

PAPER • OPEN ACCESS

CFD Modelling of Coupled Heat Transfer between Solid and Fluid in a Twin Screw Compressor

To cite this article: Hui Ding *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **604** 012005

View the [article online](#) for updates and enhancements.

CFD Modelling of Coupled Heat Transfer between Solid and Fluid in a Twin Screw Compressor

Hui Ding, Yu Jiang, and Sujan Dhar

Simerics Inc.,
1750 112th Ave NE Ste C250,
Bellevue, WA 98004, USA

E-mail: hd@simerics.com

Abstract. Temperature variation in a compressor is very critical for compression performance and efficiency. To accurately predict the thermal dynamic behaviour of a compressor, heat transfer between metal components and the working fluid needs to be properly evaluated in CFD simulation. Thermal expansion of solid components due to temperature increase may also cause excessive abrasion of metal parts and changing in leakage clearance. Solving conjugate heat transfer (CHT) of twin screw compressors not only needs to deal with complex moving rotor geometry, but also needs to consider huge difference of temperature propagation time scale inside different media. Typically, heat propagation is much slower in solid than in compressed gas. Therefore, it may take many compressor revolutions to reach a stable CHT solution if heat conduction inside solid and thermodynamic temperature change of compressed gas are directly coupled in a CFD simulation. The run time of simulation may become impractical.

In this paper, an innovative approach solving conjugate heat transfer using the Mixed Timescale Coupling method between different media will be described in detail. This approach can address time scale difference in heat propagation. The proposed approach will be applied to a twin screw compressor. Simulation results will be compared with available test data. Effects of conjugate heat transfer will be evaluated by comparing the simulation results for the cases with and without considering conjugate heat transfer. Thermal expansion of the solid rotor will also be evaluated using predicted temperature distribution. Simulation will also demonstrate that the approaches used in the paper are robust, fast, and user friendly, and can be readily applied to industrial compressor systems.

1. Introduction

Gas compression process is almost always associated with significant gas temperature change. During an adiabatic process, the temperature of a diatomic ideal gas such as air will increase 32% in kelvin when gas volume compressed to half. Additional temperature raise normally indicates loss of efficiency. On the other hand, temperature control has also been used to help improve compression efficiency. For example, oil injection was commonly used in compressor not only help seal the leakage gap but also help reduce air temperature results in reduction of the power needed to compress the gas (Ding & Jiang [1]). For most of CFD studies of twin screw compressor conducted, only the fluid part were considered. In those simulations, it was commonly assumed that no heat exchange



between solid and the compressed gas, or in other words, adiabatic boundary condition was used at fluid-solid interface. Investigation is needed to make sure this assumption is valid. Furthermore, temperature change inside solid components will introduce thermal expansion and thermal stress. They may cause other issues such as excessive abrasion, change of leakage gaps, or thermal induced fatigue. Coupling of conduction in solids with the convective and radiative heat transfer in the surrounding fluids is often called Conjugate Heat Transfer (CHT).

Rowinski et al. [2] simulated a twin screw expander coupled with solid fluid heat exchange. They used a method called “semi-transient” to resolve the issue of different heat transfer time scale. However, in their study, modeling methodology are developed based on automatic regeneration of Cartesian type of mesh at each rotor position. This type of mesh has strong disadvantage in handling of narrow gaps, and the narrow leakage gaps are very important in positive displacement machines. Therefore, traditional moving/deforming structured mesh are still more commonly used for rotor mesh in screw machine simulations. For moving/deforming mesh approach, to create conjugate heat transfer interface between fluid and solid is not easy. Moreover, another important issues for any mesh solution is the difficulty in coupling of slow heat propagation in solids with very fast movement of fluid machine.

