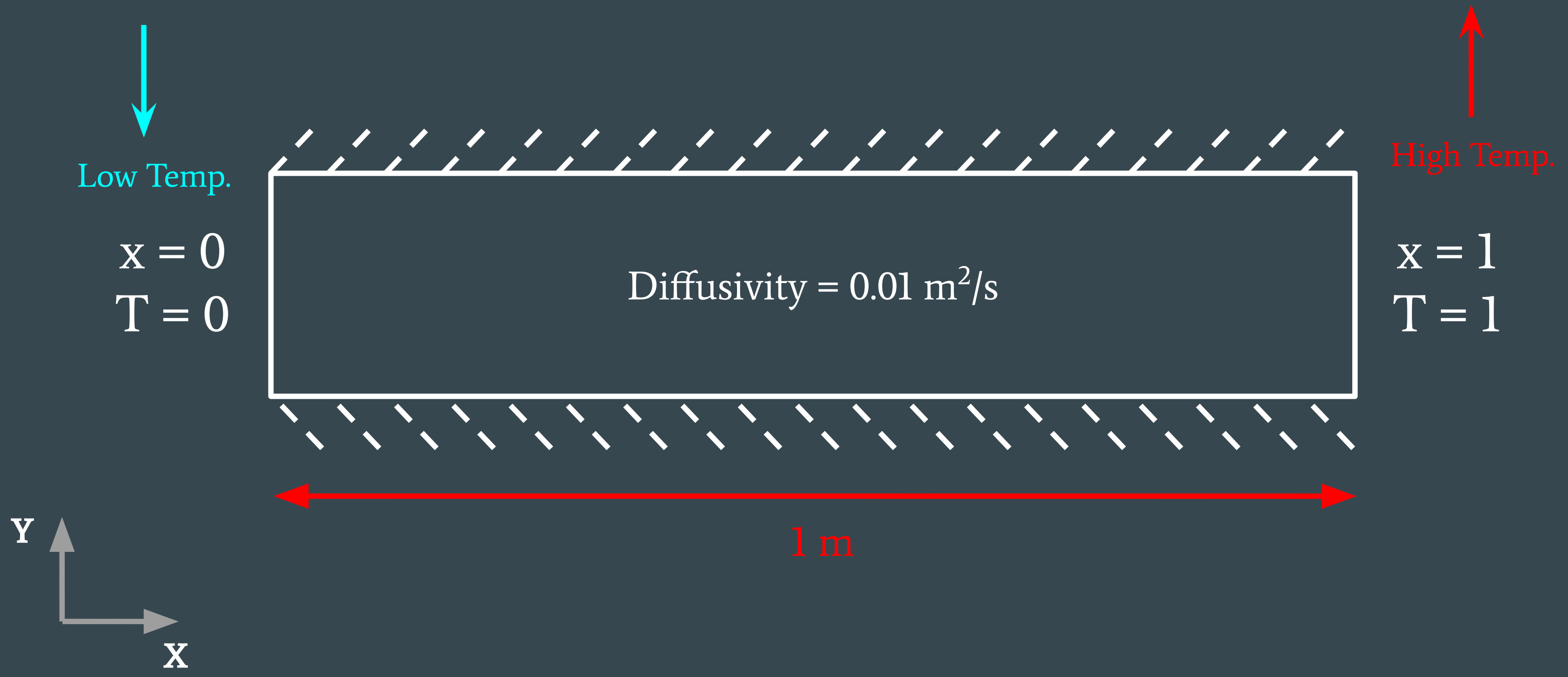
Where thermal diffusivity $\alpha = 0.01\,\mathrm{m}^2/\mathrm{s}^{-1}$, grid point spacing $\Delta x = 0.01$, and time step $\Delta t = 0.001$
You need to determine the temperature distribution at the following time points:

(i) $t = 1$ sec

(ii) $t = 5$ sec

(iii) $t = 10$ sec

Problem Statement-1



1. Setting up the case

Run following commands to setup a base case into run directory

```
cd $FOAM_RUN

mkdir conduction

cp -r $FOAM_TUTORIALS/basic/laplacianFoam/flange conduction/case1

cd conduction/case1

cp -r $FOAM_TUTORIALS/incompressible/icoFoam/cavity/cavity/system/blockMeshDict system/

rm flange.ans All*
```

