Scroll Compressor Oil Pump Analysis

S Branch
Ingersoll Rand – Trane
Compressor Development, La Crosse, WI, USA

E-mail: sbranch@trane.com

Abstract. Scroll compressors utilize three journal bearings to absorb gas, friction and inertial loads exerted on the crankshaft. To function properly, these bearings must be lubricated with a certain amount of oil. The focus of this paper will be to discuss how computational fluid dynamics can be used to predict oil flow out of a single-stage oil pump. The effects of speed and lubricant viscosity on pump output will also be presented. The comparisons will look at mass flow rates, differences in pressure, and torque at various speeds and dynamic viscosities. The computational fluid dynamic analysis results will be compared with actual lab testing where a crankshaft bench tester was built.

1. Introduction

Trane scroll compressors utilize three journal bearings to absorb the radial gas, friction and inertia loads exerted on the crankshaft. To function properly, these bearings need to be adequately lubricated by a single-stage oil pump.

The purpose of this analysis is to evaluate a crankshaft with an integral oil pump that was designed at a fixed speed and simulated at variable speeds. A scroll compressor crankshaft using RL32H oil and R410a refrigerant was used for this comparison. The oil pump was simplified to not include the oil flow to the lower main bearing or the upper main bearing. The CFD simulation results will analyze the differences in pressure, torque, and flow rates at these different speeds. Finally, a comparison will be conducted to evaluate the differences in the CFD results with a crankshaft bench tester that was built and tested.

Literature review found two papers that performed similar CFD simulations; one from Bernardi (1) in 2000 and one from Cho (2) in 2002. The study conducted by Bernardi (1) performed a CFD analysis of an oil pump used for a scroll compressor and then compared the oil mass flow rates of the CFD results with actual lab testing. The oil mass flow rates differed between the actual lab testing and the CFD analysis by 17%. The study by Cho (2) in 2002 looked at an oil pumping system of a variable speed compressor using a swing pump. The predicted results from this study had a maximum deviation of 12.8%. Both of these studies were only conducted as a steady state analysis assuming laminar flow for the oil.

2. Review of CFD Modelling Approach

The CFD simulations were conducted using CFX version 14.5.7. The simulation was built with two domains; one is stationary, and the inside one is rotating. The two domains are shown in Figure 1. Figure 2 shows the oil pump inside of the rotating domain along with the entrance and exit locations of the oil.
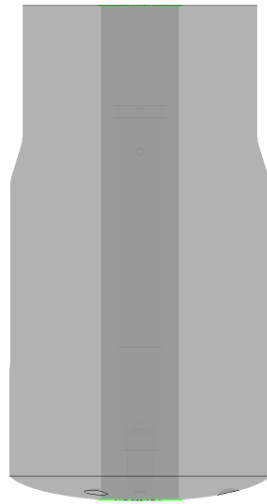

Figure 1. Stationary and Rotating Domains

Figure 2. Oil Pump

The modelling approach used by Bernardi (1) and Cho (2) is similar to the approach used in this study. In both cases oil out-gassing was not considered. Oil out-gassing occurs when the refrigerant entrained in the oil is released or absorbed as its pressure or temperature changes with the oil. For this study the turbulence was modelled as a homogenous turbulence flow using a SST model because the average Reynolds number was greater than 20000. Bernardi (1) and Cho (2) modelled the oil as laminar, which is one of the differences in the two techniques. The inside section is a rotating domain which consists of the oil pump and the crankshaft. The rotating domain meshes are 2.67 million nodes. The size of outside stationary domain mesh is 1.68 million nodes. There is a general grid interface at the inlet, outlet, and vent locations in the rotating domain. There is also a frozen rotor or transient rotor stator interface between the rotating and stationary domains. All of the simulations used the same geometry and mesh, but different setup assumptions.

