

# SCORG<sup>TM</sup> Setup for CFD Simulation of Twin Screw Machines with ANSYS FLUENT<sup>®</sup>

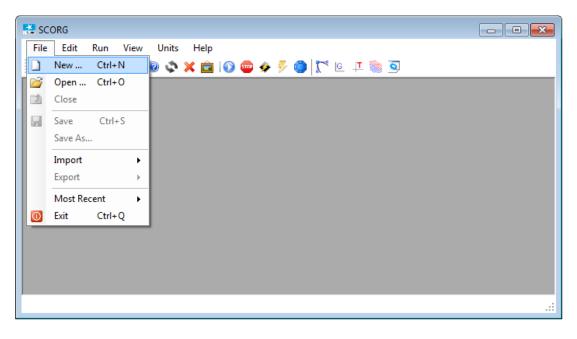
**SCORG**<sup>™</sup> is the CFD grid generation tool for rotary twin screw machines. The tool includes additional modules for designing and editing rotor profiles, executing a basic thermodynamic calculation based on quasi 1D chamber models and generating the deforming working chamber grids for selected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

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

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

## **1** SCORG<sup>™</sup> Project

- ▶ Launch SCORG<sup>TM</sup> on the Desktop.
- ▶ Select File  $\rightarrow$  New



▶ Select N35\_Template.spt  $\rightarrow$  Open





) 🖯 🗸 🖉 Vevelop	ment   SCORG(C)   SGUI   Scorg	▶ bin ▶ Templates	<ul> <li>✓ </li> <li>✓ Search Templates</li> </ul>	
Organize 🔻 New fol	der			?
🔶 Favorites	Name	Date modified		
🧮 Desktop	A46_Template.spt	27/08/2014 11:46		
🗼 Downloads	Circ46_Template.spt	27/08/2014 11:46		
Recent Places	Inv22_Template.spt	27/08/2014 11:46		
퉬 SkyDrive	Inv33_Template.spt	27/08/2014 11:46		
=	N35_Template.spt	27/08/2014 11:46		
词 Libraries	N45_Template.spt	27/08/2014 11:46		
Documents	N46_Template.spt	27/08/2014 11:46	No preview available.	
👌 Music	N56_Template.spt	27/08/2014 11:46		
Pictures	N57_Template.spt	27/08/2014 11:46		
Subversion	N67_Template.spt	27/08/2014 11:46		
📑 Videos				
🖳 Computer				
🏭 Local Disk (C:)				
👝 D (D:) 👻	•	4		
File	name: N35_Template.spt		✓ Scorg template (spt) (*.spt)	•
			Open Cance	

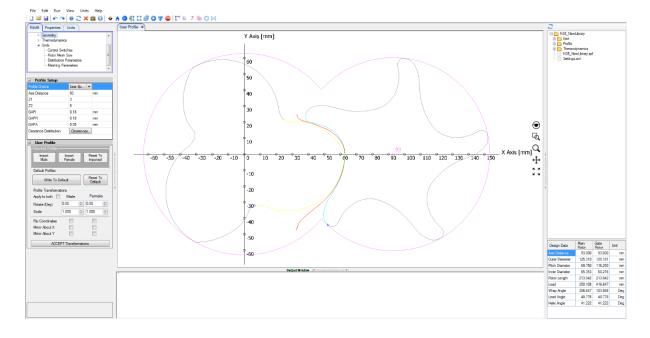
► Save the project in a new folder named TwinScrewFLUENTSetup → SCORG\_Grid\_Tutorial.spf

🚼 Save As			×
Co	omputer + D (D:) + TwinScrewCFXSetup	✓ ← Search TwinScrewCFXSetup	٩
			0
E Recent Places	Name	Date modified Type	1
🍌 SkyDrive	Noi	tems match your search.	
詞 Libraries 🖹 Documents	E		
J Music			
Pictures			
Videos			
📜 Computer	▼	III	÷.
File name:	SCORG_Grid_Tutorial		•
Save as type:	Scorg Project file (spf) (*.spf)		-
) Hide Folders		Save	

▶ The GUI of SCORG<sup>TM</sup> 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.

