MATEC Web of Conferences **328**, 04005 (2020) https://doi.org/10.1051/matecconf/202032804005
XXII. AEaNMiFMaE-2020

Quasi-unsteady Flow Field Computation and Performance Prediction of Contra-Rotating Fan

Bystrík Červenka[1*], *Michal* Holubčík[1] , and *Milan* Malcho[1]

[1]Department of Power Engineering, Faculty of Mechanical Engineering, University of Žilina, Univerzitna 1,010-26 Žilina, Slovakia

Abstract. Discussion of numerical flow field prediction with two rotor-stator interface models is presented in the paper. The mixing plane and harmonic models are applied to contra rotating fan. Both models confirmed good performance and high power density leading to compact configuration of the contra rotating fan compared to centrifugal fan. The harmonic model provides realistic wake and potential effect propagation through the rotor-rotor interface. In addition, it provides user with static pressure frequency and amplitude inputs for aerodynamic noise assessment.

1 Introduction

Fan performance and air flow rate have notable effect on overall performance of drying cycle. There are several, usually contradicting, requirements a fan must fulfil in drying process in terms of flow rate, pressure rise, noise and installation space.

Nomenclature

Symbols			
c	absolute velocity (m/s)	RS	rotor-stator
CRF	Contra Rotating Fan	t	time
E	specific internal energy (J/kg)	u	circumferential blade velocity
i	imaginary unit	U	conserved variable
k	harmonic number	w	relative velocity (m/s)
MP	mixing plane	Y	Specific energy (J/kg)
N	number of harmonics		
ω	angular speed of blade passing frequency	*Subscripts*	
Pref	reference pressure (Pa)	I, II	first and second fan stage
r	Cartesian spatial vector	1, 2	inlet to blade row
RANS	Reynolds Averaged Navier-Stokes	3, 4	outlet from blade row
ρ	density (kg/m^3)	u	circumferential component

High performance and compact fan designs are required for today domestic air-drying applications. Contra rotating axial fan (CRF) represents very compact, yet powerful

[*]Corresponding author:: cervenka2@stud.uniza.sk

alternative to the axial and centrifugal fan with classical rotor-stator stage configuration. The CRF configuration is known form long time and its advantages have been confirmed by many studies [1-3].

The accurate numerical performance prediction, making use of mixing plane and harmonic method, and flow structure of a CRF is the subject of the investigation described in the paper.

2 Design essentials of contra-rotating axial fan

There are several configurations of CRF, differing mainly in drive concepts. Two separate motor drives, with independent rotating speed, are assumed for the purpose of the study.

The essential part of the CRF design is generation of negative pre-whirl for the second stage of the fan, which increases the transmitted power to the air flow and its outlet pressure. Assuming axial flow at the inlet to the first stage and outlet of the second stage, Figure 1, the specific work of the ideal CRF could be determined by application of Euler turbomachinery equation:

$$Y = Y_I + Y_{II} = c_{u2,I}(u_I + u_{II}) \tag{1}$$

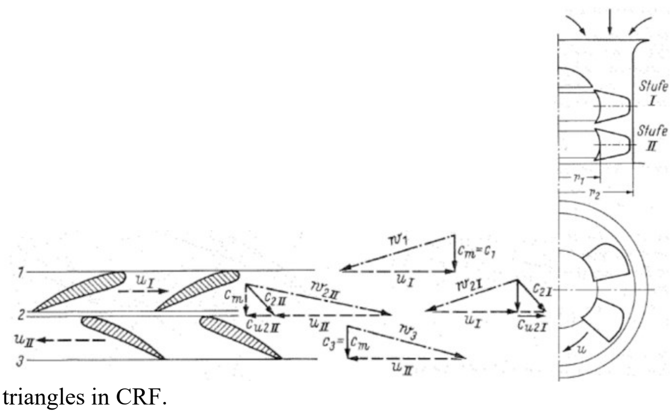

Fig. 1. Velocity triangles in CRF.

If the circumferential velocity is same in both stages, specific work is essentially double the work transmitted in the usual stage composed of impeller and stationary outlet guide vanes.

