CFD simulation of a dry scroll vacuum pump with clearances, solid heating and thermal deformation

A Spille-Kohoff, J Hesse, R Andres and F Hetze
CFX Berlin Software GmbH, Berlin, Germany
E-mail: andreas.spille@cfx-berlin.de

Abstract. Although dry scroll vacuum pumps (DSPV) are essential devices in many different industrial processes, the CFD simulation of such pumps is not widely used and often restricted to simplified cases due to its complexity: The working principle with a fixed and an orbiting scroll leads to working chambers that are changing in time and are connected through moving small radial and axial clearances in the range of 10 to 100 µm. Due to the low densities and low mass flow rates in vacuum pumps, it is important to include heat transfer towards and inside the solid components. Solid heating is very slow compared to the scroll revolution speed and the gas behaviour, thus a special workflow is necessary to reach the working conditions in reasonable simulation times. The resulting solid temperature is then used to compute the thermal deformation, which usually results in gap size changes that influence leakage flows. In this paper, setup steps and results for the simulation of a DSVP are shown and compared to theoretical and experimental results. The time-varying working chambers are meshed with TwinMesh, a hexahedral meshing programme for positive displacement machines. The CFD simulation with ANSYS CFX accounts for gas flow with compressibility and turbulence effects, conjugate heat transfer between gas and solids, and leakage flows through the clearances. Time-resolved results for torques, chamber pressure, mass flow, and heat flow between gas and solids are shown, as well as time- and space-resolved results for pressure, velocity, and temperature for different operating conditions of the DSVP.

1. Introduction
Dry scroll vacuum pumps (DSPV) are essential devices in many different industrial and academic processes. The electronic and semiconductor industry is the biggest application area of vacuum pumps, for example plasma etching and physical vapor deposition in manufacturing displays or computer chips. Vacuum pumps are also used in food industry for packaging technology or freeze-drying, in metallurgy for degassing of melt or inside a coating line. For research vacuum technology is used for electron microscopes or mass spectrometer. There are several different types of compressor mechanisms. The scroll pump is constructively the simplest solution for a vacuum pump. The advantages are high performance down to $10^{-3}$ mbar, low cost, easy maintenance, low noise and small vibrations compared with other kinds of dry vacuum pumps.

The scroll vacuum pump is quite complex from the fluid dynamics point of view. The driven orbiting scroll wrap changes the position relative to the fixed scroll wrap which results in a time changing working chamber volume. The fluid is compressible and therefore acoustic aspects could be important. Because of the vacuum characteristics the transition from the continuous flow regime to the molecular regime might have to be taken into account.
The Computational Fluid Dynamics (CFD) computation of scroll machines requires accurate grid generation because the fluid flow is transient and depends on the scroll position. Furthermore, the grid needs a high resolution especially in the gaps and their vicinity. Here, we used TwinMesh to generate high quality grids in a short time.

This paper is based on [3] which includes CFD simulation results of a dry scroll vacuum pump. In [4], we simulated the fluid region between the solid scrolls only, as done in [3], but added axial gaps to the simulation domain. Both [3] and [4] neglected the solids but set a thermal boundary condition on the surface of the solid scrolls towards the transported gas. [3] suggested a fixed temperature of 65°C at all solid surfaces but already mentioned that this causes calculation errors and the value should depend on the process parameters, in their case on suction pressure and rotation speed.

In this paper, we added the solid scrolls as given in a sketch in [2] with conjugate heat transfer between compression chamber and scrolls and a convective boundary condition at the outer surface of the scrolls. As a main advantage we get consistent temperatures in the solids depending on process parameters, and we can use the temperature distribution for the calculation of thermal deformation of the scrolls. The main problem is that the timescale of solid heating is very much longer than that of scroll rotation and fluid dynamics. Thousands of rotations would be necessary to capture the heating up to operating conditions, needing a huge amount of simulation time. Therefore the solid heating was accelerated by an increased time step size in the solids.

