Evaluation of the Volume-of-Fluid Method for the Numerical Modelling of an Aero Engine Bearing Chamber

Matthias B. Krug, Corina Höfler, Hans-Jörg Bauer  
Institut für Thermische Strömungsmaschinen (ITS)  
Karlsruhe Institute of Technology (KIT)  
76131 Karlsruhe, Germany  
Email: matthias.krug@kit.edu

Wolfram Kurz  
Rolls-Royce Deutschland Ltd & Co KG  
Eschenweg 11, Dahlewitz  
15827 Blankenfelde-Mahlow, Germany

Abstract

This paper presents results of the three-dimensional numerical modelling of the air/oil two-phase flow inside an aero engine bearing chamber. Simulations with different shaft speeds and chamber geometries were carried out and compared to experimental data in order to evaluate the potential and accuracy of the Volume-of-Fluid method for the modelling of complex multiphase flows.

For different bearing chamber scavenge offtake geometries and shaft speeds, the resulting scavenge characteristics, film thicknesses and flow pattern were analysed. Especially for the prediction of the scavenge characteristics, good agreement of numerical and experimental results was obtained. However, the accurate prediction of oil distribution and film thicknesses showed to be sensitive to the mesh resolution. Nevertheless, even with a coarse mesh, correct tendencies were observed.

With the application of a local Adaptive Mesh Refinement algorithm, the prediction of the oil film formation was improved. An additional influence of the surface-capturing scheme within the Volume-of-Fluid method on the solution accuracy has been identified.

Nomenclature

AMR [-] Adaptive Mesh Refinement
b [m] Width
d [m] Diameter
h [m] Height, Thickness
l [m] Length
m [g/s] Mass flow rate
n [rpm] Shaft speed
N [-] Quantity
p [Pa] Pressure
r [m] Radius

SR [-] Scavenge ratio
T [K] Temperature
t [s] Flow time
u [m/s] Velocity
V [l/h] Volume flow rate
VoF [-] Volume-of-Fluid

Greek symbols
α [-] Volume fraction
ρ [kg/m³] Density
η [-] Efficiency
θ [°] Contact angle
μ [Pa s] Dynamic viscosity
φ [°] Circumferential position
ω [rad/s] Angular velocity
σ [N/m] Surface tension

Subscripts
BC Bearing chamber
cell Cells
f Film
g Air
in Inwards
l Oil
sc Scavenge
seal Labyrinth seal
sh Shaft
out Outwards
vt Vent
w Wall

Introduction

Rolling element bearings in modern aero engines have a high demand for oil for lubrication and cooling. In order to supply the required oil to the bearing in modern aero engines, an oil jet is targeted onto the shaft. It is then supplied to the bearings through a feed geometry machined into the shaft.
From the bearing, the oil enters the bearing chamber continuously forming oil films at the chamber walls.

Only a certain amount of the supplied oil is caught and fed to the bearings due to the interaction of the oil jet with both the airflow and the rotating shaft, which causes jet break-up and droplet shedding prior to and at the impact [1]. In order to prevent the oil from leaking out of the chamber, secondary air is extracted at different locations of the compressor to seal the bearing chambers. Hence, a complex two-phase flow of air and oil occurs inside the bearing chamber with a liquid phase consisting of oil supply jets, ligaments, droplets and wall films, which is then scavenged from the chamber to the engine oil system or exits the chamber via the vent.

A lot of effort was put into the experimental investigation of the two-phase flow inside an aero engine bearing chamber in the past. Due to the rotating shaft, the air rotates inside the bearing chamber. Gorse et al. [2] first analysed this air flow by using 3D-LDA measurements and found an influence of both shaft speed and inflow of sealing air on the velocity profiles of the chamber flow. Further investigations focused on the oil after it passed the bearings. Glahn and Wittig [3] characterised the velocity profile inside the oil film flow at the chamber walls, which was found to be driven by a shear force due to the air flow. Gorse et al. [4] observed droplet shedding at the surface of the shear driven liquid wall films. Identifying the operating conditions leading to droplet shedding, Hashmi et al. [5] further investigated the influence of the shear forces at the film surface and proposed, depending on the direction of both gravitational force and rotation, two different flow film regimes.

