Department of Chemical Engineering, IIT Bombay
Instruction Manual for Heat Transfer problem using OpenFOAM
CFD Lab, Semester-I, Year 2017-18
Heat Transfer in a Solid Rod

Objective

To create the geometry of a pipe in blockMesh, mesh it and simulate heat transfer over it using OpenFoam.

Geometry and Conditions

Straight horizontal pipe

Radius of the rod (r) = 1 cm

Length of the rod (L) = 30 cm

Rod is aligned along the x-axis

Material of the rod : Copper.

Temperature at x=0 : 310+(10 * Group Number) K

Temperature at x=L : 310 K

Wall is fixed at temperature or Kept at Constant Heat Flux

Volumetric rate of heat generation (q) = $1 * 10^7 W/m^3$

Steps for Geometry Creation

1. In OpenFoam we can create simple geometries using the blockMesh utility.
2. Case file of OpenFoam contains three folders namely: **0, constant and system**.
3. The **0** folder consists of initial conditions, **constant** folder consists of the details of material and thermophysical properties and the **system** folder consists of mesh creation file, control file for the simulation and various discretization schemes of the solver.
4. The folder named **system** contains a file named as **blockMeshDict**
5. You need to set the parameters for geometry and meshing in the **blockMeshDict** file.
6. The steps to edit **blockMeshDict** file will be mentioned in later steps.

Step-1

1. Press **ctrl+ Alt+T** to open the terminal
2. Type the following command to connect to the server.

   For Division-1

   ```
   ssh test<GroupNumber>@10.102.1.121 –X (for groups 01–15)
   Password: test<GroupNumber>
   ssh test<GroupNumber–15>@10.102.1.122 –X (for groups 16–30)
   Password: test<GroupNumber–15>
   ```

   For example
   Group-1

   ```
   ssh test01@10.102.1.121 –X
   Password: test01
   ```

   Group-16

   ```
   ssh test01@10.102.1.122 –X
   Password: test01
   ```

   For Division-2

   ```
   ssh test<GroupNumber–15>@10.102.1.121 –X (for groups 31–45)
   Password: test<GroupNumber–15>
   ssh test<GroupNumber–30>@10.102.1.122 –X (for groups 46–60)
   Password: test<GroupNumber–30>
   ```

   For example
   Group-31

   ```
   ssh test16@10.102.1.121 –X
   Password: test16
   ```

   Group-46

   ```
   ssh test16@10.102.1.122 –X
   Password: test16
   ```

3. Now you are in the home directory of your user.
4. To create the our case folder type the following

   ```
   For Groups 1–15
   mkdir heattransfer<GN>_div1
   For Groups 16–30
   mkdir heattransfer<GN–15>_div1
   For Groups 31–45
   mkdir heattransfer<GN–30>_div2
   For Groups 45–60
   mkdir heattransfer<GN–30>_div2
   ```

For example

```
mkdir heattransfer03_div1    (For Group-3, division -1)
mkdir heattransfer08_div2    (For Group-38, division -2)
```

5. Before starting to problem we need to select our solver according to the flow conditions.

6. For this case since we are dealing with heat conduction over a rod we can choose **laplacianFoam** or **scalarTransportFoam** solver.

7. We choose **scalarTransportFoam** solver.

8. To create the case directory of our problem, we make use of the tutorial case for **scalarTransportFoam** solver, which is opened by typing the following command in the terminal

   ```
   cd $FOAM_TUTORIALS
   cd basic/scalarTransportFoam/pitzdaily
   ```

9. Now type **ls** command in the terminal to display the contents inside the folder.

10. To copy the files **0, constant and system** folders to our case folder **heattransfer** type the following command.

    ```
    For Group Number(GN) 01-15
    cp -r 0 constant system /home/test<GN>/heattransfer<GN>_div1
    For Group Number(GN) 16-30
    cp -r 0 constant system /home/test<GN-15>/heattransfer<GN-15>_div1
    For Group Number(GN) 31-45
    cp -r 0 constant system /home/test<GN-15>/heattransfer<GN-15>_div2
    For Group Number(GN) 46-60
    cp -r 0 constant system /home/test<GN-30>/heattransfer<GN-30>_div2
    ```

    For example,

    ```
    Group03, Division1
    cp -r 0 constant system /home/test03/heattransfer03_div1
    Group19, Division1
    cp -r 0 constant system /home/test04/heattransfer04_div1
    ```

Step-2

1. Now type **cd** in terminal to come to the home directory.

2. Open the case directory by typing **cd heattransfer<GN>_div<No>**.

3. Now type **cd system** to go inside the system folder and type **ls** to view the contents inside the system directory.

