Modeling a dry running twin-screw expander using a coupled thermal-fluid solver with automatic mesh generation

David Rowinski*, Alexander Nikolov², Andreas Brümmer²
1. Convergent Science, Inc., Madison, WI, USA
2. Chair of Fluidics, TU-Dortmund University, Dortmund, Germany

* Corresponding Author. Email address: david.rowinski@convergecfd.com

Abstract. Understanding the details of the internal flow processes in screw compressors and expanders is very important for their efficient and robust design. Computational fluid dynamics (CFD) provides full access to a modeled three dimensional flow field and its variation in time. However, the application of CFD to screw compressors and expanders can be difficult because of the complicated geometries involved and the need to supply the computational grid on which the modeled equations are solved over a large number of time steps. While the majority of previous research on CFD applications to screw machines features inventive techniques for generating meshes that adequately resolve the flows in the small clearances, an alternate approach is demonstrated in this work. The screw expander SE 51.2 from TU Dortmund University is analyzed here through a CFD model which generates the grid automatically based on a modified Cartesian cut-cell approach. The grid is then adaptively refined based on local gradients of velocity and temperature. At each time step, the grid is regenerated based on the geometry motion. As opposed to resolving the flow in the clearances, a model is applied so that the cells in the clearance can remain relatively large. The detailed measurements of the screw expander are used to validate the model. The operating conditions investigated include the expansion of dry air at a four-to-one pressure ratio for four different rotational speeds. The measured internal chamber pressures are compared to the results from the model, as are the average mass flow rate, indicated power, and outlet temperature. A coupled thermal-fluid approach is used to model the rotor temperatures and corresponding thermal deformation of the rotors and housing. In this approach, the fluid and solid temperatures are solved together; to deal with the problem of the disparate time scales between the fluid and solid heat transfer, the solids are periodically solved to steady state using heat transfer coefficients and near-wall temperatures computed from an energy conserving averaging over several cycles. The effects of the various leakage flow paths in the model, including the rotor-to-rotor, rotor-tip-to-housing, and bearing leakage are demonstrated and quantified. Finally, simulation and experimental results are compared in terms of different rotor-tip-to-housing clearance heights. The model considering the appropriate thermal deformation of the rotors is shown to yield the best agreement with the measurements, however there is work remaining to reduce the model calculation time, especially at low rotational speeds.

1. Introduction
Screw expanders are mechanical devices used for extracting energy from waste heat in organic Rankine cycle and trilateral flash cycle systems. As energy demands around the world grow and regulations continue to drive energy generating devices to higher efficiencies, the importance of devices such as
screw expanders for energy recovery continues to grow [1]. In order to design these machines for optimal efficiency and extract as much energy from the flow as possible, it is important to understand the nature of the flows and how they interact with the machine geometry. Mathematical models of these processes are invaluable in both optimization and analysis.

There are several different levels of fidelity at which the flow and thermodynamics in these devices can be modeled, and as the fidelity of the models increase so does the cost of each method. Chamber models [2], [3], [4], [5], also developed for the screw compressor and vacuum pump counterparts to screw expanders, are the most compact of these, and offer a very low computational cost while providing a modeled solution for the flow and thermodynamic state variables for individual homogeneous chambers. Advancements in modeling efforts have evolved some of these approaches to become even more sophisticated, including multi-phase effects [6], thermal and mechanical deformation [7], and advanced analytical models for the leakage flows [8]. Overall, though the chamber models are simplified, the cost is low enough to allow many design iterations for practical optimization use.

On the other hand, the most detailed of the modeling approaches for screw expanders is the use of computational fluid dynamics (CFD) to solve for the complete flow path in space and time using a set of modeled equations. While this approach is inherently more computationally expensive than the chamber models, providing access to a modeled three-dimensional transient flow path gives the designers and analysts the most detailed information about the flow. The primary challenge in applying CFD to screw machines is the construction of the numerical grid on which the model equations are solved. The complex rotor geometries, complex sealing lines and clearances, and the impact these have on the device performance make the direct application of CFD difficult.

The majority of CFD modeling studies on screw compressors, vacuum pumps, and expanders has been based on analytical grid generation methods, including the works in [9], [10], and [11] for screw compressors and in [12], [13], and [14] for screw expanders. These methods have been successful in computing the solutions to the modeled equations and have been validated in a wide variety of screw machines. However, these methods are not without their own limitations, some of which include a restriction on the geometry of the rotors and housing, relatively large computational cost, and numerical diffusion from moving meshes. In this study, an alternate approach for generating the grid and solving the modeled equations is used.

