



Escola Superior d'Enginyeries Industrials,  
Aeroespacial i Audiovisual de Terrassa

UNIVERSITAT POLITÈCNICA DE CATALUNYA

# Simulation of the first stages of a turbofan using OpenFOAM

---

## Report

**Degree:** Aerospace Engineering

**Course:** Application of Open-Source CFD to Engineering Problems

**Delivery date:** 09-12-2016

### Students:

Herrán Albelda, Fernando

Martínez Viol, Víctor

Morata Carranza, David

# Contents

<b>1</b>	<b>Introduction</b>	<b>2</b>
<b>2</b>	<b>Case</b>	<b>3</b>
2.1	Description of the case . . . . .	3
2.2	Hypotheses . . . . .	4
<b>3</b>	<b>Pre-processing</b>	<b>6</b>
3.1	Mesh generation . . . . .	6
3.1.1	Selection of the tutorial . . . . .	6
3.1.2	blockMesh . . . . .	6
3.1.3	Mesh refinement . . . . .	13
3.1.4	Comparison of different types of mesh . . . . .	19
3.1.4.1	Coarse mesh . . . . .	20
3.1.4.2	Standard mesh . . . . .	21
3.1.4.3	Dense mesh . . . . .	22
3.2	checkMesh . . . . .	25
3.3	Rotation . . . . .	28
3.4	Boundary conditions . . . . .	32
3.4.1	Velocity . . . . .	32
3.4.2	Pressure . . . . .	34
3.5	Properties of the flow . . . . .	36
3.6	controlDict - parameters for the simulation . . . . .	38
3.7	fvSchemes and fvSolution . . . . .	41
<b>4</b>	<b>Simulation of the turbofan</b>	<b>45</b>
4.1	Selection of the solver . . . . .	45
4.1.1	SIMPLE algorithm . . . . .	45
4.1.2	PISO algorithm . . . . .	46
4.2	Commands to run the simulation . . . . .	47
<b>5</b>	<b>Post-process</b>	<b>49</b>
5.1	Analysis of the results . . . . .	50

5.2	Conclusions . . . . .	51
<b>6</b>	<b>Difficulties that we faced</b>	<b>52</b>
<b>7</b>	<b>References</b>	<b>53</b>

# 1 Introduction

During the development of the course '*Application of Open-Source CFD to engineering problems*' we have learned the basics of how to use and solve real-world cases and situations related with fluid mechanics using OpenFOAM, an open source CFD tool. The first days of the course, several possible projects were presented and we had to choose one of them and make a report. After agreeing with the professor, we decided to simulate an option that was not on that list. Since we are really interested in propulsion, we thought that it was a good idea to try to simulate the flow inside the first stages of compression of a turbofan engine.

These type of engines are the most used propulsion system in the aerospace industry. It presents several advantages to other systems such as the turboprop or the turbojet; for example, this kind of engine takes advantage of the flow that goes through the fan (that cannot be more compressed or heated given that the combustion chamber has a limited volume) and comes out through the rear nozzle. This results in a higher thrust for the same amount of fuel that is burned. Thus, it is no surprise that state-of-the-art planes such as the Airbus 380 or the Boeing 747 use this kind of propulsion system.

In order to do a realistic simulation, we have been gathering lots of information of this type of engines and their typical working conditions. Also, we have been searching for information about how could we solve a geometry that is rotating. Several simulations and comparisons will be presented in this report to analyze the validity of the results obtained.

It will be presented in this report how to use the Arbitrary Mesh Interface utility (AMI) as well as the boundary conditions, the mesh generation and the solver that has been used for the case. Additionally, several hypotheses will be considered in order to alleviate the computation time (given that this project has been simulated in a laptop).

## 2 Case

### 2.1 Description of the case

The aim of the current project is to analyze the airflow inside the first stages of a turbofan. The turbofan consists of an initial stage where a fan is placed. This first stage increases the pressure of the air that goes through it and, as it can be seen in 2.1, a part of the incoming airflow goes to the low pressure compression stage (main or primary flow) and the other goes to the conducts (secondary flow). As the secondary flow advances through the conduct, the main flow enters the engine core where it goes through the low pressure and high pressure compression stages. By increasing the pressure of the air, the density also increases; thus, a higher mass flow can be mixed with the fuel and burned in the combustion chamber. After this process, the gases go through the turbines, interchanging energy with them (pressure and velocity, mainly), and they arrive at the nozzle, where the air is accelerated, and a thrust force is obtained.

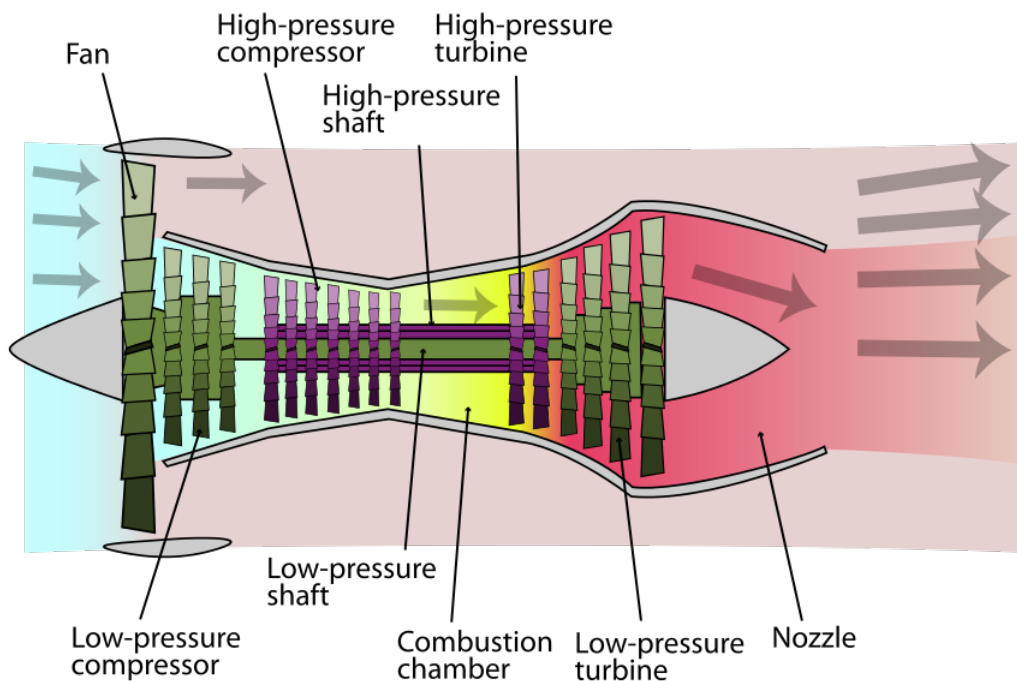


Figure 2.1: Parts of a turbofan

To simulate the flow in the first stages of the turbofan, we have downloaded the following turbine model from *GrabCad*, shown in 2.2. This model is pretty similar with the one shown above (2.1) and the simulation will take place between the back side of the fan and the back side of the second compressor. There are several realistic and potentially functional models on *GrabCad*; however, this model has been selected because it did not present incompatibilities with *SolidWorks* and *Salome*. Given that all the parts of the model have to be clearly differentiated in order to export them as *STL* files, we could not use an assembly.

