ADAPTION OF THE SINGLE-CHANNEL APPROACH TO USE THE CFD CODE MULTALL AS AN EFFECTIVE TOOL IN THE PRELIMINARY DESIGN OF RADIAL INFLOW TURBINES WITH VANELESS SPIRAL CASING

P. Danieli – M. Masi

Department of Management and Engineering, University of Padova, Vicenza, Italy,
piero.danieli@unipd.it-massimo.masi@unipd.it

ABSTRACT

The preliminary design of radial-inflow turbines is commonly supported by CFD tools which require computation time and resources dependent on the model complexity. Single-channel CFD models allow for rapid and generally sufficiently accurate computations. However, they cannot be applied in non-periodic domains such the ones of turbines with vaneless spiral casing. To overcome this limitation, the present authors suggested in a previous publication a modelling approach which reduces the non-periodic domain of a spiral volute to a periodic one. Also, it was verified that such approach fits the capabilities of the open-source CFD code MULTALL. This code was chosen for its widely validated capabilities and basic physics modelling, which permit fast computations suited to support the preliminary design phase where the rapid check of a large set of designs is most important. In the present paper, the aerodynamic performance of a small-scale turbine with vaneless spiral casing as predicted by single-channel calculations performed with MULTALL, is compared with the prediction obtained using the state-of-the-art CFD code Star CCM+. The single-channel CFD solved with Star CCM+ was calculated on a non-periodic domain made of a turbine slice including only one runner passage, being the boundary conditions extracted from preliminary simulations of the full turbine domain. The results show that the turbine's global performance as predicted in accordance with the periodic-domain approach implemented in MULTALL very well agrees with that obtained from the computations on the non-periodic single-channel domain using a state-of-the-art CFD code. These findings confirm the validity of the proposed modelling approach and demonstrate that MULTALL can be successfully used in the preliminary design of radial inflow turbines with vaneless spiral casing.

KEYWORDS

VANELESS SPIRAL CASING CFD, MULTALL, RADIAL TURBINES

NOMENCLATURE

$\dot{m}$	mass-flow rate	d_s	specific diameter	AR	volute aspect ratio
α	azimuth angle	p	pressure	h	enthalpy
A	surface area	RPM	rotational speed	u	tip speed
n_s	specific speed	η	efficiency	r_p	pressure ratio

INTRODUCTION

The turbomachine design process is based on mathematical models conceived to generate first, a preliminary design which satisfies the design operation and then, a detailed design which optimizes the performance. In the past, both the preliminary and optimized designs were verified by time-consuming and expensive experiments. Starting from the 60's [1], the diffusion of the first computers let the way to CFD computations which have become at present the main tool for the design validation, allowing for a marked reduction of the prototype experimental testing practice. Basically, CFD models can be applied to either full 360-degrees or single-channel domains. At fixed cell numbers, the latter, which includes only a periodical slice of the complete geometry, allows for faster and more accurate results than the former. From the historical point of view, the lack of large computation resources led to early-generation CFD codes able to manage calculations only on single-channel domains [2], whereas present CFD packages permit simulations of both single-channel and 360-degrees domains. However, since large models require longer computation times and higher performance computers, single-channel CFD still has an interesting role in the preliminary design of turbomachines, when it is most important to rapidly check a large set of designs, in order to limit the space to be explored in the succeeding optimization. On the other hand, single-channel computations are based on the periodic domain hypothesis, which strongly compromises their applicability to the analysis of turbomachines embedding non-periodically shaped components, such as the vaneless spiral casings. Single-channel and non-periodic models of spiral volutes can be simulated utilizing the proper boundary conditions as suggested by [3] however, this approach needs of the results of 360-degree simulations for its application and so, it is computationally very demanding. In a previous work [4], the present authors proposed a different approach to account for the effect of the spiral casing on the singe-channel CFD of a small-scale radial-inflow turbine. Differently from single-channel non-periodic models like the just recalled one ([3]), this approach does not involve the need of any prior 360-degrees simulation to be applied. As preliminary validation of the approach, the results of its application were compared to the ones obtained from 360-degrees CFD. A very good agreement was found in the pressure ratio vs mass-flow curve predictions, whereas some remarkable differences resulted in the predictions of the efficiency curves. The spiral casing simulation approach was implemented in the CFD code MULTALL, an open-source code conceived for steady-state computations of high-speed axial or radial cascades [5], to overcome some of the limitations imposed to the code by the single-channel constraint. The use of this early-generation code was motivated by its well-validated reliability coupled with a basic physical modelling, which suggest MULTALL as a good candidate to support the turbomachine preliminary design phase owing to the chance to carry out fast computations even on inexpensive today's laptops. In fact, in another paper [6], the present authors have already assessed the capability of MULTALL to simulate also low-speed axial-flow machines. They compared the performance and detailed data of the local flow field predicted by MULTALL for a tube-axial fan with the corresponding experimental data and predictions of Star-CCM+ and OpenFoam, finding a satisfactory agreement between the different results.

MULTALL is a widely validated code for the aerodynamics of high-speed flows turbines therefore, a comparison with a different code using the same computation setup is neither original from the scientific point of view nor much interesting from the turbine design practice point of view. Moreover, also a comparison based on calculations performed with a different code using more advanced physical modelling would be out of the scope of the present work. On the other hand, it is interesting to compare the results obtained from MULTALL calculations - implementing the vaneless spiral casing simulation approach - with the results obtained from a state-of-the-art CFD code which uses the same physical modelling on the single-channel geometrical domain.

