CFD ANALYSIS OF AN EVACUATED TUBE SOLAR WATER HEATER

Authors

Muhammad Humayoon Tariq (Reg # 10-NUST-BE-ME-57)

SaadatHussain (Reg # 10-NUST-BE-ME-78)

Supervisor

Dr. Muhammad Sajid

DEPARTMENT OF MECHANICAL ENGINEERING

SCHOOL OF MECHANICAL& MANUFACTURING ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY

ISLAMABAD

JUNE, 2014

CFD ANALYSIS OF AN EVACUATED TUBE SOLAR WATER HEATER

Authors

Muhammad Humayoon Tariq Reg # 10-NUST-BE-ME-57

SaadatHussainReg # 10-NUST-BE-ME-078

A thesis submitted in partial fulfillment of the requirements for the degree of

BE Mechanical Engineering

Thesis Supervisor:

Dr. Muhammad Sajid

Thesis Supervisor's Signature: ____

DEPARTMENT OF MECHANICAL ENGINEERING

SCHOOL OF MECHANICAL & MANUFACTURING ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY, ISLAMABAD

JUNE, 2014

Declaration

We certify that this research work titled "Design and CFD analysis of an Evacuated Tube Solar Water Heater" is our own work. The work has not been presented elsewhere for assessment. The material that has been used from other sources has been properly acknowledged / referred.

Signature of Student
Muhammad Humayoon Tariq
10-NUST-SMME-BE-ME-057

Signature of Student
Saadat Hussain
10-NUST-SMME-BE-ME-078

Language Correctness Certificate

This thesis has been read by an English expert and is free of typing, syntax, semantic, grammatical and spelling mistakes. Thesis is also according to the format given by the university.

Signature of Student

Muhammad Humayoon Tariq

10-NUST-SMME-BE-ME-057

Signature of Student

SaadatHussain

10-NUST-SMME-BE-ME-078

Signature of Supervisor

Copyright Statement

Acknowledgements

I am thankful to my Creator Allah Subhana-Watala to have guided me throughout this work at every step and for every new thought which You setup in my mind to improve it. Indeed I could have done nothing without Your priceless help and guidance. Whosoever helped me throughout the course of my thesis, whether my parents or any other individual was Your will, so indeed none be worthy of praise but You.

I am profusely thankful to my beloved parents who raised me when I was not capable of walking and continued to support me throughout in every department of my life.

I would also like to express special thanks to my supervisor Dr. Muhammad Sajid for his help throughout my thesis and also for Fluid Mechanics I, II and Computational Fluid Dynamics courses which he has taught me. I can safely say that I haven't learned any other engineering subject in such depth than the ones which he has taught.

I would also like to express my gratitude to all the individuals who have rendered valuable assistance to my study.

Dedicated to our exceptional parents and adored siblings whose tremendous support and cooperation led us to this wonderful accomplishment

Abstract

Evacuated Tube Solar Water Heaters are one of the most widely used solar energy devices. Efficient Heat Transfer is one of the major concerns in Solar heating devices. In an evacuated tube solar water heater, a non-toxic fluid is used inside the tube which absorbs incident heat from the sun and transfers it to the bulb inside the overhead tank. Each tube is made of a finned copper pipe surrounded by glass tube and the annular space in between is evacuated.

In this project, we have performed our analysis on the non-toxic fluid inside the tube using OpenFOAM (freeware). A 3-D mesh model of the tube and the bulb suspended inside the overhead tank was made and analysis was performed using the solver Buoyant Pimple Foam. Temperature boundary conditions at the tube surface and the bulb were assumed keeping in mind the average local temperature.

Due to computational limitations, simulations were performed up to 250 seconds. Calculations of heat flux across the tube-bulb boundary were performed and the results were plotted. Average temperature at the center plane of the tube at different time steps was also calculated and plotted.

Table of Contents

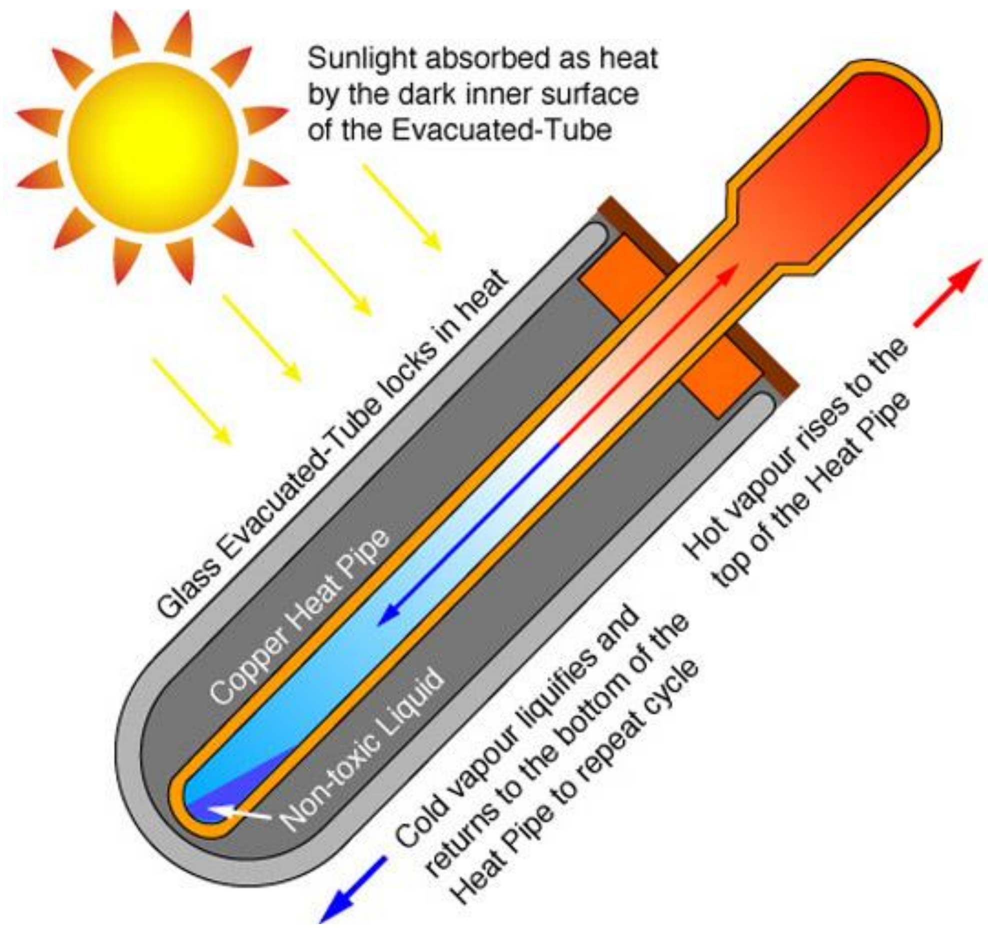
INTRODUCTION

Background:

Solar water heaters are more and more used worldwide, and the evacuated-tube designs are the most popular due to their simplicity and better overall performance over their flat-plate counterparts, especially in adverse weather conditions.

Many evacuated-tube designs have been developed and are being used among which the water-in-glass design is very popular because of its low cost and simple manufacturing and installation procedures. Another design uses a heat-pipe system with an intermediate fluid used to carry the heat from the heating elements to the tank. In this case the working fluid undergoes a phase change operation while it is transported up and down. A water-in-glass tube-based collector consists of a set of glass tubes connected to a shell or tank. Each tube is surrounded by a second glass tube of a larger diameter. The annular space between the tubes is evacuated in order to minimize the heat losses. The working fluid, generally water, flows from the tank to the tubes, captures heat, and then flows back to the tank by a natural circulation mechanism.

Various investigations have been conducted to characterize the overall performance of water-in-glass evacuated tube collectors and the results show that the overall efficiency is in the range of 50 to 60%. Advanced numerical techniques have been also used to investigate the performance and to find possible ways to improve existing designs of evacuated tubes. A heat-pipe based collector involves generally a similar set of tubes connected to a tank. Each tube is made of a finned copper pipe (heat pipe) surrounded by a glass tube and the annular space in between is evacuated. The heat pipe is a closed container consisting of a capillary wick structure and a small amount of vaporizable fluid. It is based on an evaporating-condensing cycle involving an evaporation phase using the solar heat followed by a condensation phase in which the heat is released to the water contained in the overhead tank. The working fluid flows by natural circulation between the two phases in order to transport the heat as needed. Various studies have been conducted to evaluate the thermal performance of evacuated-tube solar collectors.



Scope and Motivation:

As observed in previous publications, mostly the analysis consisted of experimental setup and observance of thermal efficiencies and temperatures dependent upon external factors like angle of inclination, ambient temperatures and local climatic conditions. However less effort has been put into the analysis using Computational fluid dynamics as a tool to investigate the temperature and pressure changes occurring inside the collector tube.

In order to carry out the above mentioned analysis, we have used OpenFOAM (freeware) as our software tool. We have carried out our analysis using the solver buoyant pimple foam to observe the temperature and pressure changes inside the collector tube. buoyantPimpleFoamis a transient solver for buoyant, turbulent flow of compressible fluid for ventilation and heat transfer.Due to computational limitations, we have performed analysis on a single tube up to 250 seconds. We have assumed the initial temperatures and pressures based on the work done previously. We have calculated the change in heat flux across the tube-bulb boundary at different time intervals as a part of post processing operations. We have also calculated the average temperature at the center plane of the collector tube which gives an understanding of the maximum and minimum temperatures that can be observed over a certain period of time.

Analysis on OpenFOAM using buoyantPimpleFoam

Mesh design:

Our mesh consisted of a single collector tube of an evacuated tube solar water heater with a bulb placed at the top which is assumed to be suspended inside the overhead water sink. The tube is surrounded by finned copper which is further surrounded by double-layered glass tubing which is evacuated from within in order to minimize heat losses to the surroundings.

To carry out the task of building the mesh, we first divided the tube-bulb geometry into three parts, namely; the bulb, the pipe and the glass evacuation structure on the outside. We used planar coordinate system and vertices were assigned taking the top-center of the pipe to be the origin. This attempt to compile the mesh was unsuccessful since a few planes were overlapping. Therefore we restructured the geometry completely and used the bottom-center of the pipe as our origin. The BlockMeshDict file of our mesh can be seen in the Appendix A.

