Optimisation techniques applied to the design of gas turbine blades cooling systems

D. Coutandin*, L. Bucchieri**, L. Brugali**, M. Galbiati**

* AVIO, Turin, Italy
** ENGINSOFT, Bergamo, Italy

Summary

The multi-objective optimisation techniques were applied to a three-dimensional CFD model for the design of the cooling system of a gas turbine blade tip.
A parametric CFD model of a high pressure rotor blade was developed in order to study the cooling efficiency in the tip area.
The model was integrated in an optimisation software and several different analyses were automatically carried out for different configurations of the system.
The application of a Multi Objective Genetic Algorithm and the use of Neural Net lead to the optimal design with a reduced number of simulations.
The comparison between the optimal and the standard configurations gives evidence that the cooling mass-flow can be considerably reduced, keeping the same wall heat flux on the blade tip surface.
The parametric CFD model was developed with ICEM-HEXA, ICEM-TETRA and CFX5. These codes were linked and integrated in the optimisation software modeFRONTIER.

Keywords

Optimisation, turbine blade, cooling system, Genetic Algorithms, Neural Net

0. Introduction

This study is part of an AVIO project concerning the development of High Pressure Turbine blades with advanced cooling systems. Due to the high gas temperatures entering the turbine of the most recent aero-engines (in general up to 2000 K at the turbine inlet), a very efficient cooling system is required in order to maintain the metal temperatures below the allowable limits. This means to use a big amount of “cold“ air directly extracted from the compressor, and so with significant impact on the engine performance.
One of the most critical area, from a thermal point of view, is the tip region of the unshrouded rotor blades. Tip regions are generally cooled using rotor internal air ejected in the flow path through a series of small holes located in the tip surfaces. The ejected air must cover all the surfaces in order to create a “cold” film between hot gas and metal. As the tip region is characterised by a very complex 3D flow field, it is very difficult to optimise the cooling system using the standard design methodologies, also considering the other blade tip requirements (to minimise the hot leakage air from pressure to suction side, which has a negative impact on turbine aerodynamic efficiency).
That’s why AVIO decided, with the support of EnginSoft, to introduce a stochastical optimisation approach in tip cooling design process.

This kind of approach means to link the optimisation software modeFRONTIER to a 3-D CFD code (CFX5) with the goal to find the optimal values of some geometrical parameters of the tip area of the high pressure rotor blade.

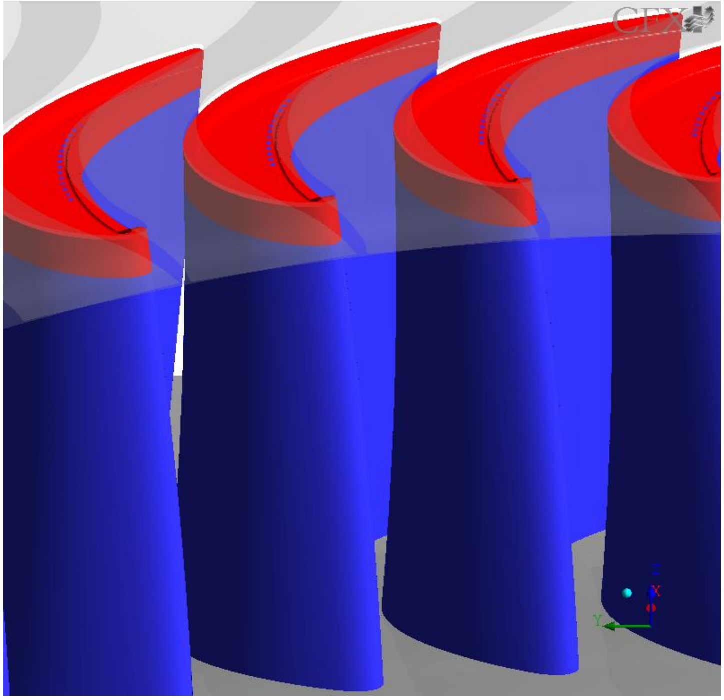

Figure 1: High pressure rotor blade

As a consequence of the geometrical complexity of the problem and of the high computational time, the use of the Response Surface Method is almost compulsory if a 3-D fluid-dynamic optimisation has to be approached.
The Response Surface Method consists in estimating a mathematical model (Neural Nets) which computes the response of the system to a given set of input values.
This way, after a preliminary series of CFD analyses and after the estimation of Neural Nets, the 3-D CFD model can be substituted by a series of mathematical functions and the computational time is considerably reduced.
In our case Neural Nets were used to describe the link between the blade configuration and its performances.
The parametric fluid-dynamic model was built and the blade cooling system was optimised with the following procedure:

1. A series of variables describes the geometry of the tip area and defines the space of possible designs which has to be investigated in order to find the optimal configuration of the system. Three objectives have to be reached at the same time: the cooling mass-flow and the wall heat flux have to be minimised, the blade efficiency has to be maximised.
2. A batch procedure allows the creation of different geometrical models, the mesh generation and the CFD analyses of the blades in an automatic way.
3. A series of preliminary CFD simulations is planned and a screening is performed in order to build an input-output database.
4. Neural Nets (NN) are estimated and are coupled to a Multi Objective Genetic Algorithm (MOGA). The MOGA investigates the space of possible solutions using Neural Nets and a virtual optimisation of the cooling system is performed without carrying out any CFD simulation.
5. The best virtual solutions are selected and the Neural Nets forecast are verified with some CFD analyses.
6. More accurate Neural Nets can now be estimated with a larger database. The virtual optimisation can be executed again and new more performing designs can be found. This procedure is repeated till convergence to the set of optimal solutions.