The first aim of the paper is to present a numerical validation of the modelling approach proposed to simulate the spiral volute of radial inflow turbines with vaneless spiral casing. The second aim is to demonstrate that MULTALL, with the vaneless spiral casing approach inclusion, permits predictions of radial turbines' global performance as accurate as those attainable by more demanding single-channel computations performed with state-of-the-art CFD codes.

The validation of the vaneless spiral casing modelling approach is the original contribution of this paper because it quantifies for the first time in the published literature advantages and drawbacks of such modelling approach.

The paper is organized as follows: firstly, an overview about MULTALL and the novel modelling approach is provided. Secondly, the validations instruments and methods are described and eventually, the results are presented and discussed.

BACKGROUND

Historical evolution of MULTALL

MULTALL was conceived and implemented by Prof. John Denton in the 70's and it was first utilized as two-dimensional CFD tool to study the internal flow of axial-flow steam and gas turbines [7]. In the following years, the code undergone several updates to exploit the advancements of the computing technology and improvements of physical modelling. Early-stage updates regarded improvements on the domain discretization - using non-overlapping grids -, and numerical methods – introducing the multi-grid computation technique [8]. In addition, the increase of available computing power led to the extension of the code to fully-3D simulations [9, 10]. In 2000, the code allowed through-flow, quasi-3D-blade-to-blade, 3D single-row, 3D multiple-rows calculations [11]. With regard to the multi-row models, the mixing-plane approach was employed to simulate sliding interfaces [12], while combinations of multiblock and structured meshes were implemented to catch the interactions between primary and secondary flow paths. Until 2006, the code was continuously improved and experimentally validated in several applications of axial-flow turbines. In particular, the "Untreast" solver was added for unsteady operations [13] and the "shroud leakage model" was successfully employed in leakage flow analyses [14]. Between 2006 and 2017 the code was extended also to radial turbines, mixed turbines, compressors and high-speed fans. Finally, in 2017, Denton et al. [5] made available an open-source version of the code named MULTALL. At present, the modelling capabilities of MULTALL are definitely suitable for the preliminary design phase of radial turbines with nozzle vanes under the hypothesis of negligible effects of the spiral casing on the aerodynamics of such turbine arrangement.

The new approach to model vaneless spiral casings by periodic single-channel simulations

In this sub-section a brief description of the modelling approach proposed by the present authors in [4] is summarised. The detailed discussion of the theoretical basis and of the accuracy limits of the modelling approach are presented in [4].

The proposed approach relies on the assumption that a well-designed spiral casing determines an even split of the flow rate between the rotating channels. Accordingly, each periodical "slice" of the spiral volute having azimuthal extension equal to the one of a single rotor blade passage should ideally deliver the same mass-flow rate to the rotor. As far as the volute circumferential throughflow is concerned, the mass-flow rate crossing any volute "slice" without entering the rotor is the entire mass-flow rate processed by the turbine (m_{tot}) minus the amount delivered to the rotor by the slices from the casing throat to the one under analysis. This ideal operation is visualized in Fig. 1 which considers, for the sake of simplicity, a turbine rotor with 4 blades whose leading edges are positioned at 0, 90, 180 and 270 degrees, respectively.

