IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

Specific Aspects in Numerical Simulation of Complex Processes in Gas Turbine Engine Bearing Chamber

To cite this article: T P Mykhailenko *et al* 2022 *IOP Conf. Ser.: Mater. Sci. Eng.* **1226** 012038

IOP Conf. Series: Materials Science and Engineering 1226 (2022) 012038 doi:10.1088/1757-899X/1226/1/012038

Specific Aspects in Numerical Simulation of Complex Processes in Gas Turbine Engine Bearing Chamber

T P Mykhailenko, A Yu Lysytsia, I I Petukhov and A V Kovalov

National Aerospace University "Kharkiv Aviation Institute", Kharkiv, 61070, Ukraine

E-mail: t.mykhailenko@khai.edu

Abstract. The complex interrelation of thermal and hydraulic processes in a gas turbine engine bearing chamber requires modelling methods based on the multiphase flow mechanics and Computational Fluid Dynamics (CFD) to predict the fluid distribution and heat transfer phenomena. This paper presents a study of different approaches to CFD modelling of multiphase oil-air flow in the bearing chamber. The Volume of Fluid and Eulerian multiphase models, Steady and Transient solvers, "Realizable k-ε" and "k-ω SST" turbulence models were analysed. The Eulerian Wall Film Model implemented in ANSYS Fluent was applied to model an oil film formation on the bearing chamber walls. The CFD results were compared with available experimental data to formulate practical recommendations for precise modelling of processes in the bearing chamber.

1. Introduction

The necessity to develop environmentally friendly and efficient gas turbine engines (GTE) forces designers to increase the thermodynamic cycle parameters while focusing on more compact and integral design. These trends significantly complicate the bearing chamber design process. On the one hand, the heavier thermally stressed engine state, higher rotational speeds, temperatures, pressures and velocities of airflow moving through the gas path present challenges for bearing chamber operation and lubrication oil protection against the temperature impact. On the other hand, minimisation of power consumption for friction zone lubrication and air consumption for bearing chamber pressurisation and protection from the hot gases are critical to ensure the overall fuel efficiency of the gas turbine engine.

Correct operation of lubrication and sealing air systems helps avoid the bearing overheating, which may lead to the failure, and ensure the lubricating oil safe temperature conditions. Only a deep understanding of the heat transfer processes in the bearing chamber within the entire range of the engine operating conditions can bring the designers to a rational solution.

The GTE bearing chamber is filled with a mixture of lubricating oil and air that leaks through the seals. The bearing and the shaft rotation generates the oil droplets. They interact with the airflow and form the oil film on the chamber wall. The oil film splashes generate the secondary droplets, which further interact with the flow core. As a result of these continuous processes, a transition region is formed near the wall, where the oil becomes a continuous phase. The flow structure in this near-wall region is very complex, and the oil film thickness is nonuniform [1, 2]. This non-uniformity affects the heat transfer between the bearing chamber wall and the air-oil mixture [3, 4].

The high intensity of the heat transfer in the multiphase flow of variable structure and the interrelation of thermal and hydraulic processes determines the complexity of the fluid distribution and heat transfer prediction problem. Application of the modelling methods based on the mechanics of multiphase flows and Computational Fluid Dynamics (CFD) is a promising solution. Many studies [1, 5-10] demonstrate the relevance of the fluid distribution and heat transfer prediction problem and modelling approach application. This approach fundamentally allows taking into account the influence of all factors in the mathematical model. However, the ambiguities associated with the CFD model structure and settings and boundary conditions specification complicate accurate prediction. Therefore, a combination of modelling methods and experimental studies is crucial to resolve the prediction problem.

Experimental studies [2-4, 11, 12,] contributed significantly to understanding the bearing chamber processes. However, these studies were performed for bearing chambers of simplified geometries. Besides, there are only a few generalised dependencies, while the similarity criteria used do not consider the interfacial interaction effects, especially the wall oil film formation and the droplet diameter influence.Therefore, their applicability requires proper verification even for geometrically similar bearing chambers.

