PAPER • OPEN ACCESS

CFD analysis of turboprop engine oil cooler duct for best rate of climb condition

To cite this article: Saurabh Kalia et al 2016 IOP Conf. Ser.: Mater. Sci. Eng. 149 012196

View the article online for updates and enhancements.

You may also like

- Corrosion Behavior of Carbon Steel in Soil with Different Particle Size Distribution during Wet-to-Dry Process Shota Ohki, Shingo Mineta, Mamoru Mizunuma et al.
- CFD Simulation of the Oil Displacement in Micromodel for Enhanced Oil Recovery Application P P Y A Jacinta, M F Majnis and S A Musa
- Heat generation of ball bearing for rocketengine turbopump in low-temperature gas hydrogen

Hiromitsu Kakudo, Takashi Yokoyama and Satoshi Takada





DISCOVER how sustainability intersects with electrochemistry & solid state science research



This content was downloaded from IP address 3.144.84.155 on 26/04/2024 at 09:53

CFD analysis of turboprop engine oil cooler duct for best rate of climb condition

Saurabh Kalia¹, Vinay CA² and Suresh M Hegde³

¹M.Tech Student, Department of Mathematical and Computational Sciences, National Institute of Technology, Surathkal, Karnataka, India

²Scientist, Centre for Civil Aircraft Design and Development, CSIR-National Aerospace Laboratories, Bengaluru, Karnataka, India

³Professor, Department of Mathematical and Computational Sciences, National Institute of Technology, Surathkal, Karnataka, India

Abstract. Turboprop engines are widely used in commuter category airplanes. Aircraft Design bureaus routinely conduct the flight tests to confirm the performance of the system. The lubrication system of the engine is designed to provide a constant supply of clean lubrication oil to the engine bearings, the reduction gears, the torque-meter, the propeller and the accessory gearbox. The oil lubricates, cools and also conducts foreign material to the oil filter where it is removed from further circulation. Thus a means of cooling the engine oil must be provided and a suitable oil cooler (OC) and ducting system was selected and designed for this purpose. In this context, it is relevant to study and analyse behaviour of the engine oil cooler system before commencing actual flight tests. In this paper, the performance of the oil cooler duct with twin flush NACA inlet housed inside the nacelle has been studied for aircraft best rate of climb (ROC) condition using RANS based SST K-omega model by commercial software ANSYS Fluent 13.0. From the CFD analysis results, it is found that the mass flow rate captured and pressure drop across the oil cooler for the best ROC condition is meeting the oil cooler manufacturer requirements thus, the engine oil temperature is maintained within prescribed limits.

Keywords: oil cooler; CFD; turboprop engine.

1. Introduction

A typical turboprop engine driving a propeller is via a two stage reduction gearbox. Adequate cooling is essential to maintain the engine operating temperatures within the prescribed limits described by engine manufacturer. The lubrication system of the engine is designed to provide a constant supply of clean lubrication oil to the engine bearings, the reduction gears, the torque-meter, the propeller and the accessory gearbox. The oil lubricates, cools and also conducts foreign material to the oil filter where it is removed from further circulation. Thus a means of cooling the engine oil must be provided and a suitable oil cooler (OC) and ducting system was selected and designed for this purpose. For sizing the oil cooler, engine heat rejection rate calculation must be done for different engine operating conditions such as ground idle, max climb and cruise. Once the engine heat rejection is known, the oil cooler must be chosen and the duct must be sized. Design of an adequate air supply system is essential to maintain the engine oil temperatures within the operating limits.

The general goals in the present work were to analyse the flow field around the oil cooler duct with the nacelle using the CFD method and to estimate pressure drop across the oil cooler and corresponding secondary mass flow captured by the NACA flush inlets for the best rate of climb flight condition. The CFD package ANSYS Fluent was used to model this problem. The finite volume method was implemented for numerical solution of the Navier-Stokes equations with choice of different models for turbulence [2]. HP Z800 four processor workstation was used for computation.

2. Methodology

2.1 Geometric Simplification

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution (cc) of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd

IConAMMA-2016

IOP Publishing

IOP Conf. Series: Materials Science and Engineering 149 (2016) 012196 doi:10.1088/1757-899X/149/1/012196

Minimization of computation time by the solver and reduction in the grid generation effort was achieved by ignoring the minute and irrelevant structural details of the fluid flow geometry. To account for the obstruction offered by the propeller to the flow of exhaust air in the downstream duct, it was replaced by a circular disk at the nacelle exit. The geometry of the nacelle and oil cooler duct was created from 2D lofting. The CATIA V5 R17 CAD system was used for digitizing the geometric data. Points, curves and surfaces were used to describe the geometric layout. The geometry involves an external part and an internal part. The external parts consist of the nacelle see figure. 1. The internal part consists of the oil cooler duct, featuring the NACA twin flush inlets, upstream duct, and downstream duct as shown in figure.2. For importing CAD data into the ANSYS Fluent pre-processor an IGES file was created. A semi cylindrical domain as shown in figure 3 was prepared for analysis as it uses the elements most efficiently and with least distortion [1]. Another reason for selecting the semi-cylindrical domain is that it eliminates edge effect at the corners. Suitable domain sizes were selected for analysis so that there will be no obstruction to the flow as per figure 3.