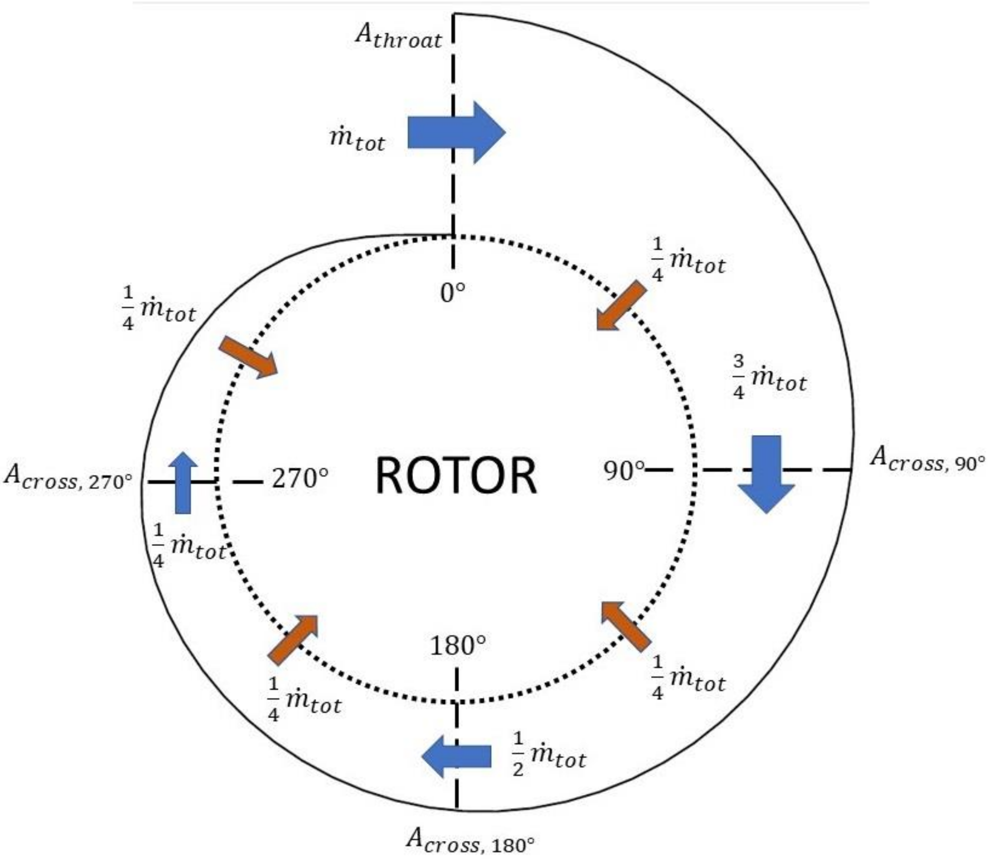


Figure 1: mass-flow rate distribution across an ideal volute

In accordance with the scheme in Fig. 1, the circumferential throughflow of each spiral casing slice corresponding to a single rotor passage (m_{cross}) can be calculated as a function of the azimuth coordinate (α):

$$\dot{m}_{cross} = \dot{m}_{tot} \times \left(1 - \frac{\alpha}{360}\right) \tag{1}$$

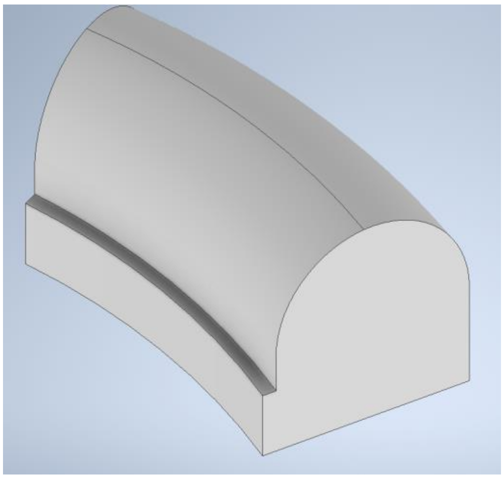
Considering that the cross-section surfaces of the spiral casing reduce almost linearly against the azimuth [15], eq. (1) can be re-written as:

$$\dot{m}_{cross} = \dot{m}_{tot} \times \frac{A_{cross}}{A_{throat}} \tag{2}$$

Where A_{throat} [m^2] is the volute cross-section area at the throat ($\alpha = 0°$).

The left side of Fig. 2 shows the slice of the real spiral casing corresponding to a single-channel of a 13 blades turbine rotor at approximately 180-degrees azimuth.

According to the assumptions and related implications summarized above, it is suggested to model the spiral casing slice of a single-channel domain as an axis-symmetric region obtained from the circular extrusion of the casing cross-section corresponding to 180-degrees azimuth. To account for the difference between the mass-flows crossing the two azimuthal surfaces bounding the real spiral casing (i.e., the flow rate entering the rotor channel), the outermost half-part of the axis-symmetric equivalent casing is removed. This permits an inflow from the cylindrical surface bounding the domain, which is not allowed by an axis-symmetric flow model. It can be noted that under the hypothesis of uniform annular flow distribution, eq. (2) remains valid also for any surface portion of a specific cross section of the volute.



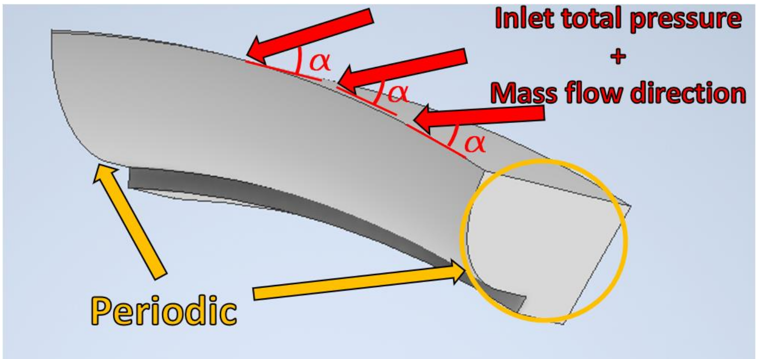


Figure 2: complete single-channel spiral volute (left) and reduced version (right) for the application of the proposed approach

The right side of Fig.2 shows the reduced version of the spiral casing with the boundary conditions required by the modelling approach at each surface. The total pressure inlet with specified flow direction imposed to the outermost cylindrical surface is complemented by the static pressure outlet must be imposed at the turbine domain exit surface. The pressure values must fulfill the pressure ratio datum of the turbine operation to be simulated. All the surfaces bounding the domain in the azimuth direction except for the blade surfaces are modelled as periodic boundaries, in accordance with the standard practice for MULTALL calculations. Note that the inlet flow direction (obliquity) in the circumferential plane must be fixed to achieve a circumferential throughflow equal to 0.5 m_{cross} to satisfy the model constraint imposed by eq.(1) at α=180-degrees.