- ANSYS CFX Version 14.5.7 was used for the CFD Analysis
- The simulation was set up with two continuous fluids, R410 and POE oil (RL32H).
- There were no phase changes
- All the walls were adiabatic.
- RPM varied (1800 RPM, 3500 RPM, 6000 RPM)
- The oil start level is 42.8 mm above the bottom of the crankshaft
- Buoyancy was turned on and was in the negative z direction.
- Turbulence was set up as a homogenous SST model.
- The reference pressure is 86.3 psia.
- Heat transfer was set to fluid dependent.
  - R410 temperature was fixed at 48 °F

- o   RL32H oil was fixed at 90 °F.
- The fluid pairs interphase transfer was set to a mixture model
  - o   Interphase length scale = 0.1 mm
- Drag coefficient = 0.44
- R410a refrigerant used an rgp table.
- The RL32H oil material properties:
- Molar Mass = 587 g/mol
- Density = 62.553 lbm/ft^3
- Specific Heat at Constant Pressure = 0.441 BTU/lbm F
- Oil Viscosity varied (2 cP, 25 cP, 50 cP)
- Thermal conductivity = 0.138 (BTU/ft^2 s (F/ft))

The simulation was started as a steady state analysis as was done by both Bernardi (1) and Cho (2). The steady state simulation ran until the mass and momentum converged to 1.0e-4. In addition, the imbalances were all less than 1%. Finally, the flows and pressures in to and out of the oil pump were not changing. After the steady state simulation converged, the results were used as the initial conditions for a transient rotor stator simulation.

The transient rotor stator simulation was simulated at 2 degrees of rotation per time step and ran until the solutions were converged. The convergence criteria were similar to the steady state method. Each time step needed to converge the mass and momentum to 1.0e-4, with imbalances less than 1% and stable flows and pressures in to and out of the oil pump.

3. CFD Results

Figure 3 shows the rotation of the crankshaft. Modelling the shaft rotation is important to predict the flow characteristics of the simulation. Figure 4 provides the streamlines to illustrate the oil flow path at one point in time. The oil travels from the inlet of the oil pump to the exit of the oil pump and then back to the oil sump. From Figure 3 where it shows the shaft rotating it can be observed in Figure 5 the vortex that is created with the oil; the lighter shade of grey is the oil which is at the bottom of the compressor shell, where the darker shade is the refrigerant. The transition region, shown in white, is a mixture of the oil and refrigerant.
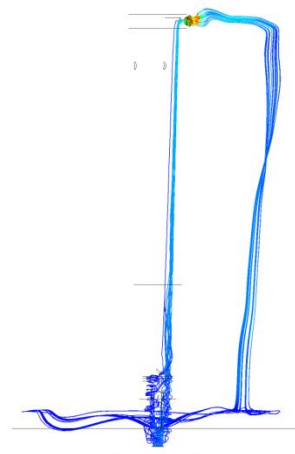

Figure 3. Crankshaft
Rotational Velocity.
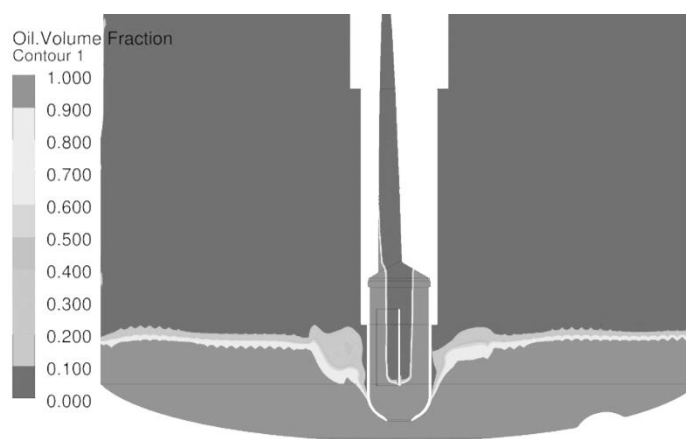
Figure 4. Oil Flow Streamline
Velocities.

Figure 5. Oil Volume Fraction @ 3500 RPM and 50 cP

A potential concern with the oil gallery being full with oil was addressed with the 6000 RPM solution. At 6000 RPM the amount of oil in the ½ inch diameter oil gallery was only 30% full as shown in Figure 6 and Figure 7. The lighter shade from Figure 7 is the oil and the darker shade is the refrigerant.
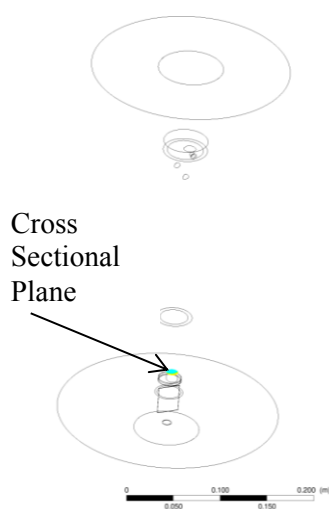

Cross
Sectional
Plane

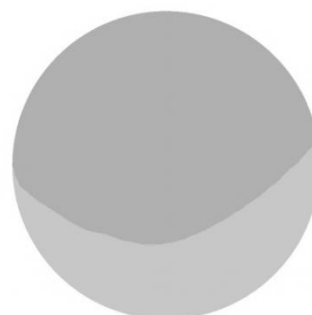**Figure 6.** Cross Sectional
Plane Location.

