

CFD Analysis of Oil Cooler Duct for Turboprop Aircraft Engine in Pusher Configuration

Abhijeet B. Chougule¹, Vinay C A.², Dr. Saleel Ismail³

M.Tech Student, SMBS, VIT University, Chennai, India¹

Scientist, C-CADD, CSIR-NAL, Belur Campus, Bangalore, India²

Associate Professor, SMBS, VIT University, Chennai, India³

ABSTRACT: CFD analysis of flow in and around the nacelle and oil cooler duct was carried out using RANS based SST $k-\omega$ model of the commercial software Fluent. Special efforts was laid on developing a good quality mesh for the computational domain with a finer boundary layer along the wall and by maintaining a higher density mesh at critical areas. Pressure drop across Oil cooler and mass flow rate captured by NACA twin inlet duct has been established from the results obtained. The results obtained from the CFD analysis have been compared with Oil cooler manufacturer supplied test data and the intake duct design data and found satisfactory.

KEYWORDS: Nacelle, Oil cooler, Oil cooler duct, CFD, Pressure drop

I. INTRODUCTION

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) was selected for this purpose. OC was located to the aft of the nacelle and a duct with twin NACA flush inlets and ejectors was designed. This allows taking advantage of the effect of propeller suction to drive the flow through the duct. However the effect of propeller suction is neither simulated nor tested as it is learnt that it will improve the secondary flow further which is neglected for the base line design. For sizing the air supply duct for the OC, a hot day climb at maximum climb rating is assumed as the flight condition. The minimum desired airspeed is 130 KEAS, and the altitude is 4500 ft (1371.6 m) above sea level. The atmosphere is ISA +35°C, corresponding to an Outside Air Temperature (OAT) of 41 °C. The required air or oil flow and the corresponding pressure drops are obtained as a function of the standard heat rejection from the OC chart provided by the manufacturer. The OC duct incorporates a NACA flush inlet designed for maximum efficiency to accommodate the required mass flow. Exit duct or the downstream duct is sized from the flight condition using the calculated efficiency of the inlet, OC pressure drop, the diffuser pressure loss factor and the exhaust depression due to propeller suction. The objective of the design is to ensure adequate cooling air supply system which meets the requirements of the OC (Heat Exchanger) pressure drop and mass flow under all the flight conditions. This in turn ensures that the engine oil temperature remains within the operating limits. The nacelle and OC system configuration are shown in Figure 1.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

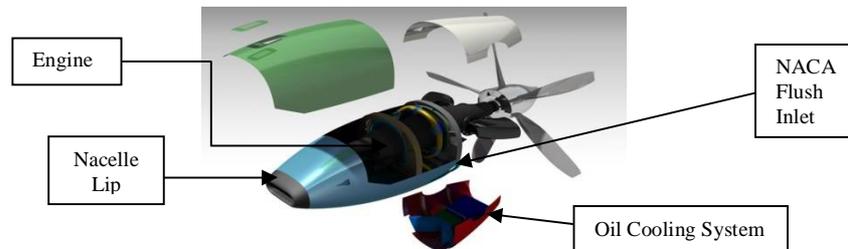


Figure 1: Nacelle and oil cooler system configuration

The general goals were to simulate the flow field around the nacelle and OC duct using CFD and thereby estimate pressure drop across the OC and secondary mass flow rate captured for minimum climb flight condition. Simulation was carried out using the commercial CFD package ANSYS Fluent. The governing equation of continuity, momentum and turbulence are solved using finite volume approach in this package. Jiyuan Tu et al [1]. Computation for solution and post processing were carried out at CSIR-NAL. HPC facility HP Z800 four processor workstation was used for this study.

II. METHODOLOGY

A) MODELLING

While accurate geometric modeling requires capturing the real geometry to the greatest extent. The meshing effort and subsequent solver time need to be minimized. Therefore it was decided to neglect minor and inconsequential structural details. Also, the propeller and spinner assembly was not considered (see Figures 2 and 3). Geometric simplification was carried out using CATIA V5 R20. Healing assistant of CATIA V5 R20 was used to create error free geometry for mesh generation. The geometry was checked for any flaws and holes and repaired when needed.

