

COUPLED PRESSURE BASED CFD SOLVER FOR TURBOMACHINERY FLOWS: OVERVIEW OF APPLICATIONS

L. Mangani, E. Casartelli, M. Darwish

Hochschule Luzern, Competence Centre for Fluid Mechanics and Hydro Machines, Horw,
Switzerland

American University of Beirut, Department of Mechanical Engineering, Beirut, Lebanon

luca.mangani@hslu.ch, ernesto.casartelli@hslu.ch, darwish@aub.edu.lb

ABSTRACT

CFD has become an indispensable tool for the design and analysis of modern turbomachinery equipment. Commercial codes usually play a major role for these processes, offering dedicated pre- and postprocessing features, but also advanced solver technology, in order to ease the approach to CFD for the development engineers. In the last decade both academic and industrial institutions have started to explore alternatives in order to achieve more flexibility in the development and implementation of new specific models and more resources in large-scale deployment of CFD procedures, especially in unsteady and transient processes as well as automatic optimization procedures. In this paper, the capabilities of a new, general-purpose coupled pressure-based CFD solver are presented. The code is developed based on the C++ object-oriented language, the discretization method employed is a finite volume cell centered approach. Pressure-based algorithms for the pressure-velocity coupling were adopted for compressible and incompressible flows with a series of new implicit developments specific for turbomachinery (mixing plane, boundary conditions, treatment of source terms), to improve convergence speed and robustness. Beside a large range of state-of-the-art turbomachinery specific features, new models like k-epsilon with enhanced wall functions and a novel all-Mach real-gas model are implemented in a coupled way.

The present publication shows an overview of the developed code and models, and a series of turbomachinery applications, ranging from subsonic to transonic internal flow, turbine cooling and conjugate heat transfer as well as transient computations in hydraulic machines.

KEYWORDS

CFD, COUPLED SOLVER, GENERAL PURPOSE, TURBOMACHINERY

INTRODUCTION

Turbomachinery flows are among the most challenging. Inherently unsteady, with (1) various types of instabilities, (2) laminar-turbulent transition, (3) incompressible to transonic behavior to name a few properties, their simulation is therefore very demanding.

Having a general purpose code able to efficiently cope with all this kind of flow situations is therefore very important. The pressure-based approach is appropriate for this purpose as it is able to capture all ranges of speeds from incompressible to supersonic flow.

From a numerical and implementation point of view, a segregated method is the most straightforward approach, solving each equation (momentum and continuity) for the independent variables in sequence while keeping the others constant, as already proposed in the 70's by [1]. On the other hand this approach can lead to difficulties in terms of robustness and simulation time when increasing the mesh size, because the algorithm unphysically decouples the flow variables.

A coupled approach, solving continuity and momentum simultaneously, can overcome these problems but leads to considerable difficulties from a numerical point of view. This is the reason why this approach is not often used for pressure based solvers.

In the present publication a completely new framework for the coupled solution of the Navier-Stokes equations is presented. It is based on the previous pioneering work presented in [2] and [3]. The new C++ framework is particularly designed for the solution of implicit algorithms for CFD problems and allows to efficiently implement new features and boundary conditions in the code, especially in an implicit way, thus allowing to achieve an improved robustness of the solution procedure.

SOLVER DESCRIPTION

In this section the basic ingredients and the advantages of the solver are briefly presented.

Numerics

The core of the proposed solver, `coupledNumerics`, is a coupled pressure-based algorithm to solve the Navier-Stokes equations. Starting from the basic idea of the SIMPLE algorithm, i.e. adopting a pressure-correction equation to enforce continuity, and using an algorithm which couples pressure and velocity, the well-known convergence deficits of the original SIMPLE approach could be overcome. Additionally a consequent implicit treatment for the pressure gradient and source terms in the momentum equation, the velocity in the continuity equation and the boundary conditions is adopted. In order to compute compressible flows, the energy equation needs to be included. A segregated implementation leads to the advantage of efficiently perform conjugate simulations (CHT) and at the same time is not affecting the convergence for flows with Mach number below 5 (hypersonic flows).

While density based coupled solvers have been developed in the last two decades, their main deficiency of computing the Jacobian to take into account the implicit coupling still remains. The linear convergence of the solver, i.e. resolving the same CFD case in the same number of iterations independently of the mesh size, deteriorates for density based versions due to this fact. Instead coupling velocity and pressure in a pressure-based approach does not require the computation of the Jacobian, is naturally linear, i.e. "only" leads to a large block-coupled matrix to solve at each iteration, for which linear convergence behavior can be achieved. The proposed solver has been designed to handle both structured and unstructured grids. Therefore an algebraic multigrid solver (AMG) for the solution of the linear systems has been adopted.

All these ingredients led to a fast and robust solver suitable for the wide range of applications present in the turbomachinery world. Moreover the speedup factor compared to state-of-the-art segregated pressure-based solvers can rise up to 40 for internal flows [2, 3]. Specific details on the solver numerics can be found in [4].

