

ISABE-2009-

Heat and Fluid Flow investigations on a heat exchanger for aero engine applications

D. Missirlis^a, O. Seite^b, S. Donnerhack^b and K. Yakinthos^a

^aLaboratory of Fluid Mechanics & Turbomachinery
Dept. of Mechanical Engineering
Aristotle University of Thessaloniki, 54124, Thessaloniki, GR

^bMTU Aero Engines GmbH
Dachauerstr. 665
80995, Munich, D

Abstract

This work presents an effort to optimize the flow field development and minimize the pressure losses in an advanced aero engine. The aero engine is using an innovative alternative thermodynamic cycle and an integrated system of heat exchangers installed in its exhaust nozzle in order to exploit the thermal energy of the turbine exhaust gas and decrease fuel consumption and pollutants. Since the presence of the system of heat exchangers enforces a significant pressure loss inside the installation it is important to optimize the flow field inside the installation. Towards this direction, 3D CFD computations and experimental data are used. For the optimization procedure, the heat exchangers are modelled as porous media with predefined pressure drop and heat transfer behaviour. The computational results are validated towards experimental measurements and provide satisfactory agreement regarding the pressure losses, the mass flow distribution and the overall flow field development. Additionally, 3D CFD computations are performed for the real engine operating conditions and based on the conclusions of these computations, modifications are

incorporated in an optimized 3D CFD model of the overall installation. The CFD results show improvement of the flow distribution inside the installation, since the pressure losses are decreased and the mass flow balance through the heat exchangers is more uniform in comparison with the initial nozzle design.

Nomenclature

a	constant coefficient in the Nusselt number correlation
D_{eq}	characteristic equivalent diameter of the elliptic tube of the heat exchanger
D'_{ref}	coefficients in the 1 st order pressure drop coefficient matrix
F'_{ref}	coefficients in the 2 nd order pressure drop coefficient matrix
m	constant coefficient in the Nusselt number correlation
$MX_{1,2,3,4}$	mass flow percentage through heat exchanger matrices 1,2,3,4
n	constant coefficient in the Nusselt number correlation

Nu	Nusselt number
P	Static pressure
Pr	Prandtl number
Re	Reynolds number
u	x-velocity component
$ \vec{v} $	velocity vector magnitude
v	y-velocity component
w	z-velocity component

Introduction

In the competitive and rapidly developing world of today, the need for environmentally friendly air transport, reliable and cost efficient, is more evident than ever before. For these reasons, alternative technologies are to be invented, developed and incorporated in the design of aero engines. At the present work, an advanced aero engine using an innovative alternative thermodynamic cycle using intercooling and recuperation and which is having an integrated system of heat exchangers installed in its exhaust nozzle is investigated, as presented in Fig.1.

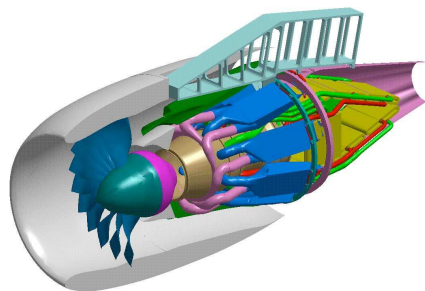


Fig. 1. The IRA - Intercooled Recuperative Aero-engine Concept

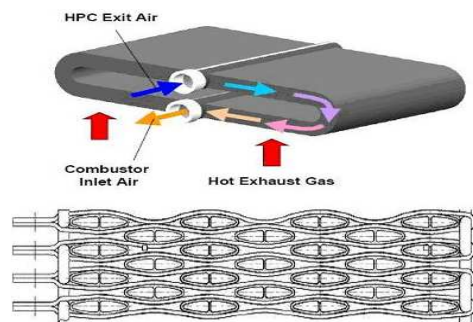


Fig. 2. The heat exchanger geometry

The heat exchangers of the installation, presented in Fig.2, are operating as heat recuperators exploiting the thermal energy of the turbine exhaust gas to pre-heat the compressor outlet air before combustion, aiming at the decrease of fuel consumption and the reduction of pollutants. Additional details about the heat exchanger geometry and of this technology are presented in the works of Broichhausen et al. [1], Wilfert and Masse [2], Schonenborn et al. [3], Missirlis et al. [4], Yakinthos et al. [5], Yakinthos et al. [6] and Albanakis et al. [7].

