

Aircraft Radial Engine CFD Cooling Model

Konrad Pietrykowski and Tytus Tulwin
Lublin University of Tech.

ABSTRACT

The article presents convective heat transfer phenomenon by analytically and empirically taken data and CFD based model analysis. 1000 hp ASz-62IR aircraft radial engine is the object of research. This engine is being continuously operated on M18 Dromader and AN-2 aircraft. To recount heat oriented phenomena a three-dimensional CFD model was developed that accounts circumfluent flow around cylinder and cylinder head engine surfaces. The geometry includes M18 Dromader frontal airframe elements to account their influence on cooling air flow. The simulation has been conducted as a steady-state flow.

Geometry and setup specific swirls and backflows were observed that increase cylinder and cylinder head rear side heat transfer coefficients.

Flow along cooling fins was analysed, connecting their heat transfer coefficient dependency. Results show that local air velocity has big influence on heat flux passed by fin walls. Additionally fin performance has been calculated in means of effectiveness and efficiency.

Model verification base on engine's inner walls heat transfer coefficients comparison with various empirical models that link different engines sizes and types heat transfer trends. The following models were presented: Eichberg's, Hohenberg's, Sitkei's. Essential physical gas parameters required by those models were calculated with aid of 0/1D engine cycle computer simulation.

CITATION: Pietrykowski, K. and Tulwin, T., "Aircraft Radial Engine CFD Cooling Model," *SAE Int. J. Engines* 8(1):2015, doi:10.4271/2014-01-2884.

INTRODUCTION

Air cooled engines are dominant on aerospace market emphasising the need for air-cooling analysis. Not many similar heat transfer studies are available in entire engine scale.

Complete elimination of water cooling system results in statistically 20% less engine malfunctions, what is very favourable in aviation because of safety. Bigger temperature difference between the coolant and the cooled surfaces cause good heat transfer in large ambient temperature range. For example in case when ambient temperature rises from 0 °C to 40 °C under assumption that the wall surface temperature is equal to 180 °C, in the water cooled engine temperature difference will drop by 50% while in the air cooled engine it will drop just by 22%. This happens despite much larger heat transfer coefficient of liquid coolant in water cooled engines because heat is transferred indirectly there-first from engine to liquid and then from liquid to air through radiator.

In water cooled engines the cooling agent flows through channels in engine block evenly receiving heat from cylinder and cylinder head. This is influenced by beneficial cooling channels distribution. The heat transfer coefficient is relatively constant throughout their length because of steady flow conditions. In air cooled engines the cooling fluid flows around surfaces being cooled unevenly, because of broad aerodynamic flow phenomena. In different locations of surfaces being cooled the flow properties can vary diametrically. In such structures conducting the flow of the coolant in even manner is limited. Changing direction, pressure and character of the flow result in increased aerodynamic drag and flow kinetic energy losses, decreasing the flow velocity, what translates to decreased heat transfer coefficient.

The heat must be taken away from the system, so the engine temperature doesn't raise above permitted values of given materials, and so the temperature stays in the range of good lubrication conditions. Otherwise excessive cylinder wear will

be present. Local areas of high temperature can cause engine knocking phenomenon negatively affecting the engine running. The value of temperature depends on equilibrium between heat transferred from hot combustion gases to inner walls and heat transferred from outer finned surfaces to the air. Knowing that by forced convections the heat transfer coefficient can vary between 50 and 500 W m⁻² K⁻¹, because of limited influence on temperature differences it is necessary to employ finned surfaces with large areas for heat transfer.

The aim of this study is to analyse the effectiveness of the air cooling system in means of heat transfer evenness and its intensity. To achieve this aerodynamic phenomena where analysed that exist on engine's outer surfaces during circumferential flow. Mathematical 3D-CFD model of engine has been developed. The geometry consist of engine components and frontal aircraft elements.

