Research Journal of Applied Sciences 9 (10): 629-634, 2014
ISSN: 1815-932X

Centrifugal Kerosene Pump CFD-Modeling

Vasily Zubanov, Leonid Shabliy and Alexandr Krivcov
Department of Aircraft Engine Theory,
Samara State Aerospace University, 443086 Samara, Russia

Abstract: This study presents the CFD-modeling of fuel pump with program ANSYS CFX. The study gives detailed description of CFD Model and techniques for improving its adequacy. Verification with experimental data allows recognize used CFD Model adequate within the experimental error. Identified CFD Model can be used to upgrade pump design for increasing efficiency.

Key words: CFD, turbopump assembly, screw centrifugal pump, calculation model, ANSYS CFX

INTORDUCTION

Powerful kerosene pumps are widely used at aircraft engines. Complexity operation processes of powerful pumps results in complexity of its design and tweaking even with using experience accumulated at research and development companies. Understanding of pumps operation processes allows reducing of the designing complexity and cost of creating new (Bolotov *et al*., 2014) and modifying existing pumps. The 3D Computational Fluid Dynamics (CFD) simulation is good tool for prediction pump performances (Shabliy and Cherniaev, 2014) but difficult to obtain accurate models (Gonzalez *et al*., 2002; Kraeva, 2010; Popov *et al*., 2014). Therefore, in this work special attention is paid to validation (identification) model by experimental data and calculated data obtained verified methods.

A number of examples of such analyses have been described in the literature. Bellary and Samad (2014) optimization was performed to maximize efficiency of centrifugal pump impeller. Results of CFD-modeling were compared with the analytical calculation (difference 9%) but not carried out a comparison with experiment and increased efficiency by 2.32% can easily be within the error. Simulations had used commercial CFD code ANSYS CFX Version 13.0 with TurboGrid for meshing and BladeGen for geometrical modeling. Limbach *et al*. (2014) modeling was carried out also with ANSYS CFX tool but main aim was in determining the proper way of modeling. So was shown small differences in integral results between steady and transient modeling. Also, two-phase modeling of cavitating flow gives model precision increasing only for cavitation pump regimes. Results of CFD-modeling differs from the experimental data: head-capacity by >17%, efficiency of pump >15%.

This short review allows determining three main distinctive features of existing approaches for pump CFD-modeling. First, the comparison with experimental data necessary. Second, as researchers of the above mentioned works, the problem can be solved using CFD commercial tools. Third, rational techniques for decreasing of calculation time are using adequate minimum of models for specific pump regime and using appropriate set of meshing and modeling tools whatever program vendors.

In this research has been performed CFD-modeling of pump flow for detail disquisition of powerful kerosene pump workflow. Especially, attention is paid to validation (identification) of CFD Model with experimental data and calculation results from other verified methods.

In general, powerful kerosene pump consists of three parts:

- Screw centrifugal pre-pump (low pressure stage): LPS: Low Pressure Screw passage; LPI: Low Pressure Impeller passage; TP: Transferring Passage
- Screw centrifugal high pressure pump (stage): HPS: High Pressure Screw passage; HPI: High Pressure Impeller passage
- Booster centrifugal pump

This study has been performed CFD-modeling of all parts of powerful kerosene pump except booster (Fig. 1).

Corresponding Author: Vasily Zubanov, Department of Aircraft Engine Theory, Samara State Aerospace University, 443086 Samara, Russia

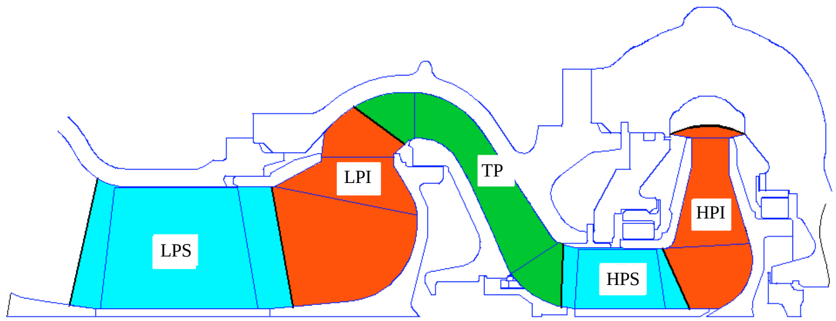

Fig. 1: Meridional section of simulated pump

Table 1: Quality metrics of structured hexagonal mesh for bladed units and of unstructured tetrahedral mesh with prism layer of Inlet and outlet ducts

Quality metrics	LPS	LPI	TP	HPS	HPI	Inlet duct	Outlet duct
Elements count ($\times10^6$)	0.67	0.51	0.40	0.61	0.62	0.67	5.00
Min. Skewness (Min. Angle) (°)	10.10	21.40	36.30	12.90	9.97	10.1	10.8
Aspect ratio							
Max. ($\times10^3$)	10.30	4.05	4.45	4.49	4.12	1000	1000
Less 1000 (% elements)	93.00	98.00	96.00	96.00	96.00	-	
Less 50 (% elements)	-	78.00	60.00				
Expansion ratio							
Max.	15.10	3.20	3.30	16.10	3.70	7	204
Less 1.8 (% elements)	90.00	97.00	96.00	88.00	97.00	83	68

METHODOLOGY

As mentioned earlier, obtaining accurate CFD Model is due to its identity by comparing the results with experimental or calculated ones which received by verified techniques.