Figure 7. Oil Gallery Volume Fraction

Results of CFD simulations are shown in Table 1, including mass flow rates, pressure drop, and torque as a function of dynamic viscosity and various compressor speeds. The mass flow rate is an average value of the oil over a complete revolution. Torque is calculated from the oil and refrigerant of the crankshaft over one revolution. These results were used to compare with the bench tester. Figures 8, 9 and 10 come from the data in Table 1.

Table 1. Mass Flow Rate, Pressure Drop, and Torque Results.

MFR (kg/s)		RPM		
cP		1800	3500	6000
	2	0.017	0.052	0.098
	25	0.011	0.049	0.091
	50	0.007	0.045	0.086

Pressure Drop (psia)		RPM		
cP		1800	3500	6000
	2	0.084	0.168	0.393
	25	0.078	0.136	0.270
	50	0.074	0.115	0.251

Torque (Nm)		RPM		
cP		1800	3500	6000
	2	0.002	0.012	0.033
	25	0.002	0.011	0.033
	50	0.001	0.010	0.032

Figure 8 shows the effects of mass flow rates. Figure 9 looks at the change in pressure from the inlet of the pump to the outlet of the pump. Figure 10 looks at the change in torque over the range of dynamic viscosities and compressor speeds.
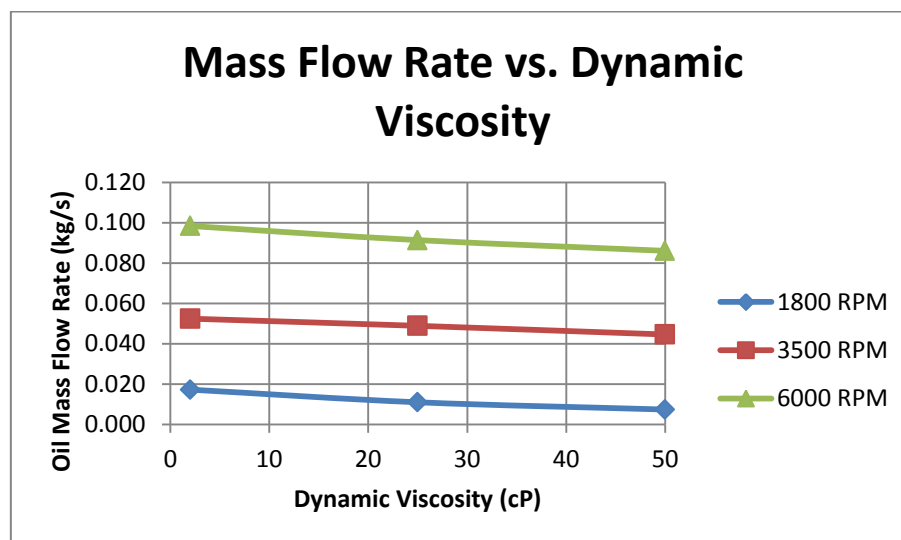
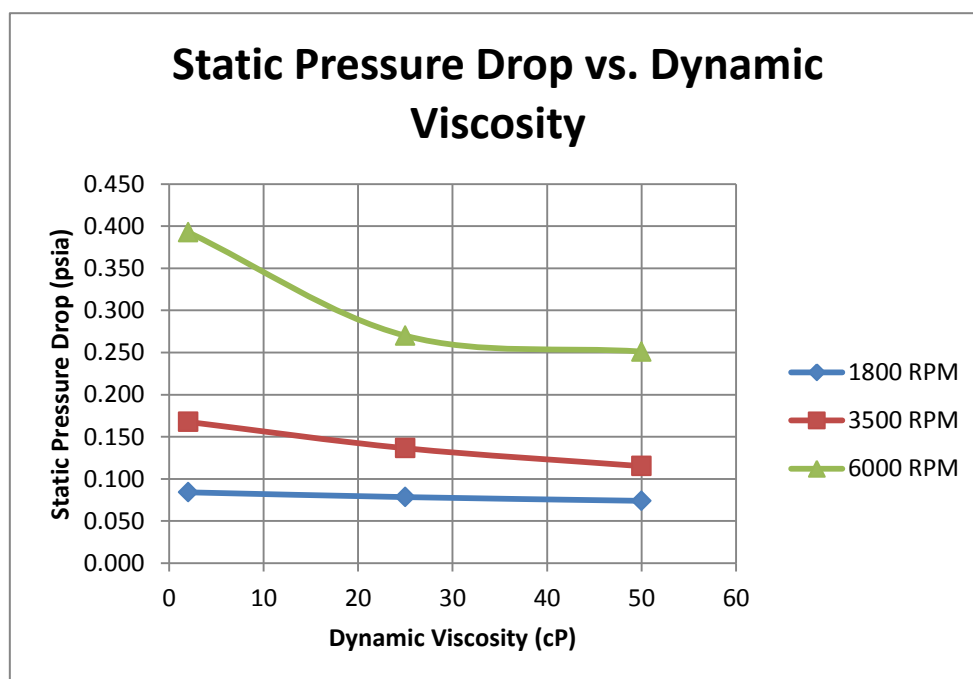**Figure 8.** Mass Flow Rate vs. Dynamic Viscosity.