2. Edit blockMeshDict file

gedit system/blockMeshDict

blockMeshDict
~/OpenFOAM/omkar-9/conduction/case1/system

```
\*---------------------------------------------------------------------------*/
FoamFile
{
    format      ascii;
    class       dictionary;
    object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices
(
    (0 0 0)
    (1 0 0)
    (1 0.1 0)
    (0 0.1 0)
    (0 0 0.1)
    (1 0 0.1)
    (1 0.1 0.1)
    (0 0.1 0.1)
);

blocks
(
    hex (0 1 2 3 4 5 6 7) (100000 1 1) simpleGrading (1 1 1)
);

edges
(
);

```

blockMeshDict
~/OpenFOAM/omkar-9/conduction/case1/system

```

defaultPatch
{
        name emptyPatch;
        type empty;
}
boundary
(
    x1
    {
        type patch;
        faces
        (
            (0 4 7 3)
        );
    }

        x2
    {
        type patch;
        faces
        (
            (2 6 5 1)
        );
    }


);

mergePatchPairs
(
);
```

4. Setup Boundary & physical Conditions

gedit 0/T

T
~/OpenFOAM/omkar-9/conduction/case1/0

```
FoamFile
{
    format      ascii;
    class       volScalarField;
    object      T;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 0 0 1 0 0 0];

internalField   uniform 0;

boundaryField
{
    x1
    {
        type            fixedValue;
        value           uniform 0;
    }

    x2
    {
        type            fixedValue;
        value           uniform 1;
    }


}

// ************************************************************************* //
```

C   Tab Width: 8   Ln 36, Col 1   INS

gedit constant/transportProperties

transportProperties
~/OpenFOAM/omkar-9/conduction/case1/constant

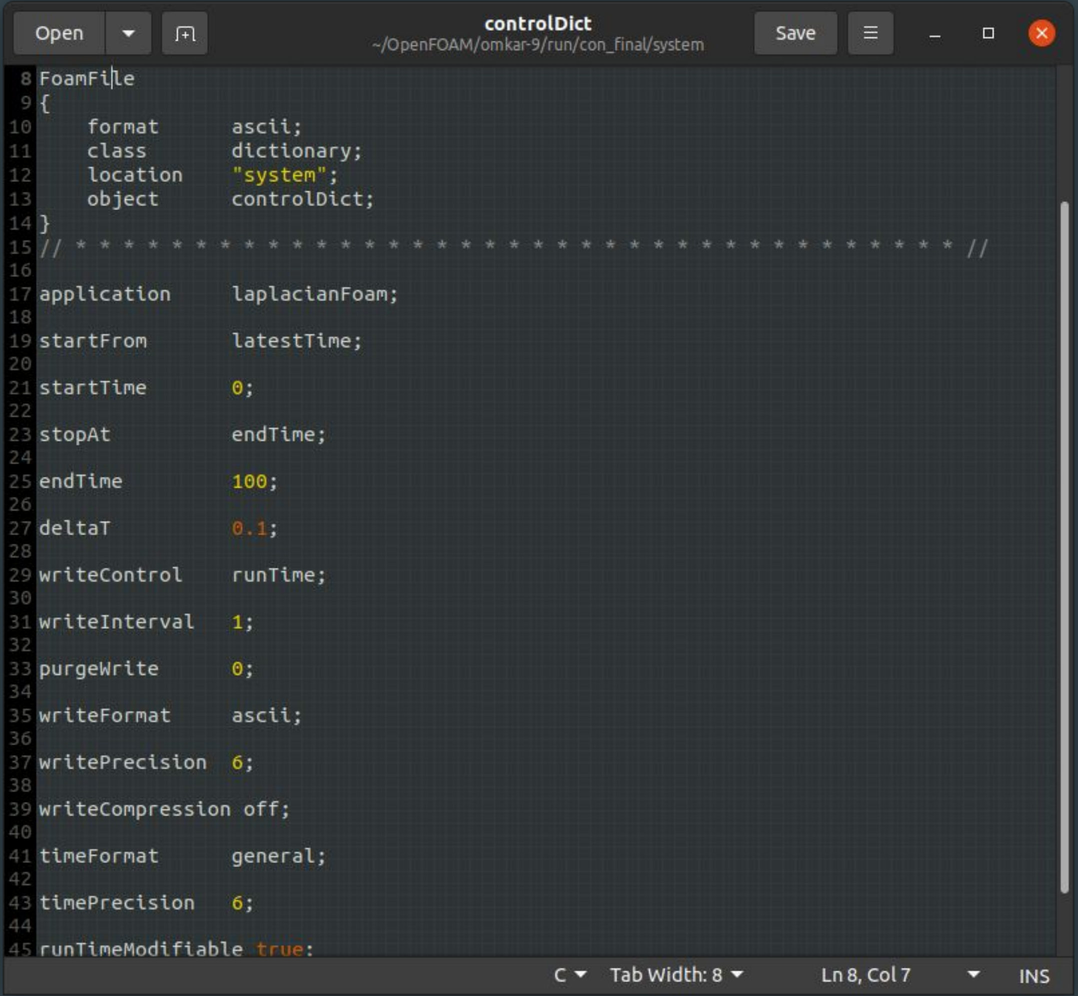
```
/*--------------------------------*- C++ -*----------------------------------*\
  =========                 |
  \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox
   \\    /   O peration     | Website:  https://openfoam.org
    \\  /    A nd           | Version:  9
     \\/     M anipulation  |
\*---------------------------------------------------------------------------*/
FoamFile
{
    format      ascii;
    class       dictionary;
    location    "constant";
    object      transportProperties;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

DT              DT [0 2 -1 0 0 0 0] 0.01;


// ************************************************************************* //
```