Based on the existing experimental data, this study aims to resolve a problem of correct selection of multiphase flow model structure and boundary conditions, modelling method and mesh parameters that, cumulatively, influence the computation results and time.

2. Methodology

Complications during simulation of heat transfer processes in the GTE bearing chamber are caused by the fluid multiphase nature and the mixture structure dissimilarity. In the flow core region, the continuous medium is air, while oil becomes a continuous phase in the near-wall region. Therefore, the computation domain should be divided into separate flow core and near-wall regions.

Heat transfer from the flow core region to the near-wall region is realised via air motion and droplet deposition. The heat transfer coefficient (HTC) in the bearing chamber determined based on the Newton-Richman law (1) is widely used to represent the heat transfer data.

$$h_{ci} = \frac{q_w}{\left(T_{nw} - T_{wi}\right)}, \tag{1}$$

where T_{nw} is a fluid near-wall temperature, T_{wi} is an internal wall temperature, q_w is a wall heat flux density.

The correct choice of the temperature that characterises the near-wall temperature is an important and ambiguous issue for heat transfer coefficient determination. This choice influences HTC values observed in the studies, in which either medium temperature measured at some distance from the chamber wall or scavenge mixture temperature was used.

The analysis shows that mass-average temperature can be quite correctly used as the near-wall temperature T_{nw} for HTC determination. The temperature T_{mi} of the oil-air mixture after the bearing can be selected as such temperature that determined by mixture enthalpy h_{mi} according to the equation

$$h_{mi} = h_l Y_l + h_a (1 - Y_l) + Q_{br} / (\dot{m}_l + \dot{m}_a), \tag{2}$$

where the oil mass fraction is

$$Y_l = \frac{\dot{m}_l}{\dot{m}_l + \dot{m}_a}, \tag{3}$$

where $\dot{m}_l, \dot{m}_a$ are mass flow rates of oil and air, respectively.

The intensity of interphase transfer in the flow core region is determined by the droplet sizes and differences in the temperatures and velocities of the phase. Two models can be used to determine the parameters of droplets in the near-wall region: homogeneous and inhomogeneous. The inhomogeneous model (for example, the Eulerian multiphase model implemented in ANSYS Fluent) allows the modelling of multiple separate yet interacting phases. An Eulerian treatment is used for each phase. The homogeneous model (for example, the Volume of Fluid model implemented in ANSYS Fluent) considers two or more immiscible fluids by solving a single set of momentum equations and tracking each fluid volume fraction throughout the domain. The inhomogeneous model allows to determine the parameters more accurately but uses a higher number of equations and boundary conditions as well as requires more powerful computation capacities.

The inhomogeneous model based on Euler's approach and two-phase flow structure inversion algorithm [13] can be used to predict processes in the near-wall region. This approach assumes that one of the phases is continuous (phase p) and the other is dispersed (phase q). The area of the interphase surface per unit of the mixture volume, known as the interfacial area density, is calculated from the following formula

$$A_{pq} = \frac{6\tilde{\alpha}_q}{d_q} \tag{4}$$

assuming that the dispersed phase is present as spherical particles of mean diameter d_q.

For model operability in the whole diapason of the volume fractions of the dispersed phase, the limits (α_{max}=0.8, $\alpha_{\min}$=10^{-7}) were imposed, and the area density (4) is defined following the algorithm:

$$\tilde{\alpha}_q = \left\{ \begin{array}{lll} max(\alpha_q, \alpha_{min}) & if & (\alpha_q \le \alpha_{max}) \\ max\left(\dfrac{1-\alpha_q}{1-\alpha_{max}} \alpha_{max}, \alpha_{min} \right) & if & (\alpha_q > \alpha_{max}) \end{array} \right\} \tag{5}$$

When the volume fraction α_q of the dispersed phase tends to zero, the lower limit of volume fraction $\alpha_{\min}$ is used to avoid the convergence problems. In the case of high volume fraction α_q, the condition of the q phase existence as a dispersed phase may be violated, which corresponds to the inversion of the two-phase flow structure. Then, the area density is decreased to reflect that it should lead to zero as α_q tends to 1.

