SCORG™ Setup for CFD Simulation of Twin Screw Machines with ANSYS FLUENT®

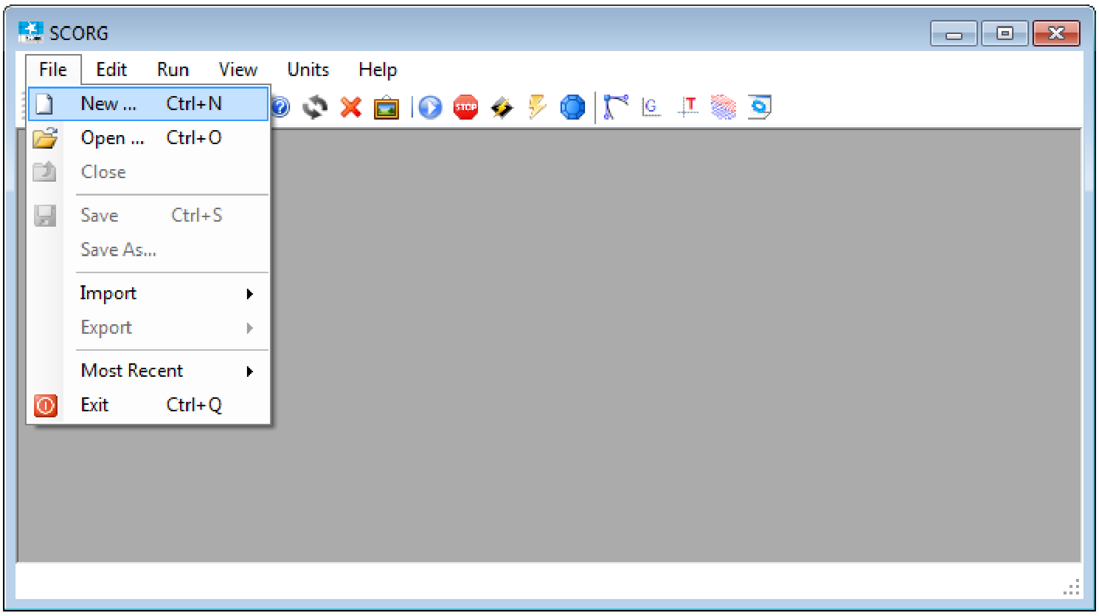
SCORG™ is the CFD grid generation tool for rotary twin screw machines. The tool includes additional modules for designing and editing rotor profiles, executing a basic thermodynamic calculation based on quasi 1D chamber models and generating the deforming working chamber grids for selected commercial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

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

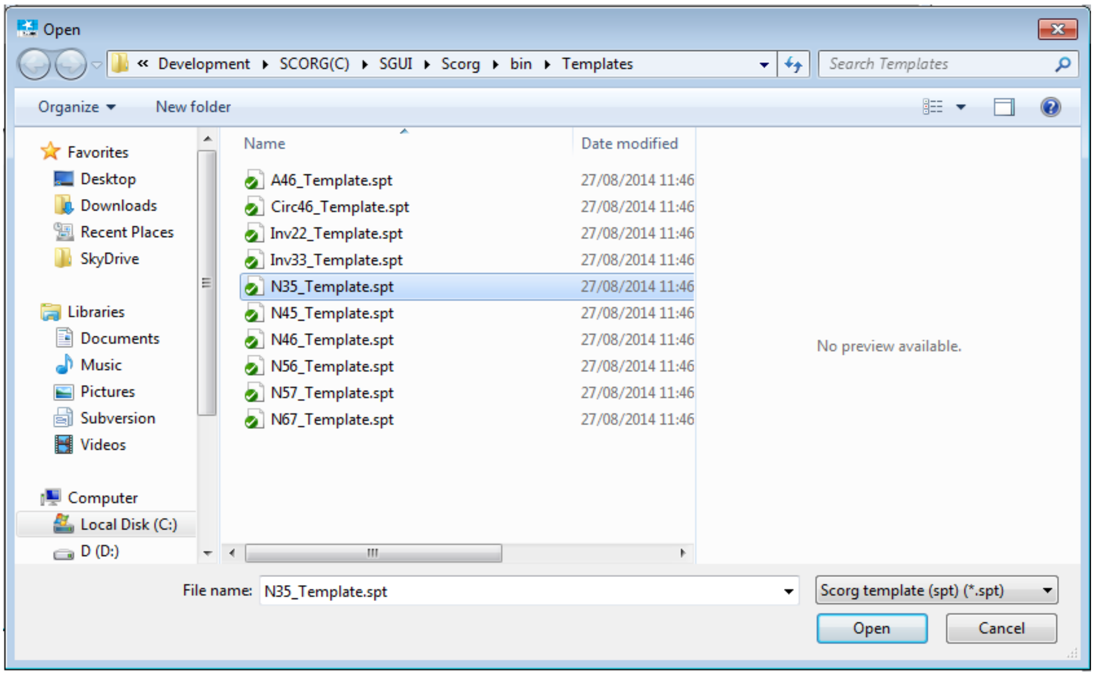
This guide lists the steps for setting up a CFD simulation for Twin Screw Compressor with SCORG™ and ANSYS FLUENT Solver. The user is expected to be familiar with screw machines, CFD and ANSYS FLUENT® in order to be able to use these procedures.

1 SCORG™ Project

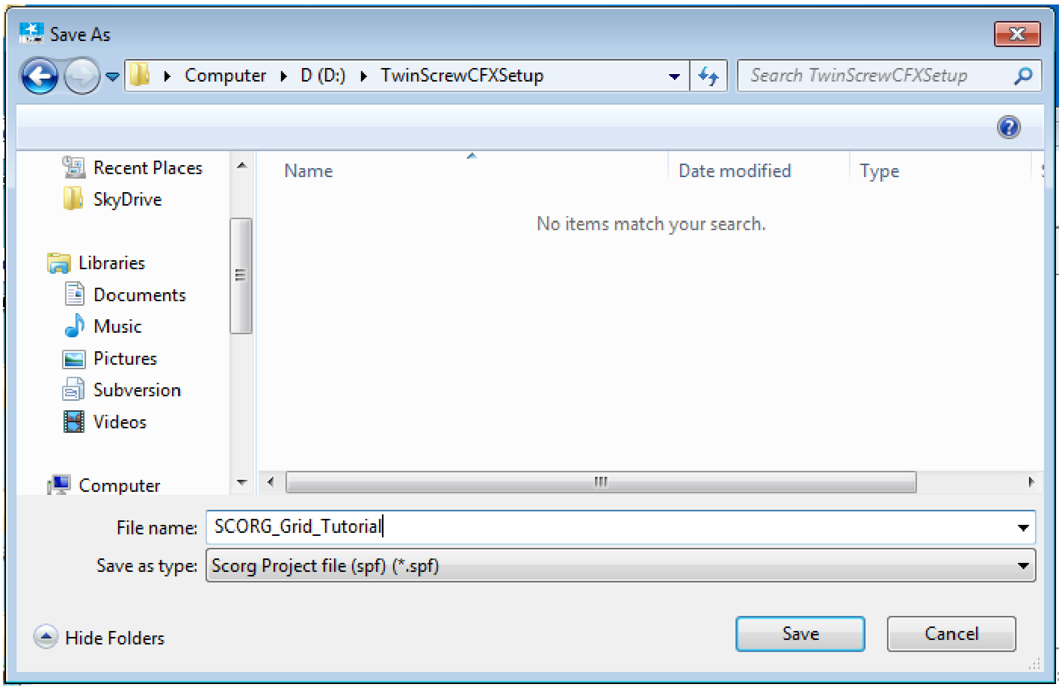
- Launch SCORG™ on the Desktop.
- Select File → New