This study uses a body-fitted Cartesian cut-cell grid to enclose the fluid volume and provide the finite volume cells on which the modeled equations are solved as demonstrated in [15]. Adaptive mesh refinement (AMR) is used to increase grid refinement at points in time and space where the gradients in velocity and temperature are large. As opposed to resolving the clearance flow, as is done in many of the aforementioned CFD studies, the clearance flow is modeled here. This permits the cells in the clearances to be relatively large. All together, these features result in a CFD model which is easily applied to arbitrary moving geometries, constructs the mesh automatically at every time step with little user effort, and computes the solutions with high stability and accuracy due to the interior grid being composed of stationary orthogonal hexahedral elements.

2. Modeling methodology

The modeling process begins with the creation of the discretized mesh where the modeled equations are solved, the description of the computational domain and boundary conditions, and the specification of the modeled equations. In this section, these processes are described in more detail.

2.1. Grid discretization

The primary feature which makes this work different from the majority of previous modeling studies of screw compressors or expanders is the way in which the numerical grid is created [16]. The procedure begins from computer-aided design (CAD) software, where a geometry of the fluid domain is created. The surface surrounding this fluid volume is discretized into a triangulated surface which is a direct import for the modeled geometry. Each of the discretized triangles is then associated to a boundary condition for the flow model.
The actual volume mesh is created at run time by the solver and is illustrated in figure 1. At every time step, an orthogonal hexahedral mesh is overlaid on the triangulated surface. The cells that lie entirely inside the surface are identified as interior cells. The cells which are intersected by the bounding surface are cut into arbitrary-sided polyhedral cells. These cells are identified as boundary cells. Every boundary cell retains exactly the geometry it has when cut by the bounding surface regardless of the cell size. It is important to point out that while the triangulated surface will determine the shape of the computational domain, it is the grid cells and not the triangles where the equations are being solved. It is also important to point out that the geometrical features of the computational domain are determined solely by the triangulated surface and not the grid; since the boundary cells are pure cuts, they always retain the same volume regardless of their size. In figure 1, the grid of the screw expander model is shown from several directions.

![Figure 1](image)

**Figure 1.** Demonstration of the grid methodology used in the CFD model. On the top left and right are cut-planes through the rotors colored by velocity and displaying the mesh lines. On the bottom left is another cut-plane, with a detail of the boxed region showing the adaptive mesh refinement resolving the velocity jet at the inlet port on the bottom right.

Any boundary in the surface may move arbitrarily so long as it does intersect another boundary. At every time step, the triangles that move are swept through the boundary cells and that swept volume along with the boundary flux is calculated and integrated into the modeled equation solution. Thus, the solution of the equations and grid generation are tightly coupled. Cells may be refined based on predefined volumes or adaptively based on local gradients as in [17] down to a user-specified size. In the close-up in figure 1, the additional refinement at the high-velocity jet in the inlet port is shown. At every cell, the gradients are evaluated and if they exceed a user-specified threshold, the cells are refined in all three directions.
2.2. Modeled equations
The flow of a compressible Newtonian fluid is considered in this study. Conservation of mass, momentum, and total energy along with an equation of state relate the six flow variables of pressure, density, temperature, and velocity vector field. An unsteady Reynolds-Averaged Navier Stokes (RANS) turbulence model is used to account for the effects of turbulence on the velocity field; in this case, the Renormalization Group (RNG) k-epsilon turbulence model [18] is employed, which requires the solution of two extra fields, the modeled turbulent kinetic energy and turbulent dissipation rate.

The equations are solved with Pressure-Implicit Splitting of Operators (PISO) pressure-velocity coupling [19], along with a bi-conditioned conjugate gradient solution for pressure and successive over relaxation (SOR) iterative solution procedure for the remaining equations. The method is formulated in the finite volume form, where each cell (including both the interior cells and boundary cut cells) is treated as a finite volume, and all quantities are computed and stored at the cell centers. The scheme of Rhie-Chow [20] is used to prevent checker-boarding on this collocated grid.