Spatial discretization is second order high resolution. Steady state computations are performed with a false transient approach, while unsteady simulations are second order accurate using a backward Euler approach.

Turbulence modeling

A series of turbulence models have been implemented in the code, in order to cope with the wide range of applications that still require different approaches.

The available turbulence models are:

- SST [5, 6]
- SST with 4 eq. transition model [7, 8]
- k- ϵ scalable
- k- ϵ realizable ewf [9]
- k- ϵ ewf
- EARSM [10, 11]

SST with automatic and k- ϵ with scalable wall functions are standard models used for the majority of the cases on a regular basis. On the other hand, there are some cases for which more sophisticated models are necessary. In this sense variations of the k- ϵ model are available with an enhanced wall function (ewf) to allow a mesh independent approach like in the SST model with the automatic wall functions. For the k- ϵ model a two-layer approach has been adopted for situations in which the mesh has a so-called low-Reynolds resolution with y^+ below 11.

For very challenging operating conditions, like instabilities in pump turbines while running in turbine-brake operation, the resulting vortical structures are dominated by turbulence anisotropy. Therefore usual models based on the Boussinesq assumption fail for such predictions. In order to accurately compute these kind of flows, an explicit algebraic Reynold Stress (EARSM) model has been implemented and successfully tested (see section “Application – Hydraulic turbine”).

Mesh-size independent convergence

A main advantage of the coupled approach is to have a mesh-size independent convergence in terms of number of iterations. This means that the solver does not suffer from detrimental effects due to mesh size increase like the original SIMPLE algorithm, thanks to the coupled linking of pressure and velocity. This fact has been shown in [3].

APPLICATIONS

In this section selected turbomachinery applications are presented, in order to show the code capabilities and underline the advantages of a coupled, implicit approach for the code.

All cases are computed with a high-resolution scheme and, if not otherwise stated, with the SST turbulence model [5, 6].

Transonic flow: NASA Rotor 37

The first application presented is a transonic flow compressor, extensively used for code validation [12], see Fig. 1. This shows the all-Mach capabilities of the code for this challenging flow problem, which, among other difficulties, also includes shock-induced boundary-layer separation. The adopted mesh consists of 400'000 cells with a low Reynolds resolution ($y^+ < 1$). The pressure ratio as well as the efficiency of the NASA Rotor 37 are presented in Fig. 2, with comparison to experimental and a series of published data. The results show good agreement with the measurements over the complete operating range for both pressure ratio and efficiency.

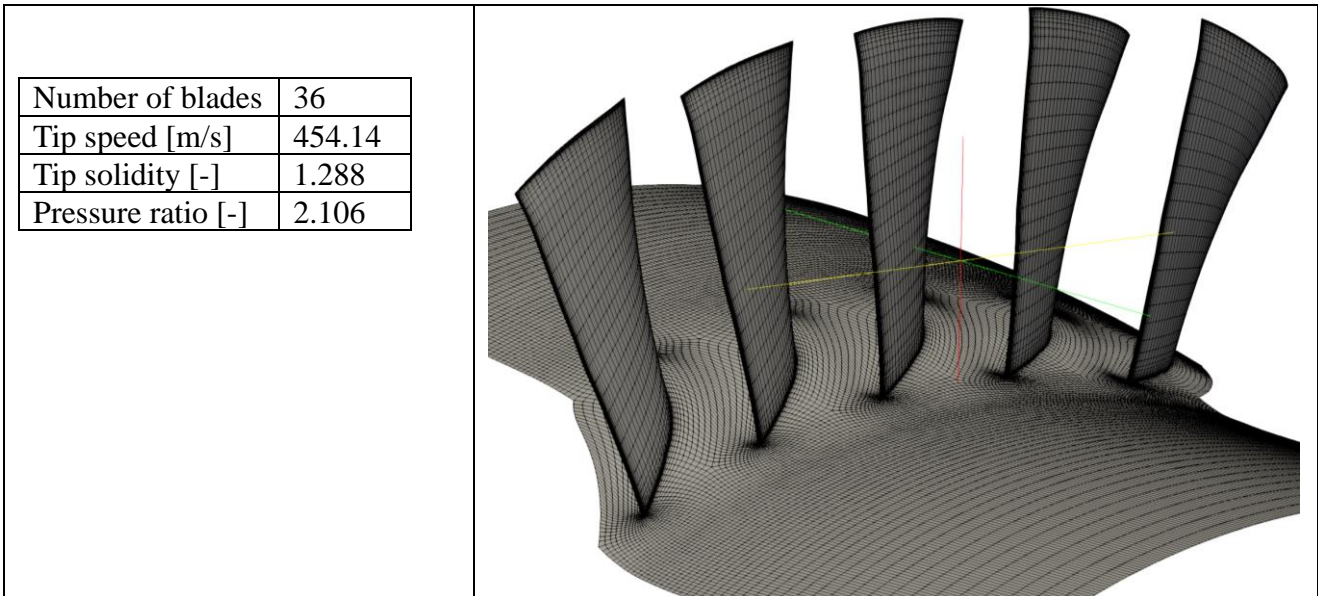


Figure 1: NASA Rotor 37 characteristic quantities (left) and geometry with mesh (right)