C   Tab Width: 8   Ln 19, Col 1   INS

5. Setup Control Parameters

- Solver = laplacianFoam
- Start Time = 0
- End Time = 10
- deltaT = 0.1

gedit system/controDict




```
FoamFile
{
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     laplacianFoam;

startFrom       latestTime;

startTime       0;

stopAt          endTime;

endTime         100;

deltaT          0.1;

writeControl    runTime;

writeInterval   1;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;
```

6. fvSchemes

gedit system/fvSchemes

fvSchemes
~/OpenFOAM/omkar-9/conduction/case1/system

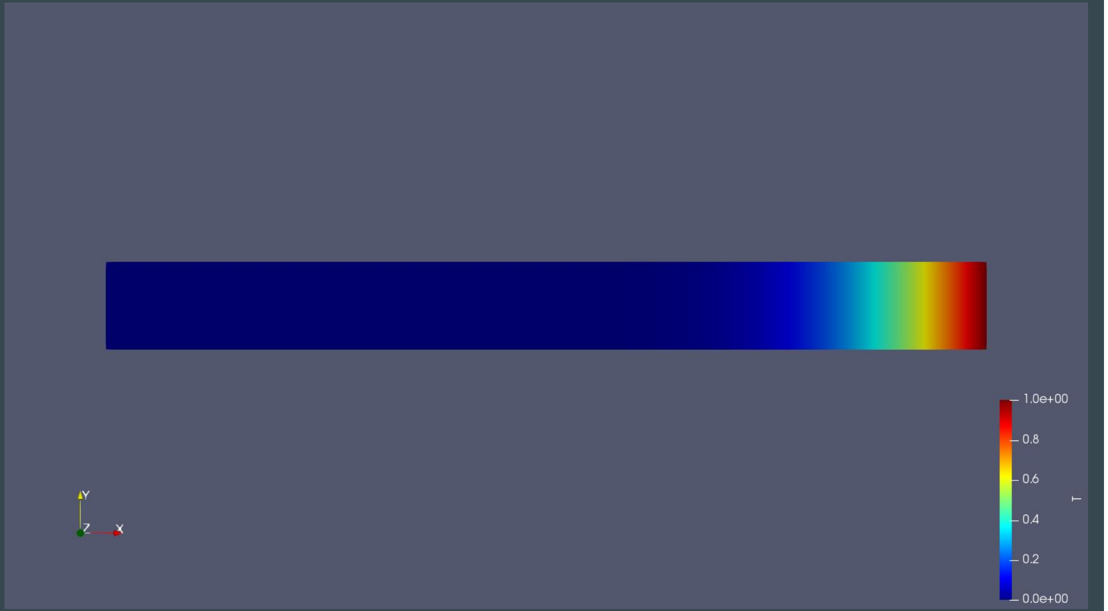
```
FoamFile
{
    format      ascii;
    class       dictionary;
    location    "system";
    object      fvSchemes;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

ddtSchemes
{
    default         Euler;
}

gradSchemes
{
    default         Gauss linear;
    grad(T)         Gauss linear;
}

divSchemes
{
    default         none;
}

laplacianSchemes
{
    default         none;
    laplacian(DT,T) Gauss linear corrected;
}

interpolationSchemes
{
    default         linear;
}

snGradSchemes
{
    default         corrected;
}
```