Following a description of the parametric CFD model, the optimisation procedure and the final results are presented.

2. Parametric CFD model

The CFD model reproduces the entire blade-to-blade channel of an axial un-shrouded turbine rotor stage. A series of cooling holes are explicitly modelled on the tip surface and their distribution is defined by a series of parameters (Table 2). In order to describe far different configurations (the number of holes can vary between 8 and 20) ICEM-TETRA was used to generate a tetrahedral mesh (with 3 prismatic layers adjacent to the walls) in the tip region.
The remaining part of the blade is fixed and was discretised with an hexahedral mesh.
The two fluid domains, with two non-matching mesh, were connected in CFX5 using the general grid interface technique (GGI).

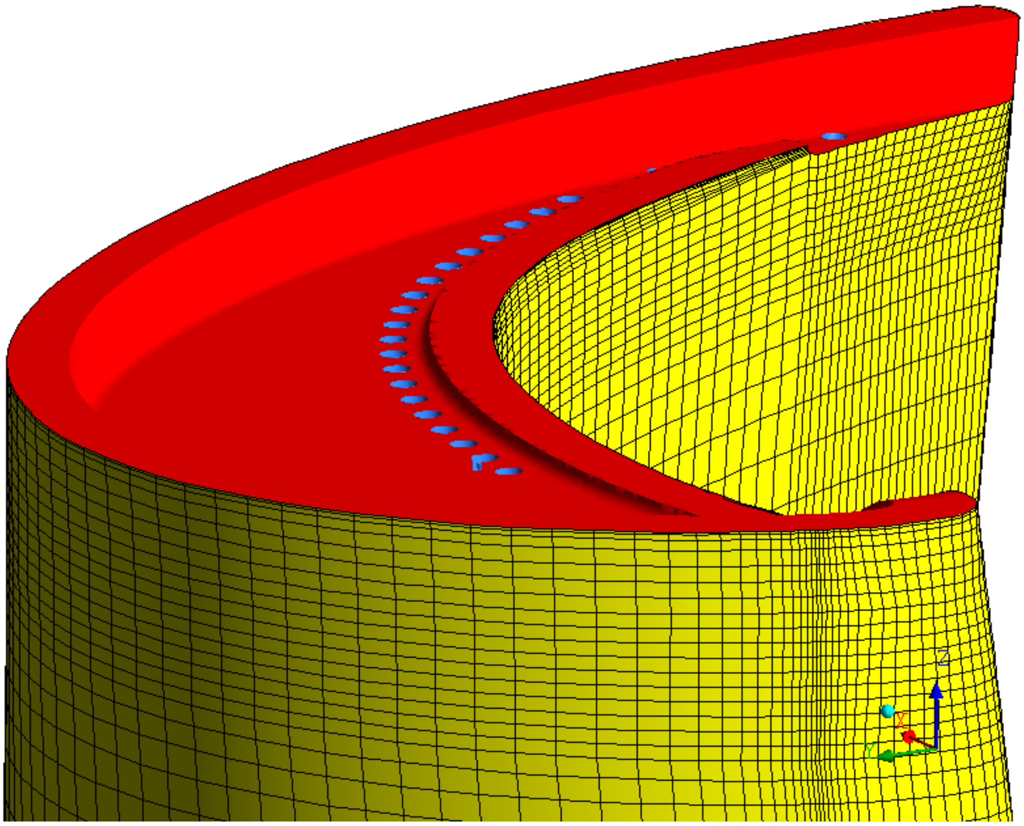

Figure 2: Cooling holes and hexahedral mesh on the blade

The following table resumes the mesh dimensions. The number of hexahedrons is fixed, while the number of prisms and tetrahedrons slightly change for different configurations of the tip clearance.

ELEMENT TYPE	Hexahedrons	Tetrahedrons	Prisms	TOTAL
Number of elements	5e+5	2.2e+6	3e+5	3e+6

Table 1: Mesh: average number of elements

The CFD model is periodic (only one of 64 blades is simulated), the flux is turbulent, compressible and in steady state conditions. The k-ε model was adopted to solve turbulence.
The fluid domain is rotating (17000 rpm). Total pressure and total temperature profiles were applied at the inlet section of the channel. Total pressure and total temperature values were assigned on the cooling holes. The blade walls were treated as fixed temperature walls. A static pressure value was applied on the outlet section of the domain.

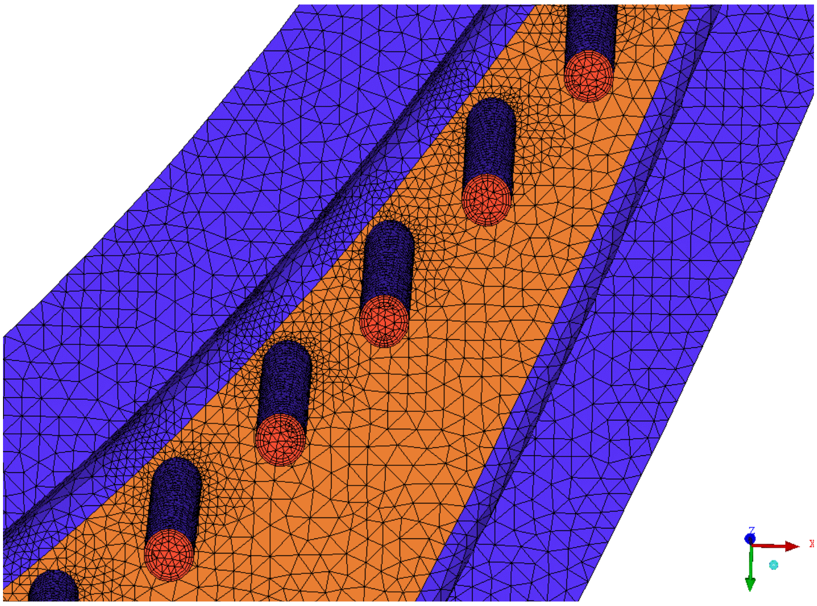

Figure 3: Tetrahedral mesh on the tip surface

1. Process Flow: input variables and objectives

The process flow (Figure 4) is the logic procedure which integrates one or more computational codes (ICEM and CFX5) with an optimisation software (modeFRONTIER), and thus allows the automatic analysis of several different configurations of the cooling system, in order to find the optimal designs with respect to the specified objectives and constraints.