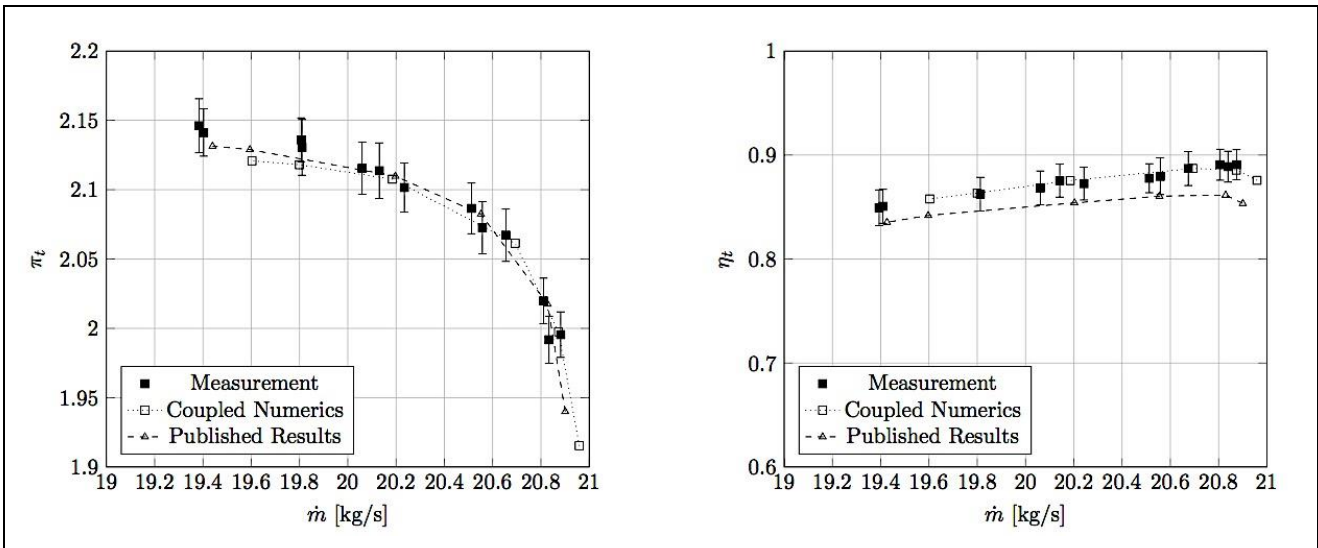


Figure 2: Performance characteristics of NASA Rotor 37. Left: pressure ratio. Right: efficiency.

Laminar-turbulent transition: ERCOFTAC cases

Turbomachinery flows can be considerably affected by laminar-turbulent transition, for example in low pressure turbines. Therefore the transition model proposed by Menter and Langtry [7, 8] has been implemented, being suitable for general 3D flows. The ERCOFTAC T3 Series Transition Test Case, configurations T3A and T3B, have been taken as validation data. These correspond to a flat-plate with sharp leading edge at zero pressure-gradient and different inlet turbulence levels: for case T3A the turbulence intensity is 3.3 % while for case T3B it is 6.5%.

Fig. 3 shows the computed results with experimental data. In general the implemented model is able to predict the laminar-turbulent transition for this case, with good agreement of the skin-friction coefficient along the flat plate. For the case T3B there is an overprediction of the skin friction in the transition region. In order to assess this apparent weakness of the code, Fig. 3 also includes data from a commercial-solver solution. It is clearly visible that also the commercial solver suffers of the same problem at this location. This discrepancy can be therefore attributed to the model itself rather than to the code.

