Annals of DAAAM for 2010 & Proceedings of the 21st International DAAAM Symposium, Volume 21, No. 1, ISSN 1726-9679
ISBN 978-3-901509-73-5, Editor B. Katalinic, Published by DAAAM International, Vienna, Austria, EU, 2010
Make Harmony Between Technology and Nature, and Your Mind will Fly Free as a Bird
3rd DAAAM-ICAT

ANALYSIS OF THE APPLICABILITY DEGREE OF MULTIPLE ROTATING REFERENCE FRAMES (MRF) METHOD FOR DESIGN OF AUTOMOTIVE WATER TURBOPUMPS

BALIC, S[enad]; DUHOVNIK, J[oze] & BESLAGIC, E[rnad]

Abstract: *Exact numerical calculations used in the paper [BalDuh04] last long time and are hardware demanding. The Multiple Rotating Reference Frames (MRF) Method eases the need for transient calculations with mowing grid. The method becomes exact if the flow on the interface between subdomains is symmetrical around a common axis of rotation.*

This paper presents degree of applicability of Multiple Reference Frames (MRF) Method for Design of Automotive Water Turbopumps by Numerical Analysis of Entire Flowing Tract, on full model of fluid flow in one turbopump. Special attention was devoted to adapting the model of the turbopumps for the mentioned request for symmetrical flow on the interface between the impeller and the spiral of the pump.

Key words: *automotive engine water turbopump, numerical flow simulation, finite volume method (FVM), multiple rotating reference frames (MRF)*

1. INTRODUCTION

Method of calculation and water turbopump design in the automotive engine circulation cooling system is defined in the paper [BalDuh04]. In this method, exact numerical calculation (based on finite volume method) has been applied on the whole flowing tract model of automotive turbopump, with rotation of impeller (transient calculations with moving grid). This method last long time and are hardware demanding. Because of that, in this paper is used the Multiple Rotating Reference Frames (MRF) Method, wich eases the need for transient calculations with mowing grid.

As an experimental model, based on which virtual models were generated and numerical calculations realized, an automotive turbo pump with an open radial impeller and cylindrical vanes was chosen. The pump is built into motor cooling systems of some types of cargo vehicles from a well-known automotive manufacturer. The meridian section and a drawing of the impeller from the chosen pump are shown in Fig. 1.

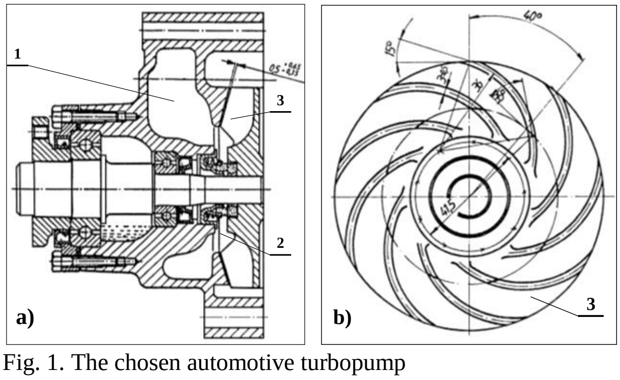

Fig. 1. The chosen automotive turbopump
a) meridian section, b) radial impeller (D_{1m} = 55,5 mm, D_2 = 107 mm, β_1 = 36°, β_2 = 15°)
1. pump collector-feeder, 2. inlet port, 3. impeller

2. MATHEMATICAL AND NUMERICAL METHOD

The water flow in a turbopump flow passage is considered viscous and turbulent.

In [BalDuh04] and [Comet01] the mathematical model of transport processes in a turbopump that can be simulated with some FVM software is presented. It includes the mass, momentum and energy balance equations in integral form, a space conservation law, constitutive relations required for the problem closure, models of turbulence in fluid flow, and boundary conditions. Equations are given for the control volume CV bounded by surface S in the integral form similar for all conserved properties.