2. Geometry

The working chamber of the scroll pump is a simple cylindrical volume including one fixed and one orbiting scroll wrap. The inlet and outlet volume is simplified as a pipe, which could have an influence on the DSPV performance in comparison to the real geometry. One important geometrical parameter of DSVP is the volume ratio, which is the ratio of the suction chamber volume to the discharge chamber volume. Here a volume ratio of 2.0 is used corresponding to the involute starting angle $\Phi_S$ of $3\pi$ like in [3] and [1], leading to a design pressure ratio of 2.64. The pressure inside the working chamber is analyzed on four different locations (orbiting angle of 300°, 570°, 840° and 1110°) next to the fixed scroll wrap, which are the pressure transducer locations of [1]. The used axial clearance size is 30 microns. The specified axial clearance in [1] of 300 microns must be a mistake.

![Geometry of the scroll wraps and total assembly](image)

**Figure 1: Geometry of the scroll wraps [3] and total assembly [2].**

**Table 1:** Basic parameters of the DSVP.

| Parameter                     | Value       |
|-------------------------------|-------------|
| Radius of the basic circle    | 3 mm        |
| Pitch of involute             | 18.85 mm    |
| Initial angle of involute     | 40°         |
| Thickness of scroll wrap      | 4.19 mm     |
| Height of the scroll wrap     | 30 mm       |
| Loops of scroll               | 5.25        |
| Axial clearance               | 0.03 mm     |
| Minimum radial clearance      | 0.036 mm    |
Figure 2: Geometry of the fluid domain (left), side and top view onto fluid and solid domains (middle and right) with dimensions in Table 2.

Table 2: Parameters of the simulation domain.

| Parameter | Value |
|-----------|-------|
| L_{In}    | 15 mm |
| D_{In}    | 8 mm  |
| L_{WC}    | 30 mm |
| D_{WC}    | 250 mm|
| L_{Out}   | 60 mm |
| D_{Out}   | 25 mm |
| L_{S1}    | 25 mm |
| R_{S1}    | 135 mm|
| R_{S2}    | 150 mm|

3. Meshing
The discretization of the fluid volume is the main challenge to get reliable simulation results. The time changing volume has to be represented by a numerical mesh including all gaps. There are several methods to model the chamber volume with different restrictions.

3.1. Chamber Modelling for Positive Displacement (PD) Machines
The overlapping-meshes method is the simplest approach to investigate the rotor influence on the fluid flow inside a PD machine. The fluid domain of the PD machine is meshed regardless of the rotor geometry, i.e. the whole fluid domain is filled up with grid elements (as if the rotors were removed, called background mesh) whereas a second mesh describes the rotor geometry. For the coupling of both meshes, two main approaches exist. In the immersed solid method, the second mesh defines the solid region only; from this, at each time step the CFD code determines the region of overlap of solid and fluid meshes and applies momentum sources to force the fluid there to follow the motion of the solid rotor. In the overset mesh method (Chimera), the second mesh describes the fluid region around the solid up to an overset boundary; each time step, the nodes of both meshes are categorized as “dead” (no intersection, no fluid flow), “solve” (solution of conservation equations), and “donor” (gives solution) and “receptor” (gets solution from donor) between. The advantage of these methods is that grid generation for the fluid region is easy since the shape of the rotors and of the gaps need not be taken into account, and that the initial grids can be used for the whole simulation as remeshing or mesh deformation are avoided. However, one of the main disadvantages is that most CFD solvers limit the choice of physical and numerical models, e.g. only incompressible single-phase flows. Furthermore, a very high number of elements is necessary to accurately resolve the gaps; since the gap position varies with time the background grid also has to be refined circumferentially everywhere the gap will be during the rotor motion. For immersed solid method, the solution quality in the near wall region lacks of boundary resolution and turbulent wall functions since there is no wall in the simulation but a rotating fluid. The overset mesh method often lacks of conservation due to interpolation between
the meshes, and a severe time step restriction due to problems when nodes change between “dead” and “solve” in one time step.