Moreover, investigations were focused on the oil film thickness at the chamber walls and scavenging characteristics of the two-phase flow. Gorse et al. [6] found an influence of the operating conditions and the chamber geometry on the oil film thickness. Kurz et al. [7] further investigated the influence of chamber geometries on the two-phase flow inside and the scavenging out of the chamber. With more detailed studies, Kurz et al. [8] proposed two different flow regimes inside a bearing chamber as a function of the shaft speed. Additionally, film thickness measurements were carried out for ventless bearing chambers [9]. Kurz and Bauer [10] finally presented an approach for predicting the flow regime as a function of various parameters, such as shaft speed, chamber pressure, oil viscosity and chamber length.

Having a good experimental insight into the overall complex two-phase flow phenomena within an aero engine bearing chamber, a more comprehensive description of certain flow phenomena, e.g., droplet shedding inside the chamber, is desired. This however leads to more complex and expensive experiments. The application of numerical methods in contrast allows a description of the complete flow field or areas of interest, such as droplet-wall film interaction, in much more detail but at comparatively low costs. Here, the Volume-of-Fluid method with its surface-capturing capability is an appropriate tool for the modelling of air/oil system multiphase flows [11].

In order to gain further knowledge regarding the two-phase flow inside the bearing chamber and the efficiency of the oil scavenging, simplified numerical investigations were conducted at the Institut für Thermische Strömungsmaschinen. Hashmi et al. [12] found that in cases of shear-driven wall films the momentum transfer between the phases is over-predicted if a turbulence model is used. An improved method is presented in order to overcome this issue. Peduto et al. [13] adapted these findings on the lower half of an aero engine bearing chamber model. Peduto et al. [14] focused on the droplet-wall film interaction numerically, and discussed the applicability of the standard Volume-of-Fluid method combined with an Adaptive Mesh Refinement algorithm.

This paper aims to evaluate the standard Volume-of-Fluid method for the full three-dimensional modelling of an aero engine bearing chamber. Therefore, different chamber geometries are modelled and influences of shaft speed, mesh size and model parameters are assessed and compared to experimental data. The findings of these investigations are presented in the following. First, the experimental setup used to gain the validation data is described. Afterwards, both the applied numerical method and the derived bearing chamber model as well as the results of the investigations are presented and discussed. The discussion starts with the presentation of the scavenging efficiency of the investigated bearing chamber configurations. Further, the influence of operating conditions and mesh resolution on the film thicknesses and film structures at the chamber walls will be shown.
Experimental Investigation

Wittig et al. [15] first introduced the highly modular test rig for investigations in bearing chamber research at the Institut für Thermische Strömungsmaschinen in Karlsruhe. It allows the variation of a range of bearing chamber geometries and roller bearings and has been used for most of the published work about bearing chamber research in the last two decades. It was also used to gain the experimental validation data for this investigation and therefore is discussed in the following.

A sectional view of the test rig configuration is displayed in Figure 1. A high speed rotor, capable of speeds up to $n_{sh} = 20,000$ rpm, is used to simulate representative aero engine bearing chamber conditions. A roller bearing supports the shaft in radial direction and separates the rig into two bearing chambers. At the very right end, a fixed ball bearing supports the shaft in axial and radial direction. The dashed line highlights the relevant part of the test rig for this investigation. Inserts can be used to modify the inner geometry of the left-hand bearing chamber, which allows the investigation of different kinds of bearing chambers. An under-cage lubrication system supplies the bearing with preheated oil. The chambers are sealed by pressurised labyrinth seals.