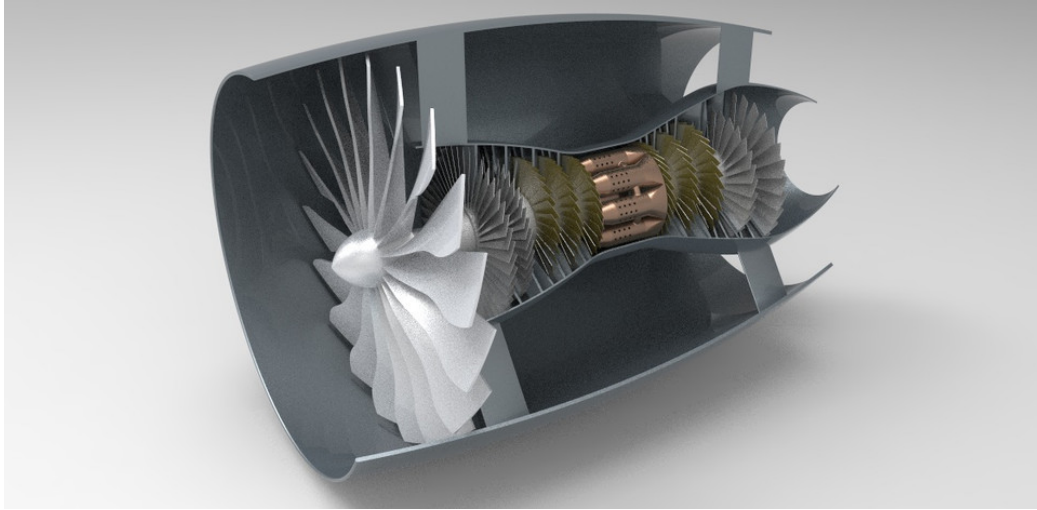


Figure 2.2: Model used

## 2.2 Hypotheses

Given that we are not able to use a supercomputer and the computational power that is available to us is very limited since we are using our own personal computers to run this simulation, several hypotheses have to be made to alleviate the calculation time, as mentioned before. Some of them can be assumed and some other will make the case a non-realistic project. These hypotheses are presented below.

- Incompressible flow
- Newtonian flow

- Laminar flow

Clearly, the flow is not incompressible in reality; the aim of the turbofan is to compress the air to seek for a more efficient combustion. However, all of the compressible solvers are really difficult to use and the computational time is also higher given that we have a high number of control volumes (as discussed in the next section) due to a really complex geometry. Thus, although it will not be a completely real flow, we will be also able to see how the low pressure section of the engine increases the pressure of the flow and how the velocity field changes as it goes through the turbine.

The simulation will take into account that it is a tridimensional flow, also that it behaves as a newtonian fluid and it will be run under transient flow conditions (that is: the velocity field as well as the values of the pressure change over time). With these hypothesis, we can begin to discuss the geometry of the *blockMesh* as well as the refined mesh, and the boundary conditions and final results in the next sections.

Finally, given that it is a fictional case, no validations of the solution will be performed.

## 3 Pre-processing

### 3.1 Mesh generation

#### 3.1.1 Selection of the tutorial

The selection of the tutorial case is the first thing that needs to be done to run the simulation of the turbofan since all the files that will be used are linked and, starting a new case would mean spending a lot of time creating files and establishing all these links in order to make *OpenFOAM* work properly. Among all the tutorials that can be found on the *OpenFOAM* folder, we have selected the *mixerVesselAMI2D* for several reasons.

#### 3.1.2 blockMesh

The definition of the *blockMeshDict* is the first part that needs to be modified. The *blockMesh* must contain the geometry that has to be simulated and we must have an idea of the vertices of the parallelogram that will contain the first stages of the compressor. In order to do that, the geometry has to be opened using either *Salome* or *Paraview*. Then the axes must be showed and the points have to be written down on the *blockMesh* file.

Now that the points enclosing the desired geometry are clear, they can be modified in the `system/blockMeshDict` file. This change is presented below:

```
vertices
(
    (0.77 0 0)
    (1.35 0 0)
    (1.35 2.3 0)
    (0.77 2.3 0)
    (0.77 0 2.3)
    (1.35 0 2.3)
    (1.35 2.3 2.3)
    (0.77 2.3 2.3)
);
```



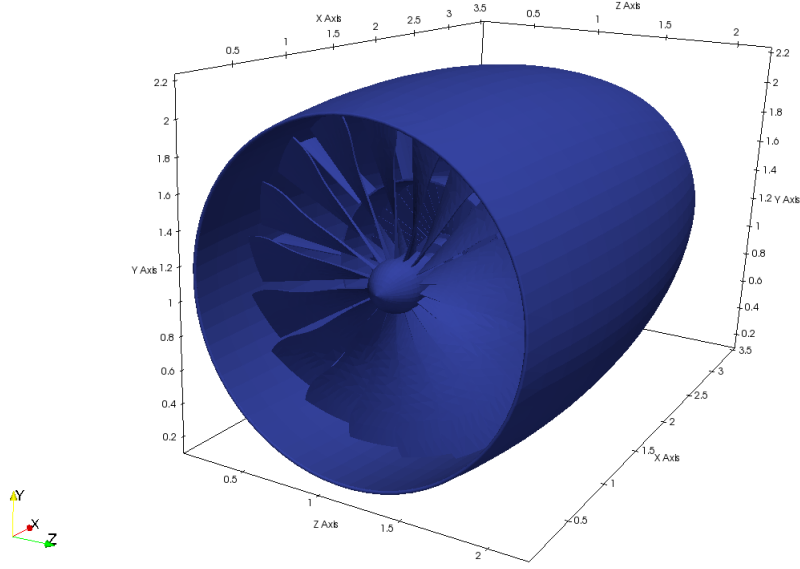


Figure 3.1: Turbofan with axes

It can be clearly seen that the domain of the mesh is a  $0.58 \times 2.3 \times 2.3m$  rectangular prism.

Once the boundaries of the *blockMesh* are defined, the number of cells that it will have has to be set. It has to be taken into account that a very dense mesh at the beginning will not be efficient when simulating the case given that we will use the *snappyHexMesh* later, so it might be over densified. On the other hand, a very coarse mesh will not be efficient either because additional divisions will have to be set when generating the refined mesh and the computational time of the *snappyHexMesh* process might grow. Thus, a solution between a mesh with a very high number of cells and a very low number of cells has to be attained.

This basic mesh has been divided every  $0.05m$ . It means that we have done 12 divisions in the x direction, 46 on the y direction and 46 more on the z direction.

blocks

```
(
    hex (0 1 2 3 4 5 6 7) (12 46 46) simpleGrading (1 1 1)
);
```

Finally, the different faces of the mesh must be defined depending on whether they are the inlet or outlet faces or the lateral faces of the geometry. To do this, *OpenFOAM* numbers the vertices according to their appearance in the *blockMeshDict* and the faces are defined by four of those numbers. Since the vertex numeration has been kept the same as the one that comes as default in every tutorial case; it is easier to define the inlet face of the *blockMesh* (this is: the face in which the flow comes in) as well as the outlet face (this is: the face in which the flow comes out) that will be used to define the boundary conditions later.

```
boundary
(
    frontAndBack
    {
        type patch;
        faces
        (
            (3 7 6 2)
            (1 5 4 0)
            (0 3 2 1)
            (4 5 6 7)
        );
    }
    inlet
    {
        type patch;
        faces
        (
            (0 4 7 3)
        );
    }
    outlet
    {
        type patch;
        faces
```

```

        (
            (2 6 5 1)
        );
    }
};

```

The *blockMeshDict* file is presented below. This is the final file that has been used to generate the *blockMesh* and it is included in the *.zip* file attached to this report. Only the parameters mentioned above have been modified; the rest is equal to the tutorial case that has been selected.

```

/*-----*- C++ -*-----*\
| ===== |
| \\      / F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O peration  | Version: 4.0 |
|  \\    /  A nd        | Web:      www.OpenFOAM.org |
|   \\/    M anipulation | |
\*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       blockMeshDict;
}

// * * * * *

convertToMeters 1;

vertices
(
    (0.77 0 0)
    (1.35 0 0)
    (1.35 2.3 0)
    (0.77 2.3 0)
    (0.77 0 2.3)
    (1.35 0 2.3)
    (1.35 2.3 2.3)
    (0.77 2.3 2.3)