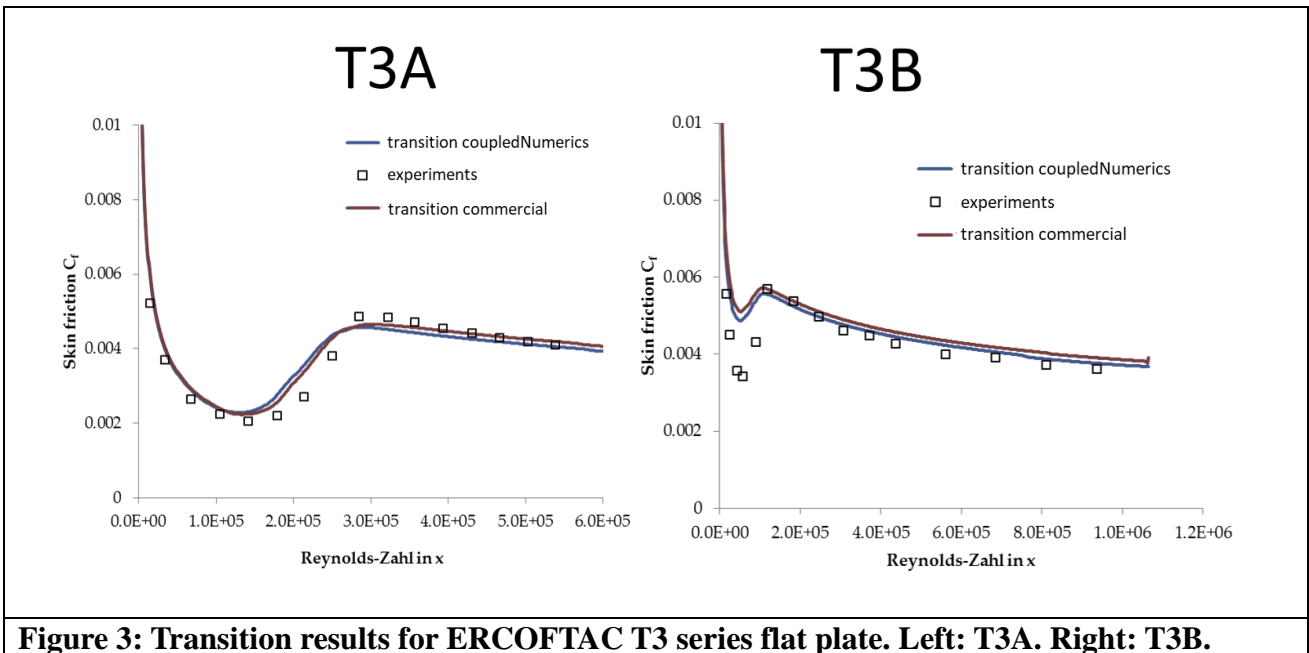


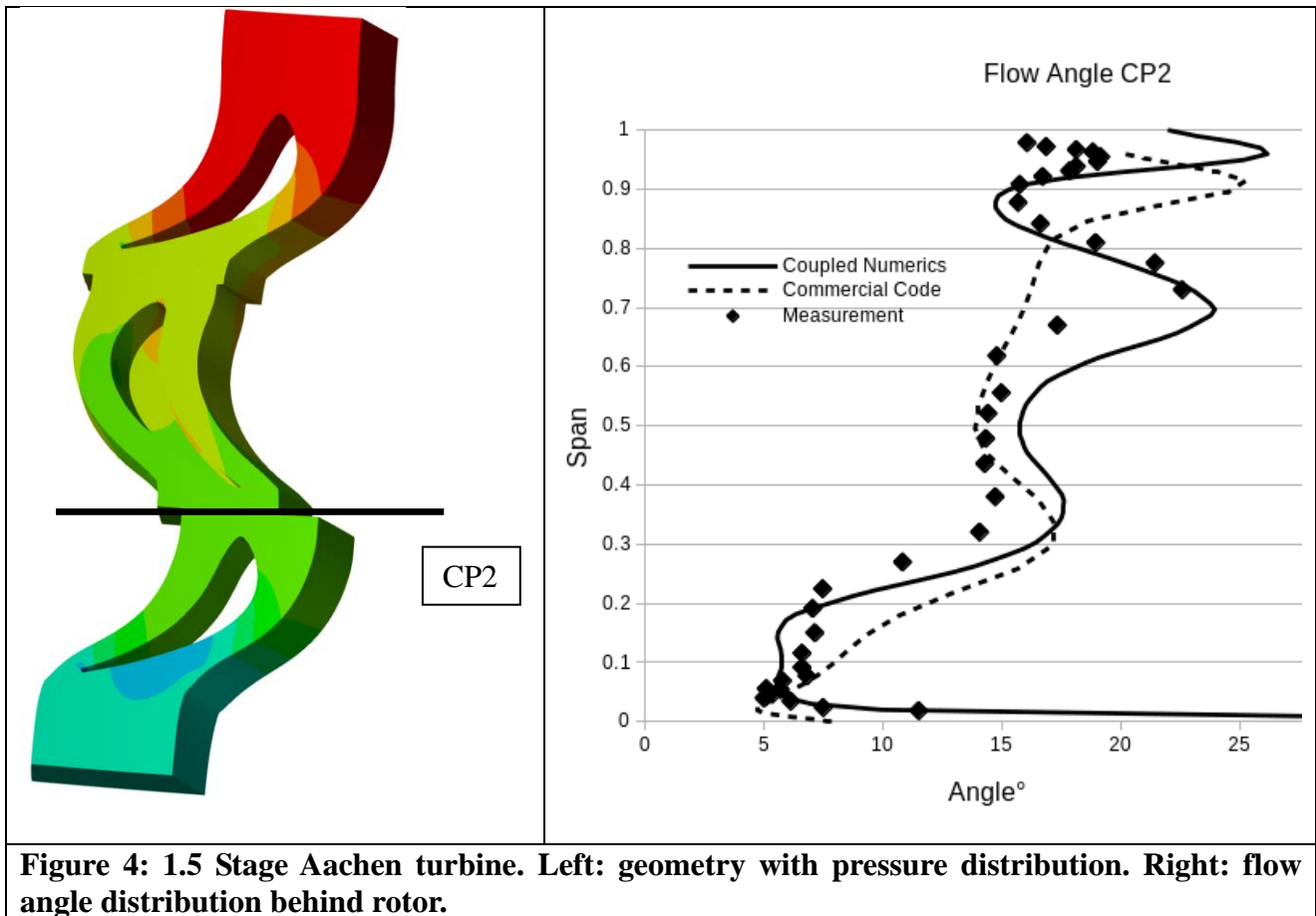
Figure 3: Transition results for ERCOFTAC T3 series flat plate. Left: T3A. Right: T3B.