However, the presence of the system of heat exchangers enforces a significant pressure loss inside the installation, which can affect the overall cycle benefit and operation of the aero engine. Thus, in order to maximize the benefit of this technology, it is important to optimize the flow field inside the installation.

An important first attempt to optimize the flow field inside the aero engine installation was presented in the works of Yakinthos et al. [5], Yakinthos et al. [6]. In these works, an isotropic porous medium model was used to describe the pressure drop behavior of the heat exchanger and investigate the flow field development. Based on this approach modifications were suggested inside the aero engine installation in order to optimize the flow field development and to minimize the pressure losses. However, these investigations were using a 2D approach in order to optimize a 3D case. Thus, even though the results were qualitatively in very good agreement towards the experimental measurements it was necessary, at the next step, to proceed to 3D CFD computations in order to quantify the actual benefit of the optimization. Additionally, it was important to proceed with CFD computations for real engine

operating conditions since so far CFD computations were performed only for isothermal laboratory conditions.

Such an effort is presented in this work where 3D CFD computations and experimental data are used to redesign and optimize the installation design in order to balance the mass flow distribution through the heat exchangers and reduce the pressure losses.

For the optimization procedure, the heat exchangers are modelled as porous media with predefined pressure drop and heat transfer behaviour based on the detailed experimental measurements presented in Albanakis et al. [7]. Based on these measurements a fully anisotropic porosity model was developed in the form of a modified formulation of the Darcy-Forchheimer pressure drop law which could be combined with a heat transfer model in the form of a Nusselt-Reynolds-Prandtl numbers correlation so as to describe accurately both the pressure drop and heat transfer behaviour of the heat exchanger. Additional details can be found in the work of Yakinhtos et al. [9]. However, at the present work an isotropic version of this model was used for simplicity reasons and since the results of the previous CFD computations showed that an isotropic model could be sufficiently accurate to be used in an optimization effort together with the heat transfer model.

The developed porosity model was applied in 3D CFD computations for a 1:1 model of the quarter nozzle installation (reference case) of the aero engine similar to the laboratory experimental setup. For this configuration, the orientation of the heat exchangers 1/2/3/4 inside the installation is at 0/20/20/0 degrees as presented in Fig.3. Then, CFD computations were performed for laboratory conditions for validation purposes and at the next stage, also for the real engine operating conditions. Based

on the conclusions of these computations, optimization suggestions were made for the nozzle installation, and modifications were incorporated in an optimized 3D CFD model of the installation in real engine operating conditions. The main targets of this effort were focused on the decrease of the pressure losses and the uniformization of the mass flow balance between the heat exchangers of the installation.

Computational fluid dynamics modelling

Due to the complexity of the aero engine installation geometry, the creation of a proper computational grid is particularly important. The computational grid should be capable of combining accurate predictions of all the important aspects of the flow field development together with a good convergence rate and grid independent results. For this reason, various computational grids were examined and evaluated. The computational grid which was finally selected was using approximately 3 million points and typical views of which are presented in Fig. 3. The computational grid was created with the use of the Numeca/IGG commercial software.

