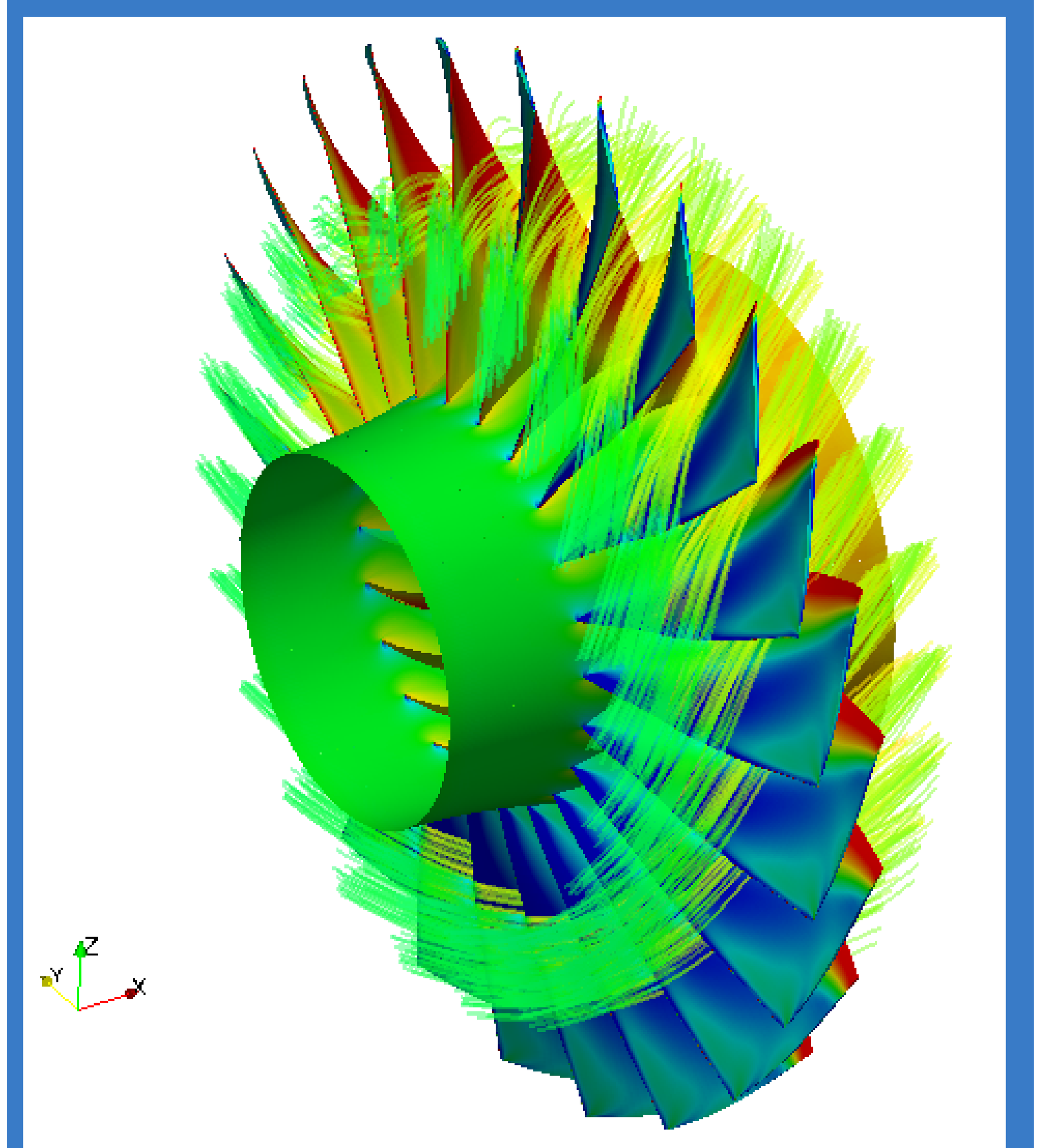
Validation of compressible flow solver with rothalpy equation on the NASA Rotor 67 transonic turbofan test case

University of Zagreb
Faculty of Mechanical Engineering and Naval Architecture
Department of Energy, Power Engineering and Environment

Josip ŽUŽUL
Gregor CVIJETIĆ
Hrvoje JASAK



ABSTRACT

This poster presents the capabilities of the OpenFOAM's new compressible solver for turbomachinery CFD simulations, developed within Prof. Jasak's CFD group.

