2006

Analysis of Lubricant Sloshing in a Rail Compressor Application

Bellur G. Shiva Prasad
Emerson Climate Technologies/Copeland Corporation

Prasad, Bellur G. Shiva, "Analysis of Lubricant Sloshing in a Rail Compressor Application" (2006). *International Compressor Engineering Conference.* Paper 1735.
http://docs.lib.purdue.edu/icec/1735

ANALYSIS OF LUBRICANT SLOSHING IN A RAIL COMPRESSOR APPLICATION

B. G. Shiva Prasad

Copeland Corporation
Emerson Climate Technologies
Sidney, OH 45365, USA
Ph: (937) 498-3371; E-mail: BGShivaPrasad@Copeland-Corp.com

ABSTRACT

Liquid sloshing is a common industrial problem affecting not only the dynamics of flow inside the container, but also of the downstream systems being fed from the container. This paper deals with the application of Computational Fluid Dynamics (CFD) for analyzing the possible effect of oil sloshing in compressors used in underground rail application on the continuous supply of lubricant to the pump to maintain a good lubrication of the system. Results are presented of a study done to track the oil/refrigerant interface near the inlet tube for some extreme conditions of acceleration, deceleration and track inclination. The study not only helped assess operating condition limits, but the calculated details of the oil motion also provided guidance for design modification for reducing sloshing. It further demonstrated the usefulness of CFD for design, analysis and operation of Positive Displacement Compressors.

1. INTRODUCTION

Liquid sloshing and chaotic free surface motion is a common industrial problem affecting not only the dynamics of flow inside the container and the downstream systems being fed from the container, but also the container itself. A major source of energy for most of the transportation systems, whether it is ground, water or air, is in the form of liquid. It also serves as a lubricant in transportation systems as well as other industrial machinery. In addition, liquids, due to their large specific heat serve as cooling media also. The containers carrying these liquids have to withstand the complex dynamics of the transportation system which they are serving. This inevitable motion of the container and the associated forces on the liquid inside it results in mostly violent and chaotic movement of the liquid/gas (mostly air or vapor) interface or free surface. It could not only make it difficult to monitor the free surface but also disrupt the continuous supply of liquid to the downstream system, whether it is the engine, bearing or the coolant passages, thus leading to system malfunctioning and even damage. This would be catastrophic, particularly in space transportation systems. In addition, the secondary effect of the forces caused by liquid sloshing on the dynamics of the space vehicle itself becomes very important. Hence a large number of studies have been done to address its cause, impact and techniques of mitigation.

Liquid sloshing has drawn attention of aerospace researchers as early as 1950. Graham and Rodriguez (1952) studied the effect of liquid sloshing in airplane fuel tanks on the dynamics of the airplane itself. Abramson (1966) has provided a review of the early analytical and experimental investigations of liquid sloshing. Von Kerczek (1975) has surveyed some numerical models incorporating the phenomenon of Rayleigh-Taylor instability. Hirt and Nichols (1981) developed the volume of fluid (VOF) method for multiphase flows for tracking the interface. Work has also been done for designing flexible structures for controlling sloshing (see Gradinscak, et al., 2002).

The interest and the need for understanding free surface dynamics is not just confined to internal flows in containers, but also exist in piping/tubing used for transporting liquids, and in external flows as in waterborne vehicles as well as air-sea interaction which impacts wave propagation and weather systems. As far as compressors are concerned, liquid sloshing and interfacial dynamics are important not only in those used in transportation systems but also in stationary compressors, due to pressure variation, phase changes and vibration which could occur inside liquid containers. Fluid dynamic analysis of free surface gets complicated not only due to its chaotic behavior but also due to the presence of multiple components and phases. Further, in addition to the forces imposed on the fluid by the external boundary, they are subjected to gravitational or surface tension forces depending on the type and size of the system. Although experimental techniques are available for monitoring free surface motion, the chaotic nature of the motion makes it difficult and intrusive or non intrusive accessibility of measurement volume is always a big impediment for making measurements in any industrial system. Computational Fluid Dynamics (CFD) is an evolving tool which is gaining ground in all industrial applications including positive displacement compressor technology. In the absence of an alternative including experimental technique, it has become the only means to analyze and understand complex fluid dynamic problems. Hence, even with questionable validity, it has started providing useful information for helping in the development and improvement of industrial machinery. This paper deals with the application of Computational Fluid Dynamics (CFD) for analyzing the possible effect of oil sloshing in Compressors used in underground rail application on the continuous supply of lubricant to the pump to maintain a good lubrication of the system.