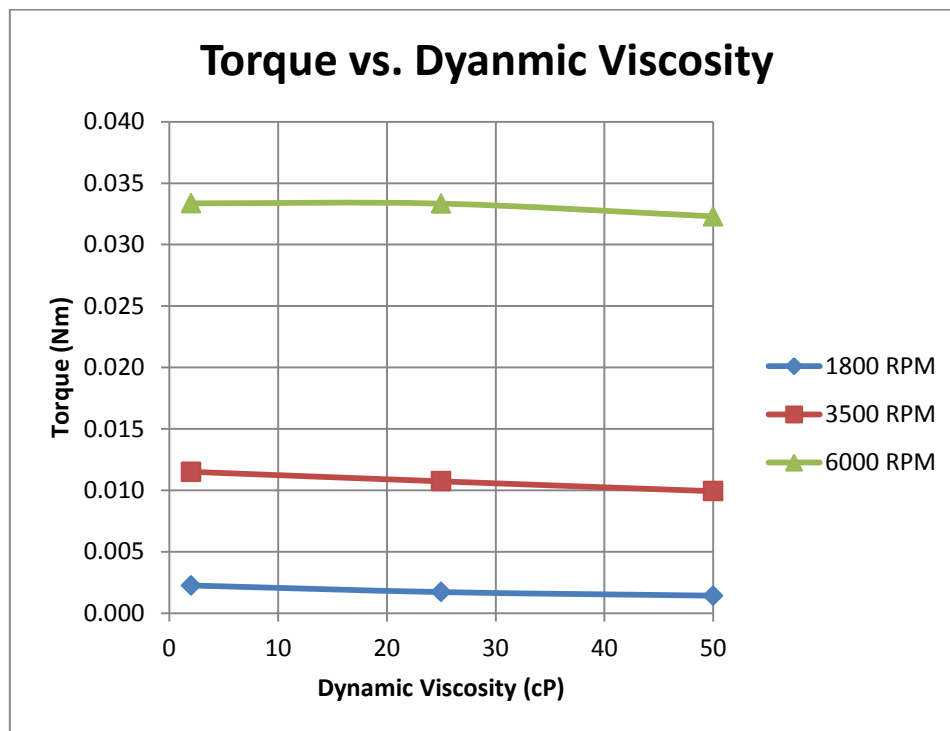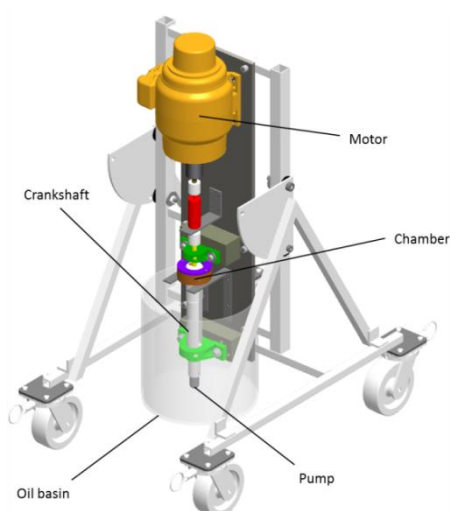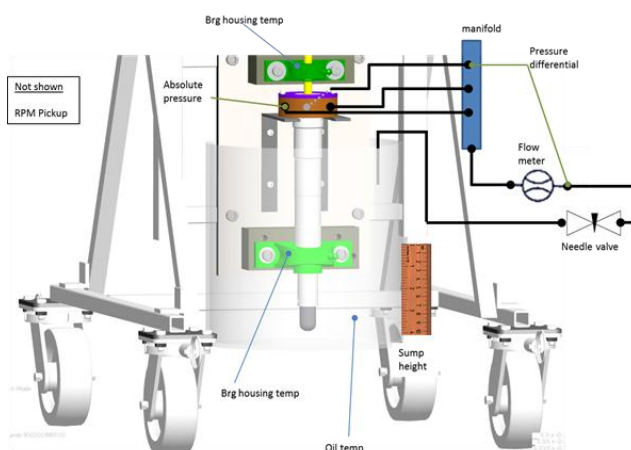**Figure 9.** Static Pressure Drop vs. Dynamic Viscosity.
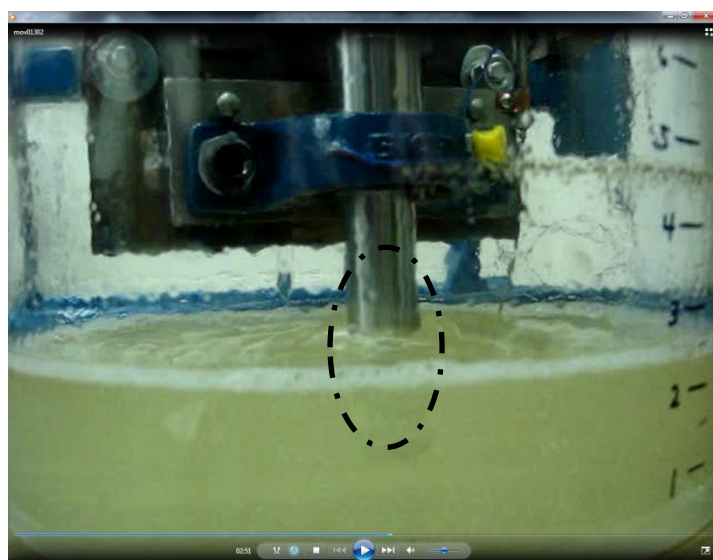
Figure 10. Torque vs. Dynamic Viscosity.