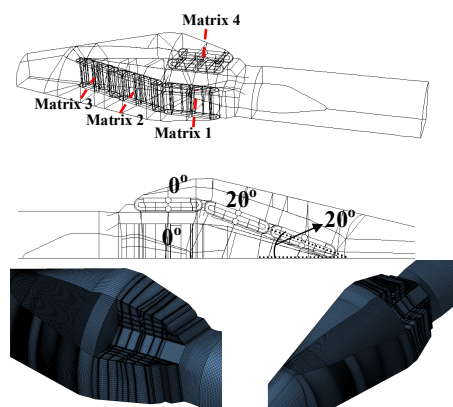


Figure 3. Perspective view of the 3-D geometry and the computational grid for the reference case

(The orientation of the heat exchangers is at 0/20/20/0)

Since, based on the conclusions of the experimental measurements, presented in Yakinthos et al. [5] and Yakinthos et al. [6] a significant amount of separation is expected, it was decided that the computations should be performed with the use of Low-Re turbulence models as being more appropriate for separated boundary layer flows. Hence, special care was given to the computational grid construction for the regions near the solid walls and to the regions for which significant flow phenomena were detected in the laboratory measurements. Due to the relative complexity of the geometry during the grid creation it was necessary to use a large number of structured computational blocks.

After the grid creation, the same boundary conditions as the ones measured for the isothermal laboratory conditions, were applied to the computational domain, as presented in Fig.4.

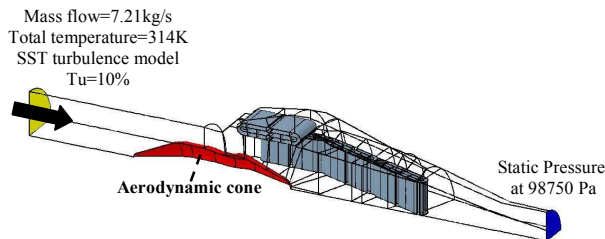


Figure 4. Boundary conditions used for the CFD computations

For the regions of the CFD model occupied by the heat exchangers a porous media model of an isotropic modified formulation of the Darcy-Forchheimer was applied, as presented in Eq.1. The CFD computations were performed with the use of the SST (Shear Stress Transport) turbulence model with the commercial CFD software FLUENT.

$$\begin{pmatrix} \frac{\partial P}{\partial x} \\ \frac{\partial P}{\partial y} \\ \frac{\partial P}{\partial z} \end{pmatrix} = \begin{pmatrix} D'_{ref} & 0 & 0 \\ 0 & D'_{ref} & 0 \\ 0 & 0 & D'_{ref} \end{pmatrix} \begin{pmatrix} u \\ v \\ w \end{pmatrix} + \frac{1}{2} |\vec{V}| \begin{pmatrix} F'_{ref} & 0 & 0 \\ 0 & F'_{ref} & 0 \\ 0 & 0 & F'_{ref} \end{pmatrix} \begin{pmatrix} u \\ v \\ w \end{pmatrix}$$

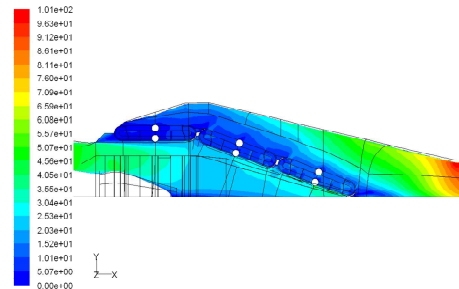
Eq.1

Comparative results between the experimental measurements and the CFD results are presented in Table 1. As it can be seen the mass flow distribution is in reasonably good agreement with the experimental measurements. Furthermore, the total pressure difference between the CFD results and the experimental measurements is approximately 3.33%

	MX1	MX2	MX3	MX4
LAB	17.4	27.1	37.9	17.6
CFD 3D	13.9	29.8	39.0	17.2

Table 1. Mass flow distribution percentage (%) between the heat exchangers of the installation

Typical plots of the flow field are presented in Fig.5 for a characteristic plane of the nozzle. The same flow patterns as presented in the experimental measurements of Yakinthos et al. [5], [6], are presented. Hence, it can be concluded that the 3D CFD model with the isotropic porosity model can be used to sufficiently describe the flow field and the pressure loss of the aero engine installation.