Compressors are used in railway trains to supply compressed air for brakes, horn, snow blower, and cooling systems, etc. Oil pumps are used for lubrication. Continuous supply of oil to the oil pump and bearings is important to keep the bearings healthy and running. The changing dynamics of railway trains result in oil sloshing inside the oil sump. This study was done to track the oil/refrigerant interface near the inlet tube inside the sump for some extreme conditions of acceleration, deceleration and track inclination. Oil sloshing in compressors involves multicomponent, multiphase and transient effects in addition to the complex internal geometry of the oil sump. These conditions not only complicate CFD analysis, but also make any experimental measurements for monitoring the oil interface level extremely difficult. This study not only helped assess operating condition limits but also provided information for possible design modification of the inlet tube and its location to reduce the effect of sloshing. It also revealed the details of the oil motion which provided guidance for possible control. The study further demonstrated the usefulness of CFD for design, analysis and operation of Positive Displacement Compressors.

2. DESCRIPTION OF ANALYSIS AND TOOL USED

The fluid volume inside the compressor geometry was simplified considering only the large scale features relevant to the problem. Also, since the oil sump was located at the bottom of the compressor shell and the oil level would never rise above the central horizontal plane, only the bottom half of the shell was considered for analysis. The geometry simplification was done in ANSYS Design Modeler and meshing was done using CFX. The simplified