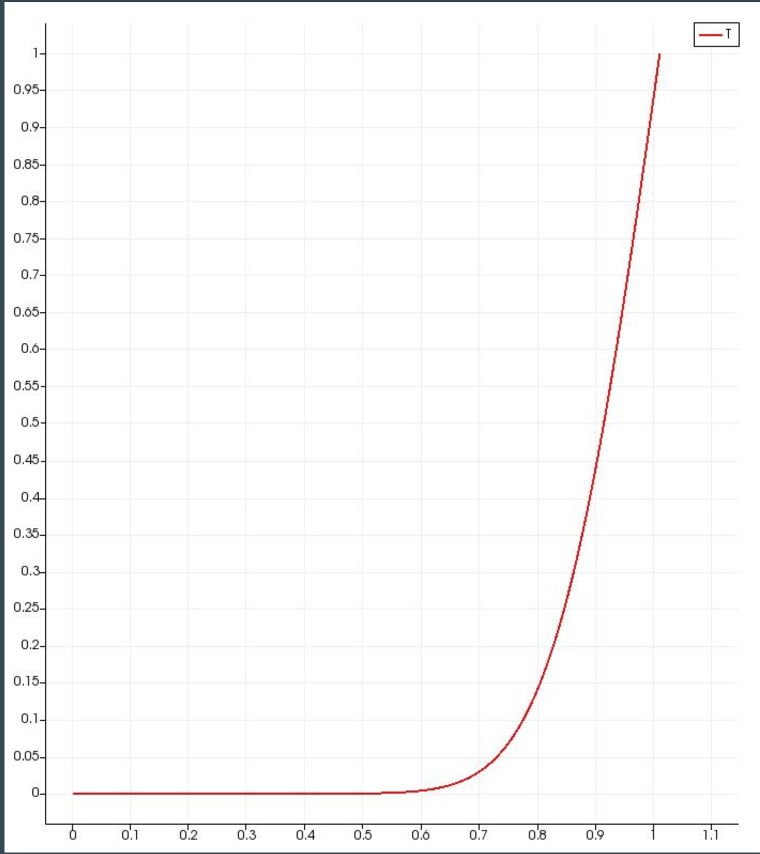
C Tab Width: 8 Ln 43, Col 1 INS

7. Run Simulation

- blockMesh - Execute geometry and mesh
- laplacianFoam - Solve Equations
- paraFoam - Shows results in ParaView