Exchangeable scavenge offtake modules allow investigations of a wide range of scavenge geometries, as displayed in Figure 2. The outflows of both air and oil are measured separately for the vent and scavenge offtakes. An oil pump at the scavenge port evacuates the oil from the bearing chamber. As the scavenge pump additionally has to remove a large amount of buffer air, the volumetric flowrate exceeds the capacity of the oil pressure feed pump by a given scavenge ratio $SR$ which is defined as:

$$SR = \frac{\dot{V}_{lsc}}{\dot{V}_{lin}} = \frac{\dot{V}_{lsc} + \dot{V}_{g,sc}}{\dot{V}_{lsc} + \dot{V}_{l,vt}},$$  \hspace{1cm} (1)

with the oil inflow $\dot{V}_{lin}$ and the scavenged flow rates of the buffer air, $\dot{V}_{g,sc}$, and oil, $\dot{V}_{lsc}$. Due to the complex flow patterns within the chamber, an additional oil outflow $\dot{V}_{l,vt}$ occurs through the vent system. The quality of the oil removal can be expressed by the scavenge efficiency [7]:

$$\eta_{sc} = \frac{\dot{V}_{lsc}}{\dot{V}_{lin}} = \frac{\dot{V}_{lsc}}{\dot{V}_{lsc} + \dot{V}_{l,vt}}.$$  \hspace{1cm} (2)

Windows incorporated into the front plate allow the optical access to the bearing chamber and the visual observation of the two-phase flow in the bearing chamber. Film thickness sensors measure the circumferential oil film distribution at the chamber walls. Figure 3 shows the positions of these sensors. Three sensors are located at the circumferential position $\phi = 135^\circ$ for a measurement of the axial distribution of the film thickness. A more comprehensive description of the applied capacitive sensors with both calibration and measurement uncertainties is given by Kurz et al. [9].

Numerical Investigation

Numerical Method

The numerical modelling of the complex air/oil two-phase flow within the chamber is achieved by applying the surface-capturing Volume-of-Fluid
(VoF) method [16] to a block structured Eulerian mesh. For surface-capturing a geometric reconstruction scheme is applied for most of the simulations, which is based on the piecewise-linear approach (PLIC) proposed by Rider and Kothe [17]. Wall adhesion and surface tension are taken into account by applying the Continuum Surface Force (CSF) model as developed by Brackbill et al. [18].

The solver (ANSYS Fluent® Release 13.0 and later) uses the Pressure-Implicit with Splitting of Operators (PISO) pressure-velocity coupling scheme. Second order accuracy is enabled for the spatial discretisation of the momentum transport equation and of both kinetic energy and dissipation rate of the applied turbulence models. Due to the application of the geometric reconstruction scheme first order temporal discretisation is needed.

Table 1: Geometry data for model construction

<table>
<thead>
<tr>
<th>Type</th>
<th>Value [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chamber height</td>
<td>$h_{BC}$</td>
</tr>
<tr>
<td>Chamber width</td>
<td>$l_{BC}$</td>
</tr>
<tr>
<td>Rotor radius</td>
<td>$r_{sh}$</td>
</tr>
<tr>
<td>Vent pipe diameter</td>
<td>$d_{vt}$</td>
</tr>
<tr>
<td>Scavenge pipe diameter</td>
<td>$d_{sc}$</td>
</tr>
<tr>
<td>Labyrinth gap length</td>
<td>$l_{seal}$</td>
</tr>
<tr>
<td>Labyrinth gap height</td>
<td>$h_{seal}$</td>
</tr>
<tr>
<td>Height oil inlet</td>
<td>$h_{lin}$</td>
</tr>
</tbody>
</table>

Numerical Model

In order to investigate the air/oil two-phase flow within the bearing chamber using a finite volume method, a numerical model of the experimental test rig as shown in Figure 1 needs to be generated. The model geometry is displayed in Figure 4 as a configuration including the covered ramp scavenge offtake as displayed in Figure 2(b). Table 1 summarises the geometrical data of the model. The labyrinth seal ① is modelled as an annular gap with the height $h_{seal}$ and the length $l_{seal}$ between chamber and rotor. The length of the gap including the part in the chamber itself totals to 9 mm. Inner and outer diameter of the resulting geometry are defined by the rotor radius $r_{sh}$ and the chamber height $h_{BC}$, respectively. The actual roller bearing was replaced by an annular gap ② with the height $h_{lin}$, which represents the oil inlet. The vent pipe ③ is located at the very top of the geometry with the vent diameter $d_{vt}$. At the bottom of the model, the scavenge offtake ④ with pipe diameter $d_{sc}$ is located. The axis of rotation is along the $x$-axis. Rotation direction is
defined as clock-wise with view on the bearing-sided chamber wall in the following.