```

```

);

blocks
(
    hex (0 1 2 3 4 5 6 7) (12 46 46) simpleGrading (1 1 1)
);

edges
(
);

boundary
(
    frontAndBack
    {
        type patch;
        faces
        (
            (3 7 6 2)
            (1 5 4 0)
            (0 3 2 1)
            (4 5 6 7)
        );
    }
    inlet
    {
        type patch;
        faces
        (
            (0 4 7 3)
        );
    }
    outlet
    {
        type patch;
        faces
        (
            (2 6 5 1)
        );
    }
);

```

```
// ***** //
```

The log obtained when the *blockMesh* has been generated is presented below. As it can be seen, no errors were found during the computation of this basic mesh. The number of cells is relatively high (**25392**) but perfectly suitable to proceed with the refinement of the mesh. Additionally, a caption of the basic mesh is presented in 3.2.

```
/*-----*\
| ===== |
| \ \      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox |
| \ \      / O peration     | Version: 4.0 |
| \ \      / A nd           | Web: www.OpenFOAM.org |
| \ \ /      M anipulation  | |
\*-----*/

Build : 4.0-665f1db4c1f1
Exec : blockMesh
Date : Dec 05 2016
Time : 14:17:05
Host : "victorPC"
PID : 4499
Case : /home/victor/Desktop/Final/pimpleDyMFoam/turbofan_std
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * //

Create time

Creating block mesh from
    "/home/victor/Desktop/Final/pimpleDyMFoam/turbofan_std/system/blockMeshDict"
Creating curved edges
Creating topology blocks
Creating topology patches

Creating block mesh topology

Check topology
```

```

Basic statistics
Number of internal faces : 0
Number of boundary faces : 6
Number of defined boundary faces : 6
Number of undefined boundary faces : 0
Checking patch -> block consistency

Creating block offsets
Creating merge list .

Creating polyMesh from blockMesh
Creating patches
Creating cells
Creating points with scale 1
    Block 0 cell size :
        i : 0.0483333 .. 0.0483333
        j : 0.05 .. 0.05
        k : 0.05 .. 0.05

There are no merge patch pairs edges

Writing polyMesh
-----
Mesh Information
-----
    boundingBox: (0.77 0 0) (1.35 2.3 2.3)
    nPoints: 28717
    nCells: 25392
    nFaces: 79396
    nInternalFaces: 72956
-----
Patches
-----
    patch 0 (start: 72956 size: 2208) name: frontAndBack
    patch 1 (start: 75164 size: 2116) name: inlet
    patch 2 (start: 77280 size: 2116) name: outlet

End

```

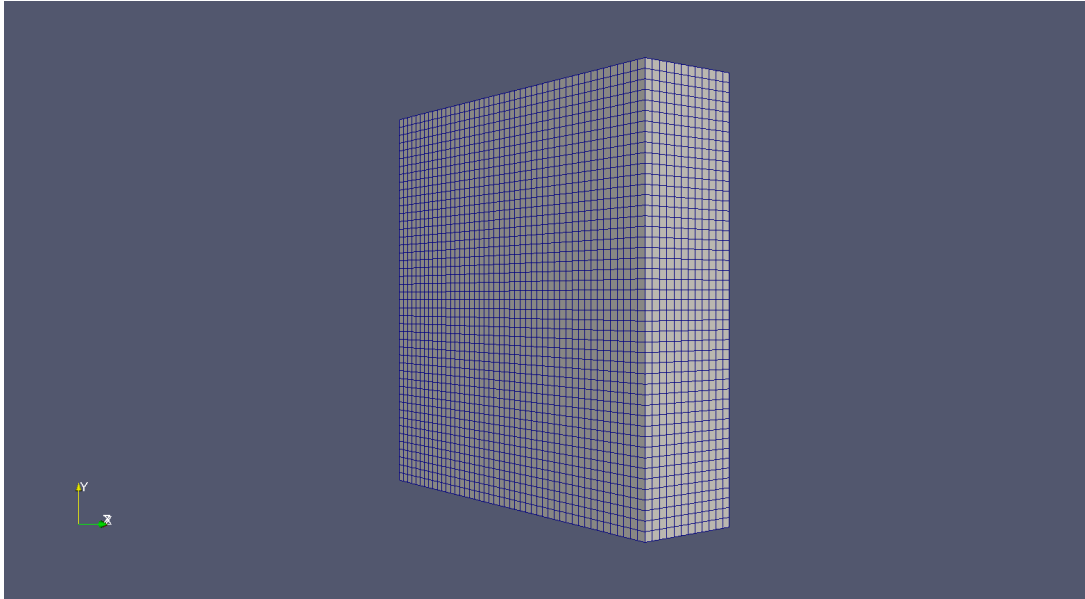


Figure 3.2: blockMeshcaption

### 3.1.3 Mesh refinement

To refine the mesh, the *snappyHexMesh* utility is used (included in *OpenFOAM*) and several parameters have to be modified in order to obtain a dense mesh that is suitable for the simulation of this complex geometry. It has to be considered that a particular geometry has a relative velocity as well; this is, the rotor is rotating, and so the first and second stages of the Low Pressure Compression of the turbofan engine, while the nacelle and the combustor are static.

In the `system/snappyHexMeshDict` several parameters have to be modified. First, the *snappyHexMesh* must be aware of the geometries that it has to take into account. As it can be seen below, the files *Fan.stl*, *HPSpool.stl*, *LPSpool.stl* and *NacelleStator.stl* have been included here.

```
geometry
{
    box1x1x1
    {
        type searchableBox;
        min (1.5 1 -0.5);
```

```

        max (3.5 2 0.5);
    }

    Fan.stl
    {
        type triSurfaceMesh;
    name Fan;
    }

    HPSPool.stl
    {
        type triSurfaceMesh;
    name HPSPool;
    }

    LPSPool.stl
    {
        type triSurfaceMesh;
    name LPSPool;
    }

    NacelleStator.stl
    {
        type triSurfaceMesh;
    name NacelleStator;
    }

    AMI.stl    //used for the dynamic mesh (the domain where the mesh rotating)
    {
        type          triSurfaceMesh;
        name          AMI;
    }
};

```

Next, the number of control volumes has to be limited to ensure that the laptop is capable of running a simulation. This number has been limited to four million cells, which is a pretty high number and the following lines have to be modified. Also, the maximum local number of control volumes has been limited to two million.



```

// Refinement parameters
// ~~~~~

// If local number of cells is >= maxLocalCells on any processor
// switches from from refinement followed by balancing
// (current method) to (weighted) balancing before refinement.
maxLocalCells 2000000;

// Overall cell limit (approximately). Refinement will stop immediately
// upon reaching this number so a refinement level might not complete.
// Note that this is the number of cells before removing the part which
// is not 'visible' from the keepPoint. The final number of cells might
// actually be a lot less.
maxGlobalCells 4000000;