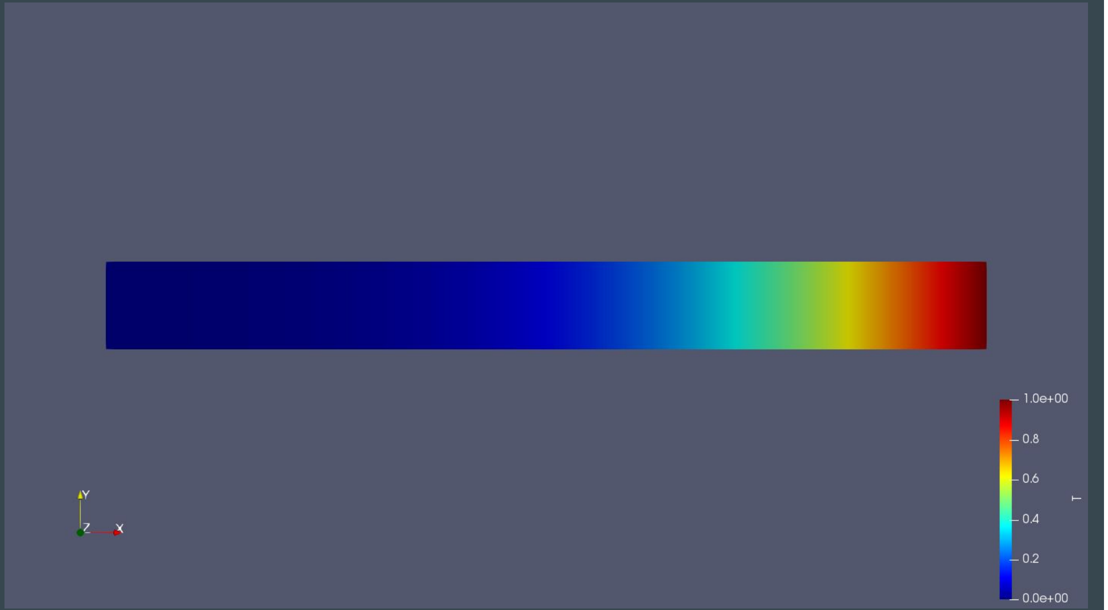
Results

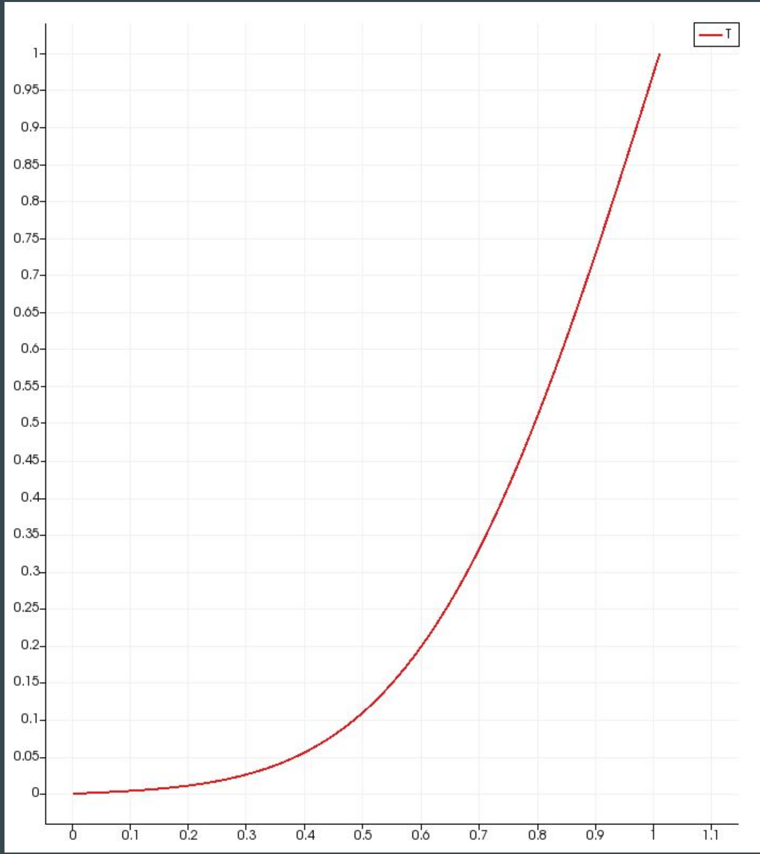




t = 1 sec

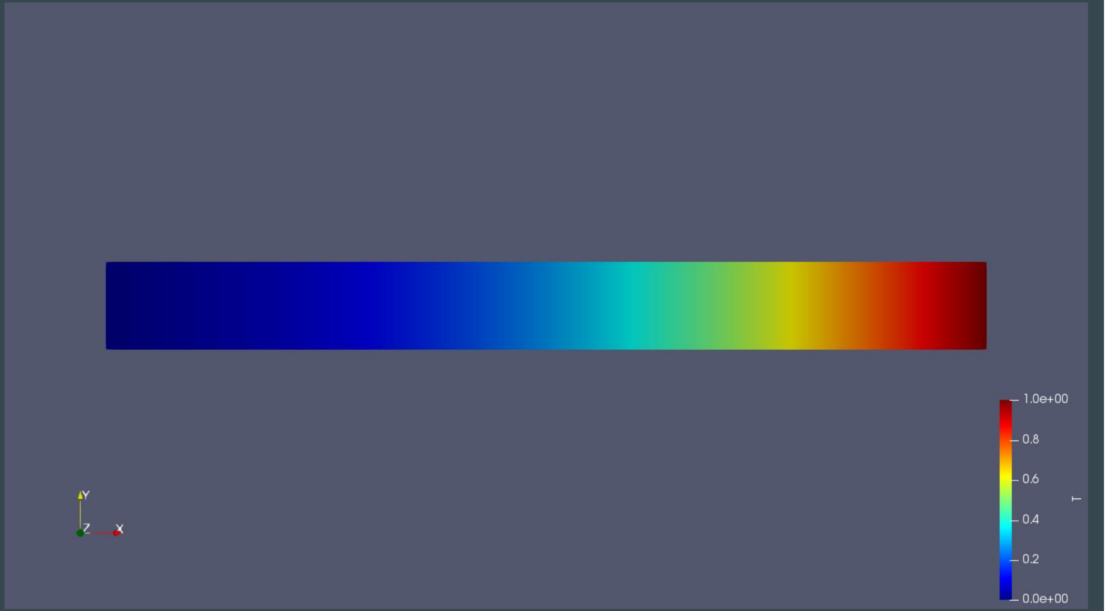
