# SCORG™ Setup for CFD Simulation of Twin Screw Machines with OpenFOAM

**SCORG™** is the CFD grid generation tool for rotary twin screw machines. The tool includes additional modules for designing and editing rotor profiles, executing a basic thermodynamic calculation based on quasi 1D chamber models and generating the deforming working chamber grids for selected commercial CFD solvers.

For more information on the product please visit the website: www.pdmanalysis.co.uk or refer to documentation help.

This guide lists the steps for setting up a CFD simulation for Roots Blower Compressor with SCORG™ and OpenFOAM Solver. The user is expected to be familiar with screw machines, CFD and OpenFOAM in order to be able to use these procedures. The setup steps here are demonstrated for Windows 10, x64 bit OS. Refer SCORG™ Installation Guide for system and hardware requirements. The OpenFOAM pre-processing, solving and post-processing steps are demonstrated for Ubuntu 18.04 and 16.04 OS. For more information about running OpenFOAM on Windows OS please visit the following websites:

https://www.openfoam.com/download/install-windows-10.php -- OpenCFD

https://openfoam.org/download/windows/ -- The OpenFOAM Foundation

## Table of Contents

# 1 Introduction

Screw Compressors are rotary positive displacement machines. Although the working principle of these machines is simple, the geometry of rotors which are in the form of multi -lobe helical screws meshing with each other, is making analysis by use of Computational Fluid Dynamics (CFD) challenging. The process starts when the lobes are engaged at one end, which creates continuous increase of the volume between the rotors and the casing which reduces pressure in the suction domain and draws the working fluid in. Further rotation of the rotors makes this volume between the rotors and the casing enclosed when the compression of fluid begins. This increases the pressure within the chamber. Further rotation exposes the pressurized fluid to the outlet port and the fluid is delivered (Stosic, et al., 2005). Similar process is occurring in other helical screw machines such as pumps, vacuum pumps, gear pumps, expanders, extruders and motors. The CFD is equally challenging in such machines due to sliding and stretching

The main objectives of CFD simulations of a screw compressor are to:

a. Obtain the pressure field inside the rotor chamber and in the suction and discharge domains. Example shown in *Figure 1-1*.
b. Obtain the velocity fields in critical regions of the computational domain.
c. Obtain temperature fields in critical regions of the computational domain.
d. Obtain integral parameters of the machine such as power, mass flow rate, discharge temperature, torques on the rotor shafts, etc.
e. Obtain the loads and temperatures on boundaries with solid parts of the machine for further structural and thermal analysis.

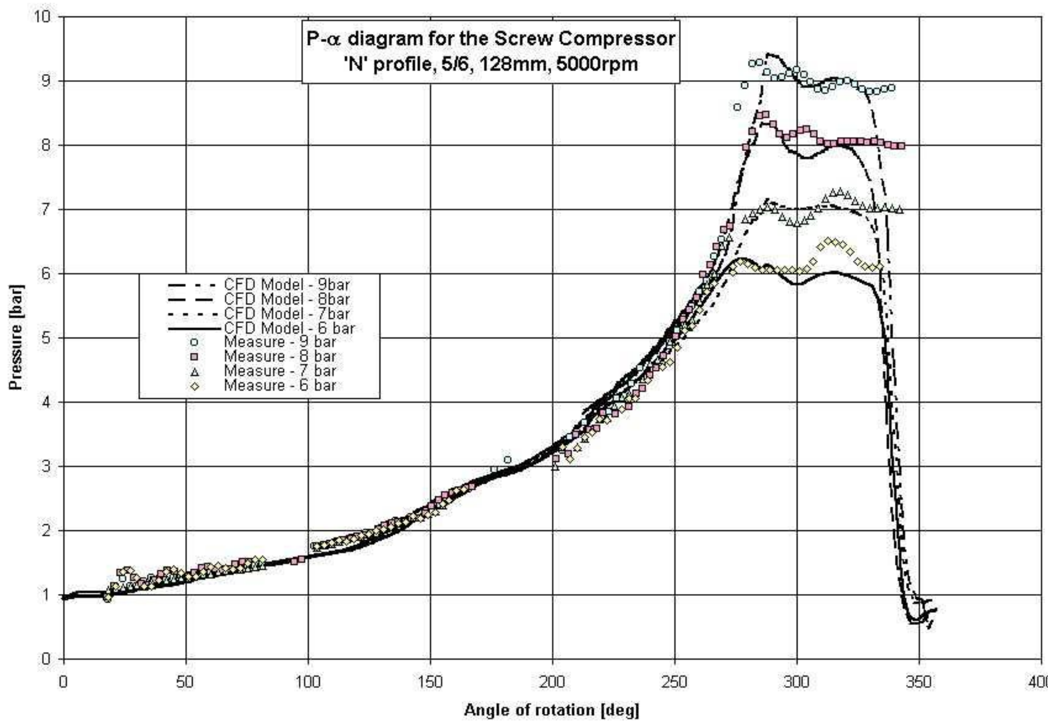
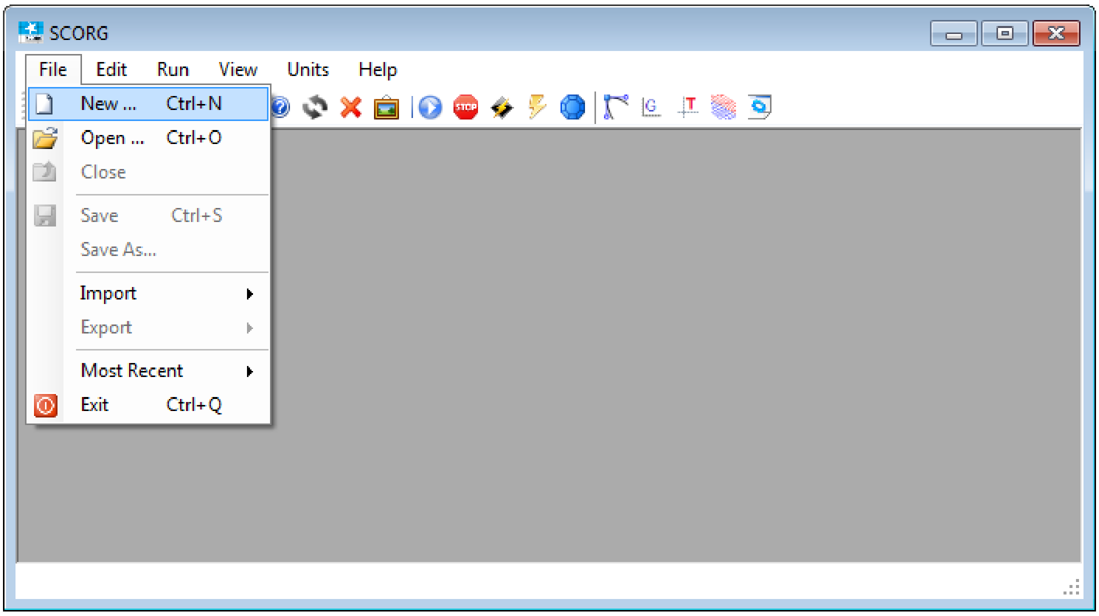



***Figure 1-1*** *Pressure Variation diagram of a Twin Screw Compressor* **(Kovacevic, et al., 2007)**

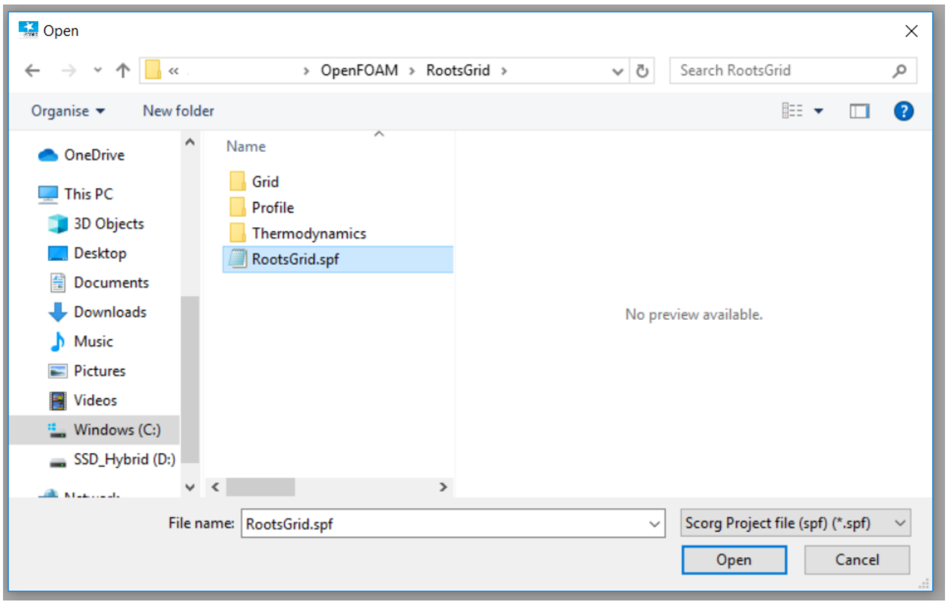
This Tutorial will provide a step by step guide for the procedure to setup and execute a typical twin screw compressor, pump or motor simulation. An example of a dry air Roots blower with 2/2 lobe combination has been considered.

