Numerical simulations of turbulent convective heat transfer

Large eddy simulations on heated rough surfaces for external cooling of fuel-cell powered aircraft engines

***D. Sotomayor-Zakharov**, Institut für Strömungsmechanik, Technische Universität Braunschweig*

In Short

- Numerical investigation of the convective heat transfer on the external flow over heated rough surfaces using large eddy simulations.
- Roughness patterns that maximize the increase of the convective heat transfer and minimize the increase of skin friction are studied.
- Modeling approaches of these surfaces are investigated to allow their representation via parameters that can be implemented into RANS tools for the numerical simulation of complex geometries such as nacelles.

The FAME project, which stands for "Fuel cell propulsion system for Aircraft Megawatt Engines", presents as an objective the multidisciplinary design optimization of a 2.5 MW fuel cell powered engine for a novel aircraft configuration, this as part of the development of technologies that offer the possibility to significantly reduce or eliminate aircraft's greenhouse gas emissions in the near future. The FAME engine presents several systems required to be designed under certain specifications. Among these systems stands the cooling system of the fuel cell, which is necessary to keep it under normal operating conditions, meaning that the excess heat that is generated by it must be eliminated. This excess heat is considerable since fuel cells can present efficiencies around 30%, implying an excess heat of more than 2.5 MW for the FAME engine. As a straightforward approach, a traditional air heat exchanger is sought to be designed. This typical system consists of disposing the excess heat into a radiator, through which an air flow is passed. Design of a proper inlet and outlet on the nacelle is required, although it can be deduced that this system would add drag to the nacelle, therefore, to the entire aircraft, apart from weight and volumetric aspects related to the relevant systems.

As an alternative to this traditional heat exchanger, an approach of using the skin of the nacelle as an external heat exchanger is explored. Initial simulations of the turbulent boundary layers on flat plates have shown that the excess heat would tend to increase the temperature of the adjacent boundary layer, resulting on a reduction of the viscous component of the total drag force, as shown in figure 1. Apart from this drag reduction, an additional drag reduction exists if this system supports the traditional heat exchanger, meaning that the latter can be reduced in size or even completely disregarded.

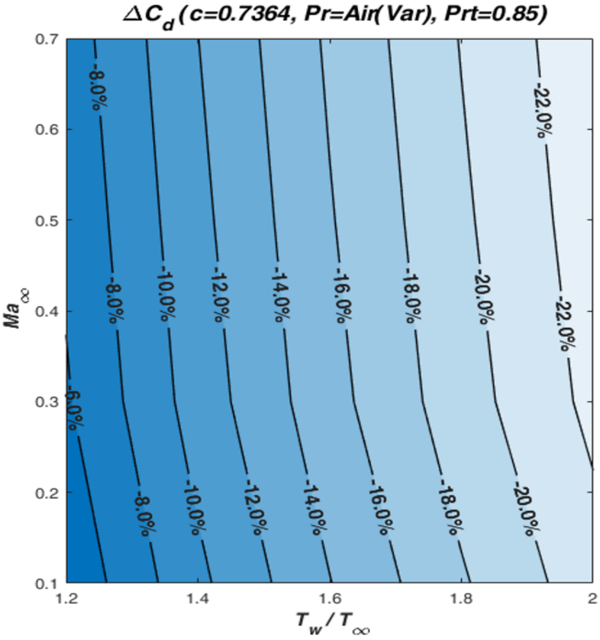

***Figure 1:** Relative reduction of the total drag on a flat plate based on the Mach number Ma_∞ and the temperature ratio between the plate and the freestream T_w/T_∞.*

Unfortunately, the external heat exchanger approach presents several constraints, which can be observed in equation 0.1. Assuming Q_W as the excess heat, this can only be conveyed through a limited surface area A_T that the nacelle presents. Also, ΔT is limited due to the maximum temperature that the fuel cell can achieve, not allowing for ratios of T_w/T_∞ larger than 2. Calculations have shown the results displayed in figure 2, showcasing the effect of said limitation: just a small portion of the excess heat can be evacuated. Nevertheless, the total heat transfer coefficient HTC_T stands as a factor that can be explored and exploited.