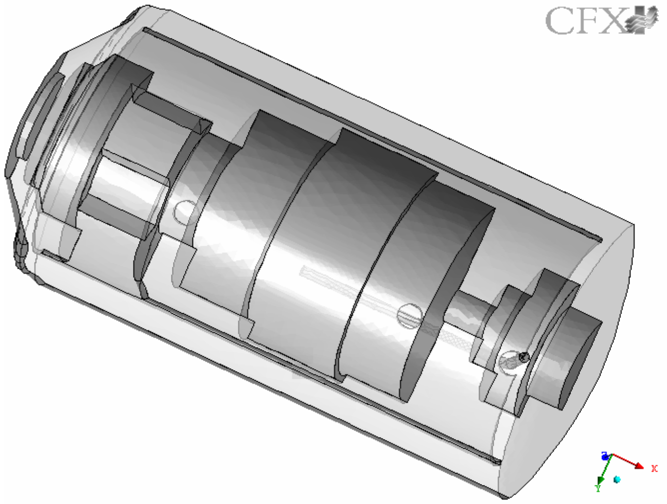

Figure 1: Simplified Model Geometry

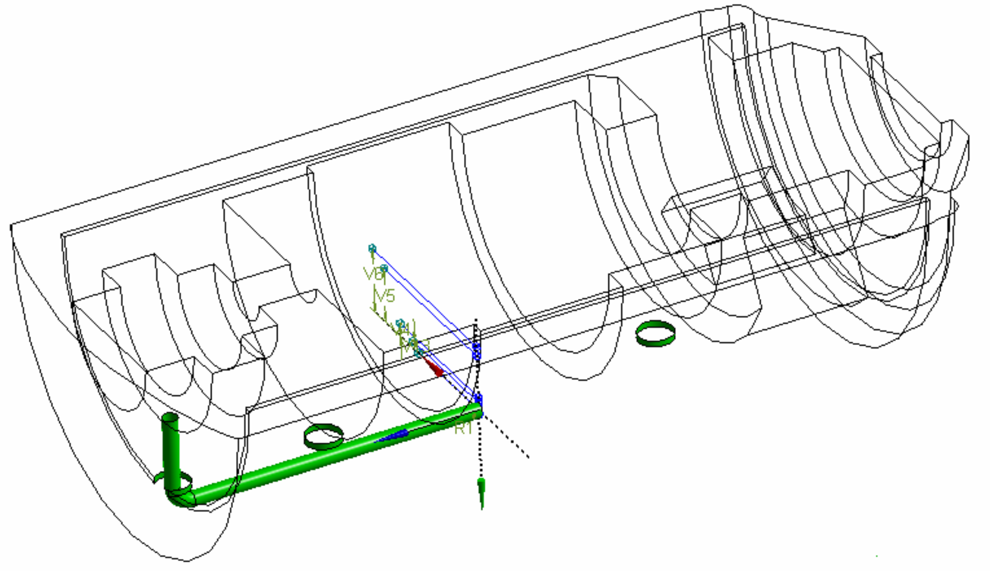

Figure 2: Geometry Showing Oil Inlet Tube and Monitor Point Locations

geometry containing the oil sump, the inlet tube and the monitor points set up along the vertical center line at the tube entrance plane for monitoring the oil level are shown in figs. (1 – 2). Since the main objective was to track the refrigerant-oil interface, and resolution of the fine scale structure of the flow was not required, the mesh was kept coarse enough to get converged solutions in a reasonable amount of time. Even this coarse mesh contained about 250,000 tetrahedrons, prisms and pyramid type elements and needed a minimum of about 3-4 weeks for obtaining a 15 sec transient event simulation. For obtaining good convergence and for limiting the Courant No. to a maximum of 25, the maximum possible time step was limited to about 1e-4 sec for most cases.

ANSYS CFX was used for solution. It uses a finite element based, finite volume method for solving the conservation equations for mass and momentum. It also uses a multiblock approach for speeding up solution time. It was decided to model and solve the flow as a transient problem assuming the body to start from rest followed by the required sequences of acceleration, coasting and deceleration events. Although moving mesh and moving reference frames, could be used for simulation, the transient method appeared to be the best considering the capabilities of CFX and the time required for computation. The moving mesh and reference frames were avoided by keeping the boundary stationary and applying the inertia forces due to acceleration/deceleration directly to the fluid as a source term along the 3 co-ordinate directions. Free surface, multiphase modeling was employed and phase homogeneity was assumed, thus neglecting interfacial mass/momentum transfer. The two fluids corresponding to the two phases considered were the oil and the refrigerant. Since the temperature inside the compressor shell remains constant, there was no need to consider any phase change of the refrigerant. Also, since the temperature surrounding a large region of the oil pump inlet was uniform, temperature gradients and heat transfer effects were not considered as their influence would be minimal.

3. RESULTS AND DISCUSSION

Several cases representing the acceleration, coasting and deceleration phases of the train motion including its motion over an inclined track were simulated. This was done by setting the required acceleration in the X and Z directions for the momentum source term. A zero fluid velocity initial condition was assumed in all the 3 co-ordinate directions. An initial height of the oil above the bottom of the shell was set by setting the oil volume fraction as 1 up to that initial height and zero above, and vice versa for the refrigerant. The basic objective was to track the motion of the oil – refrigerant interface near the entrance of the oil inlet tube for determining whether there was any instant of time during which the inlet tube was starving for oil and if so the length of such intervals and the total duration. For this purpose, monitor points were located (see fig. 2) in the entrance plane near the entrance region of the oil inlet tube as well as near the initial level, for continuous monitoring of the volume fraction of oil (ratio of volumes of oil to the refrigerant) during the solution process. Figure (3) shows the traces of oil volume fraction for one of the extreme cases near the outer limit of operation of the train, consisting of acceleration from 0 to 30 km/hr down a 10˚ incline, followed by coasting for approximately 4.5 sec and then application of emergency braking. The figure shows a continuous fluctuation of the interface near the initial oil level in the sump, while the volume fraction remains at 1 near the entrance of the inlet tube over most of the simulated event. A small reduction in oil volume fraction to 0.8 for a very short interval can be noticed near the beginning of the emergency braking phase. Since even during that short interval, the oil volume fraction remains quite high, it would not have any serious impact on oil supply.