## 2 SCORG™ Project

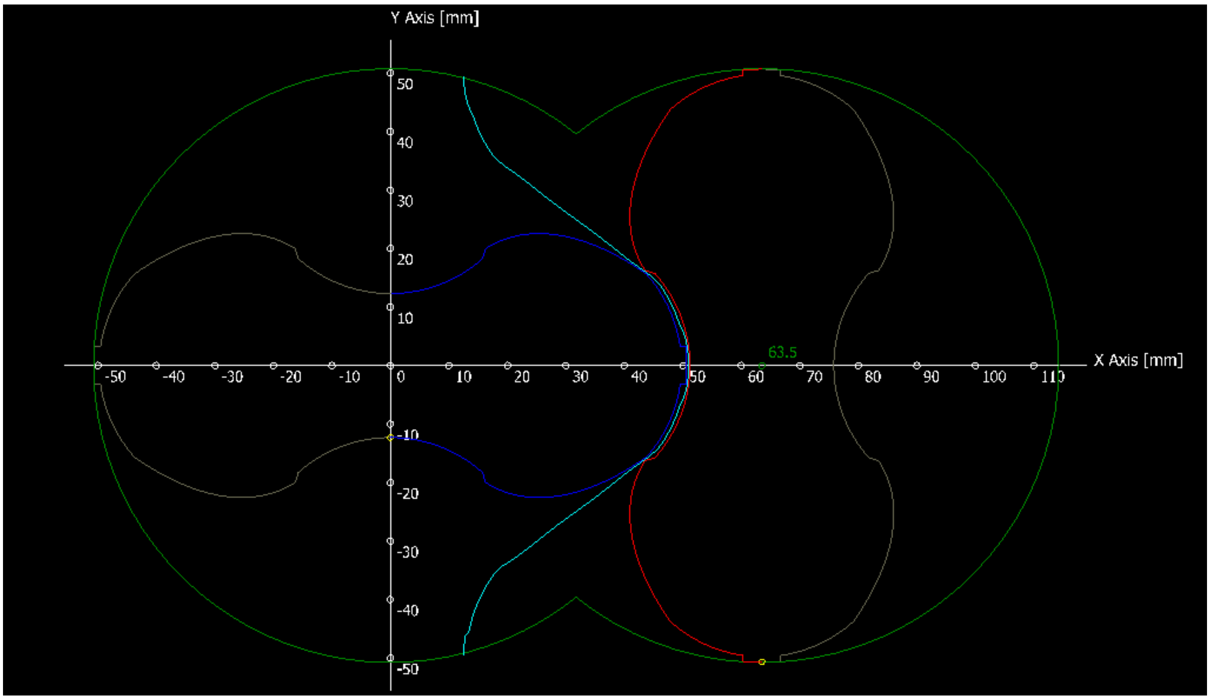
- Launch SCORG™ on the Desktop.
- Copy RootsGridProject.zip from Tutorials directory and unzip.
- Select File → Open



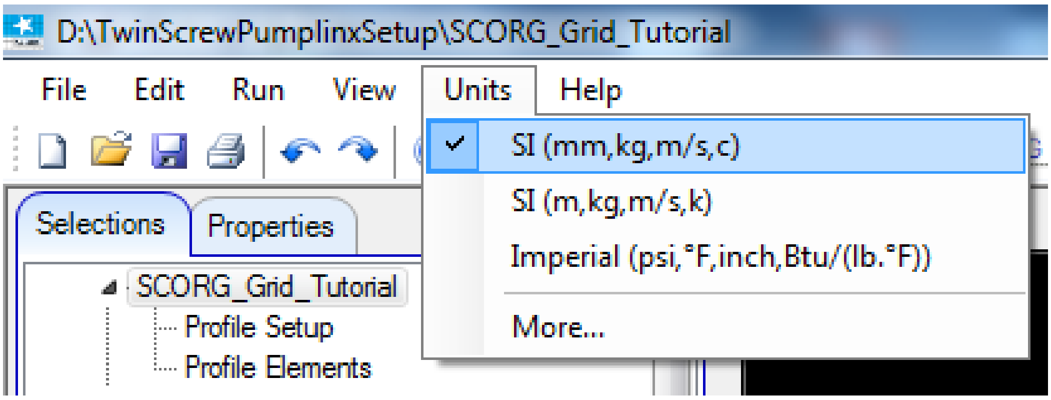

- Select RootsGrid.spf



- Inspect the Rotor Profile in the GUI for gaps in the tips, starting points of the profile indicated by the small yellow circles. Below is the required orientation.




- Set Project Units to SI




- Set the following Profile Parameters to get desired clearance size:

GAPI = 0.17mm

GAPR = 0.1mm

GAPA = 0.15mm

- Save the Project.

# 3 SCORG™ Mesh Generation

SCORG™ is stand-alone numerical CAD-CFD interface used to generate a numerical mesh of rotating parts of a machine and to transfer it to a general finite volume numerical solver. The program generates a block structured hexahedral numerical grid for rotor flow domains, solid rotor domains, inlet and outlet ports.

***Inputs Required***

In this step the rotor domain mesh is generated in SCORG™. The inputs required for this mesh generation are: (SCORG, 2021)

*Control Parameters:*

- Type of the machine.
- Number of mesh divisions along the lobe in circumferential direction.
- Number of mesh divisions in radial direction.
- Number of Angular divisions of the rotation.

*Control Switches*:

These Inputs are used to specify the method used for Rotor Profile Input and the required mesh calculation options.

- Click Grid Module in the project tree
- In Mesh Type Size set:
  - Circumferential Main = 0
  - Circumferential Gate = 150
  - Radial = 10
  - Angular = 180
  - Axial Divisions = 25
  - Interlobe Divisions = 60

| Rotor Mesh Size | |
|---|---|
| Circumferential Divisions Main Rotor | 0 |
| Circumferential Divisions Gate Rotor | 150 |
| Radial Divisions | 10 |
| Angular Divisions | 180 |
| Axial Divisions | 25 |
| Interlobe Divisions | 60 |

- Distribution Parameters:

These inputs are used for adaptation and distribution of the grid points on the boundaries of the domain and for smoothing of rack (Rack is the curve representing a rotor with infinite radius which uniquely separates the flow domains of the male and female rotors).

