

The impact of flow inlet conditions on the two phase flow pattern and the heat transfer in a scavenge pipe of an Aero Engine bearing chamber sealed with brush seals

Stratis Kanarachos, Frederick University

Michael Flouros, MTU Aero Engines

In aeronautics, gas turbine engines need the assistance of systems that have to guarantee performance throughout the whole flight envelope of the aircrafts for which they are designed. One of these systems is the lubrication system and its role is twofold: firstly to remove the heat generated in the highly loaded rolling bearings and also the gears found in the power and accessory gearboxes; and secondly to lubricate these parts.

Currently, the design and analysis process for aero-engine lubrication systems, with their bearing chambers and feeding and scavenging elements, is based on a labyrinth seals' (high air flows) architecture which is regarded to be at its technological limit not allowing for incremental improvement for future engines. A very promising new seal technology was introduced on the market in recent years. This is the brush seal. It is a circumferential brush with bristles that are made of steel, Kevlar or carbon ceramic.

When used in an aero-engine the brush seals offer a number of major advantages; one of the most important is the reduction by 60% of air due to the overlap (zero gap) between the seals and the rotating shaft. In this case, it may be possible to dispense with the bearing chamber vent pipes and thus contribute towards simplification and weight reduction. If only the scavenge pipe is left in the chamber it then has to deal with the whole amount of air and oil that needs to be evacuated from the bearing chamber. The consequences of this configuration on the behaviour of the two-phase air and oil mixture in terms of heat transfer and pressure loss are not well known.

The aim of this paper is to present results of a numerical investigation regarding the influence of the boundary conditions (inflow, geometry, performance data) at inlet and outlet of a scavenge system in an aero engine. The dependence of the resulting flow pattern and heat transfer performance of the scavenge system with respect to the boundary conditions is assessed and discussed.

The numerical investigation has been performed using the commercial CFD software ANSYS CFX and where needed empirical correlations are implemented. This work is part of the European research programme ELUBSYS (Engine

Lubrication System Technologies) and is financed by the European Commission within the 7th Frame Programme.

The numerical results will be validated in a later stage within ELUBSYS in a simulating rig.