1.1. Coupling heat transfer between fluid and solid

We can use the solid rotors of a real twin screw compressor, the compressor to be used as the final test case in this study, to do a simple heat conduction simulation to demonstrate the time scale difference in heat transfer processes. Figure 1 shows the solid rotors of the compressor. More details about this compressor can be found in section 2. The rotors are made of stainless steel. The initial rotor temperature was set to 300K, and the rotor blades exposed to surrounding fluid with 400 K temperature. Let us assume a super high heat transfer coefficient of 1000 w/m²K on those interfaces. A quick simulation solving transient heat conduction inside rotor shows that after one minute of heat conduction, the average temperature of solid rotors is only raise from 300 K to 350 K. Figure 2 shows temperature distribution after one minute of heat conduction. The male rotor of this compressor runs at around 8000 rpm. Therefore, simulation needs to run tens or even hundreds of thousands of rotor revolutions to get close enough to final temperature of 400 K. The simulation run time is too long for practical use. It was also found from transient simulation that after 0.0075 seconds, the time needed for the male rotor to rotate one revolution at 8000 rpm, the solid surface temperature only changed about 1 K. Those facts form the bases of our proposed Mixed Timescale Coupling method.

This simple test case proves that solving directly coupled conjugate heat transfer between fluid and solid for screw compressor is not affordable. The Mixed Timescale Coupling is then proposed in this study which needs much less time to reach stable solution, but can still properly capture important effect of the conjugate heat transfer.

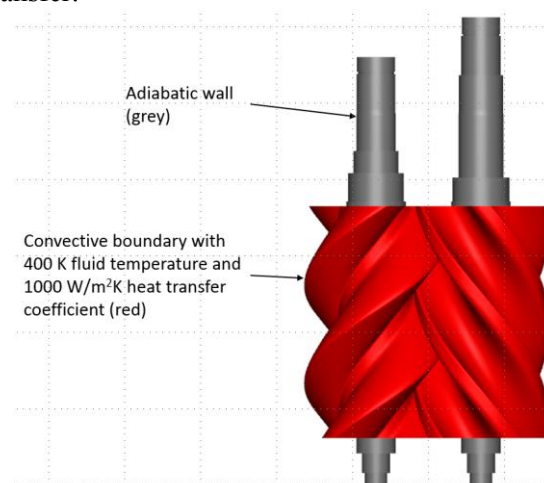


Figure 1. Rotor transient heat conduction model

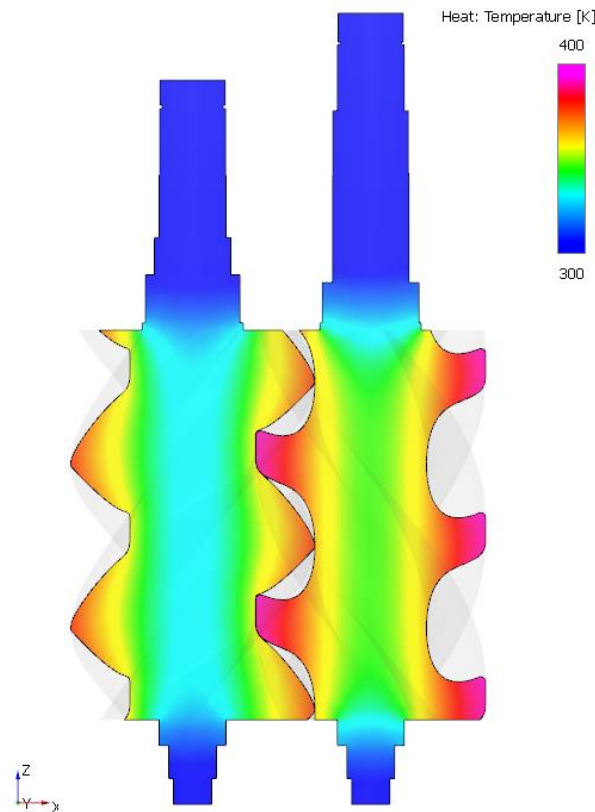


Figure 2: temperature inside rotor after 1 minutes of heat transfer from fluid interface

The proposed coupling method in this study is similar to the one used by Dhar et al. [3] for piston cooling simulation. In this approach, fluid and solid are solved in a sequence of separate simulations with heat exchanged at common interfaces as boundary conditions for each simulation. On fluid side, interface boundary was set with given temperature mapped from solid simulation results. On the solid side, the interface was set with given heat flux mapped from cycle averaged fluid solutions. The iterative process start with transient simulation for fluid part with constant initial temperature set at fluid/solid interface; after finished one cycle of fluid simulation, interface heat flux will be mapped to solid model; a steady state heat conduction will be solved for solid parts; interface temperature obtained from solid simulation will be mapped back to fluid volume as boundary condition; fluid simulation will run another cycle with updated interface temperature; this iterative process will continue until interface heat flux and temperature become stabilized. One of the assumptions for this method is that the solid temperature does not change much with in a single cycle of rotor rotation. This assumption is reasonable as demonstrated by afore mentioned transient rotor heat conduction simulation.