Figure 4 also shows a sectional view of the block structured mesh after domain discretisation. Since the main core airflow sees no major changes and behaves as a rotating flow, the grid resolution was kept coarse in this area but fine enough to resolve the velocity profile of the flow adequately. The model regions where an oil film is expected, such as the wall at the outer diameter and the bearing-sided chamber wall, were resolved with a considerably finer mesh to ensure a more accurate calculation of the film.

**Boundary Conditions**

The boundary conditions at both the inlets and outlets are displayed in Table 2 and correspond to the experimental operating conditions. Similar to the approach proposed by Peduto et al. [13], the buffer air inflow, set to \( \dot{m}_{g,\text{in}} = 10 \, \text{g/s} \), is imposed with a swirl condition depending on the shaft speed. This is valid as the labyrinth seal accelerates the buffer air in rotational direction. However, the appropriate set-up of the swirl condition is crucial as the buffer air highly influences the chamber core flow [2]. The oil is supplied to the chamber as a film via an annular gap at a constant volume flow rate of \( \dot{V}_{\text{lin}} = 100 \, \text{l/h} \).

Based on the investigations of Gorse et al. [19], this is a valid approach as droplet shedding at roller bearings occurs only for strong air crossflows. As with the buffer air before, a swirl condition based on the present shaft speed is imposed on the supplied film. This is an assumption as not much is known about the actual velocity components of the oil film after it left the bearing. Kurz and Bauer [10] for instance observed that both tangential and radial component of the oil flow depend more on the actual oil flow rate through the bearing than on the present shaft speed.

In order to impose a pressure of \( p_{\text{BC}} = 2.7 \, \text{bar} \) within the chamber, a pressure condition is set up at the vent outlet, \( p_{\text{out,vt}} \). The velocity condition \( v_{\text{out,sc}} \) at the outlet of the scavenge pipe results in the set-up of an evacuated air/oil volume flow of \( \dot{V}_{\text{lin}} = 400 \, \text{l/h} \), which corresponds to a scavenge ratio of \( \text{SR} = 4 \).

The numerical simulations are carried out at chamber operating conditions of \( p_{\text{BC}} = 2.7 \, \text{bar} \) and \( T_{\text{BC}} = 373 \, \text{K} \). Hence, in order to match experimental conditions, the fluid properties density and viscosity of oil and air are set to \( \rho_i = 929.5 \, \text{kg/m}^3 \) with \( \mu_i = 4.83 \, \text{mPa s} \) and accordingly \( \rho_g = 2.52 \, \text{kg/m}^3 \) with \( \mu_g = 22.1 \, \mu\text{Pa s} \). Surface tension at the phase interface is \( \sigma_i = 24.5 \, \text{mN/m} \).

**Numerical Range**

The evaluation of the VoF method for the modelling of an aero engine bearing chamber is carried out for the cases displayed in Table 3. At a constant oil inflow rate \( \dot{V}_{\text{lin}} \) and under engine relevant chamber conditions for \( p_{\text{BC}} \) and \( T_{\text{BC}} \), six different cases are investigated. For two different shaft speeds, \( n_{\text{sh}} = 5,000 \, \text{rpm} \) and \( n_{\text{sh}} = 15,000 \, \text{rpm} \), the scavenge offtake configurations as displayed in Figure 2 are simulated. Additionally, in order to identify influences of mesh resolution, simulations are carried out at a constant shaft speed of \( n_{\text{sh}} = 5,000 \, \text{rpm} \) with two different mesh sizes, \( N_{\text{cell}} = 621,500 \) and \( N_{\text{cell}} = 1,700,000 \). As the covered ramp scavenge offtake slightly increases the chamber volume, the resulting mesh size needs to be increased. However, the resolution of areas such as chamber core and oil covered walls remains similar to the mesh with \( N_{\text{cell}} = 621,500 \), with the smallest cell size of \( \Delta x = 0.2 \, \text{mm} \) in proximity to the oil-covered walls.