The kernel of the process flow contains two batch scripts which automatically launch ICEM and CFX5. The geometrical model of the blade is created with the current input values, the computational mesh is automatically generated and passed to CFX5.
CFX5 defines the CFD model, works out the CFD solution and computes the response of the system in terms of *Cooling Mass Flow*, *Wall Heat Flux* and *Blade Efficiency*.

The optimisation software modeFRONTIER manages all these operations, acquires the results of the simulations, evaluates the fitness of each design to the specified objectives and, using an optimisation algorithm, assigns the input values for new analyses.

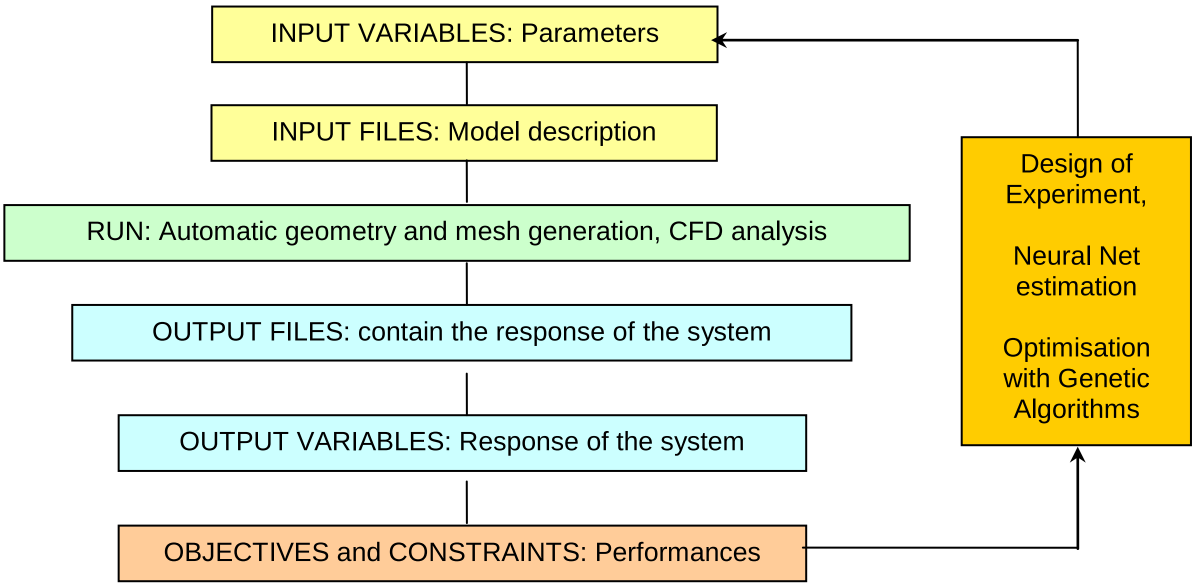

Figure 4: Process Flow

In this study the geometry of the blade contains six parameters, which define the tip configuration. The following table and Figure 5 describe the tip area of the blade and the parameters:

INPUT VARIABLE	LOWER BOUND	UPPER BOUND	STEP	DEFINITION
c	0.03	0.55	0.01	Non-dimensional distance from the pressure side
Nholes	8	20	1	Number of cooling holes
delta1	-0.07	-0.02	0.001	Non-dimensional distance between the first and second holes near the trailing edge.
alfa (degrees)	0	60	2	Holes inclination
Rholes (mm)	0.15	0.25	0.005	Radius
b (mm)	0.2	1.2	0.1	Squealer Height

Table 2: Input variables definition

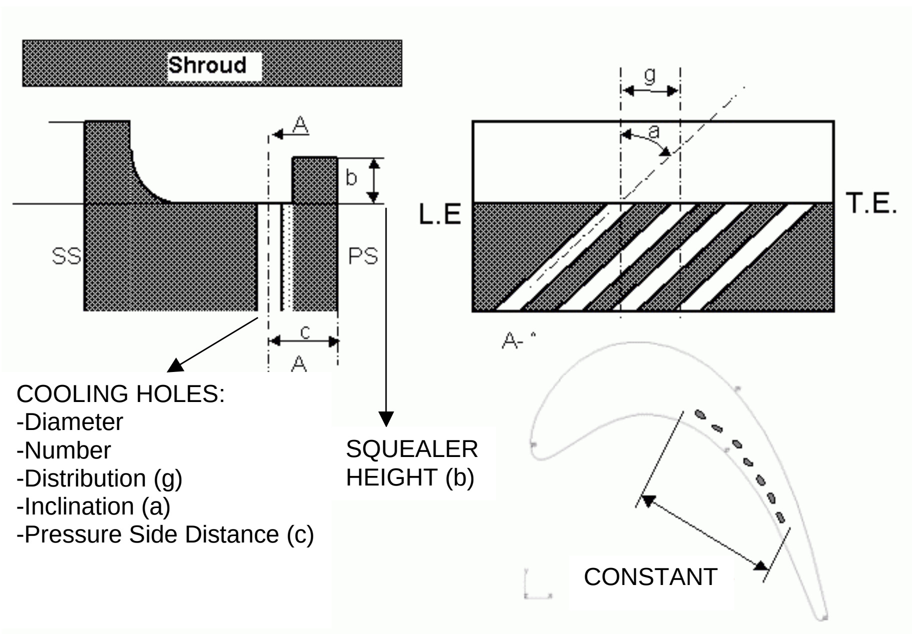

Figure 5: Input Variables (parameters)