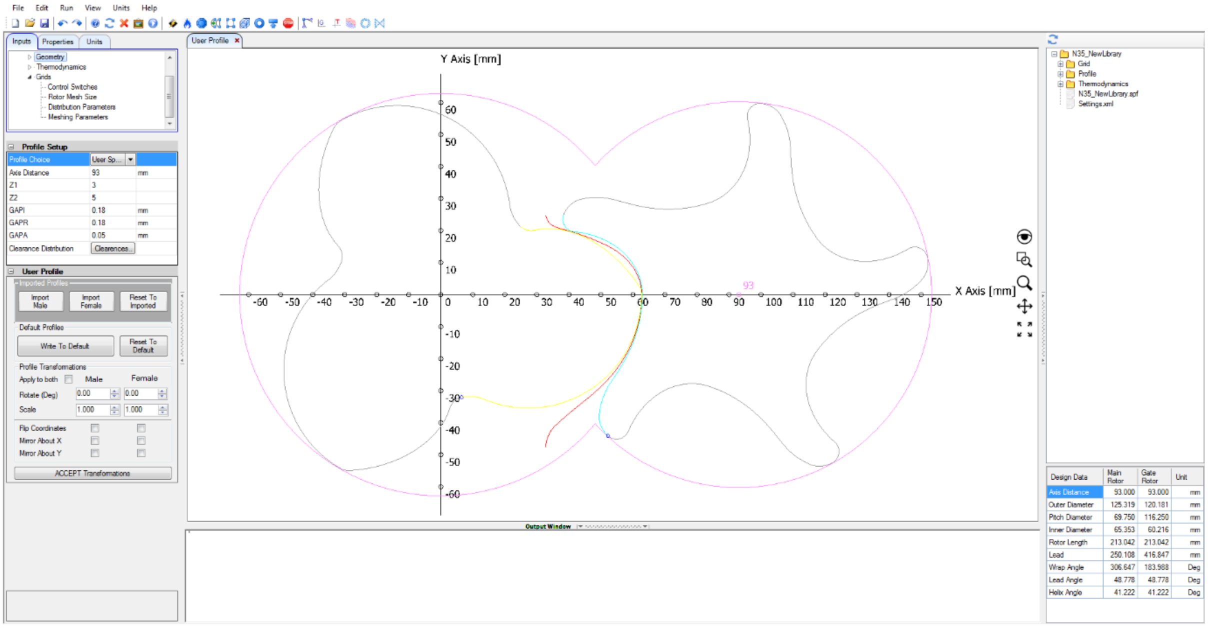
- Select N35_Template.spt → Open



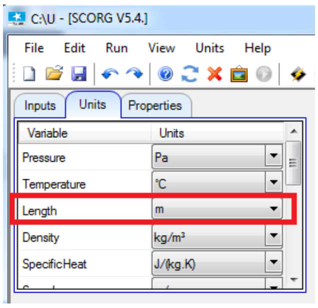
- Save the project in a new folder named TwinScrewFLUENTSetup → SCORG_Grid_Tutorial.spf



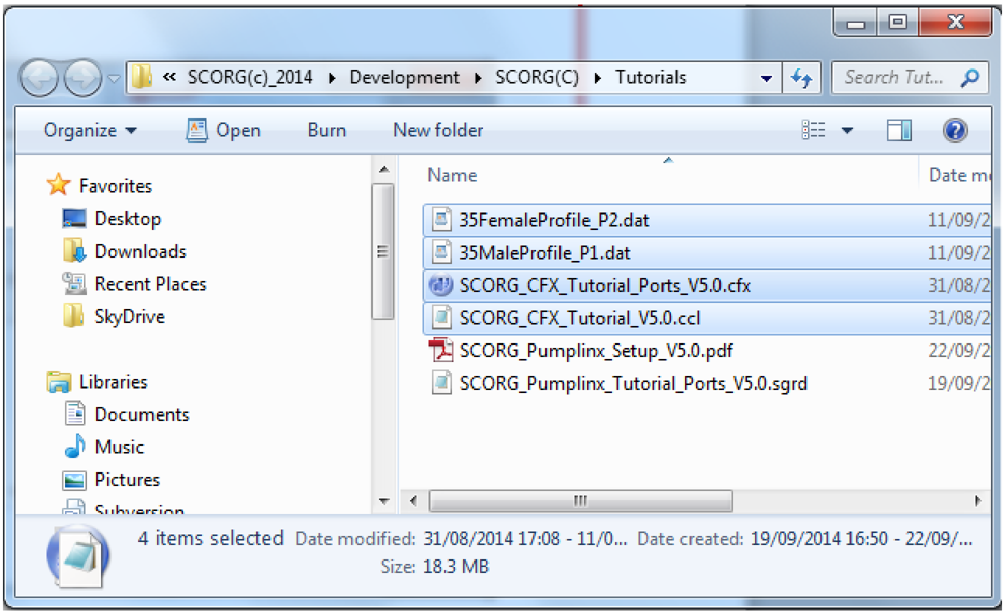
- The GUI of SCORG™ in the figure bellow shows the mains items of the front panel.



- In Units Tab, Select Length units as 'm'. This selection has to be the same as the units in which input profile coordinates are available.