2.3. Conjugate heat transfer
Many previous studies on similar cases have shown that the wall temperatures, both on the fluid and the solid, have important consequences [10], [11], [12], [14] in terms of the rotor mechanics, the fluid heat transfer to the wall, and the clearance flow characteristics. In this study, the solid domain is considered along with the fluid domain for the purposes of calculating a modeled temperature for the solid walls. The combined domain is shown in figure 2 and includes the fluid, the solid housing, and the solid rotors. In the solid, a single heat conduction equation is solved.

Figure 2. Views of a grid used in the coupled fluid-solid model. On the top and middle rows are views of the mesh through the rotor axes. On the bottom left is a view in the transverse plane across the rotors.
and on the bottom right is a zoomed in view of one of the clearances showing a location where the clearance flow is modeled. All planes are colored by region: blue (dark) is the fluid region, green (medium dark) is the joined housing elements, yellow (medium light) is the male rotor, and pink (light) is the female rotor.

At the interfaces between the solid and the fluid, both the heat flux and the temperature are coupled. At every time step, both the fluid and the solid are solved in a time-accurate fashion. An inherent difficulty with this type of problem is the disparate time scales between the fluid and the solid; while the heat transfer processes in the fluid occur on very small time scales, the time scales associated with the conduction in the solid are generally much longer. Furthermore, since the temperature fields of the fluid and solid are coupled and the flow is linked through the density, both the solid and the fluid evolution will affect one another.

To remedy this problem, a technique referred to as super-cycling is used. With super-cycling, the near-wall temperatures and heat transfer coefficients are averaged over several cycles. Once every lobe passing time, the solid is solved completely to steady state using the thermal information from the fluid over several cycles. This process is completely automated in the solver, but it does require running for several cycles since the solution of both solid and fluid affect one another. One component missing from the model is directly including the effect of the thermal deformation in the same calculation. At the conclusion of the super-cycling simulation, the rotor profiles are scaled from their axes of rotation using the computed solid temperatures, and an additional calculation proceeds using this altered rotor geometry.

2.4. Modeling of clearance flows
The mesh shown in figure 2, in particular the detail in the lower right, demonstrates that the clearance flow is not fully resolved on the grid sizes considered. In order to account of the effects of the mesh being too coarse to fully resolve the clearance flow, a simplified calibrated model [15] is used. This model relates a clearance Reynolds number to an additional dissipative term in the momentum equation to correctly capture the mass flow through the clearance. The total mass flow rate through each clearance depends on the local conditions, including the actual clearance and the grid resolution, but generally this simplified model reduces the flow between 0-20% from the under-resolved model.

![Figure 3](image_url)

Figure 3. To the left is the fluid region from the SE 51.2 screw expander CAD model. On the right are the male and female rotors; to the far right, the triangles representing the discretized surface of the rotors are overlaid.
3. Test case description
The screw expander studied in this work is SE 51.2 designed at TU Dortmund University. SE 51.2, originally based on a similar twin screw supercharger, features a relatively small size including a displacement volume of 285 cc per male rotor revolution, a rotor center distance of 51 mm, and a rotor length of 101 mm. The built-in volume ratio is 2.5, though it is designed for convenient modification through the use of interchangeable control edge plates. The expander can handle pressure ratios between the inlet and outlet as large as 6:1 and speeds as high as 20,000 based on the male rotor. The performance of the expander has been extensively characterized experimentally in [21] over a wide variety of operating conditions, and more detailed experimental measurements have been performed in [22] and [23], including both dry running and water injected conditions.

The flow path of the twin screw expander and detail of the rotors is shown in figure 3. High pressure air enters from the top left, is drawn down into the expanding rotor chambers through the inlet port as the rotors counter-rotate. Once the working chamber is disconnected from the inlet port after the rotor flanks have rotated beyond the inlet control edges, the chamber is sealed and the internal expansion processes. Around the point where the maximum chamber volume is reached, the rotors continue to rotate exposing the control edges of the outlet port, at which point the outflow from the chambers begins. The rotation of the rotors reduces the volume of the expanded fluid and it is pushed out of the outlet port. The expander SE 51.2 is a gearless expander, so the counter-rotation of the rotors occurs with rotor contact.