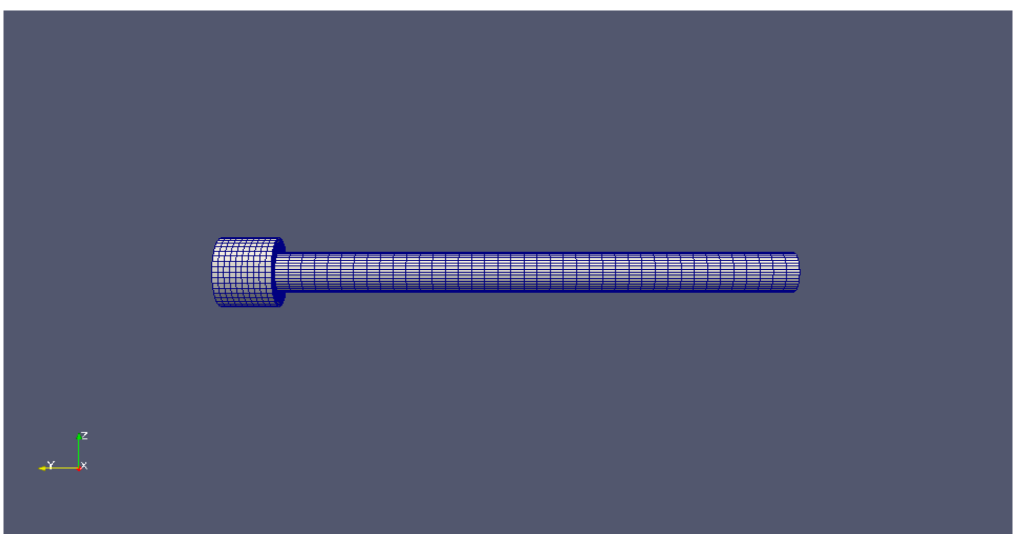


Figure 1: Pipe and Bulb Geometry

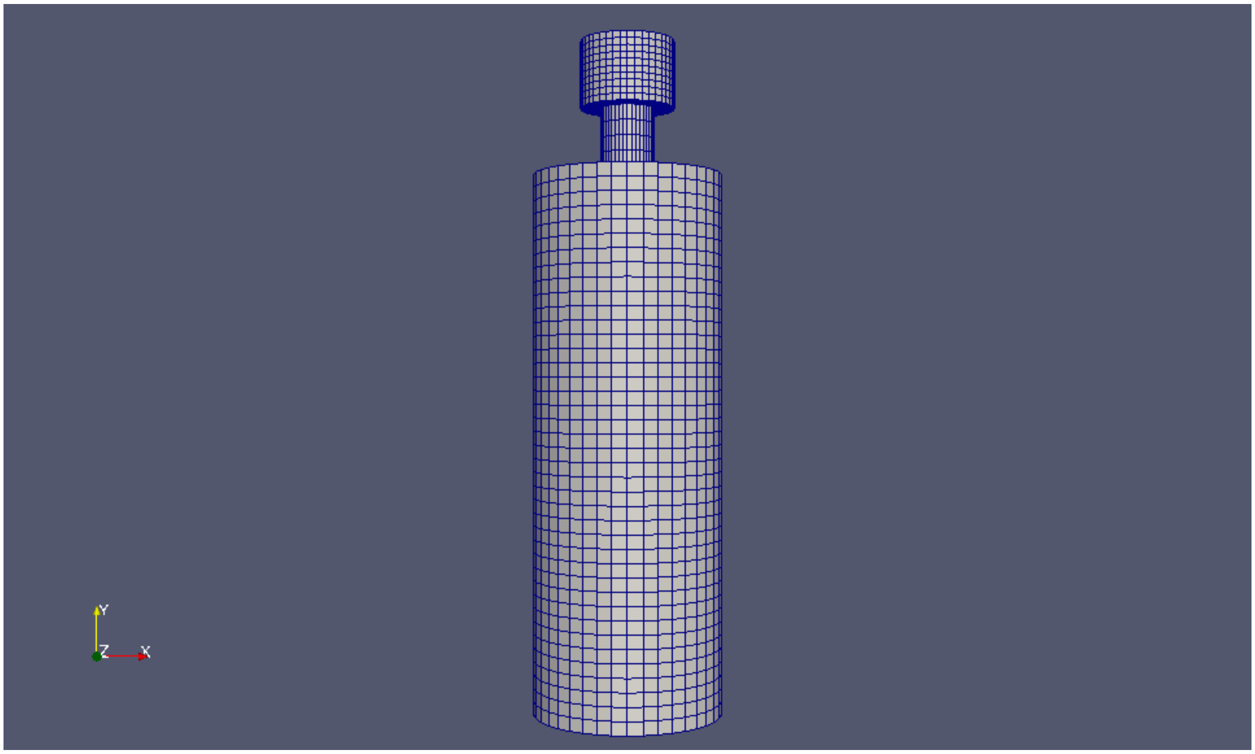


Figure 2: Complete Geometry

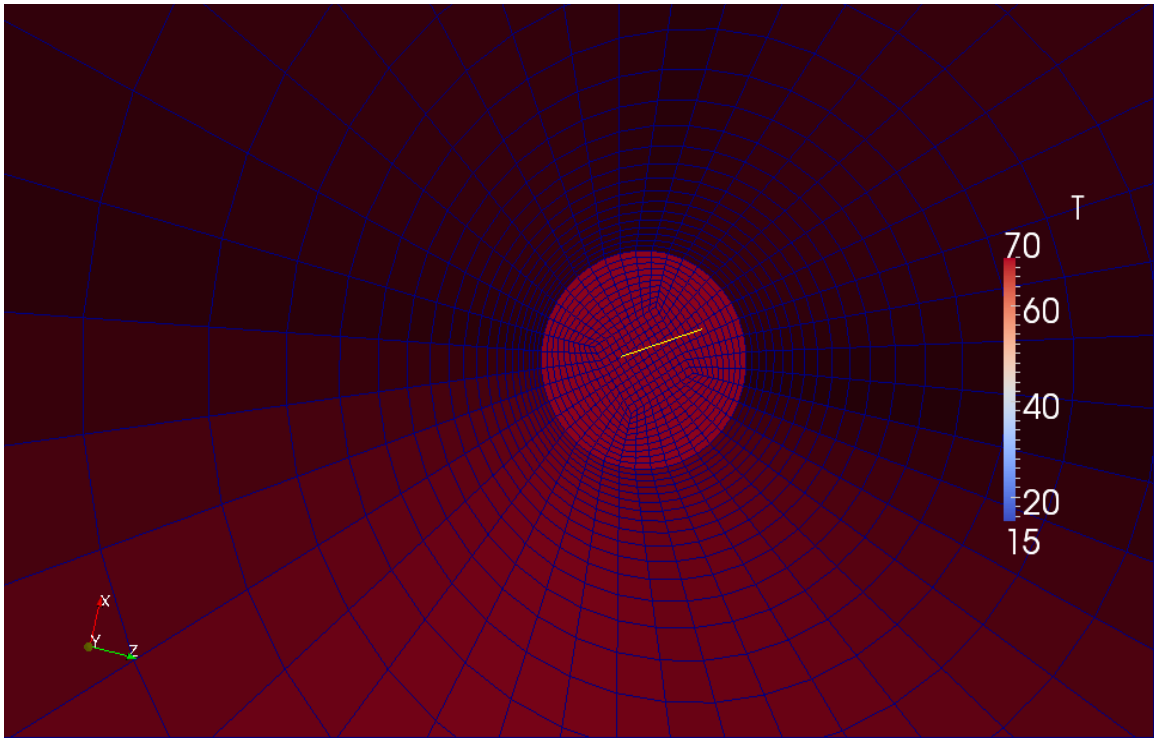


Figure 3: Internal Mesh View

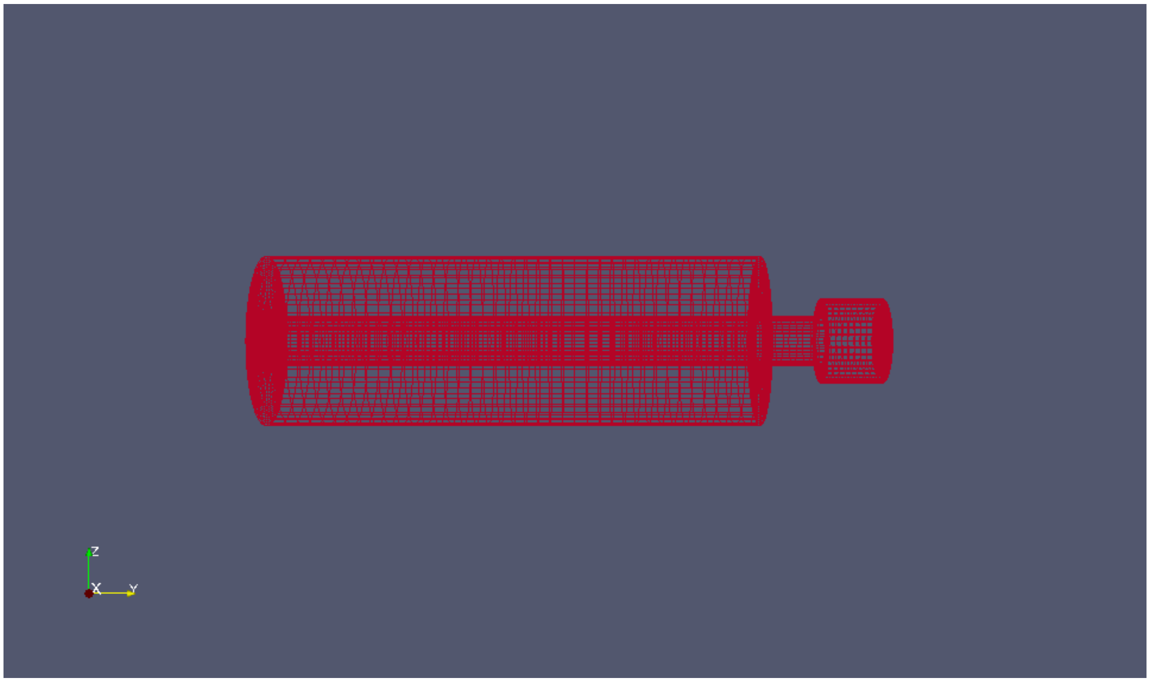


Figure 4: Wireframe Model

Boundary Conditions:

We applied different boundary conditions at different sections of the tubes using the data established by publishers previously. Temperature at the tube section was considered to be 70C as the radiation absorbed and insulation provided by the evacuated glass tubing accounted for a higher temperature at the tube boundary. Temperature of the bulb was considered to be 20C which is an estimate temperature of the water contained in the overhead sink. The boundary conditions file can be seen in Appendix A.