The mapping of boundary values are easily realized using built in functions of the CFD package used in the current study. The simulation iterations run automatically through the control of a script.

1.2. Conservation equations solving flow and heat transfer

The CFD package, Simerics-MP+, used in this study solves conservation equations of mass, momentum, and energy of a compressible fluid using a finite volume approach. Those conservation laws can be written in integral representation as

$$\frac{\partial}{\partial t} \int_{\Omega(\tau)} \rho d\Omega + \int_{\sigma} \rho(\mathbf{v} - \mathbf{v}_{\sigma}) \cdot \mathbf{n} d\sigma = 0 \quad (1)$$

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho \mathbf{v} d\Omega + \int_{\sigma} \rho ((\mathbf{v} - \mathbf{v}_{\sigma}) \cdot \mathbf{n}) \mathbf{v} d\sigma = \int_{\sigma} \tilde{\boldsymbol{\tau}} \cdot \mathbf{n} d\sigma - \int_{\sigma} p \mathbf{n} d\sigma + \int_{\Omega} \mathbf{f} d\Omega \quad (2)$$

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho E d\Omega + \int_{\sigma} \rho ((\mathbf{v} - \mathbf{v}_{\sigma}) \cdot \mathbf{n}) E d\sigma = \int_{\sigma} k \nabla T \cdot \mathbf{n} d\sigma - \int_{\sigma} p \mathbf{v} \cdot \mathbf{n} d\sigma + \int_{\sigma} (\mathbf{v} \cdot \tilde{\boldsymbol{\tau}}) \cdot \mathbf{n} d\sigma + \int_{\Omega} \mathbf{f} \cdot \mathbf{v} d\Omega \quad (3)$$

The standard $k - \varepsilon$ two-equation model [4] with wall function is used to account for turbulence,

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho k d\Omega + \int_{\sigma} \rho ((\mathbf{v} - \mathbf{v}_{\sigma}) \cdot \mathbf{n}) k d\sigma = \int_{\sigma} \left(\mu + \frac{\mu_t}{\sigma_k} \right) (\nabla k \cdot \mathbf{n}) d\sigma + \int_{\Omega} (G_t - \rho \varepsilon) d\Omega \quad (4)$$

$$\frac{\partial}{\partial t} \int_{\Omega(t)} \rho \varepsilon d\Omega + \int_{\sigma} \rho ((\mathbf{v} - \mathbf{v}_{\sigma}) \cdot \mathbf{n}) \varepsilon d\sigma = \int_{\sigma} \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) (\nabla \varepsilon \cdot \mathbf{n}) d\sigma + \int_{\Omega} (c_1 G_t \frac{\varepsilon}{k} - c_2 \rho \frac{\varepsilon^2}{k}) d\Omega \quad (5)$$

Together with equation of state, where properties are functions of temperature and pressure, to form a closed system:

$$\rho = f(p, T) \quad (6)$$

In the solver, each of the fluid properties will be a function of local pressure and temperature, and can be prescribed as an analytical formula or in a table format.

Heat conduction in solids is also solved using energy conservation equation (3).

1.3. Equations predicting thermal stress

The current CFD package also has a built in solver for solid strain-stress analysis. The base governing equation used to model structural problems is the equation of equilibrium of forces shown below,

$$\frac{\partial^2 (\rho \mathbf{u})}{\partial t^2} + \nabla \cdot \boldsymbol{\sigma} + \rho \mathbf{f} = 0 \quad (7)$$

Where \mathbf{u} is displacement, \mathbf{f} is force, and $\boldsymbol{\sigma}$ is stress. For the present work, a linearized strain definition along with linear elastic assumption was used. The governing equation for displacement solved is shown below,

$$\frac{\partial^2 (\rho \mathbf{u})}{\partial t^2} + \nabla \cdot \left[\xi (\nabla \mathbf{u}) + \xi (\nabla \mathbf{u})^T + \lambda \text{Tr}(\nabla \mathbf{u}) \mathbf{I} \right] = -\rho \mathbf{f} \quad (8)$$