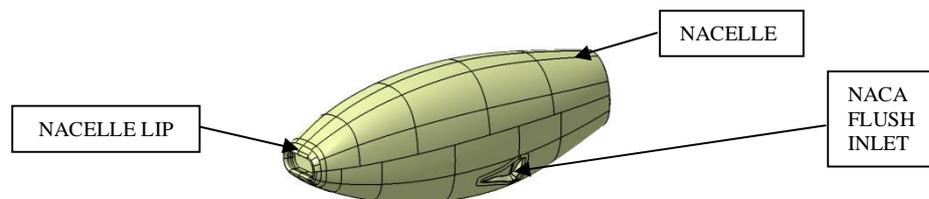


Figure 2: Assembly of Oil Cooling System with Nacelle

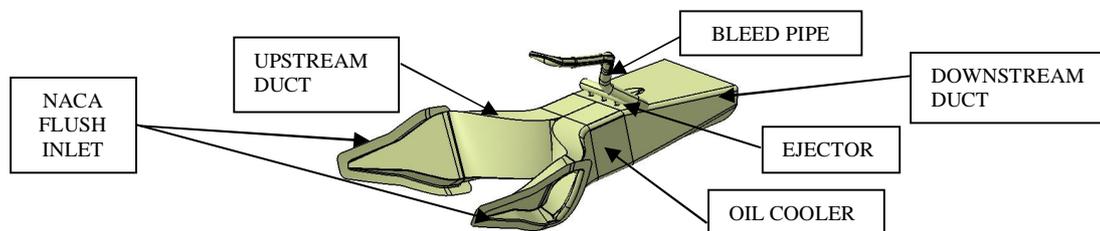


Figure 3: Oil Cooling System Model

To simulate the flow field around a nacelle it is necessary to create an external domain. There are various shapes of domain that can be modeled like rectangular, spherical, semi cylindrical, cylindrical etc. A semi cylindrical domain chosen for the current study as it uses the elements most efficiently with least distortion and eliminates edge effect at the corners. The full domain is shown in Figure 4. X and D are the length and diameter of nacelle. Manish Sharma et al [2].

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

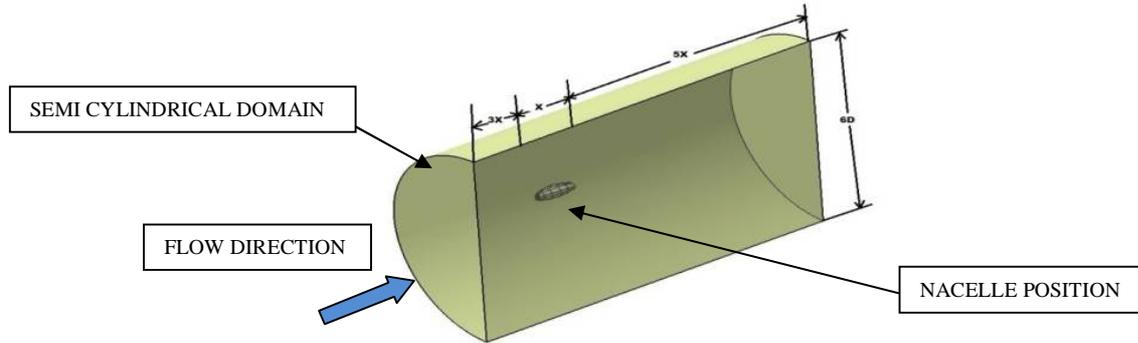


Figure 4: Flow Domain Size

B) MESH GENERATION

Geometry from CATIA V5 R20 was imported in IGES format to ANSYS ICEM CFD. Unstructured (tetrahedral) meshing was generated using patch independent Octree technique. Special emphasis was laid on obtaining a good quality mesh with a fine prismatic boundary layer mesh along the wall. Higher density mesh was used at critical areas. A grid independent study was carried out to ensure that results are independent of mesh size. For this three meshes were produced and after thorough analysis it was found that there was a negligible variation in the results with finer meshes. So a mesh with 2.33 million cell count was selected as it gave satisfactory results for minimum mesh count. Figure 5 and 6 shows the mesh with major boundaries and zones highlighted.