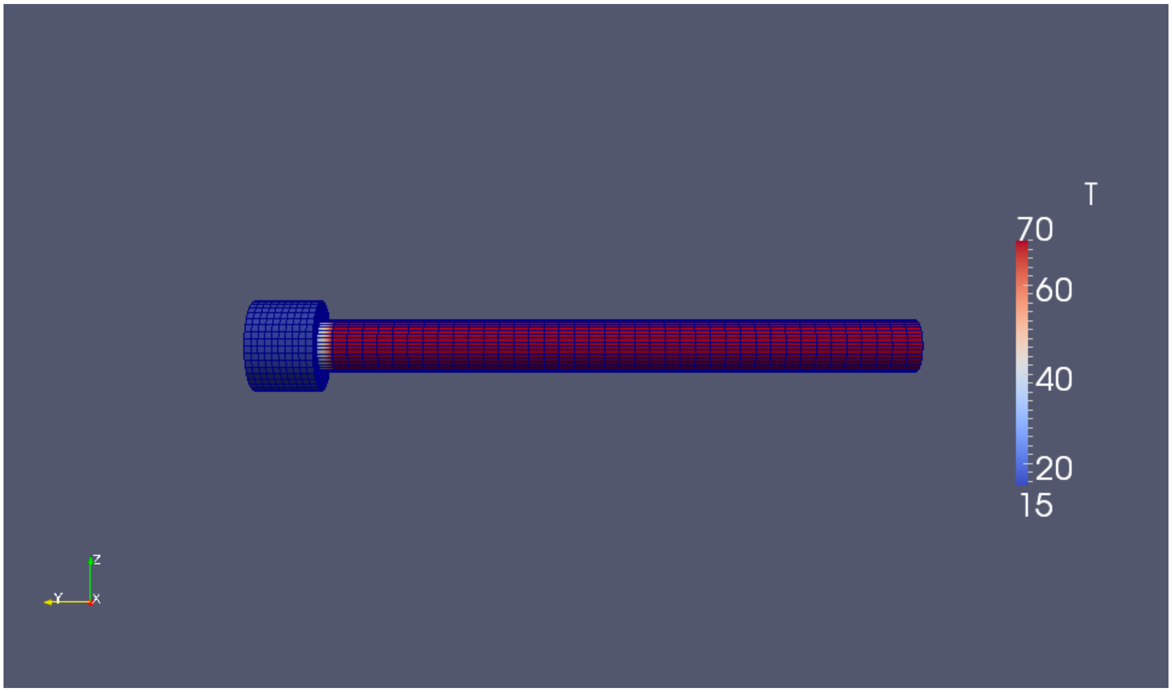


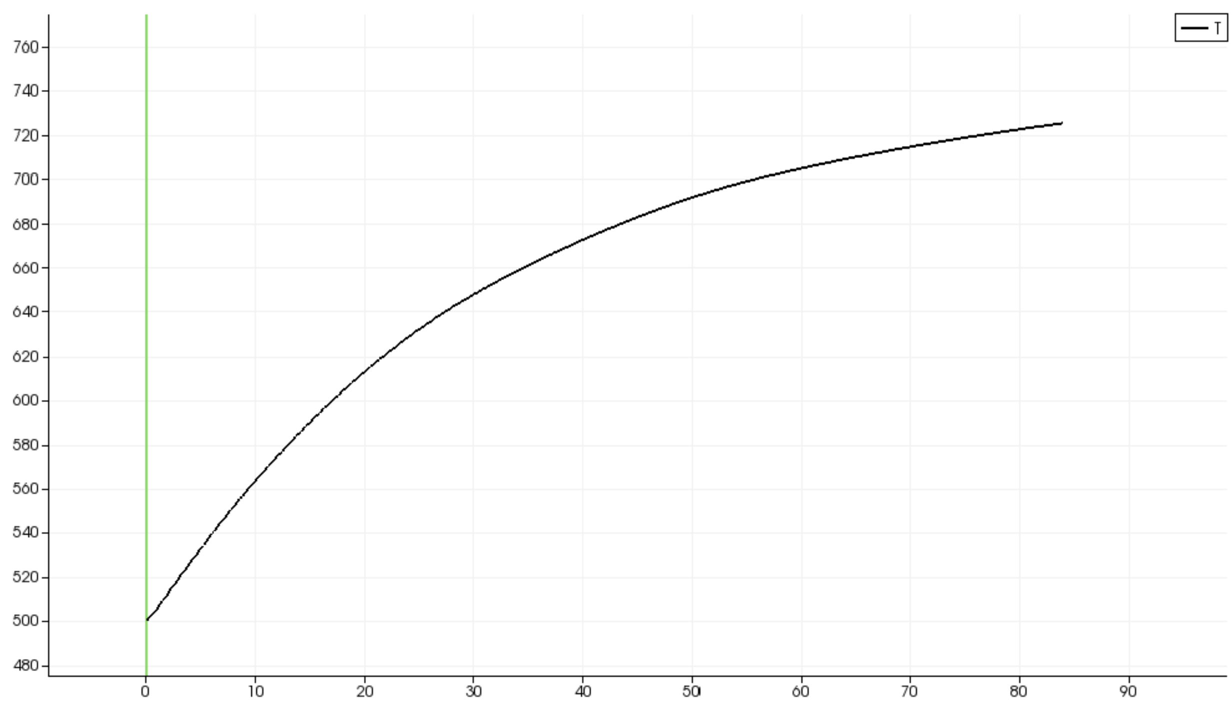
Figure:5 Initial Boundary Conditions at Pipe and Bulb Surface

Processing/Solution:

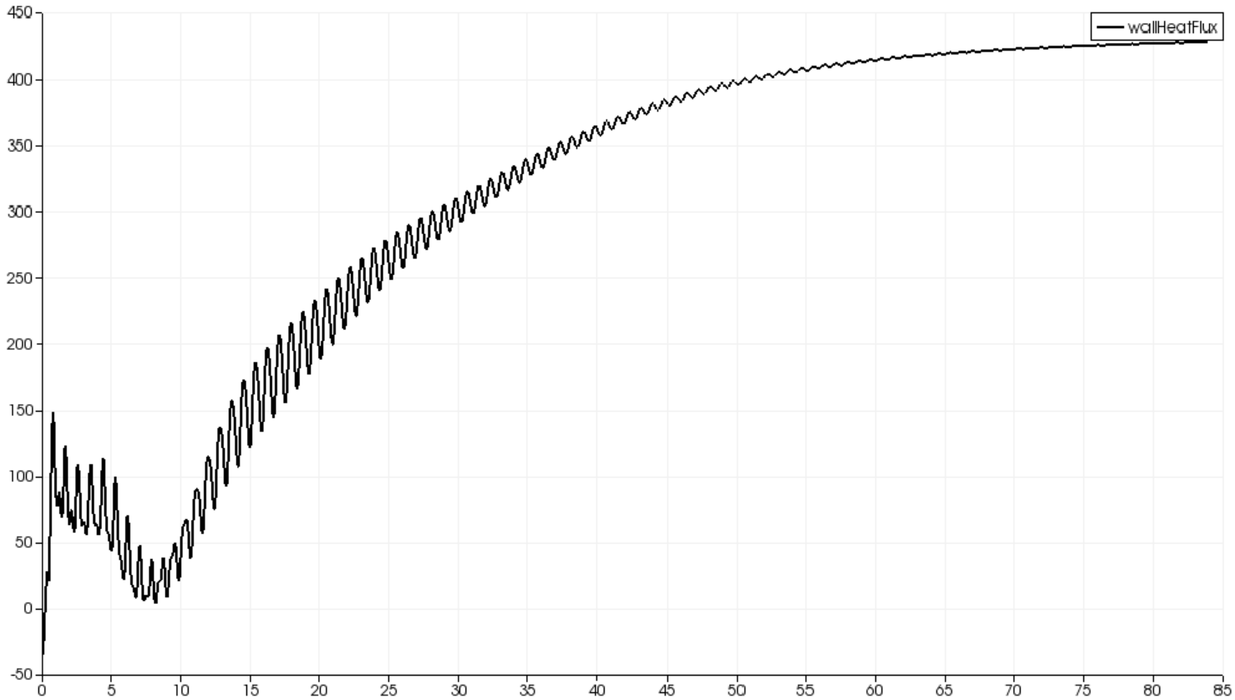
The solver used for processing the changes in temperature and pressure was buoyantPimpleFoam which is a transient solver for buoyant, turbulent flow of compressible fluid for ventilation and heat transfer. In our initial attempt, we faced certain errors such as the mesh skewness, orthagonality and aspect ratio, but these errors were later removed via the hit and trial method. Furthermore, our first processing attempt crashed as our courant number exploded. But later we removed this error by reducing deltaT to 0.01. The controlMeshDict file can be seen in Appendix A and the change in temperature results can be observed in Appendix B.

Post Processing:

In order to analyze the changing heat flux across the tube-bulb boundary, we downloaded the wallHeatFlux utility file from a CFD forum online. This utility file was integrated with OpenFOAM using wmake command. The file was run before the solver buoyantPimpleFoam so that it calculates values during the solution processing. After the solution was complete, the file was run again to compile the results of changing heat flux across the tube-bulb boundary. The results were obtained and graph was plotted by integrating the variables and slicing the geometry at the required area. The results can be observed in Appendix B.



Graph 1: Temperature Gradient



Graph 2: Heat Flux

APPENDIX A

BLOCK MESH DICT

```
/*---------------------------------*- C++ -*-----------------------------------*\
| =========                 |                                                 |
| \\      /  Field         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
version     2.0;
formatascii;
class       dictionary;
object      blockMeshDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

convertToMeters 1;

vertices
(
    (5 0 0)
    (7 0 0)
    (0 0 -7)
    (0 0 -5)
    (5 40 0)
    (7 40 0)
    (0 40 -7)
    (0 40 -5)
    (-7 0 0)
    (-5 0 0)
    (-7 40 0)
    (-5 40 0)
    (0 0 5)
    (0 0 7)
    (0 40 5)
    (0 40 7)  //15
    (2 0 0)
    (0 0 -2)
    (2 40 0)
```

```
    (0 40 -2)  //19
    (1 1 1) //garbage
    (-2 0 0)
    (1 1 1) //garbage
    (-2 40 0)
    (1 1 1)  //garbage
    (0 0 2)
    (1 1 1)  //garbage
    (0 40 2)    //27
    (0 0 1)
    (1 0 0)
    (0 0 -1)
    (-1 0 0)
    (0 50 1)
    (1 50 0)
    (0 50 -1)
    (-1 50 0)
    (0 50 2)
    (2 50 0)
    (0 50 -2)
    (-2 50 0)
    (0 45 2)
    (2 45 0)
    (0 45 -2)
    (-2 45 0)    //43
    (0 45 3.5)
    (3.5 45 0)
    (0 45 -3.5)
    (-3.5 45 0)
    (0 50 3.5)
    (3.5 50 0)
    (0 50 -3.5)
    (-3.5 50 0)
    (0 45 1)
    (1 45 0)
    (0 45 -1)
    (-1 45 0)

);

blocks
(
hex (0 1 2 3 4 5 6 7) (10 10 40) simpleGrading (1 1 1)
hex (8 9 3 2 10 11 7 6) (10 10 40) simpleGrading (1 1 1)
hex (13 12 9 8 15 14 11 10) (10 10 40) simpleGrading (1 1 1)
hex (12 13 1 0 14 15 5 4) (10 10 40) simpleGrading (1 1 1)
```

```
hex (16 0 3 17 18 4 7 19) (10 10 40) simpleGrading (1 1 1)
hex (9 21 17 3 11 23 19 7) (10 10 40) simpleGrading (1 1 1)
hex (12 25 21 9 14 27 23 11) (10 10 40) simpleGrading (1 1 1)
hex (12 0 16 25 14 4 18 27) (10 10 40) simpleGrading (1 1 1)
hex (53 41 42 54 33 37 38 34) (10 10 10) simpleGrading (1 1 1)
hex (43 55 54 42 39 35 34 38) (10 10 10) simpleGrading (1 1 1)
hex (40 52 55 43 36 32 35 39) (10 10 10) simpleGrading (1 1 1)
hex (40 41 53 52 36 37 33 32) (10 10 10) simpleGrading (1 1 1)
hex (52 53 54 55 32 33 34 35) (10 10 10) simpleGrading (1 1 1)
hex (41 45 46 42 37 49 50 38) (10 10 10) simpleGrading (1 1 1)
hex (42 46 47 43 38 50 51 39) (10 10 10) simpleGrading (1 1 1)
hex (44 40 43 47 48 36 39 51) (10 10 10) simpleGrading (1 1 1)
hex (44 45 41 40 48 49 37 36) (10 10 10) simpleGrading (1 1 1)
hex (29 16 17 30 53 41 42 54) (10 10 40) simpleGrading (1 1 1)
hex (21 31 30 17 43 55 54 42) (10 10 40) simpleGrading (1 1 1)
hex (25 28 31 21 40 52 55 43) (10 10 40) simpleGrading (1 1 1)
hex (25 16 29 28 40 41 53 52) (10 10 40) simpleGrading (1 1 1)
hex (28 29 30 31 52 53 54 55) (10 10 40) simpleGrading (1 1 1)

);

edges
(
arc 0 3 (3.535533906 0 -3.535533906)
arc 1 2 (4.949747468 0 -4.949747468)
arc 4 7 (3.535533906 40 -3.535533906)
arc 5 6 (4.949747468 40 -4.949747468)
arc 2 8 (-4.949747468 0 -4.949747468)
arc 3 9 (-3.535533906 0 -3.535533906)
arc 6 10 (-4.949747468 40 -4.949747468)
arc 7 11 (-3.535533906 40 -3.535533906)
arc 14 11 (-3.535533906 40 3.535533906)
arc 15 10 (-4.949747468 40 4.949747468)
arc 12 9 (-3.535533906 0 3.535533906)
arc 13 8 (-4.949747468 0 4.949747468)
arc 4 14 (3.535533906 40 3.535533906)
arc 5 15 (4.949747468 40 4.949747468)
arc 0 12 (3.535533906 0 3.535533906)
arc 1 13 (4.949747468 0 4.949747468)
arc 16 17 (1.414213562 0 -1.414213562)
arc 18 19 (1.414213562 40 -1.414213562)
arc 17 21 (-1.414213562 0 -1.414213562)
arc 19 23 (-1.414213562 40 -1.414213562)
arc 21 25 (-1.414213562 0 1.414213562)
arc 23 27 (-1.414213562 40 1.414213562)
arc 16 25 (1.414213562 0 1.414213562)
```

```
arc 18 27 (1.414213562 40 1.414213562)
arc 36 37 (1.414213562 50 1.414213562)
arc 37 38 (1.414213562 50 -1.414213562)
arc 38 39 (-1.414213562 50 -1.414213562)
arc 39 36 (-1.414213562 50 1.414213562)
arc 40 41 (1.414213562 45 1.414213562)
arc 41 42 (1.414213562 45 -1.414213562)
arc 42 43 (-1.414213562 45 -1.414213562)
arc 43 40 (-1.414213562 45 1.414213562)
arc 44 45 (2.474873734 45 2.474873734)
arc 45 46 (2.474873734 45 -2.474873734)
arc 46 47 (-2.474873734 45 -2.474873734)
arc 47 44 (-2.474873734 45 2.474873734)
arc 48 49 (2.474873734 50 2.474873734)
arc 49 50 (2.474873734 50 -2.474873734)
arc 50 51 (-2.474873734 50 -2.474873734)
arc 51 48 (-2.474873734 50 2.474873734)



);

boundary
(

pipe
   {
type wall;
faces
      (
         (40 41 16 25)
         (41 42 17 16)
         (42 43 21 17)
         (43 40 25 21)
         (16 17 30 29)
         (21 31 30 17)
         (21 25 28 31)
         (25 16 29 28)
         (28 29 30 31)
         );
   }

bulb
   {
type wall;
faces
```

```
        (
            (49 50 46 45)
            (50 51 47 46)
            (51 48 44 47)
            (48 49 45 44)
            (44 45 41 40)
            (41 45 46 42)
            (46 47 43 42)
            (47 44 40 43)
            (35 34 33 32)
            (34 38 37 33)
            (33 37 36 32)
            (39 35 32 36)
            (39 38 34 35)
            (38 50 49 37)
            (37 49 48 36)
            (39 36 48 51)
            (38 39 51 50)
        );
    }
 /*   fixedWalls
    {
type wall;
faces
        (
            (0 4 7 3)
            (2 6 5 1)
            (0 3 2 1)
            (4 5 6 7)
        );
    } */
);
mergePatchPairs
(
);

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

BLOCK MESH

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  Field          | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 2.1.1-221db2718bbb
Exec   : blockMesh
Date   : Jan 08 2014
Time   : 23:50:44
Host   : "custom"
PID    : 6098
Case   : /home/custom/buoyantPimpleFoam/hotRoom
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Creating block mesh from
    "/home/custom/buoyantPimpleFoam/hotRoom/constant/polyMesh/blockMeshDict"
Creating curved edges
Creating topology blocks
Creating topology patches

Creating block mesh topology
--> FOAM Warning :
    From function polyMesh::polyMesh(... construct from shapes...)
in file meshes/polyMesh/polyMeshFromShapeMesh.C at line 888
    Found 24 undefined faces in mesh; adding to default patch.

Check topology

        Basic statistics
                Number of internal faces : 41
                Number of boundary faces : 50
                Number of defined boundary faces : 50
                Number of undefined boundary faces : 0
```

```
        Checking patch -> block consistency

Creating block offsets
Creating merge list .

Creating polyMesh from blockMesh
Creating patches
Creating cells
Creating points with scale 1

Writing polyMesh
----------------
Mesh Information
----------------
boundingBox: (-7 0 -7) (7 50 7)
nPoints: 65411
nCells: 61000
nFaces: 187300
nInternalFaces: 178700
----------------
Patches
----------------
patch 0 (start: 178700 size: 2100) name: pipe
patch 1 (start: 180800 size: 1700) name: bulb
patch 2 (start: 182500 size: 4800) name: defaultFaces

End
```

