

SCORGTM Setup for CFD Simulation of Twin Screw Machines with ANSYS CFX[®]

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: <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[™] and ANSYS CFX Solver. The user is expected to be familiar with screw machines, CFD and ANSYS CFX[®] in order to be able to use these procedures. It is highly recommended that books on that topic are studied¹²

The setup steps here are demonstrated for Linux & Windows 7, x64 bit OS. Refer SCORG[™] Installation Guide for system and hardware requirements.

Table of Contents

1	1 Introduction	2
2	2 SCORG TM Project	3
3	3 SCORG TM Mesh Generation	
4	4 Compiler Environment setup [<i>One time procedure</i>]	19
	4.1 Compilation of Junction box Subroutines	20
5	5 CFX Pre case setup	24
6	6 CFX Solver Calculation	
	6.1 Restarting a Simulation from an intermediate Stop	
7	7 Distribution Options	
8	8 Summary	
9	9 Bibliography	

² A. Kovacevic. N. Stosic, I.K. Smith, Screw Compressor Three Dimensional Computational Fluid Dynamics and Fluid Solid Interaction, Springer, 2006, ISBN 3-540-36302-5



¹ N. Stosic, I.K. Smith, A. Kovacevic Screw Compressor Mathematical Modelling and Performance Calculation, Springer, UK 2005, ISBN-10 3-540-24275-9



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.

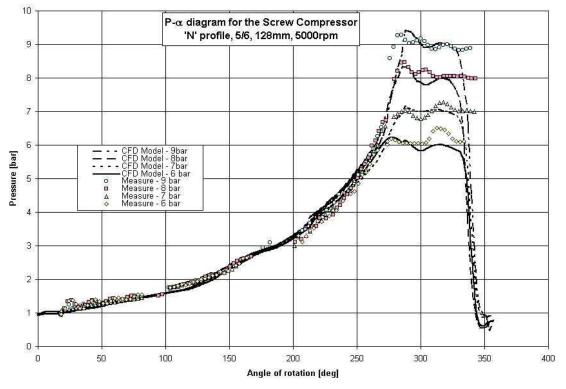


Figure 1-1 Pressure Variation diagram of a Twin Screw Compressor (Kovacevic, et al., 2007)

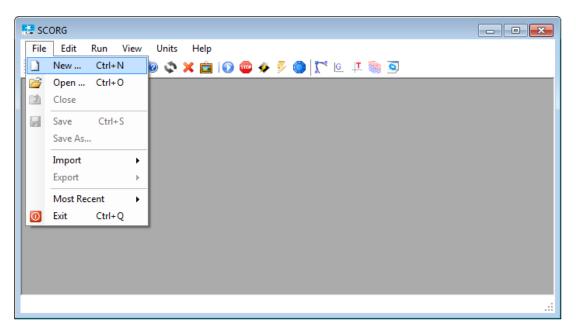




This Tutorial will provide a step by step guide for the procedure to setup and execute a typical twin screw compressor, pump or motor simulation. An example of a dry air compressor with 3/5 lobe combination, L/D ratio of 1.7 and wrap angle 285 deg has been considered.

2 SCORG[™] Project

- ► Launch SCORGTM on the Desktop.
- ▶ Select File \rightarrow New



► Select N35_Template.spt \rightarrow Open

Organize 🔻 👘 New fold	der		!≕ ▼ □ (
🛧 Favorites 🕇	Name	Date modified	
Mesktop	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	
E	N35_Template.spt	27/08/2014 11:46	
📜 Libraries	N45_Template.spt	27/08/2014 11:46	
Documents	N46_Template.spt	27/08/2014 11:46	No preview available.
J 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:) 👻	•	•	

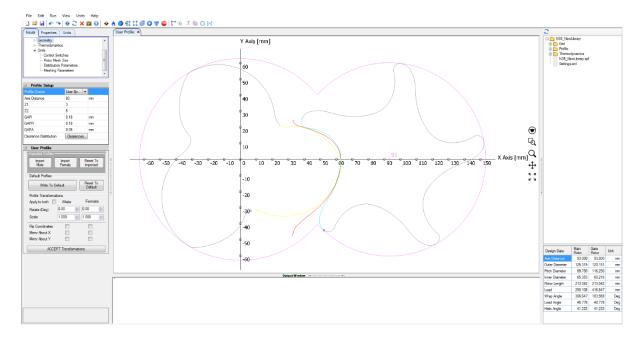




► Save the project in a new folder named TwinScrewCFXSetup → SCORG_Grid_Tutorial.spf

Save As	omputer 🕨 D (D:) 🕨 TwinScrewCFXSetup	o	nScrewCFXSetup
 Recent Places SkyDrive Libraries Documents Music Pictures Subversion Videos 		Date modified No items match your search.	₹ Type :
File name: Save as type: Hide Folders	SCORG_Grid_Tutorial Scorg Project file (spf) (*.spf)	III Save	Cancel

► The GUI of SCORGTM in the figure bellow shows the mains items of the front panel.



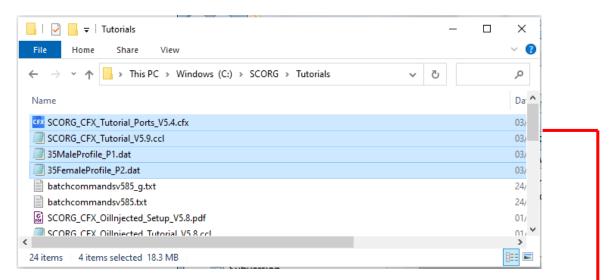




In Units Tab, Select Length units as 'm'. This selection has to be the same as the units in which input profile coordinates are available.