- Type of Distribution → Casing to Rotor Conformal (Rane, 2015)

| Distribution Parameters | |
|---|---|
| Type of Distribution | Casing to Rotor Conformal |
| K Main | 0.5 |
| K Gate | 1 |
| Rack Smoothing Factor | 1 |
| Project on Main profile | Yes |

- Meshing Parameters:

Meshing parameters provide control over the distribution of the internal mesh points in each cross section of the rotors.

| Meshing Parameters | |
|---|---|
| Mesh Orthogonality and Sm... | |
| Relaxation Factor ( 0 - 1 ) | 1 |
| Tolerance Factor ( 1 - 100 ) | 100 |
| Inflation Layer Control | |
| Radial Bias Factor ( 0 - 1 ) | 0.5 |
| Radial Bias Intensity ( 1 - 10 ) | 1 |

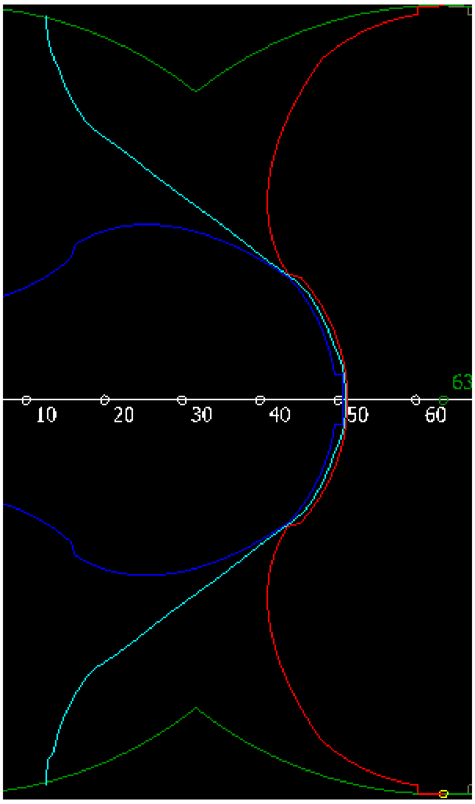
  - both the distribution and meshing parameters can be changed later

- Start Grid Generation through a three step process as below.
- Select Rack Refinement Points = 1000

| Control Switches | |
|---|---|
| Rack Generation | Off |
| Rack Refinement Points | 1000 |
| Boundary Generation | Off |
| Fluid Rotor Grid | Off |

- Click Numerical Rack Generation


This operation produces the rack curve between the two profiles. It is required to be executed only once in the grid generation process.



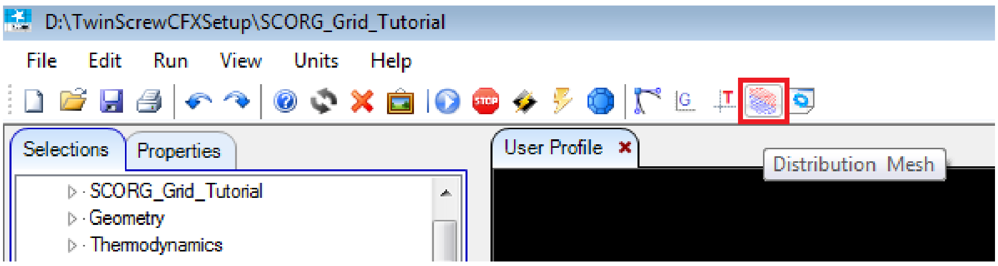

▶ Click Boundary Distribution Generation

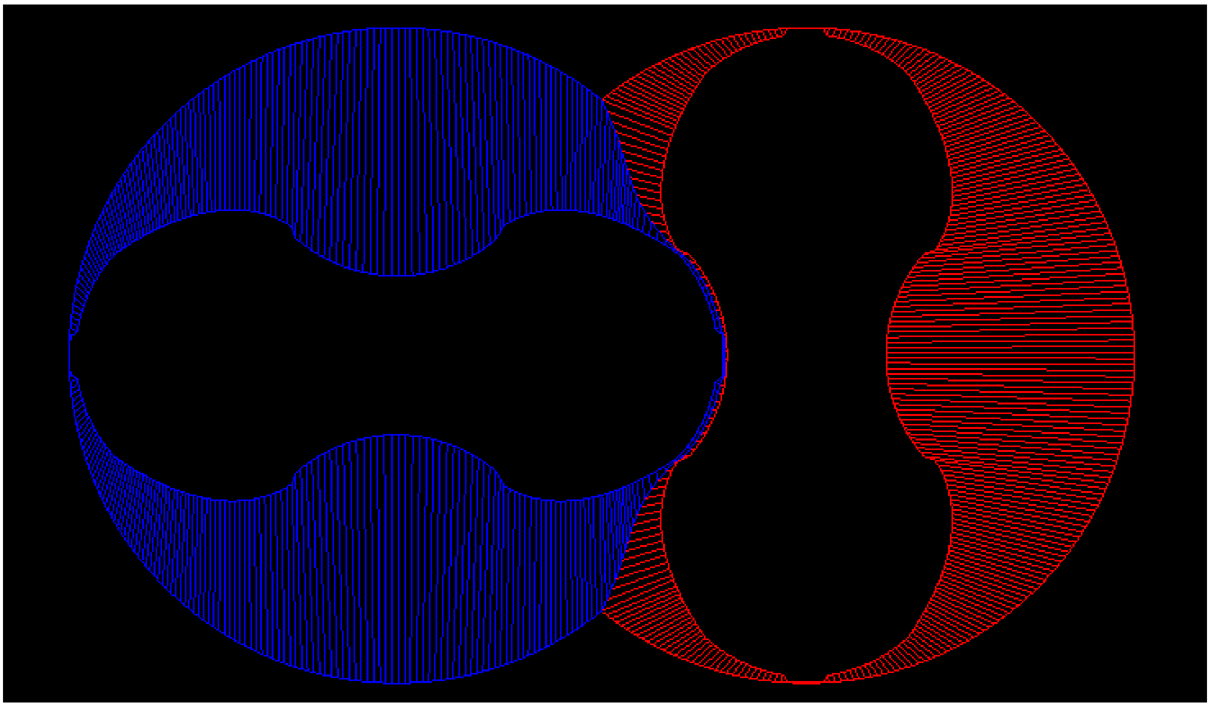

Information about the progress of the selected activities in the meshing procedure is displayed in the output window. Any warning or error and their locations are indicated. If errors occur, it is important to manually tune the input parameters which will produce a mesh without errors. Graphically the mesh distribution in each section can be visualized and checked for any deviation from requirements. The detailed description of methods used for distribution, adaptation and generation of numerical mesh is available through the Help in the drop down menu.

▶ Inspect report and check that there are no distribution warnings listed

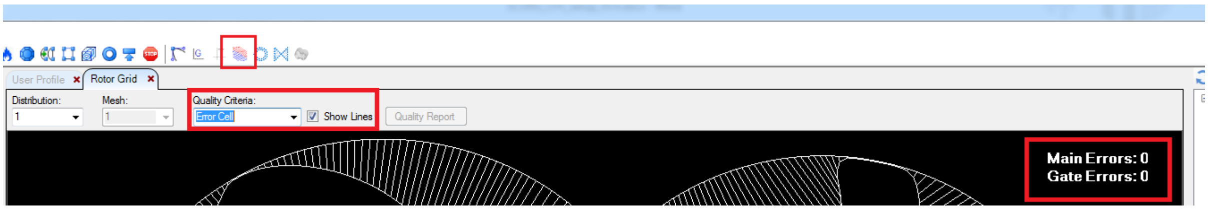
```
.......................................................................
 SCORG - Screw Compressor Rotor Geometry grid  generator V.5.7
............................................................12/11/2019...
Screw compressor/p Wrap =    0.0   RPM=12824.    Vel=  68.0   Ncel= 150000
 Z1/Z2= 2/2  d1=101.40 [mm]  d2=101.40 [mm]  a= 63.50 [mm]  len= 50.50 [mm]
.......................................................................
 Nfi   Nr   Nz Nadd  Rot Rack Boun Mesh RotM InpP OutP Prep RaSm Line  Oil
 150   10   25  180    3    1    1    0    0    0    0    0    1    0    0
.......................................................................
Calculation:  ROTOR       1:  0.00 Dist  0.00 Cos   2: 0.00 Ang.  0.00 Sin
Calculation:  RACK        Smoothing factor:  1.00   Smooth: ON
Calculation:  BOUNDARY    Male =  300               Female =  300
Initial Smoothing Distribution:Casing to Rotor Conformal
 TFI_Mesh routine - Rotor          1
 TFI_Mesh routine - Rotor          2
Initial Smoothing GRID  RelaxFac, TolFac, RadBFac, RadBInt, InterlobeBInt
                             1.0     100     0.5     1.0       2
 PDE_Interlobe_mesh routine
 Distribution Type: Casing to Rotor Conformal
.......................................................................
Distribution: Casing to Rotor Conformal
.......................................................................
Cell statistics        Overall number of cells        0
.Rotor fluid         0           .Inlet  port         0
.Rotor solid         0           .Outlet port         0
.......................................................................
 Start: 15:51:16    End: 15:52:43    Running time: 0h: 1m:26s =    86 sec
............................................................12/11/2019...
 SCORG - Screw Compressor Rotor Geometry grid generator - Ver. 5.7
```