The **deform-and-remesh method** rotates the rotor each time step and tries to deform the fluid grid to represent the new shape of chambers and gaps. Due to the big shape changes of the fluid regions in PD machines, the deformation often causes a bad quality grid or even an illegal grid. Therefore an algorithm is required that generates a new mesh during run-time depending on the rotor position and local mesh quality. As an alternative, grids may be prepared manually for certain positions (key-frame grids). Compared to the immersed solid method, the rotor surface is accounted for in a way that the near wall region can be treated more accurately. Also, compressible or multi-phase flows can be calculated. The main disadvantage is that remeshing in small gaps is unavoidable for almost every angle increment, but every remeshing requires an interpolation of the results from the old mesh onto the new mesh, which results in interpolation errors. Furthermore, the remeshing method must use an automatic grid generation program. Such programs often use unstructured mesh elements that result in high element numbers or in poor mesh quality within the gaps. This method is not efficient for a CFD simulation of a DSPV.

The **customized grid generation** can avoid the aforementioned disadvantages. Since each new rotor position results in a change of the fluid domain, the positions for each time step need to be meshed individually. These pre-generated meshes are read in at run-time into the CFD solver. If all these meshes share the same topology, i.e. the number of vertices and their connection to elements stay the same, and if the change in vertex positions from one mesh towards the next is quite small, interpolation of results is not necessary. Instead, the mesh deformation is taken into account via the Leibniz rule in the partial differential equations describing the fluid flow. The demand for a constant mesh topology can be fulfilled most easily by a block-structured mesh of hexahedral elements. The computational effort is minimized and the mesh quality is high leading to reliable simulation results.

In this project, TwinMesh was used to pre-generate the hexahedral meshes for the deforming fluid region between fixed and moving scroll of the DSVP.

### 3.2. Customized Grid Generation of Scroll compressor with TwinMesh

TwinMesh is a meshing software for PD machines with two axially parallel rotors, with complex rotor geometry, i.e. continuous (e.g. lobe pump), discontinuous (e.g. screw compressor) or with a single rotor (e.g. scroll compressor, eccentric screw pump). CAD data of the rotor and casing curvature can be imported and is used for the structured grid generation. The topology of a scroll is meshed using O-type grid around the rotor curvature. Figure 3 shows the TwinMesh GUI in the case of a scroll pump. TwinMesh generates 2D meshes for each rotation angle of the rotor with the same topology and node numbers including a refined boundary layer resolution layers towards rotor and housing walls. The meshes are smoothed with an explicit and iterative method to reach homogeneous node distribution and orthogonality. The resulting mesh of the chamber volume can be analyzed by quality criteria, such as minimum element angle, aspect ratio and volume change of cells. The 2D meshes are connected to get the 3D volume mesh. The 3D meshes are exported from TwinMesh for each rotor position (i.e. the meshes cover one complete cycle), where an angle increment of $3^\circ$ is used for this simulation of the DSVP, i.e. 120 meshes for one scroll motion cycle.

### 3.3. Simulation Domain

The simulation domain is split into six volumes to use the adequate meshing strategy for each volume (Figure 4): The two solids, the deforming fluid region between the scrolls, the stationary fluid region between outer solid and scroll with inlet and outlet pipes, and the two fluid regions in the axial gaps between scroll and opposite disc (one stationary and one orbiting).
The meshes of the deforming fluid region are created with TwinMesh whereas all other meshes are generated with ANSYS Meshing as extruded meshes with hexahedrons and wedges. The orbiting solid consists of 72,000 elements, the stationary solid of 47,000 elements. The stationary fluid region consists of 190,000 elements, each axial gap of 75,000 elements, and the deforming fluid region of 2,000,000 hexahedrons (20 in radial direction between the scrolls, 50 in axial direction). In sum, the partial differential equations are solved on 2,788,821 nodes in 2,516,640 hexahedrons and 3508 wedges. We used a reasonable mesh resolution and grid size, but so far haven’t done a grid-independence study.