C:\U - [SCORG V5.4.]					
File Edit Run	View Units Help				
🗋 💕 🛃 < 🔦	0 🎅 🗙 🧰 🔘 🔌				
Inputs Units Pro	operties				
Variable	Units				
Pressure	Pa 💌 =				
Temperature	°C 🔻				
Length	[m 🗸				
Density	kg/m³ 💌				
SpecificHeat	J/(kg.K) 💌				
	· · · · · · · · · · · · · · · · · · ·				

► Go to Help \rightarrow Tutorials \rightarrow Folder opens



Jame Date modified Type Size 35FemaleProfile_P2.dat 03/03/2021 11:05 DAT File 18 KB	
35FemaleProfile_P2.dat 03/03/2021 11:05 DAT File 18 KB	
35MaleProfile_P1.dat 03/03/2021 11:06 DAT File 23 KB	
X SCORG_CFX_Tutorial_Ports_V5.4.cfx 03/03/2021 11:07 ANSYS 2020 R2 .cf 18,696 KB	
SCORG_CFX_Tutorial_V5.9.ccl 03/03/2021 11:06 CCL File 43 KB	

- Copy the compressor rotor profile files → [35MaleProfile_P1.dat and 35FemaleProfile_P2.dat]
- ► Copy the compressor suction and discharge port grids → [SCORG_CFX_Tutorial_Ports_V5.4.cfx]





- ▶ Copy the CFX setup script \rightarrow [*SCORG_CFX_Tutorial_V5.9.ccl*]
- ▶ Paste these files in the working directory \rightarrow TwinScrewCFXSetup
- ►
- Go to User Profile \rightarrow Browse and Select the Male Rotor Profile from working directory.

User Profile				
- Imported Profiles -	Import	Reset To	 _	
Male	Female	Imported		-60
Default Profiles				
Write To D	efault	Reset To Default	and the second	
Profile Transformat	ions		. ₫	
Apply to both	Male	Female		
Rotate (Deg)	0.00	0.00		
Scale	1.000	1.000 🚔		
Flip Coordinates				
Mirror About X				
Mirror About Y				
ACCE	PT Transform	nations		

35MaleProfile_P1.dat

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 D	efault	Reset To Default
Profile Transformat	tions	
Apply to both	Male	Female
Rotate (Deg)	0.00	0.00
Scale	1.000	1.000
Flip Coordinates		
Mirror About X		
Mirror About Y		
ACCE	PT Transfo	omations

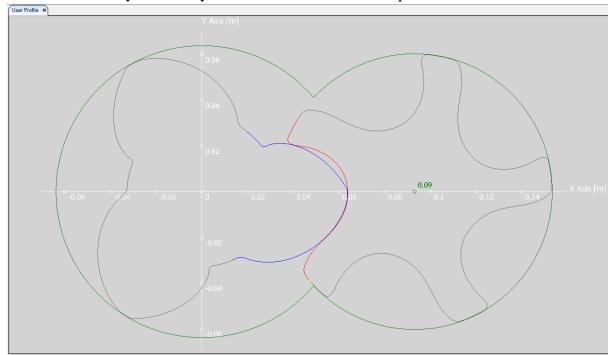
Click Refresh to view new profiles.



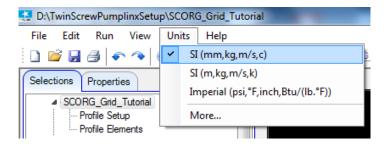




Inspect the Rotor Profile in the GUI for gaps in the tips, starting points of the profile indicated by the small yellow circles. Below is the required orientation.



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 \rightarrow Set the following parameters:

			Rotor Configurati	on	
			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
⊿ · Profile	_		Rotor Pitch	Uniform 💌	
Profile Setur			Rotor Profile	Constant 💌]
⊳ Geometry			Main Rotor Centre X	0	mm
⊳ · Thermodynamics ⊳ · Grids			Main Rotor Centre Y	0	mm
p · cinds			Main Rotor Centre Z	0	mm
			Main Rotor Start Angle	0	Deg
Profile Setup Profile Choice	User Sp 🔻]	Rotor Stage Number	0	
Axis Distance	93	mm	Main Rotor Helix	Right -	
Z1	3		Gate Rotor Position	Right 💌	
Z2	5				
GAPI	0.06	mm	Machine Configur	ation	
GAPR	0.06	mm	Machine Type	Compressor	
GAPA	0.05	mm	N Gate	1	
Clearance Distribution	Clearences	J	Compression Start	0	
User Profile		_	Compression End	161.001	

• Go to Thermodynamics \rightarrow Set the following parameters:

Wtip	66.6665	m/s
Rotor Speed	10000	RPM
P0	100000	Pa
Pr	300000	Pa
то	293	К
Tr	350	К
Теvp	268	К
Tcond	313	К
T Ambient	293	К
Ts	0	К
х	1	

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 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 SCORGTM. 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 = 60
 - \circ Radial = 8
 - \circ Angular = 50
 - \circ Interlobe Divisions = 50

Rotor Mesh Size				
Circumferential Divisions Main Rotor				
Circumferential Divisions Gate Rotor	60			
Radial Divisions	8			
Angular Divisions	50			
Axial Divisions	0			
Interlobe Divisions	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

Distribution Parameters				
Type of Distribution	Casing to Rotor Conformal			
K Main	3			
K Gate	0.3			
Rack Smoothing Factor	0.8			
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 = 400

Rack Generation	Off	~
Rack Refinement Points	400	
Boundary Generation	Off	~
Fluid Rotor Grid	Off	~
Solid Rotor Grid	Off	~
Inlet Port Grid	Off	~
Outlet Port Grid	Off	~
Preprocessor Input File	Off	~
Vertex Files Start Number	1	
Vertex Files End Number	50	

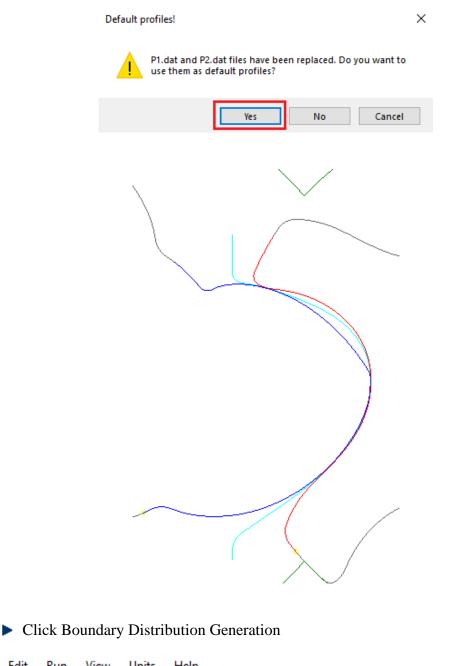
Click Numerical Rack Generation

File	Edit	Run	View	Units	Help															
	i 🚽	<u> </u>	0	2 ×	ê 🕥	ý	0	٢	C	Ц	đ	0	-	5709	1	G	T	\bigcirc	\bowtie	
<u> </u>		. V							(£1	5							

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







File Edit				_				
🗋 🚰 🛃 -	∽ へ 🖉) 🎅 🗙 💼	🕥 🐓 👌	11 109 🔘 1	Ø O 🔻	📼 🏷	' 🕒 🎩 🍉 🔿 🖂	
Inputs Prope	erties Uni	ts		Use	r Profile 🗙			