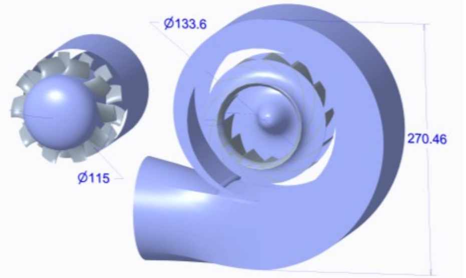

Fig. 2. Comparaison of CRF and centrifugal fan design with 6000 rpm.

Important property of the CRF configuration is possibility to design the two stages with zero whirl at outlet from CRF. The properties could be used to design a very compact CRF design in comparison to conventional centrifugal fan stage designed for same operating point and impeller rotation speed of 6000 rpm. The CRF occupies less than half the space in radial direction of that required for centrifugal fan.

2.1 CRF geometry

The CRF design, presented in **Chyba! Nenašiel sa žiaden zdroj odkazov.**, is subject to the numerical investigation.

Table 1. CRF operating point design.

	CRF	Stage I	Stage II
Flow rate (m^3/h)	300	300	300
Total pressure difference (Pa)	900	600	300
Rotation speed (rpm)		8000	4000
Number of blades		10	13

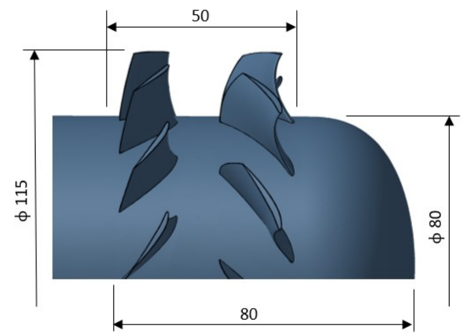

Fig. 3. CRF geometry.

3 Quasi-unsteady model for turbomachinery flow computations

There are couple of main types of the models available in the literature and commercial software dedicated to the turbomachinery flow simulation. The models essentially differ in performance prediction accuracy while considering limited computational resource for a particular single or multistage turbomachine configuration, [4].

The accuracy is essentially represented by fluid flow properties continuity across rotor-stator (RS) interface, while maintaining dynamics of interaction across RS. The computational resources are, to a greater extent, defined by domain size, which is implied by the RS model requirements. The RS model thus represent essential part of the turbomachinery modelling.

The designer is usually interested in a quasi-steady, or time averaged, performance of a turbomachine, accepting relatively small unsteady performance variation due to periodic passing of rotor blades relative to stator blades. Though, the purely steady state (RS) models, e.g. mixing plane model, introduces error into the model, due to missing dynamic RS effect on the time averaged performance.

There is a quasi-unsteady model available, located itself halfway, in terms of accuracy, between mixing plane, the least computationally demanding and sliding mesh as the heaviest, though most accurate model. The quasi-unsteady model takes advantages of the mixing plane model, in terms of computational resources. The model still provides improved accuracy of performance prediction, when taking into account dynamic effect of RS interaction on time average flow [5, 6].

3.1 Harmonic quasi-unsteady method

The quasi-unsteady model is developed is similar way as the Reynolds averaged Navier-Stokes model (RANS), [7]. The resulting unsteady flow, represented by a conserved variable $U = (\rho, \overline{\rho w}, \rho E)$ is decomposed into a time averaged flow $\overline{U}(\bar{r})$ and its unsteady part [7]. The unsteady part is further decomposed into so called perturbations, making use of Fourier analysis:

$$U(\bar{r},t) = \overline{U}(\bar{r}) + \sum_{k=1}^{N} U'(\bar{r},t) \tag{2}$$

$$U'(\bar{r},t) = \sum_{k=1}^{N} \widetilde{U_k}(\bar{r})\, e^{i\omega_k t} + \widetilde{U_{-k}}(\bar{r})\, e^{i\omega_{-k} t} \tag{3}$$

The perturbations are defined by complex conjugates of harmonic amplitudes; $\widetilde{U_k}$, $\widetilde{U_{-k}}$, fundamental blade passing frequency and their higher k multiplies ω_k, ω_{-k}.