Mixing plane

The mixing plane interface is a widely used approach to mimic the averaged rotor-stator interaction during the design phase of a stage. The main advantage is that the inherently unsteady flow in a turbomachinery is transformed into a steady situation by circumferentially averaging the flow of the upstream component and imposing it as uniform to the downstream one, considering only spanwise variations [13]. The usual approach to implement the idea of the mixing-plane in CFD codes is based on the connection of the domains by an explicit boundary condition. This straight forward approach has numerical drawbacks, so that for example non-reflecting boundary conditions (NRBC) are additionally needed in order to obtain good results. Furthermore backflow at the interface can lead to instability of the procedure affecting the convergence.

In [14] an implicit mixing-plane has been proposed. The approach does not need additional features like NRBC and is able to cope with shocks and backflow, as can be seen in [15, 16]. This is very important for the robustness of the method which allows capturing off-design conditions in an accurate way, thus allowing a reliable assessment of turbomachinery stages during the design phase also when complex flow features are present at the interface.

As a test case the Aachen 1.5 stage turbine has been used, with a mesh consisting of 700'000 cells and a y^+ between 40 and 70. Fig. 4 presents the computational domain as well as the flow angle distribution behind the rotor, at the so called CP2 position. The mixing plane is positioned at half distance between two adjacent components. The comparison with the experimental data shows a fairly good agreement. From the qualitative point of view it can be stated that all the flow features are captured, specifically the position of these features. Additionally the distribution obtained with a commercial solver is also depicted, in order to assess the code performance. From a quantitative point of view the discrepancies to the measured data are similar or even lower than with the commercial solution. This can be attributed to the implicit treatment of the mixing-plane interface, which is able to enforce circumferentially constant total conditions at every spanwise position downstream of the interface, while the classical NRBC approach can not enforce this condition directly for stability reasons.



Real gas

Usual applications of thermal turbomachinery involve air at relatively low pressure, thus allowing the use of ideal or even perfect gas relations to describe the thermophysical behavior. Nevertheless there is a wide range of machines working close to the two-phase region and/or at very high pressures, so that real gas properties have to be considered. Examples are steam turbines with superheated steam and similarly any ORC turbine, compressors in heat pumps and refrigeration plants and supercritical CO₂ applications to name a few.

Real gas capabilities have therefore been implemented [17]. As an example to show the code performance for this kind of applications, a two stage centrifugal compressor with real gas properties has been computed. Compared to the simulation with ideal gas the computational effort is about 40% higher, which is typical for this kind of problem, as can also be seen in commercial codes.

The computational domain consists of 8 domains connected by 7 mixing planes, with overall 4 million cells and a y^+ of 20. The geometry as well as the overall compressor performance are depicted on Fig. 5. Polytropic efficiency, work coefficient and static pressure ratio are compared with the results computed with a commercial code, showing that the proposed implementation is able to capture the compressor behavior. Additionally the pressure ratio is compared to the measured one, showing a fairly good agreement.

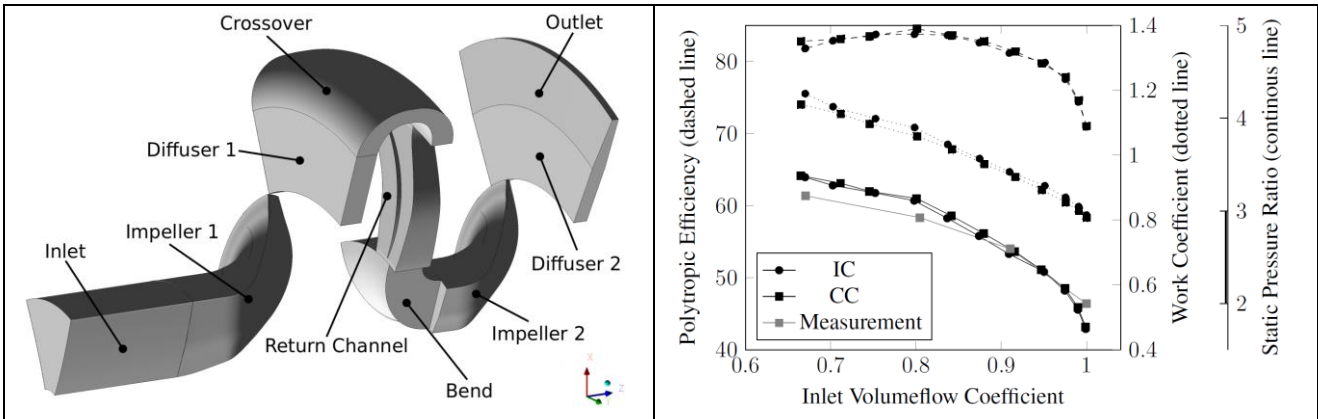


Figure 5: Two-stage compressor computed with real gas. Left: geometry with the domains connected by 7 mixing planes. Right: performance characteristic with polytropic efficiency, work coefficient and static pressure ratio. IC: in-house code (coupledNumerics), CC: Commercial code.

