Heat Transfer in Double Annular due to Natural Convection

Jan Pařez¹, Pavel Rohan², Tomáš Vampola¹

¹Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Center of Advanced Aerospace Technology, Czech Technical University in Prague. Technická 4, Praha 6, Czech Republic
²Department of Manufacturing Technology, Faculty of Mechanical Engineering, Czech Technical University in Prague. Technická 4, Praha 6, Czech Republic

Abstract. Presented paper is focused on 2D numerical simulation of turboprop engine cooling in the section of a gas turbine. The paper describes the issue of heat transfer by natural cooling of a turboprop engine in the turbine region in order to determine the time-dependent temperature field in the engine. The analysis is represented on a simplified double annular geometry, where the inner tube represents the engine rotor, the middle tube represents the flow path and the outer ring replaces the engine case. Simultaneous heat transfer by natural convection (Boussinesq model) and radiation (model S2S) is solved on this geometry. Numerical data were compared with data that was gained from measuring on an experimental stand. The surface temperatures of the tubes were measured on the experiment stand during the cooling of the engine model. The temperatures on the walls of the flow path were based on real temperatures of the running engine as initial conditions together with the temperature of the outer tube. The results will be further applied to the study of temperature-dependent deformation of aircraft engine parts, which have a significant effect on the safety and trouble-free operation of the aircraft engine.

1. Introduction

After shut off the turbine engine, the asymmetrical distribution of the temperature field on the components of the turbine engine occurs due to natural convection. This unequal distribution of the temperature field has many adverse effects that can lead to irreversible engine damage or critical engine failure. A certain time after shutting off the engine, there is a temperature difference between the upper and lower side of the shaft, and also other parts of the engine construction. This difference is changed with time and his maximal is just at one specific time. After this time, the temperature difference begins to stabilize until the engine is cooled down completely.

Due to the temperature difference caused by natural convection of hot engine parts, the thermal expansion of the parts occurs. This expansion is more noticeable with the length of the part. The solution on the rotor shaft assembly is significant. Based on the temperature difference, the shaft bends and causes i.e. thermal deflection called "Thermal Bow". Due to this bending, the balance distribution of the rotor parts is changed and this unbalance has many side effects. These effects include, for example, significant vibrations of the rotor parts, stresses on the connecting surfaces and last but not least bearing loadings. In extreme cases, unintentional contact may occur during the subsequent starting of the engine due to the reduction of the gaps between the rotor and stator parts, especially in the area of the labyrinth seals or the area of the blades.
The state of the art has long been known in the field of energy, where the development of natural convection is significant due to huge rotating mass. This mass accumulated heat energy, which is released by natural convection after engine shut off, but sufficient cooling is prevented by insulating the entire steam turbine and a large temperature gradient. After restarting the turbine, the thermal deflection may not be sufficiently eliminated and the entire system may be destroyed. In the field of aviation, this phenomenon has not been significant for a long time, due to sufficient clearances and limited temperatures. With increasing demands on engine efficiency, designers are forced to choose constructions with smaller gaps between the stator and rotor parts, temperatures are rising and, finally, there are field of military aircraft and application, where in some cases sufficient cooling of the engine after landing is impossible.

2. Analytical Model
The presented research describes the situation of engine shut off and subsequent cooling by natural convection. The solution is simplified to a 2D analysis in the section of a gas turbine, where the initial and boundary conditions defined below are assumed. The presented assembly consists of 3 tubes corresponding to the dimensions and thicknesses on the aircraft engine. Further the middle section, the hot gas flow path section of the turbine and the inner section of the shaft. The initial conditions are then prescribed for the outer tube, where is tube cooled by an external air flow. The temperatures on the middle turbine tube and shaft tube are prescribed.