😫 C:\U - [SCORG V5.4.	1
File Edit Run	View Units Help
🗋 💕 🖬   ፍ 🧇	0 2 🗙 💼 💿 🐓
Inputs Units Pro	perties
Variable	Units
Pressure	Pa 💌 📰
Temperature	°C 🗕
Length	m 🔻
Density	kg/m³ ▼
SpecificHeat	J/(kg.K)
	· · · · ·

▶ Go to Help  $\rightarrow$  Tutorials  $\rightarrow$  Folder opens





SCORG(c)_2014 ► De	elopment > SCORG(C) > Tutorials - 47 Search Tut	\ م
Organize 🔻 🔄 Open 🛛 Burn	New folder	0
🔆 Favorites	Name Da	ate me
🧮 Desktop	S5FemaleProfile_P2.dat 11	1/09/2
🚺 Downloads	S5MaleProfile_P1.dat 11	1/09/2
🔄 Recent Places	W SCORG_CFX_Tutorial_Ports_V5.0.cfx 31	1/08/2
퉬 SkyDrive	SCORG_CFX_Tutorial_V5.0.ccl 31	1/08/2
	SCORG_Pumplinx_Setup_V5.0.pdf 22	2/09/2
ز Libraries	SCORG_Pumplinx_Tutorial_Ports_V5.0.sgrd 19	9/09/2
Documents		
👌 Music		
Pictures		
Subversion	▼	- F
4 items selected Date mo	fied: 31/08/2014 17:08 - 11/0 Date created: 19/09/2014 16:50 - 22/09 Size: 18.3 MB	9/

- Copy the compressor rotor profile files → [ 35MaleProfile\_P1.dat and 35FemaleProfile\_P2.dat ]
- Go to User Profile  $\rightarrow$  Browse and Select the Male Rotor Profile from working directory.

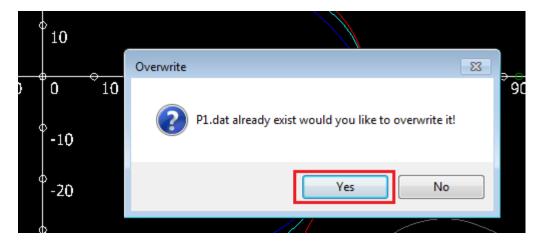
35MaleProfile\_P1.dat

User Profile		_			
– Imported Profiles –					
Import Male	Import Female	Reset To Imported		 ÷-60	
Default Profiles		_			
Write To De	efault	Reset To Default			
Profile Transformat	ions		=  *	$\langle \rangle$	
Apply to both	Male	Female		$ \rightarrow $	
Rotate (Deg)	0.00	0.00	÷		
Scale	1.000	€ 1.000	÷		
Flip Coordinates					
Mirror About X					
Mirror About Y					
ACCE	PT Transform	ations			

Click 'Yes' to overwrite P1.dat.







Similarly Select the Female Rotor Profile.

35FemaleProfile\_P2.dat

► Click Write To Default.

User Profile			
- Imported Profiles —			
Import Male	Import Female	Reset To Imported	
Default Profiles		_	
Write To De	fault	Reset To Default	
Profile Transformati	ons		3
Apply to both	Male	Female	
Rotate (Deg)	0.00	0.00	* *
Scale	1.000	1.000	*
Flip Coordinates			
Mirror About X			
Mirror About Y			
ACCEF	PT Transfo	mations	

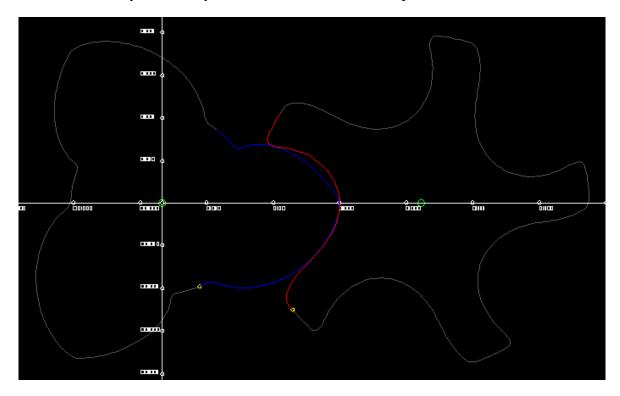
Click Refresh to view new profiles.

