

# OpenFOAM

An Open source alternative to commercial CFD Package

## Instruction Manual for creating blockMeshDict for 3-D pipe Geometry



Prepared By:

**CFD Team**

**FOSSEE**



Indian Institute of Technology, Bombay

December 2017

# Objective

To create the geometry of a pipe in blockMesh.

## Geometry

Straight horizontal pipe

Radius of the rod ( $r$ ) = 1 cm

Length of the rod ( $L$ ) = 30 cm

Rod is aligned along the x-axis

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

```
mkdir solidRod
```
4. 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 incompressible/simpleFoam/pitzdaily
```
5. Now type **ls** command in the terminal to display the contents inside the folder.

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

```
cp -r 0 constant system /home/test<UserID>/solidRod
```

## Editing blockMeshDict

1. Now type **cd** in terminal to come to the home directory.
2. Open the case directory by typing **cd pipe<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 solidRod
cd system
gedit blockMeshDict
```

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

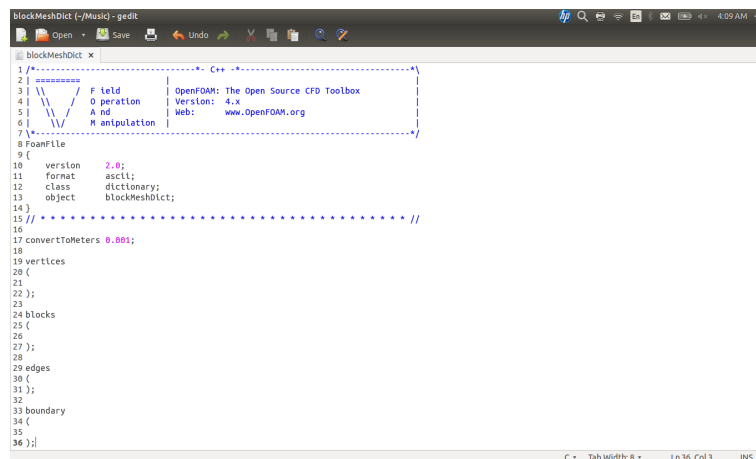


Figure 1: Blocks for the Mesh

## Scale Conversion

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 Vertices

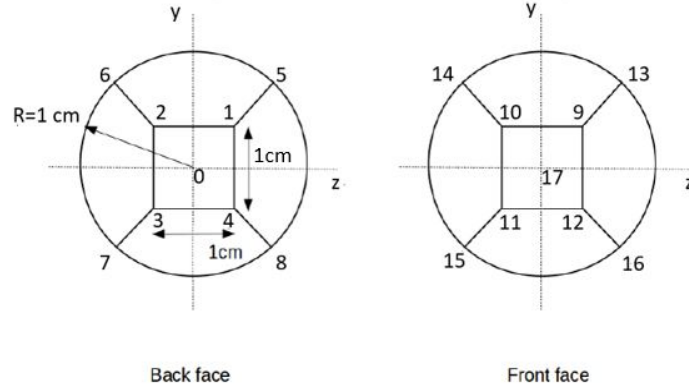


Figure 2: Node numbering for the Mesh

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).