Introduction of the conserved variable into the RANS model results into harmonic source term, as incremental part to usual Coriolis and centrifugal volume source. The harmonic perturbation source thus has effect on time mean flow and predicted performance of a turbomachine.

4 CRF aerodynamic model

4.1 Computational domain

The harmonic method, used for the simulation of the CRF, requires only one blade pitch to be meshed for each of the rotors.

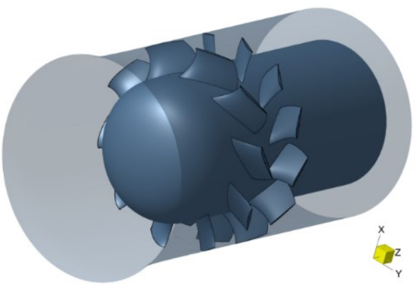

Fig. 4. Simulation domain of quasi-unsteady flow field.

Despite the reduced region meshed, it is still possible to obtain unsteady flow field solution for the full 360º region of the blade rows. The unsteady solution is reconstructed from harmonic flow field solution by applying equation (1).
The simulated domain can thus be considered full annulus of the two blade rows, including the blade tip clearance, as showed by the Figure 4.

4.2 Numerical model

The actual computational domain is discretized by fully conformal multiblock hexahedral mesh for both of the blade rows, Figure 5. The mesh contains about 1.2 and 0.9 million cells on finest grid level of stage I and II, respectively. The mesh resolution is chosen for computation of flow field quantities within boundary layer without wall function and with Spalart-Allmaras turbulence model.

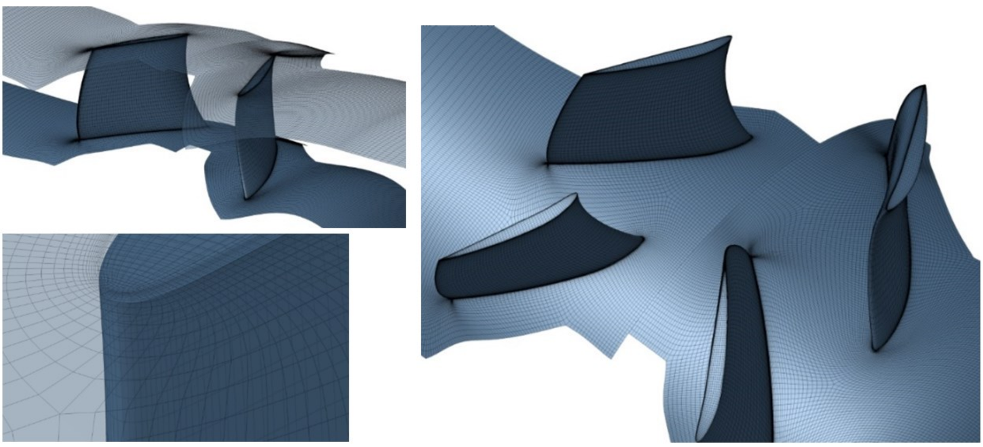

Fig. 5. Computational mesh of CRF.

The accuracy of the rotor-rotor interaction and reconstructed unsteady flow field solution is governed by selection of number of harmonic frequencies N resolved. For practical purposes it is usually sufficient to consider three harmonics in the model.

5 Global CRF performance

The aerodynamic performance of CRF is characterized by decreased shaft power when flow rate increases from best efficiency point, as shown on Figure 6. The internal aerodynamic efficiency is predicted on the level of 77 % at the best efficiency point.

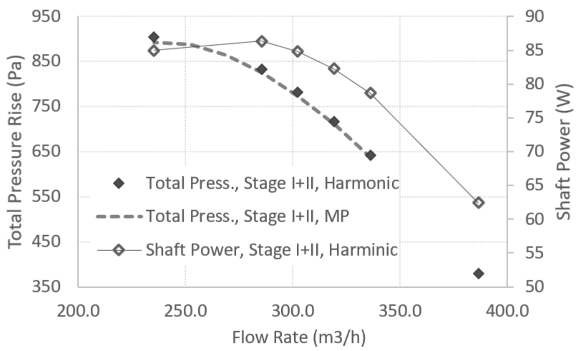

Fig. 6. Global aerodynamic performance prediction of CRF.