Even if the flow structure inversion algorithm (5) is used, the wall film modelling remains practically unsolvable, while the wall film thermal resistance is decisive in the heat transfer process. Some progress in this direction is associated with the application of the Eulerian Wall Film (EWF) model that can be used to predict the thin liquid films formation and behaviour.

2.1. Object of study

A computational domain of an analysed bearing chamber (Figure 1) represents a working volume of simplified bearing chamber II discussed in [3, 4]. Chamber II is formed by a rotating shaft and a fixed housing. The heated oil is fed to a roller bearing for lubrication purposes; part of the oil goes into chamber II. The air comes into the bearing chamber through the labyrinth seal. The air/oil mixture is removed through venting and scavenge ports located at the top and bottom of the chamber, respectively. The boundary conditions specified in [3, 4] and presented in Figure 2 were used for CFD modelling.

The mesh convergence process was preliminary studied to run an accurate CFD modelling of the analysed bearing chamber. A structured mesh of 0.7 million elements (Figure 3a) was chosen based on the results received. The mesh has a minimal step (0.04 mm) near the walls and a maximum step (4 mm) within the main computation volume.

To study film formation processes using the Eulerian Wall Film model, two variants of meshes without interfaces were created and analysed. After preliminary calculations, a mesh consisting of 2 mln tetra and hexa elements (Figure 3b) was chosen. The main characteristics of the mesh are the following: Defeature Size: 0.0195 mm; Curvature Min Size: 0.13 mm; First Layer Height: 0.1 mm; Inflation Growth rate: 1.05; Maximum Layers: 10.

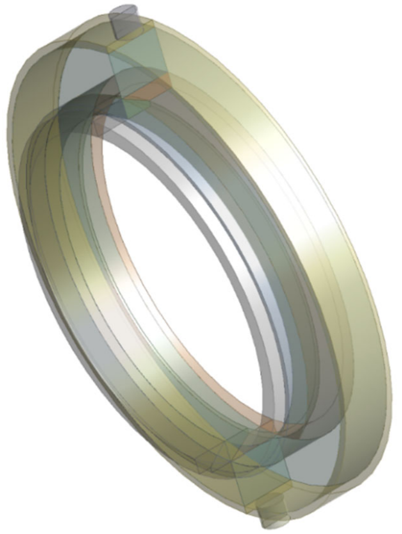


Figure 1: Geometric model.