2. Twin screw compressor test case

The compressor model used in this study was designed as an oil-free twin screw compressor with a 3/5 lobe arrangement and 'N' rotor profile rotors [5]. The operating speed on the male rotor varies from 6000 to 14000 rpm. The male rotor diameter is 127.45 mm; the female rotor diameter is 120.02 mm while the center distance between the two rotors is 93.00 mm. The length to diameter ratio of the rotors is 1.6 and the male rotor has a wrap angle of 285.0 deg. There are two independent models used in simulation: fluid model and solid model. And they are coupled together in an iterative process to solve conjugate heat transfer between the two models.

2.1. Fluid model

In the fluid model, the rotor part of the twin screw was meshed using a grid generation software SCORG [5]. SCORG creates a series of mesh files for the rotor at different rotation angles. The rotor mesh files were read into the solver through Simerics - SCORG mesh interface. The inlet and outlet ports of fluid volumes are meshed using Simerics binary tree unstructured mesher. All the fluid volumes are connected together using Mis-matched Grid Interface (MGI). The total number of cells for fluid volume is around 1.45 million. Mesh for fluid volumes are shown in Figure 5 (a).

The gas inlet is set to a fixed total pressure, and a fixed total temperature boundary condition while the outlet is set to a fixed static pressure boundary condition. Fluid-solid interface is set to a fixed

temperature boundary with temperature values mapped from solid model simulation results. The simulated fluid is air, modelled using ideal gas law. The male rotor rotation speed is 8000 RPM. In order to demonstrate the effects of conjugate heat transfer between fluid and solid, a similar case with the same parameters but only solve fluid volume with adiabatic wall at fluid-solid interface was also simulated for comparison. Simulation time is about 1 hour per male rotor revolution for fluid model on a 20 core workstation with two E5-2630 v4 CPU at 2.20 GHz.

2.2. Solid model

The solid model has 3 volumes: case, male rotor, and female rotor as shown in Figure 4. Solid volumes are all meshed using binary tree mesh with total about 0.4 million cells. Mesh for solid volumes are shown in Figure 5 (b). Solid-fluid interface is set as fixed heat flux boundary with values mapped from fluid model simulation results. The case outer surface is set as a heat convection boundary. Solid model solves steady state heat conduction. Simulation time for each run is negligible compared with simulation time for fluid model.

