A multi-objective optimisation of a transonic compressor cascade

Francesco De Vanna*, Ernesto Benini

Department of Industrial Engineering, Università Degli Studi di Padova

*francesco.devanna@phd.unipd.it

COMETES

DIPARTIMENTO DI INGEGNERIA INDUSTRIALE

Università degli Studi di Padova

1. What is a transonic stage of a compressor?

A compressor stage is said to be **transonic** if the incoming and the outgoing speed that approaches the stage are respectively higher and lower than the **local speed of sound**. This technological solution represents the state-of-art in terms compressor blade for a gas-turbine engines.

- The **efficiency** of a transonic stage is comparable to the one of a standard subsonic compressor stage;
- The **compression ratio** of a single stage is much higher ($50 \div 60\%$ more) than a subsonic technology giving you the possibility of **reducing the weights** maintaining the same propulsive performances.
- The **flow field** around a transonic blade is very complex and affected by many compressibility effects making very difficult to establish a standard dimensioning procedure;
- Lackness of **experimental data** (a supersonic wind tunnel is needed).

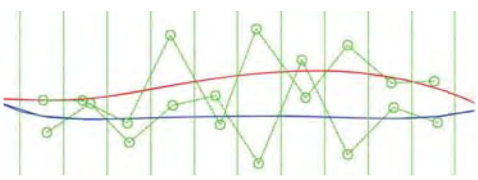
2. Aim of the study

- **Analyze the aerodynamic's field** around the most modern S-shape compressor cascade in terms of:
  - shock-wave's structure;
  - boundary layers interactions;
  - aerodynamical load distribution.
- Develop a validated **numerical model**, using both CFD commercial packages and in-house tools in order to optimise transonic airfoils with a fully automatic procedure.

3. Shape parameterization

In order to guarantee a smooth transition between optimised geometries, the entire airfoil was parametrised using a Bezier curve:

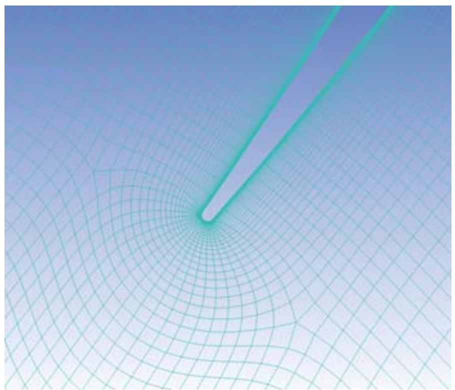
$$\begin{Bmatrix} x(t) \\ y(t) \end{Bmatrix} = \sum_{i=0}^{n} \binom{i}{n} (1-t)^{n-i} t^i \begin{Bmatrix} x(i) \\ y(i) \end{Bmatrix}$$



Those parametric curves allow you to control just a little sample of points, named *control points*, reducing the computational cost of the optimisation algorithm.

The parametrisation involved both the pressure and the suction side, maintaining constant both attack and exit angles. For each curve **10 control points** have been traced.

4. Aerodynamic model validation

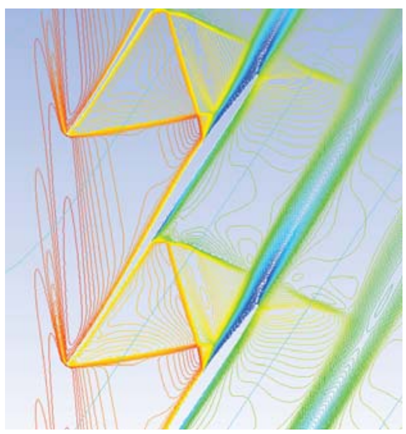


Steady RANS CFD computation with commercial packages (ICEM CFD + Ansys Fluent) on a fully structured grid. The numerical model was validated in terms of both global

- inlet-outlet Mach number (M_1, M_2);
- absolute outlet flow angle (α_2);
- loss factor (ω);
- pressure ratio (p_2/p_1)

and local parameters (isentropic Mach number, M_{is}, along the airfoil), comparing numerical results with experimental data. For all the aerodynamic parameters a **grid sensitivity analysis** was performed and **turbulence model influence** was checked.

5. Multi-objective otimisation for ω and $-p_2/p_1$ minimisation



Two-dimensional Mach field around the cascade.

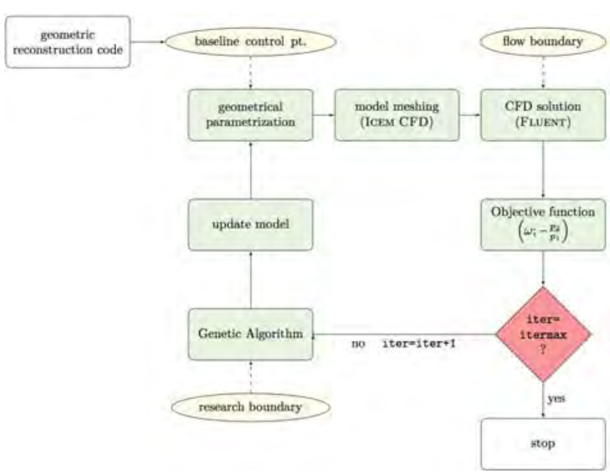
	ω	$\Delta\omega\%$	p_2/p_1	$\Delta p_2/p_1\%$
opt ω	0.1225	-11.87	2.2218	2.5
opt p_2/p_1	0.13817	-0.6	2.2238	2.6

Two new geometries have been found:

- one is optimal in terms of the losses coefficient;
- one is optimal in terms of pressure ratio.

In particular, the once that optimises the loss coefficient shows a much higher efficiency compared to the baseline. This new geometry is able to produce a **complete different set of shock waves in the pre-compression zone**, reducing the speed of the flow that is approaching the inner normal shock-waves. Having a weaker shock wave in the blade-to-blade channel reduces both the shock drag and the entropy production.

6. Optimisation loop



The entire optimisation loop was develop in a MATLAB environment and consists in the following parts:

- An external code **initialises the initial population**; this one consists in the baseline geometry and some random individuals. It also sets all the parameters need by the the main optimisation loop;
- A dedicated module **parametrises the geometry** with the Bezier curve strategy;
- Knowing the parametrised geometry, an automatic procedure write all the mesher instructions in order to **build the computational grid**;
- The **CFD solution** is obtained by Ansys Fluent and the objected functions $\omega, -p_2/p_1$ are computed;
- If the number of the iterations reaches the maximum value, the loop stops, otherwise, the ***GAmultibj algorithm*** is called in order to produce a new geometry to test.