In order to investigate the capability of an Adaptive Mesh Refinement (AMR) in combination with the VoF method and its applicability to modelling complex two-phase flows in bearing chambers, case F is set up. With the covered ramp scavenge offtake and a constant shaft speed of \( n_{\text{sh}} = \)

<table>
<thead>
<tr>
<th>Table 2: Boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Case</strong></td>
</tr>
<tr>
<td>Buffer air</td>
</tr>
<tr>
<td>Oil supply</td>
</tr>
<tr>
<td>Vent pipe</td>
</tr>
<tr>
<td>Scavenge pipe</td>
</tr>
<tr>
<td>Bearing chamber</td>
</tr>
<tr>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3: Investigated cases</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Case</strong></td>
</tr>
<tr>
<td>A</td>
</tr>
<tr>
<td>B</td>
</tr>
<tr>
<td>C</td>
</tr>
<tr>
<td>D</td>
</tr>
<tr>
<td>E</td>
</tr>
<tr>
<td>F</td>
</tr>
</tbody>
</table>
5,000 rpm a simulation is conducted applying AMR in accordance with the gradient of the volume fraction $\alpha$ of the liquid phase. This approach leads to a variable mesh size with a minimum cell size of $\Delta x = 25 \, \mu m$ at the phase interface. The applied AMR method with all its settings is described in detail in Peduto et al. [14] and Peduto [20].

An adaptive time-stepping based on a global CFL number around 3 is used for the simulations of cases A - E, leading to a maximum average time step size of $\Delta t = 1.5e^{-5} \, s$. As the oil film at the walls moves relatively slow compared to the main core air flow, this time step size is sufficiently small to resolve the overall oil motion properly within the bearing chamber. For case F, an average time step of $\Delta t = 3.25e^{-6} \, s$ is set up due to the smaller cell size and applied CFL number smaller than 1.

Results and Discussion

Scavenge Efficiency

The numerical predictions of the scavenge efficiency in comparison with the experimentally determined results for different shaft speeds and scavenge offtake geometries are shown in Figure 5. The shaft speed is of major importance for the formation of the secondary flow field and hence for the formation of the oil film, which eventually affects the oil removal due to its axial and circumferential position and velocity. Higher shaft speeds lead to an increased momentum transfer between airflow and oil film and gravity becomes less important. As a result, more oil is dragged towards the vent offtake, where it is then removed. Hence different flow regimes develop [8]. The design of the covered ramp offtake (s. Figure 2(b)) in contrast to the baseline configuration (s. Figure 2(a)) copes with the effects of increased shaft speeds, which results in an almost stable and higher scavenge efficiency over the investigated range of shaft speeds. For high shaft speeds, $\eta_{sc}$ remains almost constant at a high level as the covered ramp reduces the influence of the primary airflow field at the scavenge offtake and thereby results in a relatively slow moving oil film which is directed straight into the scavenge pipe.

The numerical prediction of the scavenge efficiencies for both baseline and covered ramp offtake shows good agreement with the experimentally determined data. However, a slight deviation of numerical and experimental data for high shaft speeds is observed, especially for the baseline configuration. An explanation for this behaviour might be an over-predicted wall film velocity, due to the false momentum transfer as mentioned by Hashmi et al. [12] and Peduto et al. [13]. The efficiency of the covered ramp design, however, is not influenced by these higher velocities, as mentioned.

Wall Film Development – Film Thickness

Figure 6 displays the film thicknesses for the two investigated scavenge offtake geometries at a shaft speed of $n_{sh} = 15,000 \, rpm$. The time-averaged film thicknesses $\overline{h_f}$ were taken at the eight axially centred sensor positions (s. Figure 3), $n_{sh}$ indicates the rotational direction of the shaft. At low shaft speeds, the variation of the offtake geometry has only a negligible influence on the formation of the film thicknesses as already shown by Kurz et al. [8]. The dominating gravitational force leads to an increased film thickness at the bottom of the chamber at $\varphi = 180^\circ$. However, significant differences between the
designed to occur at higher shaft speeds. The oil split directly affects the residence oil volume and hence the film thicknesses. Since the covered ramp efficiently collects and removes the oil, the average film heights are comparatively small. In addition, the oil distribution is of uniform nature in circumferential direction. In contrast, the reduced scavenging efficiency of the baseline offtake is obvious and represented by increased film thickness.