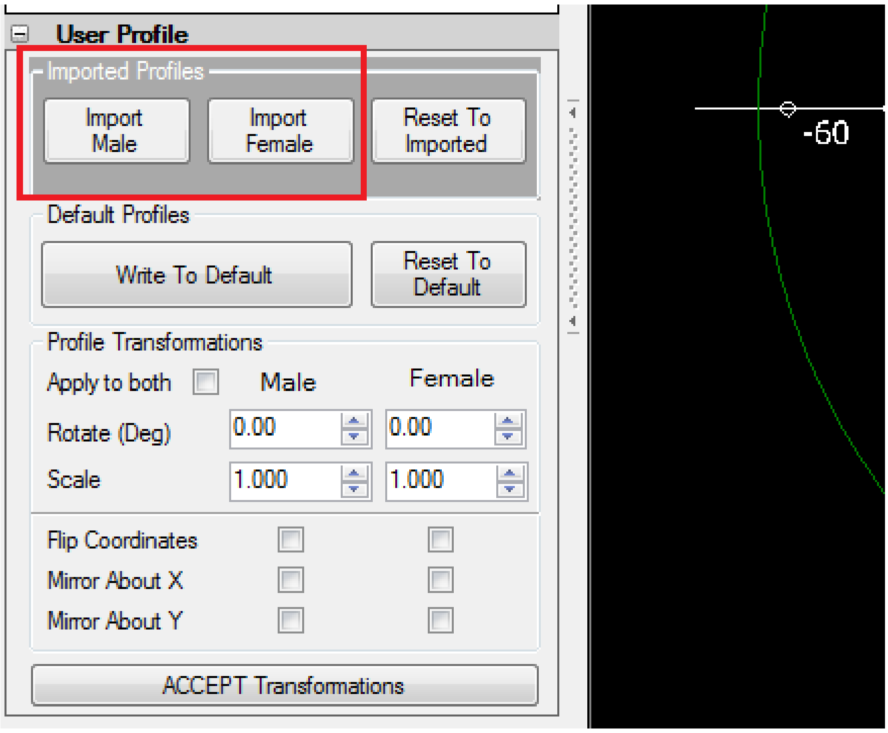


- Go to Help → Tutorials → Folder opens

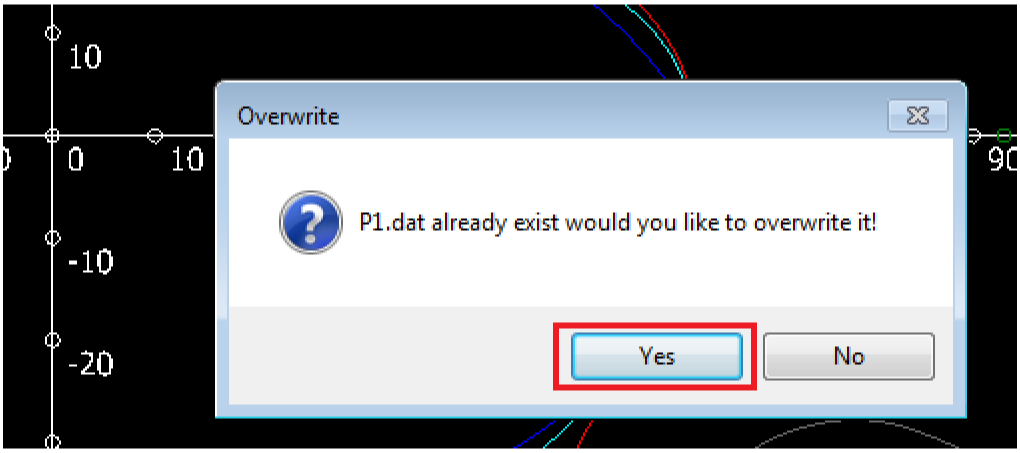


- Copy the compressor rotor profile files → [*35MaleProfile_P1.dat* and *35FemaleProfile_P2.dat*]
- Go to User Profile → Browse and Select the Male Rotor Profile from working directory.

35MaleProfile_P1.dat



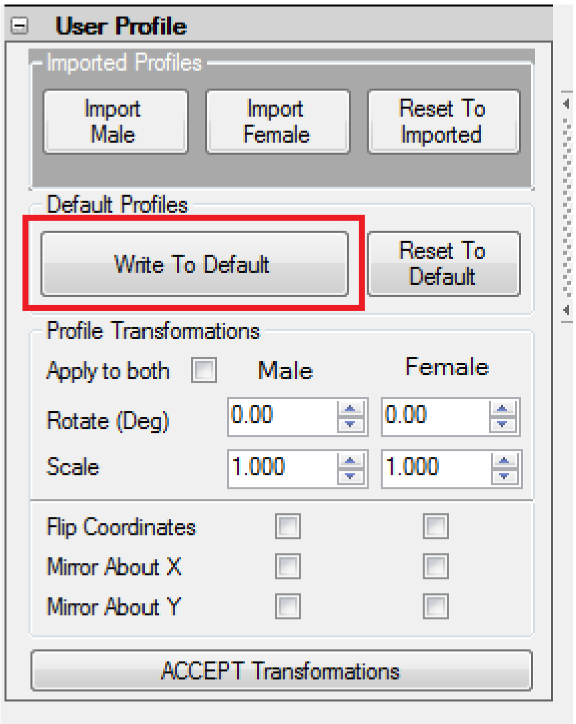
- Click 'Yes' to overwrite P1.dat.



- Similarly Select the Female Rotor Profile.

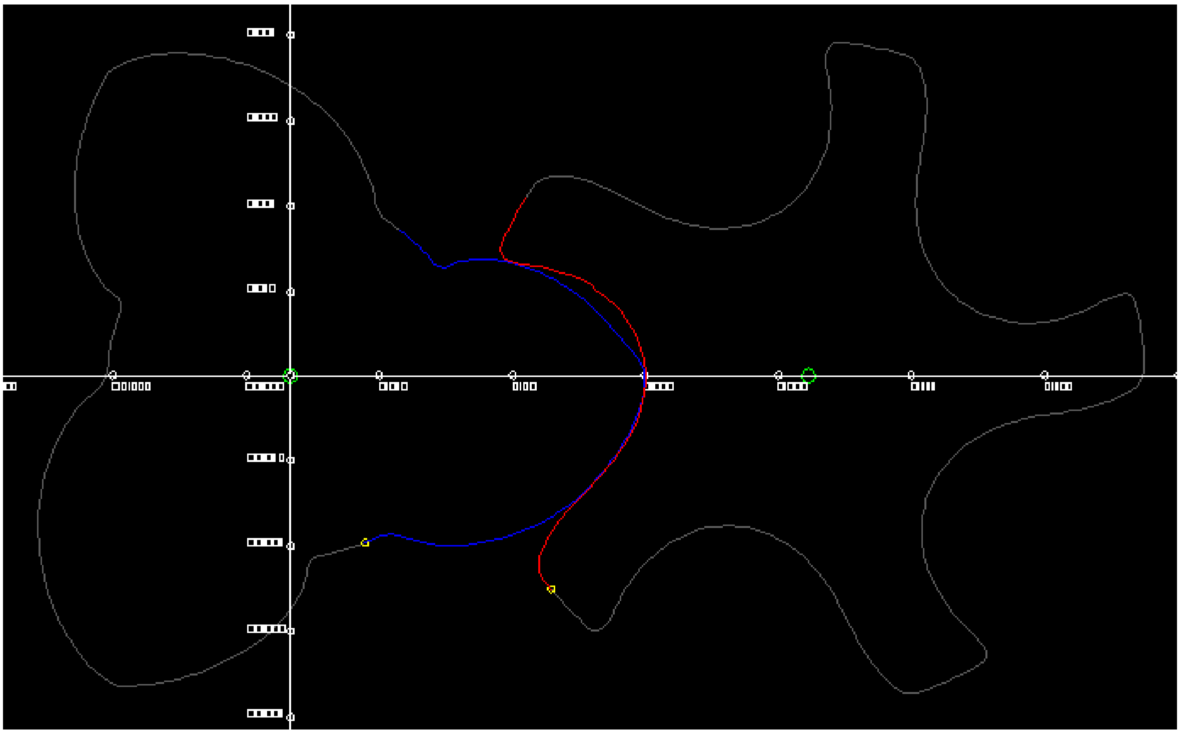
  35FemaleProfile_P2.dat

- Click Write To Default.

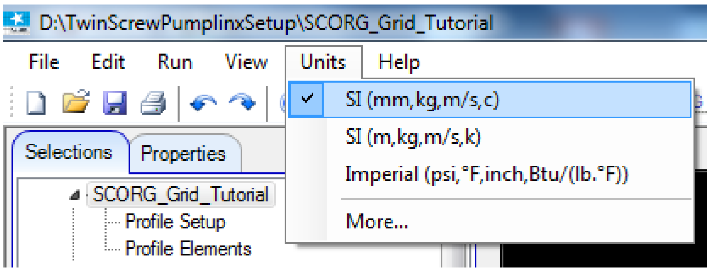


- Click Refresh to view new profiles.

- 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 = 0.06mm

GAPR = 0.06mm

GAPA = 0.05mm

*Setting GAPI = 0.06 sets the minimum interlobe clearance as the GAPI.

- Go to Geometry → Set the following parameters:

- Profile
  - Profile Setup
  - User Profile
- Geometry
- Thermodynamics
- Grids

Profile Setup

Profile Choice	User Sp...	
Axis Distance	93	mm
Z1	3	
Z2	5	
GAPI	0.06	mm
GAPR	0.06	mm
GAPA	0.05	mm
Clearance Distribution	Clearences..	