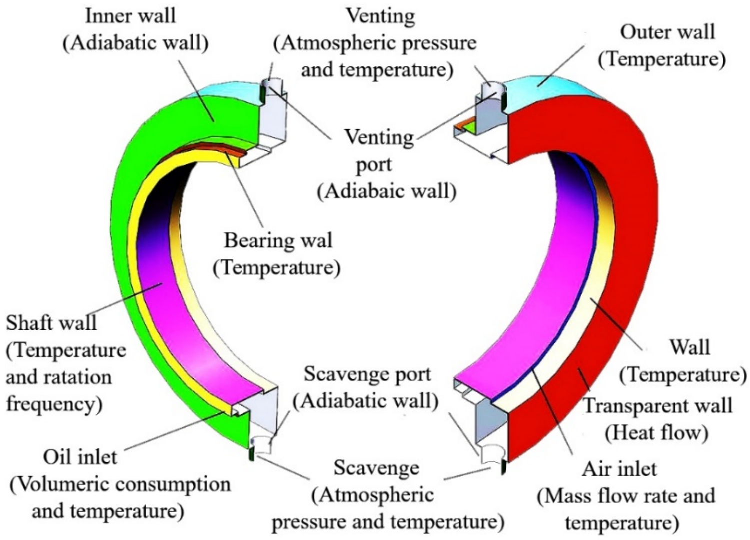
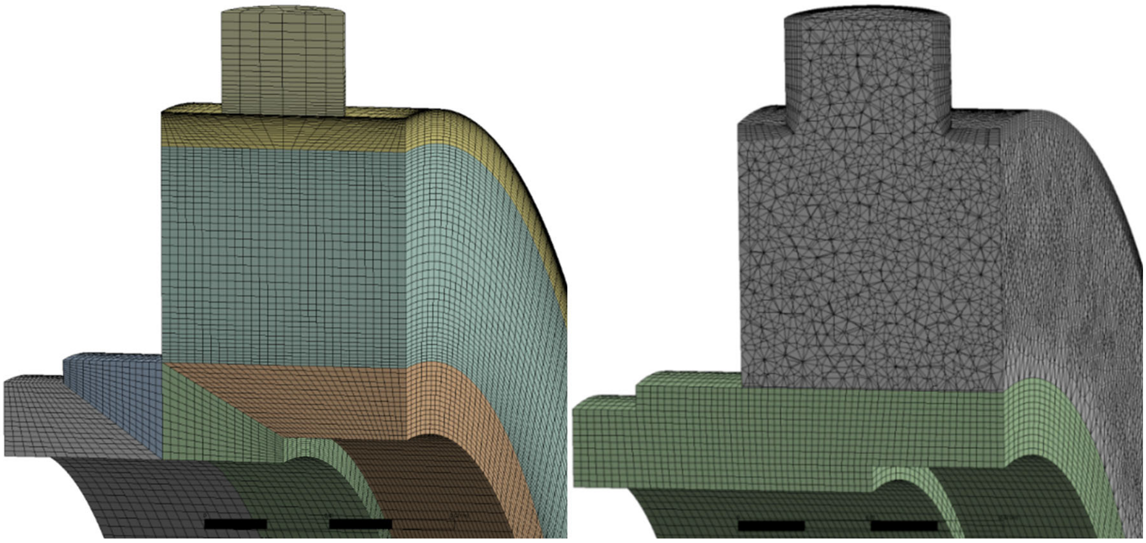


Figure 2: Boundary conditions.



a) b)

Figure 3: Mesh for the bearing chamber: a - hexa; b - tetra/hexa.

To optimise the computational time, a fine mesh was created only on the inner surface of the bearing chamber housing, where the heat transfer coefficient and oil film were studied. Additional calculations showed that the absence of a boundary layer on the remaining surfaces has a minimal effect on the flow distribution near the chamber wall.

2.2. Methodology validation

The numerical simulation was carried out using the ANSYS Fluent. The Eulerian based CFD results and the experimental data [3, 4] are presented in Figure 4. Comparative analysis shows that the difference between the CFD results and experimental data does not exceed 12% for most of the points.

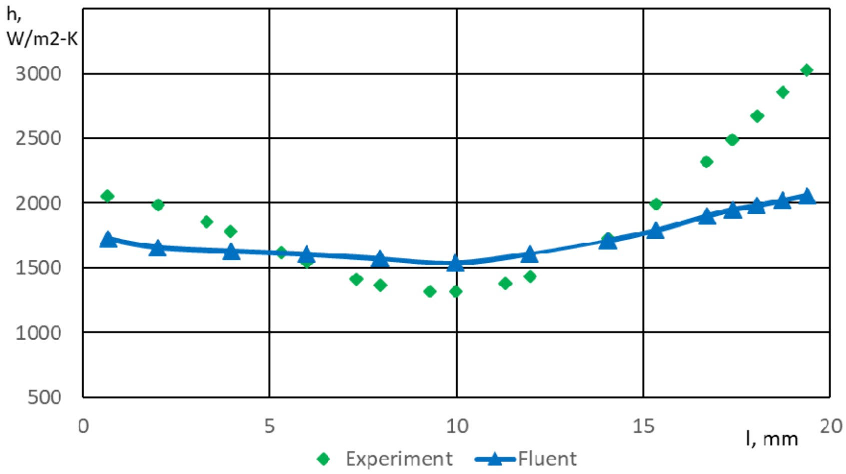


Figure 4: Heat transfer coefficient in section θ=15°.