In this investigation, only dry-running operating conditions are considered as have been the focus of previous CFD studies of this device in [12] and [14]. For the conditions investigated in this work, air enters at a pressure of 4 bar absolute and 90 °C and exits at a pressure of 1 bar absolute. These are incorporated into the model through setting a constant total pressure and temperature at the inlet and a constant static pressure at the outlet. Measurements of mass flow rate, indicated power, outflow temperature, and internal chamber pressure have been previously conducted for male rotor speeds between 500 and 18000 rpm, and the model considers four of these speeds (1000, 4000, 10000, and 16000 rpm) to compare calculated and measured indicator diagrams. The nominal rotor-tip-to-housing clearance in the base cases is 0.08 mm. Additional measurements have been conducted in modified housing geometries where the bore is increased to yield rotor-tip-to-housing clearances of 0.16 mm and 0.24 mm, and these conditions are the focus of the second part of the CFD modeling study.

4. Results and discussion
The model described in section 2 is applied to the test case described in section 3, and the results are presented and discussed here. First presented is the steady state thermal predictions, followed by grid and leakage treatment, and lastly the comparison of different rotor-tip-to-housing clearances.

4.1. Base case temperature comparison
Since there is a close coupling between the solid and fluid solutions and a large discrepancy in their time scales, it is very important to show that the model has reached an appropriately temporally converged steady state (or a statistically stationary state). Figure 4 shows the inflow, outflow, and rotor temperatures for a case to demonstrate how this is achieved and how the results compare to the measured fluid temperatures.

As shown in the temperatures plotted on the left side of figure 4, the model is first run for a number of cycles. After five revolutions of the male rotor, the solid is solved to steady state to yield new wall temperatures seen by the fluid (and evident by the jump in rotor temperature at 5 revolutions). From there, the fluid and solid are continued to be solved together at each time step in a time-accurate fashion, and at every lobe passing the solid is again solved to steady state using averaged fluid temperatures for the previous five lobe passing times. It takes about an additional five revolutions to arrive at steady state for most of the solid temperatures and fluid temperatures around the rotors. However, because there is some additional time required to flow through to the outlet, it takes longer (about twenty revolutions of the male rotor in total) to reach a steady state for the average temperature at the outflow.
Figure 4. On the left is the evolution in time of temperatures (outlet fluid measurement shown in a brown dashed line, calculated outlet fluid spatial mean in the solid brown line, female rotor spatial mean temperature in a solid dark green line, male rotor spatial mean in a dark blue line, and inlet temperature in a dark purple line) in the coupled fluid-solid model as they reach steady state through the super-cycling procedure. On the right are the outlet temperatures at steady state comparison between measurements (black squares) and CFD model at various locations (outlet in blue circles, monitor point DA6 on the housing side in green triangles, and monitor point DA7 on the outlet housing underside in orange diamonds) as functions of male rotor speed.

The right side of figure 4 shows the modeled temperatures compared to the predicted temperatures at the outlet. In general, at high rotational speeds the leakage is proportionally less, so the outlet temperature is controlled primarily by the balance between the isentropic outlet temperature and the solid wall temperatures. As the speed decreases, the internal leakage effects become more important, and more high temperature gas from the inlet makes its way to the outlet, resulting in both higher outlet temperatures and higher solid temperatures. In general, this trend is well-produced by the model. Figure 4 shows the modeled temperature at three different locations: the actual domain outlet, monitor point DA6 near the outlet control plate, and monitor point DA7 slightly downstream of the outlet plate (where the measured temperature is actually reported).

At 16000 rpm, the agreement at DA7 is very good, but for 10000 and 4000 rpm the temperature is over-predicted by about 10 K. At 1000 rpm, the temperature is over-predicted slightly more, by nearly 20 K. However, at this point in the curve there is a large sensitivity of outlet temperature to rotor speed. As seen by the other modeled temperatures, there is some sensitivity to the spatial location of the monitor point and the solid wall temperature there. More work could also be done to characterize the convective boundary condition applied to the outer housing. An additional consideration which is also very important is the variation in time to steady state with rotor speed. While the model time for the low rotational speeds represented a greater physical time, it also corresponded to running fewer lobe passings, so more work should also be done to determine the quality of the transient solution reaching steady state for these lower speed conditions.