User Profile

Rotor Configuration

Relative Length	1.7	
Rotor Length	216.45	mm
Wrap Angle	285	Deg
Pitch Low Pressure Port	0	mm
Pitch High Pressure Port	0	mm

Machine Configuration

Machine Type	Compressor	
N Gate	1	
Compression Start	0	Deg
Compression End	161.001	Deg
Volume Index	1.8	
E Rotor	211	GPa
αL Rotor	1E-05	m/m/°C
E Casing	211	GPa
αL Casing	1E-05	m/m/°C
Wall Roughness	0	mm

- Go to Thermodynamics → Set the following parameters:

Working Conditions

Wtip	66.6665	m/s
Rotor Speed	10000	RPM
P0	100000	Pa
Pr	300000	Pa
T0	293	K
Tr	350	K
Tevp	268	K
Tcond	313	K
T Ambient	293	K
Ts	0	K
X	1	

- Save the Project.

2 SCORG™ Mesh Generation

SCORG™ is stand-alone numerical CAD-CFD interface used to generate a numerical mesh of rotating parts of a screw 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: (Kovacevic, et al., 2007).

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:

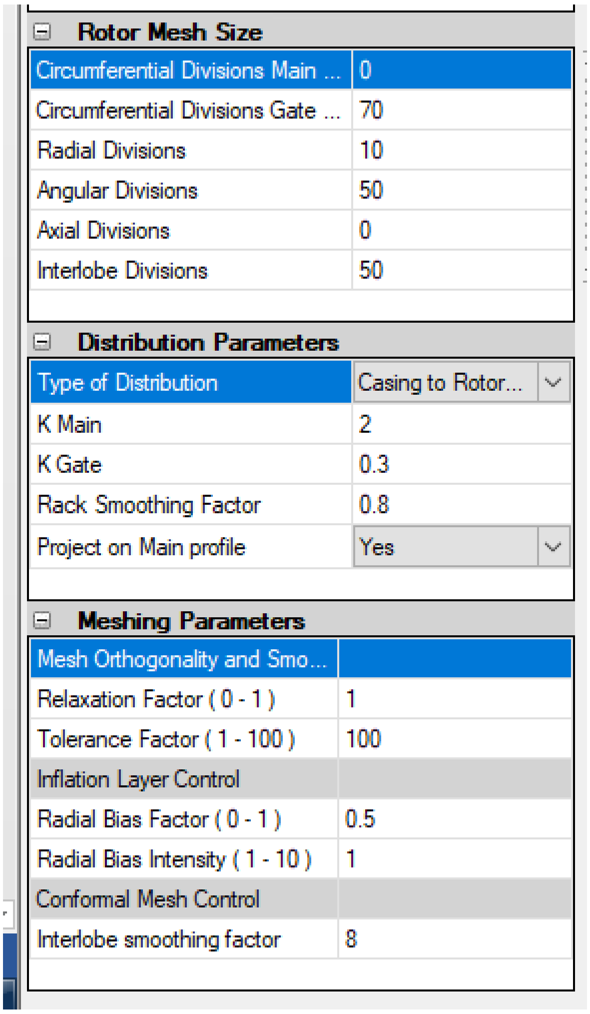
These Inputs are used to specify the method used for Rotor Profile Input and the required mesh calculation options.

- Click Grid Module in the project tree
- In Mesh Type Size set:
  - Circumferential Main = 0
  - Circumferential Gate = 70
  - Radial = 10
  - Angular = 50

- Distribution Parameters:

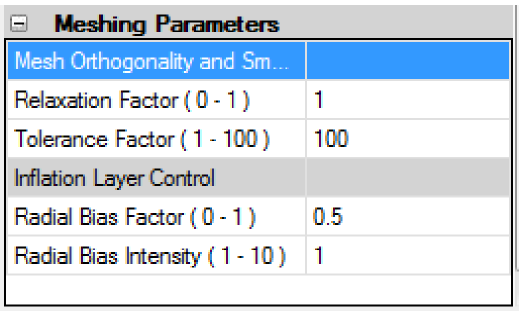
These inputs are used for adaptation and distribution of the grid points on the boundaries of the domain and for smoothing of rack (Rack is the curve representing a rotor with infinite radius which uniquely separates the flow domains of the male and female rotors).

  - Type of Distribution →Casing to Rotor Conformal



- Meshing Parameters:

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



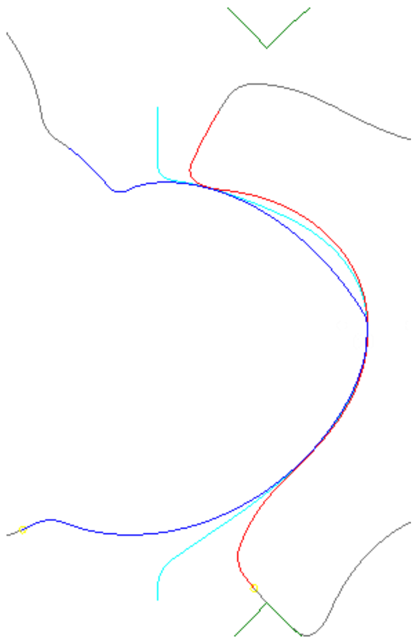
  - both the distribution and meshing parameters can be changed later

- Start Grid Generation through a three step process as below.
- Select Rack Refinement Points = 500

Control Switches	
Rack Generation	Off
Rack Refinement Points	500
Boundary Generation	Off
Fluid Rotor Grid	Off
Solid Rotor Grid	Off
Inlet Port Grid	Off
Outlet Port Grid	Off
Preprocessor Input File	ANSYS FLUENT
Vertex Files Start Number	1
Vertex Files End Number	50

- 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.