- Click Distribution Mesh to visually inspect the distribution in each cross section





- In the Distribution Display → Select Quality Criteria = Error Cell

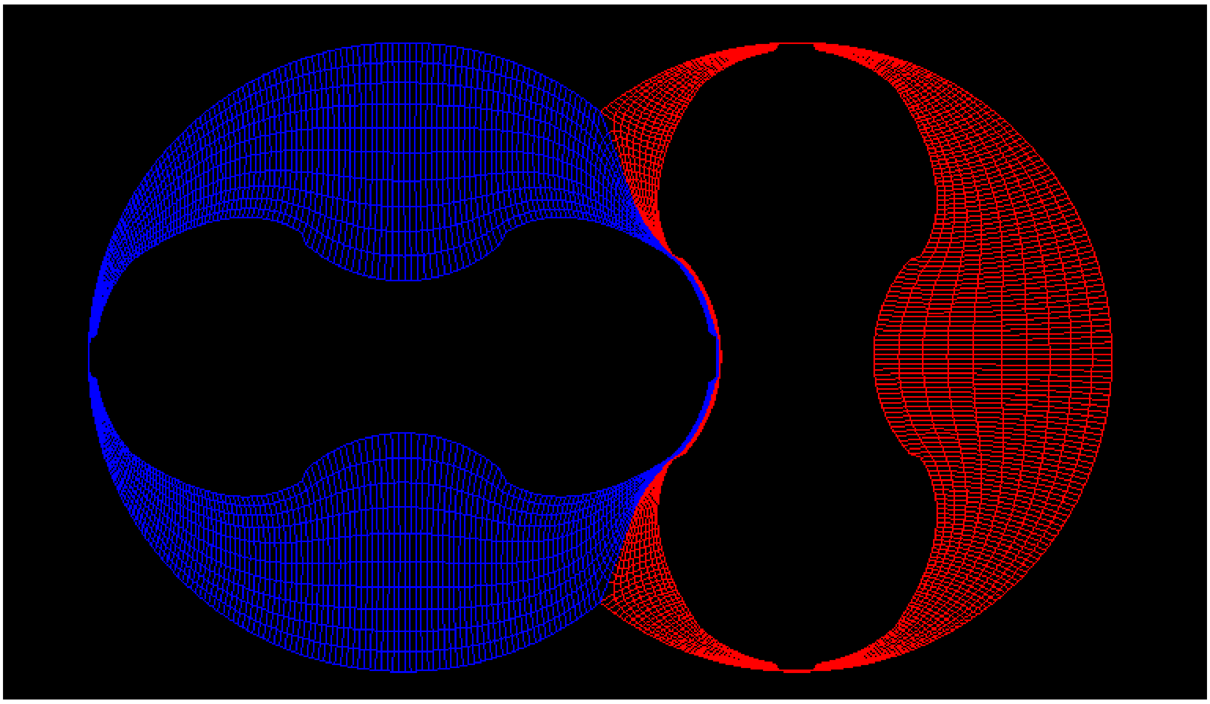



- Inspect all the distribution positions and ensure that 0 error are reported in each position.

- Click Rotor Grid Generation


- Inspect report and check that there are no grid errors listed

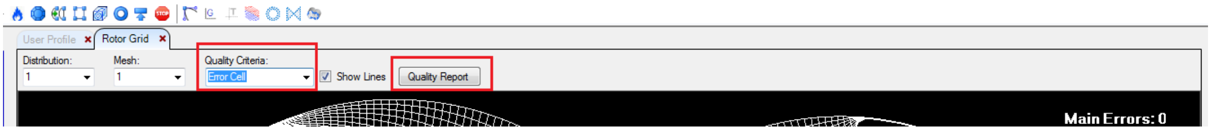
- Click Distribution Mesh to visually inspect the grid in each cross section



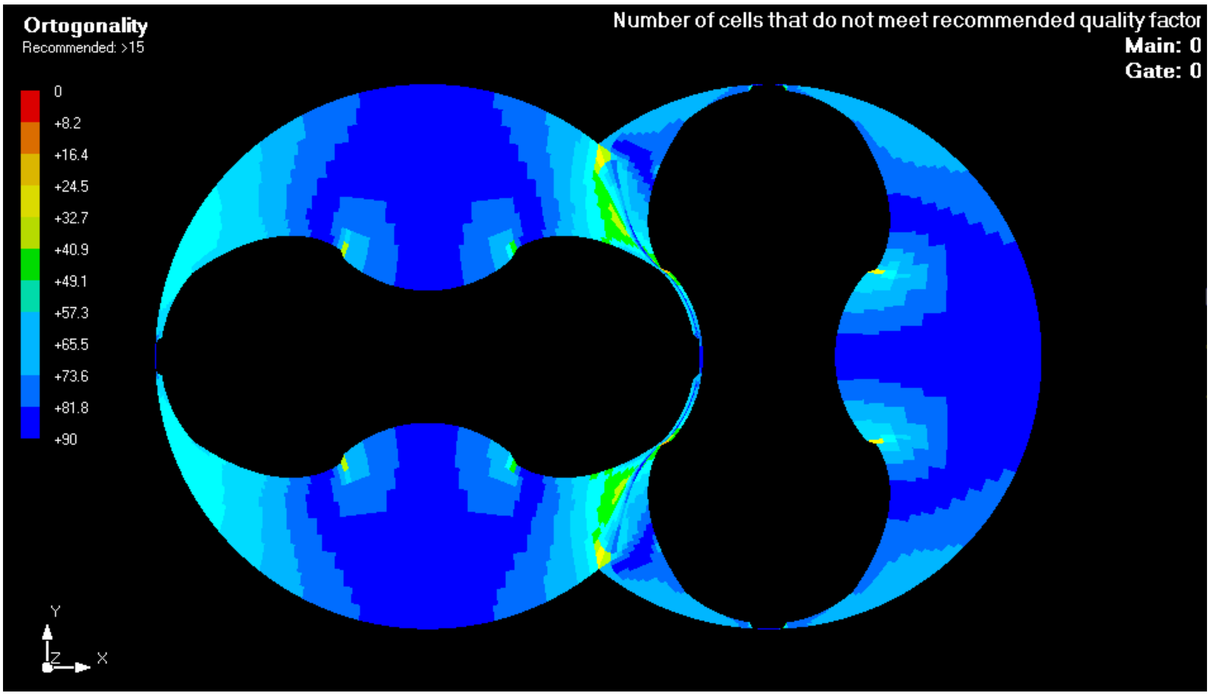
```
..........................................................................

 SCORG - Screw Compressor Rotor Geometry grid  generator V.5.7
..............................................................13/11/2019...
Screw compressor/p Wrap =    0.0   RPM=12824.    Vel=  68.0   Ncel= 150000
 Z1/Z2= 2/2  d1=101.40 [mm]  d2=101.40 [mm]  a= 63.50 [mm]  len= 50.50 [mm]
..........................................................................
 Nfi   Nr   Nz Nadd  Rot Rack Boun Mesh RotM InpP OutP Prep RaSm Line  Oil
 150   10   25  180    0    0    0    1    0    0    0    0    1    0    0
..........................................................................
Calculation:  FLUID GRID  RelaxFac, TolFac, RadBFac, RadBInt, InterlobeBInt
                              1.0     100     0.5     1.0       2
 TFI_Mesh routine - Rotor           1
 TFI_Mesh routine - Rotor           2
 PDE_mesh routine - Rotor           1
 PDE_mesh routine - Rotor           2
 PDE_Interlobe_mesh2 routine: Smooth Interlobe
..........................................................................
 Check_Grid - Rotor:            1
 Check_Grid - Rotor:            2
 Write 2D Grid Data

 Grid Data Count:
  Male rotor domain, Vertices:   85800, Cells   75000
Female rotor domain, Vertices:   85800, Cells   75000
Written Control.dat
..........................................................................
Cell statistics        Overall number of cells        0
.Rotor fluid         0            .Inlet  port        0
.Rotor solid         0            .Outlet port        0
..........................................................................
 Start: 11:56:43    End: 11:56:57    Running time: 0h: 0m:13s =    13 sec
..............................................................13/11/2019...
 SCORG - Screw Compressor Rotor Geometry grid generator - Ver. 5.7
```