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

C:\SCORG\Grid>echo off

. InstallPath = C:\SCORG ProjectPath = D:\TwinScrewCFXSetup\SCORG_Grid_Tutorial SCORG - Screw COmpressor Rotor Geometry grid generator V.5.9 Screw compressor/p wrap = 283.2 RPM=12344. Vel= 82.3 Ncel= 566400 Z1/Z2= 3/5 d1=127.38 [mm] d2=120.32 [mm] a= 93.00 [mm] len=216.45 [mm] Nfi Nr Nz Nadd Rot Rack Boun Mesh RotM InpP OutP Prep RaSm Line Oil 60 8 118 50 3 1 1 0 0 0 0 0 1 1 0 1: 0.00 Dist 0.00 Cos Calculation: ROTOR 2: 0.00 Ang. 0.00 Sin Calculation: RACK Calculation: BOUNDARY Smoothing factor: 0.80 Male = 300 Smooth: ON Female = 300Initial Smoothing Distribution:Casing to Rotor Conformal TFI_Mesh routine - Rotor TFI_Mesh routine - Rotor Initial Smoothing GRID RelaxFac, TolFac, RadBFac, RadBInt, InterlobeBInt 1.0 100 0.5 1.0 PDF Interlobe mesh routine Distribution Type: Casing to Rotor Conformal . Distribution: Casing to Rotor Conformal Overall number of cells Cell statistics 0 .Inlet port .Outlet port 0 0 .Rotor fluid .Rotor solid 0 0 Start: 11:23:18 End: 11:24: 2 Running time: Oh: Om:43s = 42 sec 3/ 3/2021... SCORG - Screw COmpressor Rotor Geometry grid generator - Ver. 5.9

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

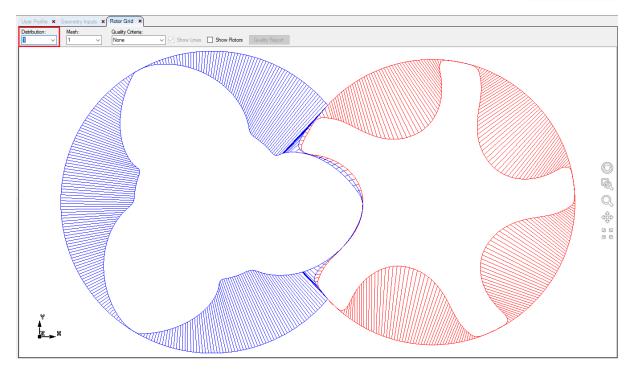
 Image: Selections
 Properties

 Image: Selections
 Image: Selections

 Image: Selections
 Image: Selections







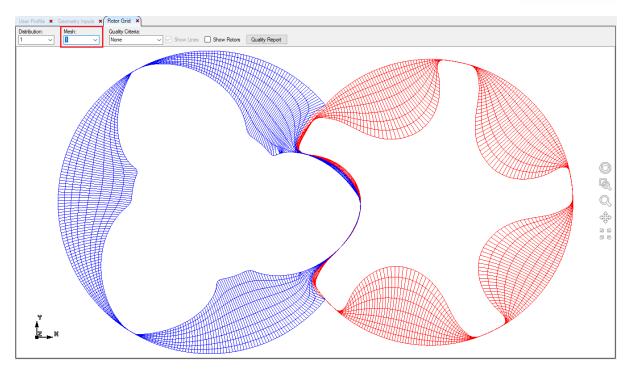
▶ In the Distribution Display \rightarrow Select Quality Criteria = Error Cell

User Profile x Rotor Grid x
Distribution: Mesh: Quality Citeta: 1 1 Immediate Immediate Quality Report Guality Report
Main Errors: 0 Gate Errors: 0
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
D 🖻 🖬 < < @ C 🗙 🖻 0 < < < < < < < < < < < < < < < < < <
Insuita Decention Unite (Hear Profile ¥)

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







C:\SCORG\Grid>echo off

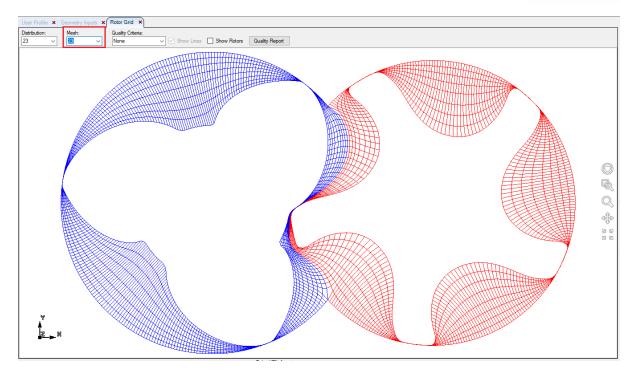
```
. . . . .
                                                            InstallPath = C:\SCORG
ProjectPath = D:\TwinScrewCFXSetup\SCORG_Grid_Tutorial
SCORG - Screw COmpressor Rotor Geometry grid generator V.5.9
Screw compressor/p Wrap = 283.2 RPM=12344. Vel= 82.3 Ncel= 566400
Z1/Z2= 3/5 d1=127.38 [mm] d2=120.32 [mm] a= 93.00 [mm] len=216.45 [mm]
                               .....
Nfi Nr I
              . . . . . . . . . . . . . . . . .
 NFi Nr Nz Nadd Rot Rack Boun Mesh RotM InpP OutP Prep RaSm Line Oil
60 8 118 50 0 0 0 1 0 0 0 0 1 1 0
                                                                                0
         . . . . . . . .
                        . . . . . . . . . . . . . . . .
                                           . . . . . . . . .
Calculation: FLUID GRID RelaxFac, TolFac, RadBFac, RadBInt, InterlobeBInt
                                 1.0
                                         100
                                                0.5
                                                            1.0
 TFI_Mesh routine - Rotor
TFI_Mesh routine - Rotor
                                        1
                                         2
 PDE_mesh routine - Rotor
PDE_mesh routine - Rotor
                                         1
                                         2
 PDE_Interlobe_mesh2 routine: Smooth Interlobe
            . . . . . . . .
                                                            Check_Grid - Rotor:
Check_Grid - Rotor:
                                   1
                                   2
 Write 2D Grid Data
 Grid Data Count:
Male rotor domain, Vertices: 321300, Cells
Female rotor domain, Vertices: 321300, Cells
                                                     283200
                                                     283200
Written Control.dat
                         Overall number of cells
Cell statistics
                                                              0
                       0
                            .Inlet port
.Rotor fluid
                                                              0
.Rotor solid
                      0
                                                              0
                                      .Outlet port
Start: 11:29:31 End: 11:29:42 Running time: Oh: Om:10s = 2 sec
3/ 3/2021...
SCORG - Screw COmpressor Rotor Geometry grid generator - Ver. 5.9
. . . . . . . .
```