CHECK MESH

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  Field         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 2.1.1-221db2718bbb
Exec   : checkMesh
Date   : Jan 08 2014
Time   : 23:51:38
Host   : "custom"
PID    : 6139
Case   : /home/custom/buoyantPimpleFoam/hotRoom
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create polyMesh for time = 0

Time = 0

Mesh stats
points:           65411
faces:            187300
internal faces:   178700
cells:            61000
boundary patches: 3
point zones:      0
face zones:       0
cell zones:       0

Overall number of cells of each type:
hexahedra:     61000
prisms:        0
wedges:        0
pyramids:      0
tet wedges:    0
tetrahedra:    0
```

```
polyhedra:     0

Checking topology...
    Boundary definition OK.
 ***Total number of faces on empty patches is not divisible by the number of cells in the mesh.
Hence this mesh is not 1D or 2D.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
   *Number of regions: 2
    The mesh has multiple regions which are not connected by any face.
<<Writing region information to "0/cellToRegion"

Checking patch topology for multiply connected surfaces ...
    Patch              Faces    Points   Surface topology
pipe               2100     2121     ok (non-closed singly connected)
bulb               1700     1721     ok (non-closed singly connected)
defaultFaces       4800     4800     ok (closed singly connected)

Checking geometry...
    Overall domain bounding box (-7 0 -7) (7 50 7)
    Mesh (non-empty, non-wedge) directions (0 0 0)
    Mesh (non-empty) directions (0 0 0)
 ***Number of edges not aligned with or perpendicular to non-empty directions: 122300
<<Writing 65411 points on non-aligned edges to set nonAlignedEdges
    Boundary openness (-2.53666e-16 1.09409e-16 2.46416e-16) OK.
    Max cell openness = 1.4252e-16 OK.
    Max aspect ratio = 0 OK.
Minumum face area = 0.0125056. Maximum face area = 1.09843.  Face area magnitudes OK.
    Min volume = 0.00625279. Max volume = 0.227612.  Total volume = 6386.44.  Cell volumes
OK.
    Mesh non-orthogonality Max: 31.4636 average: 4.3786
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 0.96984 OK.
    Coupled point location match (average 0) OK.

Failed 1 mesh checks.

End
```

Boundary Conditions:

```
/*--------------------------------*- C++ -*----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      T;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

dimensions      [0 0 0 1 0 0 0];

internalField   uniform 40;

boundaryField
{
    pipe
    {
        type            fixedValue;
        value           uniform 70;
    }

    bulb
    {
        type            fixedValue;
        value           uniform 15;
    }

   /* fixedWalls
    {
        type            zeroGradient;
    }
*/
}

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

BLOCK MESH WITH BOUNDARY CONDITIONS

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  Field          | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 2.1.1-221db2718bbb
Exec   : blockMesh
Date   : Jan 08 2014
Time   : 23:52:39
Host   : "custom"
PID    : 6146
Case   : /home/custom/buoyantPimpleFoam/hotRoom
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Creating block mesh from
    "/home/custom/buoyantPimpleFoam/hotRoom/constant/polyMesh/blockMeshDict"
Creating curved edges
Creating topology blocks
Creating topology patches

Creating block mesh topology

Check topology

	Basic statistics
		Number of internal faces : 29
		Number of boundary faces : 26
		Number of defined boundary faces : 26
		Number of undefined boundary faces : 0
	Checking patch -> block consistency

Creating block offsets
Creating merge list .
```

Creating polyMesh from blockMesh
Creating patches
Creating cells
Creating points with scale 1

Writing polyMesh

Mesh Information
----
boundingBox: (-3.5 0 -3.5) (3.5 50 3.5)
nPoints: 30971
nCells: 29000
nFaces: 88900
nInternalFaces: 85100

Patches
----
patch 0 (start: 85100 size: 2100) name: pipe
patch 1 (start: 87200 size: 1700) name: bulb

End

CHECK MESH WITH BOUNDARY CONDITIONS

```
/*---------------------------------------------------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
Build  : 2.1.1-221db2718bbb
Exec   : checkMesh
Date   : Jan 08 2014
Time   : 23:52:47
Host   : "custom"
PID    : 6147
Case   : /home/custom/buoyantPimpleFoam/hotRoom
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Disallowing user-supplied system call operations

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //
Create time

Create polyMesh for time = 0

Time = 0

Mesh stats
points:           30971
faces:            88900
internal faces:   85100
cells:            29000
boundary patches: 2
point zones:      0
face zones:       0
cell zones:       0

Overall number of cells of each type:
hexahedra:     29000
prisms:        0
wedges:        0
pyramids:      0
```

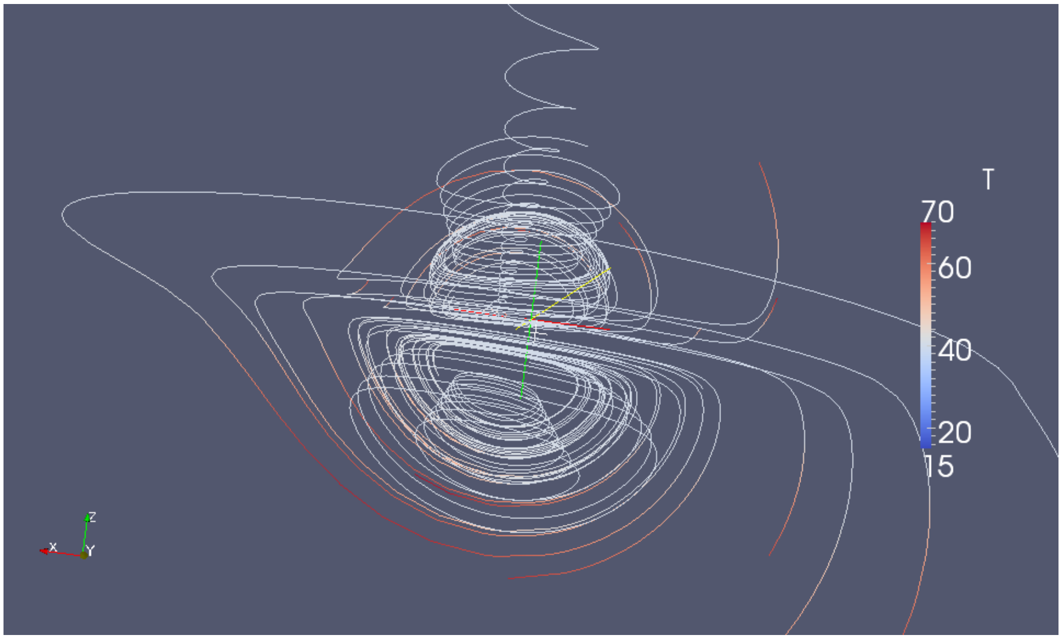
```
tet wedges:    0
tetrahedra:    0
polyhedra:     0

Checking topology...
    Boundary definition OK.
    Cell to face addressing OK.
    Point usage OK.
    Upper triangular ordering OK.
    Face vertices OK.
    Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces ...
    Patch             Faces    Points   Surface topology
pipe              2100     2121     ok (non-closed singly connected)
bulb              1700     1721     ok (non-closed singly connected)

Checking geometry...
    Overall domain bounding box (-3.5 0 -3.5) (3.5 50 3.5)
    Mesh (non-empty, non-wedge) directions (1 1 1)
    Mesh (non-empty) directions (1 1 1)
    Boundary openness (-4.88178e-16 4.9998e-16 7.73934e-16) OK.
    Max cell openness = 1.4252e-16 OK.
    Max aspect ratio = 17.992 OK.
Minumum face area = 0.0125056. Maximum face area = 0.353066.  Face area magnitudes OK.
    Min volume = 0.00625279. Max volume = 0.0440642.  Total volume = 754.796.  Cell
volumes OK.
    Mesh non-orthogonality Max: 31.4636 average: 6.34672
    Non-orthogonality check OK.
    Face pyramids OK.
    Max skewness = 0.96984 OK.
    Coupled point location match (average 0) OK.
Mesh OK.