Case Description

The geometrical model consist of cylinder and cylinder head with thickness defined for heat conduction numerical calculations. Rest of the walls defined such as aircraft elements, push rod covers, transmission cover, rocker box covers, exhaust and intake manifolds are set as adiabatic walls. Those elements form the flow aerodynamic properties.

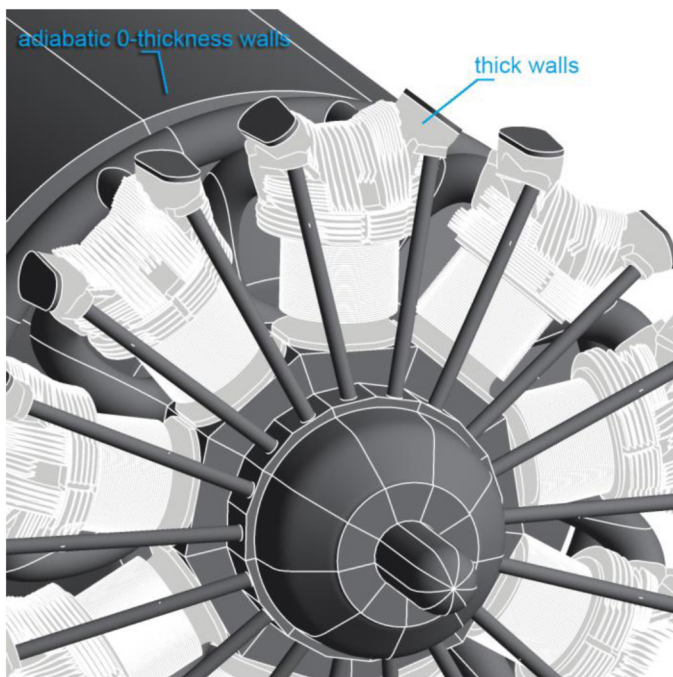


Figure 1. CFD geometrical model composition: Thick walls representing domain for conductive heat transfer analysis; Adiabatic 0-thickness walls are mainly for the aerodynamic flow properties formulation.

Considering that the radial engine consists of symmetrically distributed cylinders relative to the crankshaft axis, developing the model of one cylinder sector can be justified and doesn't lead to large error. Because of that assumption the computational time required for analysis time by the factor of 9.

Table 1 presents the ASh-62IR engine specifications. In simulation study the engine running has been conducted under take-off power conditions. In that state the engine heat transfer is most intense. (Table 2)

Table 1. ASh-62IR engine specifications

Designation	ASh-62IR
Cooling system	air-cooled
Number of cylinders	9
Bore diameter	155,5 mm
Stroke	174,5 mm
Summarized displacement	29,87 l
Compression ratio	6,4 ± 0,1
Fuel type	aviation gasoline Avgas 100 LL

Table 2. Engine working conditions.

Rotational speed n	2100 rpm
Take-off power Ne	721 kW
Brake specific fuel consumption Ce	300 g/KMh
Air-fuel ratio λ	0.85

3D-CFD Model Setup

In order to recreate thermal and flow conditions in engines cooling simulation beside the geometry proper boundary conditions had to be set. The propeller situated in front of the engine forces the flow of air. It's crucial to know the velocity of air under take-off conditions in order to properly set the boundary conditions. The velocity has been calculated using Froude's momentum theory. Constant temperatures has been set on inner cylinder, cylinder head and exhaust channel surfaces (Figure 2). Cylinder head temperature come from empirical measures in spark plug location [6]

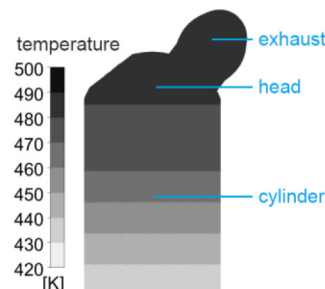


Figure 2. Temperature boundary conditions on exhaust channel, cylinder and cylinder head inner surfaces. Front view.