Figure 5 shows the rotors and a cut-plane over the fluid and housing colored by the temperature. The difference in outlet temperature shown in figure 4 is shown more qualitatively here. The higher fluid outlet temperatures, as well as the higher solid wall temperatures, can easily be seen here as the rpm decreases from 16000 to 1000.
4.2. Effects of the grid resolution, leakage paths, and clearance treatment

Once the procedure for obtaining the steady-state thermal solution is validated, the behaviour of the mass flow rate, indicated power, and internal chamber pressure is examined for different setup configurations of the base case (4:1 pressure ratio, 90 °C inlet temperature, and 0.08 mm cold rotor-to-housing clearances). Three different configurations are examined: Case 1 which has cold rotor clearances and under-resolved clearance flow, Case 2 which has hot rotor clearances (determined from the steady state thermal load) and a clearance model described in reference [15] for modeling the clearance flow, and Case 3 which is identical to Case 2 but also includes an additional bearing leakage path on the inlet side of the rotors and the bearing casing which is visible in the top-left of figures 2 and 3. The pressure in this additional region is supplied based on measurements and is referred to here as the cushion pressure.

First, the mass flow rate and indicated power for these three cases are compared in figure 6 for varying rotational speeds. The simplest and least expensive configuration, Case 1, results in the mass flow rate and indicated power in closest agreement with the measurements. However, it is clear when looking in deeper detail at the internal chamber pressures in figure 7, that those models do not yield the correct mass flow rate and indicated power for the right reason; likely offsetting errors resulting from over-predicting the leakage rates out of the chamber during the initial part of the expansion process and then over-predicting the leakage rates into the chamber during the end of expansion. So much can be learned from Case 1: even though the mass flow rate and indicated power may yield good agreement with the measurements, the offsetting errors are deceiving in predicting the right answers for the wrong reason. It is very valuable to have more complete description of the flow, particularly the internal

![Figure 5. Rotors and a cut-plane across the rotor axes colored by temperature near steady state for male rotor speeds of (from top-left) 1000, 4000, 10000, and 16000 rpm.](image-url)
chamber pressures for a thorough analysis and making sure the model results are correct for the right reasons.

Figure 6. Mass flow and indicated power for measurements (black squares), and CFD models for cases 1 (blue circles), 2 (green triangles), and 3 (orange diamonds) as functions of male rotor speed.

Progressing from Case 1 to Case 2, one clear difference is that the mass flow rate drops consistently from 1 to 2. This occurs because of a generally greater restriction, both in the clearance flow model and the generally smaller clearances in Case 2. Since the flow between chambers is nearly choked for a large part of the expansion process, the clearance model mainly reduces the flow rate from that on the under-resolved grid, which is at a point around plug flow at the sonic velocity. There is a slight drop in indicated power from Case 1 to Case 2, which can be better understood by examining the internal chamber pressure traces in figure 7. From an angle around 240°, there is less refilling of the chamber (from the higher pressure chamber on the inlet side) due to the restricted leakage in Case 2, so the pressure is generally lower around this point.

From Case 2 to Case 3, where the bearing leakage is added into the model, the mass flow rate increases slightly, closer to the measured values, but still less than those occurring in Case 1. This is mainly contributed at the point in the cycle where the internal chamber pressure decreases beyond the cushion pressure but the chamber is still in communication with the bearing leakage passage. The effects on the internal chamber pressure are strongest in Case 1, but the effect on the mass flow is present in all cases. The average error in the mass flow rate for Case 3 is about 4% too low, however considering the overall agreement in all compared metrics, there is significant improvement overall from other less sophisticated configurations, Cases 1 and 2.

The model does a good job at predicting that, in general, as the rotational speed increases so do the pressure wave dynamics in the inlet port. However, there is considerable deviation in the actual pressure signal for the points with higher rotational speeds, specifically at 10000 and 16000 rpm. This is perhaps a result of the inlet boundary condition, which allows pressure reflections. Further investigations with non-reflective boundary conditions at the inlet and outlet, as used in [14], are currently being examined.

In general, the prediction of indicated power (which in the CFD model is derived from the pressure and viscous torque on the rotors and the rotational speed) follows that of the mass flow rate; the models which predict the higher flow rate also yield higher power. This is explainable through the comparison of the internal pressure, where the cases with higher mass flow rate also generally had higher internal chamber pressures, which translates to higher power consumption when computing the indicated work from the pressure and volume integral.

The refilling of the chamber from the inlet side higher pressure chamber that occurs around 250° and is most evident at the low rotational speeds is in general well-predicted by the model. At some of the conditions (specifically at 4000 and 16000 rpm), the decrease in the pressure is offset from the
measurements by about 10°. This occurs from the start of expansion, so it is possibly a result of incorrect phasing of the pressure fluctuations at the inlet and not as much in the expansion itself.