Elements of the numerical method: discretization principles, derivation of algebraic equation systems, solution procedure and implementation of boundary conditions are also given in [Comet01].

2.1 Boundary conditions

In the present study, the equations are solved for a special case of flow in full model of the automotive turbopump's entire flowing tract.

The pump flow passage calculation domain is bounded mainly by walls, where no-slip boundary condition was applied. The fluid velocity in the vicinity of the wall was approximated by using wall function [Comet01], which assume logarithmic region in the velocity profile.

Boundary conditions implemented at inlet and outlet for two numerical models (Fig. 4 and 5) are given in Tab. 1 and 2.

3. MODEL OF TURBOPUMP'S ENTIRE FLOWING TRACT

For a realization of 3D MRF numerical analysis calculation procedure, it is necessary to provide that the flow at the interface between the subdomains is symmetric around the axis of rotation.

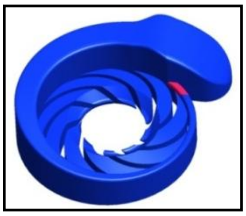

Fig. 2. Modified model of spiral’s tail

Therefore, for selected water turbopump model, spiral tail is extended to the head of the spiral, as shown in Fig. 2. The results showed that this change did not affect on the accuracy of results.

MRF numerical calculations are realised for two numerical models. First model (Fig. 3) is designed without a *collector-*

feeder [BalDuh04]. Second model has a *collector-feeder* (Fig. 1 and 4).

In first numerical model (Fig. 3), pump's inlet is indicated with number 1. In second numerical model (Fig. 4), pump's inlet is the motor inlet 1. In both numerical models, model's outlet is the spiral's passage 3 shown in Fig. 3 and 4.

4. GRID GENERATION

Because of complex geometry of flow parts models trough turbopump, the specific type of grid generation mentioned models was used. The grids of the analyzed automotive model's entire flowing tract, for the analyses with Multiple Reference Frames (MRF) Method, are shown in Fig. 3 and 4.

Numerical grid is created using hexahedron control volumes (CVs), and locally using prism CVs.

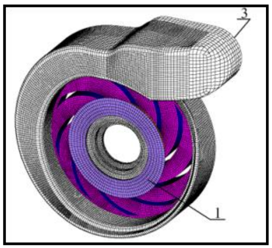

Fig. 3. The mesh of 1st model of turbopump flowing tract
1. pump inlet, 3. pump outlet

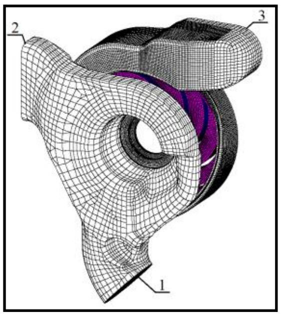

Fig. 4. The mesh of 2nd model of analysed automotive turbopump entire flowing tract
1. motor inlet, 2. radiator inlet, 3. pump outlet

5. RESULT AND DISCUSSION

5.1 3D-calculation of fluid flow

Based on numerical calculation results it’s possible to obtain a presentation of fluid flow’s pressure and velocity distribution, values of all relevant parameters (flow Q, effort H, general effort Δp, efficiency factor η, resulting radial force F_r and torque M_z on fictive impeller), and Q-Δp diagrams.

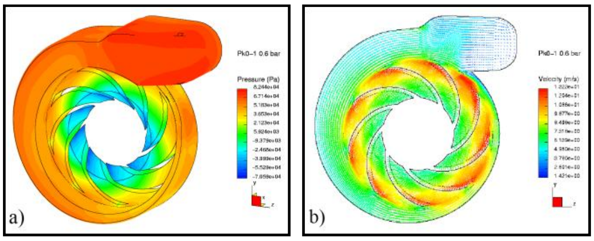

Fig. 5. Distribution of pressure (a) and velocity (b) in one model's part of analyzed turbopump's

Distribution of pressure and velocity in one model's part of analyzed automotive turbopump's entire flowing tract are shown in Fig. 5.