Figure 3 compares the pressure ratio (r_p) vs mass-flow rate and adiabatic efficiency (η_{TS}) vs tip speed ratio (u/C_0) curves obtained from the single-channel calculations performed using MULTALL with the vaneless spiral casing implementation (blue markers), and the corresponding performance curve obtained from full 360-degrees turbine domain calculations performed with the state-of-the-art CFD code Star CCM+ (green markers). The dimensionless performance parameters are defined in accordance with the following eqs. (3) to (5), in which u [m/s] is the tip speed of the runner and Δh_{is} [kJ/kg] is the isentropic specific work, h_{in}^0 [kJ/kg] is the total enthalpy at the domain inlet, h_{out} [kJ/kg] is the static enthalpy at the rotor outlet, $h_{is,out}$ [kJ/kg] is the isentropic static enthalpy at the rotor outlet, p_{in}^{0} [bar] is the inlet total pressure, p_{out} [bar] is the outlet static pressure and C_0 [m/s] is the isentropic tip speed.

$$\eta_{TS} = \frac{(h_{in}^{0} - h_{out})}{(h_{in}^{0} - h_{is,out})} \tag{3}$$

$$r_p = \frac{p_{in}^{0}}{p_{out}} \tag{4}$$

$$\frac{u}{C_0} = \frac{u}{\sqrt{2 * \Delta h_{is}}} \tag{5}$$

Note that, to keep the computation time to a level reasonable for the support of the turbine preliminary design phase, the cell numbers in the 360-degrees domain are not much higher than twice the cell numbers in the single-channel domain. Accordingly, the 360-degrees domain grid is very coarse, and the predictions shown in Fig. 3 are certainly dependent on the grid size.

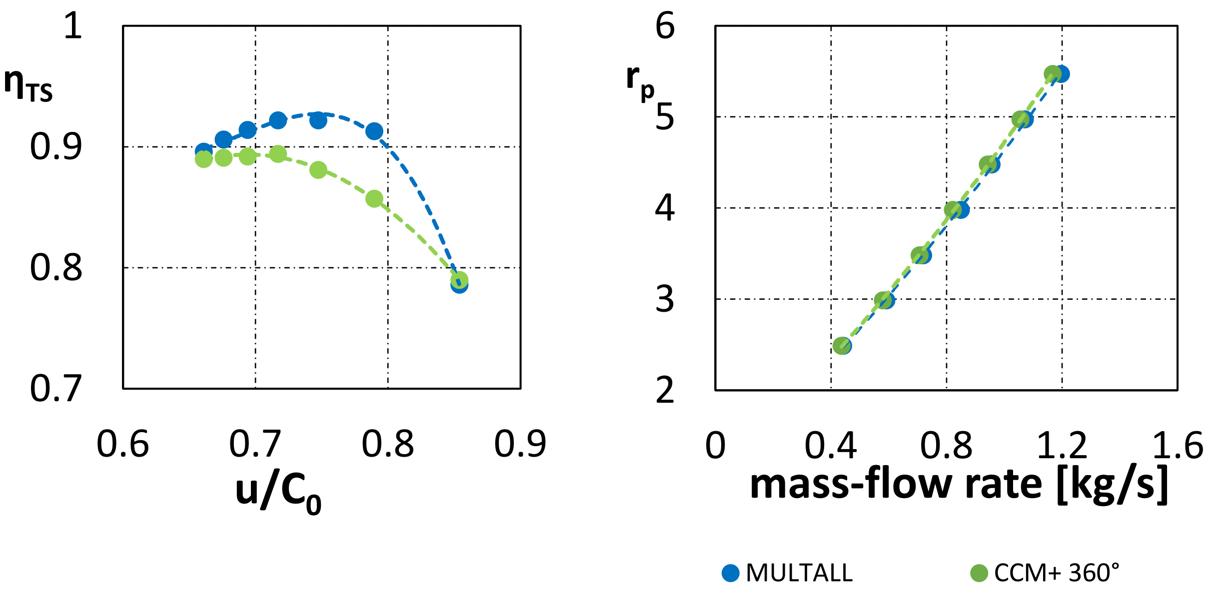


Figure 3: comparison of the simulation results (adapted from [4])

A very good agreement was found in the pressure ratio vs mass-flow curve predictions, whereas some remarkable differences in the efficiency predictions are clearly visible.

As stated in the Introduction, these results (discussed in detail in [4]) represent a partial validation of the vaneless spiral casing modelling approach. In fact, they do not permit to clearly assess its reliability because it is not possible to quantify how much of the difference in the predictions is due to the grid density, the uneven distribution of the mass-flow between different rotor channels or other oversimplifications of the spiral casing modelling approach.

INSTRUMENTS AND METHODS

This Section is divided into three sub-sections. The first provides the main geometrical parameters of the turbine utilized as case of study, the second describes the single-channel model implemented in MULTALL and the third illustrates the single-channel model generated with the state-of-the-art code Star-CCM+.

Case of study

The selected case of study is a nano-radial inflow turbine designed in accordance with a well-known design procedure [16] to match the optimal design specifications reported in Tab.1. The table also reports the main geometrical parameters of the turbine. More details on the turbine geometry can be found in [4].

Table 1: main design parameters of the radial inflow turbine

Radial inflow turbine design parameters	Specific speed, n_s *[-]*	0.56
	Specific diameter, d_s *[-]*	3.62
	Rotational speed, *RPM [rev/min]*	60000
	Absolute inlet total pressure, p_{in} *[bar]*	8.01
	Absolute outlet static pressure, p_{out} *[bar]*	2.01
Runner geometrical parameters	External diameter *[mm]*	50.39
	Blade count	13
Spiral casing geometrical parameters	Throat centroid radius, r_1 *[mm]*	61.73
	Throat Area, $A_{throuat}$ *[mm²]*	468.90
	Aspect ratio, *AR [-]*	1.23

MULTALL 3D single-channel model

The domain of the 3D single-channel model is generated in accordance with the guidelines summarised in the Background Section. The fully-structured grid includes near-wall refinement. The numerical validation has been carried out by comparing the results obtained from simulations with different grid density and checking the grid dependency of the global performance parameters. The one-equation Spalart-Allmaras [17] has been utilized for the turbulence closure. Each computation was stopped once the average residual (ratio between the percentage change in velocity and the RMS velocity as defined in [4]) reached values lower than 0.001 and the continuity error dropped below 0.01. Figure 4 shows the predicted mass-flow rate and total-to-static efficiency of the turbine (defined in eq. (3)) as a function of the cell numbers.