a) Velocity magnitude colour contour

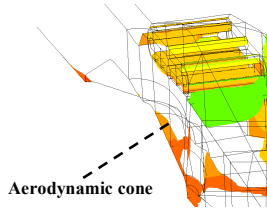
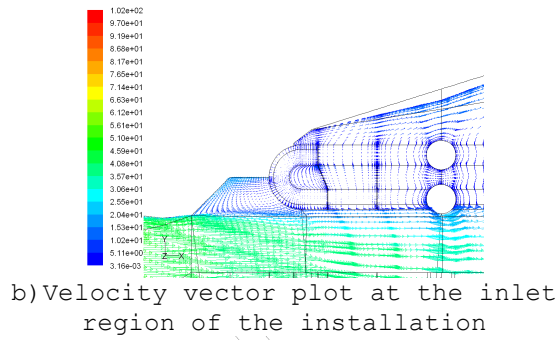


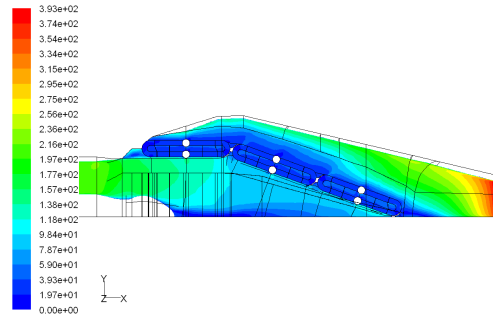
Figure 5. Typical plots of the flow field development inside the installation reference geometry

At the next step of the investigation it was necessary to proceed with CFD computations for real engine operating conditions where the heat transfer effect should be included. For this reason the heat transfer model, presented in detail in the work of Yakinhtos et al. [9], was used together with the isotropic porosity model. The heat transfer model is in the form of a Nusselt-Reynolds-Prandtl numbers correlation, presented in Eq.2.

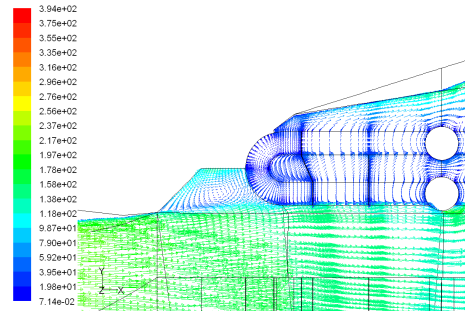
$$\overline{Nu}_{D_{eq}} = a Pr^m Re^n \tag{Eq.2}$$

Regarding the boundary conditions applied for the CFD computations with the real engine operating conditions at the inlet of the computational domain the mass flow and the total temperature was prescribed while at the outlet the static pressure was defined. For the heat transfer model, the

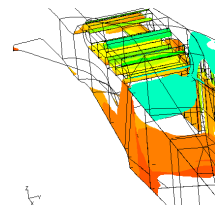
necessary values of the wall temperature were provided through computations with the use of a finite element model which was developed with MSC/Patran and presented in detail in the work of Schonenborn et al. [3].



a) Velocity magnitude colour contour



b) Velocity vector plot at the inlet of the installation



c) Isosurfaces of zero axial velocity
(the size of the coloured regions indicates the separation size on the aerodynamic cone surface)

Figure 6. Typical plots of the flow field development inside the installation reference geometry for engine conditions

As it can be seen the flow field is very similar to the one presented in the experimental measurements of the laboratory conditions since the mass flow distribution through the heat exchangers is not uniform (even though not as non-uniform as in the laboratory conditions case) and the same flow patterns are presented. Based on the CFD results, modifications were suggested and incorporated inside the computational model, as presented in Fig. 7, and the computation was performed again.