Results

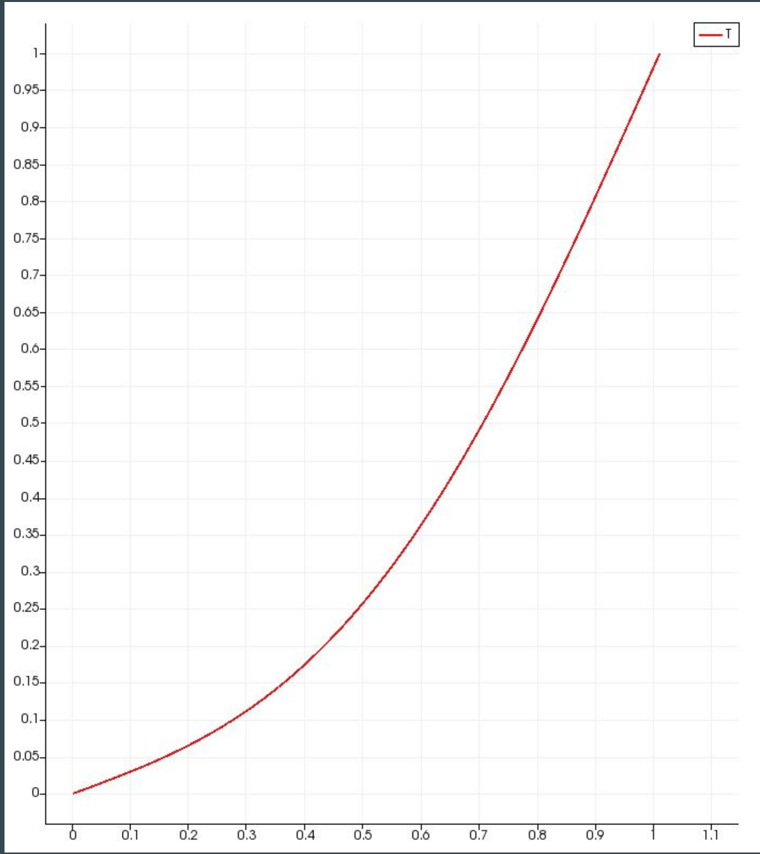




t = 5 secs

Results

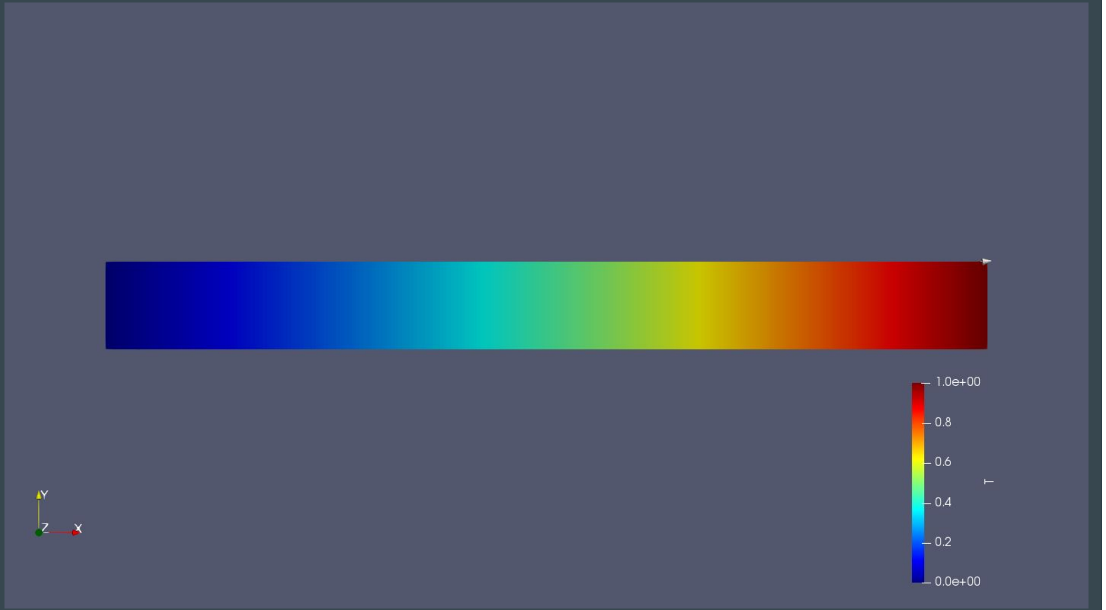