If the aircraft fly at cruising altitude (CRUISE), the installed engine is cooled by a forced air flow flowing in the longitudinal direction. Due to the high speed of the air and its low temperature (corresponding for example to the temperature of the standard atmosphere at cruising altitude), cooling is usually sufficient in this flight mode. The worst case occurs when the aircraft is in standby mode or taxiing on the ground at low speed. Even in this case, however, the heat transfer from the surface of the engine is sufficient to cool the engine and its lower parts due to forced convection. In this case, the air temperature can reach much higher values than the corresponding temperature at cruising altitude, e.g. the temperature corresponding to an extremely hot day can be up to 50 °C. The analysis of the cooling of the outer tube has been previously studied and measured on a real turbine engine and is presented in previous research [1]. The results of previous paper are used for comparison and validation with the current study. The initial state is assumed to be shutting of the running engine and the associated heating of the middle and shaft tubes from hot gas streams to an isothermal field of 600 °C. The outer tube is heated to an isothermal field at a temperature of 150 °C, this condition approximately corresponds to the external cooling of the engine due to forced convection of air during flight. Figure 1 shows a 2D diagram of the monitored area including dimensions, temperatures and emissivity values for heat radiation.
All areas of the gas are stationary in the initial area of the solution and a temperature of 25 [°C] is prescribed for the middle part, then a temperature of 150 [°C] for the area of the flow path and the shaft space. This simplification is chosen with regard to the subsequent experimental validation of the simulated data. When the heat source is removed, the areas are blown through and then the measuring of natural convection is recorded, the initial condition is based on this assumption.

3. CFD Analysis
Numerical analysis of heat transfer by natural convection and heat radiation is compared and validated with experimental data measured on a measuring stand. The stand model is based on the real dimensions of the turbine section of the turboprop engine and it was presented in the previous section. Material properties and emissivity parameters are considered.

The CFD software ANSYS 2019R2 was used to calculate the heat transfer. The problem is solved in 2D with RANS (Reynolds Averaged Navier-Stokes) transport equations supplemented by the equilibrium energy balance. It used the Boussinesq model and precise density model to express the temperature dependence of thermophysical properties. The turbulent flow is modeled using the $k - \omega$ SST (Shear Stress Transport) turbulence model. It used the S2S (surface-to-surface) radiation model to predict the radiative heat transfer.

The meshing was chosen with the assumption of double annular profiles and smoothing "Inflation" layers around the walls in the area of the boundary layers. The meshing scheme is shown in Figure 2.

![Figure 2. Scheme of 2D turbine section](image)

The research of heat convection in the annulus was presented, for example, in articles [2-4]. The heat transfer due to the natural convection in the section between two coaxial cylinders was studied by Kuehn and Goldstein [5, 6].

Kuehn and Goldstein experimentally study heat transfer during natural convection between two horizontal cylinders using an optical method (Mach–Zhender interferometer). Authors state that most of the papers are about study for a limited range of Rayleigh’s number $10^1 < R_{al} < 10^6$, where Rayleigh’s number is based on radius difference, i.e. $R_{al} = g\beta L^3 \Delta T / \mu a$, where $L = R_{outer} - R_{inner}$ and $\Delta T$ is the temperature difference between cylinders.

Due to the huge temperature differences between the individual tubes, especially between the outer tube and the middle tube after the start of the measurement and before the temperatures stabilized, a large Rayleigh number is assumed. The study of high Rayleigh numbers in natural convection in annuloid was presented by Fant [7] and Charter-Moitabi. [8]
Natural convection is the driving force in the system and it is caused by a change in air density. These changes in the density of the air in the gravitational field cause the fluid to flow and move. The density differences are mostly due to temperature differences. In natural convection is areas, where temperature is higher than means the density is smaller and the fluid moves up. The tube is thus unevenly cooled and reheated at the same time. In the case of the double annulus component, this phenomenon is even more pronounced due to the dependences of the partial annulus and the reheating of the outer parts by the inner ones.

\[ T_2 > T_1 \Rightarrow \rho_2 > \rho_1 \]  \hspace{1cm} (1)

\[ \frac{d\rho}{dz} < 0 \Rightarrow \frac{d\rho}{dz} > 0 \]  \hspace{1cm} (2)

Boussinesq approximation assume the density depends on temperature and its approximate by the following Equation 3.

\[ \rho \approx \rho_0 + \frac{\partial \rho}{\partial T} \bigg|_{T_0} (T - T_0) + \cdots, \]  \hspace{1cm} (3)

where \( \rho_0 \) is initial density and \( \frac{\partial \rho}{\partial T} \) is time depend density gradient. This Equation 3 is may be present with thermal expansion coefficient \( \beta \) to Equation 4.