Conjugate heat transfer

A further application field of CFD in turbomachines is the analysis of heat transfer due to large temperature differences between the gas and the mechanical component. To analyse such applications conjugate heat transfer (CHT) is employed, where a joint computation of gas and solid is performed in order to determine the interaction and the temperature behavior in the solid as well as in the gas, for example in gas turbine blades and liners. The solver is able to directly resolve fluid and solid interaction in an implicit way. Thanks to this treatment, no additional FEM solver for the solid is necessary.

In order to show this capability the film-cooled NASA C3X vane has been used, which is representative for high-pressure stators in gas-turbines. The mesh uses a low Reynolds resolution with max y^+ below 1 and consists of a total of 1.4 M cells.

Fig. 6 shows the mesh of the analyzed configuration, as reported in [18]. It can be seen that the wall temperature is well captured along the blade. Only at the suction-side leading-edge (S_{Norm} between 0.3 and 0.5) there is a discrepancy to the experiments. A detailed investigation suggests that in the experiment there is a thermal barrier between the film cooling region and the radial channels which is not completely adiabatic.

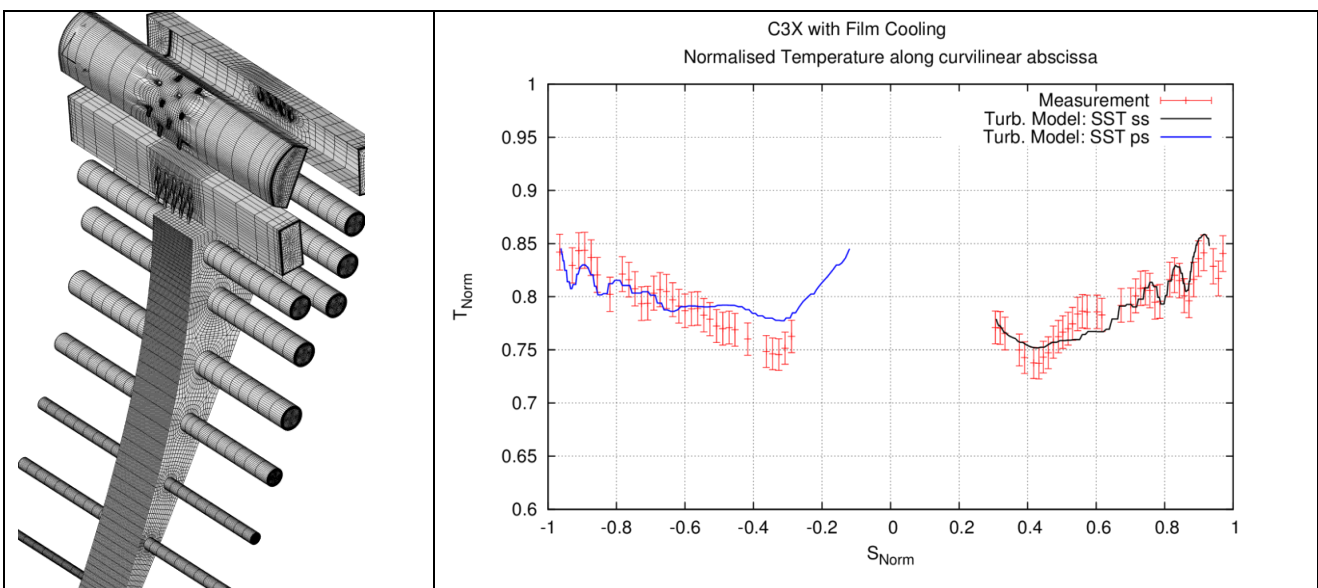


Figure 6: Conjugate heat transfer for the NASA C3X case. Left: geometry with the domains. Right: wall temperature.

Unsteady flow in hydraulic turbine

As mentioned before, turbomachines are inherently unsteady. Nevertheless for a wide range of analysis a steady-state approach is sufficient, using corresponding procedures and simplifications. There are, however, particular flow situations and operating conditions for which unsteady computations are necessary, like (1) rotor-stator interaction to gain information on excitation forces like in adjacent blade-rows or between runner and spiral casing, (2) particular operating conditions like rotating-stall or (3) general instabilities in machines.