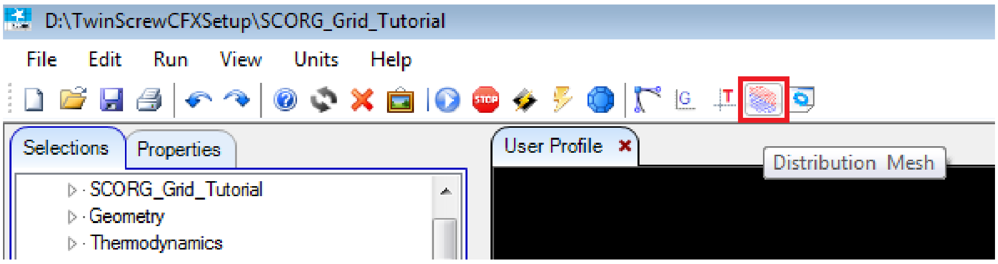


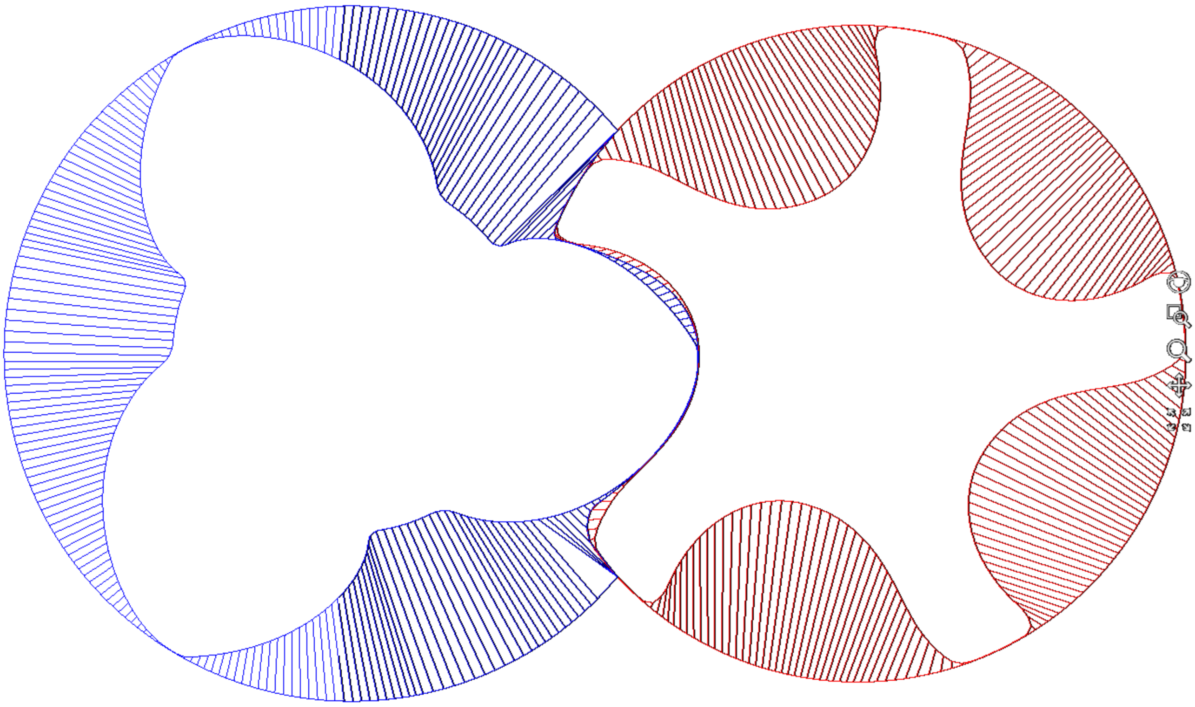
- Click Boundary Distribution Generation

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.

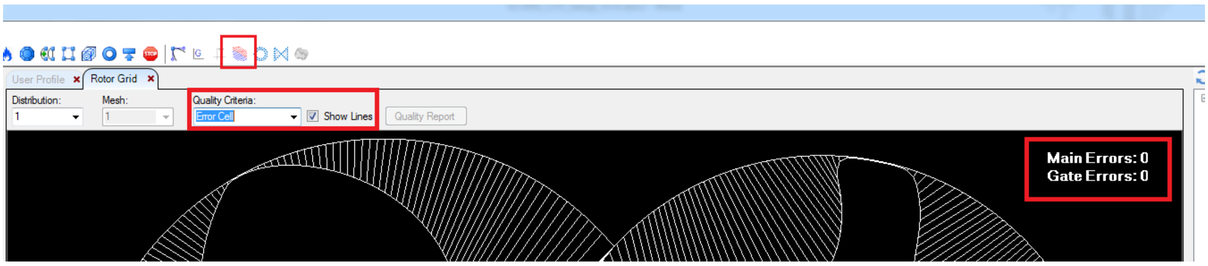
- Inspect report and check that there are no distribution warnings listed

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





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

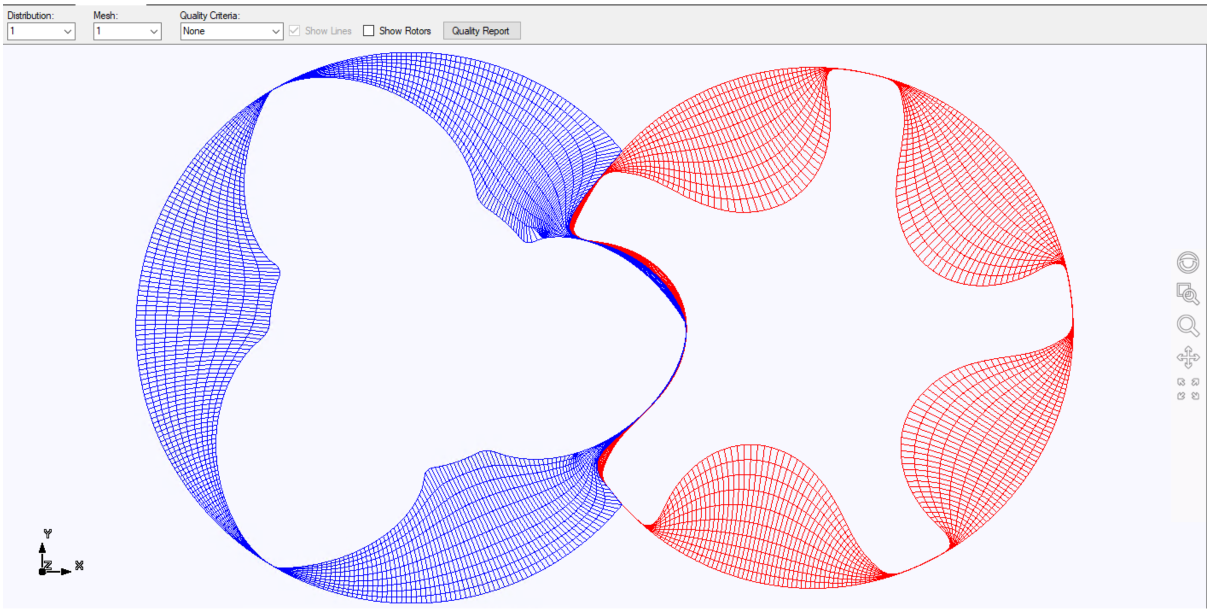


- Inspect all the distribution positions and ensure that 0 error are reported in each position.

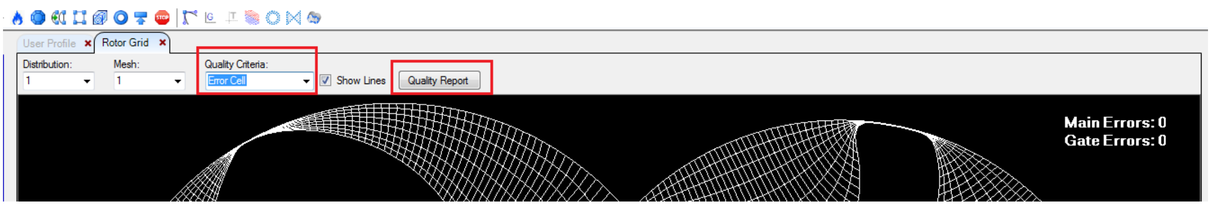
- Click Rotor Grid Generation

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

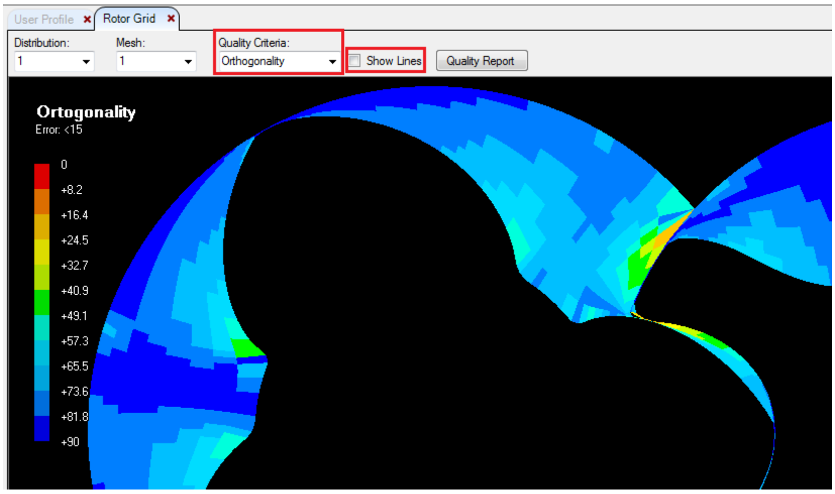
- Click Rotor Grid 2D Mesh to visually inspect the grid in each cross section