\[ \rho = \rho_0 + \beta \rho_0 (T - T_0) \]  \hspace{1cm} (4)

In some case is this approach not valid because of high temperature differences. Precise approach of calculation exact dependency density on temperature for this case is used. For transient simulation with \( \rho = \rho(T, p \ldots) \) and high temperature differences is used incompressible ideal gas model correspond to temperature ideal gas model.

\[ \rho v = R n T \]  \hspace{1cm} (5)

Analysis of NS equation using dimensionless variables in gravitation field is gained dimensionless velocity profile

\[ \bar{u} = f(\bar{x}^*, Gr, Pr), \]  \hspace{1cm} (6)

where \( \bar{x}^* \) is coordinate, \( Gr \) is Grasshof number and \( Pr \) is Prantl number.

\[ Gr = \frac{g \beta \Delta T L^3}{\mu^2}, \]  \hspace{1cm} (7)

where \( g \) is gravitation constant, \( L \) represent characteristics length and \( \mu \) kinematics viscosity.

\[ Pr = \frac{\mu}{a} \]  \hspace{1cm} (8)
The previous equation 8, where \( \nu \) is kinematics viscosity and \( \alpha \) is thermal conductivity. Multiple of Grassholf number and Prandtl number is Rayleigh’s number gained. This number to heat transfer and calculation natural convection is used. Nusselt number is then calculated with Rayleigh’s number and another geometry depend parameters.

\[
R_{al.} = Gr \ast Pr = \frac{\beta g \Delta T L_3}{\mu a}
\]  

(9)

ANSYS software allows the selection of many types of equations and models described and presented in [9] [10]. The solver can choose the required turbulence model and approach to the solution of natural convection and heat transfer by radiation. The used turbulence model determines the "expected" flow regime in the solved geometry mentioned in the previous chapters. In the case of problems with natural convection, the Rayleigh number determines the flow mode [7]. The value of the Rayleigh number may not be accurate, because the temperature-dependent parameters in the case of natural convection cannot be determined for eat time step. The outer diameter of the middle tube of the annulus, shown in Figure 1, and the properties of air at an average temperature of 25 [°C] were used as a characteristic dimension. The calculated value of the Rayleigh number is of the order of \( 7 \ast 10^7 \) to \( 9 \ast 10^8 \) and this value is between the turbulent mode with the transition after initial natural convection to the laminar flow. From the point of computational it is a model of turbulence.

A comparison of different approaches to turbulence modeling has been made in academic work [11]. Based on numerical experiments, the Transition SST (Shear Stress Transport) model was recommended for turbulence modeling. The author also dealt with the selection of a suitable approach to modeling natural convection. He compared the model based on the Boussinesq linearization of the ascending force and the model of the incompressible ideal gas. When comparing the results of both models, he states that both models provide practically the same results in a given task. Finally, the author compared two models of radiant heat transfer - model S2S (surface to surface) and P1.

The results show that the P1 model overtakes about 12% of the radiant heat flux above the S2S model. However, when comparing heat fluxes on both surfaces, the P1 model shows better results. Using the S2S model, the task could not be brought closer, so the total heat fluxes on both sides of the gondola were not the same. This imbalance was caused by the radiation component of the heat flux and it was approx. 6% of the total heat flow. On the other hand, the P1 model managed to calculate the task with zero resulting heat flow imbalance. For these reasons, the author recommends using the radiation model P1.

4. Experimental Stand

4.1. Stand Design

The numerical analysis is compared and validated with experimental data measured on a measuring stand. The construction of the stand is based on the real dimensions of the turbine section of the turboprop engine and corresponds to the model given in the analytical model. Stainless steel W.Nr. was chosen as the pipe material. 1.4301 (AISI304) with defined parameters considered in CFD analysis. The solution of the 2D problem on the measuring stand was realized by thermal insulation of the measured area from the surrounding contact surfaces and lateral transition surfaces of the intermediate tube. Ceramic Fiber Insulation was chosen for the insulation.