In order to assess the unsteady capabilities of the code, a full annulus water turbine including inlet spiral casing with stay vanes, all guide and runner vanes as well as the exit draft tube has been simulated, computing an operating line with constant guide vane opening.

The mesh used is about 6 million cells with a y^+ of 50, and corresponds to a typical size used for industrial applications during the design phase, when computations of the full machine with URANS are performed. For this computations both SST as well as $k-\varepsilon$ turbulence model were used. The use of the $k-\varepsilon$ model was necessary after computation of the characteristic with the SST model showed bad agreement with the results [19]. It has to be mentioned that the flow is largely separated for all computed operating points, with complex vortical structures in the guide vanes as well as in the runner. For these operating points it is found that the SST model, usually more prone to separation than the $k-\varepsilon$ model, generates stable vortex structures which do not capture the unsteady behavior of the machine, thus leading to a stable and therefore wrong characteristic.

Fig. 7 shows the computed characteristic with both models. It can be clearly seen that in these conditions the $k-\varepsilon$ model shows a superior performance than the SST model. On the other hand the full behavior is also not captured by this model. Further investigations [19] have shown that the best results are achieved with an anisotropy model (EARSM). The authors believe that to take in to account turbulence anisotropy for these conditions is necessary, in order to completely capture the machine instability.

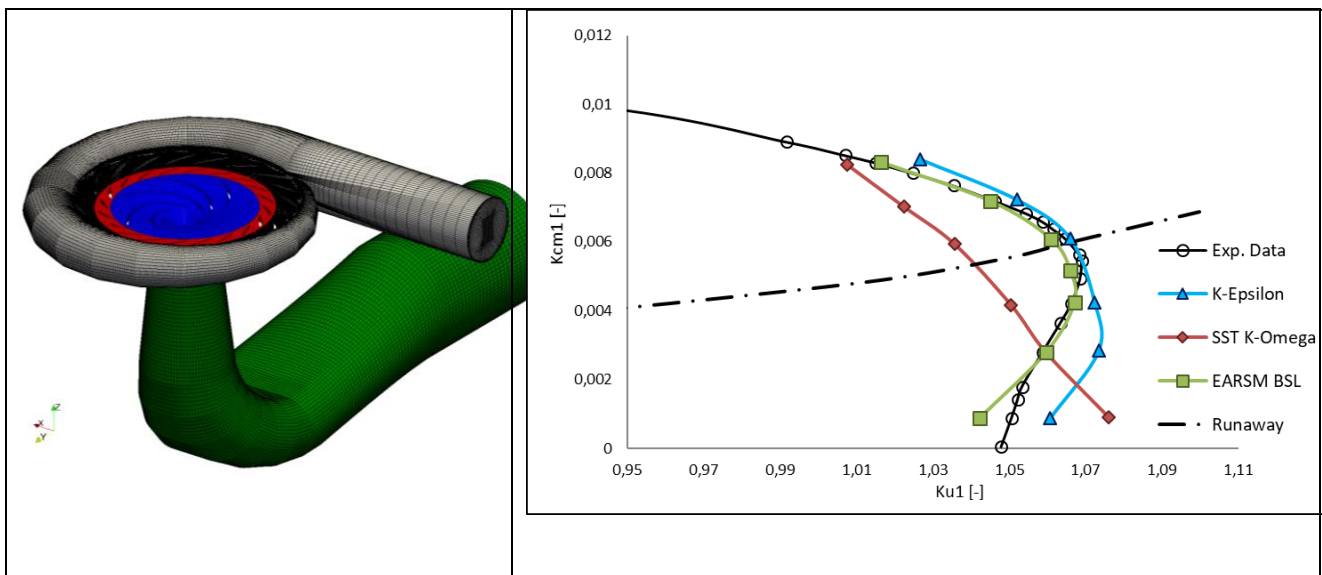


Figure 7: hydraulic turbine. Left: geometry with mesh. Right: characteristic for small opening of guide vanes for turbine and turbine brake operation in 4 quadrant graph.

Adopting a coupled approach largely stabilizes the computational procedure, making very large time steps possible during the transient initial phase before reaching periodic conditions, which often takes several runner revolutions. Steps of up to 15 degrees have been tested showing that the global behavior still can be captured, see [20]. In order to resolve more details after the start-up phase, smaller time steps (in the order of 1 to 3 degrees) are advisable.

CONCLUSIONS

In this publication a new pressure-based, all-Mach solver named coupledNumerics has been presented. Adopting a coupled and implicit implementation of the numerics lead to a fast and robust general purpose code. This code has been used to compute a wide range of turbomachinery flows, showing the capabilities of the code and validating its results against experimental and computed data.

The results show that the proposed approach is suitable to efficiently compute incompressible and compressible turbomachinery flows.

ACKNOWLEDGEMENTS

The authors would like to thank the Institute of Jet Propulsion and Turbomachinery (IST) of RWTH Aachen, Germany for providing the data of the test case "Aachen Turbine".

