

# Special Topics in CFD

DAY 9

Flow over a Sphere in OpenFOAM

---

*Kumares*

# Exercise – 7

1. Create a surface file “Sphere”
2. Create a case setup file for the problem, “Flow over a sphere”
3. Modify the case setup file
4. Run your new case file
5. Upload your new solver, case files, and results in GITHUB

# 1. Create a surface file “Sphere”

1) Use any modelling software (AutoCAD, CATIA, FreeCAD or any tool) to model sphere

2) Sphere dimensions are as follows:

Radius = 0.1mm

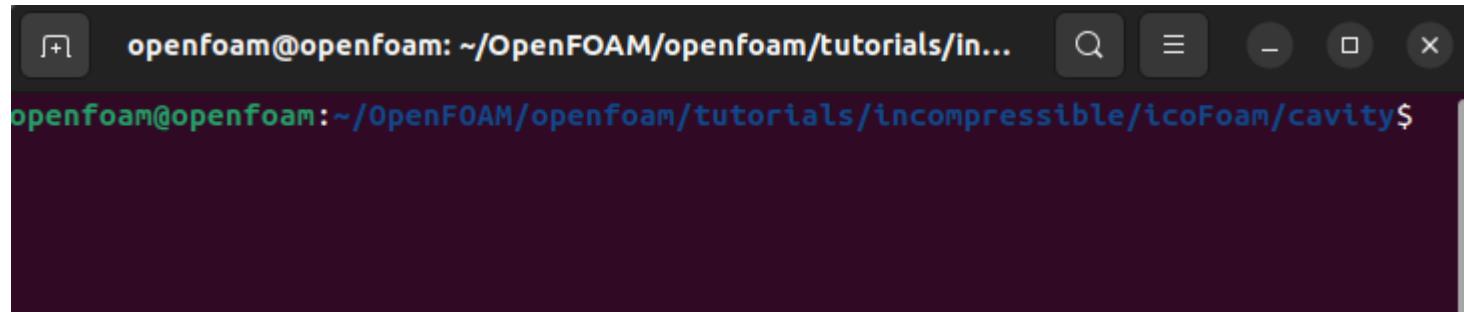
Location of the sphere = (0.5mm, 0.5mm, 0.5mm)

3) Export the sphere file with the file name, “**sphere.obj**”

## 2. Create a case setup file for the problem, “Flow over a sphere”

Use **icoFoam** solver

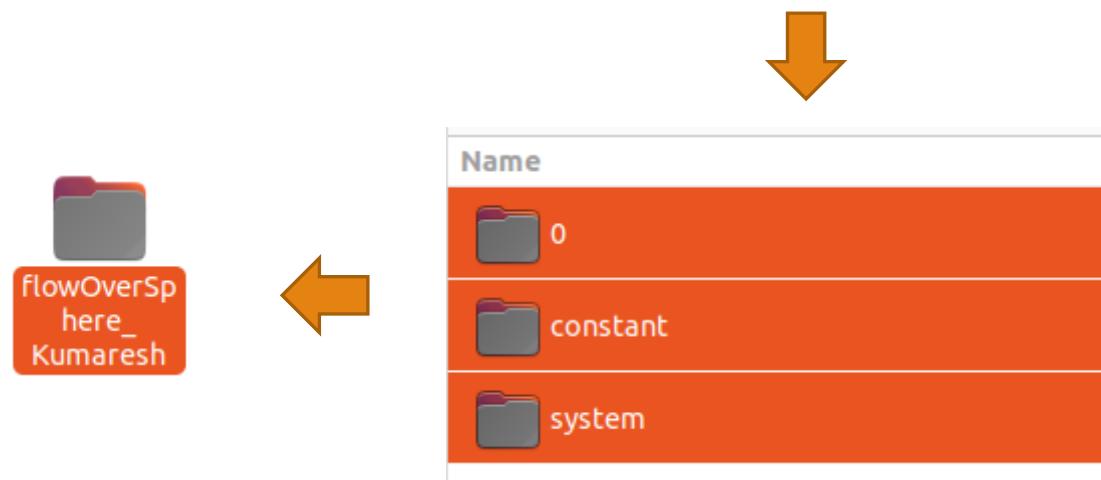
From the location:



```
openfoam@openfoam:~/OpenFOAM/openfoam/tutorials/incompressible/icoFoam/cavity$
```

Copy 0, constant, and system  
files in the new file name

“flowOverSphere\_YourNAME”



### 3a. Modify the case setup file

Changes in system/blockMesh file

icoFoam file

```
scale  0.1;

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

blocks
(
    hex (0 1 2 3 4 5 6 7) (20 20 1) simpleGrading (1 1 1)
);
```

flowOverSphere



```
scale  1;

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

blocks
(
    hex (0 1 2 3 4 5 6 7) (20 20 20) simpleGrading (1 1 1)
);
```