🛃 D:\TwinScrewPumplinxSetup\SCORG_Grid_1	Tutorial
File Edit Run View Units Help	
📄 🗅 📂 🖬 🚑 🍖 🤏 🏽 🚳 🖄	I 💽 🤹 🌾 🥘 🚺 🔀 🖉 🤹 💽
Selections Properties	User Profile ×
SCORG_Grid_Tutorial	

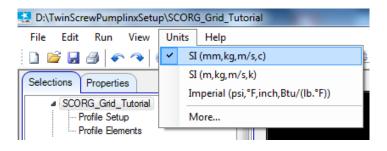




Inspect the Rotor Profile in the GUI for gaps in the tips, starting points of the profile indicated by the small yellow circles. Below is the required orientation.



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.





<ul> <li>▲ · Profile</li> <li>Profile Setup</li> <li>User Profile</li> <li>▷ · Geometry</li> <li>▷ · Thermodynamics</li> <li>▷ · Grids</li> </ul>	-	
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		

# • Go to Geometry $\rightarrow$ Set the following parameters:

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

• Go to Thermodynamics  $\rightarrow$  Set the following parameters:

Wtip	66.6665	m/s
Rotor Speed	10000	RPM
PO	100000	Pa
Pr	300000	Pa
то	293	К
Tr	350	К
Теур	268	K
Tcond	313	K
T Ambient	293	К
Ts	0	К
х	1	

► Save the Project.





# 2 SCORG<sup>™</sup> Mesh Generation

SCORG<sup>™</sup> 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<sup>TM</sup>. 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:
  - $\circ$  Circumferential Main = 0
  - $\circ$  Circumferential Gate = 70
  - $\circ$  Radial = 10
  - $\circ$  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  $\rightarrow$ Casing to Rotor Conformal





Circumferential Divisions Main .	0
Circumferential Divisions Gate .	70
Radial Divisions	10
Angular Divisions	50
Axial Divisions	0
Interlobe Divisions	50
Distribution Parameters	8
Type of Distribution	Casing to Rotor 🗸
K Main	2
K Gate	0.3
Rack Smoothing Factor	0.8
Project on Main profile	Yes ~
Meshing Parameters	
Mesh Orthogonality and Smo	
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
Conformal Mesh Control	

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

 $\circ$  both the distribution and meshing parameters can be changed later





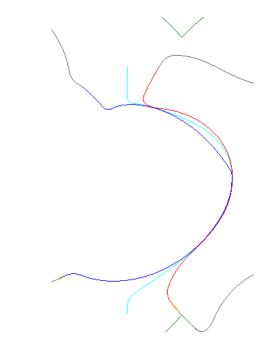
- ▶ Start Grid Generation through a three step process as below.
- ▶ Select Rack Refinement Points = 500

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

				Units															
	j 🔒	<b>~</b> ^	0	<b>2 x</b>	ê 🕥	<b>%</b>	8	٢	C	I	Ø	0	-	STOP	1	G	T	0 >	4
· ·	$\gamma_{\overline{-}}$								(		- D	£1	2						

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

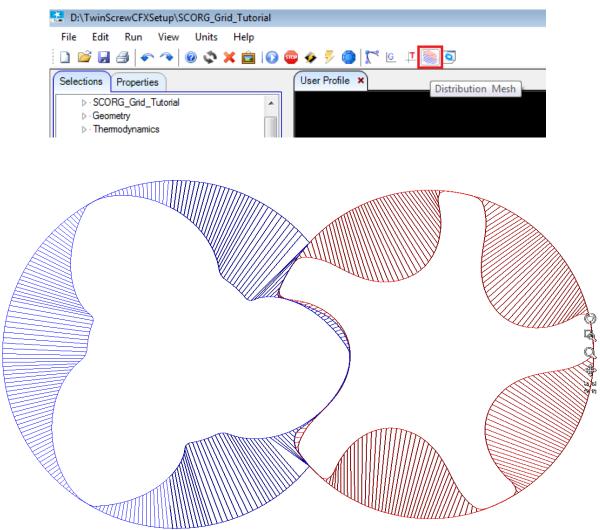






Information about the progress of the selected activities in the meshing procedure is displayed in the output window. Any warning or error and their locations are indicated. If errors occur, it is important to manually tune the input parameters which will produce a mesh without errors. Graphically the 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.