## Creation of blocks

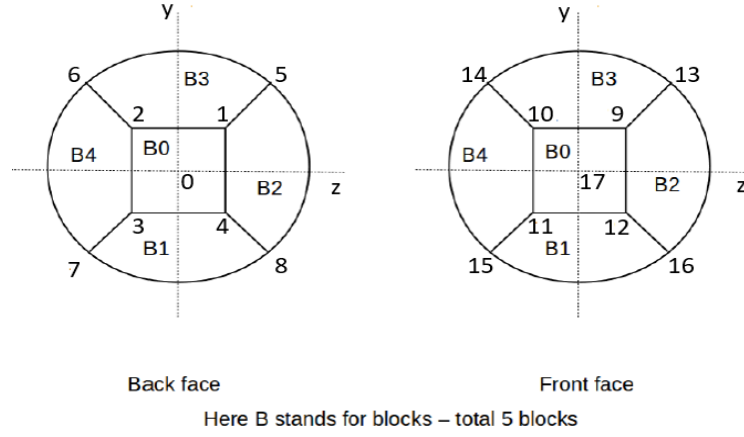


Figure 3: Blocks for the Mesh

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 using the coordinate as **(X Y Z)**.
13. Though we can define coordinate 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)
```

## Setting up of Boundary Patches

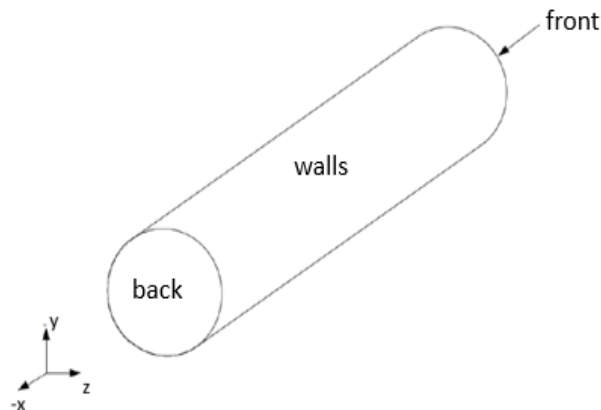


Figure 4: Boundary Patch Names for the Geometry

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.

## Meshing the Geometry

1. Type **cd** to return to home directory.
2. Open the case file by typing **cd pipe<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 solidRod
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** if it has any errors. In case of any error have a better look at the error in the terminal and make changes accordingly.

# Viewing the Mesh in paraFoam

1. Now type **cd** to return to home directory.
2. Open the case file by typing **cd solidRod** and type **paraFoam** in the terminal (Note that F is capital) and press enter.

For example

```
cd
cd solidRod
paraFoam
```

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

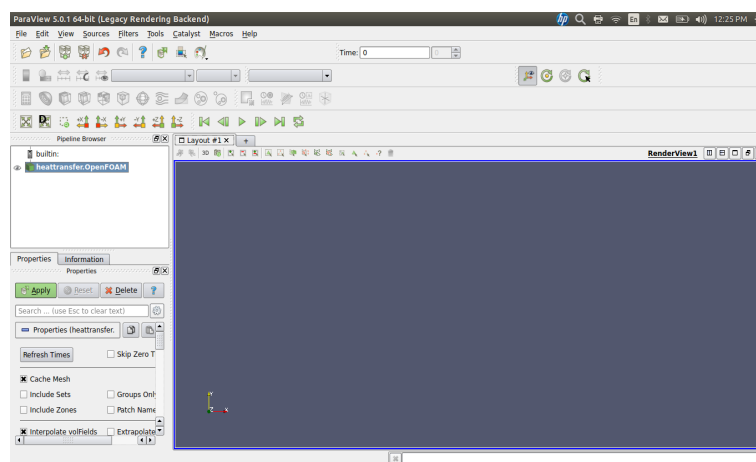


Figure 5: Paraview window

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

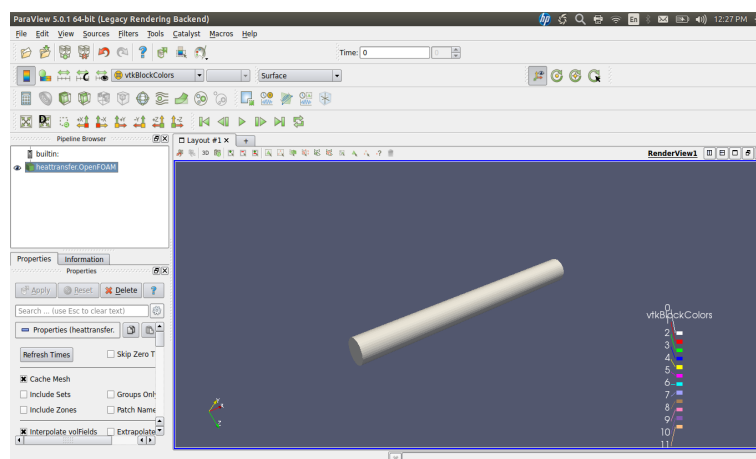


Figure 6: Solid model of the Geometry

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



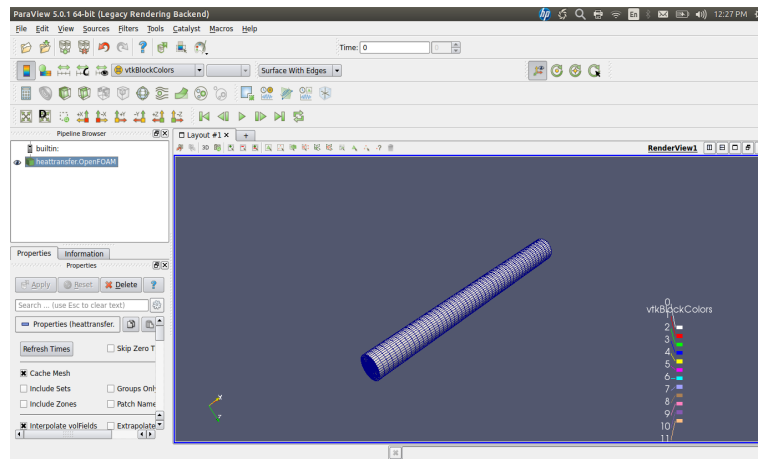


Figure 7: Mesh model of the Geometry

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