Typical property of CRF is nearly axial outlet direction of absolute velocity vector. The average value of outlet blade to blade angle downstream the blades of the second stage is -9º despite missing stationary outlet guide vanes, Figure 7.

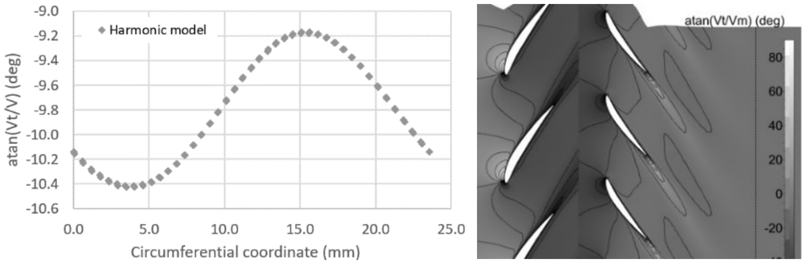

Fig. 7. Blade to blade angle variation in time mean flow downstream stage II along 50 % span.

6 CRF Flow field prediction

There is essential difference in predicted time mean CRF flow field structure between the two model used. While the harmonic method allows for, up to certain degree, continuous convection of transport quantities across the rotor-stator (RS) interface in time, the MP model assumes flow quantities to be mixed out at the RS interface without variation in circumferential direction. The real time mean flow is thus approximated by circumferential averaging of flow in MP model, what essentially ammends the flow quantities and the two blade row interaction across RS interface.

There is potential effect of downstream blade leading edge to the RS interface. The circumferential variation of fluid flow properties, e.g. total pressure, is thus influenced by the potential effect, Figure 8**Chyba! Nenašiel sa žiaden zdroj odkazov.**. Detail variation of the of the total pressure and computed entropy in short distance from RS interface is shown on Figure 9.

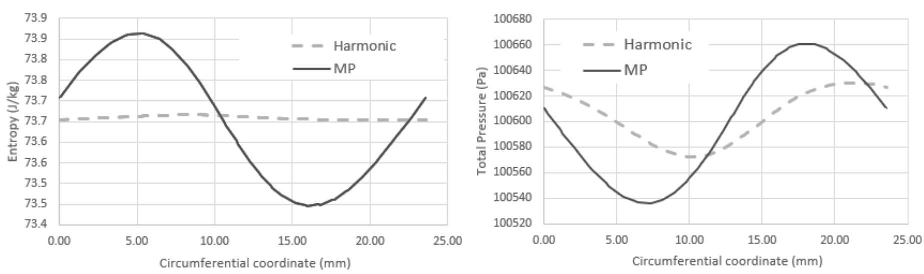

Fig. 8. Total pressure variation across RS interface for MP (left) and time mean flow of harmonic model (right), 50 % span.

The time mean flow prediction of harmonic model results in less upstream effect compared do MP model, as the downstream potential effect is captured in dynamic part of the flow field.

There is, indeed, relatively weak interaction between the rotors as their axial distance, of about length of second stage axial chord, is relatively high, [8]. Despite overestimation of potential effect by steady solution of the MP model, it provides reasonable prediction of the fan global performance.

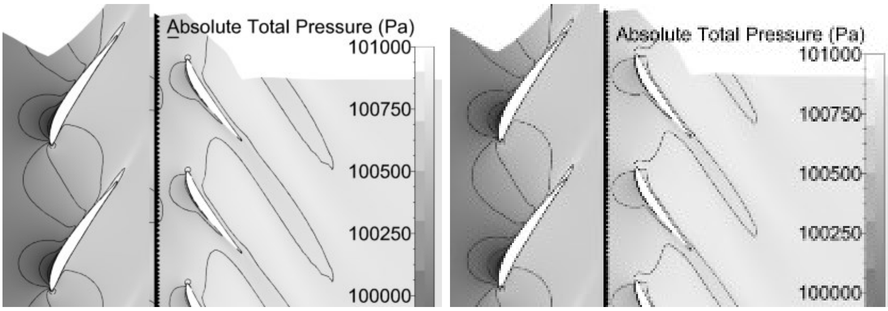

Fig. 9. Potential effect on total pressure variation at downstream side of RS interface, 50 % span.