To predict the oil film distribution, the VOF model in combination with the EWF model was used. Gravity force and surface shear force were chosen for momentum options. A comparison of the obtained CFD results and experimental data of Kurz W. et al. [12] is shown in Figure 5.

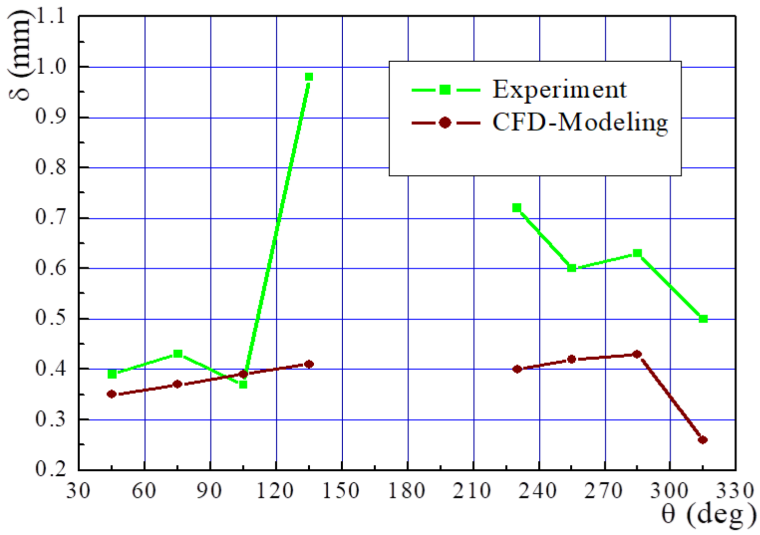


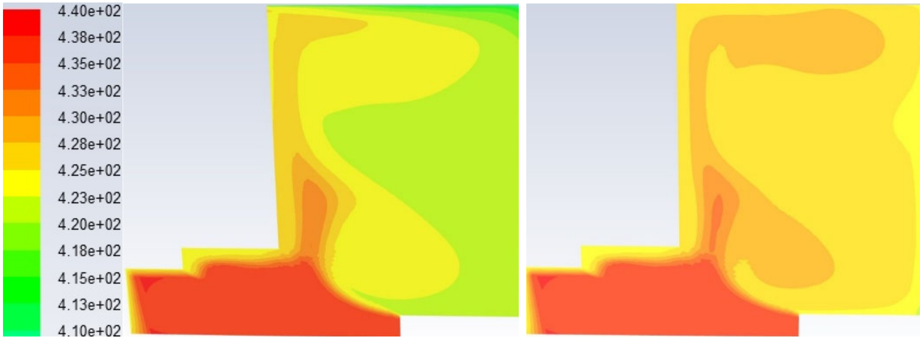
Figure 5: Distribution of oil film thickness over the chamber cross-sections.

A significant deviation is observed in the 135° section. A similar deviation between calculations and experiments was also observed in [7]. It can be explained by the peculiarities of the film thickness measurements during the experimental studies. Therefore, the EWF and VOF models can be applied successfully to define the oil film parameters from the hydrodynamic point of view.