End
```

CONTROL DICT

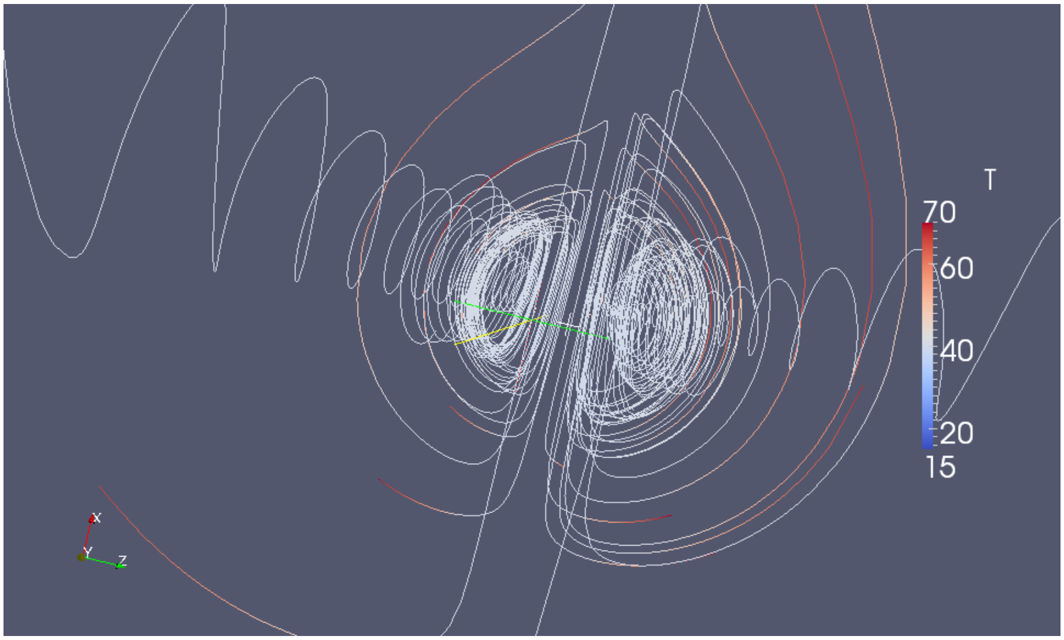
```
/*---------------------------------*- C++ -*-----------------------------------*\
| =========                 |                                                 |
| \\      /  F ield         | OpenFOAM: The Open Source CFD Toolbox           |
|  \\    /   O peration     | Version:  2.1.1                                 |
|   \\  /    A nd           | Web:      www.OpenFOAM.org                      |
|    \\/     M anipulation  |                                                 |
\*---------------------------------------------------------------------------*/
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    location    "system";
    object      controlDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

application     buoyantPimpleFoam;

startFrom       startTime;

startTime       0;

stopAt          endTime;

endTime         20000;

deltaT          0.001;

writeControl    timeStep;

writeInterval   50;

purgeWrite      0;

writeFormat     ascii;

writePrecision  6;

writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;
```

```
adjustTimeStep  no;

maxCo           0.5;

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

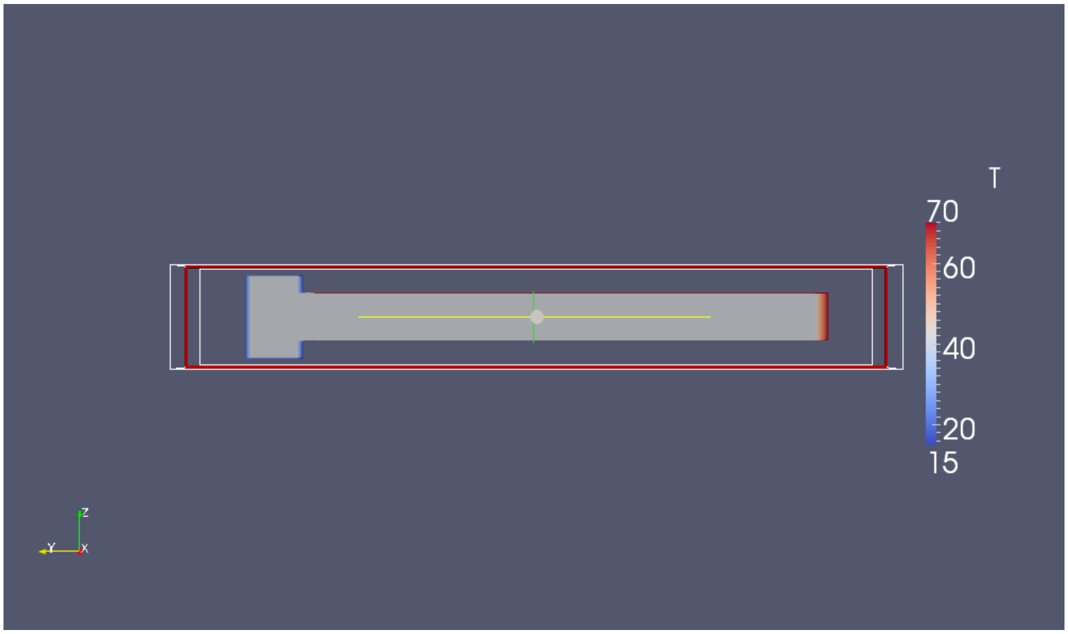
Appendix B



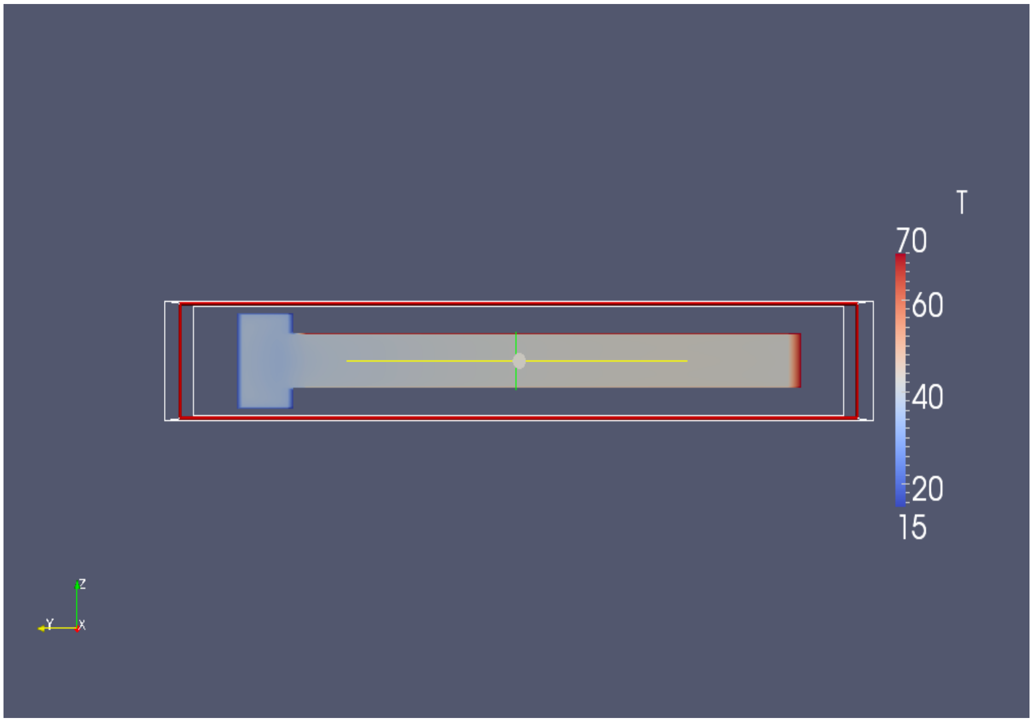
Stremlines (Top View)



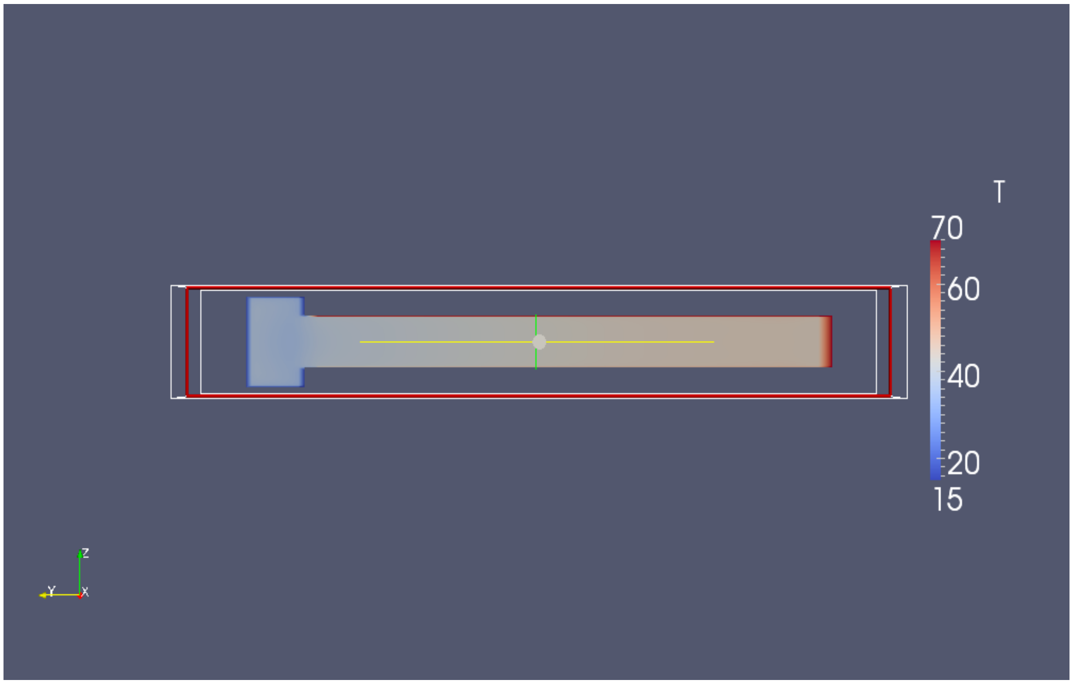
Streamlines (Bottom View)