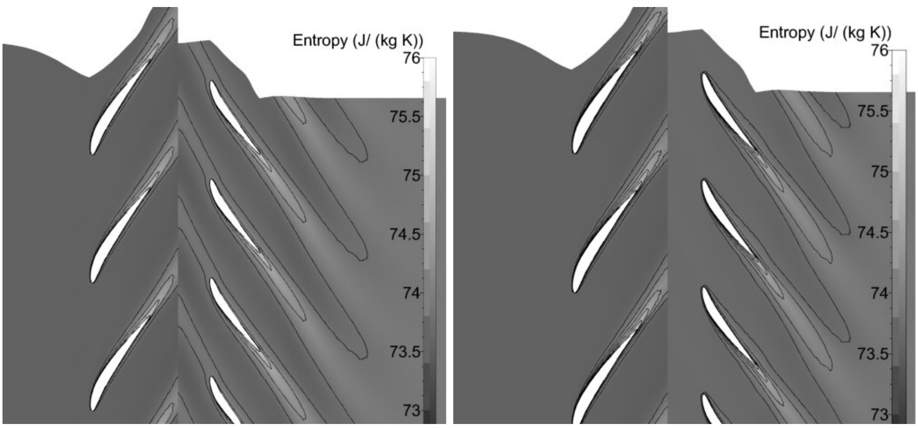

Fig. 10. Entropy prediction at 50 % span of CRF for MP (left) and Harmonic model (right).

The effect of MP is that there is discontinuity of flow field across RS interface and MP might be source of artefacts and fake wakes. The cut wake of first stage and unreal wakes interacting with the second stage blade row could be seen on Figure 10**Chyba! Nenašiel sa žiaden zdroj odkazov.** for the MP model.

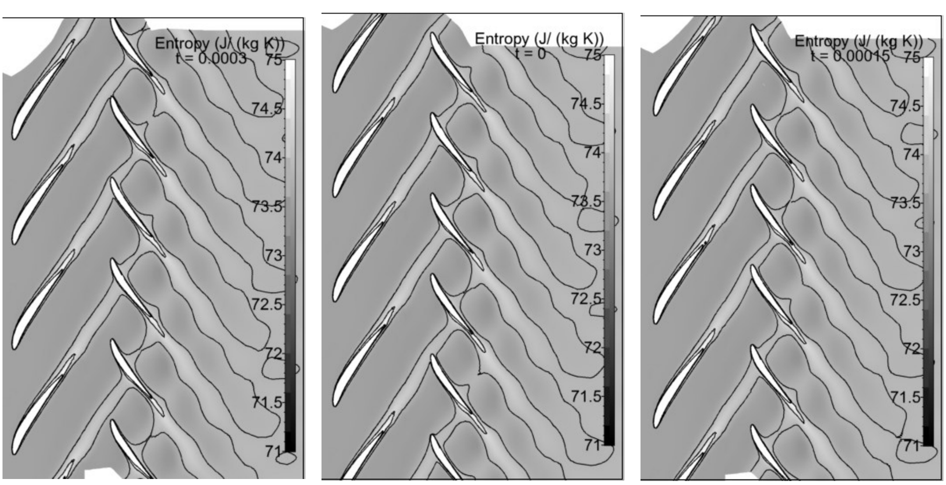

Fig. 11. Continuity of reconstructed flow across harmonic RS model for several time steps.

The harmonic RS model allows for real convection of first stage wake into the second rotor, as could be seen from reconstructed harmonic entropy evolution on Figure 11.