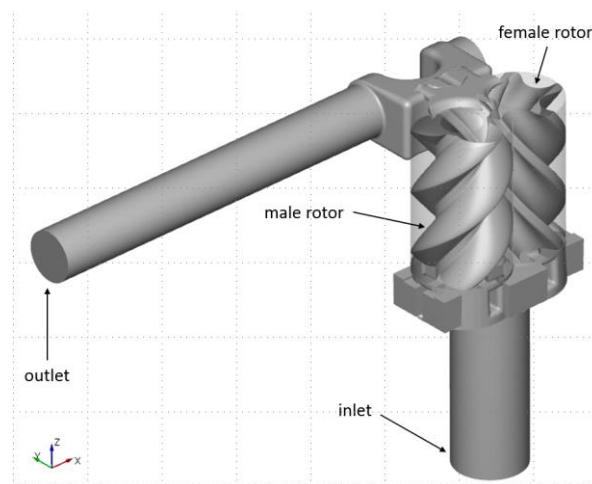


Figure 3: fluid volumes

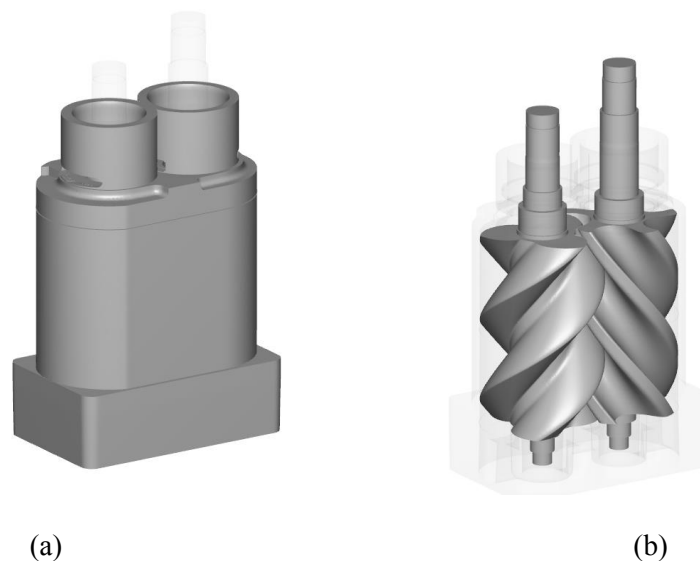


Figure 4: Solid volumes: (a) case (b) rotors

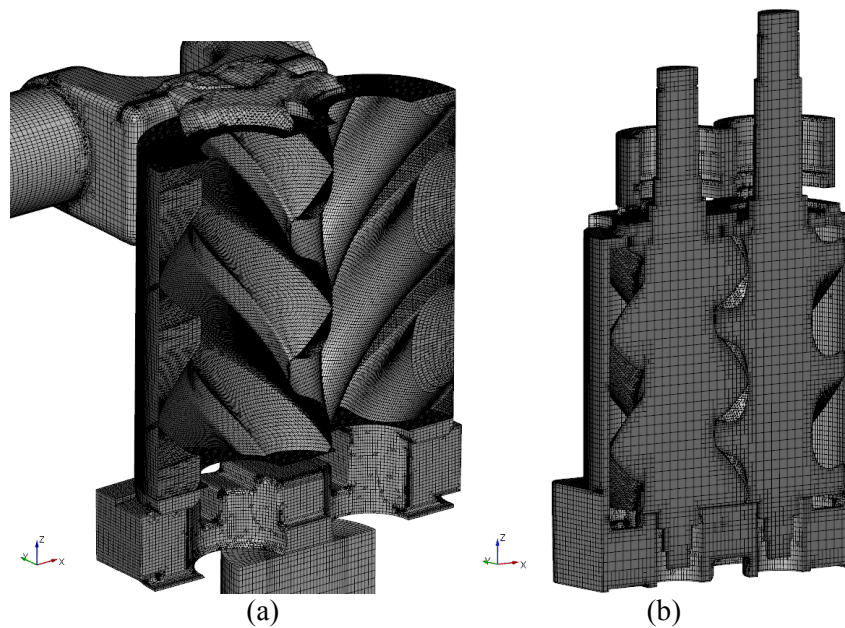


Figure 5: Mesh: (a) fluid volumes (b) solid volumes

3. Results and discussion

In the simulation, the fluid inlet total pressure and the outlet static pressure are set to 1 bar and 2 bar respectively. The inlet total temperature is set to 300K. Both rotors are assigned with correct rotational speed. Table 1 summarizes simulation parameters used for the conjugate heat transfer simulation. Other boundaries of the solid rotor are assumed fully insulated. The out surfaces of solid case is assumed to have a 10 W/m²K heat convection with 300 K environment temperature. The initial temperature for both fluid and solid are all set to 300 K.

Table 1: Simulation parameters

Parameters	Values
Gas	Air (using ideal gas law)
Gas inlet total pressure	1 bar absolute
Gas inlet total temperature	300 K
Outlet static pressure	2 bar absolute
Solid	Stainless steel
Solid density	7800 kg/m ³
Solid conductivity	30 W/mK
Solid heat capacity	450 J/KgK
Solid Young's Modulus	180 GPa
Solid Poisson Ratio	0.305
Solid thermal coefficient	1.6 x 10 ⁻⁵ m/mK
Compressor speed	8000 rpm (male rotor)

Simulation takes about 1 hour per revolution on a 20 core workstation with two E5-2630 v4 CPU at 2.20 GHz. During the simulation, results start to become periodic after about 5 male rotor revolutions. Figure 6 shows instantaneous and cycle averaged heat flux between fluid and rotors. The maximum instantaneous heat flux between fluid and solid at rotor interface is about 400 w. The average heat carried away from case outer surface is about 100 w.