![Internal chamber pressures for measurements and CFD models](image)

**Figure 7.** Internal chamber pressures for measurements (black solid lines), and CFD models for cases 1 (dark blue dashed line), 2 (medium green dashed-dotted lines), and 3 (thin light orange lines) for male rotor speeds of (from top-left) 1000, 4000, 10000, and 16000 rpm.

4.3. **Effects of rotor-tip-to-housing clearances**

In the final investigation, rotor-to-housing clearances of the base case geometry at 0.08 mm are compared to clearances of 0.16 mm and 0.24 mm using newly acquired experimental measurements. In the experiments, the rotor bores are increased so that the rotor-to-housing clearances are increased equally around the bore. For the CFD model, the configuration equivalent to Case 3 (with both higher grid resolution, hot clearances, modeled clearance flow, and bearing leakage paths) is assessed for the two new clearances.

The mass flow rate and indicated power for the measurements and CFD models are shown in figure 8. As the clearances increase, the measured flow rate increases nearly proportionally to the clearance for most of the points, especially at lower rotor speeds. For the higher speeds, there is slightly more variation possibly due to the varying throttling and acoustic effects in the inlet port. The CFD models yield flow rates with very similar trends, that is, increase in flow rate proportionally to the clearance. At low rotor speeds of 1000 and 4000 the results coincide very well for all clearances. At 10000 rpm, the CFD model prediction is significantly lower than the measured values, which will be
evaluated and explained after examining the internal chamber pressure. At 16000 rpm, there is a little more deviation in the opposite direction, with the CFD model over-predicting the flow rate.

Figure 8. Effect of rotor-tip-to-housing clearances on global measurements and model results. To the left is the mass flow rate and to the right is the indicated power, both as functions of male rotor speed. The measurements are shown in the small gray symbols for clearances of 0.08 mm (dark circles), 0.16 mm (triangles), and 0.24 mm (light diamonds). The model results are shown in larger colored symbols of 0.08 mm (blue circles), 0.16 mm (green triangles), and 0.24 mm (orange diamonds).

The comparison between the measured and modeled indicated power is also shown in figure 8. In the measurements, there is a very slight decrease in the indicated power as the clearances increase. This physically represents the power lost from the excess leakage mass. In the CFD models, the difference is slightly exaggerated. At all the different speeds there is a decrease in indicated power as the clearance increases. For all points except 10000 rpm, the models yield a slightly higher indicated power. More insight on the power generation will be extracted from the internal chamber pressure comparison in figure 9.

Figure 9 shows the measured and computed chamber pressure for all four rotor speeds. At the lowest speeds, 1000 and 4000 rpm, the effects of increasing the clearance are strongest. As the expansion process begins (when the inlet port control edges are closed around 200°), the pressure inside the chamber decreases more rapidly with larger clearances. This flow is presumably directed mainly at the adjacent low pressure chamber, so there is some of the decrease in pressure which is not due to expansion itself but instead inter-chamber leakage. Very similarly, around 270° the pressure in the chamber can be seen to increase more (though it is still lower) when the clearance is large due to the chamber receiving more mass from the adjacent high pressure chamber.

At the higher speeds of 10000 and 16000 rpm, there is considerably less difference in the internal pressure due to the leakage being less important at high speeds. There is, however, considerably more effect of the acoustic waves and the throttling. For the 10000 rpm case, which had an unexpectedly low mass flow rate and indicated power across all clearances, the reason is more obvious when examining the chamber pressure. A large difference in the pressure behavior in the inlet port is the reason, as was observed in the previous cases at 10000 rpm. At 16000 rpm, the differences in inlet port pressure are not so extreme, however there is still a large decrease in pressure observed between the inlet port and the rest of chamber during filling which is due to the throttled flow.
Figure 9. Effect of rotor-to-housing clearances on measured and model internal chamber pressures. Solid thick gray lines are from measurements at clearances of 0.08 mm (dark), 0.16 mm (medium), and 0.24 mm (light). Dashed colored lines are from the model at clearances of 0.08 mm (dark blue), 0.16 mm (medium green), and 0.24 (light orange). Male rotor speed differs between each figure at (from top left) 1000, 4000, 10000, and 16000 rpm.