4. Simulation Setup
The commercial CFD solver ANSYS CFX is used for the simulation. The meshes for the six different parts are connected with GGI's (Generalized Grid Interfaces); the motion of the orbiting solid and of the axial gap between the orbiting scroll and the stationary disc is described with expressions, the de-
formation of the fluid region between the scrolls by the reading of the pre-generated TwinMesh meshes at run-time via a FORTRAN routine. The GGIs are automatically updated each time step to consider the time dependent connections.

The working fluid of the DSVP is air which is modelled as ideal gas, the discs and scrolls consist of steel. Energy conservation is solved in all regions, additionally Navier-Stokes equations with SST turbulence model in fluid regions. Though the DSVP is a vacuum pump, the lowest pressure of 17 kPa used here gives a Knudsen number of 0.011 so that continuum approach and no-slip boundary conditions at walls are still usable; for lower pressures, the usage of a slip model is possible.

We simulated the DSVP for three process parameters given in [1-3]: Suction pressures of 17 kPa, 42 kPa and 95 kPa at a fixed discharge pressure of 95 kPa at a rotational speed of 1704 rpm. Since design pressure ratio is 2.64, we expect under-compression for 17 kPa, slight over-compression for 42 kPa and high over-compression for 95 kPa.

For the energy equation, 20°C inlet temperature is used at inlet pipe. Due to the compression in the DSVP, temperature rises with pressure in the working chamber and heat flows into the solids. At the outside of the solid discs, we assume a convective boundary condition by setting a heat transfer coefficient of 16.8 W/(m² K) towards an outside temperature of 20°C. Some solid components as shaft and frame are neglected since full CAD data was not available. To get comparable solutions to [3] and [4], we made further simulations where we neglected all solids and fixed the wall temperatures in the fluid regions to 65°C, for 17 kPa suction pressure also for 45°C and 85°C.

Time step size of the transient simulations is 0.293 ms due to 120 meshes for one cycle of oscillation time 35.2 ms at 1704 rpm. The advection scheme is high resolution and the transient scheme is second order backward Euler. With this setup, simulation time for one oscillation cycle of the DSVP is 16 hours on 8 cores of Intel(R) Xeon(R) E5-2637 with 3.5 GHz with a demand for 35 GB RAM if solved in double precision. The residuals converged to a root mean square value below 0.001; improved convergence is possible by using a smaller time step size, i.e. more meshes for one rotation.

Simulation was run over a lot of rotations until a periodic state was reached. Since the heating of the solid structures (discs and scrolls) is slow compared to cycle time, i.e. in the range of 10 minutes compared to 35 ms for one cycle, thousands of cycles would be necessary with simulation time of several months. Since only the periodic state in working condition is of interest here, we accelerated the solid heating by an increased time step in the solids (up to a factor 10,000) and decreased this factor in steps to 1 when the temperatures in the solids reached their final level. With this approach, we got stagnant solid temperatures and periodic results in 10 to 20 cycles, i.e. one to two weeks of simulation time for each case.

5. Simulation results
In this paper, we extended the simulations described in [4] by taking into account the solids (discs and scrolls) and their temperature field. Therefore, we focus on the effect on working chamber pressures, on the temperature distribution in the solids and on their thermal deformation. Results for pressure, velocity and temperature distributions in the fluid region were given in [4] and did not change qualitatively. Figures 5 and 6 show working chamber pressures over orbiting angle for the suction pressures 17 kPa, 42 kPa and 95 kPa. In Figure 5, experimental results and ideal process curves from [2] are compared to our simulations without solids, i.e. with wall temperature set to 65°C, and with solids. The ideal process assumes adiabatic compression without gap losses.