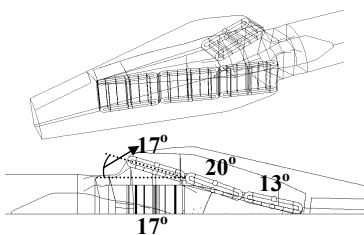
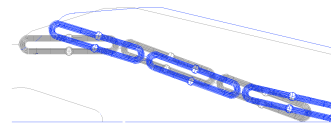


Figure 7. Optimized nozzle geometry

The most important modifications, presented in Fig.8 can be summarized as:

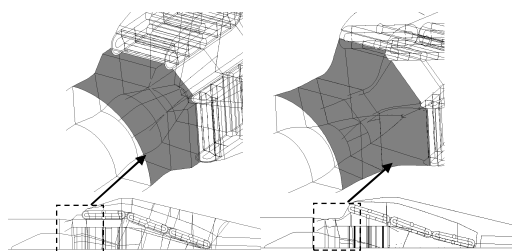
- the orientation of the heat exchanger matrices is now at 17/20/13/17 degrees instead of 0/20/20/0 in the reference nozzle geometry
- the aerodynamic cone has been redesigned so as to minimize the recirculation downstream the cone
- the guiding walls at the inlet region of the installation and the guiding walls at the outlet region have been remodified
- the covers of the bow region of the heat exchangers have been modified in order to uniformize the flow velocity inside each heat exchanger



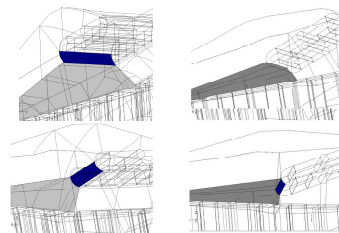
Modifications in the orientation of the heat exchangers
(Grey:Reference Blue:Optimized)



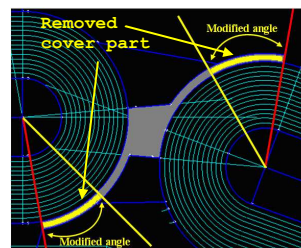
Modifications in the design of the aerodynamic cone
(Grey:Reference Blue:Optimized)



Inlet region guiding walls modifications
(Left:Reference Right:Optimized)

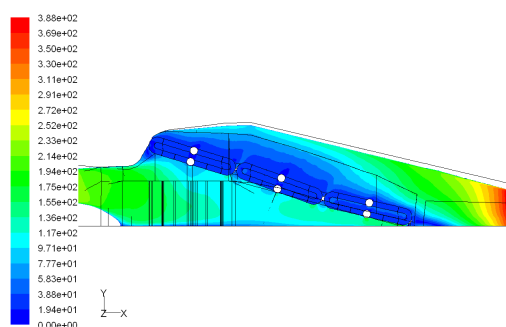


Outlet region guiding walls modifications
(Left:Reference Right:Optimized)

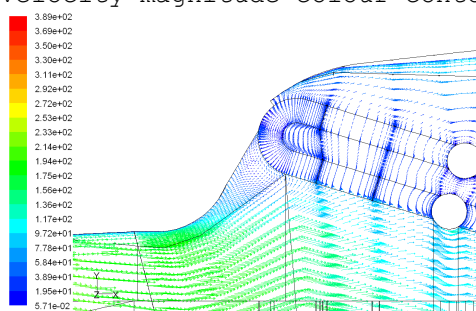


Modifications in the covers of the bow region of the heat exchangers
(Grey and yellow part: Reference cover, Yellow part: Removed cover part after modification, Grey part: remaining cover part used in Optimized case,)

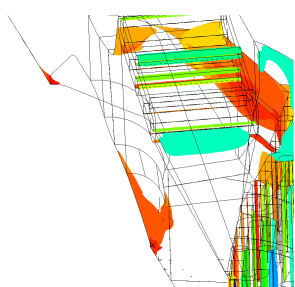
Fig.8 Modifications in the optimized nozzle geometry in relation to the reference nozzle geometry



a) Velocity magnitude colour contour



b) Velocity vector plot at the inlet of the installation



c) Isosurfaces of zero axial velocity

(the size of the coloured regions indicates the separation size on the aerodynamic cone surface)

Figure 9. Typical plots of the flow field development inside the installation optimized geometry for engine conditions

Typical plots of the CFD computations are presented in Fig.9.