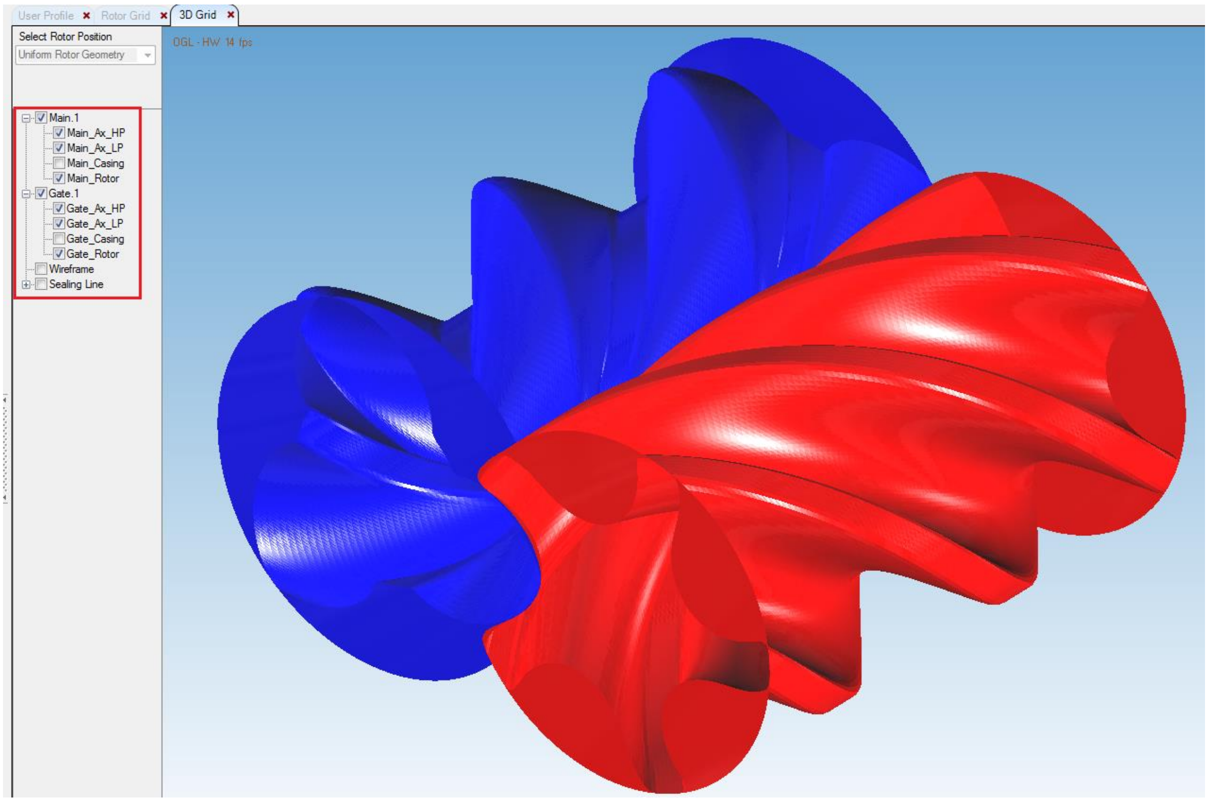
- Click Quality Critera → Error Cell and Inspect.



- Click Quality Critera → Orthogonality and Inspect.



- Inspect the 3D mesh



- In Control Switches → Preprocessor Input File select → FLUENT
- Set Vertex Files Start = 1
- Set Vertex Files End = 50
  [= Number of Angular Divisions]

- Calculate Preprocessor Files Generation

3 Code Compiler

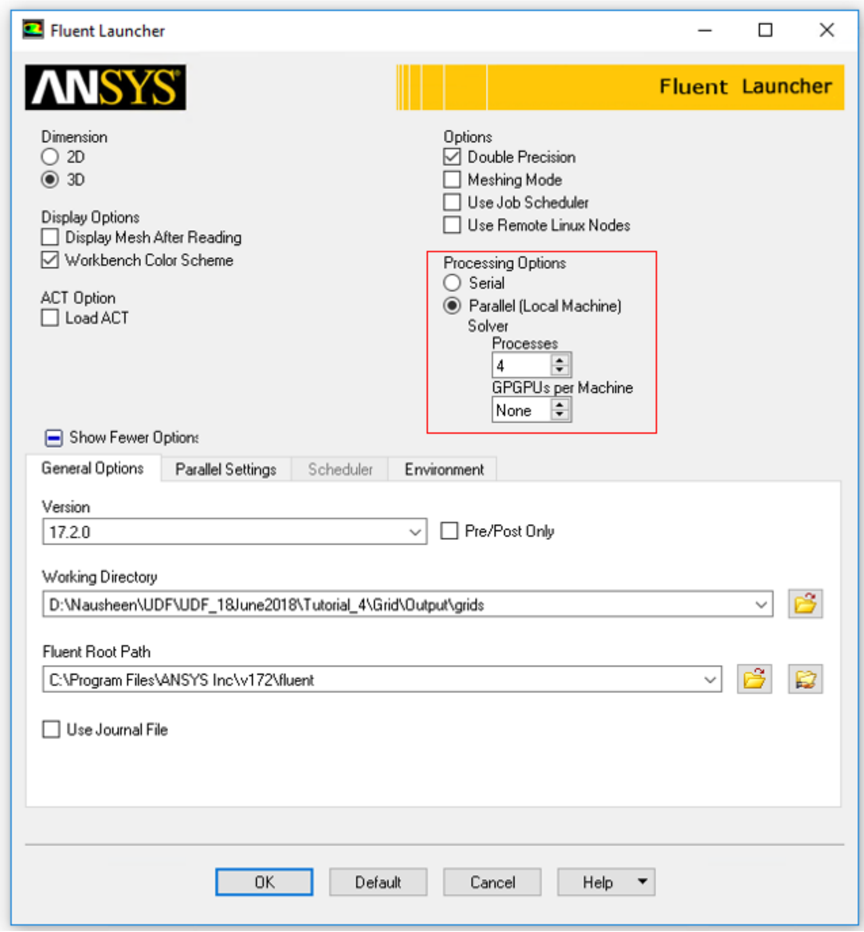
The case requires Microsoft Visual Studio to compile User Defined Functions (UDF), please ensure that this software is installed before proceeding forward.