```

The next step is to define the required minimum and maximum refinement levels of the mesh for the different geometries that will be simulated. It can be clearly seen in this section that we have included the same *STL* files that we did before. The higher the level of the refinement, the denser the mesh will be and the better it will resemble to the real geometry. But the limitation here is the computational power available so, the maximum refinement number cannot be as high as we would like to. Thus, depending on the complexity of the geometry, several minimum (the first number) and maximum (the second number) refinement levels have been defined.

```

// Surface based refinement
// ~~~~~

// Specifies two levels for every surface. The first is the minimum level,
// every cell intersecting a surface gets refined up to the minimum level.
// The second level is the maximum level. Cells that 'see' multiple
// intersections where the intersections make an
// angle > resolveFeatureAngle get refined up to the maximum level.

refinementSurfaces
{
    LPSpool
    {

```

```

        // Surface-wise min and max refinement level
        level (3 4);
        patchInfo
        {
            type wall;
            inGroups (movingWalls);
        }
    }

    NacelleStator
    {
        // Surface-wise min and max refinement level
        level (2 3);
        patchInfo
        {
            type wall;
            inGroups (staticWalls);
        }
    }

    Fan
    {
        // Surface-wise min and max refinement level
        level (1 2);
        patchInfo
        {
            type wall;
            inGroups (movingWalls);
        }
    }

    HPSpool
    {
        // Surface-wise min and max refinement level
        level (1 2);
        patchInfo
        {
            type wall;
            inGroups (movingWalls);
        }
    }
}

```

```

AMI
{
    level (2 3);

    faceType boundary;
    cellZone innerAMI;
    faceZone innerAMI;
    cellZoneInside inside;
}
}

```

It is worth mentioning that new zones have been created in the file. As it can be seen above, the 'AMI' zone has been defined given that it will be the rotating zone of the turbofan. Also, besides the level of refinement for each geometry, it can be seen that it has been defined a type of group for each of them. So, the zones are now assembled into either the rotating geometry or static geometry.

```

LPSPool
{
    // Surface-wise min and max refinement level
    level (3 4);
    patchInfo
    {
        type wall;
        inGroups (movingWalls);
    }    //~MODIFIED PART
}

```

The next step is to select a point within the mesh. So, the *locationInMesh* has to be modified with the x, y, and z-coordinates of a point with that feature.

```

// Mesh selection
// ~~~~~

// After refinement patches get added for all refinementSurfaces and
// all cells intersecting the surfaces get put into these patches. The

```

```

// section reachable from the locationInMesh is kept.
// NOTE: This point should never be on a face, always inside a cell, even
// after refinement.
// This is an outside point locationInMesh (-0.033 -0.033 0.0033);
locationInMesh (.97443222 1.40534444343 1.24221211); // Inside point

// Whether any faceZones (as specified in the refinementSurfaces)
// are only on the boundary of corresponding cellZones or also allow
// free-standing zone faces. Not used if there are no faceZones.
allowFreeStandingZoneFaces false;

```

Finally, the *surfaceFeatureExtract* has been used. What this option does it to refine even more the mesh near the points that have complex geometries such as the edges of the blades. This is particularly useful for the turbofan given that it has a high number of blades and a twisting geometry that will resemble more to the reality when using this option. Thus, the following lines within the *snappyHexMeshDict* must be modified.

```

// Explicit feature edge refinement
// ~~~~~

// Specifies a level for any cell intersected by its edges.
// This is a featureEdgeMesh, read from constant/triSurface for now.
features
(
    {
        file "NacelleStator.eMesh";
        level 2;
        // levels ((0.0 2) (1.0 3));
    }
    {
        file "LPSPool.eMesh";
        level 3;
        // levels ((0.0 2) (1.0 3));
    }
);

```

All of the other parameters of the *snappyHexMeshDict* have not been modified.

### 3.1.4 Comparison of different types of mesh

A comparison between the meshes generated is presented below. Several meshes have been made in order to select the best one to perform the simulation of the given case.

The mesh that has been used to simulate the case is the **Standard Mesh**. Since the computational power that we have is limited, a very dense mesh would end up in a very tedious process that could need several hours to be simulated. Thus, given that the standard mesh is pretty good already, we have selected it because the results will also be quite similar.

The aim of this section is to show that the difference between the dense mesh and the standard mesh is relatively small. The standard mesh has 2500000 number of cells, approximately, while the dense mesh has about 3500000 number of cells. It might seem a big difference but the critical geometries are really well meshed in both cases and only the non-critical parts of the turbofan (this is: the flow inside the fan duct) has a smaller number of cells in the standard mesh.

#### 3.1.4.1 Coarse mesh

The number of control volumes in the mesh is 40, which is a very small number given the size of the turbofan. It can be seen in 3.3 and 3.4 that this mesh cannot be used to run a simulation.

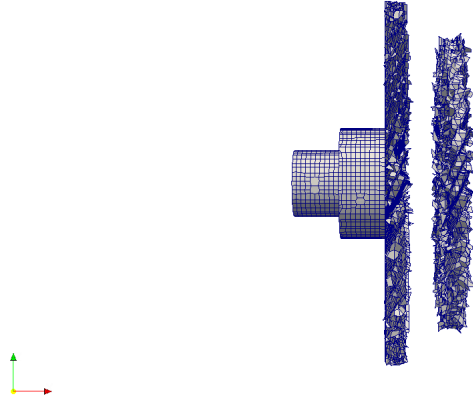


Figure 3.3: Detail of the coarse mesh

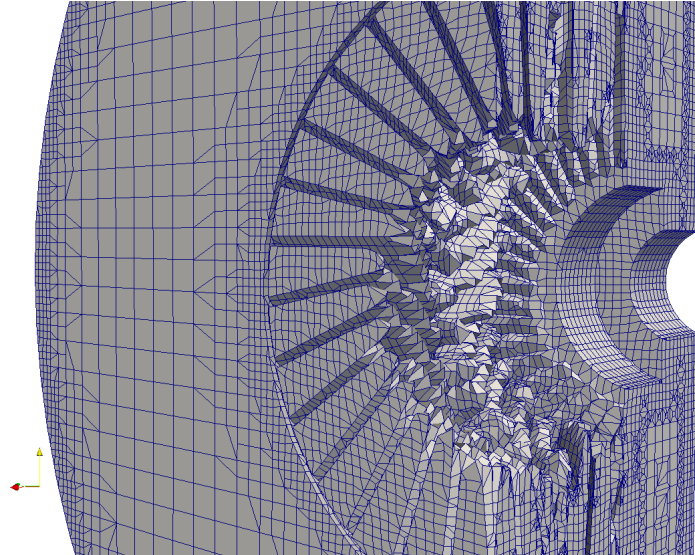


Figure 3.4: Detail of the coarse mesh

#### 3.1.4.2 Standard mesh

Next, it is presented the standard mesh. It has a higher number of cells compared to the coarse mesh, but it cannot be used either given that the size of the turbine is pretty big. The figures 3.5 and 3.6 are a proof of that.