The parameter *c* defines a curve on the tip surface at a certain distance from the pressure side of the blade. The cooling holes are distributed on this curve using a linear function defined by the input variables *Nholes* and *delta1* (some possible configurations are shown in Figure 7)
The cooling holes direction is defined by *alfa* (Figure 6). The radius is *Rholes*.
The parameter *b* describes the squealer height on the pressure side of the blade (Figure 6).

Some geometrical constraints have to be respected for the pitch/diameter ratio $2 \leq \dfrac{pitch}{diameter} \leq 6$.

During the geometrical phase ICEM checks this ratio for the cooling holes and stops the entire process if the constraints are not respected.

The process flow contains six input variables, four geometrical constraints and three independent objectives:

1. ***Cooling Mass Flow***: to be minimized.
2. ***Wall heat flux***: to be minimized. It is the heat flux transferred from the hot gas to the blade wall in the tip area (red region in Figure 1).
3. ***Aerodynamic blade efficiency***: to be maximized

In addition to these objectives, AVIO assigned also a constraint on the maximum value of *Cooling Mass Flow*. Solutions with higher mass flow values have to be considered unfeasible.

We have to point out that the solution of a multi-objective problem is not unique, but is defined by a series of Pareto dominating designs: the Pareto criterion states that design_X dominates design_Y, if design_X improves at least one objective, without worsening any other objective with respect to design_Y.

The following figures show some possible configurations of the tip area:

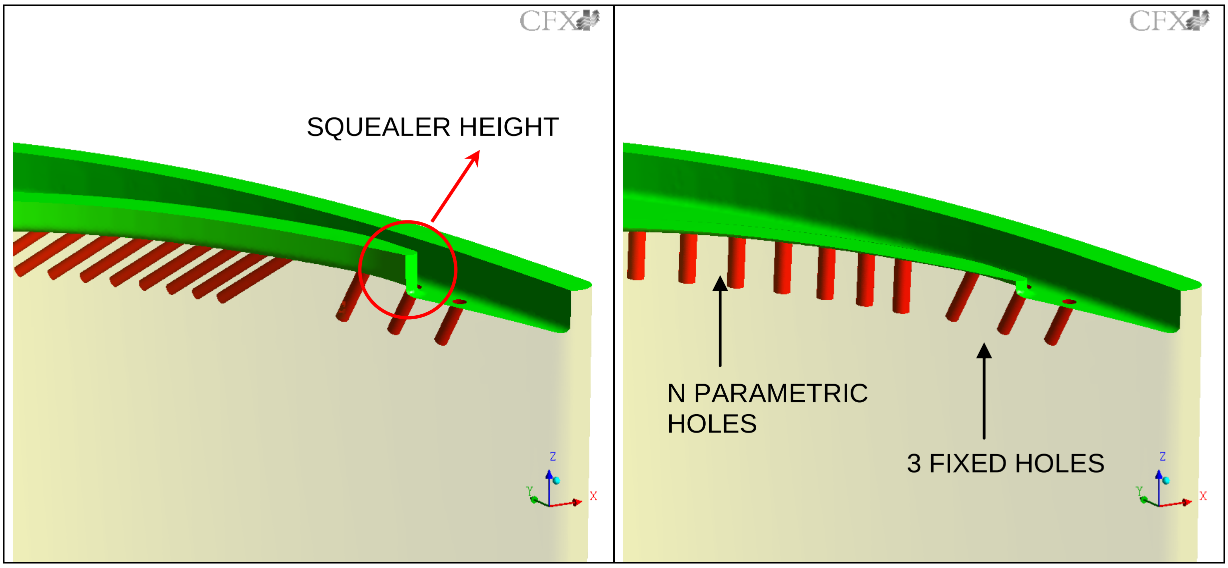

Figure 6: Different holes inclination and squealer height

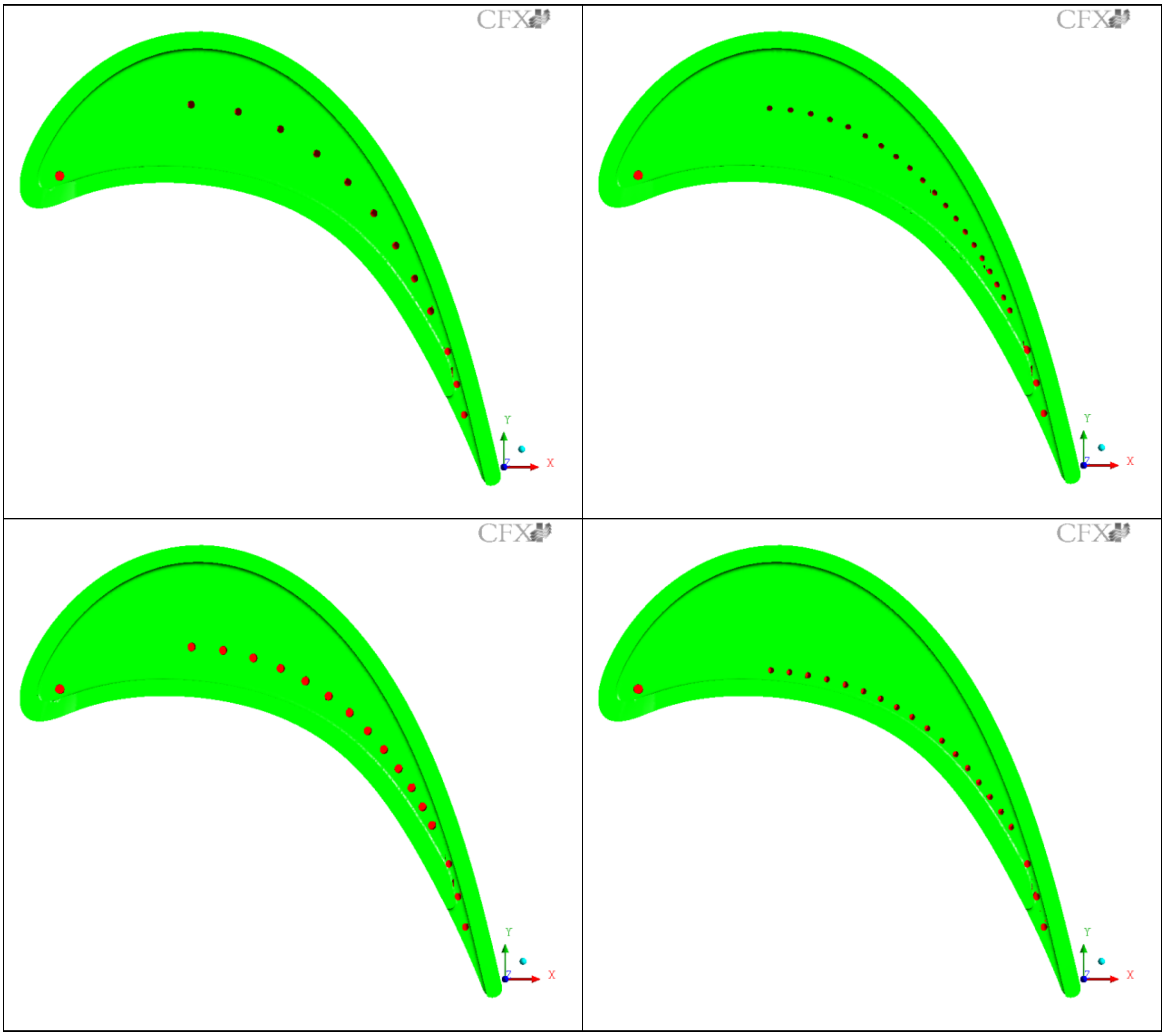

Figure 7: Possible distributions of the cooling holes on the tip surface

3. Optimisation procedure

Figure 8 illustrates the procedure followed to optimise the tip cooling system.

After the integration of the CFD model in the process flow, an initial series of 24 CFD analyses was planned using the SOBOL method. This algorithm distributes a set of points in a pseudo-random way and assures a uniform coverage of the space of possible solutions.

This input-output database was used to estimate a first set of Neural Nets. The accuracy of Neural Nets is given by the relative error, which is the difference between the CFD results and the Neural Net forecasts expressed in relative terms.
The maximum relative error was computed for the objective *Cooling Air Mass Flow* and was lower than 5%: $RelativeError = \frac{|Mcooling(NN) - Mcooling(CFD)|}{Mcooling(CFD)} * 100 < 5\%$

The Neural Nets were then coupled to a Multi Objective Genetic Algorithm (MOGA) and the first optimisation step was performed with 3 generations of virtual designs. The evaluation of this series of 24*3 solutions had a computational time of few seconds, while the same 72 CFD analyses would have required more than three weeks in terms of CPU time.

Six designs were chosen from the virtual Pareto Frontier (which collects the best solutions found by the MOGA and by the NN) and six CFD analyses were carried out.
The comparison between the Neural Nets forecasts and the CFD results is reported in Figure 9 in the (*Wall Heat Flux – Cooling Mass Flow*) plane. This figure shows that the points computed by the CFD analyses were very close to the NN forecasts and confirms the accuracy of the Neural Nets.

A second set of Neural Nets was estimated using 30 CFD points and the same procedure was followed: optimisation with MOGA and verification of 9 solutions taken from the new Pareto frontier.

Before the last step the problem was reduced to a single objective optimisation using the Multi Criteria Decision Making method (MCDM). The following weights were assigned to the initial objectives:

- *Wall Heat Flux* = 0.5
- *Cooling Mass Flow* = 0.4
- *Blade efficiency* = 0.1

These weights correspond to the relevance of the objectives and allow the construction of a unique function (TOTAL UTILITY) which is the weighted sum of the initial objectives. The TOTAL UTILITY is an indicator of the fitness of the solution to all the objectives at the same time and has to be maximised.
The B*lade Efficiency* is less relevant than the W*all Heat Flux* and than the *Cooling Mass Flow* because very low efficiency changes were computed for far different tip configurations.

The last optimisation phase was carried out using the SIMPLEX algorithm coupled to the last set of Neural Nets (estimated with 39 CFD points).
In the final series of 9 CFD simulations also a solution proposed by AVIO, and obtained with standard design approach, was included in order to compare it with the modeFRONTIER optimal design.

Globally 48 CFD analyses were carried out and more than 250 virtual points were evaluated using the Neural Nets.