3. Results and discussion

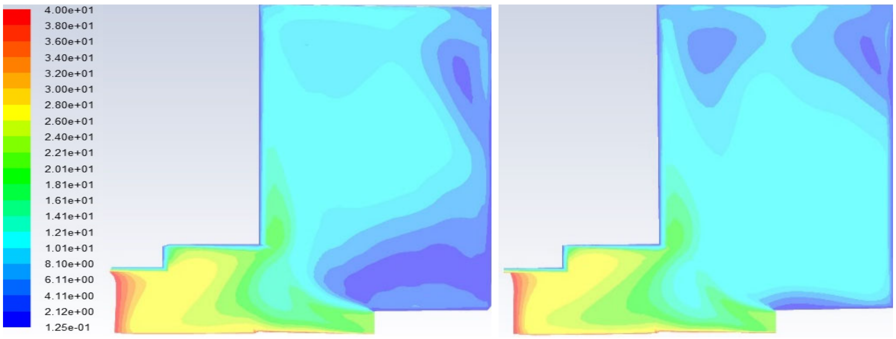
At the moment, there is no single approach to modelling the processes in the bearing chamber. Thus, the identification of the most suitable models and settings is a very urgent problem. The Eulerian model allows determining the parameters of each phase separately, while the VOF model determines the parameters of the oil-air mixture. The Eulerian and VOF models application gives similar results when used to simulate processes in the bearing chamber if the droplets in the flow core are about 30 µm. However, the average computation time for Eulerian based simulations is almost two times higher in this case compared to the VOF model application.

The steady and unsteady VOF based simulation results are shown in Figures 6, 7. The comparative analysis demonstrates qualitatively close results for steady and transient solvers, while the difference in quantitative values can reach 10 K for the mixture temperature and 12 m/s for the velocity. Therefore, it is recommended to use the transient solver to have a higher accuracy of results.



a) b)

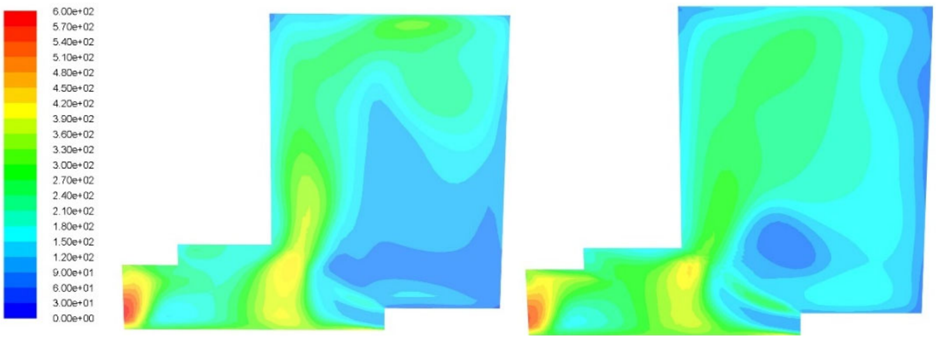
Figure 6: Contours of mixture total temperature (K): a – transient; b – steady.



a) b)

Figure 7: Contours of mixture velocity (m/s): a – transient; b – steady.

The complexity of the multiphase flows modelling in the GTE bearing chamber lies in the accurate description of the flow in the near-wall region and main volume. To achieve a maximum simulation accuracy, it is essential to use a proper turbulence model for the bearing chamber processes prediction. Analysis of the corresponding studies [6, 14-16] and authors' experience show that "*Realizable k-ε*" and "*k-ω SST*" turbulence models are the most appropriate for this issue. The application of these models for the problem under analysis shows very similar results (Figure 8).



a) b)

Figure 8: Turbulent Intensity (%) for "*Realizable k-ε*" (a) and "*k-ω SST*" (b) turbulence models.

Summarising the results of this study, a block diagram that presents the critical components of CFD modelling of the processes in the bearing chamber was developed (Figure 9). Some of these components (incl. models and their settings) will be additionally analysed and refined in further studies.