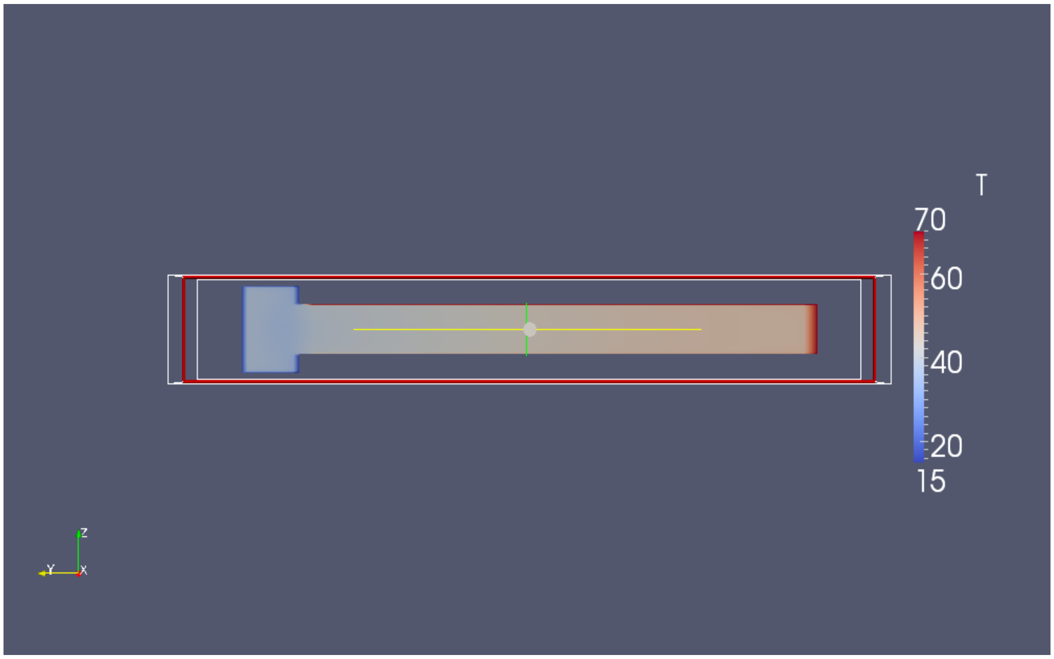
Temperature Profile t=0 seconds



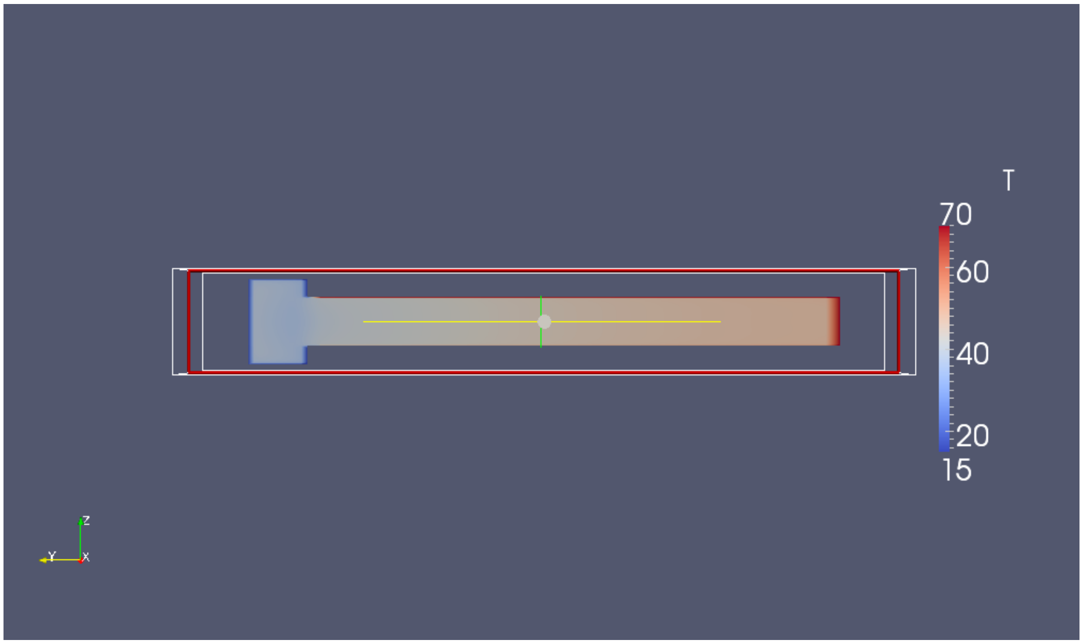
Temperature Profile t=50 seconds



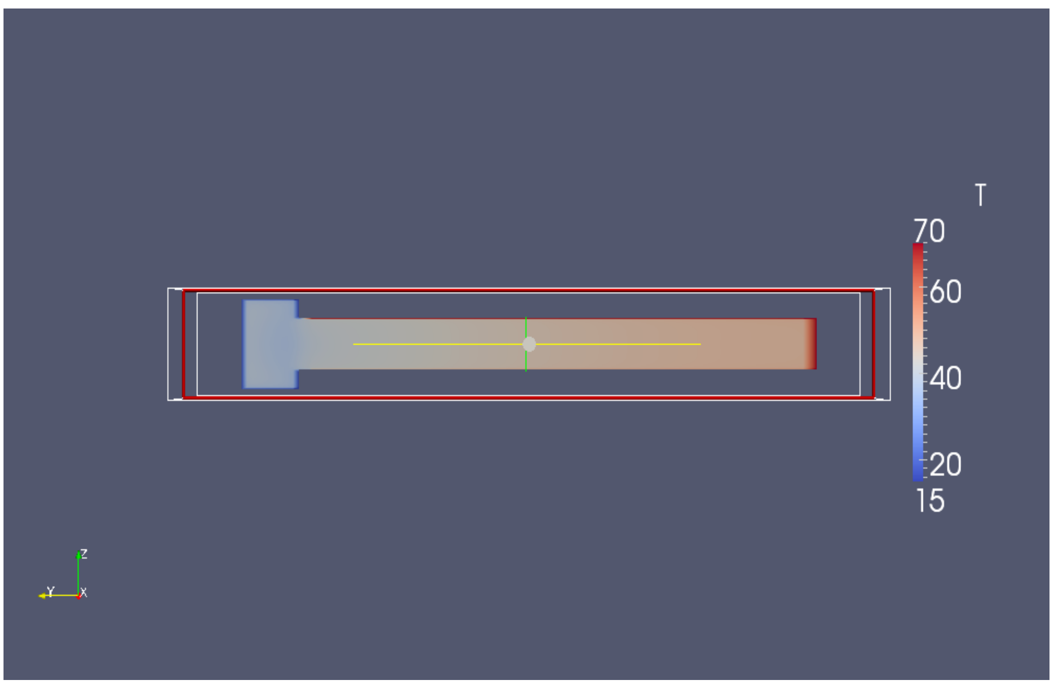
Temperature Profile t=100 seconds



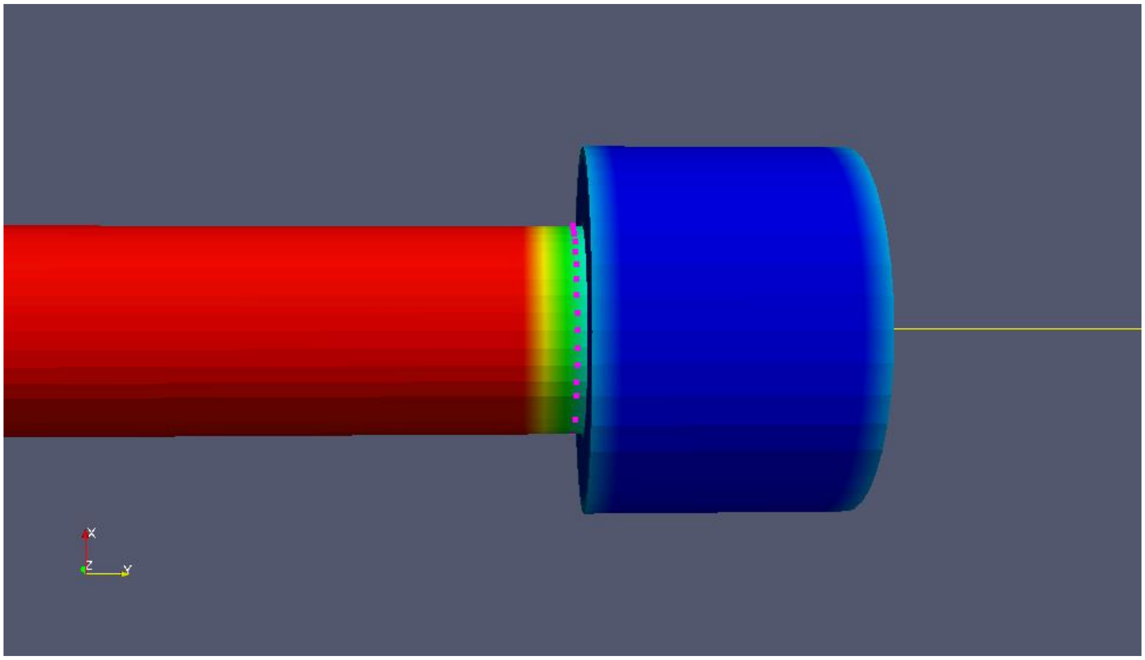
Temperature Profile t=150 seconds



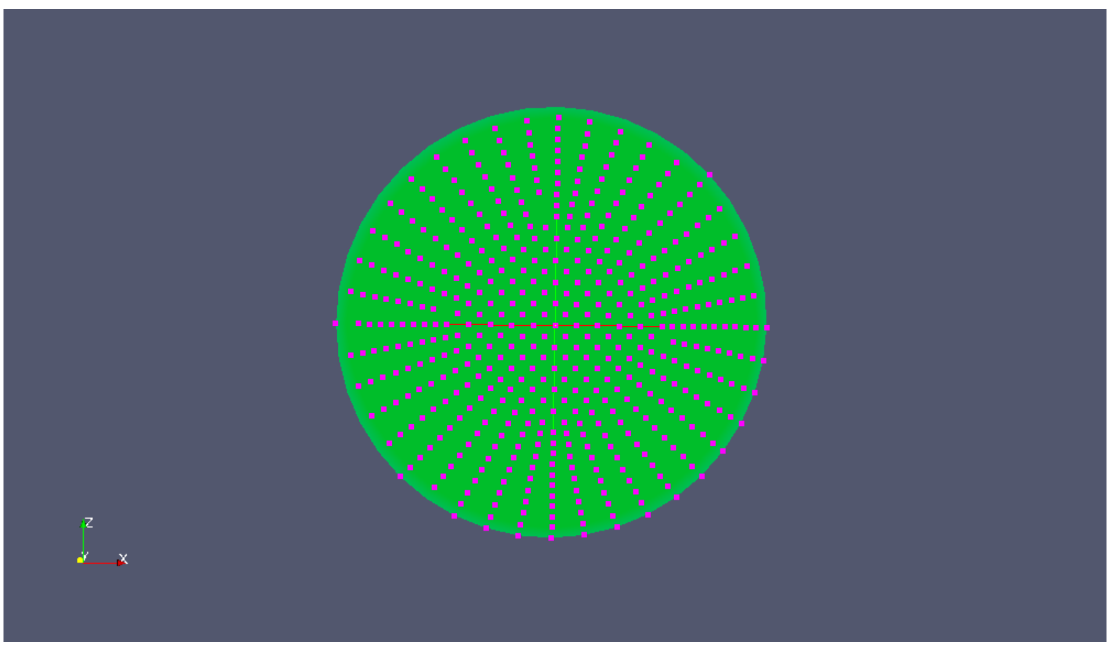
Temperature Profile t=200 seconds



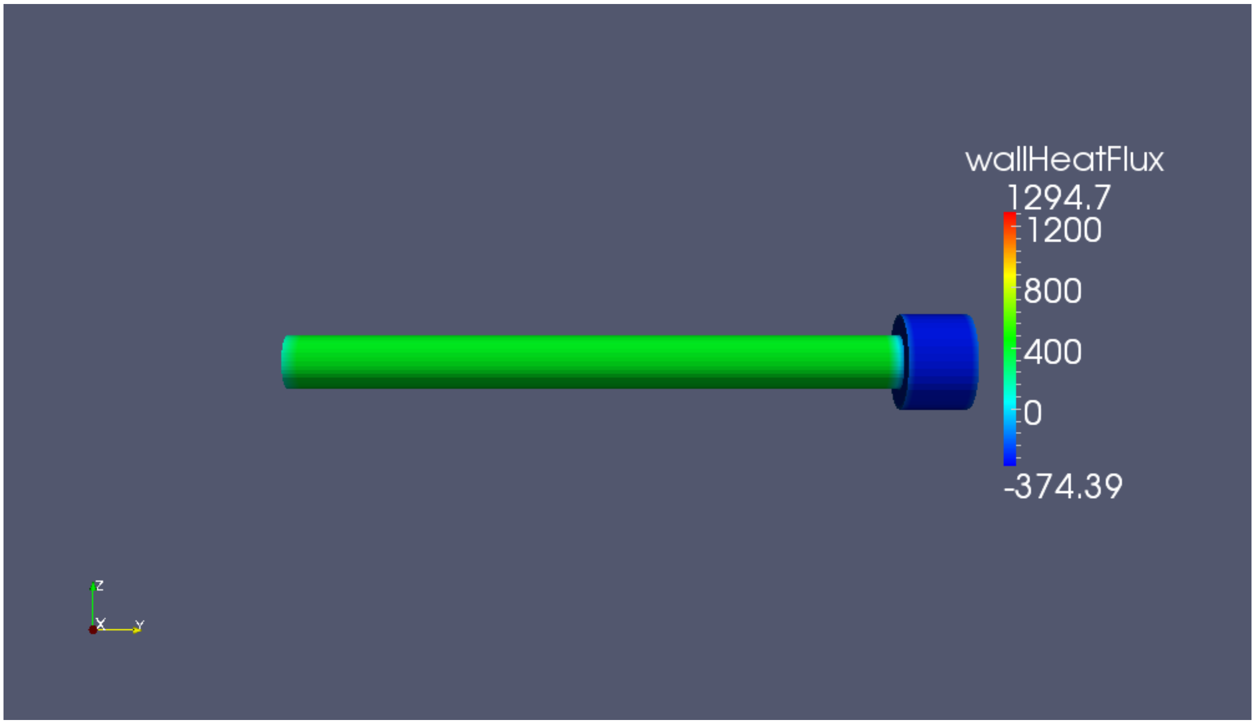