Engine Conditions CFD	$RMS\ deviation(MX_i,25) = \sqrt{\frac{\sum_{i=1}^{i=4} (MX_i - 25)^2}{4}}$
Reference nozzle installation	2.502
Optimized nozzle installation	1.129

Table 2. Mass flow balance improvement for the optimized geometry

The improvement in the mass flow distribution between the heat exchangers of the installation is presented in Table 2. The comparison is performed based on the root-mean-square (RMS) deviation value of the mass flow percentages through each one of the heat exchangers, MX_i , in relation to the 'ideal' mass flow balance of 25% per heat exchanger. As it can be seen, the optimized case has a lower RMS deviation value indicating the improvement in the uniformization of the mass flow balance which is now more uniform in comparison with the initial nozzle installation. Furthermore, it must also be mentioned that in relation to the initial design an improvement of approximately more than 9% has been achieved for the total pressure losses.

CONCLUSIONS

At the present work an effort to optimize the flow field inside the exhaust nozzle of an aero engine using an integrated system of heat exchangers is presented. The optimization was performed with the use of CFD modelling. The presence of the heat exchangers has been modelled with the use of a porous medium methodology. The CFD results were validated towards experimental measurements with satisfactory results. Since the computational model was sufficient in accurately describing the flow field inside the aero engine, additional computations were performed for real engine operating conditions.

Based on the conclusions of the CFD results, modifications were suggested and incorporated in the exhaust nozzle installation. At the final step, CFD computations were performed for the optimized nozzle installation. The modifications proved to be successful since in relation to the reference nozzle installation the mass flow balance was improved and the total pressure losses were reduced by approximately 9%.

ACKNOWLEDGMENTS

A major part of this work has been financially supported by the E.U. under the 'Sixth framework programme Priority 4 - Aeronautics and Space FP6-2005-Aero-1' in the integrated research project 'NEWAC - NEW Aero Engine Core concepts', contract no. FP6-030876.

REFERENCES

- [1]. K. Broichhausen, H. Scheugenpflug, Ch. Mari, A. Barbot, 2000, "Clean The European Initiative Towards Ultra Low Emission Engines", *ICAS 2000*, Harrogate, UK.
- [2]. G. Wilfert, B. Masse, 2001, "Technology integration in a low emission heat exchanger engine", *Proceedings of the 8th CEAS European Propulsion Forum*, Nottingham, UK.
- [3]. H. Schonenborn, B. Simon, E. Ebert, P. Storm, 2004, "Thermomechanical design of a heat exchanger for a recuperative aero engine", *Proceedings of ASME Turbo Expo 2004, Power for Land, Sea and Air*.
- [4]. D. Missirlis, K. Yakinthos, A. Palikaras, K. Katheder, A. Goulas, 2005, "Experimental and numerical investigation of the flow field through a heat exchanger for aero-engine applications", *Int. J. of Heat and Fluid Flow*, Vol. 26, pp. 440-458.
- [5]. K. Yakinthos, D. Missirlis, A. Palikaras, P. Storm, B. Simon, A. Goulas, 2007, "Optimization of the design of recuperative heat exchangers in the exhaust nozzle of an aero engine", *Applied Mathematical Modelling*, Vol. 31, pp. 2524-2541.
- [6]. K.J. Yakinthos, D.K. Missirlis, A.C. Palikaras, A.K. Goulas, 2006, "Heat exchangers for aero engine applications", *IMECE2006-13667, Proceedings of IMECE 2006, ASME International Mechanical Engineering Congress and Exposition*, November 5-10, Chicago, Illinois USA.
- [7]. C. Albanakis, K. Yakinthos, K. Kritikos, D. Missirlis, A. Goulas, 2009, "The effect of heat transfer on the pressure drop through a heat exchanger for aero engine applications", *Applied Thermal Engineering*, Vol. 29, pp. 634-644.
- [8]. Kyros Yakinthos, Stefan Donnerhack, Dimitrios Missirlis, Olivier Seite, Paul Storm, Derivation of an anisotropic model for the pressure loss through a heat exchanger for aero engine applications, *Proceedings of ASME Turbo Expo 2009: Power for Land, Sea and Air, GT2009*, June 8-12, 2009, Orlando, Florida, USA