5. Conclusions and future work
This study has several important findings relevant to the topic of modeling flows in screw machines. The first of these is the validity of the cut-cell based method for CFD modeling. Key advantages of this approach include general applicability regardless of geometry or motion types, and relatively easy process of beginning a simulation from CAD without the need to create a volume mesh. The method conserves volume and mass, and it utilizes stationary orthogonal hexahedral grid for low numerical diffusion, high accuracy, and high stability. A key limitation to this approach relevant to this application is the need to model the clearance flow in order to keep the computational cost manageable. This issue was addressed in comparing the under-resolved flow to cases in which a simplified model developed in another work was utilized.

The use of a semi-transient approach called super-cycling is shown to solve a problem of the disparate time scales between the solid and the fluid without the need to artificially change the solid or fluid thermal properties or conduct multiple simulations. Approximately 10 revolutions of the male rotor were required to arrive at a steady state solution in the solid, after which the flow and solid could be continued either with super-cycling or in a purely transient, time-accurate way. The thermal solutions were validated against measurements of the temperature and found to be in very good agreement for the
highest speed, but the agreement deviated more as the speed was reduced. Current work is ongoing to
determine if this is a result of differences in the flow rate predictions, interaction with the leakage flow,
heat transfer model applied through thermal boundary layer profiles, or an artifact of not actually
reaching a steady state condition.

The model behavior is compared to experimental results over a wide range of rotor speeds and
clearances. The model results which considered the thermal deformation, clearance flow model, and
bearing leakage demonstrated the best overall agreement with measurements. As was observed early
on in Case 1, one must be careful to determine the model validity from mass flow rate and indicated
power alone. The internal chamber pressure measurements conducted at TU Dortmund University are
very valuable in determining whether or not the model is correct for the right reasons. There is still
more improvement to be made in the prediction of the internal chamber pressures for two main reason:
(1) the clearance flow model must be generalized and evaluated specifically for the operating conditions
evaluated here, and (2) the pressure reflections at the inflow boundary must be evaluated and alternative
boundary conditions, including a Navier-Stokes characteristic boundary condition with controlled
degree of impedance at the inlet, should be investigated.

The measurements with varying rotor-to-housing clearances are very helpful in understanding the
model behavior. It is clear from the results of the low speed cases that there is still more work required
to predict accurately the internal chamber pressures and leakage flows, though the current results show
levels of agreement which are roughly similar or less error than other reported works on the same case.
More work is currently required to quantify precisely the leakage flow rates into distinct categories of
blowhole leakage, axial rotor-to-housing leakage, radial rotor-to-housing leakage, and rotor-to-rotor
leakage.

One additional improvement necessary for the solver in these cases is to improve the computationally
efficiency. While these runs were not necessarily optimized for fastest turn-around time, there is much
room for improvement here. Especially at the low rotational speeds (the run time scales almost inversely
proportionally to rotor speed), the run times could become prohibitively long for application in an
engineering context (run times at 1000 rpm were generally around 2 weeks total turn-around time
running in parallel on 32 cores; the goal is typically to have a simulation result in under 15 hours, a
target which is currently achieved only in the 16000 rpm cases). Current work is ongoing in several
areas, including tighter coupling to allow running with larger time steps and higher parallel efficiency
related to moving geometries and surface storage.