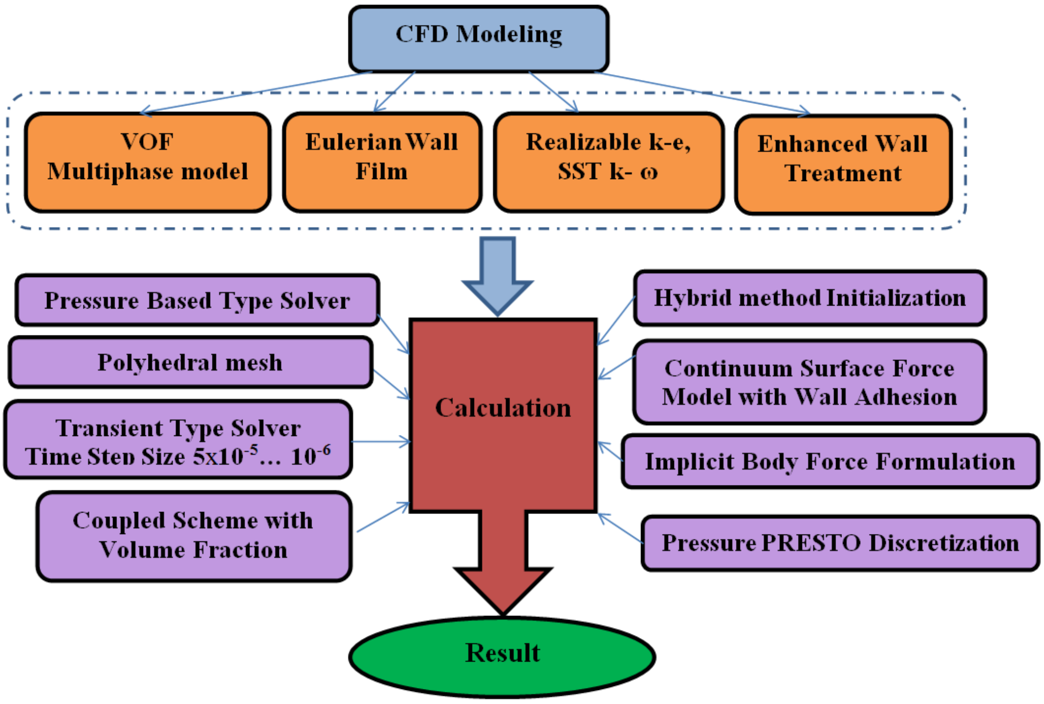


Figure 9: Block diagram of CFD modelling.

4. Conclusion

Two regions in the bearing chamber volume should be considered: a flow core with a gas-droplet structure and a near-wall region, where a liquid becomes a continuous phase.

The VOF and Eulerian models, as a whole, demonstrate qualitatively similar results. Considering that the Eulerian model requires more input data and, on average, 2-3 times longer computation time, it is reasonable to use the VOF model as the base one. Moreover, the convergence process analysis shows that the VOF based numerical schemes are more stable and converges faster. It is especially relevant for the coupled problems, which involve both heat transfer and fluid and gas dynamics calculations. A rational combination of the flow core and film models is of great importance for such problems. In this regard, the combination of VOF and EWF models is a promising solution.

Comparative analysis of the CFD modelling results demonstrates a necessity of transient solver application to model complex processes in the bearing chamber. The applicability of a particular turbulence model depends on the grid size and quality, the bearing chamber geometry complexity, the vortex zones size, flow rates, etc. As a result, now it is recommended to take into account both turbulence models (“*Realizable k- ε*” and “*k-ω SST*”), while the final choice between these models should be based on the comparison of CFD results and experimental data.

5. Acknowledgements

The research leading to these results has been performed in the frame of the “Advanced Modelling Methodology for Bearing Chamber in Hot Environment (AMBEC)” project. This project has received funding from the Clean Sky 2 Joint Undertaking under the European Union’s Horizon 2020 research and innovation programme under grant agreement No 785493.