4. Comparison between the Bench Tester Results and the CFD Results

A bench tester was built to experimentally measure oil flow rates as a function of speed and lubricant viscosity, which is shown in Figure 11. The method that was used to measure the flow out of the crankshaft is shown on Figure 12. The flow exits the crankshaft and fills up the chamber. The change in pressure causes the oil to flow to the flow meter by flowing out of the chamber through three tubes, collect in the manifold and go through a flow meter. After the flow meter, the oil drains back into the bottom of the bench tester by gravity. The measurements were then compared to the CFD output to understand differences between the CFD model and the bench tester results. The bench tester consisted of an oil basin, the pump, a crankshaft to house the pump, and a chamber to collect oil discharged by the pump. A 2 hp electric motor provided torque to the assembly. A grade 68 cSt mineral oil was used for testing. Tests were conducted in air.

Figure 11. Bench Tester.

Figure 12. Bench Tester Flow Measurement Method.

Three areas were investigated to compare the CFD results with the bench tester. The first area looked at the vortex shape around the pump. The second area was the functionality of the gas vent at higher speeds. The third area was the oil flow rates at various speeds at a 50 cP viscosity.

Figure 13 shows the vortex around the pump. The vortex from Figure 13 does exist and is similar to what was predicted by the CFD on Figure 5.

Figure 13. Vortex of the Pump.

CFD predicted that the oil gallery was not full with oil at the 6000 RPM flow rates as shown in Figure 6 and Figure 7. When the bench tester ran at 6000 RPM it confirmed these results. The gas vent provides a method for the gas to leave the oil gallery without slowing the flow of the oil. The location of the gas vent is provided in Figure 14.
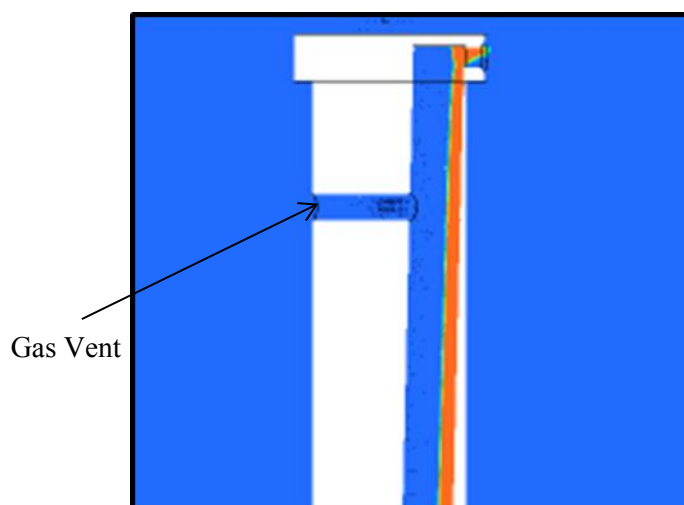
Figure 14. Gas Vent.