5.2 Comparison of the numerical results for two used numerical methods

The following tables give a comparative review of the results of CFD calculation of three-dimensional models, using the exact calculation procedure with rotation of the fictive impeller (Tab. 1), and with MRF method (Tab. 2).

CFD ANALYSIS WITH ROTATION OF IMPELLER
n=3000 rpm

Model	Boundary cond.		Flow Q (m^3/s)	Torque M_z (Nm)	Useful Power P_u (W)	Power at Shaft P (W)	Effic. factor η
	Motor Inlet p_i (bar)	Outlet p_o (bar)					
Without spiral	0	0,4	0,01210	2,04139			
		0,6	0,01012	2,76708			
		0,8	0,00775	3,11239			
		1,0	0,00382	2,36655			
1st model Fig. 3	0	0,4	0,00856	2,88862	342,28	907,49	0,377
		0,6	0,00765	2,90632	458,97	913,05	0,502
		0,8	0,00657	2,80817	525,60	882,21	0,595
		1,0	0,00488	2,44171	488,46	767,09	0,636
2nd m. Fig. 4	0	0,6	0,00565	2,03816	338,70	640,31	0,529
		0,8	0,00475	1,99823	378,00	627,76	0,602

Tab. 1. The results of CFD with rotation of impeller

CFD ANALYSIS USING MRF CALCULATION PROCEDURE
n=3000 rpm

Model	Boundary cond.		Flow Q (m^3/s)	General effort Δp (bar)	Torque M_z (Nm)	Useful Power P_u (W)	Power at Shaft P (W)	Effic. factor η
	Inlet c_i (m/s)	Outlet p_o (bar)						
Without spiral	5,318	0,4	0,01210					
	4,448	0,6	0,01012					
	3,405	0,8	0,00775					
	1,679	1,0	0,00382					
1st model Fig. 3	3,761	0,4	0,00855	0,5148	3,33875	440,48	1048,90	0,419
	3,362	0,6	0,00765	0,7677	3,44288	587,03	1081,61	0,542
	2,887	0,8	0,00657	0,9582	3,34907	629,30	1052,14	0,598
	2,147	1,0	0,00488	1,1222	2,88733	547,94	907,08	0,604

Tab. 2. The results of CFD analysis using MRF procedure

The presented results show that the MRF method, which requires considerably less time than an exact procedure, represents the best solution for the analysis before the final calculation. The character of the value change of observed parameters is the same for both calculation procedures. Deviations of these values, obtained by an approximate procedure, compared to an exact procedure, not exceeding the limit acceptable for engineering calculations.

6. CONCLUSION

Using the Multiple Rotating Reference Frames (MRF) Method for CFD analysis in flow calculation of entire flowing tract of automotive water turbopumps, in a manner suggested in this paper, can be successfully included in engineer’s calculation procedure and design, because they enable flow simulations with acceptable coincidence with real condition.

7. REFERENCES

Balić, S. & Duhovnik, J. (2004). Design of Centrifugal Turbopumps by Numerical Analysis of Entire Flowing Tract, *Proceedings of 3rd DAAAM International Conference on Advanced Technologies for Developing Countries ATDC’04*, Katalinic, B. and others (Ed.), pp. 3-10, ISBN 953-6114-68-2, Croatia, June 2004., University of Split and DAAAM International Vienna, Split

Comet User Manual and Tutorials, Version 2.00, ICCM Institute of Computational Continuum Mechanics GmbH, Hamburg, 2001

Balić, S. & Duhovnik, J. (2003). Analiza protočnega trakta centrifugalne črpalke DN 200 za agresivne medije, Dr. Duhovnik d.o.o., Ljubljana

Didandeh H.; Toksoy, C.; Cutler, F. and the other (1997), Computer-Aided Design of a Water Pump Impeller for the Chrysler 4.0 Liter 6 Cylinder Engine, *SAE International Congress & Exposition*, pp 82-93, Michigan, February 1997., Detroit