

Special Topics in CFD

DAY 9

Convection problem in OpenFOAM

Kumaresh

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”


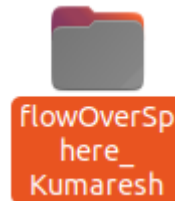
Use **icoFoam** solver

From the location:

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

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

“flowOverSphere_YourNAME”



Name
0
constant
system

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];
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];
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

Changes in 0/U boundary 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;
    }
}
```



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;
        outletValue uniform (1 0 0);
    }
}
```

```
front
{
    type      noSlip;
}

back
{
    type      noSlip;
}

sphere
{
    type      noSlip;
}
```

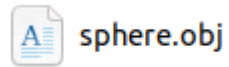

3c. Modify the case setup file

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)



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