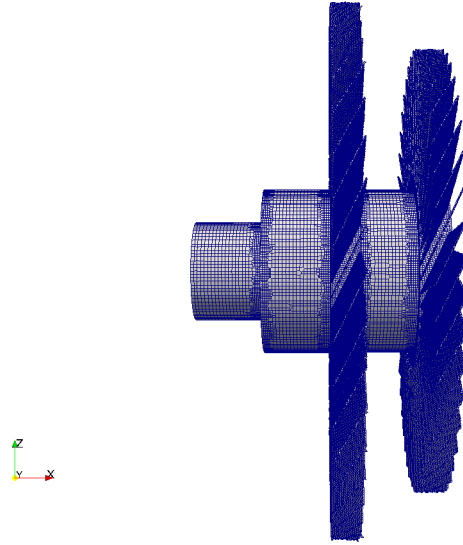


Figure 3.5: Detail of the standard mesh

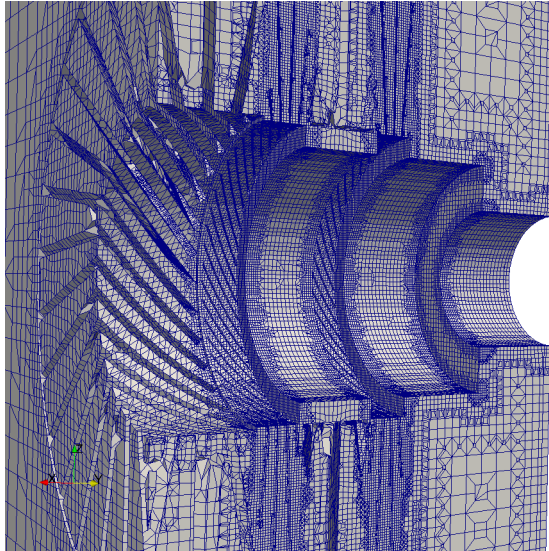


Figure 3.6: Detail of the standard mesh

### 3.1.4.3 Dense mesh

Finally, the dense mesh is presented in the figures 3.7 and 3.8. This is the best mesh that has been generated for the case but for the reasons commented below, this is not the mesh used to run the simulation.

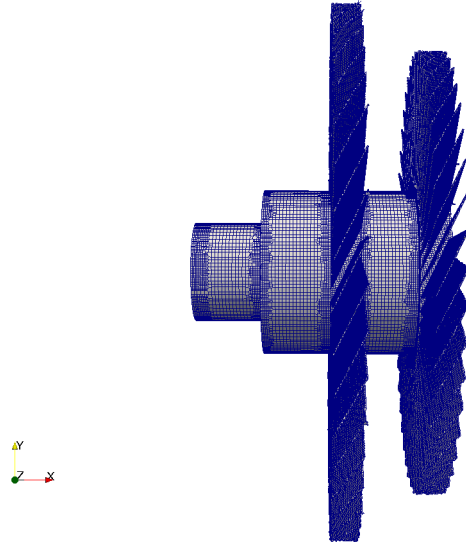


Figure 3.7: Detail of the dense mesh

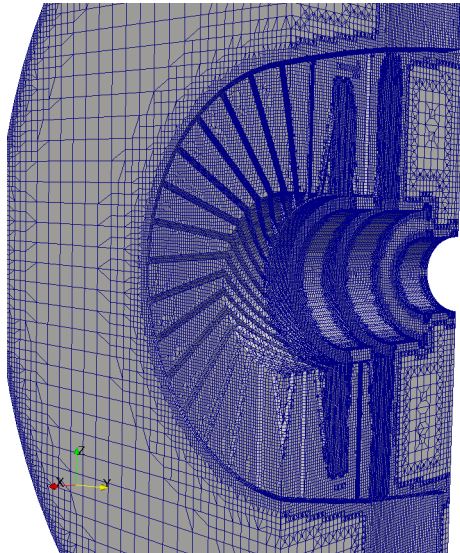


Figure 3.8: Detail of the dense mesh



To end with this section, the *log* file that has been obtained when the mesh has been defined with the *snappyHexMesh* utility is presented below. It has to be taken into account that it is a huge file since the number of iterations is really big and only the final part of the *log* file is shown.

```

/*-----*\
| =====|
| \\      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O p e r a t i o n      | Version: 4.0 |
|  \\    /   A n d      | Web:      www.OpenFOAM.org |
|   \\/      M a n i p u l a t i o n      | |
\*-----*/
Build   : 4.0-665f1db4c1f1
Exec    : /opt/openfoam4/platforms/linux64GccDPInt32Opt/bin/snappyHexMesh -overwrite
Date    : Dec 05 2016
Time    : 14:17:06
Host    : "victorPC"
PID     : 4504
Case    : /home/victor/Desktop/Final/pimpleDyMFoam/turbofan_std
nProcs  : 1
sigFpe  : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Allowing user-supplied system call operations

// * * * * *

[...]
```

Snapped mesh : cells:3511624 faces:11081993 points:4202883

Cells per refinement level:

```

0 4602
1 38356
2 198997
3 1304463
4 1965206
```

Writing mesh to time constant

Wrote mesh in = 153.87 s.

Mesh snapped in = 1259.68 s.

Checking final mesh ...

Checking faces in error :

```

non-orthogonality > 65 degrees : 0
faces with face pyramid volume < 1e-13 : 0
faces with face-decomposition tet quality < 1e-15 : 0
faces with concavity > 80 degrees : 0
faces with skewness > 4 (internal) or 20 (boundary) : 0
faces with interpolation weights (0..1) < 0.02 : 0
faces with volume ratio of neighbour cells < 0.01 : 0
faces with face twist < 0.02 : 0
faces on cells with determinant < 0.001 : 0
Finished meshing without any errors
Finished meshing in = 2418.34 s.
End

```

## 3.2 checkMesh

To view whether the mesh has been generated with error or not, the command `checkMesh` has been typed. This will show some information about the mesh, including the number of faces or cells, as well as some other information. The log obtained is presented below.

```
/*-----*\
| ===== |
| \\      / F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O peration  | Version: 4.0 |
|  \\    /  A nd        | Web:      www.OpenFOAM.org |
|   \\/      M anipulation |
\*-----*/
Build : 4.0-665f1db4c1f1
Exec : checkMesh
Date : Dec 05 2016
Time : 19:24:31
Host : "victorPC"
PID : 4598
Case : /home/victor/Desktop/turbofan
nProcs : 1
sigFpe : Enabling floating point exception trapping (FOAM_SIGFPE).
fileModificationChecking : Monitoring run-time modified files using timeStampMaster
allowSystemOperations : Allowing user-supplied system call operations

// * * * * * //
Create time

--> FOAM Warning :
    From function static Foam::instantList Foam::timeSelector::select0(Foam::Time&, const Foam
    in file db/Time/timeSelector.C at line 270
    No time specified or available, selecting 'constant'
Create polyMesh for time = constant

Time = constant

Mesh stats
  points:          3041731
  faces:           7969935
  internal faces:  7230304
```

cells: 2522441  
faces per cell: 6.026  
boundary patches: 6  
point zones: 0  
face zones: 1  
cell zones: 1

Overall number of cells of each type:

hexahedra: 2085085  
prisms: 105950  
wedges: 0  
pyramids: 0  
tet wedges: 1267  
tetrahedra: 27  
polyhedra: 330112

Breakdown of polyhedra by number of faces:

faces	number of cells
4	116290
5	77755
6	36807
7	2382
8	2453
9	51174
10	607
11	770
12	22688
13	274
14	389
15	15309
16	30
17	72
18	3111
21	1

Checking topology...

Boundary definition OK.  
Cell to face addressing OK.  
Point usage OK.  
Upper triangular ordering OK.  
Face vertices OK.  
Number of regions: 1 (OK).

Checking patch topology for multiply connected surfaces...

Patch	Faces	Points	Surface topology
inlet	11685	12488	ok (non-closed singly connected)
outlet	6410	7707	ok (non-closed singly connected)
LPSpool	537648	631640	ok (non-closed singly connected)
NacelleStator	172722	217913	multiply connected (shared edge)
AMI1	5583	5775	ok (non-closed singly connected)
AMI2	5583	5811	ok (non-closed singly connected)

<<Writing 2 conflicting points to set nonManifoldPoints

Checking geometry...

Overall domain bounding box (0.77 0.115495 0.106656) (1.35 2.21443 2.20599)

Mesh has 3 geometric (non-empty/wedge) directions (1 1 1)

Mesh has 3 solution (non-empty) directions (1 1 1)

Boundary openness (4.12283e-16 -1.46559e-16 1.42131e-16) OK.