Figure 6. Heat flux convergence

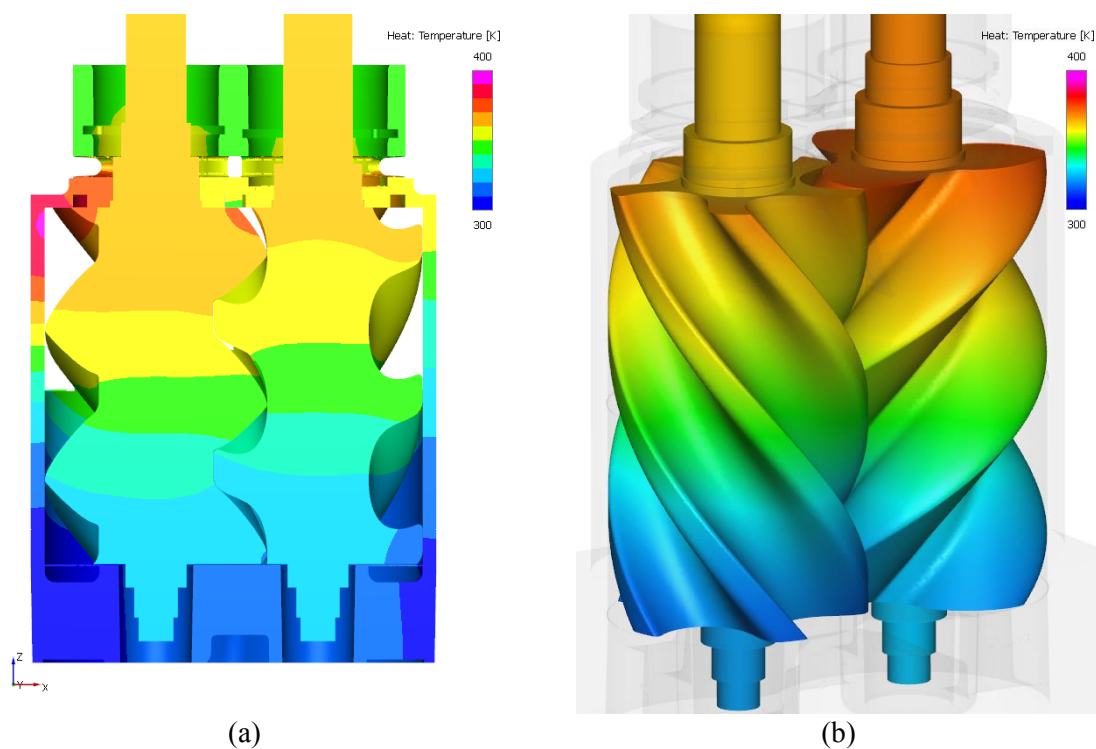


Figure 7. Temperature distribution in solids: (a) cutting plane (b) rotors

The final averaged solid temperatures are 345.3 K, 349.0 K, and 329.6 for male rotor, female rotor, and case respectively. Figure 7(a) shows solid temperature distribution in a cutting plane. Figure 7(b) shows rotor surface temperature. The colour map ranges from 300 K to 400 K with magenta represents high temperature and blue represents low temperature. The temperature inside solid shows a layered distribution with temperature goes from low to high when moving from inlet to discharge. As a comparison, a fluid solution assuming adiabatic wall at fluid-solid interface was also carried out. In this case, interface temperature keeps changing at different crankshaft angles. Figure 8 shows

temperature distribution of the rotors at 5 different male rotor crankshaft angles. The instantaneous temperature distribution no longer has layered pattern. Instead, temperature has similar values in each fluid “pocket”. Also, the temperature range is significantly higher. This means due to the huge heat inertia of metal, rotor surface temperature is actually milder, more uniform, and has a layered distribution than one can get from adiabatic wall assumption.