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



© PDM Analysis Ltd, 2021

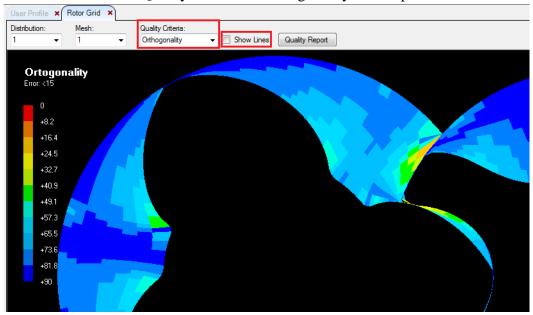




▶ Click Quality Critera \rightarrow Error Cell and Inspect.

🔥 🔘 👯 🖬 🖉 🔿 🐺 😁 🕽	7 L I 🛸 O M 👁		 	
User Profile × Rotor Grid ×				
Distribution: Mesh:	Quality Criteria:	ow Lines Quality Report		
				Main Errors: 0 Gate Errors: 0

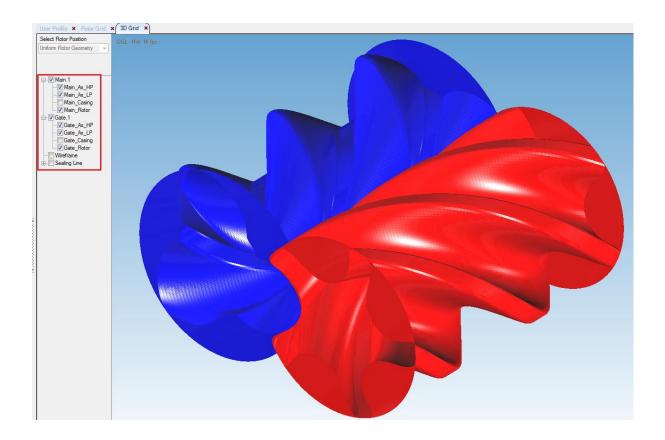
▶ Click Quality Critera \rightarrow Orthogonality and Inspect.



► Inspect the 3D mesh







- ▶ In Control Switches \rightarrow Preprocessor Input File select \rightarrow ANSYS CFX
- Set Vertex Files Start = 1
- Set Vertex Files End = 50 [= Number of Angular Divisions]

Rack Generation	Off	~
Rack Refinement Points	400	
Boundary Generation	Off	~
Fluid Rotor Grid	On	~
Solid Rotor Grid	Off	~
Inlet Port Grid	Off	~
Outlet Port Grid	Off	~
Preprocessor Input File	ANSYS CFX	~
Vertex Files Start Number	1	
Vertex Files End Number	50	

Calculate Preprocessor Files Generation



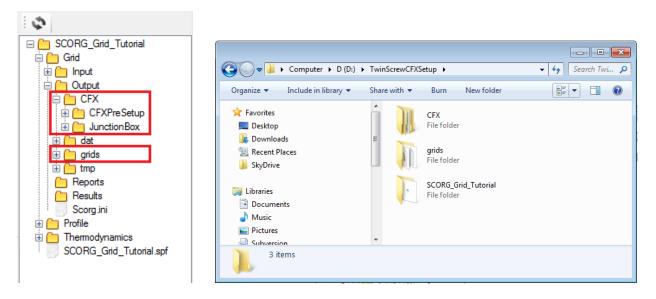


□ □
Installeath - Ci\Scopc
$T_{restall_path} = c_1 c_{resp}$
ProjectPath = D:\TwinScrewCFXSetup\SCORG_Grid_Tutorial
All control parameters for grid generation are disabled
All concrol parameters for grid generation are disabled
Conformal
Generation of Port Pre Processor files
Low Pressure Port not calculated
Pre-Processing Skipped High Pressure Port not calculated
Pré-Processing Skipped
Generation of Rotor Pre Processor files
Checking volumes in Male
Min/Max Volume= 1.0739071E-11 1.4677777E-08 Checking volumes in Female
Min/Max Volume= 8.2917484E-12 1.8342320E-08
SEX SETUR Saids units a
CFX SETUP Grids written
Generation of time step grid files
Start time step: 1 End time step: 50
end enne seep. 50
Rotor 1, Grid position 1
Rotor 1, Grid position 2 Rotor 1, Grid position 3

- ▶ With this the SCORGTM Project is complete and the CFX setup can be started.
- ► In the directory structure of SCORGTM Project → Grid → Output with consist of CFX and grids folder.
- ▶ Copy these two folders in the project working directory \rightarrow TwinScrewCFXSetup
- ► The CFX folder consists of two sub folders
 - CFXPreSetup
 - o JunctionBox







JunctionBox folder consists of the Fortran codes that need to be compiled to generate an external library that will be linked to the CFX solver during executions in order to be able to read the set of grids generated by SCORGTM

Organize 🔻 🛛 Include in library	y ▼ Share with ▼ Burn New folder	= 🕇 🗍 🤅
🔆 Favorites	^ Name	Date modified
🧮 Desktop	Compile.txt	07/10/2011 17:56
〕 Downloads	mycal_easymap.F	22/08/2014 19:26
🖳 Recent Places	mycal_meshmap.F	22/08/2014 19:26
퉬 SkyDrive	set_mesh_user.F	22/08/2014 19:26
	upd_crdvx_user.F	22/08/2014 19:26
词 Libraries	update_crdvx_user.F	22/08/2014 19:26
Documents	update_mesh_user.F	22/08/2014 19:26
🚽 Music	📄 userpart.F	22/08/2014 19:26
Pictures		
Subversion	▼	

4 Compiler Environment setup [One time procedure]

Intel Fortran Compiler is required in the initial stages of the case setup. It is not required to repeat this step for every CFX case setup.

For Windows OS,

• Install Microsoft Visual Studio 2008 or any later version.