A Dawell Inverter Heater shown in the Figure 6 was used for heating the tubes Heating of the inner and middle tubes to the isothermal heat field was performed by resistive removable "blankets" with which the tube was wrapped. After heating and stabilizing the temperature field, these heating elements were removed and the measuring space was closed with an insulated cover. Temperatures were measured in a perpendicular section of six equally spaced K-type thermocouples on each tube showed
on Figure 4. The results were recorded in the Almemo measuring center. Numerical and experimental data are further compared and evaluated at the end of this paper. (Figure 5)

![Figure 4. Thermocouples equally spaced on tubes](image)

![Figure 5. Experimental stand](image)

4.2. Thermal Camera

The course of the measurement was recorded with a FLIR Thermal Camera to determine heat fluxes and areas with heat loss and transition. Due to the dependence of the emissivity parameter $\varepsilon$ on the temperature, the angle of reflection and the variable state of the surface on the assembly, it was not possible to precisely set the emissivity value in each time step. The Thermal Camera was controlled by the temperature at a specific point, where this temperature was detected using a placed K-type thermocouple. Thermal analysis with FLIR Thermal Camera is shown in Figure 6.

![Figure 6. Thermal analysis with FLIR Thermal Camera](image)

5. Results

A CFD model of turboprop engine cooling was described, which was further compared with the results from experimental measurements. Time step 1 [sec] was selected for the numerical calculation with the number of iterations 50 in each time step. The measurement was performed in the range of 0 to 60 minutes, however, the most significant phenomenon takes in the range between 0-45 minute, then the courses are decreases exponentially until engine is completely cooled. The temperature and velocity profiles for the selected time steps are shown in Figure 7 to Figure 10. Next, the points corresponding to the location of the thermocouples in Figure 4 have been defined, in which graphs of temperature versus time are shown for each tube in Figure 11, Figure 12 and Figure 13. The temperature versus time for experimental measurements is shown and compared in the same graph.
The graph shown on Figure 11 to Figure 13 compares the corresponding numerical and experimentally measured values. It can be evaluated that under the same initial and boundary conditions, the numerical values of the temperature are higher and at the same time the decrease of temperatures with time is lower than the experimental values. This is due to several influences. The main reason is the spread of heat transfer in a real 3D assembly. It is not structurally possible to ideally insulate all connecting members and pipe lengths.

The heat dissipation in the assembly was observed by a FLIR Thermal Camera shown in Figure 6. Furthermore, the difference between the numerical and experimental data is due to the cooling surface and the manipulation when removing the heating elements. Last but not least, this is due to non-compliance with all temperature-dependent factors such as emissivity or heat transfer coefficient. These parameters would have to be studied in detail for each specific case and temperature. Finally, the numerical analysis showed good agreement with the measured values from the experimental stand and at the same time the external temperature field on the engine case with the characteristic corresponds to the previously measured temperature field on a real turboprop engine. Difference between measured and numerical values of the engine temperature field is about 20°C after 30 minutes. From this result it is possible to evaluate the possibility to use numerical analysis to obtain the temperature field of the engine after switching off. The temperature field can be used directly for strength analysis and shaft deflection analysis.
Figure 11. Comparison of numerical and experimental temperatures on shaft tube

Figure 12. Comparison of numerical and experimental temperatures on middle tube
6. Conclusions
The problem of cooling the turboprop engine after shut off in the area of the gas turbine was studied, assuming simplification to 2D problems. The cooling of the engine due to natural convection and the effect of heat radiation in a double annulus consisting of three concentric tubes of different diameters and thicknesses were solved. The analysis was performed in two steps. First, an analysis of natural convection with the effect of heat radiation was performed in the Ansys CFD program. The geometry corresponding to the actual dimensions was designed, and the material and initial conditions corresponding to the actual values were defined. The results of the numerical simulation also show that the effect of radiation is significant and must be considered.

The numerical simulation was compared with experimental data from measurements on a measuring stand. The construction of the experimental stand is similar to the conditions in the numerical simulation. The geometries and materials were chosen similar to the numerical simulation. The conversion of the 3D measuring stand into a 2D task was performed using ceramic fibre insulation and partitions separating the interior space. Contact surfaces and sidewalls were insulated to achieve heat sharing only in the radial direction. Heating was provided by a Dawell Inverter Heater. Temperatures were recorded with 6 K-type thermocouples equally spaced on each tube. The obtained experimental data were compared with data from numerical analysis.