$$Q_W = HTC_T \, A_T \, \Delta T \qquad (0.1)$$

Studies[1] have shown that HTC_T depends on the dynamics between the boundary layer and the surface, which can be altered by the presence of rough patterns: roughness tends to increase the

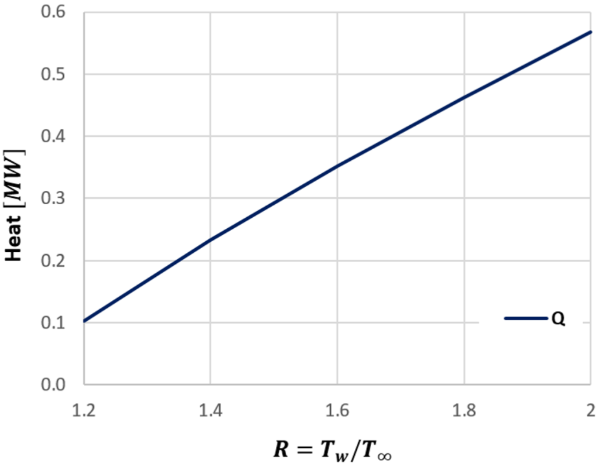

Figure 2: *Amount of excess heat Q_W that can be evacuated based on the ratio T_w/T_∞.*

wetted area of the surface, implying an enhancement of HTC_T. This can prove advantageous, although a consequence is that the wall skin friction is enhanced as well, therefore, the drag on the surface eventually increases. Consequently, it is necessary to look for roughness patterns that can maximize the increase of the convective heat transfer while minimizing the increase of the wall skin friction.

The issue of this quest is that the interaction of turbulent boundary layers with rough surfaces presents complex dynamics, for which modeling attempts have been made, implying roughness characterization and correlations of statistical surface parameters to physical effects. While these approaches are somehow inaccurate on the side of the wall skin friction, they are almost scarce on the side of the wall heat transfer. This is why simple tools such as RANS present issues due to the lack of precise modeling and understanding of the complex mechanisms present in the interactions of the flow with the rough surface, especially on heat transfer. This justifies using high-fidelity approaches such as LES to obtain a better description of the flow and, by extension, an accurate quantification of the effect of roughness on heat transfer.[2]

The project looks towards the calculation of the heat transfer on generated rough surfaces via embedded LES, aiming to look for a surface pattern that can be used to evacuate a large portion of the fuel cell excess heat under the prescribed constraints. Initially, rough patterns are simulated on simple flat plate configurations, from which roughness characterization[3] can be applied to correlate geometrical properties to the heat transfer enhancement via parameters such as the equivalent sand-grain height k_s, as displayed in figure 3. Later, these rough patterns can be implemented on the nacelle geometry, obtaining more realistic results.

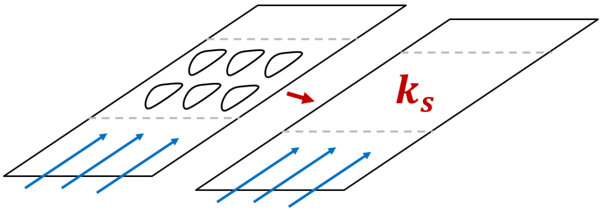

Figure 3: *Roughness characterization: Simplification of a complex rough surface into a smooth surface with an equivalent sand-grain height k_s.*

All of the results given by the simulations can act as a database from which RANS roughness models can be improved for their application in the design of the nacelle and, eventually, for the accurate calculation of the effect of the boundary layer heating on its aerodynamic characteristics such as the previously observed drag reduction, while managing to evacuate as much excess heat as physically possible.

WWW

`https://cordis.europa.eu/project/id/101140559`

More Information

[1] W.M. Kays, M.E. Crawford, *McGraw-Hill* (1993).

[2] D. Sotomayor-Zakharov, *Ph.D. Thesis, NFL* (2024). `doi:10.24355/dbbs.084-202401191241-0`

[3] D. Sotomayor-Zakharov, E. Radenac, M. Gallia, A. Guardone, I. Knop, *J. Aircraft* **61**, 1 (2024). `doi:10.2514/1.C037403`

Project Partners

AIRBUS, DLR and other companies and universities across countries such as Germany, France, Italy and Spain.

Funding

Horizon Europe, Grant agreement ID: 101140559

DFG Subject Area

404-03 Fluid Mechanics