New pressure-based formulation of the compressible flow solver for turbomachinery purposes is specific regarding energy conservation equation, which is described in terms of the conservation of rothalpy. Solver is implemented in foam-extend, a community-driven fork of the CFD package OpenFOAM, and is validated against the experimental data on the NASA Rotor 67 transonic turbofan test-case.Since rothalpy is conserved over a certain blade row (but not over a stage), a special care needs to be taken in cases of rotor-stator interaction in order to accurately resolve a fluid flow. Therefore, if the rothalpy equation is solved, rothalpy is continuous over domain, but the quantities that are calculated from rothalpy (enthalpy and temperature) have a jump over the rotor-stator interface, due to the change in angular velocity. Thus, a rothalpy jump on the interface is used in order to manipulate rothalpy in a way that ensures continuous evaluation of enthalpy and temperature field.
Rotor-only configuration of the NASA 67 transonic turbofan doesn't have rothalpy jump interfaces, therefore a temperature field is directly evaluated from rothalpy. NASA Rotor 67 is a common test case for compressible flow in turbomachinery due to complex three-dimensional flow features which occur in the flow, such as tip-leakage flow and trailing edge vortices. Successful evaluation of these complex fluid flow phenomena implies, in addition to agreement between numerical and experimental overall performance parameters, a proper implementation of the numerical solver for turbulent, compressible and transonic flow in turbomachinery.

NASA Rotor 67 case setup

NASA Rotor 67 transonic turbofan is the axial-flow fan consisting of 22 rotor blades. Within this work, results were evaluated by taking the advantage of the periodic boundaries where only a single blade passage was examined, while the effect of rotation is obtained using the *Multiple Reference Frame* (MRF) approach. Additionally, a `cyclicGgi` interface is used for coupling of non-conformal periodic mesh boundaries on the matrix level. Prescribed rotational speed is the nominal rotational speed of 16 043 rpm, while the turbulence was modelled with the $k-\omega$ SST turbulence model.

Results

Numerical simulations were carried out for operating points with known experimental data (nominal and near stall operating point) and many other operating points in order to generate the performance curve and compare it with the provided experimental data (*Strazisar et al.,1989*).
Figure 1 shows the good agreement of relative Mach number contours between experiment (left) and simulation (right) for the nominal operating point near the blade-tip.

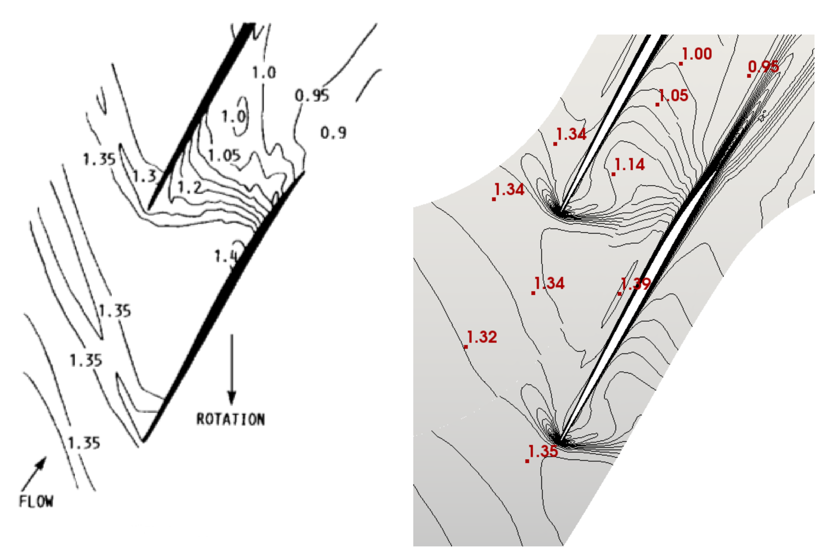


Figure 1: Comparison between relative Mach number contours.