Temperature Profile t=250 seconds



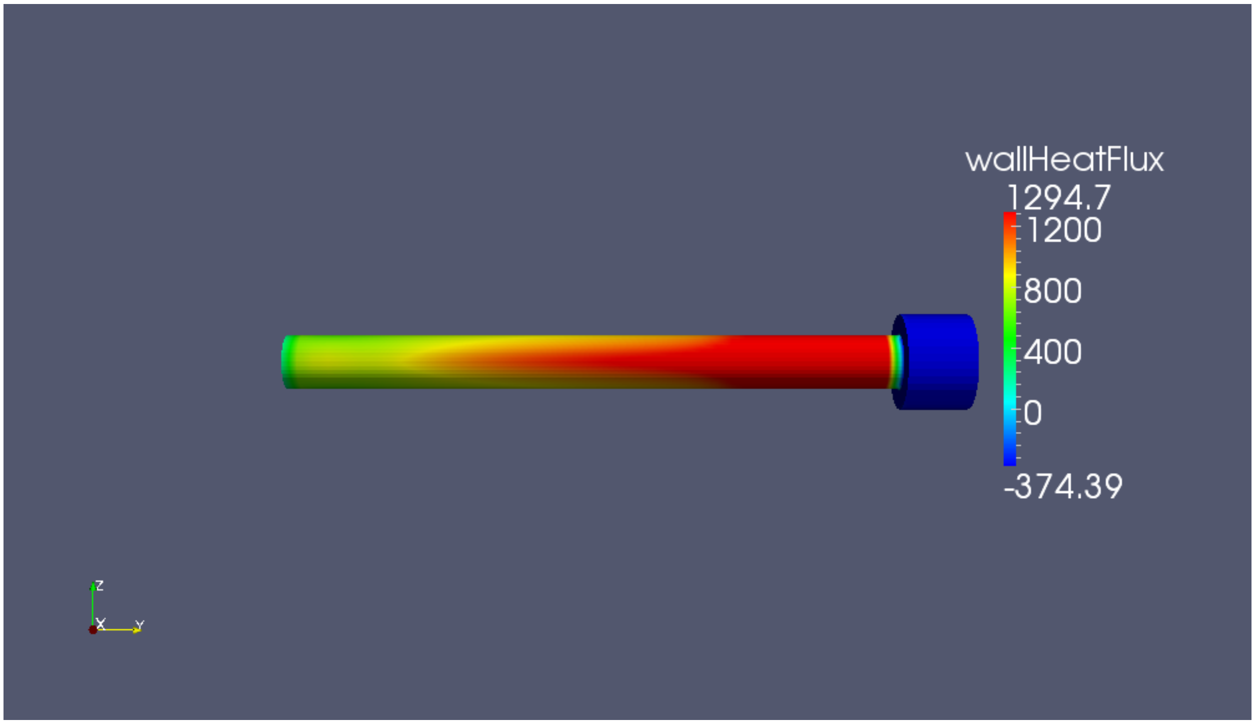
Sliced Section



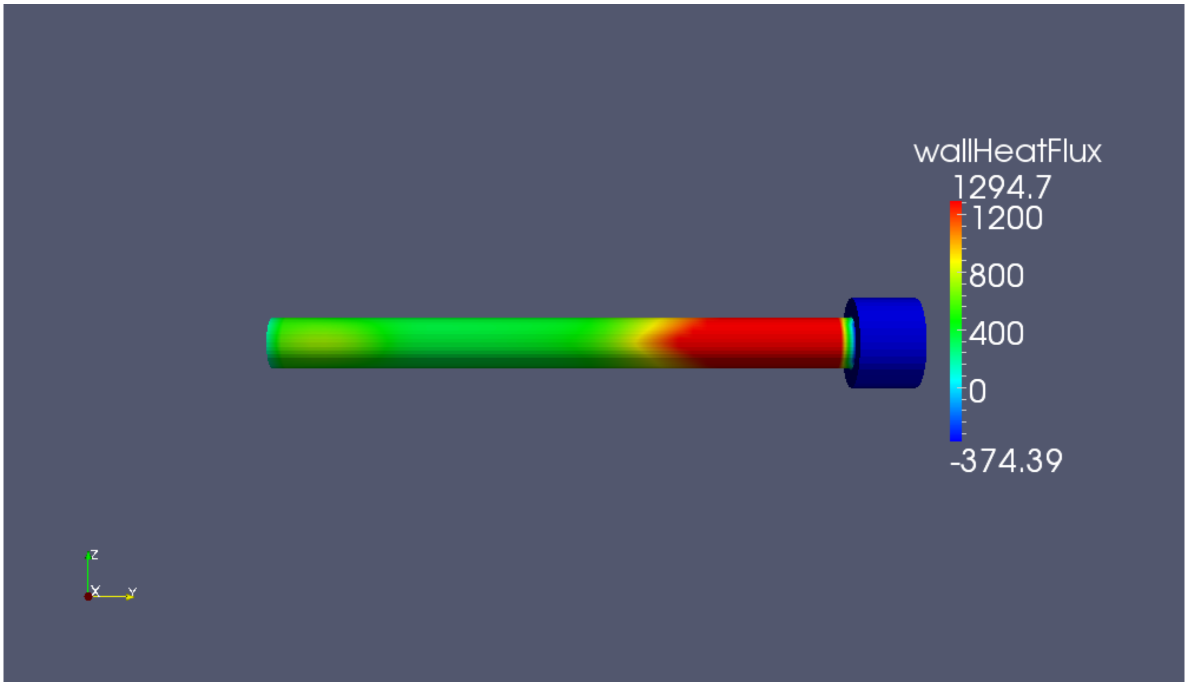
Sliced Plane



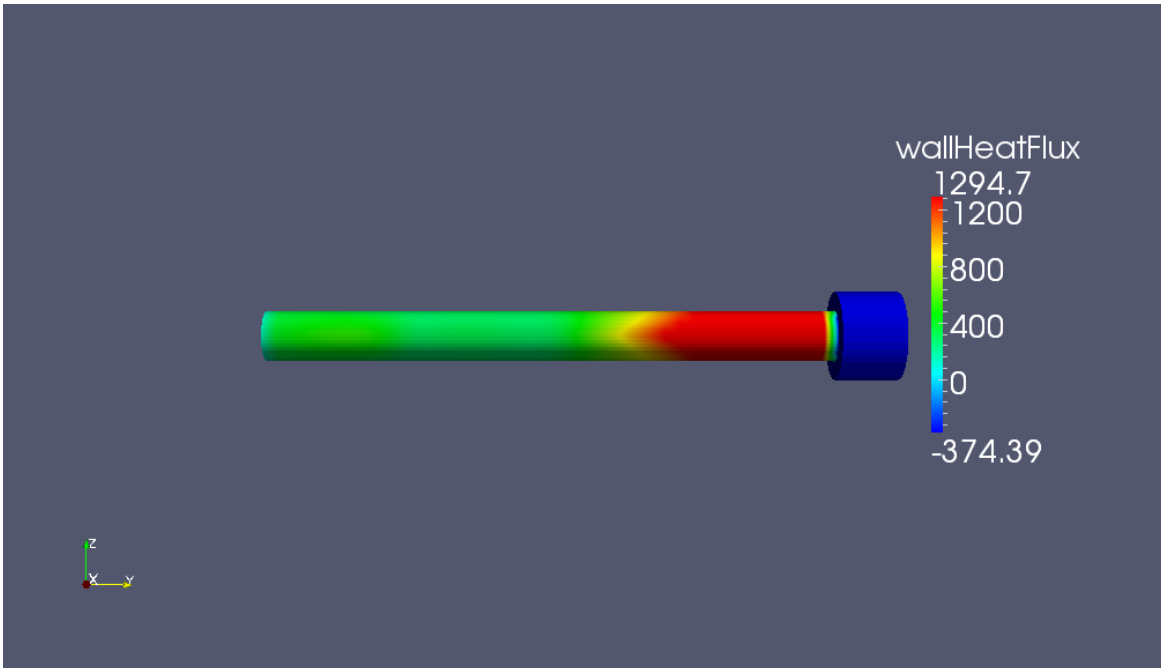
Heat Flux Profile t= 0 seconds



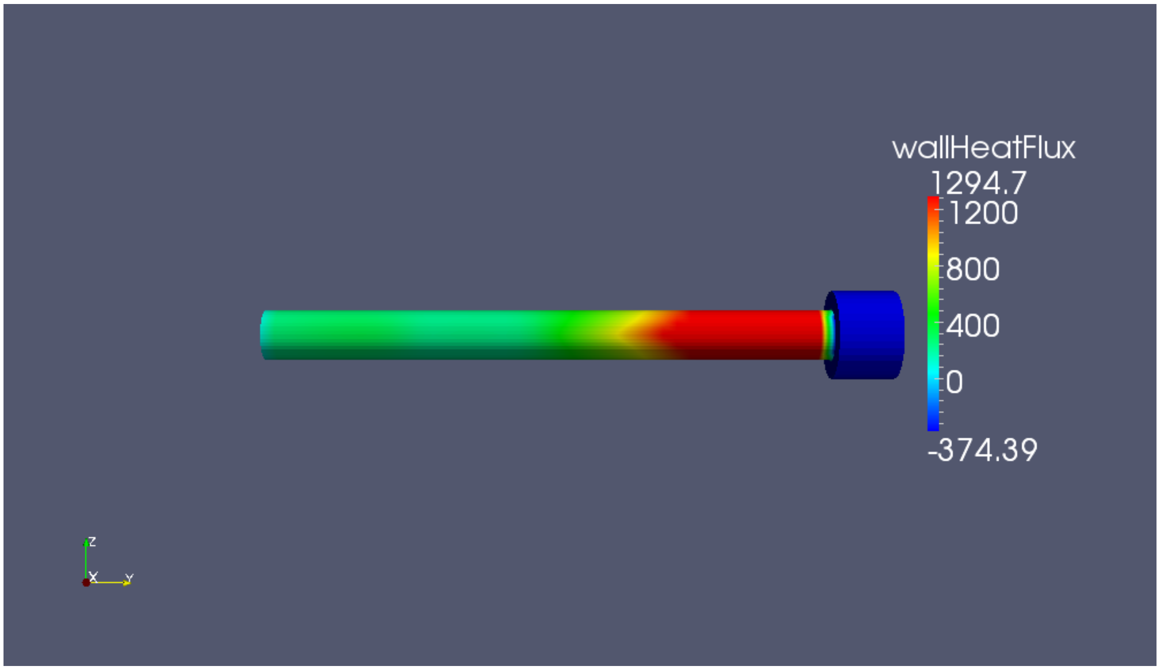
Heat Flux Profile t= 20 seconds



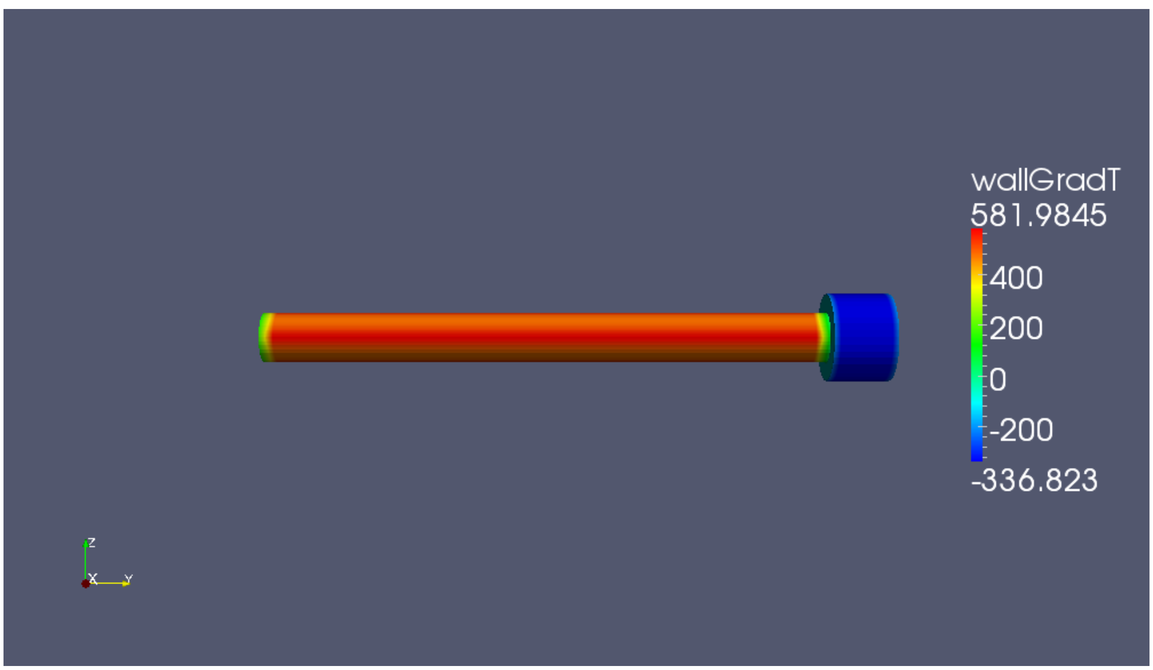
Heat Flux Profile t= 40 seconds



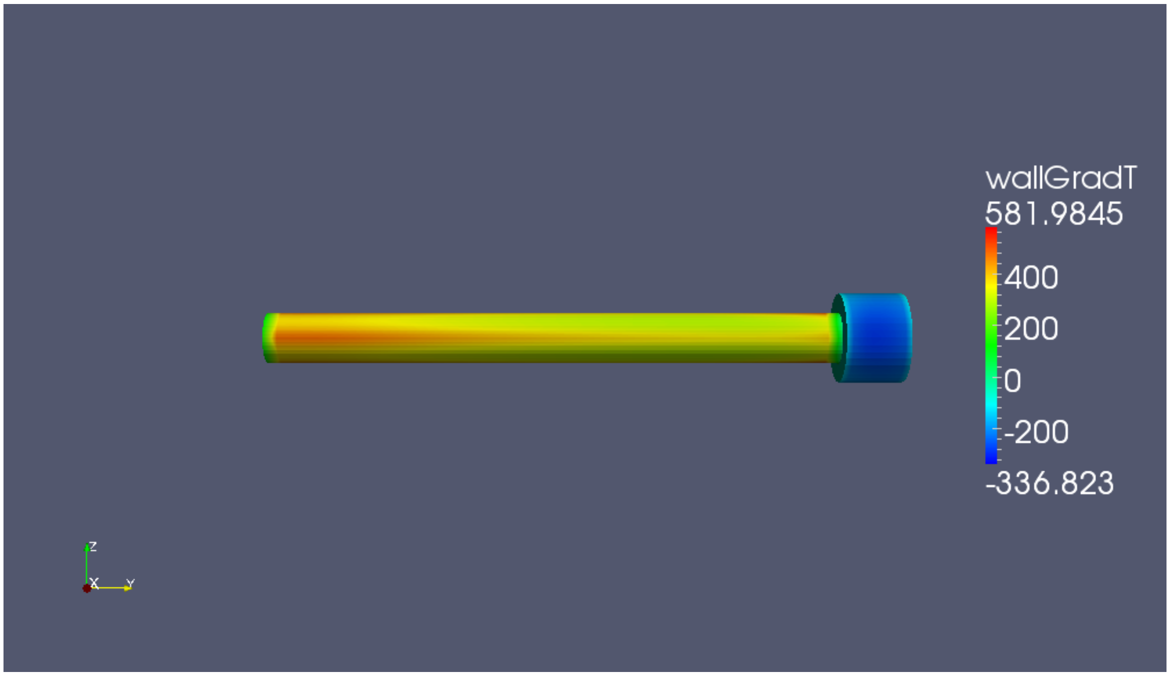
Heat Flux Profile t= 60 seconds