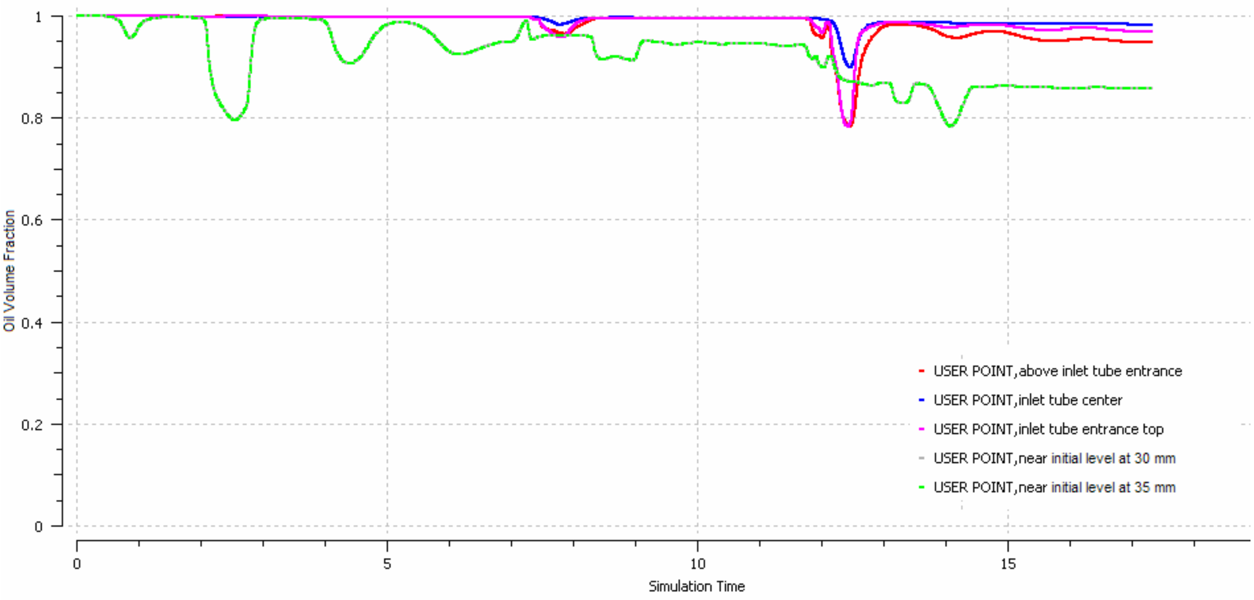

Figure 3: Oil Volume Fraction Variation with Time at 5 Monitored Locations

Figures (4 – 8) show the variation of the volume fraction over the entire volume at some important instants during