Max cell openness = 4.10842e-16 OK.

Max aspect ratio = 10.7936 OK.

Minimum face area = 8.56533e-10. Maximum face area = 0.002683. Face area magnitudes OK.

Min volume = 1.88435e-09. Max volume = 0.000135167. Total volume = 1.87506. Cell volumes

Mesh non-orthogonality Max: 64.9921 average: 11.5854

Non-orthogonality check OK.

Face pyramids OK.

\*\*\*Max skewness = 13.9612, 292 highly skew faces detected which may impair the quality of the

<<Writing 292 skew faces to set skewFaces

Coupled point location match (average 0) OK.

End

### 3.3 Rotation

An important consideration for this project is that the rotor is spinning at a high speed and this condition has to be somehow communicated to *OpenFOAM*. To work with this condition there are three possible approaches, which are widely described in [1], the Single Rotation Frame (SRF), the Multiple Reference Frame (MRF) and the Arbitrary Mesh Interface (AMI). In this particular case AMI is going to be used. The method itself is based on defining a sliding mesh where the part of the mesh that moves is rotating in every timestep and the values of the cells lying on the interface are interpolated to update the mesh in every timestep.

To apply this method, a solid containing the "moving" domain has to be defined as a `cellZone`. How to do that has been shown in the previous section. In our case the moving domain is the one contained between the low-pressure compressor and the nearest wall as shown in the figure 3.9.

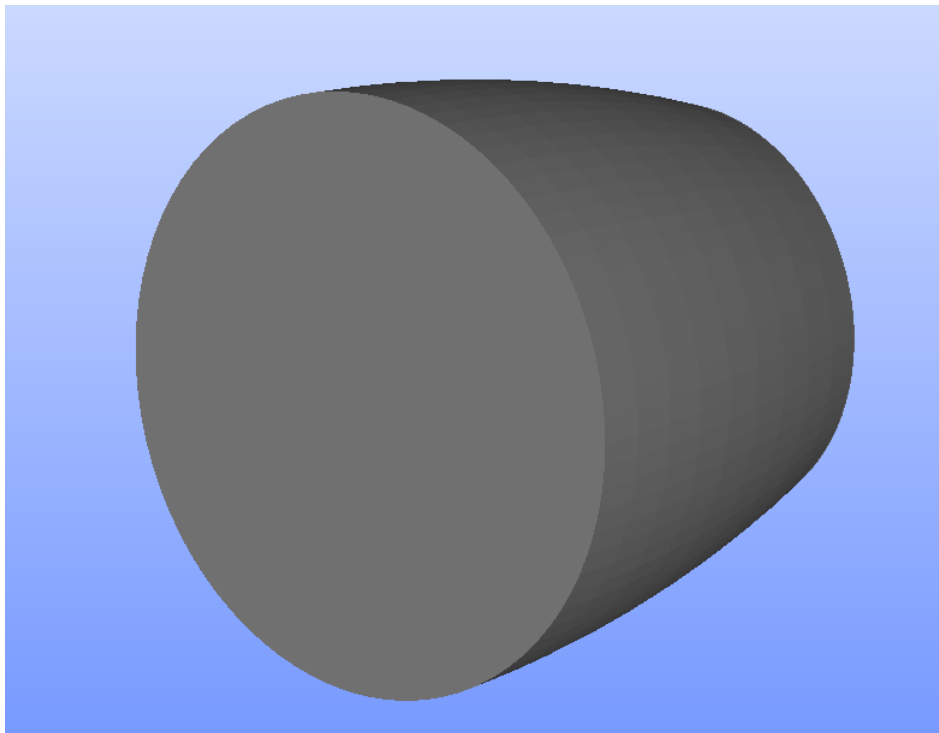


Figure 3.9: Rotating zone solid

To define the parameters related to the rotating zone such as the origin, the axis of rotation, the angular velocity and of course the zone which will be moving the *constant/dynamicMeshDict* file has to be modified with the appropriate parameters for the case. Since the axis of *OpenFOAM* are based on the right hand rule, the rotation axis must be defined as follows: (1 0 0). From information found in different references, it has been chosen 5000 *rpm* as the angular velocity of the rotor. It should be noted that the units used must be those of the International System, so the 5000 *rpm* must be converted to *rad/s*. Furthermore, an origin point has to be indicated in the axis of rotation; in this case, the origin point is as follows: (0 1.1675354 1.15542633). Finally the file will look like:

```

/*-----*- C++ -*-----*\
| =====|
| \\\ / F i e l d | OpenFOAM: The Open Source CFD Toolbox |
| \\\ / O p e r a t i o n | Version: 4.x |
| \\\ / A n d | Web: www.OpenFOAM.org |
| \\\ / M a n i p u l a t i o n | |
\*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       dynamicMeshDict;
}
// * * * * *

dynamicFvMesh    solidBodyMotionFvMesh;

motionSolverLibs ( "libfvMotionSolvers.so" );

solidBodyMotionFvMeshCoeffs
{
    cellZone      innerAMI;

    solidBodyMotionFunction    rotatingMotion;
    rotatingMotionCoeffs

```

```

    {
        origin      (0 1.1675354 1.15542633);
        axis        (1 0 0);
        omega       523; // rad/s
    }
}

```

```
// ***** //
```

Where `cellZone innerAMI;` is used to define the rotating zone. It can clearly be seen that the modification of any parameter is pretty straight-forward in the file since there is no possible confusion with the names. Another important thing to do when working with AMI is to define 2 patches related to this *"moving zone"*. These patches can be created using the *createPatch* routine. To do so, a dictionary (*createPatchDict*) has to be included in the system folder. For this case, the dictionary file can be as follows:

```

/*-----*- C++ -*-----*\
| ===== | |
| \\      / F i e l d | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O p e r a t i o n | Version: 2.3.1 |
|  \\    / A n d | Web: www.OpenFOAM.org |
|   \\ / M a n i p u l a t i o n | |
\*-----*-*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    object       createPatchDict;
}
// * * * * *

```

```

// Do a synchronisation of coupled points after creation of any patches.
// Note: this does not work with points that are on multiple coupled patches
//       with transformations (i.e. cyclics).
pointSync false;

```



```

// Patches to create.
patches
(
    {
        //- Master side patch
        name          AMI1;
        patchInfo
        {
            type          cyclicAMI;
            matchTolerance 0.0001;
            neighbourPatch AMI2;
            transform      noOrdering;
        }
        constructFrom patches;
        patches (AMI);
    }

    {
        //- Slave side patch
        name          AMI2;
        patchInfo
        {
            type          cyclicAMI;
            matchTolerance 0.0001;
            neighbourPatch AMI1;
            transform      noOrdering;
        }
        constructFrom patches;
        patches (AMI_slave);
    }
);

// *****

```

This method might seem more complicated than simply studying the case with MRF but the results obtained for this particular case are better with the scheme used in this report.