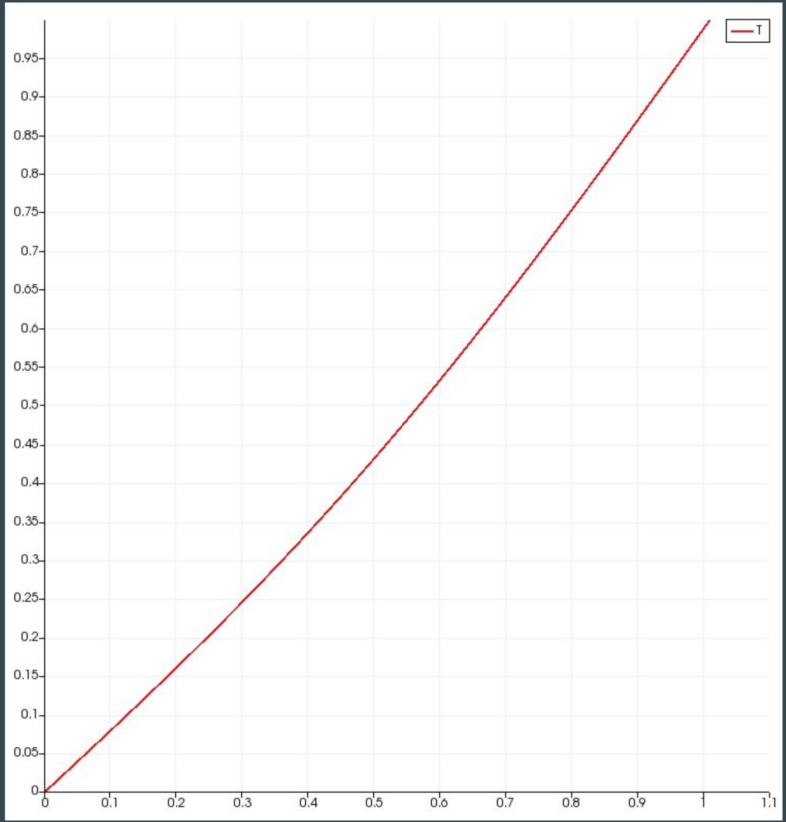


t = 10 secs

Results

If Simulation is run Long enough it attains a steady state solution





t = 100 secs

Thank you!