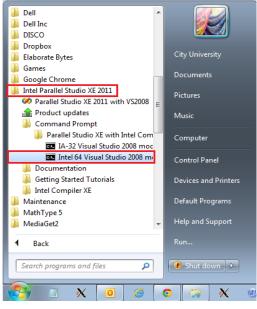
• Install Intel Parallel Studio XE 2011 for the Fortran compiler.

Below are some links that provide these installers (Evaluation Editions), but if you have other Fortran compiler then it can be used.

http://www.microsoft.com/en-gb/download/details.aspx?id=40787

https://software.intel.com/en-us/intel-parallel-studio-xe/

You can then access the Intel Fortran compiler command prompt as shown below. Try to run **ifort** command to check if it is recognized and properly works.



For Linux, check if either of F77, F90, G77, G90 or a Portland compiler is installed.

4.1 Compilation of Junction box Subroutines

Junction box subroutines are used by CFX solver to read the new mesh coordinates of the rotor domain every time step. The source code and compilation command used for this purpose is available in the [**TwinScrewCFXSetup\CFX\JunctionBox**] folder.

For Windows,

In order to create the library and link the object files compiled by Fortran compiler we will use the cfx command **'cfx5mkext'**. For this you need to first set the environment variable 'Path' and point to the CFX installation directory.

System Properties \rightarrow Advanced \rightarrow Environment Variables \rightarrow System Variables \rightarrow Path.

Add the path of [..*AnsysInc*v130CFXbin] to the variable separated by a semicolon from others.





System Properties					X				
Computer Name	Hardware	Advanced	System Protection	Remote					
Environment Va		niversity		(23	-			
Variable AWP_LOCAL FLUENT_INC path	LE130 er		s\ansys inc\v130\flu Sham_2011\Disco_t						
TEMP			LE%\AppData\Local		-	ſ	Edit System Variable		×
System variab	les	New	Edit	Delete			Variable name: Variable value:	Path C:\Program Files\ANSYS	Inc\v130\CFX\bin:
Variable Path PATHEXT	C		es (x86)\Intel\Comp					ОК	Cancel
PROCESSOR	R_A AI	MD64	AT;.CMD;.VBS;.VBE; 6 Model 42 Stepping		-		-		
		New	Edit	Delete					
_			ОК	Cancel					

Once the path is set, launch Intel Fortran Compiler command prompt. Change the working directory. Issue the command mentioned in **Compile.txt** file.

-double is a flag for double precision CFX simulation. You need to remove it for Single precision simulations.

-name is a flag for the routine object and is called by the solver.

► Twi	nScrewCFXSetup 🕨 CFX 🕨 Jui	nctionBox				
nt	Burn New folder					
	Name	Da	te modified	Туре	Size	
	Compile.txt	07/	/10/2011 17:56	Text Document	1 KB	
	🗎 mycal_easymap.F	22/	/08/2014 19:26	Fortran Source	3 KB	
	mycal_meshmap.F set_mesh_user.F	Administrator:	Intel Composer)	KE 2011 Intel(R) 64 Vis	ual Studio 2008	- • •
	upd_crdvx_user.F	C:\Program Fi	iles (x86)\I	ntel\Composer	XE 2011 SP1>D:	Â
	update_crdvx_user.F	D:\TwinScrew(FXSetup\CFX	\JunctionBox>c	fx5 &_	
	update_mesh_user.F userpart.F					





🛤 Administrator: Intel Co	mposer XE 2011 I	ntel(R) 64	Visua	al Studio 2008 - cfx5		
C:\Program Files (D:\TwinScrewCFXSet						▲ III
	ANSYS CFX-13 File Edit CFX	.0 Launch	ner (or Tools	n SHAM-PC) User Help ANSYS Client Licensing Utility	3.0	- • 🔀 »
						h.

Administrator: C:\Windows\system32\cmd.exe
Microsoft Windows [Version 6.1.7601] Copyright (c) 2009 Microsoft Corporation. All rights reserved.
D:\TwinScrewCFXSetup\CFX\JunctionBox>cfx5mkext -double -name meshread mycal_easy map.F mycal_meshmap.F set_mesh_user.F upd_crdvx_user.F update_crdvx_user.F updat e_mesh_user.F_

 Copy the created library "winnt-amd64" in the C:\windows folder for easy access and subsequent reuse





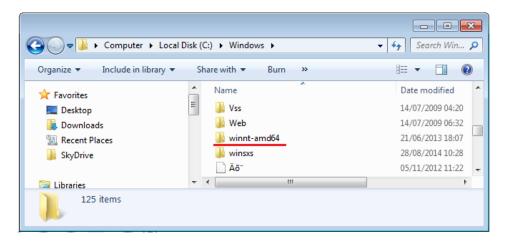
CFX > JunctionBox >	
Organize 👻 Include in library 🔻	Share with 🔻 🛛 Burn ᠉ 🔠 🔻 🗍 🔞
 ★ Favorites ■ Desktop Downloads ™ Recent Places SkyDrive Clibraries ■ Documents 	Name Name Compile.txt mycal_easymap.F mycal_meshmap.F set_mesh_user.F upd_crdvx_user.F update_crdvx_user.F
Music Pictures Subversion 9 items	update_mesh_user.F userpart.F ✓ ✓ ✓ ✓ ✓

For Linux,

Go to the working directory using Linux command prompt and issue the following commands.

- ▶ Use Intel Fortran Compiler 2015
- ▶ The procedure for compilation is same as for windows.
- A folder **linux-amd64** will be created when this step is successful.

It is not required to repeat this step for every CFX case setup and the folder **winnt-amd64/linux-amd64** can be just copied and used again in another case. But this library is specific to a given operating system and a given architecture. So in case you are running on a 32 bit OS or any other OS (Win 8) etc. you need to execute this step and provide the library so created to the CFX solver definition file. The library is common for Serial and Parallel Simulations.







5 CFX Pre case setup

Launch CFX from [**TwinScrewCFXSetup****CFX****CFXPreSetup**] folder

ANSYS CFX 2020	22 Launcher (on DESKTOP-M89HFJ0)	_		×
File Edit CFX Sh	ow Tools User Help			
PRE SI	и РST			
CFX-Pre CFX-Solver	Manager CFD-Post			
Working Directory	D:/TwinScrewCFXSetup/SCORG_Grid_Tutorial/Grid/Output/CFX/CFXPreSetup		~ 🖻	
				^

► Select New \rightarrow General

CFX-Pre	💼 New Case	
File Edit Session Insert Tools	Circulation Trans	. 0 d = 1: » (0 »
	General	
	Turbomachinery	
	Quick Setup	
	Library Template	
	OK Cancel	
	component	łł.