The harmonic RS model is working only with blade passing frequencies and their higher harmonics. It can not capture unsteady effects of other frequency, e.g. rotating stall. The full unsteady sliding mesh model should be used in this case, though at much higher resource costs, due to much larger 360º mesh and small time step. This usually lead to two orders of magnitude longer computational time, [5].

7 Unsteady flow field solution

One of the main effect of harmonic RS model is in improved prediction of aerodynamic performance of time mean flow, when allowing for more realistic periodic unsteady interaction among the blade rows.

The available static pressure frequency spectrum in the harmonic solution could be used for prediction of noise propagation. **Chyba! Nenašiel sa žiaden zdroj odkazov.**

Figure 12 shows locally reconstructed static pressure variation at a control pint in time, based on its pressure frequency spectrum.

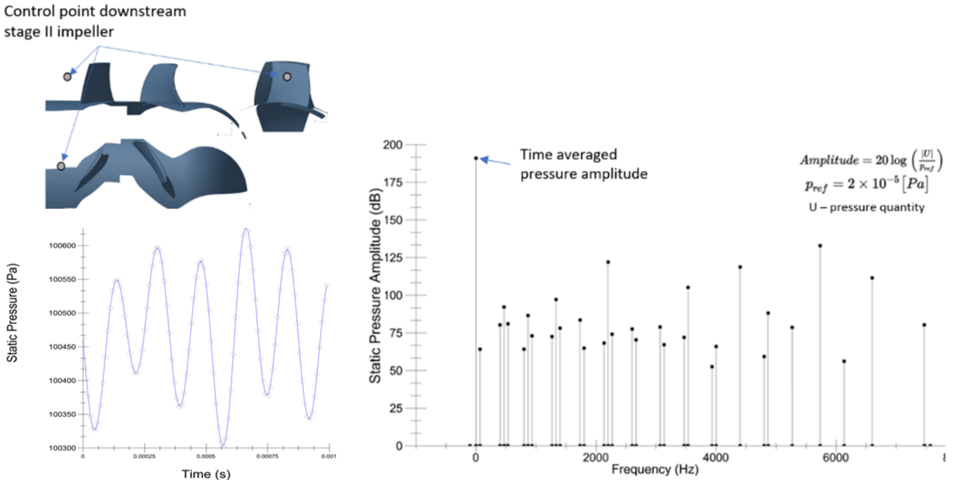

Fig. 12. Static pressure frequency spectrum at a control point.

8 Conclusion

The main purpose of the paper is to discuss differences in numerical flow field prediction by steady state mixing plane and quasi-unsteady harmonic model of rotor-rotor interaction in a contra rotating fan stage.

The global time mean performance of the contra rotating fan was predicted on similar level by the two models, as the relatively high axial distance of the rotors does not induce intensive unsteady interaction. The harmonic model provides more realistic prediction of unsteady flow field and avoids artefacts in time mean flow structure, like cut and artificial wakes, inherent to mixing plane model.

The harmonic model solution, with relatively small computational overhead, compared to the mixing plane model, might be used for aerodynamic noise prediction.

Acknowledgement: This work has been supported by the project KEGA 033ŽU-4/2018 “Heat sources and pollution of the environment” and APVV-17-0311 “Research and development of zero waste

technology for the decomposition and selection of undesirable components from process gas generated by the gasifier".

References

1. R. P. Mueller, O. Velde, C. Friebe, 34th Thermal Measurement, Modeling & Management Symposium (2018)
2. M. Heinrich, Ch. Friebe, R. Schwarze, Proc. IMechE Part A: J. Power Energy, **0**, 1-10, (2016)
3. H. Nouri, F. Ravelet, F. Bakir, C. Sarraf, R. Rey, J. Fluids Engineering **134**, 10 (2012)
4. J. J. Adamczyk, J. Turbomachinery **122,** 189-217 (2000)
5. T. Hildebrandt, P.Thiel, S. Albert, S. Vilmin, ASME Turbo (2014)
6. L. He, W. Ning, AIAA Journal, **36**, 11 (1998)
7. Numeca User Manual Fine^{TM}/Turbo (2019)
8. H. Luan, L. Weng, Y. Luan, PLoS ONE **13**, 7 (2018)