- Click Rotor Grid 2D Mesh to visually inspect the grid in each cross section

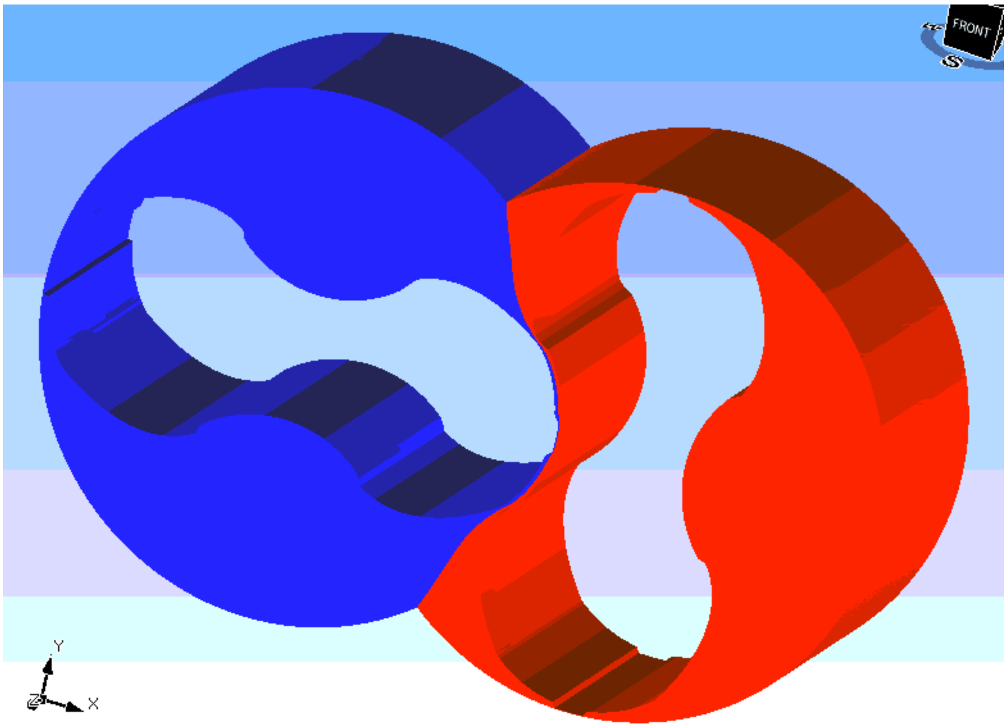
- Click Quality Critera → Error Cell and Inspect.




- Click Quality Critera → Orthogonality and Inspect.




- Inspect the 3D mesh



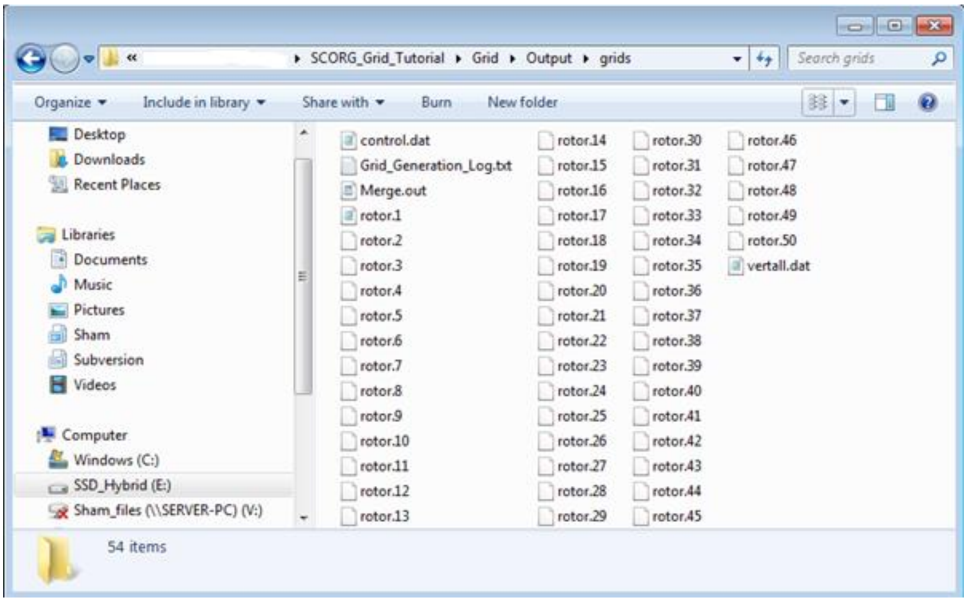
- In Control Switches → Preprocessor Input File select → OpenFOAM
- Set Vertex Files Start = 1
- Set Vertex Files End = 180 [ = Number of Angular Divisions for Casing to Rotor type of distribution]

| Control Switches | |
|---|---|
| Rack Generation | Off |
| Rack Refinement Points | 1000 |
| Boundary Generation | Off |
| Fluid Rotor Grid | Off |
| Solid Rotor Grid | Off |
| Inlet Port Grid | Off |
| Outlet Port Grid | Off |
| Preprocessor Input File | OpenFOAM |
| Vertex Files Start Number | 1 |
| Vertex Files End Number | 180 |

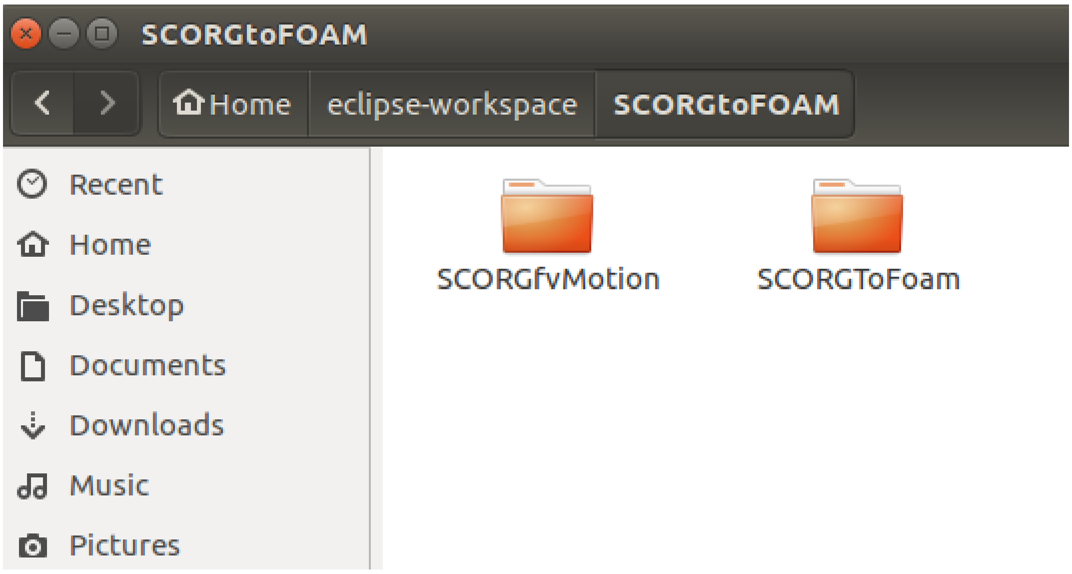
- Calculate Preprocessor Files Generation


```
..........................................................................
InstallPath = C:\SCORG
ProjectPath = C:\Users\OpenFOAM\RootsGrid
...........................................................................
 All control parameters for grid generation are disabled

...........................................................................

----Generation of Port Pre Processor files
----Generation of Rotor Pre Processor files
Checking volumes in Male
Min/Max Volume=  1.4394324E-11  8.4670049E-09
Checking volumes in Female
Min/Max Volume=  1.4281043E-11  9.1572163E-09
Generation of time step grid files
  Start time step:           1
  End   time step:           1

Rotor      1,  Grid position     1
Rotor      2,  Grid position     1

Grids written

CALL Single Rotor Domain Grids

Number of grids =           1
Single Rotor Domain Grids written

...........................................................................
Cell statistics        Overall number of cells          0
.Rotor fluid          0           .Inlet  port          0
.Rotor solid          0           .Outlet port          0
..........................................................................
 Start: 12:14:54    End: 12:15: 2    Running time: 0h: 0m: 8s =     8 sec
.................................................................. 5/ 5/2020...
 SCORG - Screw Compressor Rotor Geometry grid generator - Ver. 5.8
```