By applying the VoF method for the modelling of bearing chamber wall films, the correct tendencies were predicted for both offtake geometries. For the covered ramp offtake (case E), the numerically predicted values are in good agreement with the experimentally determined ones. Major deviations occur for the baseline offtake geometry (case B). Especially at the counter-current chamber side ($\varphi < 180^\circ$) the film thickness is overestimated. At the counter-current chamber side shedding of ligaments occurs due to the subcritical nature of the film [5]. The separated ligaments are either vented from the chamber or enhance the oil film at the co-current chamber side ($180^\circ < \varphi < 360^\circ$). However, due to the coarse mesh, the ligament size is over-predicted and hence no reliable information can be obtained with regard to the influence of the ligaments on oil film formation. High fluctuations of the film thicknesses at the measurement positions lead to considerable deviations from the time-averaged value. An additional influence on the oil film prediction might be caused by the false momentum transfer between the phases. If the momentum transfer is over-predicted too much oil is transferred from the co-current to the counter-current chamber side which leads to an overestimated oil film thickness. Nevertheless, the tendency of the experimentally determined film thicknesses is matched in both cases but differs in accuracy due to coarse mesh resolution. The influence of mesh resolution is therefore discussed in the following.

Wall Film Development – Influence of Mesh Resolution

The influence of a global mesh refinement on the film thickness at $n_{\text{sh}} = 5,000$ rpm for the baseline scavenging offtake configuration is displayed in Figure 7. A comparison between experimental results and results from simulations with 621,500 cells (case A) and 1,700,000 cells (case C) domain resolution without AMR is drawn. The influence of mesh resolution on wall wetting is obvious but not as expected. Due to the finer mesh, case C on one hand resolves the oil film in a more accurate manner but on the other hand, the film is disrupted in smaller pieces and hence the oil covered surface area of the bearing chamber decreases, as displayed in Figure 8(b). Wall adhesion effects might be a reason for these results. For case A (s. Figure 8(a)), the oil film is rather smeared over the chamber walls due to insufficient grid resolution in tangential direction. However, the smearing of the oil film and hence numerical diffusion seems to falsely increase the predicted film thickness for $255^\circ < \varphi < 315^\circ$.

Figure 8(c) shows the influence of a local AMR (case F) based on the gradient of the volume fraction of the liquid phase. Here, a completely different wall film structure occurs. On the one hand, smaller flow features such as oil droplets, ligament shedding and film waves are predicted due to the much smaller cell size, leading to a mesh size of around $N_{\text{cell}} = 7,000,000$. On the other hand however, the wetting of the chamber walls differs from the ones observed in cases A and C, considerably. A reason for this behaviour might be the solver settings necessary for the adaptive mesh refinement. Based on Peduto [20], an implicit compressive scheme is used for the discretisation of the volume fraction $\alpha$ with case F. Cases A and C in contrast use an explicit geometric reconstruction scheme for surface-capturing. This difference in solver settings could influence the capturing of the phase interface in a way that the prediction of the oil wall film structure is strongly affected.

However, a comparison of the oil film formation according to Figure 8 and experimental data as shown in Figure 9 reveals that the prediction
of the oil film formation of case F represents the oil film flow phenomena observed during experiments. The experiments were carried out on the test rig described by Krug et al. [1] at the Institut für Thermische Strömungsmaschinen ensuring comparable boundary conditions. Here, the supply of the required oil is provided by a system consisting of an oil jet targeted onto the shaft, which is then supplied to the bearings through a feed geometry machined into the shaft. The oil is then fed to the bearing through feed holes via a combination of under-cage and under-race lubrication and enters the chamber through the bearing. Figure 9 displays the oil film at the bearing-sided chamber wall.