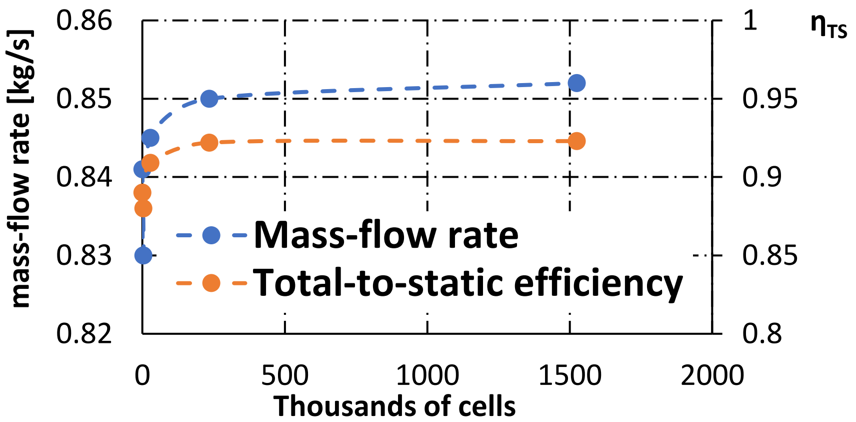


Figure 4: number of grid cells against predicted performance parameters

Table 2 lists the features of the 5 grids used for the numerical validation.

Table 2: grid features specification for the numerical validation

Grid ID	Total number of grid cells	Volute model	Runner	Downstream
1	324	6x5x4§	6x5x12§	6x5x2§
2	3780	12x9x8§	12x9x23§	12x9x4§
3	28550	23x18x16§	23x18x45§	23x18x8§
4	234890	46x37x32§	46x37x90§	46x37x16§
5	1525000	92x74x64§	92x74x128§	92x74x32§
				§ *span x pitch x streamwise*

According to the simulation results shown in Fig. 4, the differences between mass-flow rate and η_{TS} predicted using either ID4 or ID5 grid were below 0.2% and 1%, respectively. Thus, the ID4 grid was deemed as accurate enough for the current study. In the selected case, the wall-y^+ was equal to 0.5. The wireframe plots in Fig.5 show some details of the ID 4 grid [4].

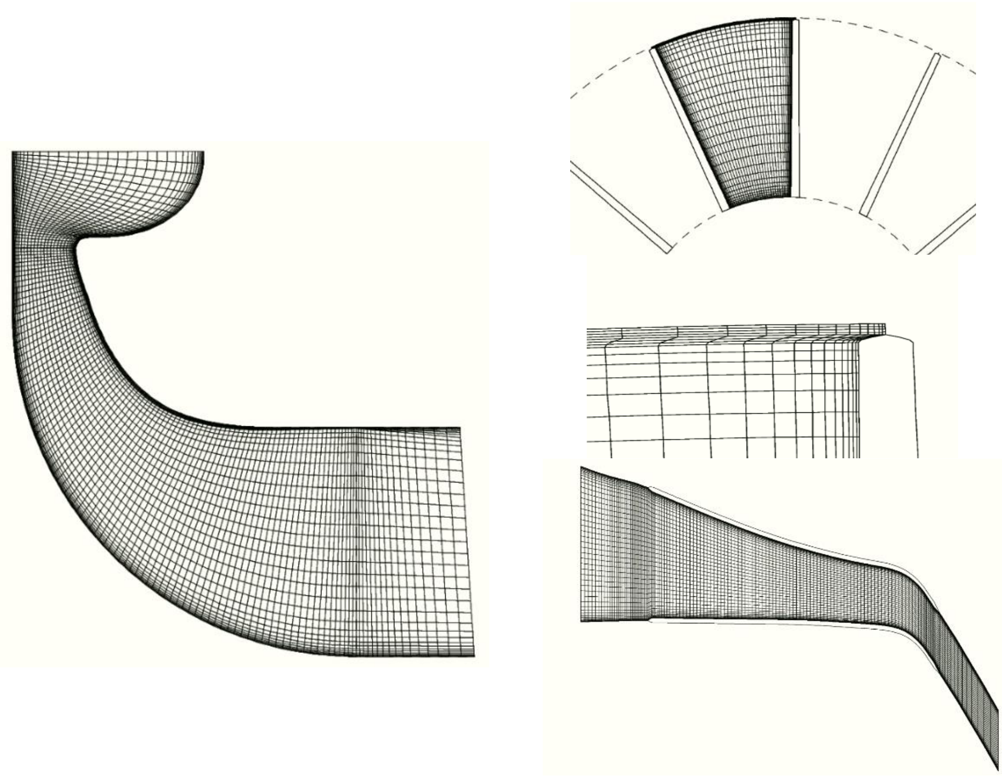


Figure 5: wireframe plots (taken from [4]) of the ID 4 grid used for the single-channel computations with MULTALL

