Numerical Study of the Transonic Flow in the NASA Rotor 37

15/11/2026

CHIKHI Abdelilah
abdelilahchikhi08@gmail.com

FlowThermoLab

Objective

The objective of this work is to numerically investigate the transonic compressible flow in the NASA Rotor 37 using ANSYS CFX, in order to:

- Reconstruct the geometry of the NASA Rotor 37 in BladeGen
- Generate a mesh suited to a single blade-to-blade passage
- Simulate the transonic compressible flow through the rotor
- Analyze the main aerodynamic phenomena (shocks, tip leakage, wakes)
- Compare the CFD performance predictions to literature data
- Identify the limitations of the numerical model used

Problem Statement

The NASA Rotor 37 is a transonic axial-compressor rotor (36 blades) in which the relative inlet flow is supersonic. As the flow crosses the blade passage, it undergoes a complex compression process that standard RANS turbulence models struggle to capture accurately:

- A detached shock wave forms near the blade leading edge
- The shock interacts strongly with the blade boundary layer
- Tip-clearance leakage generates a leakage vortex and blockage
- Wakes form behind the blades, affecting downstream mixing
- Secondary flows develop near the hub and shroud
- The interaction of these effects governs the achievable pressure rise and efficiency

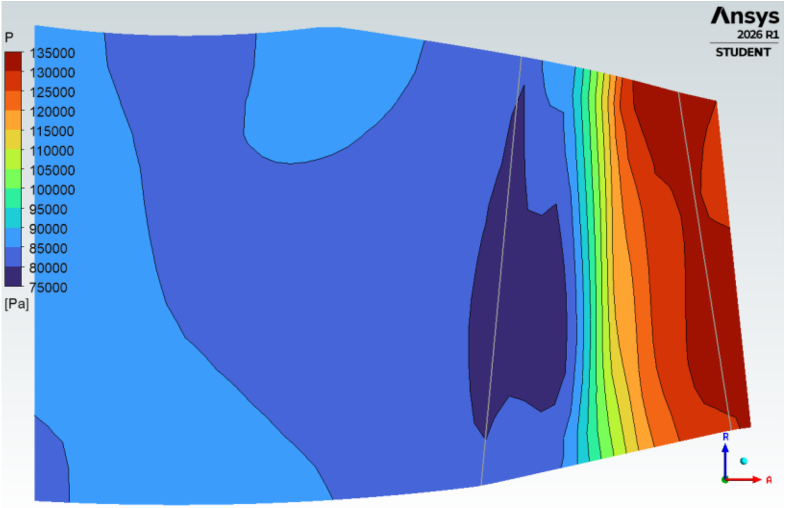
CFD Setup

- **Phase:** Steady-state, single-phase (compressible air)
- **Turbulence model:** k–ω SST
- **Energy equation:** Total Energy (on)
- **Solver:** ANSYS CFX Solver (coupled, pressure-based)
- **Temperature unit:** °C
- **Mesh type:** Structured mesh (ANSYS TurboGrid), single blade-to-blade passage
- **Cell size:** ~104,282 nodes / 95,304 elements (no dedicated mesh-independence study)
- **Wall condition:** No-slip, smooth, adiabatic walls
- **Initialization method:** Automatic initialization from boundary conditions (CFX default)

Results

These results show that the rotor transfers energy to the fluid and produces an increase in total pressure and total temperature across the passage.

19.6378 kg/s Mass flow rate	**1.8886** Total pressure ratio	**1.2297** Total temperature ratio
88.58% Isentropic efficiency (T-T)	**89.51%** Polytropic efficiency (T-T)	**1.306 MW** Power transmitted

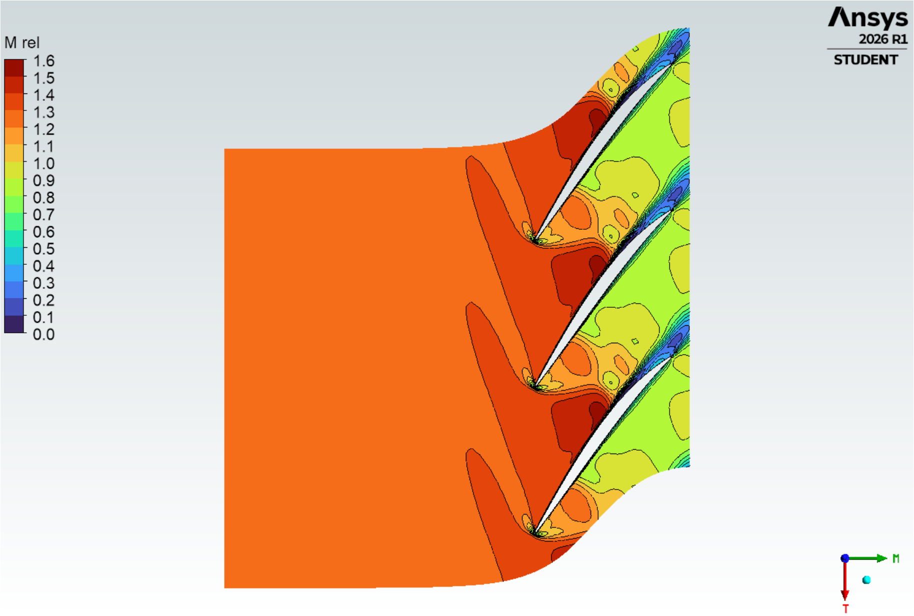


Results

The transition from a supersonic relative flow to a subsonic flow indicates strong transonic compression within the rotor.

FLOW EVOLUTION (relative Mach number)

INLET	LEADING EDGE	OUTLET
1.2802	1.3486	0.8361
Supersonic	*Supersonic*	*Subsonic*



Comparison with the Literature

Quantity	CFD	NASA Reference	Deviation
Mass flow rate (kg/s)	19.6378	20.19	**−2.74%**
Total pressure ratio	1.8886	2.106	**−10.32%**
Isentropic efficiency	88.58%	≈ 88.9%	**−0.36%**

- The mass flow rate is close to the design value
- The overall efficiency is correctly reproduced
- The total pressure ratio is underestimated
- The simulation reproduces the general operating trend

Conclusion

- A steady-state CFD simulation of the NASA Rotor 37 was carried out in ANSYS CFX
- The relative flow is transonic, decelerating from supersonic to subsonic across the passage
- The mass flow rate and efficiency are close to the reference (NASA) values
- The total pressure ratio remains underestimated (−10.3%)
- Overall, the main aerodynamic trends are correctly reproduced

Discussion — Limitations

- No mesh-independence study was performed
- Approximate geometry reconstruction and coarse tip-clearance resolution
- Steady-state hypothesis neglects unsteady shock / wake interactions
- k–ω SST has known limits for shock position and shock–boundary-layer interaction
- Comparison should ideally use corrected mass flow and corrected speed