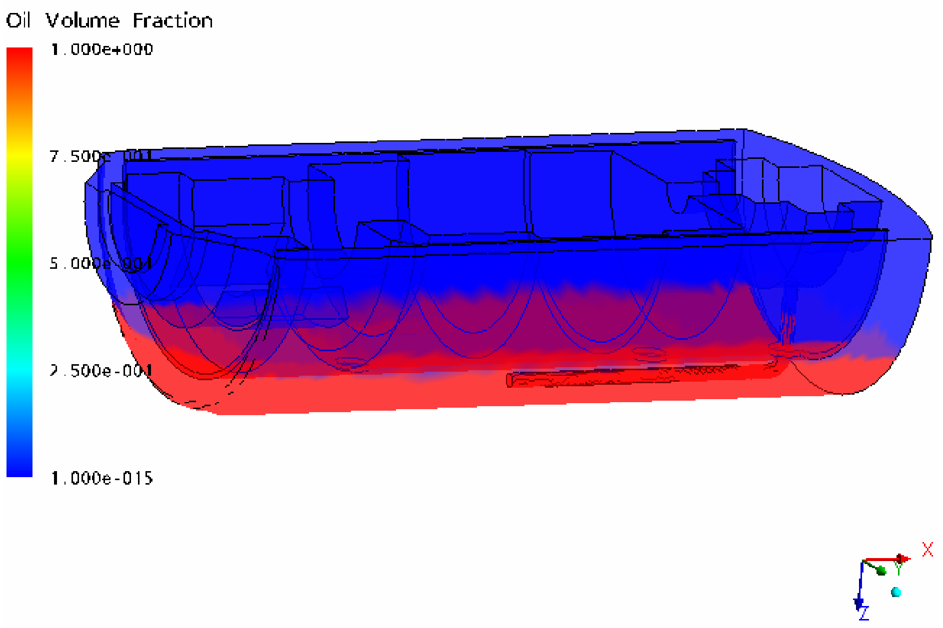

Figure 4: Global Oil Volume Fraction at t = 0 sec

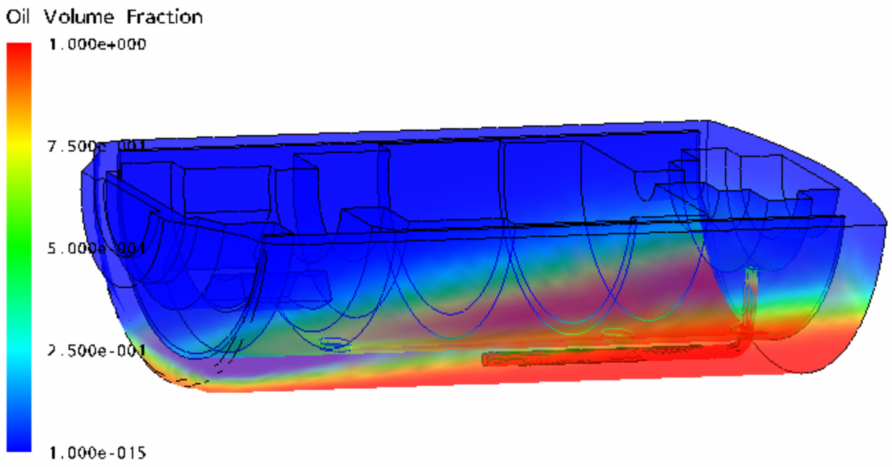

Figure 5: Global Oil Volume Fraction at end of Acceleration (t = 7.01 sec)

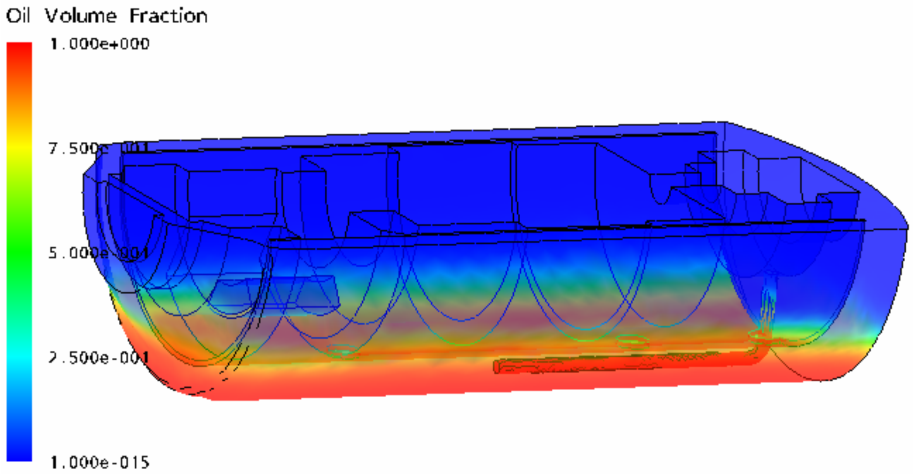

Figure 6: Global Oil Volume Fraction at the end of Coasting (t = 11.68 sec)