4. To edit the **blockMeshDict** file type **gedit blockMeshDict**,

```
cd
cd heattransfer03_div1
cd system
gedit blockMeshDict
```

5. Delete the contents of the blockMeshDict file to edit it for our case as shown below

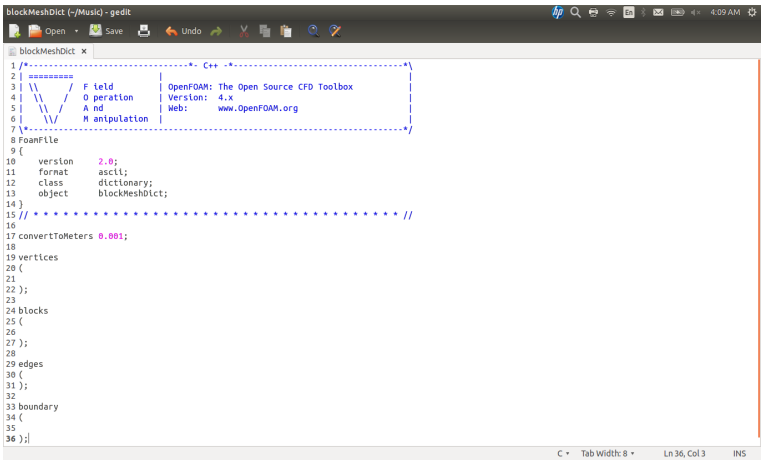

Figure 1: Blocks for the Mesh

Step-3

1. In OpenFoam we create geometry using point in space. This is similar to the way we use a graph paper and plot points on it.

2. Check for **convertToMeters** on the very first line of blockMeshDict.

3. By default the units used in OpenFOAM are in meters.

4. Since our geometry is in cm we need to use the conversion factor from meters to centimetres and replace 0.001 by 0.01.

Creation of Geometry

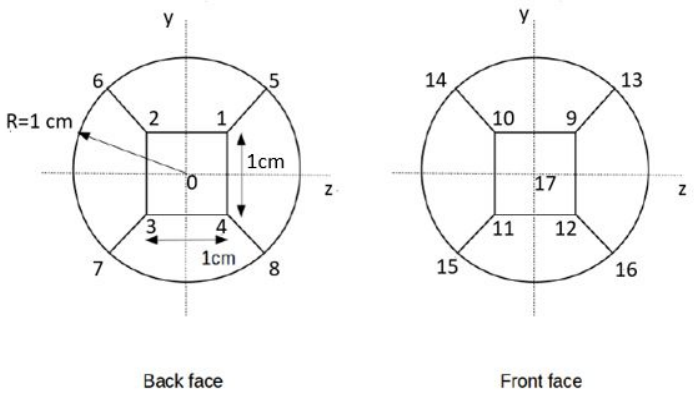

Figure 2: Node numbering for the Mesh

1. In OpenFOAM the numbering start from 0 and continues till the last point number 17. Here we have a total 18 points. The same geometry can be created using many blocks to have better mesh, which required more vertices.

2. Enter the co-ordinate of vertices as **(X Y Z)**

3. In vertices start entering the co-ordinates for these points. For example

```
(0     0          0)          //0
(0     0.5        0.5)        //1
(0     0.5       -0.5)        //2
(0    -0.5       -0.5)        //3
(0    -0.5        0.5)        //4
(0     0.7071     0.7071)     //5
(0     0.7071    -0.7071)     //6
```

4. Co-ordinates for the points on the outer periphery can be obtained using simple trigonometry relations. Enter the co-ordinates for the back face till point number 8.

5. For back face you should do this by keeping the x-coordinate as 0.

6. Since pipe is 30 cm, along x-axis we need to set the x coordinate as 30 for the front face.

7. To do this copy the points of the back face and paste it below point number 8.

8. Now starting from point number 9, add 30 to the x coordinate value instead of 0. Eg. point 9 **(30 0.5 0.5)**, point 10 **(30 0.5 -0.5)** and continue till the point 17.

9. The vertices are created in the order as we done numbering for it as shown in fig. The numbering can be done in any order (clockwise or anticlockwise).

Step-4

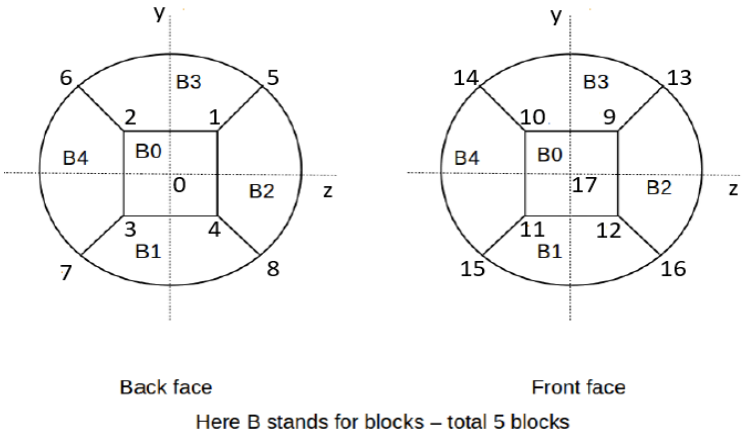

Figure 3: Blocks for the Mesh

1. Our geometry is divided in to blocks for Meshing purposes.
2. In the figure shown above we can see the block numbers starting from 0 upto 4 and hence total of 5 blocks.
3. Inside blocks we need to enter the following line for block number B0 :