References
[1] Smith I K, Stosic N and Kovacevic A 2014 *Power recovery from low-grade heat by means of
screw expanders* (Chandos Publishing)
[2] Stosic N, Kovacevic A, Hanjalic K and Milutinovic L 1988 Mathematical Modelling of the Oil
Influence Upon the Working Cycle of Screw Compressors *Proc. of the International
Compressor Engineering Conference at Purdue* (Paper 645) West Lafayette, USA: Purdue
University
[3] Kauder K and Rau B 1994 *Auslegungsverfahren fur Schraubenkompressoren, Design Procedure
for Screw Compressors* (Düsseldorf: VDI-Verlag)
[4] Hanjalic K and Stosic N 1997 Development and Optimization of Screw Machines with a
Simulation Model, Part II: Thermodynamic Performance Simulation and Design
Optimization. *Transactions of the ASME* 664(119)
[5] Janicki M 2007 *Modellierung. und Simulation von Rotationsverdrängermaschinen*. Dissertation,
TU Dortmund University, Dortmund, Germany
[6] Feng C, Yueyuan P, Ziwen X and Pengcheng S 2001 Thermodynamic Performance Simulation
of a Twin-Screw Multiphase Pump *Proc. of the Institution of Mechanical Engineers, Part E:
Journal of Process Mechanical Engineering* 215(2) 157-163
[7] Nikolov A, Huck C and Brümmer A 2012 Influence of Thermal Deformation on the Characteristic
Diagram of a Screw Expander in Automotive Application of Exhaust Heat Recovery
*Proc. of
the 22nd International Compressor Engineering Conference at Purdue (Paper 2128) West Lafayette, USA: Purdue University [8] Gräßer M and Brümmer A 2014 An Analytical Model of the Incompressible One-Phase Clearance Flow in Liquid Injected Screw Expanders Proc. of the International Conference on Screw Machines Dortmund, Germany: TU Dortmund [9] Kovacevic A 2002 Three-Dimensional Numerical Analysis for Flow Prediction in Positive Displacement Screw Machines Ph.D. Thesis, School of Engineering and Mathematical Sciences, City University London, UK [10] Voorde Vande J, Vierendeels J, Dick E 2005 Development of a Laplacian-Based Mesh Generator for ALE Calculations in Rotary Volumetric Pumps and Compressors Comput. Methods Appl. Mech. Eng., 193, 4401–4415 [11] Kovacevic A, Rane S, Stosic N, Jiang Y, Furmanczyk M and Lowry S 2014 Influence of Approaches in CFD Solvers on Performance Prediction in Screw Compressors Proc. of the 22nd International Compressor Engineering Conference at Purdue (Paper 2252) West Lafayette, USA: Purdue University [12] Kovacevic A and Rane S 2013 3D CFD Analysis of a Twin Screw Expander Proc. of the 8th International Conference on Compressors and Their Systems London, UK: City University London [13] Papes I, Degroote J and Vierendeels J 2015 Numerical Simulation of a Twin Screw Expander for Performance Prediction Proc. of the 9th International Conference on Compressors and their Systems London, UK: City University London [14] Rainer A, Hesse J, Babic H, Salecker U, Spille-Kohoff A, Nikolov A and Brümmer A 2016 CFD Simulation of a Twin Screw Expander including Leakage Flows Proc. of the 23rd International Compressor Engineering Conference at Purdue (Paper 2497) West Lafayette, USA: Purdue University [15] Rowinski D H, Li Y and Bansal K 2018 Investigations of Automatic Meshing in Modeling a Dry Twin Screw Compressor Proc. of the 24th International Compressor Engineering Conference at Purdue West Lafayette, USA: Purdue University [16] Senecal P K, Pomraning E, Richards K J, Briggs T E, Choi C Y, McDavid R M, Patterson M A, Hou S and Shethaji T 2007 A New Parallel Cut-Cell Cartesian CFD Code for Rapid Grid Generation Applied to In-Cylinder Diesel Engine Simulations SAE Technical Paper #2007-01-0159 [17] Pomraning E, Richards K J and Senecal P K 2014 Modeling Turbulent Combustion Using a RANS Model, Detailed Chemistry, and Adaptive Mesh Refinement SAE Technical Paper 2014-01-1116 [18] Yakhot V and Orszag S A 1986 Renormalization Group Analysis of Turbulence. I. Basic Theory. J. Sci. Comput., 1, 3 [19] Issa R I 1985 Solution of the Implicitly Discretised Fluid Flow Equations by Operator-Splitting Journal of Computational Physics, 62 [20] Rhie C M and Chow W L 1983 Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation AIAA J., 21, 1525-1532 [21] Hütker J and Brümmer A 2013 Physics of a Dry Running Unsynchronized Twin Screw Expander Proc. of the 9th International Conference on Compressors and their Systems London, UK: City University London [22] Nikolov A and Brümmer A 2014 Influence of Water Injection on the Operating Behaviour of Screw Expanders: Experimental Investigation Proc. of the International Conference on Screw Machines Dortmund, Germany: TU Dortmund [23] Nikolov A and Brümmer A 2016 Analysis of Indicator Diagrams of a Water Injected Twin-shaft Screw-type Expander Proc. of the 23rd International Compressor Engineering Conference at Purdue (Paper 2492) West Lafayette, USA: Purdue University