4 ANSYS FLUENT

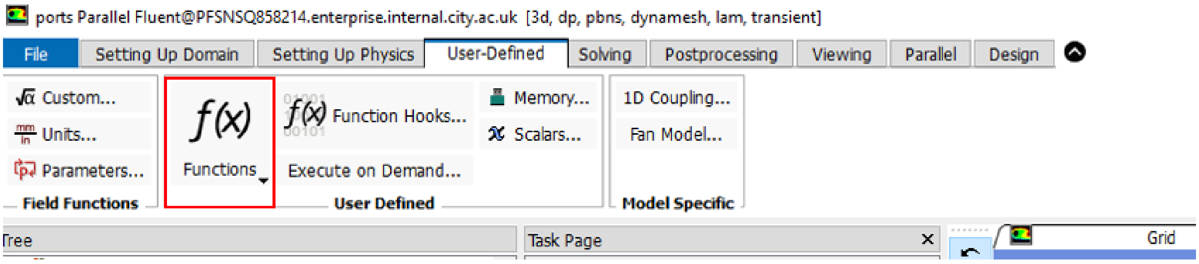
1. In the folder Grid-> Output-> FLUENT, is the file named 'Rotor_screw.msh'. Copy this file and place it into Grid-> Output-> grids folder
2. Depending up on the available FLUENT license decide if the case needs to be solved with serial or parallel solver

4.1 PARALLEL

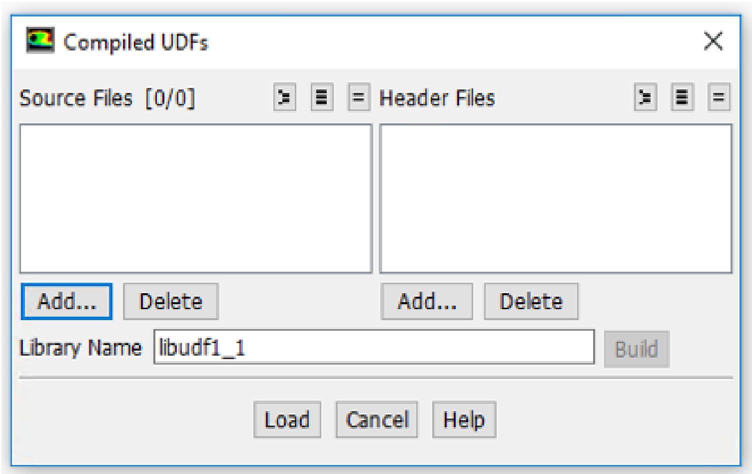
1. Open ANSYS FLUENT in parallel (ex. 4 nodes)



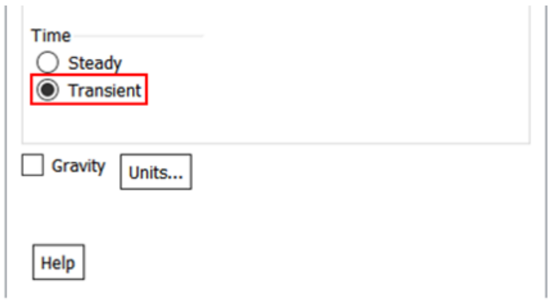
2. Choose User-Defined->functions tab and select Compiled option



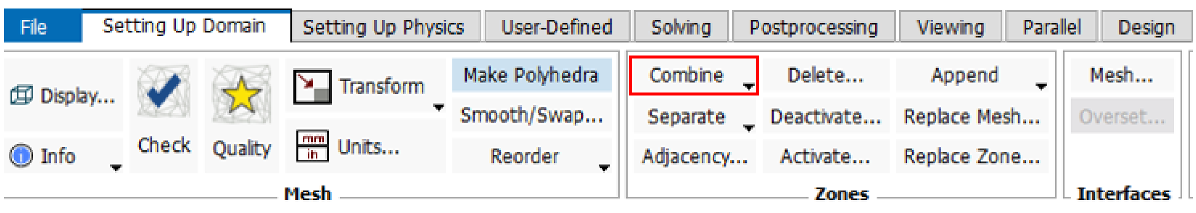
3. With the compile option click add and choose the file 'Node_Mapping_Parallel_1.c', click build and load



4. Read Rotor_screw.msh file
5. Click Setup->General and pick time as 'Transient'



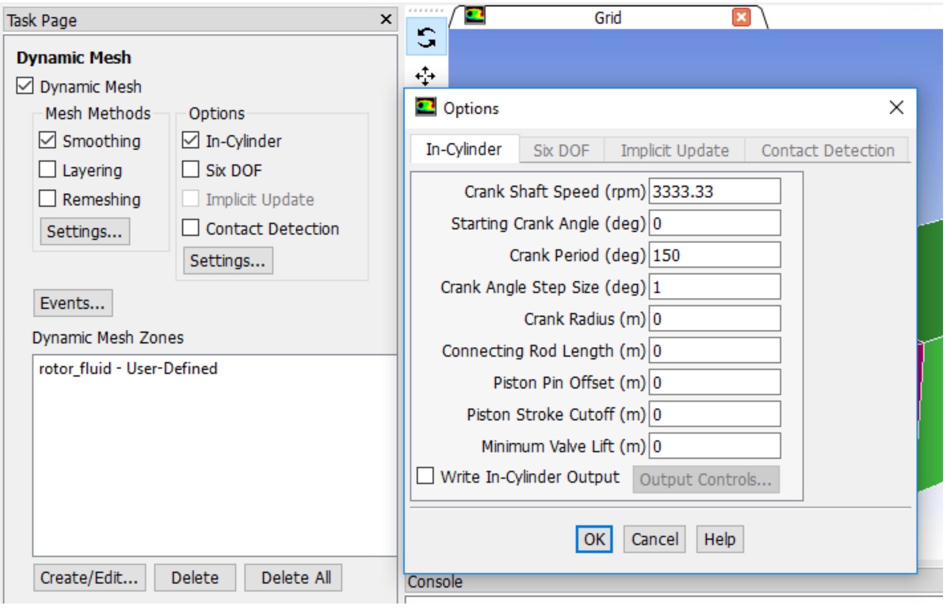
6. From Combine option, select Merge. Merge male_ax_1 and female_ax_1. Merge male_ax_2 and female_ax_2 surfaces
7. From Combine option, select Fuse and Fuse male_intf and female_intf zones



8. Load scheme file ‘01_input_data.scm’. Update scheme file according to the case

   (rp-var-define ‘txt_nbr 50 ‘integer #f) – Number of node file to be used in a cycle
   (rp-var-define 'om_full 8000 'real #f) – Main rotor RPM
   (rp-var-define 'om_ratio 1.6667 'real #f)- Ratio of male rotor to gate rotor speed

9. Go to Dynamic Mesh and populate In—Cylinder settings



   This is used to get the parameter 'crank-angle' for node position interpolation.
   Preferably, generate SCORG grids with 1 degree angle per step.
   Grids without 1 degree per step are not valid for interpolation

   a. Crank Shaft Speed = (Desired main rotor rpm/SCORG grid degree per step)
      Returns the time step size
      Example if Main rotor rpm = 8000
      If Grids have 3 degree per step rotation, then set Crank Shaft Speed = 2666.6667
      If Grids have 1 degree per step rotation, then set Crank Shaft Speed = 8000.0
      This will return timestep size = 6.25e-05 sec.