- With this the SCORG™ Project is complete and the OpenFOAM setup can be started.
- In the directory structure of SCORG™ Project → Grid → Output with consist of OpenFOAM and grids folder.
- The grids folder consists of all time step mesh files named as rotor.1, rotor.2 ,,,etc



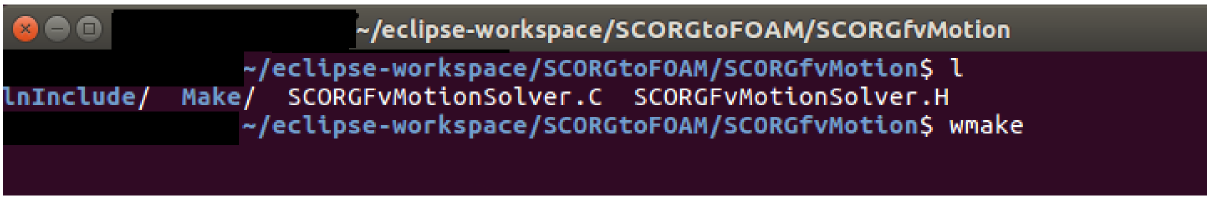

- The directory SCORG™ Project → Grid → Output→ OpenFOAM contains one sub-directory for every OpenFOAM version supported.
- Each of these directories contains two folders: rootsBlowerTutorial is the tutorial template directory, while SCORGtoFOAM contains the user libraries to be compiled in order to run the case.



# 4 Compile OpenFOAM user libraries [ *One time procedure,* (Casari & Fadiga, 2019) ]

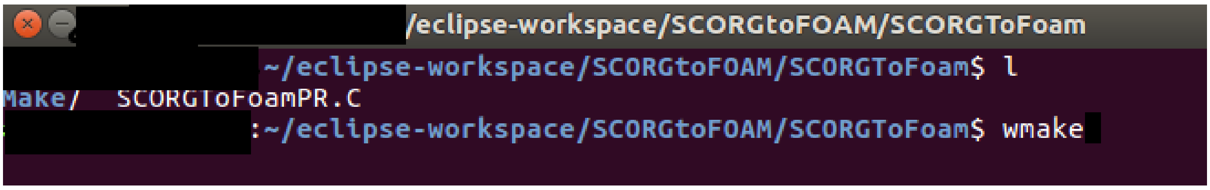
**Ubuntu 16.04 and 18.04**

- Move the outputs of SCORG™ to the computer in which OpenFOAM is compiled.
- Open the terminal, source the OpenFOAM environment and move to the SCORGToFOAM directory.
- Move to the SCORGfvMOTION directory




- Exectute the command **wmake** to compile the user library that handles the SCORG dynamic mesh process.

- Move to the SCORGToFoam directory




- Exectute the command **wmake** to compile the user library that handles the SCORG mesh conversion process.

# 5 OpenFOAM case setup (Casari & Fadiga, 2019)

- Copy the content of the rootsBlowerTutorial folder into the working directory

- Copy the grids folder into the working directory (in case of conformal distribution, only the rotor.# files and the roa.1 and rog.1 files are needed)

