3d Centrifugal Rotating Pump

Federico Piscaglia *

Dept. of Aerospace Science and Technology (DAER), Politecnico di Milano, Italy

Abstract. Lab handout for study of incompressible flow inside a centrifugal pump with a steady solver. Topics covered: patch manipulation, creation of sets and zones, MRF, decomposition and parallel running.

1 Learning outcome

The software used is the open-source CFD software OpenFOAM-8 by the OpenFOAM Foundation. In this Lab you will learn how to:

- Manipulate patches using `createPatch`

- Create sets and zones with `topoSet`
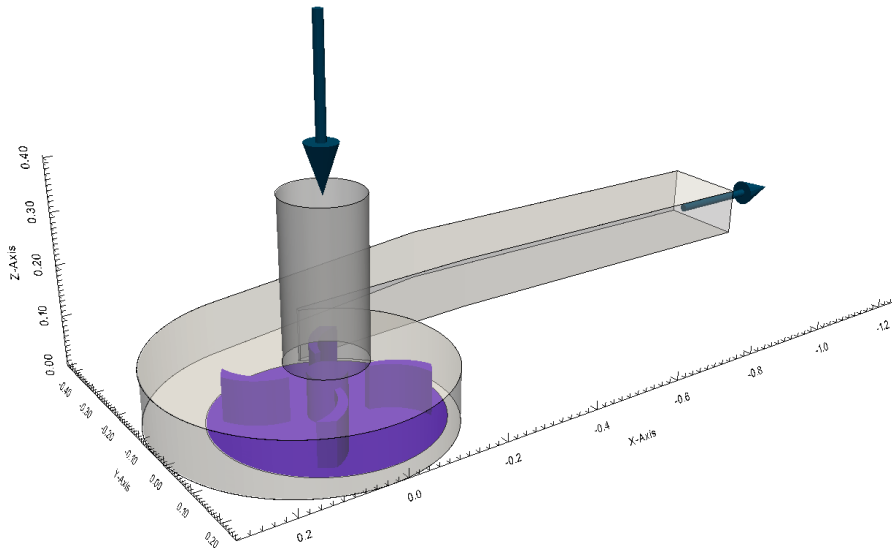
- Decompose the mesh for a parallel run

Figure 1: geometry

---

*Tel. (+39) 02 2399 8620, E-mail: federico.piscaglia@polimi.it

2   Prepare the mesh

The geometry to be simulated is represented in Fig. 1. Flow will be simulated first using a steady-state incompressible solver (`simpleFoam`), modeling source terms with the Moving Reference Frame (MRF) approach.

Impeller is rotating at 100 rad/s. Backpressure on the outlet is 1 m^2/s^2

Copy the provided folder into your `$FOAM_RUN` and extract the mesh

```
user@host:centrifugalPump$ tar xzf centrifugalPump-mesh.tgz
```

2.1   Create sets for joining top and bottom patch

1. Copy the example file from the utility folder
   ```
   $ cp -r $FOAM_ETC/caseDicts/annotated/topoSetDict system/topoSetDict.createPatch
   ```
   or alternatively:
   ```
   $ foamGet topoSetDict
   ```
   ```
   $ mv system/topoSetDict system/topoSetDict.createPatch
   ```

2. Delete all but the header

3. Add the following lines:
   ```
   actions
   (
       {
           name     topBottomPatch;
           type     faceSet;
           action   new;
           source   patchToFace;
           sourceInfo
           {
            name lowerWall;
           }
       }

       {
           name topBottomPatch;
           type faceSet;
           action add;
           source patchToFace;
           sourceInfo
           {
               name topBottom;
           }
       }

   );
   ```

4. run topoSet
   ```
   user@host:centrifugalPump$ topoSet -dict system/topoSet.createPatch
   ```

2.2   Adjust patch names

1. Copy the example file:
   ```
   user@host:centrifugalPump$ cp $FOAM_ETC/caseDicts/annotated/createPatchDict system
   ```

2. Delete what is inside the list "patches" and add the following lines:

```
    {
        name topBottom;
        patchInfo
        {
            type wall;
        }
        constructFrom set;
        set topBottomPatch;
    }

    {
        name outlet;
        patchInfo
        {
            type patch;
        }
        constructFrom patches;
        patches (inlet);
    }

    {
        name inlet;
        patchInfo
        {
            type patch;
        }
        constructFrom patches;
        patches (upperWall);
    }
```

3. run "createPatch"

```
user@host:centrifugalPump$ createPatch -overwrite
```

2.3   Renumber the mesh

```
user@host:centrifugalPump$ renumberMesh -overwrite
```

2.4   Copy the BCs

```
user@host:centrifugalPump$ cp -r 0.orig 0
```

2.5   Create MRF zone

1. Copy the topoSetDict:

```
$ cp system/topoSetDict.createPatch system/topoSetDict
```

2. Delete all actions and add the following lines:

```
    {
        name    movingCells;
        type    cellSet;
```

```
            action   new;
            source   cylinderToCell;
            sourceInfo
            {
                p1 ( 0 0  -1);
                p2 ( 0 0 0.09);
                radius 0.180;
            }
        }
```

3. create the set:

   ```
   user@host:centrifugalPump$ topoSet
   ```

4. Convert the cellSet to a cellZone: `user@host:centrifugalPump$ setsToZones -noFlipMap`

3 Set up MRF

1. Copy the template file:

   ```
   $ cp $FOAM_TUTORIALS/incompressible/simpleFoam/mixerVessel2D/constant/MRFProperties constant
   ```

2. Modify the MRFProperties files and set appropriate parameters (sign follows the right-hand rule):

   ```
   MRF1
   {
       cellZone     movingCells;
       active       yes;

       // Fixed patches (by default they 'move' with the MRF zone)
       nonRotatingPatches (topBottom);

       origin    (0 0 0);
       axis      (0 0 1);
       omega     -100;
   }
   ```

4 Decompose the mesh

1. Copy the example file

   ```
   $ cp $FOAM_ETC/caseDicts/annotated/decomposeParDict system
   ```

2. Set 2 subdomains and SCOTCH decomposition method

   ```
   numberOfSubdomains 2;
   method          scotch;
   ```

3. Decompose

   ```
   user@host:centrifugalPump$ decomposePar
   ```

5 Run the case (steady-state)

Run the solver `simpleFoam`:

```
user@host:centrifugalPump$ mpirun -np 2 simpleFoam -parallel > log.simpleFoam 2>>&1 &
```

Print out the log file on the screen while the simulation is running:

```
user@host:centrifugalPump$ tail -f log.simpleFoam
```

(note: to stop the run-time visualization of the file, type `Ctrl+C`. This does not stop the simulation, it just stops the bash command, i.e. the visualization of the log file at run-time!)

6   Hands-on

6.1   Set post-processing

Set the functionObjects to extract the following quantities:

- p, $\boldsymbol{U}$ on a y- normal plane

- p on impeller blades

- Streamlines

- p along the outlet channel

- Torque (on impeller)

- p_2, p_1

- $\dot{V}_2$, $\dot{V}_1$

- Q-criterion, vorticity

- iso-Q surfaces

- residual history

6.2   Extract characteristic curve

Produce the graph $H = f\left(\dot{V}\right)$, where $[H] = $ m is the pump head.

Hint:

$$H = \frac{p_2 - p_1}{\rho g} + \frac{v_2^2 - v_1^2}{2g}$$