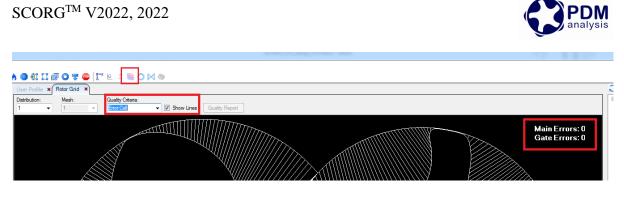
Inspect report and check that there are no distribution warnings listed



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

▶ In the Distribution Display  $\rightarrow$  Select Quality Criteria = Error Cell



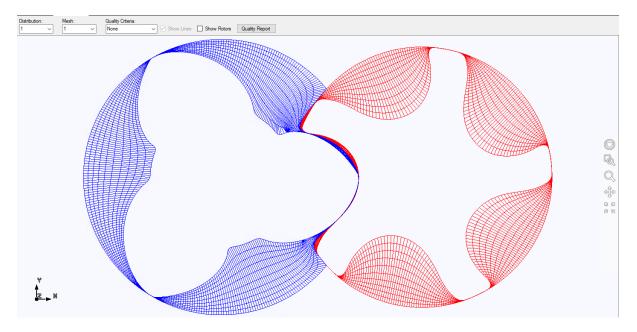


- Inspect all the distribution positions and ensure that 0 error are reported in each position.
- Click Rotor Grid Generation

File	Edit	Run	View	Units	Help										
	iii 📔	<b>~</b> ^		<b>2 x</b>	ê 💽 😽	۵	٢	60	II 🖉 O	) Ţ	STOP	G	T	Ô	$\bowtie$
	to 0		11-3-					(	Llear Profila	¥					

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

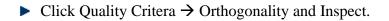


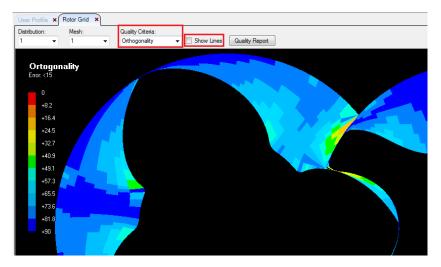
• Click Quality Critera  $\rightarrow$  Error Cell and Inspect.

🔥 🔿 👯 🖬 🖉 🔿 署 🚭 🕅	* 🕒 프 🐌 🔿 🕅 🕸			
User Profile × Rotor Grid ×				
Distribution: Mesh:	Quality Criteria: Error Cell  ✔ Show Line	s Quality Report		
				Main Errors: 0 Gate Errors: 0

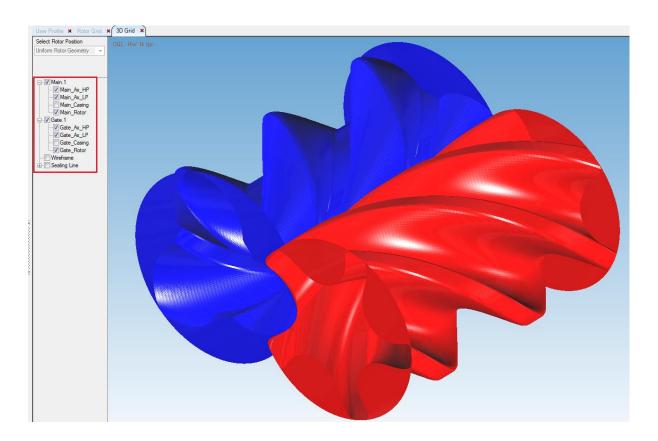








Inspect the 3D mesh



- ▶ In Control Switches  $\rightarrow$  Preprocessor Input File select  $\rightarrow$  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)

Eluent Launcher	>
AN <mark>SYS</mark>	Fluent Launcher
Dimension 2D 3D Display Options Display Mesh After Reading Workbench Color Scheme ACT Option Load ACT	Options Double Precision Meshing Mode Use Job Scheduler Use Remote Linux Nodes  Processing Options Solver Processes 4 GPGPUs per Machine None
Show Fewer Options     General Options     Parallel Settings     Scheduler	Environment
Version	
17.2.0	✓ Pre/Post Only
Working Directory D:\Nausheen\UDF\UDF_18June2018\Tutorial_4\Gri	d\Output\grids v 🔁
Fluent Root Path C:\Program Files\ANSYS Inc\v172\fluent	✓
Use Journal File	
OK Defa	ılt Cancel Help ▼