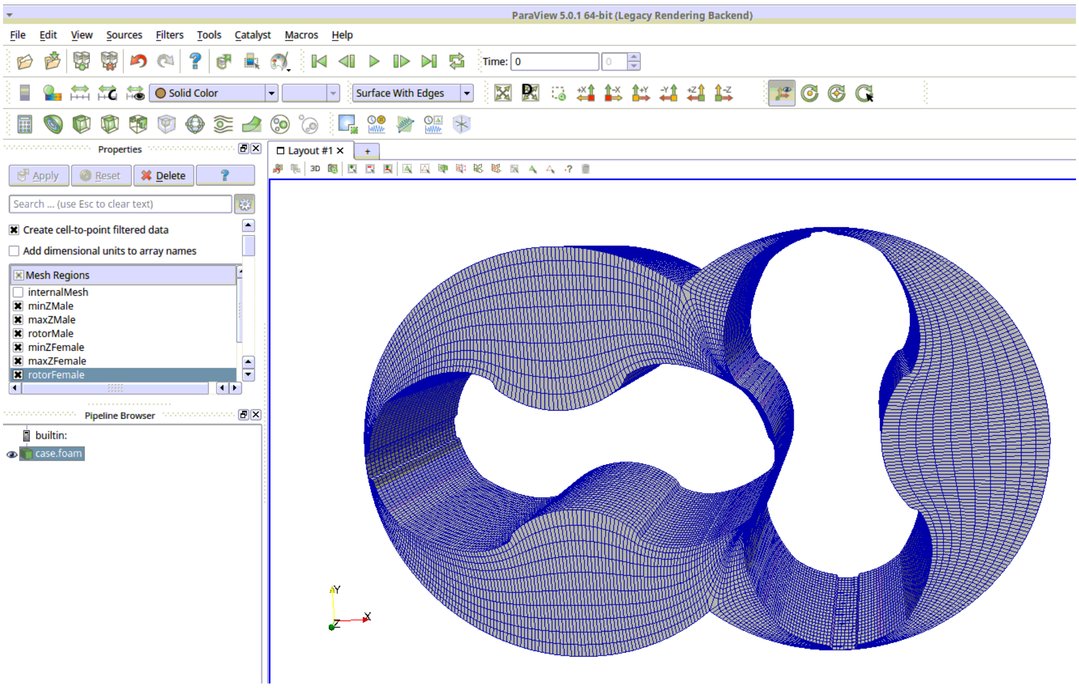
- Open the file **workingDirectory/system/SCORGdict** with a text editor

```
FoamFile
{
    version     2.0;
    format      ascii;
    class       dictionary;
    object      SCORGDict;
}
// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * //

conformalInterface true;
nonConformalInterface false;
rotorToCasing false;
singleRotor false;
lowPressurePort false;
highPressurePort false;
prisms  false;

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

- Make sure that the required flag (conformalInterface) is set to true. In order to convert different meshes, the user must correctly fill the SCORGdict dictionary.
- If the user wants to convert also the port meshes, the SCORG output files for the ports must be included in the grids directory as well.

- Open the terminal in the case main directory (the working directory), source the OpenFOAM environment and run the command "**SCORGToFoam**"

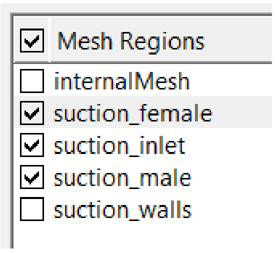
- Check the mesh in Paraview in order to visualize the result, loading the file case.foam contained in the case main directory:

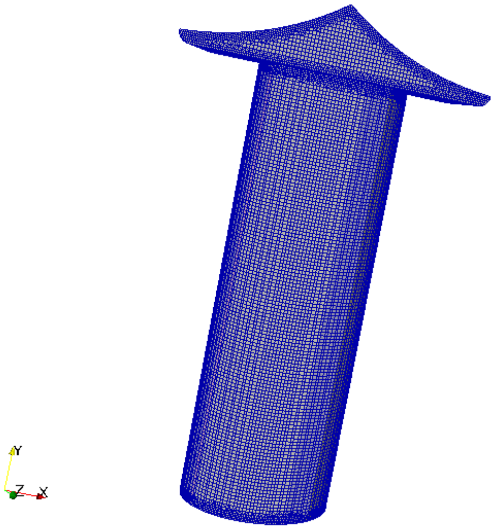


- Check the mesh quality running "**checkMesh"** in the terminal from the case main directory.
- Move to the **workingDirectory/suction** directory to generate the suction port mesh.
- The mesh will be generated with the **snappyHexMesh** utility, starting from the STL files into the **workingDirectory/suction/constant/triSurface** folder.
- Open the terminal and run "**surfaceCheck constant/trisurface/suction.stl"**, checking if the surface is closed.
- Run "**blockMesh"** to generate a background mesh for the snappy procedure.
- Run "**surfaceFeatureExtract"** to extract the edges of the surface that must be meshed
- Check and update the number of processors in the **workingDirectory/suction/system/decomposeParDict** file
- Run "**decomposePar"** to prepare for the parallel generation of the mesh (ignore this line if a serial run is preferred)
- Run "**snappyHexMesh"** for serial mesh generation, or "**mpirun -np 4 snappyHexMesh -parallel"** for parallel generation in **workingDirectory/suction**
- Run "**reconstructParMesh -latestTime -mergeTol 1e-06"** to reconstruct the generated mesh.
- Run "**rm -r constant/polyMesh"**, then "**mv 2/polyMesh constant/**" and "**rm -r 2**" to move the new mesh in the constant directory.
- Open the **workingDirectory/suction/constant/polyMesh/boundary** text file and change the suction_inlet type from **wall** to **patch**

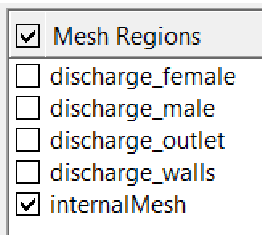
```
suction_inlet
 {
     type            patch;
     nFaces          2558;
     startFace       466583;
 }
```

- Visualize the mesh in paraview and check it with the terminal command "**checkMesh**".

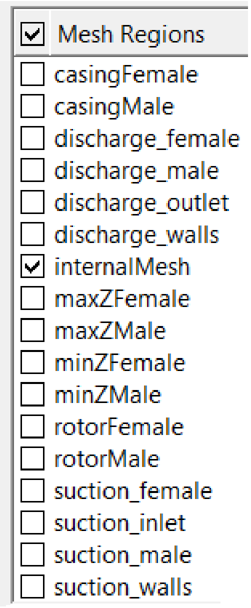





- Move to the **workingDirectory/discharge** directory to generate the discharge port mesh, following the same procedure.
- In this case, when "**surfaceCheck"** is run, be sure of checking constant/triSurface/**discharge.stl** file.
- For the discharge mesh, the discharge_outlet patch in **workingDirectory/discharge/constant/polyMesh/boundary** has to be changed into **patch type**

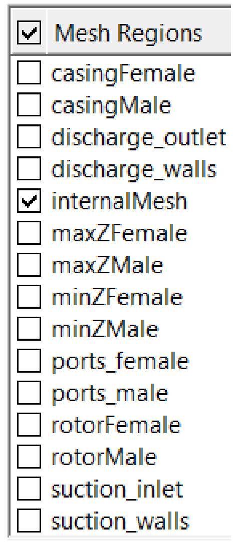



- Move back to the **workingDirectory**
- Run "**mergeMeshes . discharge/**" to merge the main mesh with the discharge port mesh
- Run "**mergeMeshes . suction/**" to merge the main mesh with the discharge port mesh
- Run "**cp -r 0.00071839/polyMesh constant**" and "**rm -r 0.000***" to copy the new mesh into the constant folder and remove the time directories created by mergeMeshes




- Run "**createPatch**" to join the ports patches togheter

- Run "**cp -r 0.000359*/polyMesh constant**" and "**rm -r 0.000***" to copy the new mesh into the constant folder and remove the time directories created by createPatch.



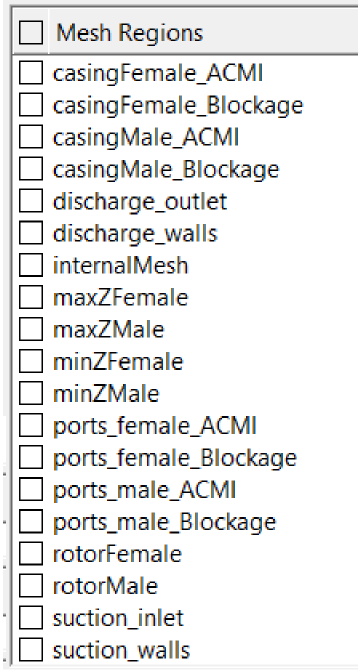

- Run "**topoSet -dict system/topoSetDict.ports"** to create faceZones starting from the new patches.

- Edit the text file **workingDirectory/constant/polyMesh/boundary** and change the type of the following patches **from wall to symmetry**: minZMale, maxZMale, minZFemale, maxZFemale.

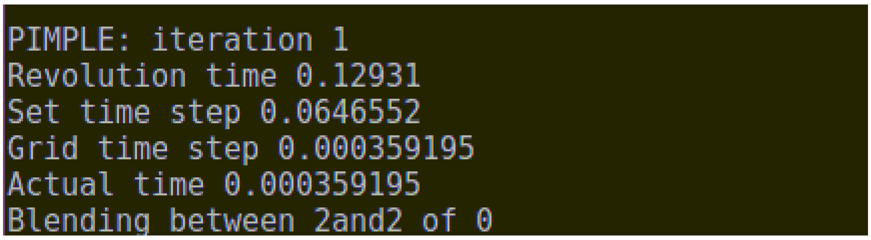
- Run "**createBaffles**" to create ACMI interfaces.

- <u>**N.B.**</u> In order to run simulations with the non-conformal distribution the user has to create an AMI interface between the two rotors, using the patches interlobeMale and interlobeFemale created by the SCORG mesh converter. This feature could introduce some conservation error and it is still under investigation. More information about AMI interfaces are available in the **createPatchDict.AMI** dictionary in the **workingDirectory/system** folder.