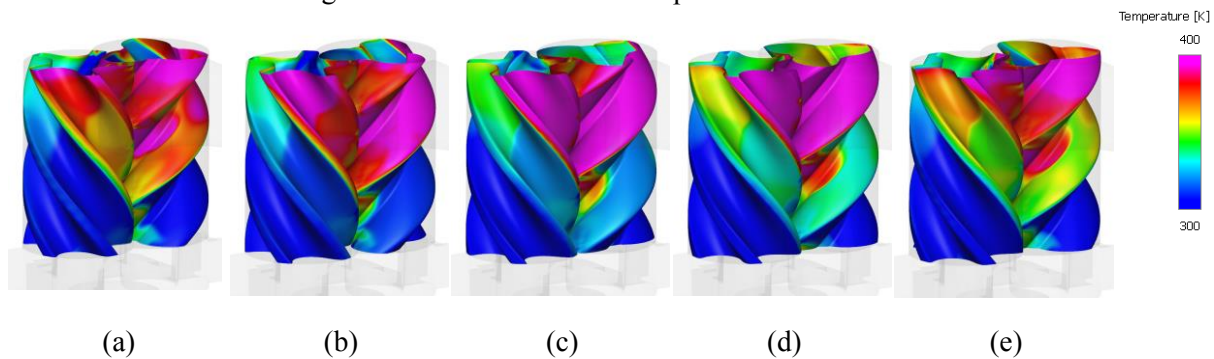


Figure 8: Rotor surface temperature without conjugate heat transfer at different male rotor crankshaft angles: (a) 24 degree (b) 48 degree (c) 72 degree (d) 96 degree (e) 120 degree

Figure 9 shows pressure contour of rotors at 5 crankshaft angles. The colour map ranges from 1 bar to 2.5 bar with magenta represents high pressure and blue represents low pressure for the case with CHT. The pressure at each fluid pocket has similar values as expected. And pressure increases when the “pocket” moves from inlet to discharge due to gradual reduction of fluid volume. Unlike temperature distribution, the pressure distribution on rotor surfaces are almost identical for the cases with/without CHT. This implies the influence of CHT on the compressor performance could be minimal.

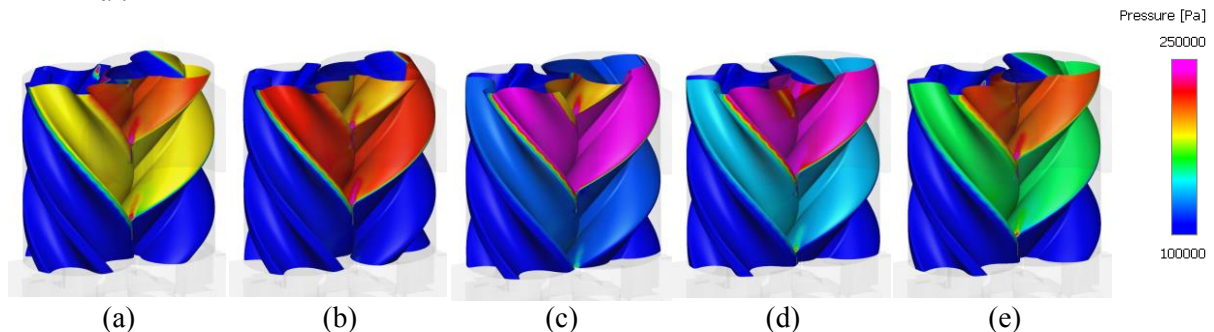


Figure 9: Pressure contour at different male rotor crankshaft angles for simulation with CHT: (a) 24 degree (b) 48 degree (c) 72 degree (d) 96 degree (e) 120 degree

Table 2 compares gas mass flow rate and rotor power for cases with and without considering CHT. From the table, the difference in predictions of flow rate and rotor power are less than 1%. When comparing with experiment, both results over predict the flow rate by about 4-5%. This discrepancy may be introduced by the inaccuracy of clearance size [5]. Power predictions are about 1% difference with experiment. For this particular case, conjugate heat transfer only has minimum influence on compressor performance, and therefore simulation without considering conjugate heat transfer is acceptable.

Table 2: Mass and energy balance

	Without CHT	With CHT	Experiment
Gas mass flow rate (kg/min)	12.4	12.3	11.8
Rotor power (kw)	22.0	21.9	22.2

Based on solid temperature simulation results, solid thermal stress/expansion was predicted using a strain-stress solver built in the used CFD package. Figure 10 plots rotor solids displacement due to thermal expansion in radial direction. The colour map ranges from 0 to 50 micron with magenta represents high displacement and blue represents low displacement. The maximum displacement in radial direction is about 50 microns. Please note that currently thermal expansion is a one way coupled prediction. The thermal expansion results were not fed back to fluid simulation.