In terms of the relative velocity, flow conditions for the nominal operating point are considered supersonic at the inlet, transonic in the blade passage and subsonic at the outlet boundary. Relative Mach number field (right) and the corresponding pressure field (left) is presented in the Figure 2.

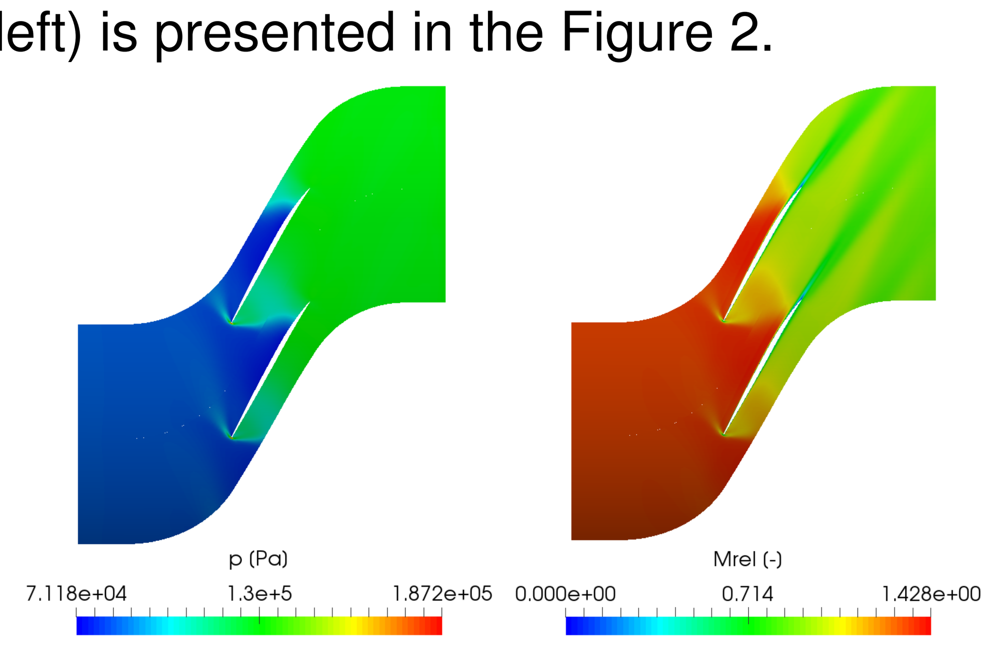


Figure 2: Pressure and relative Mach number field near blade-tip.

Table 1 and Table 2 present the comparison of obtained numerical and experimental data for nominal and near-stall operating point. It can be seen that the calculated values of mass flow rate, total pressure ratio and adiabatic efficiency are slightly underestimated with respect to the experimental values. For the complex near-stall operating point from the physical and numerical point of view, the maximal observed relative error of 3.47% for the total pressure ratio is more than satisfactory.

Table 1: Comparison of numerical and experimental data for the near peak efficiency operating point.

Parameter	CFD	Experiment	Relative Error, [%]
Mass flow rate, $\dot{m}$	33.65 $\mathrm{kg/s}$	34.57 $\mathrm{kg/s}$	2.66
Total pressure ratio, Π	1.615	1.642	1.64
Adiabatic efficiency, η_{ad}	90.03%	93%	3.19

Table 2: Comparison of numerical and experimental data for the near-stall operating point.

Parameter	CFD	Experiment	Relative Error, [%]
Mass flow rate, $\dot{m}$	32.23 $\mathrm{kg/s}$	32.31 $\mathrm{kg/s}$	0.25
Total pressure ratio, Π	1.668	1.728	3.47
Adiabatic efficiency, η_{ad}	88.49%	90.1%	1.79

By simulating multiple operating points, a performance curve is evaluated and compared with the performance curve provided in the literature. Comparison is presented in Figure 3, where quite good agreement can be observed along the whole operating range.