With 17 kPa, the scroll pump works in under-compression mode, i.e. the pressure in the working chamber is not increasing to discharge pressure before the chamber opens to discharge pipe at 90° orbiting angle. The simulations capture the pressure increase well; the chambers are heated by the warmer walls or the warm solids and pre-filled from discharge side through the axial and radial gaps, so that real pressure is higher than that from ideal process. When the chamber opens to discharge side, chamber is filled from there until discharge pressure is reached. In the ideal process, filling occurs suddenly, whereas in the real process, filling is slower due to the slow opening of the gap between scrolls. In the experiment, pre-filling is stronger so that working chamber pressure is higher when
chamber opens to discharge side. Most probably, the axial and/or radial gaps in experiment are greater than the 30 µm used in the simulations. The simulations with and without solids show almost no difference since the assumed wall temperature of 65°C is approximately the solid temperature.

While the 42 kPa case has small over-compression in ideal case, experiment and simulations show almost 95 kPa in working chamber at opening to discharge pipe. For the over-compression case with suction pressure 95 kPa in Figure 6, no experimental data was given. Here, differences between the simulations without and with solids occur since the solid temperature does not reach 65°C for this case. In all simulations, oscillations can be seen in the pressure of the working chamber just after opening to the discharge pipe. This comes from the usage of standard, i.e. reflective opening boundary conditions at the pipe end; Figure 6 shows simulation results for 17 kPa suction pressure with three different pipe lengths: Oscillations with small period for the 60 mm pipe, oscillations with larger period for a 1 m pipe, and no oscillations for a 100 m pipe.

Deviations between experiments and simulations come from (a) not fully described geometry in [1-2], (b) incorrect gap sizes of 30 µm in the simulations due to measurement errors and thermal deformation, (c) influence of inlet and outlet boundary models, and (d) pressure measurement by pressure transducers with finite size and transducer film instead of monitor point data in the simulation.

Figure 5: Chamber pressure for suction pressure 17 kPa (left) and 42 kPa (right) at 1704 rpm: Comparison of experimental and ideal process data from [2] with simulations without solids (wall temperature 65°C) and with solids. Differences between both simulations are very small (see text).

Figure 6: left: Comparison of chamber pressure for suction pressure 95 kPa at 1704 rpm for simulation without solids (wall temperature 65°C) and with solids; right: Comparison of chamber pressure for suction pressure 17 kPa at 1704 rpm for simulation without solids (wall temperature 65°C) with different outlet pipe lengths to prevent reflection.
Table 3: Comparison of mass flow rate and power consumption for different suction pressures and temperature boundary conditions in the simulations.

| Suction pressure in kPa | Wall temperature in °C (no solids) | Heat transfer coefficient in W/(m² K) | Mass flow rate in g/s | Power in W |
|-------------------------|-----------------------------------|--------------------------------------|-----------------------|------------|
| 17                      | 45                                | -                                    | 0.803                 | 113.9      |
| 17                      | 65                                | -                                    | 0.770                 | 114.4      |
| 17                      | 85                                | -                                    | 0.727                 | 114.1      |
| 17                      | -                                 | 8.4                                  | 0.707                 | 114.0      |
| 17                      | -                                 | 16.8                                 | 0.738                 | 114.1      |
| 17                      | -                                 | 33.6                                 | 0.797                 | 114.4      |
| 42                      | 65                                | -                                    | 1.83                  | 93.6       |
| 42                      | -                                 | 16.8                                 | 1.91                  | 96.5       |
| 95                      | 65                                | -                                    | 4.02                  | 50.8       |
| 95                      | -                                 | 16.8                                 | 4.31                  | 51.2       |

Table 3 shows mass flow rates and power consumption for all simulated cases: Mass flow rates decrease with wall temperature in cases without solids, and increase with the heat transfer coefficient for the outer solid walls in cases with solids, since higher heat transfer coefficients lead to colder solids. The temperature distribution on the fluid side of both discs and scrolls can be seen in Figures 7 to 9 for a heat transfer coefficient of 16.8 W/(m² K). While the temperatures reach 63 to 94°C for 17 kPa suction pressure, in the 42 kPa case the solids are 43 to 60°C warm, and in the 95 kPa case only 28 to 39°C with the warmest region in the middle of the scrolls and not, as in the other cases, at the innermost ends towards discharge pipe. Since this case shows strong over-compression up to 140 bar, the gas decompresses towards the discharge pipe and therefore becomes colder again.