Figure 1. Assembly of nacelle and oil cooling system.



Figure 2. Complete oil cooling system.



Figure 3. Semi-cylindrical domain.

3. Discretization

This is the most important step in any engineering analysis, as results may vary depending on the meshing strategies. For current case meshing was carried out in ICEM CFD. Before meshing, it was ensured that geometry is free from errors. As structured meshing is a very time consuming process and hard to apply for complex geometries, unstructured meshing technique was adopted [3]. Special emphasis was laid on for developing a good quality mesh for the computational domain with a finer boundary layer mesh along the wall and by maintaining a higher density mesh at critical areas. Element type used for analysis was TRI element with Patch independent Octree mesh method [4]. Elements mesh quality is an important part of controlling discretization error. The quality of the mesh is determined by the shape of the individual element, if the quality of one element is poor then it can lead to inaccurate results. Efforts were taken to produce a mesh of high quality. A grid independent study was also carried out on meshes having 1.7 million, 2.4 million and 2.9 million elements to ensure that results are independent of mesh size. For this three meshes produced and after thorough analysis it was found that there was a negligible variation in the results with finer meshes. So a mesh with 2.4 million cell count was selected as it gave satisfactory results for minimum mesh count. Figure 5, 6 and 7 gives various mesh views.



Figure 4. Fully meshed domain.

IOP Conf. Series: Materials Science and Engineering 149 (2016) 012196 doi:10.1088/1757-899X/149/1/012196



Figure 5. Cross-sectional view of nacelle and oil cooling system.



Figure 6. Cross-sectional View (Z-cut plane).

4. Boundary Conditions

The boundary conditions (BC) were chosen as indicated in Table 1.

| S/N | Part Name | Boundary Condition | S/N | Part Name | Boundary Condition |
|-----|------------------|-----------------------|-----|------------------|-----------------------|
| 1 | Inlet | Velocity Inlet | 7 | Oil Cooler | Wall |
| 2 | Outlet | Pressure Outlet | 8 | Lip Cover | Pressure Outlet |
| 3 | Domain | Pressure Outlet | 9 | Nacelle | Wall |
| 4 | All Fluid Bodies | Interior | 10 | OC Upstream | Wall |
| | | | | Cover | |
| 5 | Upstream Duct | Wall | 11 | OC Downstream | Interior |
| | | | | Cover | |
| 6 | Downstream Duct | Wall | 12 | Ram Pressure | Wall |
| | | | | Pipe and Ejector | |

Table 1. Boundary conditions for parts and fluid bodies.

IConAMMA-2016 IOP Publishing

IOP Conf. Series: Materials Science and Engineering 149 (2016) 012196 doi:10.1088/1757-899X/149/1/012196

5. Flow parameters and solver set-up

The fluid flow was modelled as an incompressible flow because of low Mach number and calculations were run as per the given flight condition. Operating pressures used for analysis and flight conditions are mentioned in Table 2 and 3 respectively. A turbulence intensity value of five percent behind the nacelle was assumed. Fluent uses a control volume based technique to convert differential governing equation to algebraic form, which is solved numerically in Fluent. The discretization method used for gradient and pressure was "Green Gauss Node Based" and "Standard" respectively whereas for momentum, turbulent kinetic energy, turbulent dissipation rate, turbulent viscosity and energy a "first order upwind" scheme was used [4]. All solver runs were realized on the local workstation. To maintain a good balance between speed of convergence and stability of solution, under relaxation factors were maintained below 1. Convergence criterion used for the analysis was 10^{-3} . The standard k- ω model which is a two equation model is one of the popular models due to its simplicity and stability however, for the current case SST k-ω (shear stress transport) model was used as it is very good for solving flow near wall cases. While in operation oil cooler (heat exchanger) comprised by NACA flush inlets creates obstruction to flow, adding a pressure drop across heat exchanger. To account this, pressure drop was modeled by considering heat exchanger as a porous medium. Porous medium was modeled based on two coefficients inertial resistance and viscous resistance. These coefficients were derived based on experimental pressure and velocity data. Table 4 gives the inertial and viscous resistances that were used for the analysis.