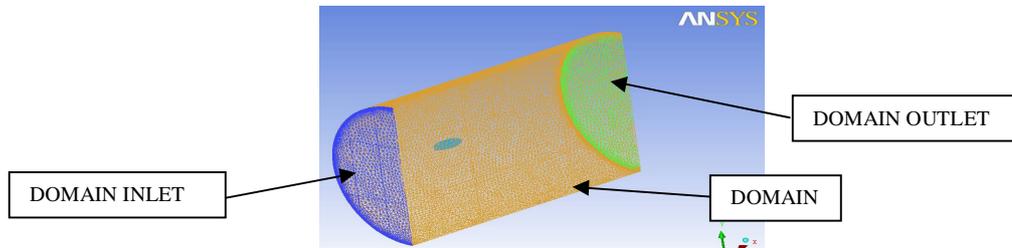


Figure 5: Full meshed geometry in ANSYS ICEM CFD 13.0

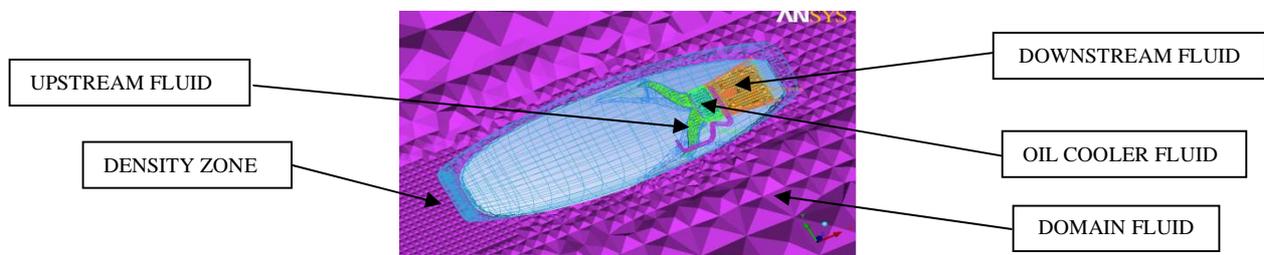


Figure 6: Variation in Element sizes of Oil Cooling System and Domain

C) CFD ANALYSIS

Pressure based solver type with double precision was selected for the current study. The model used for analysis was SST $k-\omega$ model. This is a two equation RANS model which is good for solving flow near wall cases. It is a hybrid model combining the Wilcox $k-\omega$ and the $k-\epsilon$ models. The working fluid in this analysis is air which is

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Special Issue 9, May 2016

assumed to be incompressible. There is an internal flow in the OC duct and in engine intake duct, while the flow over the nacelle is external. The boundary types are listed in Table 1. Prathapanayaka R et al [3]. Flight condition and operating pressures used in this analysis are mentioned in Tables 2 and 3.

Table 1: Boundary types

Part name	Type of boundary
Domain Inlet	Velocity inlet
Domain outlet	Pressure outlet
Nacelle	wall
Upstream duct	wall
Downstream duct	wall
NACA inlets cover	Interior
Porous in& out	Interior
Oil cooler	wall
Downstream duct	wall
Downstream Duct cover	Pressure outlet
Domain	Pressure outlet

Table 2: Flight condition

Flight Condition	Altitude, m (ft)	OAT, °C (K)	Speed, m/s (Mach No.)	Nacelle AOA, degree
Min. Climb (4500ft)	1371.6 (4500)	6.1 (279.25)	71.46 (0.21)	6.60

Table 3. Air Properties for Flight Condition

Case	Density kg/m ³	Dynamic Viscosity N.s/m ²	Static Pressure (Pa)	Dynamic Pressure (Pa)	Total Pressure (Pa)
Min. Climb (4500ft)	1.0717	1.746e-05	85908.6	2669.4	88578

To account the pressure drop across OC (Heat exchanger), it 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 calculated using experimental pressure and velocity data. The values used are provided in Table 4.

Table 4: Resistances for Porous medium Calculation

Case	Inertial Resistance, 1/m	Viscous Resistance, 1/m ²
Min. Climb Condition (4500ft)	4.6590	2.140x10 ⁶

III. RESULTS AND DISCUSSION

The solution was carried out for nearly 14000 iterations. The mass imbalance was found to be less than 0.5%, implying that continuity equation was satisfied.

Figure 7 represents the contours of static pressure on nacelle. Stagnation point at the intake lip can be seen. Low pressure region was noticed on OC intake cutout portion of the nacelle due to the aerodynamic profile (contour) of the nacelle design. This low pressure region in the lower portion of the nacelle helps to drive the flow through OC intake. A relatively high pressure region was observed near tail portion of the nacelle indicating adverse pressure gradient (results in drag).

Figure 8 shows the contours of static pressure on OC ducts. Due to contour of NACA flush inlets, low pressure region was observed which is blue in color. Further downstream of the duct static pressure rise was seen proportionately due to the obstruction for the flow by OC. Exit pressure of the downstream duct was slightly above the atmospheric pressure.

Figure 9 shows the velocity vector plot. Intake velocity accelerated (suction pressure) which is seen orange in color near NACA flush intake. Due to the contour of the Y shaped duct, velocity was reduced gradually up to the downstream duct. It was found that the NACA flush intake design was very much efficient. In the vicinity of ejector nozzle tubes flow was diverted and retarded. In the downstream duct, magnetic chip detector access scoop had negligible disturbance in the overall flow path. From the static pressure plots pressure drop across the OC was calculated. The OC Manufacturer supplied graph data and captured mass flow rate has been tabulated in Table 5.