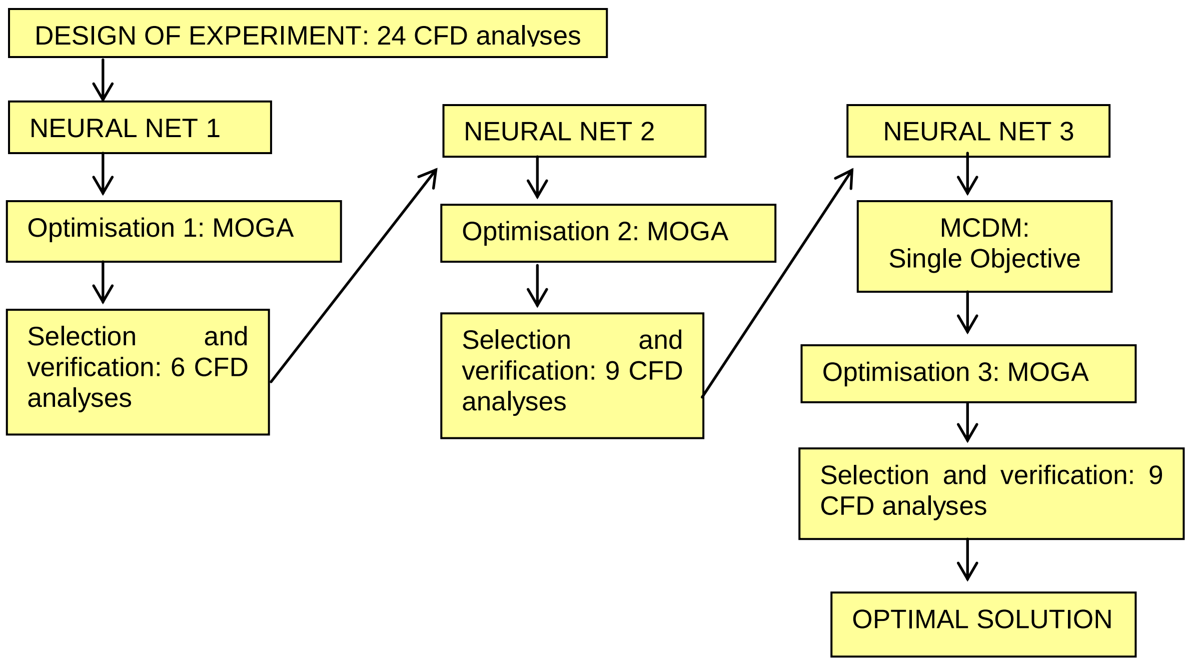

Figure 8: Optimisation process

In the following figure a comparison between NN forecasts and CFD results (at the end of the first optimisation phase) is reported in the (*Wall Heat Flux – Cooling Mass Flow*) plane. Both *Wall Heat Flux* and *Cooling Mass Flow* are expressed as non-dimensional quantities referring to the performances of the AVIO standard solution.

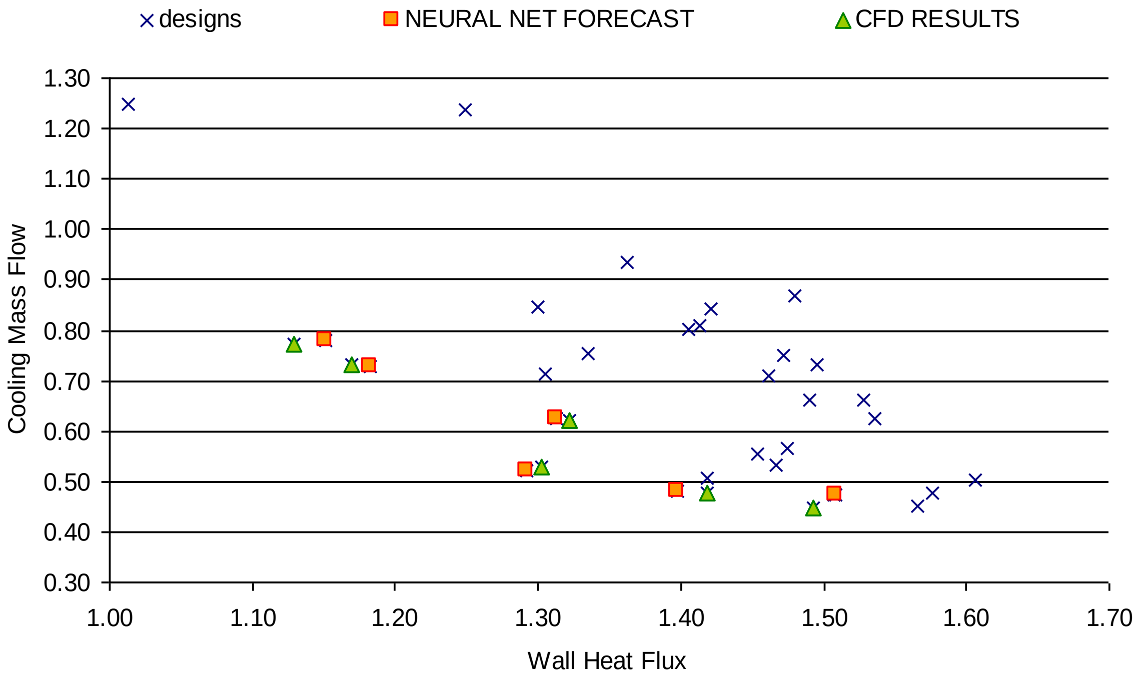

Figure 9: Comparison between NEURAL NET forecasts and CFD results

7. Results

At the end of the third optimisation step, the optimal design was chosen according to the objective *max(TOTAL UTILITY)*.