▶ Go to \rightarrow Session \rightarrow Play Session \rightarrow Select SingleRotorDomainImport.pre





PRE CFX-Pre: Unnamed		
File Edit Session Insert Tools Help		
	〕 ♪ / + ѷ 毋 沢 & ♦ ೫ ₪ 凾 ァ ┃ 0 ਗ਼ ■ 吽 ㅋ ѷ ㅋ ば い ⊵ 昴 ⊁ 喘 ▦ ⑮	: 📷 🔊 🔊
		3 🛄 🖂 🧠
Outline 🗋 New Session	PRE Play Session File	×
💙 🙆 M 🚨 Start Recording	They bession the	~
Stop Recording	Look in: 📙 D:\TwinScrewCFXSetup\SCORG_Grid_Tutorial\Grid\Output\CFX\CFXPreSetup 🔹 🔾 🔾 👔	爲 📰 🔳
V 😥 Flow Analysis 1	Documents Name Size Type Date Modified	
C Analysis Type	SingleRotorDomainImport pre 2 KB pre File 09/08/2019 09:14	
Interfaces	a shamr	
✓ I Solver		
a ^{ks} Solution Units		1
Solver Control		
User Locations		
Transformations		
A Materials		
Reactions		
 Keactions Keactions Expressions, Functions and Variations 		
Additional Variables		
🕅 User Functions		
Juser Routines		
✓ not simulation Control		
Configurations	File name: SingleRotorDomainImport.pre	Open
> 🙆 Case Options		
	Files of type: CFX-Pre Session Files (*.pre)	Cancel

- This imports the rotor grid in starting position and single domain rotor is created in the setup.
- Save case as Rotor.cfx

		🔄 🍾 💁 🗣 🍭 👩 🗆] 🗕 ?🗖	
Mesh i roa.1 Connectivity Simulation		View 1 🔻		
😰 Flow Analysis 1	Save Case			×
Analysis Type Image: Analysis Type Image: Type				_
✓ J‡ rotor Default		SCORG_Grid_Tutorial\Grid\Output\CFX\CFXPreSet	up 🔹 🔾 🔾 🙀 🖽	
 pil Interfaces ✓ Solver 	Documents Name	Size Type	e Date Modified	
 Solver solver solution Units 	a shamr			
Solver Control	·••			the second of th
A Output Control				and the state of t
Luser				and the second sec
Coordinate rrames User Locations				
Transformations				The manufacture of the second
🙆 Materials				and the second s
Reactions				1 A Manual 1
Expressions, Functions and Variables Additional Variables				
Expressions				
User Functions				
🗸 🙀 User Routines				
📫 Mesh Read				the stand of the s
Simulation Control Configurations				
Case Options	File name: Rotor.cfx		Save	
case options	Files of type: CFX-Pre Case Files (*.cfx)		- Cance	
				· · · · · · · · · · · · · · · · · · ·
				and the second
				3 - 32
				- Harrison market
				A start and a start and a start

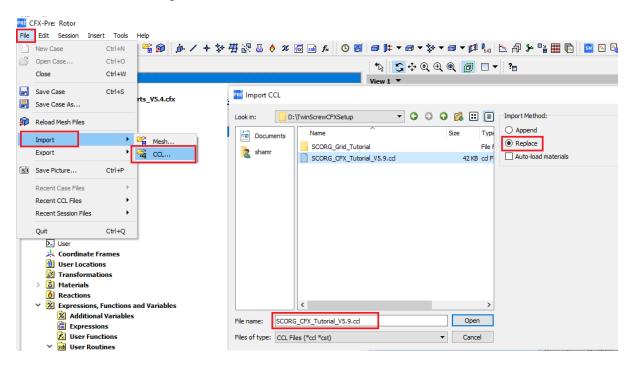
▶ Right click Mesh → Import Mesh → CFX Mesh → Select SCORG_CFX_Tutorial_Ports_V5.4.cfx





File Fi PRE CFX-Pre:								
File Edit S	Sessio			あくキシ留言	R 🕹 👌 🗴 🖟	sub f	: 0 3 6 # 6 * 6 # 4 * 6 * 8 * 8 * 8 * 8	
Outline							*\$ \$ \$ \$ \$ \$ \$ \$ \$ \$ \$	
✓ <u>© Mer</u> > <u>©</u>	e:	Import Mesh		CFX Mesh			View 1 🔻	
v 😧 Sim		View by		ANSYS Meshing	PRE Import I	Mesh		×
~ 🖻		Transform Mesh Define Connection Glue Adjacent Meshes		CFX-Solver Input ICEM CFD ANSYS FLUENT	Look in:	D:	TwinScrewCPXSetup V O O Ø Ø E E	
~	₽ ■	Reload Mesh Files Mesh Statistics	er:	CGNS	and the shame of t	r	SCORG_Crid_Tutorial SCORG_CFX_Tutorial_Ports_V5.4.cfx 1	
	_	Import Region Data Export Region Data						
× 🗠	*	Insert Render Show Hide	ble	·s				
~	ъ Ч	Delete All Mesh Expand Sub-Branches Collapse Sub-Branches					< >>	
	ulatio Confi	on Control gurations			File name: Files of type:	<u> </u>	S_CFX_Tutorial_Ports_V5.4.cfx Open esh ("gtm "cfx) Cancel Use settings next time	

▶ Go to File \rightarrow Import \rightarrow CCL \rightarrow Select SCORG_CFX_Tutorial_V5.9.ccl



Save case as Testrun1.cfx





🕒 Save Case			×
Look in:	🕕 D: \TwinScrewCFXS\CFX\CFXPreSetup 👻 🔇	00	📑 🖽 🗐
🔊 Му	Name Si	ze	Туре
Doc	Rotor.cfx SCORG(C)_CFX_Tutorial_Ports_V5.0.cfx		3 cfx File 3 cfx File
	۰ III		4
File name:	TestRun1.cfx		Save
Files of type:	CFX-Pre Case Files (*.cfx)	•	Cancel