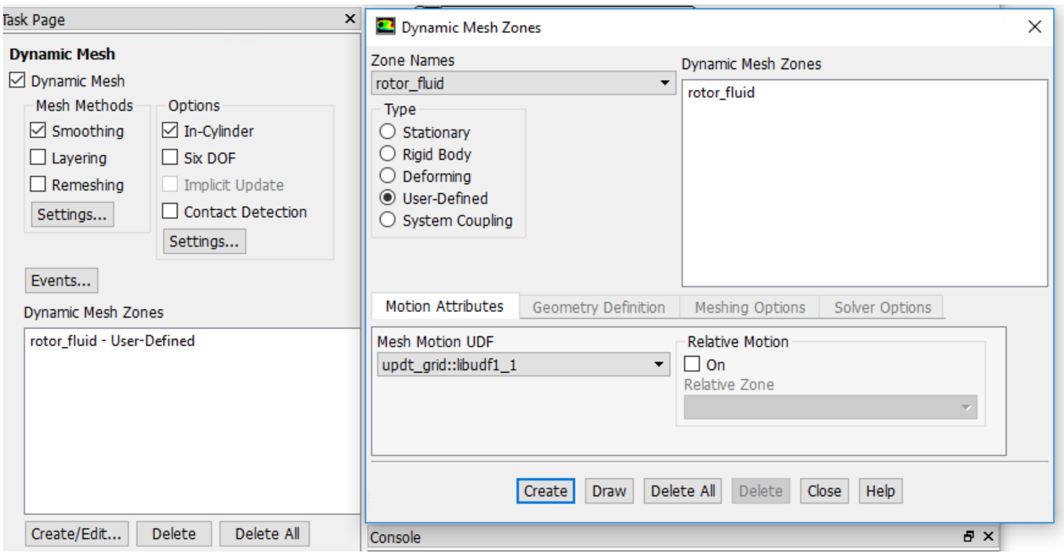
   b. Crank Period = (Number of Grid files*Number of Main rotor lobes)
      Example for 3 lobes and 40 grid files, Crank Period = 120
      Not used by Grid Deformation.
      Used by addaptive time stepping function.

   c. Crank Angle step size <= 1 degree and (1/Crank Angle step size) should return an integer.
      This setting is independent of SCORG grid angle per step.
      Example: 0.01, 0.25, 0.5
      If set > 1, grid files will get skipped (Not valid).

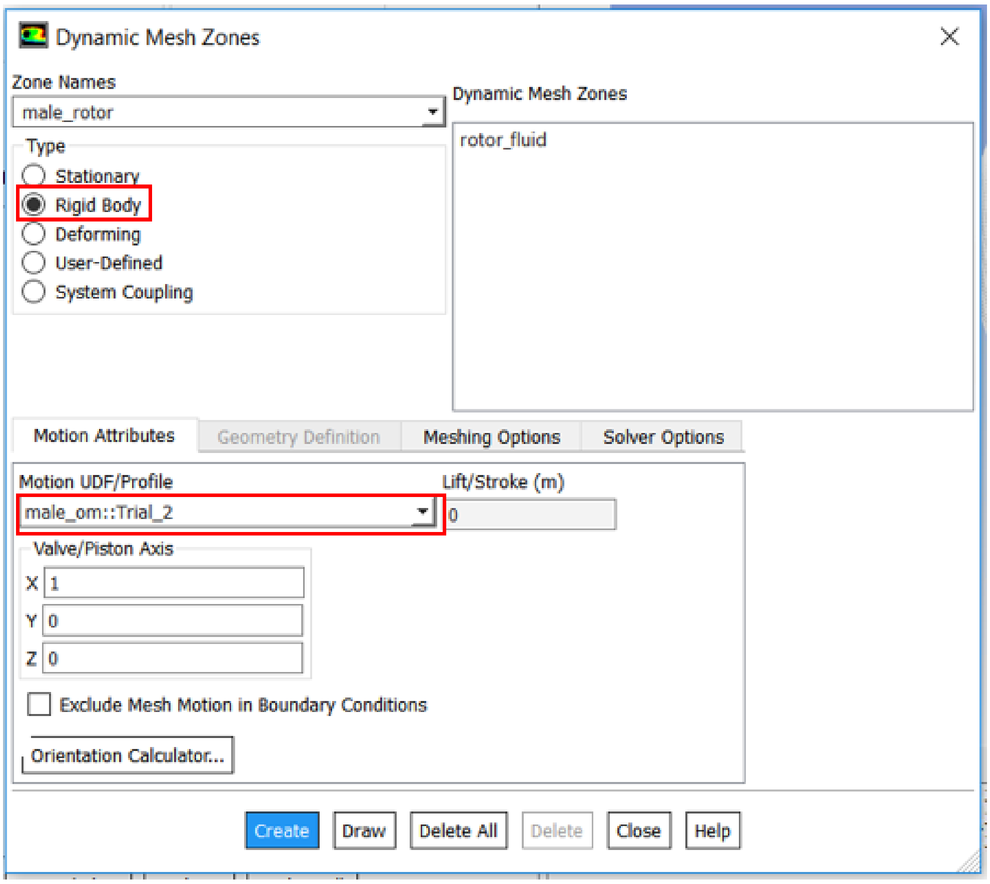
If set as 1 degree, No grid node Interpolation is done (Preferred).

If set as < 1 degree, Intermediate angles are interpolated (In this case it is necessary to have SCORG grids at 1 degree angle per step)

10. Create rotor_fluid as a Dynamic zone by choosing type as 'User-Defined'



11. Create male_rotor as a Dynamic zone by choosing type as 'Rigid Body'. Attach motion UDF.

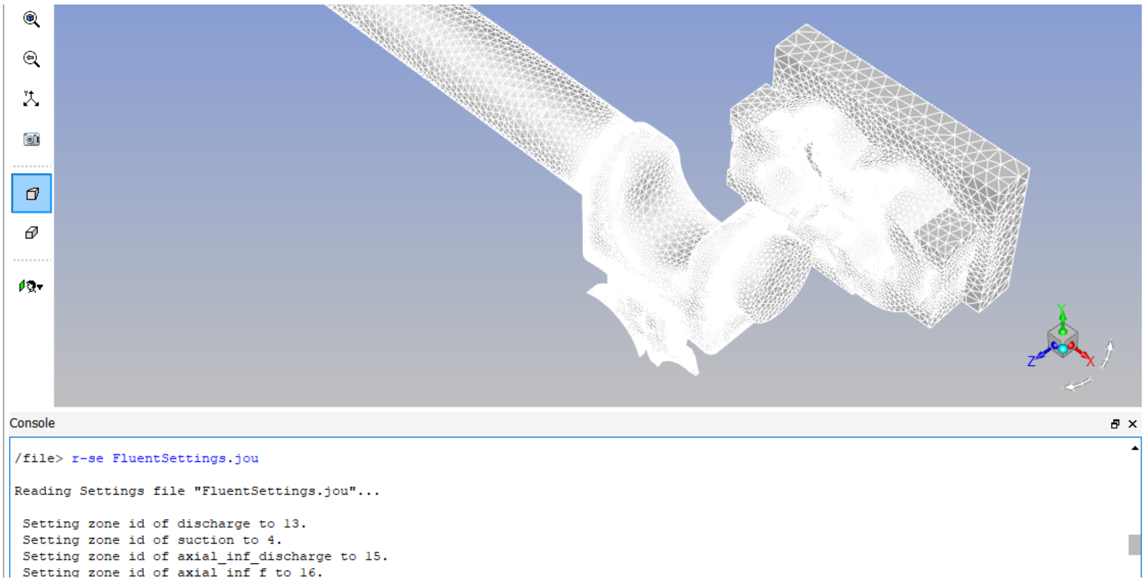


12. Do the same with female_rotor

13. Go to Execute on Demand, choose the UDF function named ‘distance’ and execute. Notice that the file named ‘Nodemap_para_05.txt’ is written to the folder

14. Load the file for suction and discharge ports as a case file. Zones->Append case file->ports.cas.gz

15. Load ready settings using TUI command File→read-setting→FluentSettings.jou.



16. Setup solution monitors for pressure points and temperature
17. Initialise solution and run calculation with number of iterations per time step as at least 10-20. Run the simulation for around 1200 time steps

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