The best solution is represented by the green point in the (*Wall Heat Flux – Cooling Mass Flow*) plane (Figure 10) and it corresponds to the 42th design, evaluated during the last optimisation step.
Figure 10 compares the performances of the following designs in the *(Wall Heat Flux – Cooling Mass Flow)* plane:

- Blue crosses: all the designs evaluated with a CFD simulation
- Light blue triangles: Pareto Frontier at the end of the DOE phase
- Red squares: Pareto frontier at the end of the optimisation
- Black circles: AVIO standard design
- Green rhombus: modeFRONTIER best solution (DESIGN_42)

Figure 10 demonstrates that, starting from the DOE frontier, the Pareto points moved towards lower *Cooling Mass Flow* values and lower *Wall Heat Flux* values at the same time.

The final Pareto frontier is composed by designs evaluated during the second and third optimisation phases. This table highlights the improvements obtained by DESIGN_42 with respect to the AVIO standard design:

OBJECTIVE	AVIO SOLUTION	ModeFRONTIER SOLUTION	DIFFERENCE
MIN(*Cooling Mass Flow*)	1.0	0.785	-21.5 %
MIN(*Wall Heat Flux*)	1.0	0.985	-1.5 %
MAX(*Efficiency*)	94.6%	94.4%	-0.2 %

Table 3: Comparison between AVIO solution and modeFRONTIER solution

The main conclusion which can be inferred from Figure 10 and from Table 3 is that the same cooling effectiveness (-1.5%) can be kept with a reduced *Cooling Air Mass Flow* (-21.5%).

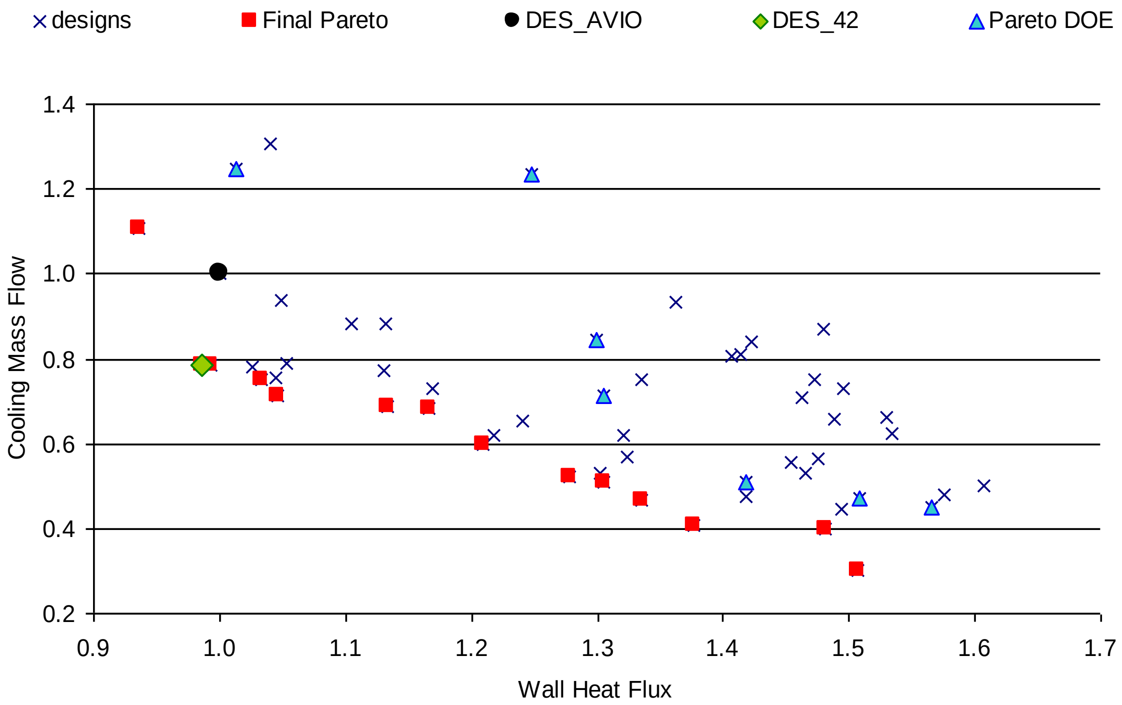

Figure 10: Final results in the (Wall Heat Flux – Cooling Mass Flow) plane

This result was obtained with the following tip configuration:

- 14 cooling holes are distributed with a constant pitch along a curve which is adjacent to the pressure side of the blade (*c* = 0.03)
- the holes are vertical (*alfa* = 0.0 degrees) and the radius is maximum (*Rholes* = 0.25 mm)
- the squealer height is minimum (*b* = 0.2 mm) on the pressure side of the blade

The cooling holes for DES_42 and for the AVIO standard design turned out to have the same total area, but DES_42 has a lower squealer (*b* = 0.2 mm with respect to 1.2 mm).
As a consequence of the lower squealer, and due to different pressure distribution and velocity components on the tip surface, also the penetration of the cold jets is lower (Figure 11).
This aspect affects the mass flow rate (lower in DES_42) and reduces the hot vortex under the cold jets.