Star-CCM+ 3D single-channel model

To perform a validation of the modelling approach more rigorously than the preliminary one summarized in the Background Section, the single-channel modelling approach used among others by [3] was implemented in Star CCM+. In particular, the 360-degree 3D model of the turbine presented in [4], and previously recalled, was used to extract the values of the boundary conditions to be applied on the boundary surfaces of a single channel domain. The latter was generated by slicing the 360-degree model at α=180 degree and keeping only one rotor channel and the corresponding parts of the real spiral casing and turbine discharge. A first set of computations was performed on the 360-degree model composed of approximately 600-thousand grid cells fixing as boundary conditions: i) rotational speed of the rotor, ii) total-inlet-pressure (at the spiral casing entrance), and iii) static-pressure (at the domain exit). The local flow field predicted by the computations at the boundary surfaces of the spiral casing corresponding to single-channel domain was extracted and applied to the latter. Then, a second set of computations was performed on the single-channel model. The latter, which counted approximately 46000 cell numbers in the original 360-degree domain, was refined up to 235000 cells to achieve grid features practically equal to those of the MULTALL model. In fact, a grid sensitivity analysis was performed for the Star CCM+ single-channel model. The results are showed in Fig. 6 from which it can be observed that the differences between mass-flow rate and η_{TS} predicted using 235,000 and 2,500,000 cells were below 0.3% and 0.2%, respectively. Accordingly, a grid (made of a polyhedral core and 4 Near-wall prism layers) counting approximately 235,000 cell numbers was chosen as final grid for the comparison (validation) with the periodic single-channel approach implemented in MULTALL. Figure 7 shows the 360-degree and single-channel domains on the left-side frames and the contour plots of the static pressure extracted from the complete domain and applied to the single-channel domain on the right-side frames. The convergence criterion used for CCM+ 3D single-channel model calculations was the achievement of residuals lower than 10^{-5} for all the conservation equations.

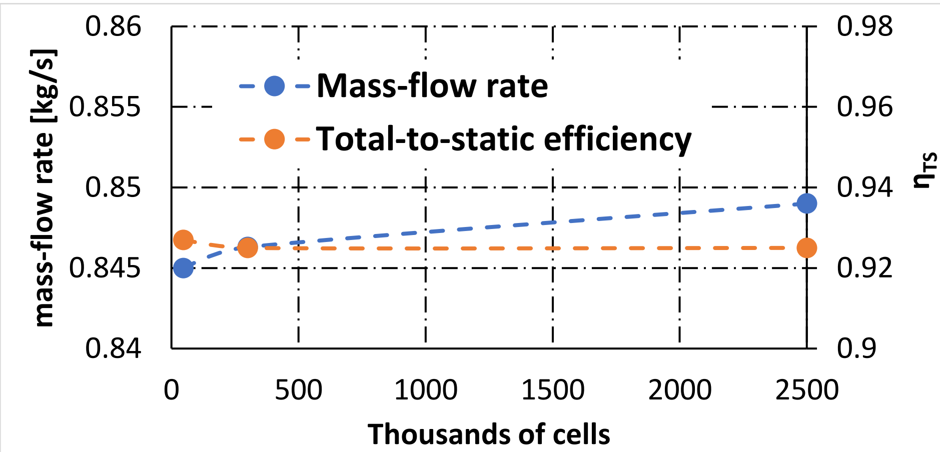


Figure 6: number of grid cells against predicted performance parameters

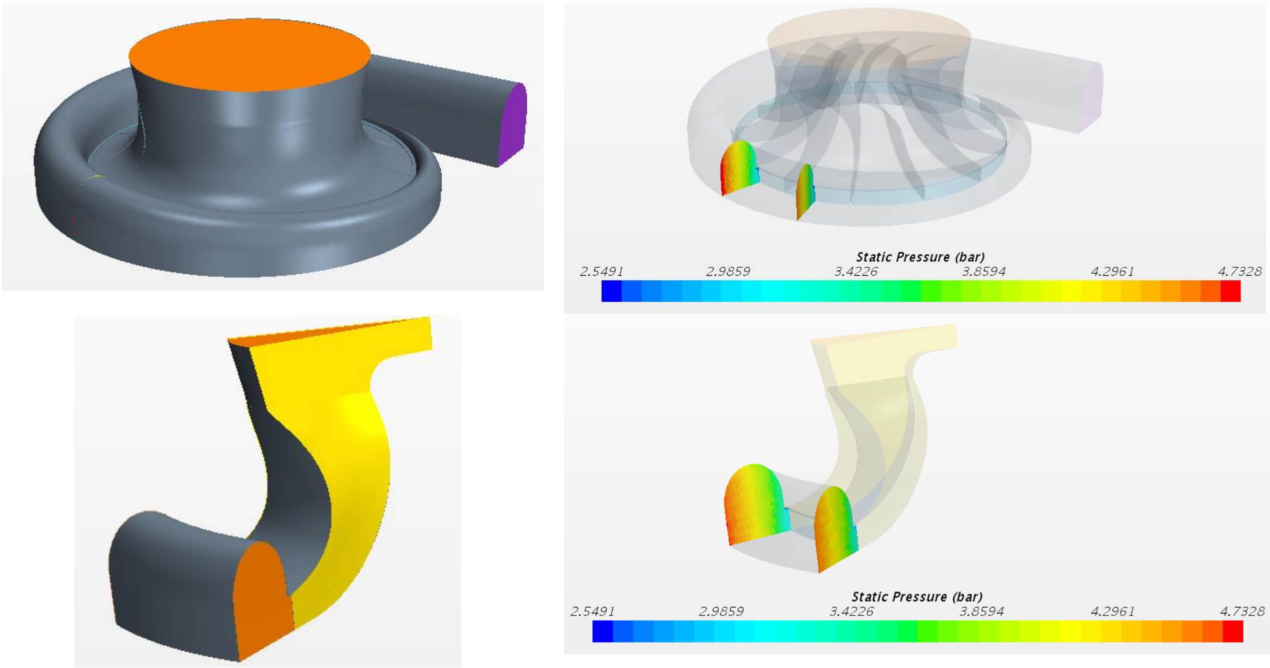


Figure 7: complete and single-channel numerical domain (left) and extracted boundary conditions (right)