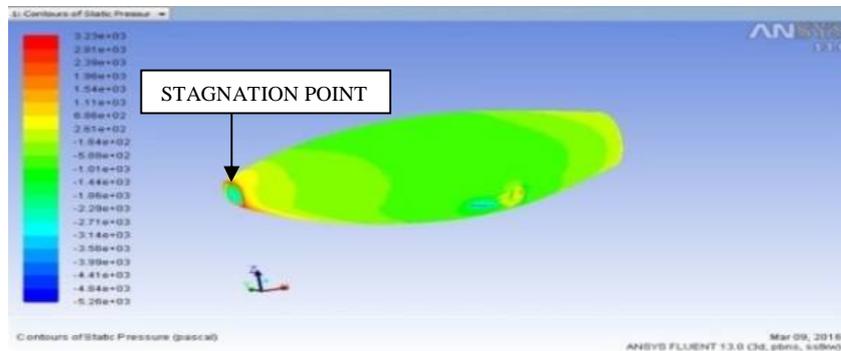


Figure 7: Static pressure contour of Nacelle

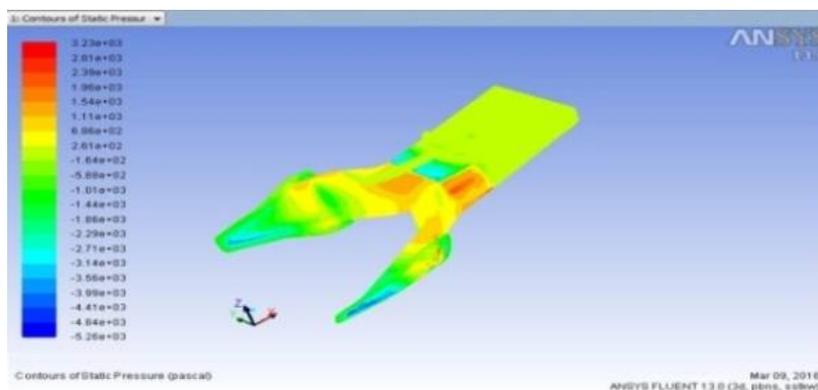


Figure 8: Static pressure contour of Oil Cooling System

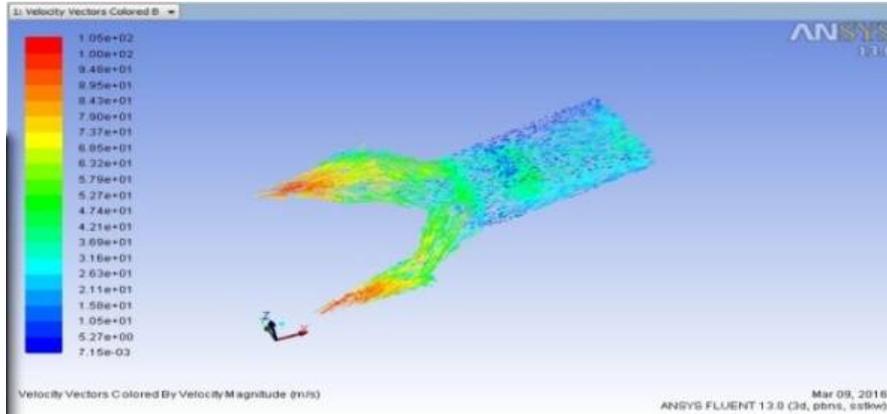


Figure 9: Velocity vector contour of Oil Cooling System

Table: 5 Static pressure drop and mass flow rate

OC Manufacturer Data		CFD Analysis Results	
Mass Flow rate	Pressure Drop across OC	Mass Flow rate	Pressure Drop across OC
1.8 Kg/s (250 lb/min)	1 474.60 Pa (5.92”of H ₂ O)	1.7kg/s (224.871 lb/min)	1346.28 Pa (5.41”of H ₂ O)

IV. CONCLUSION

This paper shows computational fluid dynamics study using ANSYS FLUENT has been successfully carried out for the nacelle and OC duct of LTA. Simulations were carried out for minimum climb flight condition to determine the OC intake secondary mass flow rate captured and pressure drop across the OC. CFD results as obtained shows an under prediction of 8.7 % in the pressure drop when compared with the OC manufacturer supplied experimental data is a good prediction thus the results are satisfactory. The same model will be used for carrying out further simulation for different flight conditions. These initial CFD predictions have supported the design to get clearance from certification agencies for conducting the ground/flight tests.

REFERENCES

[1] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, “Computational Fluid Dynamics: A Practical Approach”, First Edition, Elsevier, USA, 2008.
 [2] Manish Sharma, T. Ratna Reddy, and Ch. Indira Priyadarsini, “Flow Analysis over an F-16 Aircraft Using Computational Fluid Dynamics”, International Journal of Emerging Technology and Advanced Engineering, Volume 3, Issue 5, May 2013.
 [3] Prathapanayaka R, Agnimitra Sunkara SN, Balamurugan M and Kishor Kumar, “CFD Analysis of a Highly Loaded Gas Turbine Stage”, 17th Annual CFD Symposium, August 11-12, 2015, Bangalore.