   ```
   hex (1 4 3 2 9 12 11 10)(10 10 50) simpleGrading (1 1 1)
   ```

4. Here hex stands for Hexahedral Mesh, the next line with (10 10 50) stands for number of meshing points in **Z,Y and X directions** (since we are creating the faces of rod in Z-Y plane and length is aligned along X-axis).
5. The term simpleGrading is the ratio of the size of the end cell to the size of initial cell. Here ratio is kept as 1 in **Z Y X** axis and hence kept (1 1 1), since boundary layer refinement is not needed.
6. The **block numbering should be in clockwise direction**, since the rod is aligned along positive x-axis and we numbered the vertices by viewing from negative x axis.

   For example in the line **hex (1 4 3 2 9 12 11 10)**, since we choose (1 4 3 2) in clockwise direction in back plane, (9 12 11 10) is also chosen in clockwise direction in the front plane for the the creation of block.
7. Repeat this for remaining blocks.
8. Since this is a circular pipe we need to specify arcs. In case of a square pipe this part can be kept empty.
9. We need to edit the edges section in blockMeshDict file. which is shown below,

   ```
   edges
   (
   );
   ```

10. In our geometry we have total 8 arc and hence we need to specify them.

11. From the figure check for the points 5 and 6. They form two end points of the arc.

12. We need to define one point in between the points 5 and 6, to make an arc connecting 5 and 6 through the newly defined point suing the coordinate as **(X Y Z)**.

13. Though we can define co ordinate of any point in the arc between points 5 and 6, defining that point in axis is more simple.

14. Inside the arcs we need to specify the **arc 5 6 (0 1.0 0)**, arc along with the end points. The points in the brackets are any coordinate between two points 5 and 6 but which lies on that arc, **1.0** is the radius of the arc.

15. You have to repeat this for remaining arcs. For the front face we need to add the distance of x-axis. For example, **arc 13 14 (30 1.0 0)**

    For example

```
arc 5 6 (0 1 0)
arc 6 7 (0 0 -1)
arc 13 14 (30 1 0)
```

Step-5

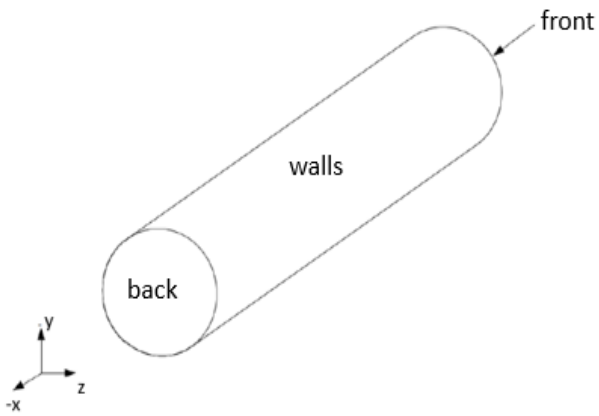

Figure 4: Boundary Patch Names for the Geometry

1. We need to enter the boundary faces and type here.

2. Now looking at the figure in step number 6, we need to set the faces for **front, back and walls** here.

3. Here we type the name of the boundary as **front**, on the next line we see the type as patch and then faces.

4. Faces are to be named carefully, since majority of the mistake take place here.

5. Since back face is divided in to 5 blocks, the **front** face consists of 5 faces.

6. While creating the faces either use a clock wise or anti-clock wise convention. In the example shown below anti-clockwise convention is followed by viewing the rod from **back** face (negative x-axis).

```
front
    {
        type patch;
        faces
        (
            (9 13 14 10)
            (10 14 15 11)
            (11 15 16 12)
            (12 16 13 9)
            (9 10 11 12)
        );
    }
```

7. Similarly do this for back and walls.

Step-6

1. Type **cd** to return to home directory.
2. Open the case file by typing **cd heattransfer<GN>_div<No>**
3. Rename the **0** to other name as **0.orig** (can be any name), since we have not edited the **0** folder. This can be done by typing the command.