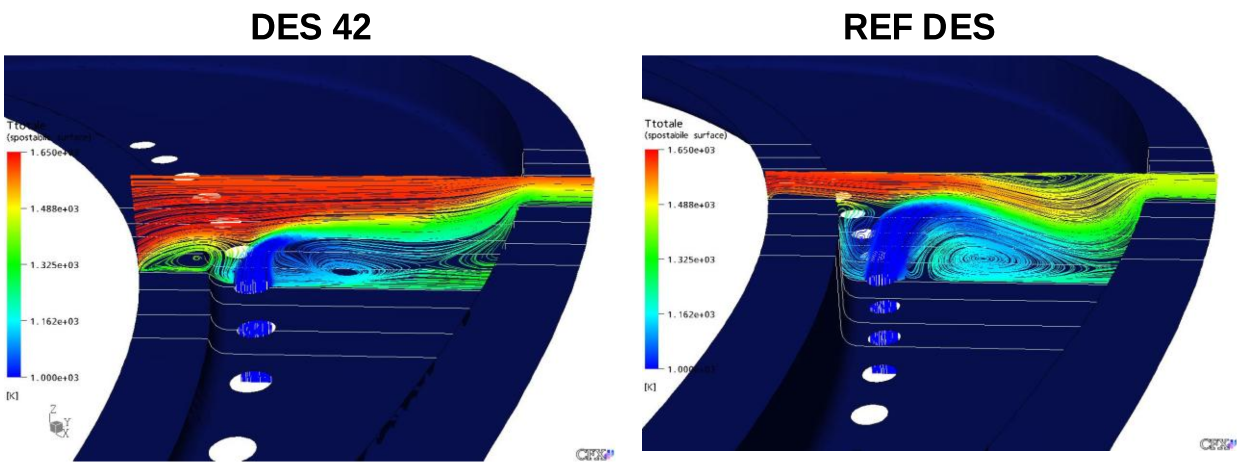

Figure 11: DESIGN_42 versus DESIGN_AVIO: streamlines and temperature in the tip region

Moreover the lower squealer on the pressure side of the blade seems to be able to avoid hot gas impingement on the suction side squealer.

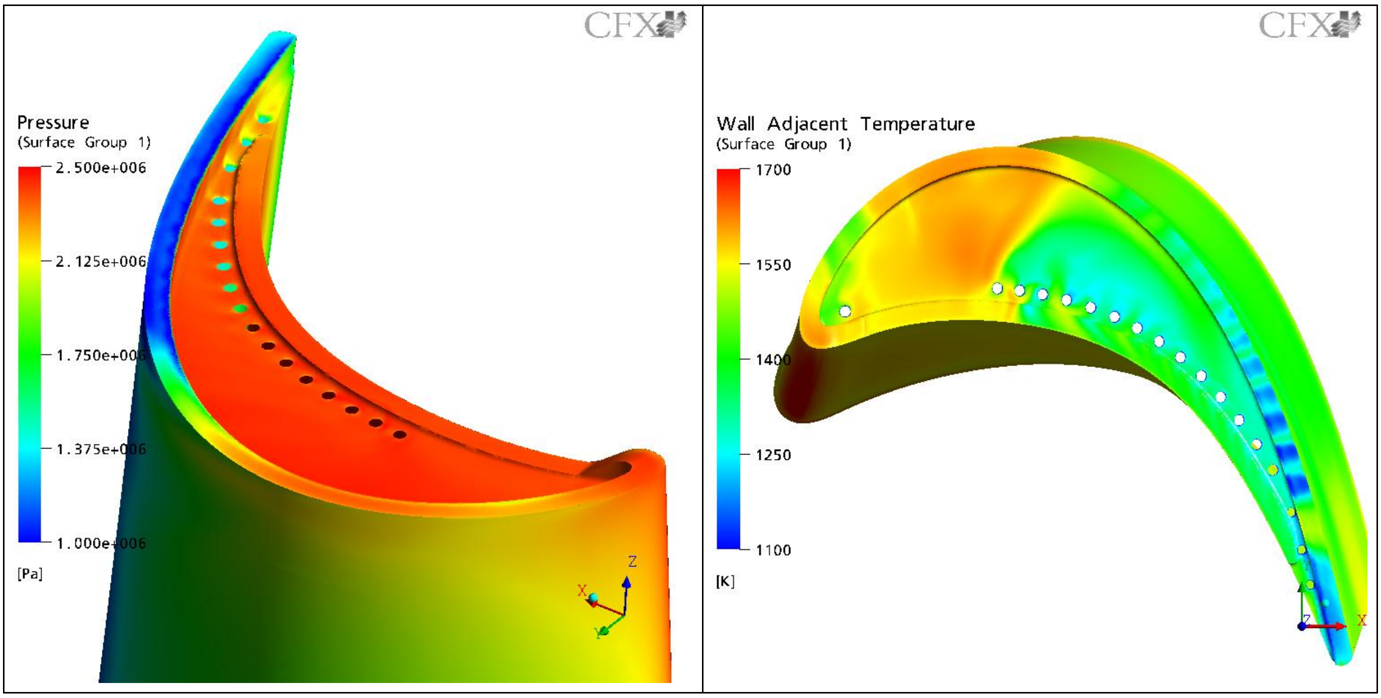

Figure 12: Optimal solution (DESIGN_42): pressure and temperature distributions on the tip wall

Conclusions

This study demonstrates that the multi-objective optimisation techniques can be applied with success to complex and high computational costing 3-D models.
The flexibility of the process flow in modeFRONTIER allows the integration of different codes and the management of a complex logic containing several batch scripts.
The use of Neural Nets was crucial in order to reduce the number of CFD simulations and the computational time. Neural Nets turned out to be accurate even if estimated with a low number of points, gave reliable forecasts of the response of the system and directed the process towards the optimal solutions.
The comparison of the optimal and the standard configurations gives evidence that the cooling mass-flow can be considerably reduced (-20%), keeping the same wall heat flux on the tip surface.
Moreover the parametric CFD study highlighted the influence of the geometrical configuration of the tip region on the local fluid-dynamic behaviour of the system and lead to a new cooling configuration of the blade tip.

References

[1] Florschuetz, Truman, Metzger, “Streamwise flow and heat transfer distributions for jet array impingement with crossflow“, 1981, Journal of Heat Transfer, Vol. 103.

[2] Florschuetz, Truman, Metzger, “Effects of initial crossflow temperature on turbine cooling with jet arrays“, 1984, Heat and mass transfer in rotating machinery.

[3] ModeFrontier, User Manual