Extended Abstract

In the competitive and rapidly developing world of today, the need for environmentally friendly air transport, reliable and cost efficient, is more evident than ever before. For these reasons, alternative technologies are to be invented, developed and incorporated in the design of aero engines. At the present work, an advanced aero engine using an innovative alternative thermodynamic cycle and an integrated system of heat exchangers installed in the exhaust nozzle of the engine is investigated. The heat exchangers of the installation are operating as heat recuperators exploiting the thermal energy of the turbine exhaust gas to pre-heat the compressor outlet air before combustion, aiming at the decrease of fuel consumption and, hence, the reduction of pollutants. However, the presence of the system of heat exchangers enforces a significant pressure loss inside the installation, which can affect the overall operation of the aero engine. Thus, in order to maximize the benefit of this technology, it is important to optimize the flow field inside the installation. Such an effort is presented in this work where 3D CFD computations and experimental data are used to redesign and optimize the installation design in order to balance the mass flow distribution through the heat exchangers and reduce the pressure losses. For the optimization procedure, the heat exchangers are modelled as porous media with predefined pressure drop and heat transfer behavior. For the porosity model, a modified formulation of the Darcy-Forchheimer pressure drop law is combined together with a heat transfer model in the form of a Nusselt-Reynolds-Prandtl numbers correlation. For the derivation of the pressure loss and the heat transfer coefficients and the validation of the porosity model, various sets of isothermal and anisothermal experimental data are used. The developed porosity model is applied in 3D CFD computations for a 1:1 model of the quarter nozzle installation of the aero engine. The computational results are validated towards experimental measurements and provide satisfactory agreement regarding the pressure losses, the mass flow distribution and the overall flow field development. Additionally, 3D CFD computations are performed for the real engine operating conditions and based on the conclusions of these computations, optimization suggestions are made inside the nozzle installation, and modifications are incorporated in an optimized 3D CFD model of the installation in real conditions. The CFD results show significant improvement of the flow distribution inside the installation, since the pressure losses are decreased and the mass flow balance through the heat exchangers is more uniform in comparison with the initial nozzle design.

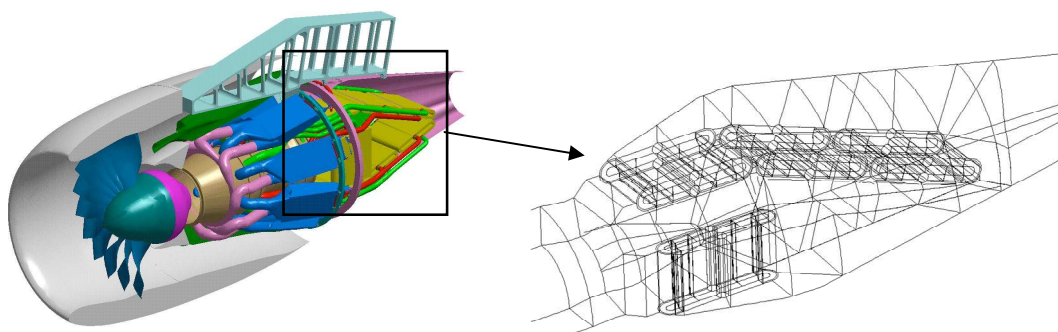


Figure 1. IRA-intercooled recuperative engine (left) and detailed view of the modelled region of the exhaust nozzle with the integrated system of the heat exchangers (right).