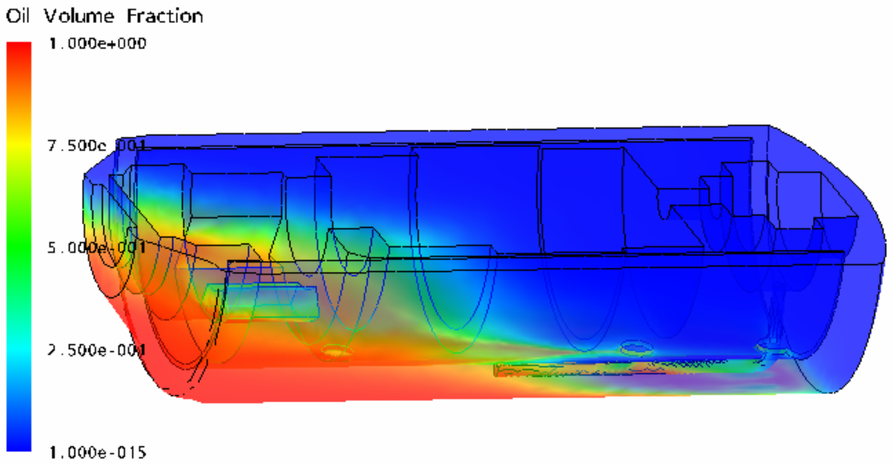

Figure 7: Global Oil Volume Fraction at t = 12.44 sec (during Emergency Braking)

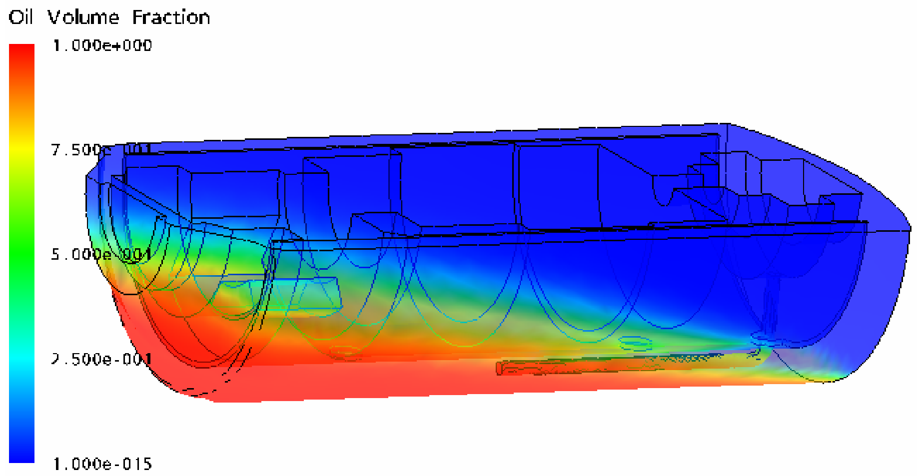

Figure 8: Global Oil Volume Fraction at the end of Emergency Braking (t = 17.33 sec)

the simulation. As the train starts accelerating in the –X direction, the oil sloshes and migrates towards the right side of the sump as shown in fig. (5). After the acceleration was cut off and the train started to coast the oil flowed back in to the sump almost to the original level as shown in fig. (6). When the emergency braking was applied, the violent sloshing caused an instantaneous starvation of the inlet tube near its entrance as shown in fig. (7). Figure (8) shows the oil migrating to the left of the sump at the end of the emergency braking phase. This type of analysis could not only help ascertain whether the oil pump is getting a continuous supply of oil or not, but could also help set the outer limits of acceleration, deceleration, track inclination etc., for safe operation of a train or any other transportation system.

Figure (9) shows a sample picture of some organized motion near the beginning of the emergency braking phase. Consistent with figs. (7) and (8), the velocity vectors show a reasonably fast oil motion from the bottom of the sump to the top in a clockwise direction tending to transport the fluid to the left side of the sump. The analysis of the detailed motion helped in conceiving design modifications which could help reduce oil sloshing and migration of oil towards the ends of the sump.