## 3.4 Boundary conditions

### 3.4.1 Velocity

Some initial conditions and values have to be defined in order to run the simulation. The initial velocity field and pressure are especially important since these inputs are the starting values of the simulation. So, it is clear that the values at the *inlet*, which was defined within the previous sections, have to be defined. Additionally, the values at the *outlet* of the geometry have to be defined as well, but they can be, unlike the *inlet*, not fixed values. Thus, the following modifications have to be made in the `0.orig/U`.

```
/*-----*- C++ -*-----*\
| =====|
| \\      / F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O peration  | Version: 4.0                          |
|  \\     / A nd        | Web:      www.OpenFOAM.org            |
|   \\/    M anipulation |                                     |
\*-----*-*/

FoamFile
{
    version      2.0;
    format       binary;
    class        volVectorField;
    location     "0";
    object       U;
}

// * * * * *

dimensions      [0 1 -1 0 0 0 0];

internalField    uniform (0 0 0);

boundaryField
{
    inlet
    {
        type      fixedValue;
        value      uniform (200 0 0);
    }
}
```

```

outlet
{
    type            inletOutlet;
    inletValue      uniform (0 0 0);
    value           uniform (0 0 0);
}
LPSPool
{
    type            movingWallVelocity;
    value           uniform (0 0 0);
}
NacelleStator
{
    type            noSlip;
}
AMI1
{
    type            cyclicAMI;
    value           uniform (0 0 0);
}
AMI2
{
    type            cyclicAMI;
    value           uniform (0 0 0);
}
}

// *****

```

As it can be seen above, the value of the velocity at the inlet has been set to  $200\text{ m/s}$  (about  $720\text{ km/h}$ ). The *inletOutlet* condition has been set to the outlet (this is: the flow is restricted to move always forwards and reverse flow cannot appear at this region). The NacelleStator file has the noSlip condition given that the fluid is supposed to be Newtonian. Finally, the initial velocity field for the rotational parts AMI1 and AMI2 have been set to 0 (this is: the flow is not moving at the beginning of the simulation).

### 3.4.2 Pressure

The initial conditions for the pressure have to be evaluated as well in the 0.orig/p file.

```
/*-----*- C++ -*-----*\
| ===== |
| \\      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O p e r a t i o n | Version: 4.0 |
| \\      / A n d | Web: www.OpenFOAM.org |
|  \\/      M a n i p u l a t i o n |
\*-----*-*/
FoamFile
{
    version      2.0;
    format       binary;
    class        volScalarField;
    location     "0";
    object       p;
}
// * * * * *

dimensions      [0 2 -2 0 0 0 0];

internalField    uniform 0;

boundaryField
{
    inlet
    {
        type      zeroGradient;
    }
    outlet
    {
        type      fixedValue;
        value      uniform 0;
    }
    LPSpool
    {
        type      zeroGradient;
    }
}
```

```

NacelleStator
{
    type          zeroGradient;
}
AMI1
{
    type          cyclicAMI;
    value         uniform 0;
}
AMI2
{
    type          cyclicAMI;
    value         uniform 0;
}
}

// *****

```

The *zeroGradient* condition simply extrapolates the quantity to the current surface from the nearest cell value meaning that the gradient of the quantity is equal to zero in the direction perpendicular to the patch. This condition will not fix a value for the pressure, it will just let the pressure be the same as the calculated using the Navier Stokes equations (provided that the velocity field is known).

Finally, the boundary conditions for the *AMI1* and *AMI2* should not be modified since they are specifically set for a geometry that is rotating.

### 3.5 Properties of the flow

The properties of the flow have to be defined taking into account the hypotheses made for the case. As mentioned in previous sections, the flow is assumed to be Newtonian. This property can be modified in the `constant/transportProperties` file. Also, since the flow we are dealing with is air, the kinematic viscosity has to be set to  $1e6 m^2/s$ .

```

/*-----*- C++ -*-----*\
| =====|
| \\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |
| \\\ / O peration | Version: 4.x |
| \\\ / A nd | Web: www.OpenFOAM.org |
| \\\ / M anipulation | |
\*-----*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       transportProperties;
}
// * * * * *

transportModel  Newtonian;

nu              [0 2 -1 0 0 0 0] 1e-6;

// * * * * *

```

Another hypotheses that has been made in order to alleviate the computation time is that the flow is laminar, although it will not be a very realistic flow. To impose this condition, a modification has to be made in `constant/turbulenceProperties` file and it is presented below.

```

/*-----*- C++ -*-----*\
| =====|
| \\\ / F ield | OpenFOAM: The Open Source CFD Toolbox |
| \\\ / O peration | Version: 4.x |

```

```

|  \ \ /  A nd          | Web:      www.OpenFOAM.org          |
|  \ \ /  M anipulation |                                     |
\*-----* /
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "constant";
    object       turbulenceProperties;
}
// * * * * *

simulationType  laminar;

// *****

```

Since we are only interested in the topology of the flow of this file, other parameters have not been modified.

### 3.6 controlDict - parameters for the simulation

This file is really important for the simulation since it contains the timesteps and other variables such as the `endTime` or `startTime`. To begin, the initial and final times of the simulation have to be defined. In this case, the `startTime` is 0 s and the `endTime` has been set to  $1e^{-3}$ s. These changes can be made in the `system/controlDict` file. Furthermore, this file also states the application chosen (*pimpleDyMFoam*), a modified `pimpleFoam` that handles dynamic meshes.

```
application      pimpleDyMFoam;

startFrom        startTime;

startTime        0;

stopAt           endTime;

endTime          1e-3;
```

The next thing that has to be done is to define the time step for the simulation as well as the write interval (this is; the number of time that must pass in order to make a folder for the simulation in the current simulation time).

```
deltaT .5e-4;
writeControl     timeStep;
writeInterval 1;

purgeWrite       0;

writeFormat      binary;

writePrecision   6;

writeCompression off;

timeFormat       general;

timePrecision    6;
```



The *runTimeModifiable* modifies the deltaT in each iteration to make possible for the solver to converge. Also, the maximum Courant number has been set to 2. Finally, the file that has been used to run the simulation is presented below.

```

/*-----*- C++ -*-----*\
| ===== |
| \\ / F i e l d | OpenFOAM: The Open Source CFD Toolbox |
| \\ / O p e r a t i o n | Version: 4.x |
| \\ / A n d | Web: www.OpenFOAM.org |
| \\ / M a n i p u l a t i o n | |
\*-----*-*/
FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "system";
    object       controlDict;
}
// * * * * *

application      pimpleDyMFoam;

startFrom        startTime;

startTime        0;

stopAt           endTime;

endTime          1e-4;

deltaT           .5e-4;
writeControl      timeStep;
writeInterval     1;

purgeWrite       0;

writeFormat       binary;

writePrecision    6;

```

```
writeCompression off;

timeFormat      general;

timePrecision   6;

runTimeModifiable true;

adjustTimeStep  yes;

maxCo          2;

// ***** //
```

### 3.7 fvSchemes and fvSolution

The fvSchemes dictionary contains the numerical schemes that they are used in the simulation. In the hypotheses section we have considered that the flow is transient To impose this condition we must modify the `system/fvSchemes` file, the section of *ddtSchemes*, which is the temporal integration scheme. We must write Euler. Finally, the file that has been used to run the simulation is presented below.

```

/*-----*- C++ -*-----*/
| ===== |
| \ \      / F i e l d      | OpenFOAM: The Open Source CFD Toolbox |
| \ \      / O peration     | Version: 4.x |
| \ \      / A nd           | Web:      www.OpenFOAM.org |
| \ \      / M anipulation  | |
/*-----*- C++ -*-----*/

FoamFile
{
    version      2.0;
    format       ascii;
    class        dictionary;
    location     "system";
    object       fvSchemes;
}

// * * * * *

ddtSchemes
{
    default      Euler;
}

gradSchemes
{
    default      Gauss linear;
    grad(p)      Gauss linear;
    grad(U)      cellLimited Gauss linear 1;
}

divSchemes
{

```

```

        default          none;
//    div(phi,U)          Gauss upwind;
    div(phi,U)          Gauss linearUpwind grad(U);
    div((nuEff*dev2(T(grad(U)))) Gauss linear;
}

laplacianSchemes
{
    default              Gauss linear limited corrected 0.33;
}

interpolationSchemes
{
    default              linear;
}

snGradSchemes
{
    default              limited corrected 0.33;
}

// *****