▶ In the setup there are three non-conformal grid interfaces

- o Domain Interface 1
- o Domain Interface 2
- Domain Interface 3

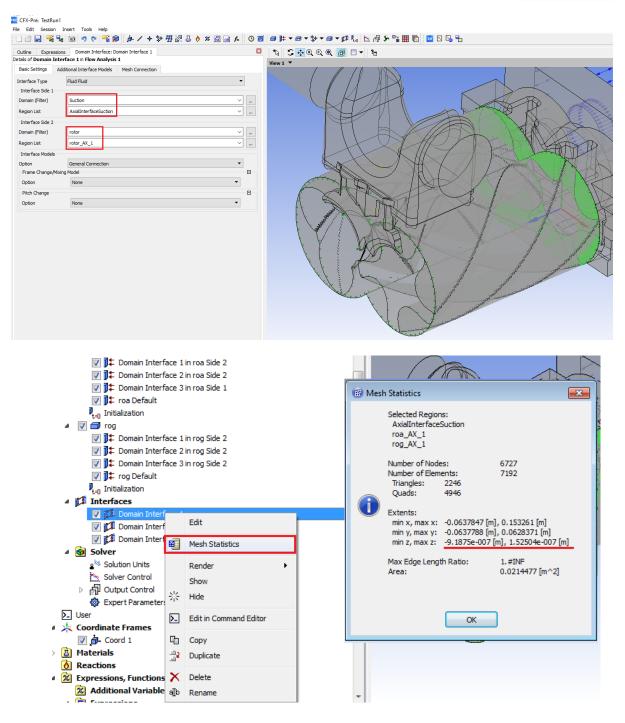
Interface 1 is between the Suction Port and the two rotors.

- Inspect the interface settings.
- Right click Domain Interface $1 \rightarrow$ Mesh Statistics \rightarrow min / max z should match so that there is no gap between the faces.



SCORGTM V5.9, 2021



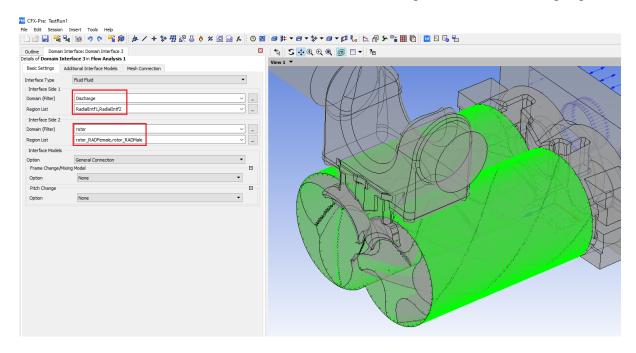


- **Interface 2** is between the Discharge Port and the two rotors.
 - Inspect the interface settings.
 - Right click Domain Interface $2 \rightarrow$ Mesh Statistics \rightarrow Min / Max Z should match so that there is no gap between the faces.





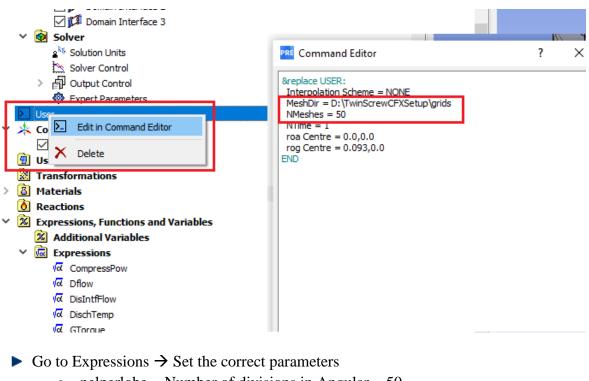
- ▶ Interface 3 is between the two rotors and also with the Discharge Port and a special zone selection is required in such type of domain connection.
 - Inspect the interface settings.
 - Notice that the face zones form a cross exchange of boundaries as highlighted.



▶ Go to User → Edit in Command Editor → Set the correct MeshDir, NMeshes and rog Centre.







- \circ nelperlobe = Number of divisions in Angular = 50
- \circ nlobe = Number of Lobes on Male Rotor = 3
- \circ revpermin = rpm of Male rotor = 8000

	YCL	Sucurraniow	ווומצאי וטיינר שטטוומוו בווגרוומנכ ב וווויטמ טוטכ בדווומ	
	(d	то	298.15 [K]	Ξ
	Val	nelperlobe	50	
	d	ngrids	nelperlobe*nlobe	
	Val	nlobe	3	
	Val	revpermin	8000 [rev/min]	
	(d	timestep	1.0 [rev] / (revpermin * ngrids)	-
•			4	
Detail	s of n	elperlobe		
Defi	inition	Plot Evaluate		
50				

Go to Mesh Read → Set correct Library Path → Start of time Step
 ○ Library Path is the folder where winnt-amd64 has been placed

yus i	nusanuu		
(a 1	MonitorABSP		
(<mark>ਕ</mark> (OUTLETMASSFLOWNEW		
<u>√ल</u> (OUTLETMASSFLOWOLD	Outline User Routin	e: Mesh Read 🛛 🔀
<u>(</u> त्र (OUTLETTEMPNEW I	Details of Mesh Read	
(<mark>ਕ</mark> (OUTLETTEMPOLD	Basic Settings	
Val F	PO		
va s	Sflow	Option	Junction Box Routine 👻
(d . 5	SuctIntfFlow	Callina Nama	update_mesh_user
(<mark>a</mark> 1	то	Calling Name	upuate_mesn_user
(od r	nelperlobe	Library Name	meshread
v <mark>a</mark> r	ngrids		
√ਕ ਾ	nlobe	Library Path	C:\Windows
(ਕ ਾ	revpermin	Junction Box Location	Start of Time Step
v <mark>a</mark> t	timestep	Juncuon box Location	Start of Hind Step
🐔 Use	er Functions		
⊿ 👊 Use	er Routines		
sub	Mesh Read		





- ► Move TestRun1.cfx and grids folder \rightarrow TwinScrewCFXSetup \rightarrow Open TestRun1.cfx
 - Only rotor.1, rotor.2 etc file from grids folder will be used, other files can be deleted.