In particular, the boundary conditions fixed for the single-channel computations were:

- Flow direction, Mach number, static pressure and static temperature distributions (from the 360-degree simulations) at the entrance surface of the spiral casing slice.
- Static pressure distribution (from the 360-degree simulations) at the exit surface of the spiral casing slice.
- Constant static pressure at the domain exit.
- Rotating walls at the blade and rotor hub surfaces.
- No slip walls at the metal surfaces of the turbine's shroud and casing.
- Periodicity at the remaining side surfaces.

RESULTS

Triangle and circle markers in Fig.8 indicate the turbine operation points simulated with CCM+ and MULTALL single-channel models, respectively.

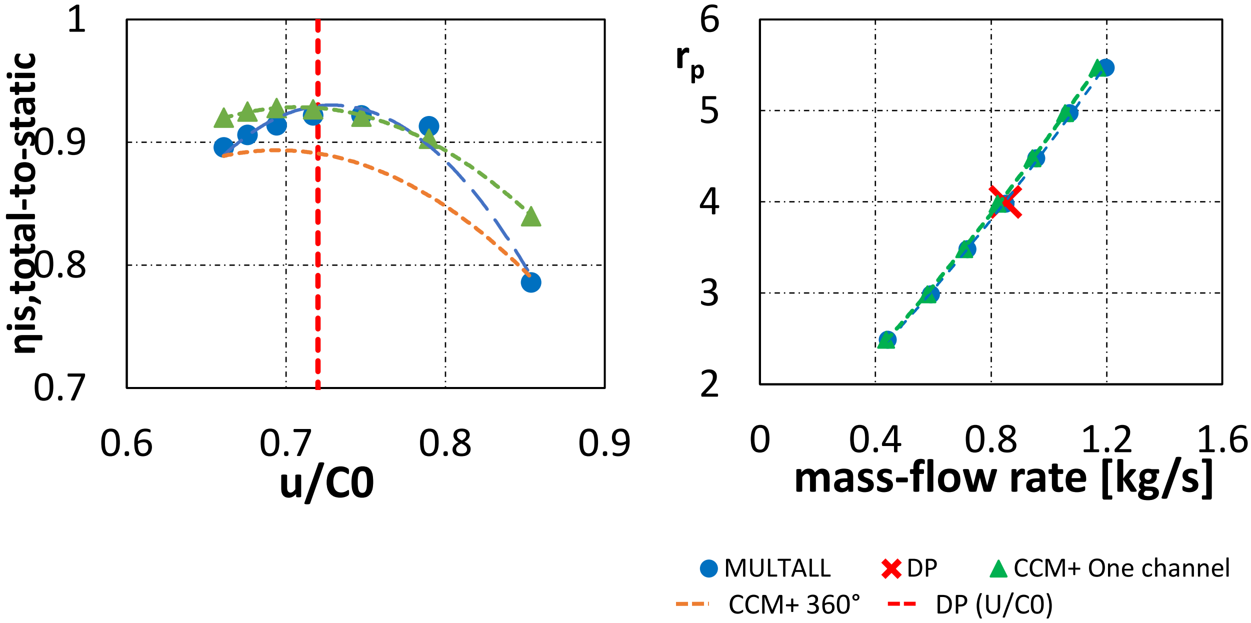


Figure 8: comparison of the simulation results

The good agreement of the pressure ratio vs mass-flow rate curves already achieved by the results shown in Fig. 3 is confirmed also for the CCM+ single-channel model (see the curves on the right side of Fig. 8). In particular, the predictions of the two single-channel models differ each other always by less than 1%, being the maximum difference occurred at the highest simulated pressure ratio. Comparing the efficiency trends calculated with Star-CCM+, it is evident that the single-channel model overestimates the prediction of the 360-degree version by approximately 3% over the entire operation range. This difference can be attributed to the grid differences (refined grid for the single channel domain) and to the flow field distribution imposed by single-channel simulations compared to the full actual domain.

Most important, also the efficiency predictions of the two single-channel computations are almost superimposed in the $0.68 < u/C_0 < 0.8$ operation range, whereas the difference in the predictions at turbine operations far from the maximum-efficiency u/C_0 remains in between 0.5% and 3%.

Interesting to note from the preliminary design point of view, that both the single-channel models capture rather well the design point operation, indicated by the red cross and red line in Fig. 8.

The comparison between the performance predicted by the 360-degrees and single channel CCM+ model confirms that most part of the discrepancy between MULTALL and CCM+ shown in Fig.3 (at least in the surroundings of the design point) is not due to the vaneless spiral casing modelling approach and can be attributed to the coarseness of the 360-degrees grid. In particular, it is demonstrated that the uneven distribution of the turbine mass-flow rate between different rotor channels, which is neglected by the spiral casing simulation approach in MULTALL and accounted in the CCM+ single-channel model, plays (at least for the case studied here) an appreciable role on the efficiency predictions only at turbine operations far from design. Thus, it can be stated that the suggested spiral casing simulation approach for single-channel computations demonstrated to be well suited to support the turbine preliminary design with CFD.