```

The `system/fvSolution` file contains the resolution methods and the tolerances for each equation, but in our case it has not been modified. So, the file `fvSolution` for the current case is the following:

```

/*-----*- C++ -*-----*\
| ===== |
| \\      / F ield      | OpenFOAM: The Open Source CFD Toolbox |
| \\      / O peration  | Version: 4.x |
|  \\    / A nd         | Web:      www.OpenFOAM.org |
|   \\/    M anipulation |
\*-----*-*/

FoamFile
{
    version      2.0;
    format       ascii;

```

```

        class      dictionary;
        object      fvSolution;
    }
    // * * * * *
    solvers
    {
        pcorr
        {
            solver      GAMG;
            tolerance    1e-2;
            relTol       0;
            smoother     DICGaussSeidel;
            cacheAgglomeration no;
            maxIter       50;
        }

        p
        {
            $pcorr;
            tolerance     1e-5;
            relTol        0.01;
        }

        pFinal
        {
            $p;
            tolerance     1e-6;
            relTol        0;
        }

        U
        {
            solver      smoothSolver;
            smoother     symGaussSeidel;
            tolerance    1e-6;
            relTol       0.1;
        }

        UFinal
        {

```

```

        solver      smoothSolver;
        smoother     symGaussSeidel;
        tolerance    1e-6;
        relTol       0;
    }
}

PIMPLE
{
    correctPhi       no;
    nOuterCorrectors 3;
    nCorrectors       1;
    nNonOrthogonalCorrectors 0;
}

relaxationFactors
{
    U      0.5;
    UFinal 1;
}

cache
{
    grad(U);
}

// ***** //

```

## 4 Simulation of the turbofan

### 4.1 Selection of the solver

The solver used is the *pimpleDyMFoam*. It solves large timestep transient, incompressible flow problems including dynamic meshes using the PIMPLE algorithm which is based on the combination of *SIMPLE* and *PISO* solvers. To correctly understand how the solver works, a briefly explanation of the two solvers will be done in the following pages.

#### 4.1.1 SIMPLE algorithm

The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm uses the Navier-Stokes equations for a single-phase flow with a constant density and viscosity written below:

$$\nabla \vec{u} = 0 \quad (4.1)$$

$$\frac{\partial U}{\partial t} + \nabla \Delta(\vec{u}\vec{u}) - \nabla \Delta(\nu \nabla \vec{u}) = -\nabla p \quad (4.2)$$

The pressure equation is deduced from the momentum equation (the procedure can be reviewed in the *OpenFoam* literature) and is:

$$\nabla \left( \frac{1}{a_p} \nabla p \right) = \nabla \left( \frac{H(\vec{U})}{a_p} \right) = \sum_f \vec{S} \left( \frac{H(\vec{U})}{a_p} \right)_f \quad (4.3)$$

The SIMPLE algorithm is used for solving steady-state problems iteratively. In this situations, is not necessary to solve completely the coupling between velocity and pressure, so the simple algorithm works as follows:

1. Solving the momentum equation gives an approximation of the velocity using as the pressure gradient the previous iteration pressure distribution.
2. The current timestep pressure is later calculated using the pressure equation.

3. The velocities are corrected using the new pressure gradient.

As it can be seen, it uses an implicit method for solving the momentum equation and the pressure equation, but then, it corrects. If a steady-state problem is being solved iteratively, it is not necessary to fully resolve the linear pressure-velocity coupling, as the changes between consecutive solutions are no longer small. The SIMPLE algorithm:

#### 4.1.2 PISO algorithm

The PISO (Pressure Implicit with Split Operator) algorithm is an algorithm used for non-iterative, large time-steps and unsteady problems. It involves one predictor step and two corrector steps and it is designed to fulfill the mass conservation equation. The algorithm is the following.

1. Solve a discretized momentum equation to compute an intermediate velocity field.
2. Compute the mass fluxes at the cell faces.
3. Solve the pressure equation.
4. Correct the mass fluxes.
5. Correct the velocities using the new pressure distribution.
6. Recalculate the boundary conditions.
7. Repeat the process a prescribed number of times.
8. Increase the time step and repeat from 1.

As it can be seen the predictor step is set when computing the  $p$  distribution and then the two corrector steps are applied. With PISO the timestep can be larger than the used with SIMPLE.

In summary, the PIMPLE algorithm is a combination between these two schemes, acquiring the capacity to use larger timesteps than SIMPLE and speeding up the simulation since it can use PISO (non iterative) for some of the cells.



## 4.2 Commands to run the simulation

The first thing to perform the simulation is to copy the tutorial folder case into our current directory. To do so, open a terminal window and type the following command:

```
cp -r /opt/openfoam4/tutorials/incompressible/  
pimpleDyMFoam/mixerVesselAMI2D/* .
```

Then we have to modify all the parameters explained during this report. The first thing that must be done when the *blockMeshDict* has been modified is to type the following lines in a terminal. As it can be seen, it will save the log on a new file named *blockMesh.log*.

```
blockMesh > blockMesh.log
```

When the *blockMesh* has been generated, the next thing to do is the refinement of the mesh. So, the *snappyHexMesh* utility will be used, provided that all the parameters in *system/snappyHexMeshDict* have been modified. In the terminal, the following command has to be typed:

```
foamJob decomposePar  
foamJob -p surfaceFeatureExtract  
foamJob -s -p snappyHexMesh -overwrite
```

The previous command will make the mesh refinement to run in parallel (notice the *decomposePar* and the *-p*). Additionally, a log will be created when running the command that will contain all the information about the mesh refinement. The case can be runned without using parallel computing by typing:

```
surfaceFeatureExtract > surfaceFeatureExtract  
foamJob -s snappyHexMesh -overwrite  
mv log snappyHexMesh.log
```

In order to run the desired simulation, a patch with the AMI surface has to be made. Thus, the following command has to be typed. It will also create a *log* file named *createPatch.log*.

```
createPatch -overwrite > createPatch.log
```

Next, the simulation can be run.

```
cp -r 0.orig 0
foamJob -s PimpleDyMFoam
mv log PimpleDyMFoam.log
```

And finally, if we were working in parallel (in that case a `-p` must be added to the `pimpleDyMFoam` command as we did in the previous steps), the case has to be reconstructed to show the results.

```
reconstructParMesh -constant
reconstructPar
```

There has also been added into the folder the `Allrun` file, which will run the case automatically.

To view the results, typing `paraFoam` in the `terminal` window should be enough. The `.zip` folder of the case delivered along with this report contains an `Allrun` and an `Allclean` scripts to automatize all these commands.

## 5 Post-process

## 5.1 Analysis of the results

NO SE QUE ESCRIBO...TENGO SUEÑO

## **5.2 Conclusions**

**NO SE QUE ESCRIBO...TENGO SUEÑO**

## 6 Difficulties that we faced

We have faced with several difficulties during the simulation of this case.

## 7 References

- [1] Fumiya Nozaki. CFD for Rotating Machinery. (May), 9240.
- [2] Jordi Casacuberta. Study of fluid dynamics applications in the field of engineering by using OpenFOAM. 2014.
- [3] Learner Feedback. Running in parallel. pages 45–54.
- [4] Eric Paterson. PENN STATE The icoFoam “ Cavity Tutorial ”. (April), 2010.
- [5] Olivier Petit. Mesh generation: different ways of creating the mesh. 2011.
- [6] Roberto Pieri. A tool for pre-processing : snappyHexMesh. (June), 2014.
- [7] Jone Rivrud Rygg. CFD Analysis of a Pelton Turbine in OpenFOAM. *Thesis*, (June):119, 2013.