📙 🛃 📕 🖛 TwinScrewCFXSetup			– 🗆 ×
File Home Share View			~ 🕐
\leftarrow \rightarrow \checkmark \uparrow \frown \rightarrow This PC \rightarrow Work (D:)	> TwinScrewCFXSetup >	· · · · ·	ල Search Tw , P
Name	Date modified	Туре	Size
	03/03/2021 11:51	File folder	
SCORG_Grid_Tutorial	03/03/2021 11:14	File folder	
35FemaleProfile_P2.dat	03/03/2021 11:05	DAT File	18 KB
35MaleProfile_P1.dat	03/03/2021 11:06	DAT File	23 KB
SCORG_CFX_Tutorial_Ports_V5.4.cfx	03/03/2021 11:44	ANSYS 2020 R2 .cf	18,696 KB
SCORG_CFX_Tutorial_V5.9.ccl	03/03/2021 12:01	CCL File	42 KB
CFX TestRun1.cfx	03/03/2021 12:07	ANSYS 2020 R2 .cf	34,858 KB

7 items | 1 item selected 34.0 MB

6 **CFX Solver Calculation**

▶ Click Define Run \rightarrow Save Definition file as TestRun1.def

5 🚾 sub f (0 3 - 1 ‡ • ∂ • o • 1 [‡] • ∂		
	▲ View 1 ▼		
🐵 Write Solver	Input File		—
Look in:	D:\TwinScrewCFXS\CFX\CFXPreSetu	p 🗸 🗘 💭 📑 🎛 🔳 quit CFX-Pre	
🔊 Му	Name	Size Type D	
Doc	📄 TestRun1.def	25.1 MB def File 3	
📕 Sham			
-			
	<	•	
File name:	TestRun1.def	Save	
Files of type:	CFX-Solver Input Files (*.def)	✓ Cancel	
			it.

▶ Select → Double Precision → HP MPI Local Parallel → Start Run



:::



💶 Define Run	? ×
Solver Input File D:\TwinScrewCFXSetup\TestRun1.def Global Run Settings	2
Run Definition Initial Values Partitioner Solver Interpolator	
Run Settings	
Type of Run Full	-
Double Precision	
Large Problem	
Parallel Environment	Ξ
Submission Type Direct Start	-
Run Mode Intel MPI Local Parallel	-
Host Name Partitions	
desktop-m89hfj0 10	+
	-
Run Environment	
Working Directory D:\TwinScrewCFXSetup	6
Show Advanced Controls	
Start Run	Cancel

In the Time stepping Information, mesh map generation and replacement with consecutive meshes should be reported

 +	Timestep	1	Courant Number Acoustic Courant Number					umber				
ļ		÷.	F	MS		MAX			RMS		MAX	
	5.0000E-05	i		0.10		2.75			724.04		999.99	
DOMA	IN: rotor							_				
File Numb DOMA File	ling mesh for e read: D:\Tw per of vertic AIN: rotor e read: D:\Tw per of vertic	vinSci es = vinSci	ewCE	XSet	up\gr 63 up\gr	6650						
				Me	sh St	atisti	lcs					
	omain Name		ļ	Orth	og. A	ngle	Exp	. Fac	tor	Asp	ect F	latio
			ļ	Mini	mum [deg]	м	aximu	um	+ 	Maxin	num
rc	rotor		1	14.0 !				99	990 ! 106			54 OK
Global		T	1.1 !				1314	3142 !		11204 ok		
			i	8!	%ok	%OK	% !	%ok	%OK	%!	%ok	%OK
	otor Lobal					90 91				0 0		100 100
Doma	Ain interface Non-overlap Non-overlap Ain interface Non-overlap Non-overlap Ain interface Non-overlap	area area : Dor area area : Dor	frac frac nain frac frac nain	tion tion Inte tion tion Inte	on s on s rface on s on s rface	ide 1 ide 2 2 ide 1 ide 2 3				+ = = = =	2. 4. 8.	48E-0 69E-0 13E-0 11E-0 .90E-0
	Non-overlap	area	frac	tion	on s	ide 2				=	9.	84E-0





6.1 Restarting a Simulation from an intermediate Stop

You can stop the simulation at an intermediate step by hitting the stop button in CFXSolver GUI or through command line. This will complete the current coefficient loop iteration and close the solver. Results file is written for the current completed time step.

To restart from the same flow time specify this result file as the initialization file and check on the continue history from initial file. This will continue the residual monitors from the same time steps and also the junction box routine will call the corresponding mesh file from the grids directory.

🔞 Define Run					8 23
Solver Input File				6	
Global Run Settings					,
Run Definition Par	titioner	Solver	Interpolato	r	
Initial Values Spe	cification				^
Initial Values		•			
Initial Values 1					
				*	
Initial Values 1 Set	tings				
File Name	M_MF	BC_wallVe	locity_003.re	s 🚰	Е
Continue Histor	y From				
Continue History Fro	om Initial	Values 1		•	
Use Mesh From	Solver	Input File		•	
Type of Run	Full			-	
Double Precision Parallel Environment					
Run Mode	HP MPI	Local Para	allel	•	
Host Name		Partitio	ns		
SHOPBUILD6059		2		+	-
Start Run				C	ancel





7 Distribution Options

There are three options available in the Type of Distribution (Rane, 2015)

a. Rotor to Casing

Generates a smooth rotor surface and node distribution on the rotor can be controlled using Distribution Adaptation factors. This gives good quality rotor profile in 3D.

- b. Casing to Rotor Nonconformal Generates an orthogonal cell structure with possibility to independently refine the interlobe leakage region. This gives better leakage predictions.
- c. Casing to Rotor Conformal Generates an orthogonal cell structure with a single domain containing both the rotors.

Distribution Parameters				
Type of Distribution	Rotor to Casing 🔹 🔻			
K Main	Rotor to Casing			
K Gate	Casing to Rotor Nonconformal Casing to Rotor Conformal			
Doole Creathing Easter	n o			

The selection of the type of distribution is dependent on the type of screw machine being solved and the CFD solver in consideration. For ANSYS CFX all the three options are available.

The preferred option is Casing to Rotor Conformal as this will generate a single block structured grid that has both the rotors and eliminates the interface between them

Refer to SCORG Help Manual for more details.

8 Summary

This document describes the steps to setup an ANSYS CFX model for Screw compressor CFD analysis starting from output data generated by SCORG[™] Meshing tool. More detailed information on using SCORG and Screw compressor mesh generation can be found in user guide (SCORG, 2021). As mentioned earlier the compilation of junction box routines is a onetime process but has to be done whenever the operating system or its architecture changes. 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. It is also possible to stop and restart the simulation in between, change certain Boundary conditions, Solver control parameters or save the intermediate results. More details information on using ANSYS CFX, Transient simulations and Post-Processing can be found in user guide (ANSYS CFX, 2021).





9 Bibliography

ANSYS CFX, 2021. User Guide, 2021-R1: ANSYS Inc..

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