The oil leaves the aero engine ball bearing as a closed film without any non-wetted spots. Due to wall adhesion and gravity, the rotational momentum of the oil decreases quickly. The oil film expands horizontally and flows down the bearing-sided wall vertically. Above the bearing, the interaction of oil and gravitational forces leads to a flapping of the oil film and hence ligament shedding out of the film. Wavy structures can be identified on the film surface while it is flowing down the wall. All the effects mentioned are observed as well for the prediction of the oil wall film formation in case F, as displayed in Figure 9 on the right. Additionally, small droplets are identified in the main air core flow, which interact with the rotating oil wall film. Further refinement of the phase interface would allow the investigation of droplet-wall film interaction as discussed by Peduto et al. [20], but would also increase the numerical effort as shown in the next section. Cases A and C, however, fail to predict the proper oil film due to reasons mentioned above.

**Numerical Effort**

An important aspect of numerical investigations is the computational effort of the flow simulations. Considering case A with an average time step of $\Delta t = 1.41e-5$ s, the computational effort for calculating a physical flow time of $t = 1$ s equals to 1,500 CPUh/s which corresponds to 48 time steps per CPUh. Increasing the mesh size globally for case B, the computational costs are increased to 4,000 CPUh/s, corresponding to 16 time steps per CPUh with an average time step of $\Delta t = 1.48e-5$ s. The application of local AMR increases mesh size, up to $N_{cell} = 7,000,000$, and computational costs considerably up to 63,000 CPUh/s, corresponding to 5 time steps per CPUh with an average time step of $\Delta t = 3.25e-6$ s. However, it yields physically reasonable results.

**Conclusions**

Numerical investigations were carried out in order to assess the Volume-of-Fluid method for the three dimensional modelling of complex two-phase flow phenomena in an aero engine bearing chamber. For different shaft speeds and scavenge offtake geometries the numerical results were compared with experimental data with regard to scavenge efficiency and oil film formation. Additionally, the numerical effort was analysed.
The results of the previous sections revealed that the standard formulation of the VoF method provides promising results as it calculates the correct tendencies of the flow behaviour for both gaseous and liquid phase. Especially for the prediction of scavenge efficiencies, proper values were obtained for a wide range of shaft speeds and scavenge offtake geometries. However, some other issues were identified. Oil dry-out at the chamber walls and a deviation in film thicknesses were observed. They may be a result of a too coarse mesh and the shortcoming of the VoF method when it comes to the calculation of shear driven liquid wall films in a combination with a turbulence model. Nevertheless, the tendencies of the experimentally determined film thicknesses were predicted correctly. A strong influence of the mesh size, and hence resolution of the phase interface, on the appropriate prediction of the wall film formation was identified.

The resulting scavenge characteristics proofed to be in good agreement with the experimental findings and could be yield with the standard VoF method under comparatively inexpensive calculations. In order to predict proper oil film formation the numerical effort needed to be increased considerably due to the much finer mesh resolution at the phase interface required.

Further research is necessary in order to investigate the influences of the model parameters, which were identified in this study, such as surface-capturing scheme, shear stress correction and wall adhesion, more comprehensively.

This study showed that, for a deeper insight into the oil wall film formation, the application of local Adaptive Mesh Refinement is required. The coupling of the Volume-of-Fluid method for wall film modelling and an Euler-Lagrange technique for droplet tracking in combination with a droplet-wall film interaction model [20] could be a promising approach to provide a numerical method for the modelling of complex two-phase flow phenomena in an aero engine bearing chamber. If the mesh resolution can be kept fine enough to resolve the film with VoF-AMR, but coarse enough for droplet tracking via Euler-Lagrange, this approach can be seen as well-balanced between computational effort and solution accuracy. Results with a proper prediction of the overall air/oil two-phase flow within the chamber can be achieved with a relatively low resolution applying the standard VoF method.

Acknowledgements

The research leading to these results has received funding from the European Commission’s Seventh Framework Programme within the research project "Engine Lubrication System Technologies" (ELUBSYS) under grant agreement No. ACP8-GA-2009-233651. This financial support is gratefully acknowledged.

References


Anemometer (LDA) Measurements in an Aero-Engine Bearing Chamber”, ASME Paper No. GT2003-38376