THEORY

Conduction and Convection

In the process of air cooling the heat is transferred from hot combustion gases to inner cylinder surfaces, then it flows through material by conduction according to the Fourier's law in the temperature gradient direction. After that it's again transferred by convection from outer engine's finned surfaces to the air. The direction and intensity of heat transfer in engine is dependent on many variables. It's difficult and non-practical to accurately depict designate them analytically, especially when it comes to complicated geometries. Effectively the heat transfer can be calculated with aid of CFD iterative methods.

Convective Heat Transfer Coefficient

Convective heat transfer coefficient is dependent on many variables (Even 15 according to some sources [2]) making it difficult to designate. After discounting the indirect influence of temperature on the value of the heat transfer coefficient it can be assumed that it is not temperature dependant.

In CFD simulation resultant heat transfer coefficient had been assumed basing on heat flux calculations according to [Eq. \(1\)](#). Because that coefficient had been calculated "backwards", there is an error present in that methodology as the known heat flux is temperature gradient dependant. The biggest error arise in areas of low heat flux values. Furthermore the validity of the heat transfer coefficient is dependant on reference temperature T_g . The method of defining its value is dependent on flow type. For engine cooling circumferential flow it can be assumed as ambient air temperature. For inner cylinder surfaces the reference temperature is set as an average gas temperature.

$$h = \frac{q}{T_g - T_w} \quad (1)$$

Fins Performance

Heat flux along the length of the fin decreases the further away from its base because the temperature gradient between fin and air decreases. Use of oversized fins is not reasonable in means of the heat transfer as the efficiency gets very low. Ideal fin with efficiency value of one has constant temperature along its length. In order to find out dependencies that depict the efficiency of the fin, the case of infinite length tip boundary condition fin has been analysed. The efficiency equation can be then simplified to:

$$\eta_f = \frac{1}{L} \cdot \sqrt{\frac{k \cdot A_c}{h \cdot P}} \quad (2)$$

Thus its clear, that by increasing the thermal conductivity of the fin its efficiency increases. Therefore use of materials with high thermal conductivity like copper or aluminium is most beneficial in means of heat transfer. For efficiency it is also advisable to maximize fins cross-sectional area to its circumference length ratio. However increasing that value can lead to adverse heat flow character (decreased heat transfer coefficient) and decreasing fin effectiveness. Therefore fin design is the matter of compromises and optimizations based on project demands.

Increasing the fin length L decrease the fin efficiency but increase its effectiveness. Because the effectiveness can be presented as a function of efficiency, its growth decrease as the length is increased as seen in [eq. \(3\)](#).

$$\varepsilon_f = \eta_f \cdot \frac{\text{surface area of the fin}}{\text{cross-sectional area of the fin}} \quad (3)$$

In case of infinite length tip BC fin the [eq. \(3\)](#) can be transformed to :

$$\varepsilon_f = \sqrt{\frac{k \cdot P}{h \cdot A_c}} \quad (4)$$

[Equation \(2\)](#) shows that by increasing the heat transfer coefficient the efficiency decrease. It may seem non-intuitive but can be explained by the purpose of the fin definition - In high heat transfer coefficient flows there is less need of finned areas in means of heat transfer. [Equation \(4\)](#) shows that to increase the fin effectiveness the circumference of cross-section to cross-sectional area ratio need to be increased. Thus the most effective would be dense, closely spaced thin fins (keeping in mind that too small gap can lead to decreased heat transfer coefficient). It can be assumed that in order to justify using fins their effectiveness has to be greater than 2 [3] [4].

Model Validation

The greatest advantage using the dimensional analysis is to develop a functional form of relationships which govern a special process mechanism. Therefore, even in an extremely complex phenomenon, like the engine heat-transfer, it is possible to identify a limited number of critical variables, which are predominately responsible for the process [5]. Following this phenomenological heat transfer models the choice of characteristic length, characteristic velocity and relevant gas temperature are made. The model validation is based on inner cylinder walls heat transfer coefficient comparison of many heat transfer models to 3D-CFD model.