| Table 2. (| Given i | nputs | for | best | rate | of | climb | flight | operation. |
|------------|---------|-------|-----|------|------|----|-------|--------|------------|
|------------|---------|-------|-----|------|------|----|-------|--------|------------|

| Case No. | Flight Condition | Altitude , m (ft.) | Free Stream Total Pressure, P _{t.} , Pa | Free Stream Static Pressure P _s , Pa | Outside Air Temperature T, °C (K) | Speed, m/s (M) | Engine Mass Flow Rate, kg/s | AOA _{nac} elle [°] |
|-------------|-----------------------|-----------------------|--|---|---|-------------------|---|---|
| 1 | Best Rate of Climb | 3810 (12500) | 66222 | 63190.5 | 9.75 (263.4) | 88.47 (0.26) | 3.43 | 5.45 |

Table 3. Air properties.

| Case | Static | Dynamic | Total |
|------|---------------|---------------|-----------|
| | Pressure, psi | Pressure, psi | Pressure, |
| | (Pa) | (Pa) | psi (Pa) |
| 1 | 9.165 | 0.439 | 9.604 |
| | (63190.5) | (3031.5) | (66222) |

| Table 4. | Resistances | for | porous |
|----------|-------------|-----|--------|
| | | | |

| | medium formulation. | | | | | |
|------|--------------------------------|--|--|--|--|--|
| Case | Inertial Resistance, 1/m | Viscous Resistance, 1/m ² | | | | |
| 1 | 5.9740 | 2.2419 x 10 ⁶ | | | | |

IOP Conf. Series: Materials Science and Engineering 149 (2016) 012196 doi:10.1088/1757-899X/149/1/012196

IOP Publishing

6. Computed Cases

6.1 Flow Visualisation

The main flow characteristics were monitored at the upstream duct, oil cooler, downstream duct and the pressure loss between the upstream and downstream faces of the oil cooler was determined see figure 7,8 and 9. A stagnation point was noted in the vicinity of the lip because of the NACA profile. As the fluid passes over the nacelle, the pressure goes on decreasing due to the nacelle contour and the negative pressure gradient developed on the NACA inlets helps to draw cold atmospheric air into the oil cooling system. Relatively high pressure region is observed near tail portion of the nacelle indicates adverse pressure gradient (results in drag).



Figure 7. Static Pressure contours of Nacelle.



Figure 8. Static Pressure Maps of Oil Cooling System.

IOP Conf. Series: Materials Science and Engineering 149 (2016) 012196 doi:10.1088/1757-899X/149/1/012196



Figure 9. Velocity Vector Maps of Oil Cooling System Case 1: Best Rate of Climb Z, m (ft.) =3810 (12500); Speed, m/s (M) = 88.47 (0.26); OAT, $^{\circ}$ (K) = -9.75 $^{\circ}$ (263.4); AOA_{nacelle} = 5.45 $^{\circ}$; Mass flow rate, kg/s (lb/s) = 3.43 (7.61).

| Data supplied by | oil cooler manufacturer | CFD Analysis Results | | |
|--|--|---------------------------------|---|--|
| Mass flow rate Pressure drop across oil cooler | | Mass flow rate | Pressure drop across oil cooler | |
| 1.725 kg/s (230 lb/min) | 1450.24 Pa (5.83 " of H ₂ O) | 1.730 kg/s (230.5731 lb/min) | 1572.67 Pa (6.32216 " of H ₂ O) | |

7. Conclusion

This research paper shows the CFD analysis and results carried out using commercial software package ANSYS FLUENT 13.0 for the nacelle and complete oil cooling system of pusher configured LTA. The choice of RANS based SST k-omega model for capturing turbulence effects proved to be successful. As per the input conditions listed in Table 2, 3 and 4 for aircraft best ROC flight operation, the case was computed for around 30000 iterations to determine the secondary mass flow rate captured by the oil cooler and the pressure loss across its upstream and downstream faces.

8. Discussion

Qualitative results obtained theoretically show an over prediction of 8.44% in the pressure loss as shown in Table 5 when compared against the experimental data provided by the OC manufacturer which is in close agreement, thus the results are satisfactory. Based on these results, oil cooling duct design clearance can be obtained from certification agencies in order to proceed and conduct ground/flight tests.

9. References

- [1] Manish Sharma, T. Ratna Reddy, and Ch. Indira Priyadarsini 2013 *Flow Analysis over an F-16 Aircraft Using Computational Fluid Dynamics* International Journal of Emerging Technology and Advanced Engineering **3** 5
- [2] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu 2008 *Computational Fluid Dynamics: A Practical Approach* First Edition (USA: Elsevier) chapter 6 pp 258-271
- [3] Anderson J D 1995 *Computational Fluid Dynamics: The Basics with Applications* International Edition (Singapore: McGraw-Hill Book Co.) chapter 4 and 5 pp 125-215
- [4] ANSYS Fluent 13.0 User Guide Manual