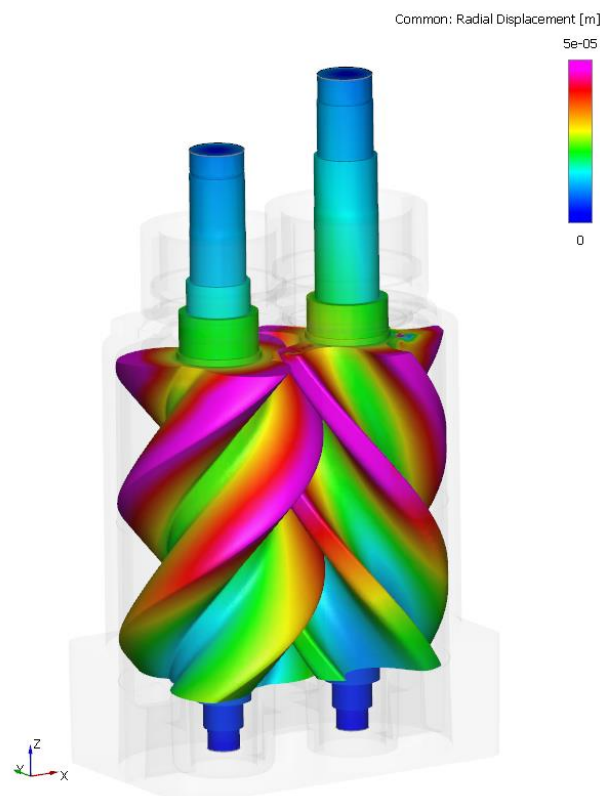


Figure 10. Thermal expansion in radial direction

4. Conclusion and Future Works

An innovative Mixed Timescale Coupling method solving conjugate heat transfer has been proposed and described in detail. The new approach has been successfully applied to the modelling of a twin screw compressor. This approach properly resolved time scale difference issue. Simulation run time is at the same level as simulation without CHT. In the current approach, CHT simulations solves fluid and solid in a sequence of separated models. With the help of built in capability of used CFD package, setup of such simulation is very straightforward. The simulation results match very well with experiment. For the particular test case, simulation results show that the influence of CHT on compressor performance is small, and therefore simulation without considering CHT is acceptable. CHT simulation also predict temperature distribution inside solid parts. Thermal stress and thermal expansion of solid parts are predicted using temperature distribution simulation results in a one way coupled fashion. All the simulation functionalities used in this project are built in the current CFD package.

In the future, CHT simulation will be integrated into a single model to further reduce user setup effort. Predicted thermal expansion will be further coupled with geometry change in fluid simulation

to improve simulation accuracy by automatically take into account all the changes introduced by CHT during operation.

Acknowledgement

The authors would like to express our gratitude to Dr. Kovacevic and Dr. Rane of City University London for letting us use their twin screw rotor geometry and mesh in this study.

References

- [1] Ding, H. and Jiang, Y., 2017, "CFD simulation of a screw compressor with oil injection, " IOP Conference Series: Materials Science and Engineering, Volume 232, conference 1.
- [2] David Rowinski, Alexander Nikolov, Andreas Brümmer, 2018, "Modeling a dry running twin-screw expander using a coupled thermal-fluid solver with automatic mesh generation," IOP Conference Series Materials Science and Engineering 425:012019.
- [3] Dhar, S., Godavarthi, R., Mishra, A., Bedekar, S., and Ranganathan, R., 2019, "A Transient, 3-Dimensional Multiphase CFD/Heat Transfer and Experimental Study of Oil Jet Cooled Engine Pistons," , SAE conference 2019.
- [4] Launder, B.E., and Spalding, D.B., 1974, The numerical computation of turbulent flows, *Comput. Methods Appl. Mech. Eng.*, 3, pp. 269-289.
- [5] Kovacevic, A., Rane, S., Stosic, N., Jiang, Y., and Lowry, S., 2014, Influence of approaches in CFD Solvers on Performance Prediction in Screw Compressors, *Int. Compressor Engrg. Conf. at Purdue*.