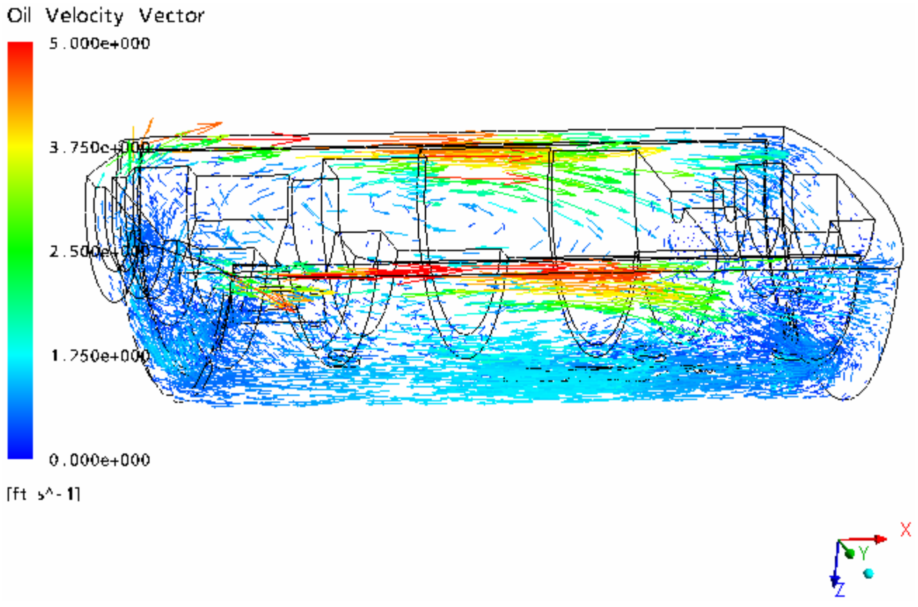

Figure 9: Global Oil Motion at the beginning of Emergency Braking (t = 12.14 sec)

4. CONCLUSIONS

This study helped assess the extent of oil migration if any, away from the entrance of the oil pump inlet tube, due to sloshing caused by the changing dynamics of train motion. It helped ascertain that continuous oil supply was assured during the normal operating conditions studied. It further helped set outer limits for acceleration, deceleration and track inclination beyond which sloshing could cause considerable interruption of the oil supply. The details of the oil motion obtained from the study provided guidance for possible design modification for reducing sloshing and its consequences. This investigation further demonstrated the usefulness of CFD for design, analysis and operation of Positive Displacement Compressors.

REFERENCES

Abramson, H. N., 1966, Dynamic Behavior of Liquids in Moving Containers with Application to Space Vehicle Technology, *NASA-SP - 106* .

Gradinscak, M., Semercigil, S. E., and Turan, O. F., 2002, Design of Flexible Containers for Sloshing Control, *Proc. of ASME Fluids Engrg. Div. Summer Meeting,* Montreal, Canada, Paper No. FEDSM2002-31424.

Graham, E. W., and Rodriguez, A. M., 1952, The Characteristics of Fuel Motion which Affect Airplane Dynamics, *Journal of Applied Mechanics,* Vol. 19, p. 381-388.

Hirt, C. W., and Nichols, B. D., 1981, Volume of Fluid Method for the Dynamics of Free Boundaries, *Journal of Computational Physics,* Vol. 39, p. 201-225.

Von Kerczek, C. H., 1975, Numerical Simulation of Naval Free-surface Hydrodynamics Problems, *Ist Int. Conf. on Numerical Ship Hydrodynamics*, Gaithersburg, USA.

ACKNOWLEDGEMENT

The author appreciates the interest shown and the support provided by Donald C. Draper for the application of CFD. He would like to thank Mark D. Bills for his help in providing all the required data and Steve P. Roberts for his help in generating the geometry. Finally, the author thanks Emerson Climate Technologies’ management for encouraging R & D work and for permitting dissemination of technology.