   ```
   mv 0 0.orig
   ```

4. Type **blockMesh** in the terminal (Note that M is capital) and press enter.

   For example

   ```
   cd
   cd heattransfer03_div1
   mv 0 0.orig
   blockMesh
   ```

5. Your terminal window will display your geometry parameters and also the total number of cells in the geometry.
6. Then type **checkMesh** command.
7. It will display whether the **mesh is OK** of it has any errors. In case of any error have a better look at the error in the terminal and make changes accordingly.

Step-7

1. Now type **cd** to return to home directory.

2. Open the case file by typing **cd heattransfer<GN>_div<No>** and type **paraFoam** in the terminal (Note that F is capital) and press enter.

   For example

   ```
   cd
   cd heattransfer03_div1
   paraFoam
   ```

3. This will Open up the paraview window as shown in the figure.

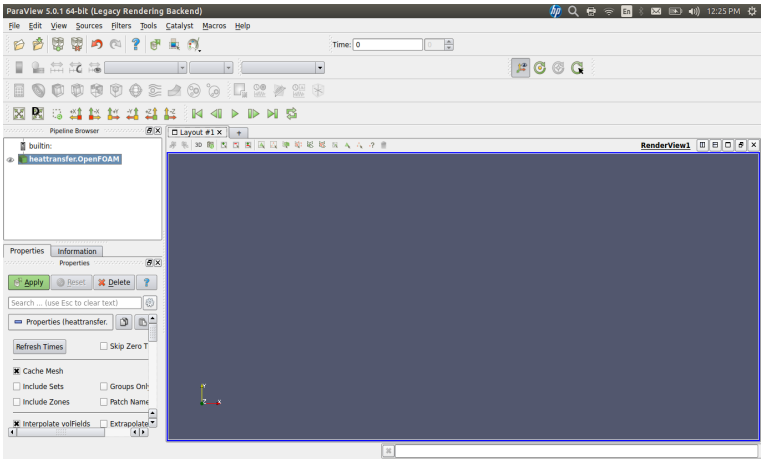

Figure 5: Paraview window

4. Now click on **Apply** in the pipeline browser to view the geometry. The geometry will be displayed as shown below

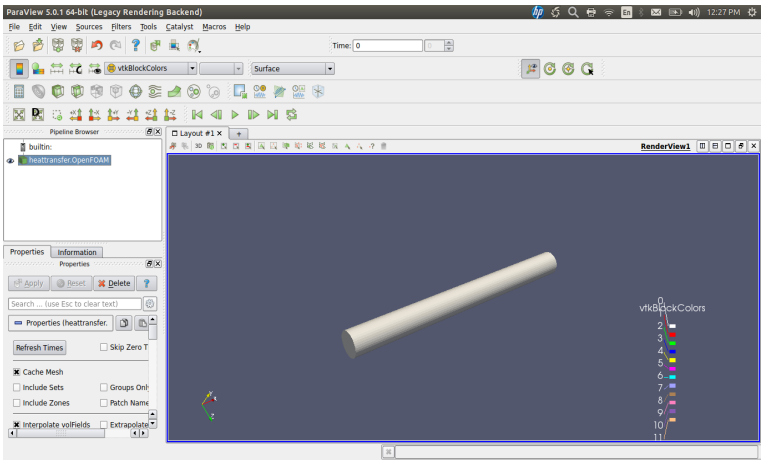

Figure 6: Solid model of the Geometry

5. In the drop down list select **Surface with Edges** to display the mesh as shown below.

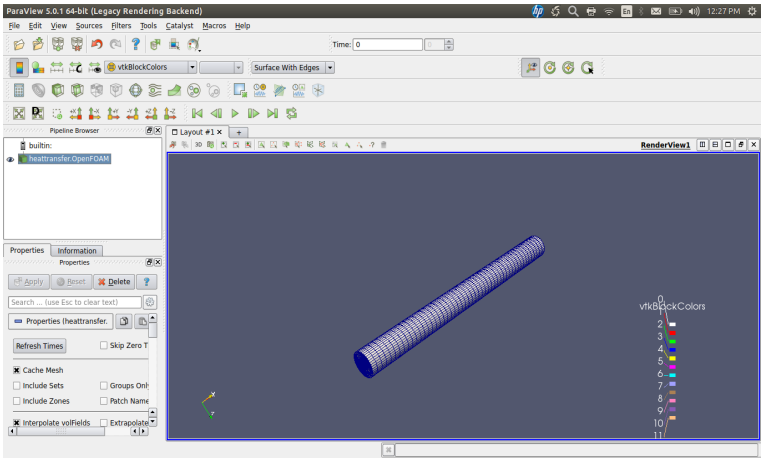

Figure 7: Mesh model of the Geometry

6. Go to **File** > **Save Screenshot** and give names according to your group id.

You have now finished creating the geometry, meshing it and setting up the boundary faces.