- Run "**cp -r 0.000359*/polyMesh constant**" and "**rm -r 0.000***" to copy the new mesh into the constant folder and remove the time directories created by createBaffles.




- Run "**topoSet -dict system/topoSetDict.pZones**" to create faceSets
- Run "**decomposePar**" for the parallel execution of the numerical calculation.
- It is possible to change and set thermophysical properties, turbulence properties, numerical schemes and time settings from the dictionaries in **constant and system** directories.
- In order to use exactly the grids generated by SCORG, the deltaT in **system/controlDict** dictionary must be consistent with the angular velocity and the number of grids set up in **constant/dynamicMeshDict.** Otherwise, the software will perform a linear interpolation between grids, generating intermediate positions. The grid used for every time step is reported in the log file as follow:
  "*Blending between 2and2 of 0*" indicates that the software is using the grid stored in rotor.2 file




  "*Blending between 2and3 of 0.4563*" indicates that the software is interpolating the nodal positions between grid 2 and grid 3

The interpolation feature must be treated carefully: the interpolation could produce low quality mesh elements.

- Run "**foamJob -p rhoPimpleFoam"** to start the simulation. The log file will be automatically saved in the working directory.

- Use **pyFoamPlotWatcher** or **foamLog** to check the residuals during the simulation. pyFoam must be installed by the user, while foamLog is already compiled in OpenFOAM standard versions.

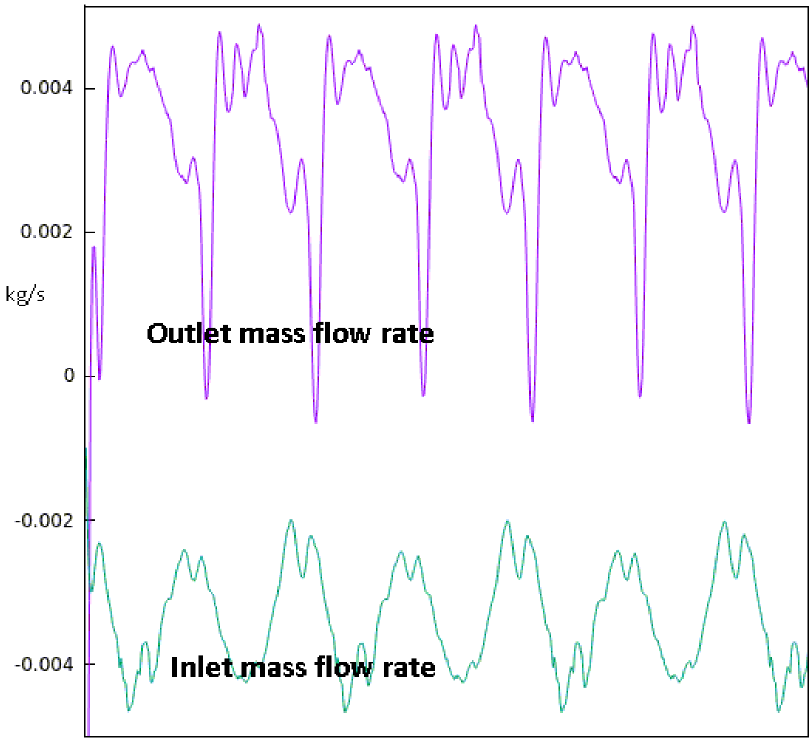
  https://openfoamwiki.net/index.php/Contrib/PyFoam -- pyFoam

  https://www.cfdsupport.com/OpenFOAM-Training-by-CFD-Support/node88.html - foamLog

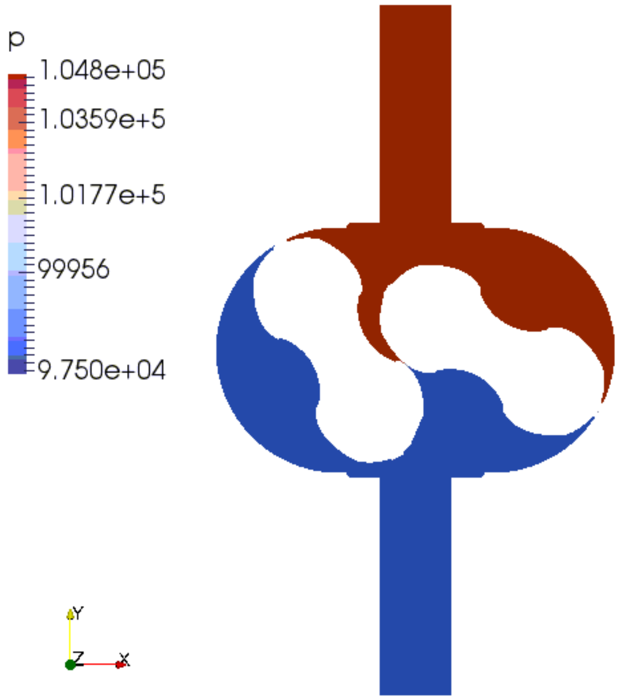
- It is possible to add monitor points in different positions of the rotor and create plots for Pressure on these points. The trace of the chamber pressure rise can be seen.

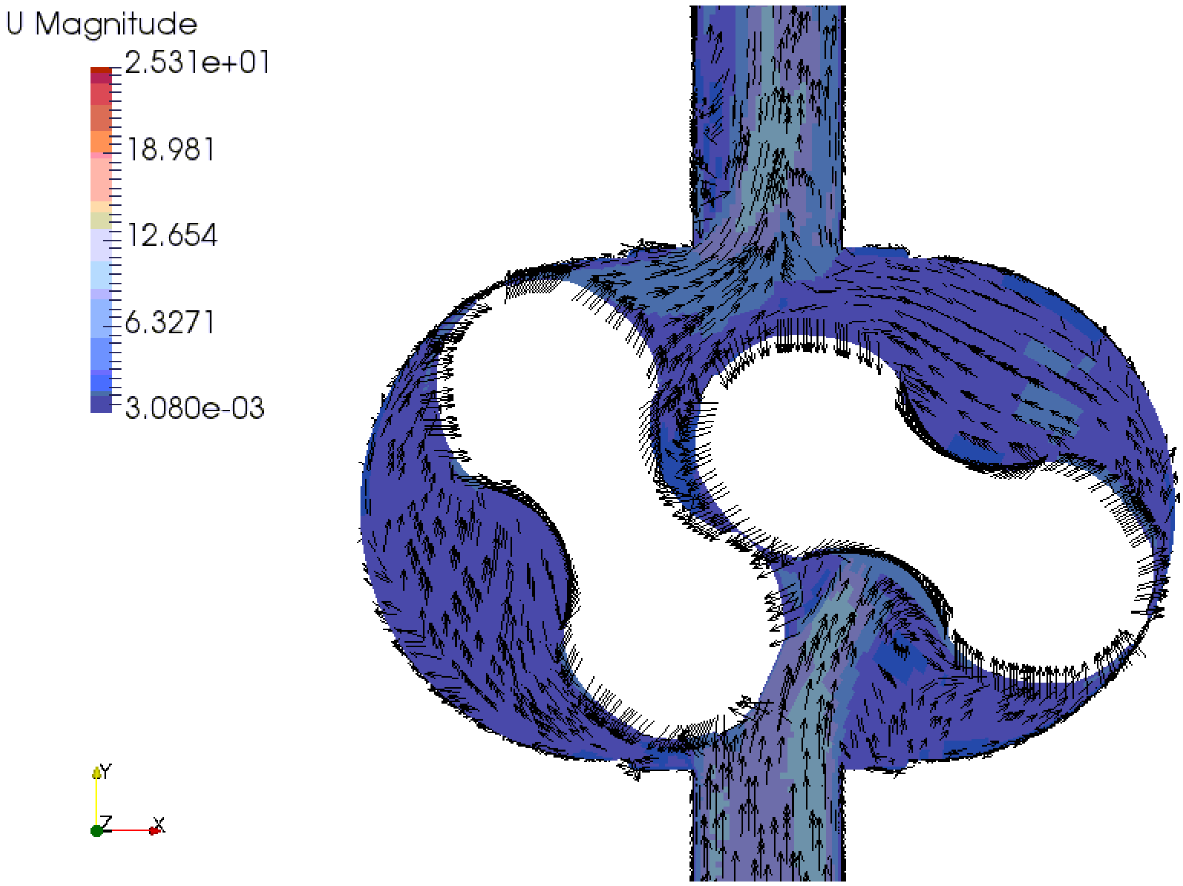
  (To be completed)

- Different surface reports for the mass flow rate through the In and Out boundaries are already set up in the **system** directory. It is possible to plot the results in the **workingDirectory/postprocessing** folder using **gnuplot**.



- It is possible to create contour plots, glyphs representations and many more post processing features in **Paraview**.



# 6 Summary

This document describes the steps to setup an OpenFOAM model for Roots Blower CFD analysis using grids generated by SCORG$^{TM}$ Meshing tool. More detailed information on using SCORG and Screw compressor mesh generation can be found in user guide (SCORG, 2021). The set of mesh files generated for a complete cycle are reused cyclically when the simulation is run for more than one cycle. Thus, it is possible to continuously run the simulation until repeatable results in the monitors and good convergence is obtained.

# 7 Bibliography

Casari, N. & Fadiga, E., 2019. *CFD Simulations of Single- and Twin-Screw Machines with OpenFOAM.* Italy, MDPI - designs.

DISCO, 2007. *DISCO, User Help Manual*, London: City University.

Kovačević, A. & Rane, S., 2017. Algebraic generation of single domain computational grid for twin screw machines Part II – Validation. *Advances in Engineering Software,* Volume 107.

Kovacevic, A., Stosic, N. & Smith, I. K., 2007. *Screw compressors - Three dimensional computational fluid dynamics and solid fluid interaction, ISBN 3-540-36302-5.* 1 ed. New York: Springer-Verlag Berlin Heidelberg.

Rane, S., 2015. *Grid Generation and CFD analysis of Variable Geometry Screw Machines,* London: City University London.

Rane, S. & Kovačević, A., 2017. Algebraic generation of single domain computational grid for twin screw machines. Part I. Implementation. *Advances in Engineering Software*, Volume 107, pp. 38-50.

Rane, S., Kovačević, A. & Stošić, N., 2016. *CFD Analysis of Oil Flooded Twin Screw Compressors. Paper 2392.*. Purdue, Int. Compressor Eng. Conference.

SCORG, 2021. *SCORG, User Help Manual,* London: City University.

Stosic, N., Smith, I. K. & Kovacevic, A., 2005. *Screw compressors: Mathematical modeling and performance calculation, ISBN 3540242759.* 1 ed. London: Springer.

End of Document

PDM Analysis Ltd
Bourne House, 475 GodstoneRoad, Whyteleafe, Surrey, CR3 0BL, United Kingdom
+44 20 7040 8780; +44 78 2781 8689
SCORG@PDMAnalysis.co.uk http://www.pdmanalysis.co.uk