The results of the numerical simulation of the temperature field are higher than the measured application temperatures. The reasons for these differences were evaluated. An adverse effect in terms of experimental and calculated temperature differences is probably non-compliance with the conditions of pure natural convection during the measurement (especially at the beginning of the measurement). Furthermore, it was not possible to maintain the heat transfer by conduction at the connecting screws and the outer shell of the assembly. However, the results show the possibility of numerical simulation of cooling a turboprop engine, determination of its temperature field for the purposes of other temperature-dependent phenomena under real environmental conditions. The results can significantly affect the service life of individual engine parts and its closer understanding of the issue of temperature-dependent processes.

![Figure 13. Comparison of numerical and experimental temperatures on case tube](image-url)
Acknowledgment

Authors acknowledge support from the ESIF, EU Operational Programme Research, Development and Education, and from the Centre of Advanced Aerospace Technology (CZ.02.1.01/0.0/0.0/16_019/0000826), Faculty of Mechanical Engineering, Czech Technical University in Prague.

References

[1] S. Solnař, L. Popelka, M. Dostál, Cooling turboprop engine during shutdown. 13th International Conference on Experimental Fluid Mechanics (EFM 2018): EPJ Web of Conferences Volume 213, 2018. ISBN 9781510889668.

[2] E. Abu-Nada, Z. Masoud, and A. Hijazi, Natural convection heat transfer enhancement in horizontal concentric annuli using nanofluids [online]. International Communications in Heat and Mass Transfer 35, pp. 657-665. Jordan: elsevier, February 2008. Available at: https://doi.org/10.1016/j.icheatmasstransfer.2007.11.004.

[3] D. F. Francis, T. M. Itamura, W. S. Webb, L. D. James, CFD Calculation of Internal Natural Convection in the Annulus between Horizontal Concentric Cylinders [online]. Sandia National Laboratories Albuquerque, New Mexico and Livermore, California: Sandia National Laboratories, October 2002 [Accessed 2021-5-23]. Available at: https://core.ac.uk/download/pdf/208903821.pdf

[4] K. Ranganathan, Study of natural convection in horizontal annuli [online]. Int. J. Heat Mass Transfer Vol. 30 No. 6, pp. 1137-1148. Clemson, U.S.A.: Peergamon Press, 1988. Available at: https://doi.org/10.1016/0017-9310(88)90056-7.

[5] T. H. Kuehn, R. J. Goldstein, An experimental and theoretical study of natural convection in the annulus between horizontal concentric cylinders, .Z. Fluid Mech. 74, pp. 695-719, 1976.

[6] T. H. Kuehn, R. J. Goldstein, Correlating equations for natural convection heat transfer between horizontal circular cylinders, Int. Heat Mass Transfer 19, pp. 1127-1134, 1976.

[7] D. B. Fant, The numerical and analytical study flow instability due to natural convection between narrow horizontal isothermal cylindrical annuli at high Rayleigh numbers [online]. Iowa: Iowa State University Capstones, 1987 [Accessed 2021-5-10]. Available: https://lib.dr.iastate.edu/cgi/viewcontent.cgi?referer=https://www.google.com/&httpsredir=1&article=12676&context=rtd.

[8] M. C. Charter-Mojtabi, A. Mojtabi, J. P. Caltagirone, Numerical solution of a flow due to natural convection in horizontal cylindrical annulus, Jounal of Heat Transfer 101, pp. 171-173 (1979)

[9] ANSYS, Inc. Natural Convection and Buoyancy-Driven Flows - ANSYS Fluent. Section 13.2.5, 2006a.

[10] ANSYS, Inc. Surface-to-Surface (S2S) Radiation Model Theory - ANSYS Fluent. Section 13.3.7, 2006c.

[11] M. Hodek, Bachelor’s thesis, Czech Technical University in Prague (2018) Available: https://dspace.cvut.cz/handle/10467/79943.