### 3a. Modify the case setup file

Changes in system/blockMesh file

#### icoFoam file

```
boundary
(
    movingWall
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }
    fixedWalls
    {
        type wall;
        faces
        (
            (0 4 7 3)
            (2 6 5 1)
            (1 5 4 0)
        );
    }
    frontAndBack
    {
        type empty;
        faces
        (
            (0 3 2 1)
            (4 5 6 7)
        );
    }
);
```



#### flowOverSphere

```
front
{
    type wall;
    faces
    (
        (4 5 6 7)
    );
}
back
{
    type wall;
    faces
    (
        (0 3 2 1)
    );
}
```

```
boundary
(
    top
    {
        type wall;
        faces
        (
            (3 7 6 2)
        );
    }
    bottom
    {
        type wall;
        faces
        (
            (1 5 4 0)
        );
    }
    left
    {
        type patch;
        faces
        (
            (0 4 7 3)
        );
    }
    right
    {
        type patch;
        faces
        (
            (2 6 5 1)
        );
    }
);
```

### 3b. Modify the case setup file

Changes in 0/p boundary file

icoFoam file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * *

dimensions   [0 2 -2 0 0 0 0];
internalField uniform 0;
boundaryField
{
    movingWall
    {
        type          zeroGradient;
    }

    fixedWalls
    {
        type          zeroGradient;
    }

    frontAndBack
    {
        type          empty;
    }
}
```



flowOverSphere

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volScalarField;
    object      p;
}
// * * * * *

dimensions   [0 2 -2 0 0 0 0];
internalField uniform 0;
boundaryField
{
    top
    {
        type          zeroGradient;
    }

    bottom
    {
        type          zeroGradient;
    }

    left
    {
        type          zeroGradient;
        //type          fixedFluxPressure;
        //value         uniform 0;
    }

    right
    {
        type          zeroGradient;
        //type          totalPressure;
        //p0            uniform 0;
    }
}
```

```
    front
    {
        type          zeroGradient;
    }

    back
    {
        type          zeroGradient;
    }

    sphere
    {
        type          zeroGradient;
    }
}
```

### 3b. Modify the case setup file

#### icoFoam file

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * *
dimensions   [0 1 -1 0 0 0];
internalField uniform (0 0 0);
boundaryField
{
    movingWall
    {
        type          fixedValue;
        value         uniform (1 0 0);
    }

    fixedWalls
    {
        type          noslip;
    }

    frontAndBack
    {
        type          empty;
    }
}
```



#### Changes in 0/U boundary file

#### flowOverSphere

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       volVectorField;
    object      U;
}
// * * * * *
dimensions   [0 1 -1 0 0 0];
internalField uniform (1e-8 1e-8 1e-8);
boundaryField
{
    top
    {
        type          noslip;
    }

    bottom
    {
        type          noslip;
    }

    left
    {
        type          fixedValue;
        value         uniform (1 0 0);
    }

    right
    {
        //type          zeroGradient;
        //type          pressureInletOutletVelocity;
        //value         uniform (0.1 0 0);
        type          outletInlet;
        value         uniform (1 0 0);
    }
}
```

```
front
{
    type          noslip;
}

back
{
    type          noslip;
}

sphere
{
    type          noslip;
}
```

### 3c. Modify the case setup file

Changes in constant file

1) Create a **triSurface** file by the following command:

```
mkdir -p triSurface
```

2) Inside the triSurface file, copy the surface file named “sphere.obj” (Carried out in step 1)



sphere.obj

### 3d. Modify the case setup file

Changes in system/controlDict file

#### icoFoam file

```
application      icoFoam;  
  
startFrom       startTime;  
  
startTime       0;  
  
stopAt          endTime;  
endTime         0.5;  
deltaT          0.005;  
writeControl    timeStep;  
writeInterval   20;  
purgeWrite     0;  
writeFormat     ascii;  
writePrecision  6;  
writeCompression off;  
timeFormat      general;  
timePrecision   6;  
runTimeModifiable true;
```



#### flowOverSphere

```
application      icoFoam;  
  
startFrom       startTime;  
  
startTime       0;  
stopAt          endTime;  
endTime         1;  
deltaT          0.0001;  
writeControl    adjustable;  
writeInterval   0.002;  
purgeWrite     0;  
writeFormat     ascii;  
writePrecision  6;  
writeCompression off;  
timeFormat      general;  
timePrecision   6;  
runTimeModifiable true;  
adjustTimeStep yes;  
maxCo          1;  
maxDeltaT      0.1;
```

### 3d. Modify the case setup file

Changes in system file

1) Keep the remaining files, fvSchemes, fvSolution, and decomposeParDict as such.

2) Download the **following files from GitHub** and copy inside system file:

> meshQualityDict

> snappyHexMeshDict

> surfaceFeatureExtractDict

## 4. Run your new case file

1. blockMesh
2. surfaceFeatureExtract
3. snappyHexMesh -overwrite
4. icoFoam

## 5. Upload your new solver, case files, and results in GITHUB

### Exercise – 7