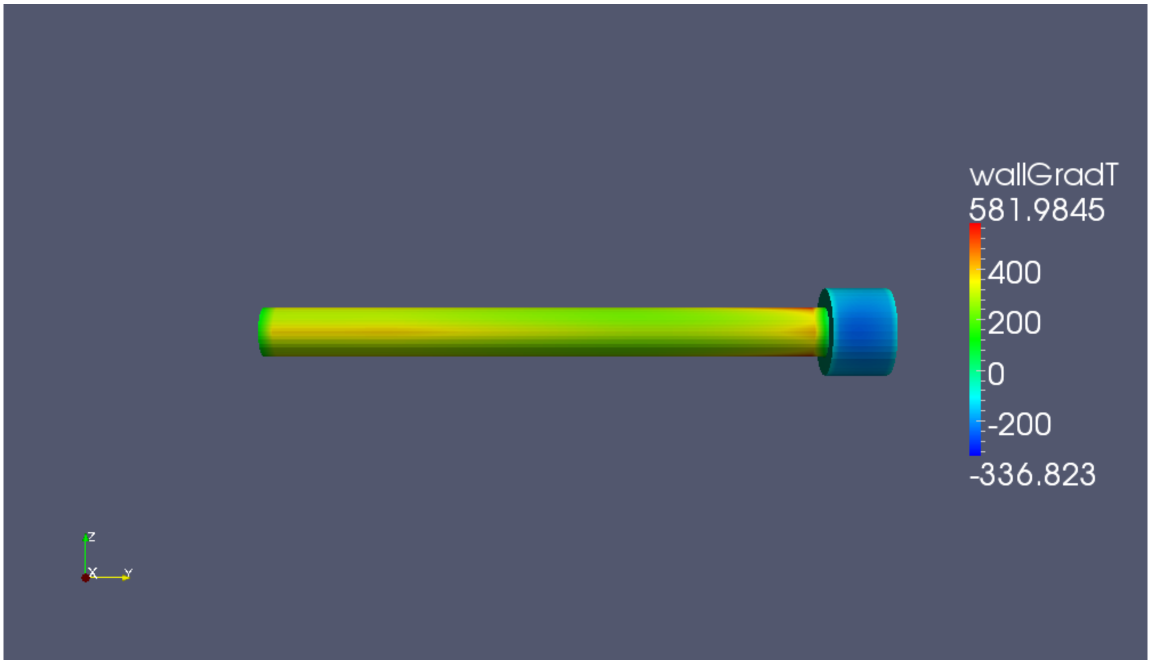
Heat Flux Profile t= 80 seconds



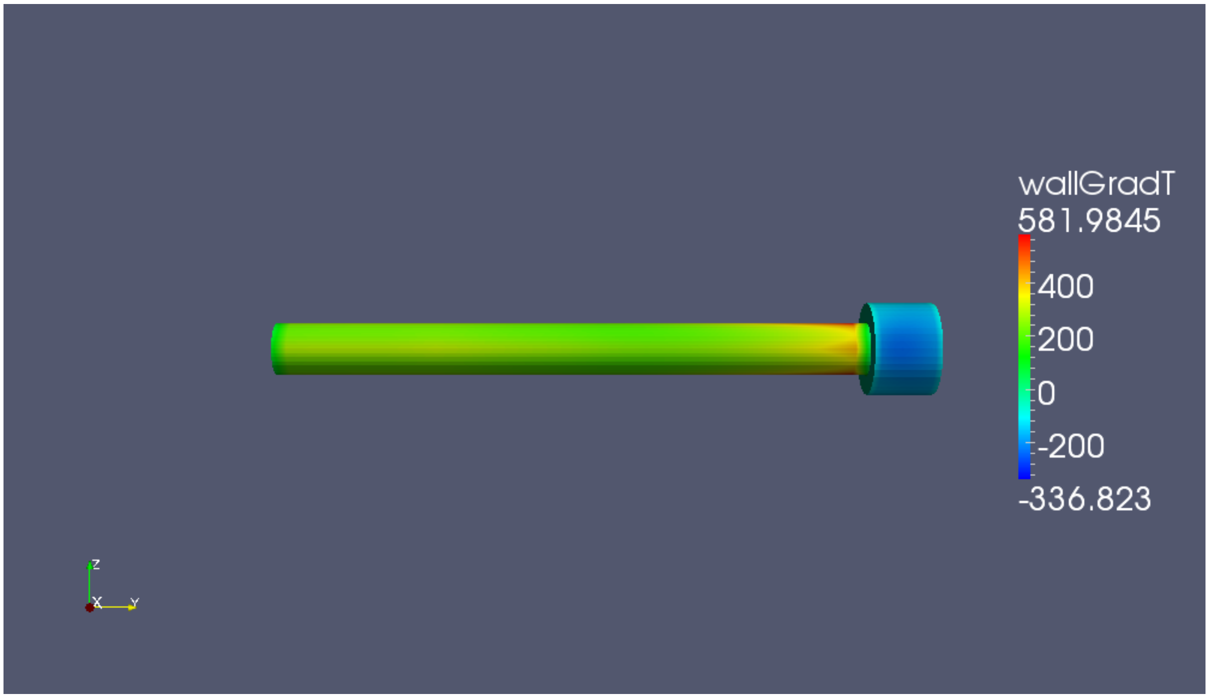
Temperature Gradient Profile t=0 seconds



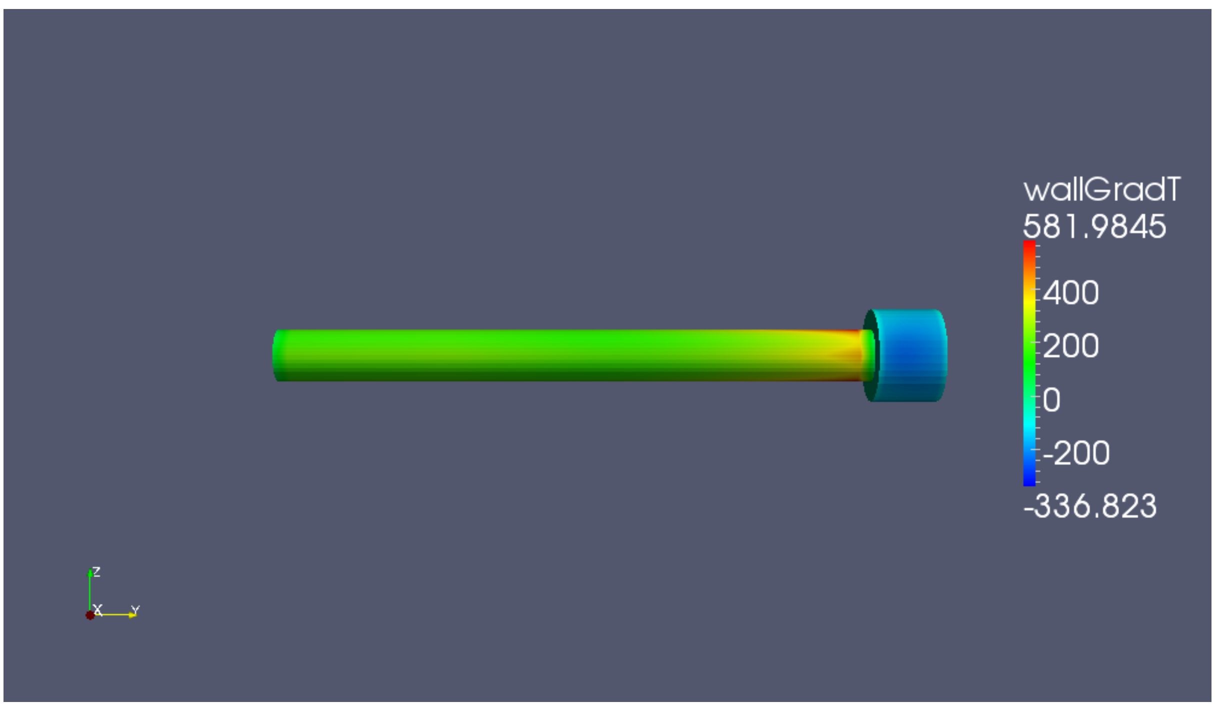
Temperature Gradient Profile t=20 seconds



Temperature Gradient Profile t=40 seconds



Temperature Gradient Profile t=60 seconds



Temperature Gradient Profile t=80 seconds

References

- http://www.reuk.co.uk/Evacuated-Tube-Solar-Water-Heating.htm