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

💶 ports Parallel Fluent@PFSNSQ858214.enterprise.internal.city.ac.uk [3d, dp, pbns, dynamesh, lam, transient]

File Setting U	Ip Domain	Setting Up Physics Use	r-Defined Sol	ving Postprocessing	Viewing Para	allel Design	
√α Custom	fa	f(X) Function Hooks	🛔 Memory	1D Coupling			
mm Units	f(x)	50101 Function Hooks	X Scalars	Fan Model			
Parameters	Functions	Execute on Demand					
— Field Functions —		User Defined		Model Specific			
Tree			Task Page		×		Grid

3. With the compile option click add and choose the file 'Node\_Mapping\_Parallel\_1.c', click build and load

Compiled UDFs			×
Source Files [0/0]	) E =	Header Files	> = =
Add Delete		Add Delete	
Library Name libudf1_1			Build
	Load Car	Help	

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

Time Steady Transient	
Gravity Units	
Неір	

- 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

File Set	tting Up	Domain	Setting Up Physic	cs User-Defined	Solving F	ostprocessing	Viewing	Parallel	Design
E Disabu			Transform	Make Polyhedra	Combine 🖕	Delete	Append	•	Mesh
🗇 Display	10:25-7-31	$\sim$		Smooth/Swap	Separate 🖕	Deactivate	Replace Mes	h (	Overset
🕕 Info 🖕	Check	Quality	in Units	Reorder 🖕	Adjacency	Activate	Replace Zon	e	
			Mesh				nterfaces		





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

Task Page	×	Grid 🛛	
Dynamic Mesh		<b>4</b>	
Dynamic Mesh Mesh Methods Smoothing Layering Remeshing Settings Events	Options DI-Cylinder Six DOF Implicit Update Contact Detection Settings	Crank Shaft Speed (rpm) 3333.33 Starting Crank Angle (deg) 0 Crank Period (deg) 150 Crank Angle Step Size (deg) 1	
Dynamic Mesh Zor rotor_fluid - User-		Crank Radius (m) 0 Connecting Rod Length (m) 0 Piston Pin Offset (m) 0 Piston Stroke Cutoff (m) 0 Minimum Valve Lift (m) 0 Write In-Cylinder Output Output Controls	
Create/Edit	Delete Delete All	Console	

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'

Task Page ×	Dynamic Mesh Zones	×
Dynamic Mesh Dynamic Mesh Smoothing Layering Remeshing Settings Events	Zone Names Dynamic Mesh Zones       Totor_fluid       Type       Stationary       Rigid Body       Deforming       I User-Defined       System Coupling	
Dynamic Mesh Zones	Motion Attributes Geometry Definition Meshing Options Solver Options	
rotor_fluid - User-Defined	Mesh Motion UDF updt_grid::libudf1_1  Create Draw Delete All Delete Close Help	
Create/Edit Delete Delete All	Console 8 2	×

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

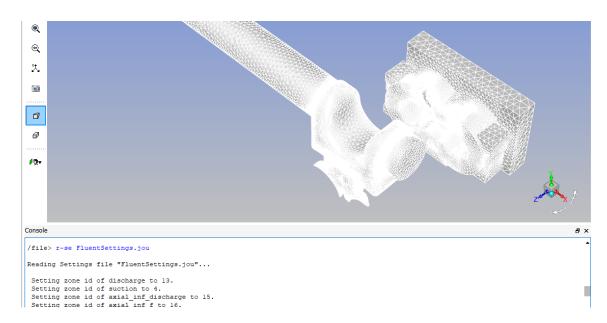
Dynamic Mesh	Zones		$\times$
Zone Names		Dynamic Mesh Zones	
male_rotor Type Stationary Rigid Body Deforming User-Defined System Coupling		rotor_fluid	
Motion Attributes Motion UDF/Profile male_om::Trial_2	Geometry Definition	Meshing Options Solver Options	
Valve/Piston Axis		0	
X 1 Y 0 Z 0 Exclude Mesh Mo	otion in Boundary Conditio	ons	
Orientation Calculate	or		
	Create	Delete All Delete Close Help	

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 $\rightarrow$ read-setting $\rightarrow$ 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 <u>SCORG@PDMAnalysis.co.uk</u> <u>http://www.pdmanalysis.co.uk</u>