REFERENCES

- [1] Patankar, S.V., Spalding, D.B., (1972). *A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows*. International Journal of Heat and Mass Transfer, vol. 15, no. 10, pp. 1787–1806.
- [2] Mangani, L., Buchmayr, M., Darwish, M., (2014). *Development of a Novel Fully Coupled Solver in OpenFOAM: Steady-State Incompressible Turbulent Flows*. Numerical Heat Transfer, Part B: Fundamentals, vol. 66, no. 1, pp. 1-20.
- [3] Mangani, L., Buchmayr, M., Darwish, M. (2014). *Development of a Novel Fully Coupled Solver in OpenFOAM: Steady-State Incompressible Turbulent Flows in Rotational Reference Frames*. Numerical Heat Transfer, Part B: Fundamentals, 66(6), 526–543. doi:10.1080/10407790.2014.894372.
- [4] Mangani, L., Darwish, M., Moukalled, F., (2016). *An OpenFOAM pressure-based coupled CFD solver for turbulent and compressible flows in turbomachinery applications*. Numerical Heat Transfer, Part B: Fundamentals, 69(5). doi.org/10.1080/10407790.2015.1125212.
- [5] Menter, F. R., (1994). *Two equation eddy viscosity turbulence model for engineering applications*. AIAA Journal, 32, pp. 1598–1604.
- [6] Menter, F. R., (1993). *Zonal two equation $k-w$ turbulence models for aerodynamic flows*. AIAA Paper (93-2906).
- [7] Menter, F. R., Langtry, R. B., Likki, S. R., Suzen, Y. B., Huang, P. G., and Völker, S., (2004). *A correlation-based transition model using local variables: Part I — model formulation*. ASME Conference Proceedings, 2004(41693), pp. 57–67.
- [8] Langtry, R. B., Menter, F. R., Likki, S. R., Suzen, Y. B., Huang, P. G., and Völker, S., (2004). *A correlation-based transition model using local variables: Part II — test cases and industrial applications*. ASME Conference Proceedings, 2004(41693), pp. 69–79.
- [9] Shih, T.-H., Liou, W. W., Shabbir, A., Yang, Z., Zhu, J., (1995). *A new $k-\epsilon$ eddy viscosity model for high reynolds number turbulent flows*. Computers & Fluids, Volume 24, Issue 3, March 1995, Pages 227-238

- [10] Menter, F.R., Garbaruk, A.V., Egorov, Y. (2012). *Explicit algebraic Reynolds stress models for anisotropic wall-bounded flows*. Progress in Flight Physics (3)
- [11] Hellsten, A., (2005). *New advanced k-omega turbulence model for high-lift aerodynamics*. AIAA Journal, 1857 (43)
- [12] Denton, J., (1997). *Lessons from Rotor 37*. J. of Thermal Science, Vol.6, No.1.
- [13] Denton, J., (1992). *The calculation of three-dimensional viscous flow through multistage turbomachines*. Journal of Turbomachinery, January 1992, Vol. 114/19.
- [14] Hanimann, L., Mangani, L., Casartelli, E., Mokulys, T., Mauri, S., (2014). *Development of a Novel Mixing Plane Interface Using a Fully Implicit Averaging for Stage Analysis*. ASME. J. Turbomach. 2014;136(8):081010-081010-14. doi:10.1115/1.4026323.
- [15] Hanimann, L., Casartelli, E., Mangani, L., (2014). *Reviewing the Implicit Mixing Plane Approach: Theoretical and Applied Cases*. ASME Turbo Expo: Power for Land, Sea, and Air, Volume 2B: Turbomachinery ():V02BT39A017. doi:10.1115/GT2014-25869.
- [16] Mangani, L., Casartelli, E., Hanimann, L., Wild, M., Spyrou, N., (2014). *Assessment of an implicit mixing plane approach for pump-turbine applications*. IOP Conf. Series: Earth and Environmental Science 22 (2014) 022003
- [17] Hanimann, L., Mangani, L., Casartelli, E., Vogt, D., (2019). *Robust integration of real gas models into a pressure-based coupled solver*. Proceedings of the 13th European Conference on Turbomachinery, Lausanne, Switzerland.
- [18] Hylton, L. D., Mihelc, M. S., Turner, E. R., Nealy, D. A., York, R. E., (1983). *Analytical and Experimental Evaluation of the Heat Transfer Distribution Over the Surfaces of Turbine Vanes*. Tech. Rep. 182133, NASA, Lewis Research Center, Cleveland Ohio.
- [19] Deniz, S., Del Rio, A., Casartelli, E., (2018). *Experimental and Numerical Investigation of the Speed-no-Load Instability of a Low Specific Speed Pump-Turbine with focus on the Influence of Turbulence Models*. IOP Conf. Series: Earth and Environmental Science 2018.
- [20] Casartelli, E., Mangani, L., Romanelli, G., Staubli, T., (2014). *Transient Simulation of Speed-No Load Conditions With An Open-Source Based C++ Code*. IOP Conf. Series: Earth and Environmental Science 22 (2014) 032029.