Eventually, it is worth to consider that, although the computation times required by the two single-channel models are almost equal, two relevant advantages are awarded to the adoption of MULTALL with the spiral casing modelling approach. First, the MULTALL version used in this work is not implemented with parallel computing technology so that the equivalence in computation time was achieved by the CCM+ single-channel model only because of the ability of such commercial package to exploit the multi-core processors (4 threads in the present case). Obviously, this represents a drawback for MULTALL, but remarks the large potential of this code. Second, the CCM+ single-channel computations need for an additional set of preliminary computations of the 360-degrees domain to obtain the boundary conditions necessary for the reliable solution of the non-periodic single-channel domain. Further investigations will be conducted to validate the approach by experiments.

CONCLUSIONS

The presented work aims to validate rigorously an innovative technique which permits to account for the non-axis-symmetric geometry of turbines' vaneless spiral casing in single channel computations. A Star-CCM+ single-channel non-periodic model was implemented and solved using as boundary conditions the local flow field quantities extracted from computations performed on the 360-degrees turbine domain. The computation results were compared to the ones obtained by using MULTALL, in which the novel modelling approach is implemented to include the spiral casing in the single-channel domain. The results confirm the validity of the modelling approach in the surrounding of the design-point operation and most important, they emphasize the potential of MULTALL. In particular, the two compared codes shown equivalent computation times even though MULTALL is not implemented with parallel computation technology. This indicates that MULTALL has the capabilities to compete with state-of-the-art codes in applications where the accuracy of basic physical models is sufficient. Accordingly, this work demonstrated that MULTALL can be successfully used as supporting tool for the preliminary design of radial inflow turbines with vaneless spiral casing and also, that the code has the potential to carry out faster simulations than present CFD codes.

REFERENCES

[1] Hess, J.L.; A.M.O. Smith (1967). "Calculation of Potential Flow About Arbitrary Bodies". Progress in Aerospace Sciences. 8: 1–138.
[2] Harlow, F.H. (1955). "A Machine Calculation Method for Hydrodynamic Problems". Los Alamos Scientific Laboratory report LAMS-1956.
[3] Simpson A.T., Spence S.W.T., Watterson J.K. (2009). A Comparison of the Flow Structures and Losses Within Vaned and Vaneless Stators for Radial Turbines. Journal of Turbomachinery. JULY 2009, Vol. 131 / 031010-15.
[4] Danieli, P., Masi M., Lazzaretto A., Carraro G. (2021). "An engineering approach for the fast simulation of radial inflow turbines with vaneless spiral casing by single-channel CFD models". (Accepted for conference publication). ATI 2021 – 76th Conference of the Italian Thermal Machines Engineering Association

[5] Denton, J.D., (2017), "Multall - an open source, computational fluid dynamics based, turbomachinery design system", Journal of Turbomachinery, Volume 139, Issue 12, Article number 121001.

[6] Danieli, P., Masi M., Delibra G., Corsini A., Lazzaretto A. (2021). "Assessment of MULTALL as computational fluid dynamics code for the analysis of tube-axial fans." Journal of Turbomachinery 143(7).

[7] Denton, J.D., (1978). "Throughflow Calculations For Transonic Axial Flow Turbines", J Eng Power Trans ASME, Volume 100 (1978), Issue 2, Pages 212-218.

[8] Denton, J.D., (July 1983). "An improved time-marching method for turbomachinery flow calculation", Journal of Engineering for Gas Turbines and Power, Volume 105, Issue 3, Pages 514-521.

[9] He, L., Denton, J.D., (1994). "Three-dimensional time-marching inviscid and viscous solutions for unsteady flows around vibrating blades", Journal of Turbomachinery, Volume 116, Issue 3, Pages 469-476.

[10] Denton, J.D., Xu, L., (1999). "The exploitation of three-dimensional flow in turbomachinery design", Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering, Volume 213, Issue 2, Pages 125-137.

[11] Denton, J.D., Dawes, W.N., (1999). "Computational fluid dynamics for turbomachinery design", Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering. Volume 213, Issue 2, Pages 107-124.

[12] Denton, J.D., (1992). "The calculation of three-dimensional viscous flow through multistage turbomachines", Journal of Turbomachinery, Volume 114, Issue 1, Pages 18-26.

[13] Pullan, G., Denton, J.D., and Dunkley, N., (2003). "An experimental and computational study of the formation of a streamwise shed vortex in a turbine stage", Journal of Turbomachinery, Volume 125, Issue 2, Pages 291-297.

[14] Rosic, B., Denton, J.D.,and Pullan, G., (2006). "The importance of shroud leakage modeling in multistage turbine flow calculations", Journal of Turbomachinery, Volume 128, Issue 4, Pages 699-707.

[15] Chappie, P. M., Flynn, P. F., and Mulloy, J. M. January 1, 1980. "Aerodynamic Design of Fixed and Variable Geometry Nozzleless Turbine Casings." ASME. J. Eng. Power. January 1980; 102(1): 141–147.

[16] Whitfield, A., Baines, N. C. 1990. "Design of radial turbomachines". Harlow, Essex, England, Longman Scientific& Technical.

[17] Spalart, P. R., Allmaras, S. R. 1992. "A One-equation Turbulence Model for Aerodynamic Flows", AIAA, Paper 92-0439.