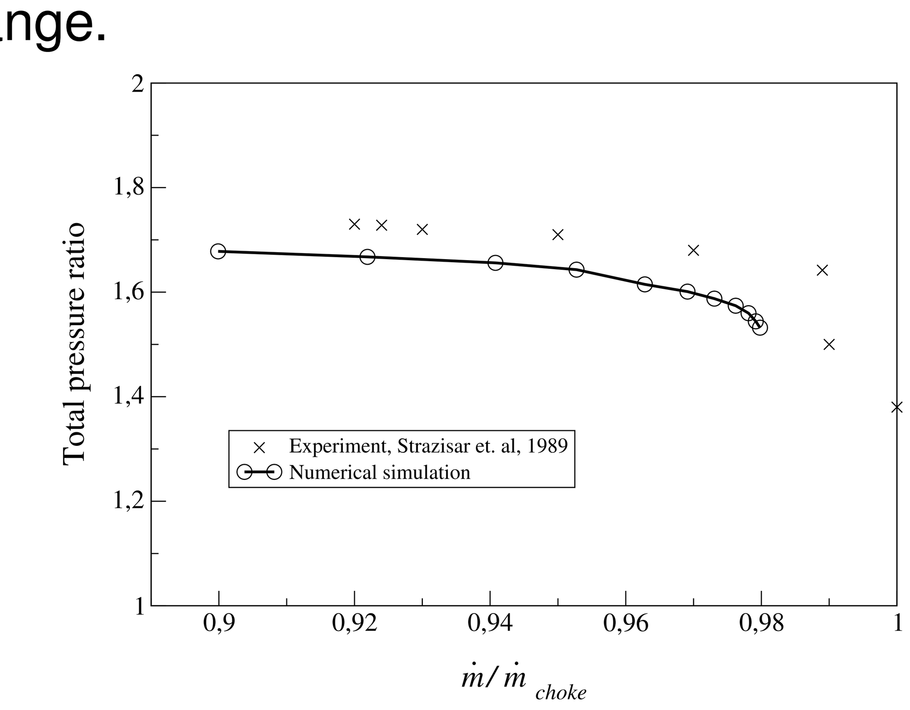


Figure 3: Comparison of the evaluated performance curve with the experiment.

Trailing edge vortex evaluated for the nominal operating point and visualised in the Figure 4 confirms the complex nature of turbulent flow downstream of the NASA Rotor 67 transonic turbofan.

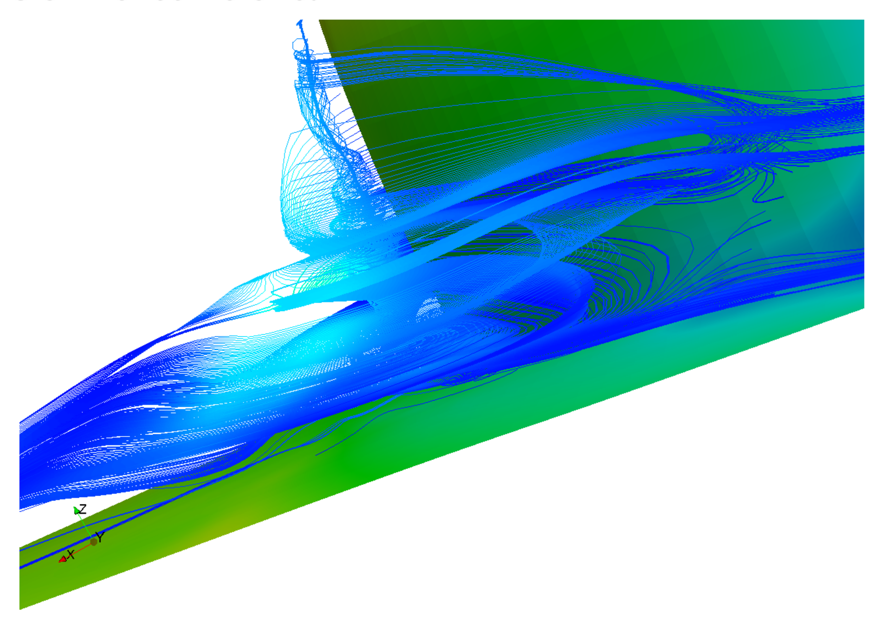


Figure 4: Trailing edge vortex.

Conclusion

Good agreement between numerical and experimental data shows that the new compressible solver for turbomachinery is capable of solving complex transonic flow with reasonable accuracy. The most complex three-dimensional flow features such as vortices and tip-leakage flow were also adequately captured.