The bench tester was limited to a dynamic viscosity of 50 cP, therefore the flow rate comparisons were only conducted at that dynamic viscosity. Figure 15 shows the differences in the results from the CFD analysis and the bench tester. It is believed that a large portion of the error can be attributed to the measuring system of the bench tester putting extra pressure drop on the pump. If the CFD model had incorporated this extra pressure drop, there would have been less difference in the results.
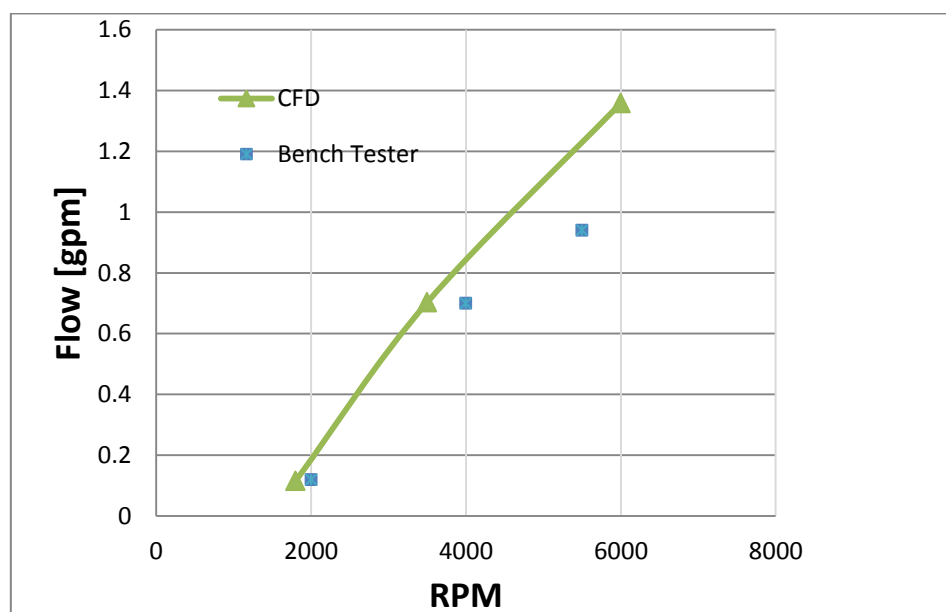

Figure 15. Model Validation Flow Rates at 50 cP.

5. Conclusion

- Although the crankshaft was designed for a fixed speed it performed adequately at variable speeds. The flow rates of the oil were sufficient to provide adequate lubrication to the bearings.

- The three areas that were compared between the CFD model and the bench tester showed the following.

  o In the CFD analysis the vortex shape appeared similar to the bench tester results. The main difference was in the CFD results it showed the transition region that was not visible from the bench tester.

  o The CFD simulation predicted that the gas vent would still function at 6000 RPM and was confirmed with the bench tester. This was verified by the percent of oil in the oil gallery in Figure 7.

  o The oil flow rates of the CFD analysis and the bench tester agreed well at low flow rates. As the flow rate increased the results deviated. It is believed that the differences are primarily due to the measuring system that was used for the bench tester which caused a larger pressure drop at higher flow rates.

- Further investigation would be to determine if there is a more appropriate turbulence model to better capture the behaviour of the oil and refrigerant.

Acknowledgement

References

[1] Bernardi, J.de, 2000,"CFD Simulation of a Scroll Compressor Oil Pumping System", Proceedings of the International Compressor Engineering Conference at Purdue, West Lafayette, IN,1454

[2] Cho,H, Yoo,B.K, Kim,Y, 2002,"CFD Simulation on the Oil Pumping System of a Variable Speed Scroll Compressor", Proceedings of the International Compressor Engineering Conference at Purdue, West Lafayette, IN,1605