References

[1] Farral M, Simmons K, Hibberd S and Gorse P 2004. A numerical model for oil film flow in an aero-engine bearing chamber and comparison with experimental data. *Proceedings of ASME Turbo Expo 2004* (GT2004-53698), Vienna, Austria

[2] Wittig S, Glahn A and Himmelsbach J 1994 Influence of high rotational speeds on heat transfer and oil film thickness in aero engine bearing chambers. *Proceedings of Journal of Engineering for Gas Turbines and Power*. Vol. 116, iss. 2. pp. 395–401

[3] Busam S, Glahn A and Wittig S 2000 Internal bearing chamber wall heat transfer as a function of operating conditions and chamber geometry. *J. Eng. Gas Turbines Power*, Vol. 122, ASME, pp. 314-320

[4] Glahn A, Busam S and Wittig S 1997. Local and mean heat transfer coefficients along the internal housing walls of aero-engines bearing chambers. *Proceedings of International Gas Turbine and Aero-Engine Congress & Exposition*, Colorado, Florida

[5] Adeniyi A A, Chandra B and Simmons K 2017 Computational study of a customised shallow-sump aero-engine bearing chamber with inserts to improve oil residence volume. *Proceedings of ASME Turbo Expo 2017* (GT2017-64410), Charlotte, NC, USA

[6] Berthold M, Morvan H, Jefferson-Loveday R, Rothwell B and Young C 2018 Multiphase CFD modeling of external oil flow from a journal bearing. *Proceedings of ASME Turbo Expo 2018* (GT2018-77130), Oslo, Norway

[7] Bristot A, Morvan H and Simmons K 2016 Evaluation of a volume of fluid CFD methodology for the oil film thickness estimation in an aero-engine bearing chamber. *Proceedings of ASME Turbo Expo 2016* (GT2016-56237), Seoul, South Korea

[8] Kakimpa B, Morvan H P and Hibberd S 2015 Thin-film flow over a rotating plate: an assessment of the suitability of VOF and Eulerian Thin-Film Methods for the numerical simulation of isothermal thin-film flows. *Proceedings of ASME Turbo Expo 2015* (GT2015-43506), Montréal, Canada

[9] Singh K, Sharabi M, Ambrose S, Eastwick C, Jefferson-Loveday R, Cao J, and Jacobs A 2019 Assessment of an enhanced thin film model to capture wetting and drying behavior in an aero-engine bearing chamber. *Proceedings of ASME Turbo Expo 2019* (GT2019-91323), Phoenix, Arizona, USA

[10] Singh K, Sharabi M, Ambrose S, Eastwick C and Jefferson-Loveday R 2019 Prediction of film thickness of an aero-engine bearing chamber using coupled vof and thin film model. *Proceedings of ASME Turbo Expo 2019* (GT2019-91314), Phoenix, Arizona, USA

[11] Gorse P, Dullenkopf K, Bauer H J and Wittig S 2008 An experimental study on droplet generation in bearing chambers caused by roller bearings. *Proceedings of ASME Turbo Expo 2008* (GT2008-51281), Berlin, Germany

[12] Kurz W, Dullenkopf K and Bauer H 2012 Influences on the oil split between the offtakes of an aero-engine bearing chamber. *Proceedings of ASME Turbo Expo 2012* (GT2012-69412) Copenhagen, Denmark

[13] *ANSYS Help*. Release 2020 R1.

[14] Le J, Zhenxia L, Yaguo L and Pengfei Z 2019 Numerical study on flow characteristics of liquid jet in airflows. *Proceedings of ASME Turbo Expo 2019* (GT2019-91076), Phoenix, Arizona, USA

[15] Peyton-Bruhl A, Belton D, Walker A D, Snowsill G and Young C 2019 Development of a CFD based methodology for predicting oil auto-ignition in gas turbine bearing chambers. *Proceedings of ASME Turbo Expo 2019* (GT2019-92050), Phoenix, Arizona, USA

[16] Prabhakar A, Ambrose S and Morvan H 2019 Numerical investigation of two phase flow in a dual drive booster. *Proceedings of ASME Turbo Expo 2019* (GT2019-90347), Phoenix, Arizona, USA