This temperature distribution leads to a thermal deformation of discs and scrolls; since some parts of the solid geometry were neglected (e.g. shaft and frame), adequate boundary conditions for the thermal deformation simulation were needed. At the red lines in Figure 10, we fixed the position of the centre of gravity for these boundary areas. The resulting deformation for 17 kPa suction pressure and heat transfer coefficient of 16.8 W/(m² K) is shown in Figure 10 also; the effect on the axial gap sides between the scrolls and the adjacent disc is shown in Figure 11: While the gaps near the discharge pipe decrease by 10% from 30 µm to 27 µm, they increase at suction side to 36 µm.

Figure 7: Temperature on surface of orbiting disc and scroll (left) and on stationary disc and scroll (right) for suction pressure 17 kPa for heat transfer coefficient 16.8 W/(m² K).
Figure 8: Temperature on surface of orbiting disc and scroll (left) and on stationary disc and scroll (right) for suction pressure 42 kPa for heat transfer coefficient 16.8 W/(m² K).

Figure 9: Temperature on surface of orbiting disc and scroll (left) and on stationary disc and scroll (right) for suction pressure 95 kPa for heat transfer coefficient 16.8 W/(m² K).

Figure 10: Sketch of location of boundary conditions for the simulation of thermal deformation (left, red lines), and total thermal deformation calculated for the case with suction pressure 17 kPa.
6. Conclusions and outlook

This paper shows CFD results for a dry scroll vacuum pump including axial gaps and solid components as scrolls and disc. Different suction pressures from 17 kPa to 95 kPa were applied using an inlet temperature of 20°C. The CFD results of the working process are compared with ideal process data and experimental data. In addition, average values of the mass flow and power consumption are shown for different suction pressures. For the CFD simulation ANSYS CFX was used and the meshes for the deforming working chamber were created with the meshing software TwinMesh, all other meshes with ANSYS Meshing.

The comparison of the CFD results with the experimental data indicates that the working mechanism and flow conditions within the dry scroll vacuum pump are well captured by the simulation. The heat transfer from the process gas to the casing and the leakage flow through axial and radial gaps has a main impact on the increase of the pressure over the rotation angle. The differences between experiment and simulation are mainly caused by geometrical uncertainties and model assumptions.

The inclusion of the solids with conjugate heat transfer allows a consistent simulation for different process parameters without the necessity to adjust the assumed wall temperature manually as in the cases without solids. The temperature distribution in the solid components is non-uniform with a higher temperature in the scrolls, especially in the center of the DSVP near discharge. The thermal deformation changes gap sizes by 10 to 20%.

We used a reasonable mesh resolution and grid size in this paper, but plan further mesh refinements to estimate grid dependencies of the solutions. Further investigation will adjust the meshes in the CFD simulations according to the thermal deformation to account for the change of gap sizes. The usage of a slip boundary condition for lower suction pressures towards the ultimate pressure is also planned.

References

[1] Li Z, Li L, Zhao Y, Bu G and Shu P 2010 Theoretical and experimental study of dry scroll vacuum pump Proceedings of the Vacuum 84, (415-421), Elsevier
[2] Li Z, Li L, Zhao Y, Bu G, Shu P and Lui J 2010 Test and analysis on the working process of dry scroll vacuum pump Proceedings of the Vacuum 85, (95-100), Elsevier
[3] Yue X, Lu Y, Zhang Y, Ba D and Wang G 2015 Computational fluid dynamics simulation study of gas flow in dry scroll vacuum pump Proceedings of the Vacuum 116, (144-152), Elsevier
[4] Hesse J and Andres R 2016 CFD simulation of a dry scroll vacuum pump including leakage flows Proceedings of 23rd International Compressor Engineering Conference