In this study to identify the workflow model of pump used two types of data:

- Experimental results of spilling using water as the working fluid instead of kerosene
- The results of a one-dimensional analytical calculation with working body kerosene

The main objective of the work is to create a CFD Model of pump which giving results fully complies with both the above data.

MESH MODEL

Mesh creation for bladed units (rotors and stator transition duct) has been performed using program NUMECA AutoGrid5. For calculation simplification was used sector models flow around only one blade or screw (Shabliy *et al*., 2012). For CAD Model simplification method (Shabliy and Dmitrieva, 2006) has been used. This method allows structural meshes for blade rows with first element side size 1 micron.

For non-bladed units (inlet and outlet ducts) meshing has been performed using program ANSYS ICEM CFD. Due to non-periodicity of these units were used full models. Complexity of outlet spiral duct geometry especially in the throat of snail results in some worse grid quality. Nevertheless, problems for solving are not caused. Quality metrics for meshes are shown in Table 1.

WATER TEST BOUNDARY CONDITIONS

CFD-modeling was implemented using program ANSYS CFX with water because experimental data was only for this working fluid. There are input data for pump modeling based on experiment data:

- Low pressure rotor speed 3620 rpm, high-pressure rotor speed 13300 rpm
- Inlet total pressure 0.4 MPa
- Inlet water temperature 293 K
- Standard water properties: density 997 kg/m^3, the molar mass 18 kg kmol^{-1}, isobaric heat capacity of 4182 J kg.K^{-1}, dynamic viscosity 0.00089 kg (m.sec^{-1}), thermal conductivity 0.607 W mK^{-1}

METHOD FOR DETERMINING THE PUMP CHARACTERISTIC

As been pointed above experimental data presents so-called pressure (head)-characteristic. And for validation we need same head-characteristic consisted of several CFD-points. For obtain results for each calculation point can be used ANSYS CFD Post in GUI-mode but faster and more convenient way using CFD-Post in batch mode with extracting values of predefined mathematical expressions in the CEL-language (CFX Expression language). Also, using expressions helps to obtain integral parameters estimating convergence of solution. For example, consumed power formula:

$$N_{consumed} = M_{ALL}.\omega \quad (1)$$

Where:

M_{ALL}[N.M] = Drive shaft torque
$\omega[c^{-1}]$ = Rotor angular velocity

Drive shaft torque is sum of torques of all blades of each rotor. For example, for the discussed pump CEL-expression for consumed power is:

Power consumed = (torque_z()@rk_bd Default×12+
torque_z()@rk_md Default×15+
torque_z()@shn_bd Default×5+
torque_z()@shn_md Default×3)×
abs(n)/1 [rad]

CFD-MODELING OF WATER TEST

Additional data required for the simulation are:

- Output static pressure set: from 15 up to 21 MPa in steps of 0.5 MPa
- Inlet turbulence intensity 10%
- Processes are steady-state
- Turbulence model k-ε
- "Stage" interface type with velocity averaging at all rotor-stator pairs; rotational periodicity for bladed units; no vibration
- No surface roughness (smooth wall)
- No cavitation

Solution convergence was estimated by minimization of mathematical discrepancies and constancy of integral parameters: overall mass flow, total head (pressure rise), consumed pump power.

To reduce computer calculation time during simulation process have been used specific methods: stepwise changing of timescale factor and stepwise changing of boundary conditions.

COMPARISON WITH EXPERIMENTAL DATA IMPROVING CFD-MODEL ADEQUACY

At the first stage, model represents only the hydrodynamic path of pump, inlet and outlet were in place of their geometrical position (Fig. 2a). Analysis of the calculation showed a change calculation parameters from iteration to iteration, the solver also limited outlet area. In this case, the mathematical residuals were unacceptably high. The reason for this was the use uniform flow parameters at the inlet and at the outlet which in reality are pretty uneven.

An attempt was made to get rid of this error by creating additional cylindrical nozzle at the inlet and outlet of the pump and calculated the modified model (Fig. 2b).

Proof of the correctness of the decision confirmed found vortex flow in the diffuser (Fig. 3) which does not

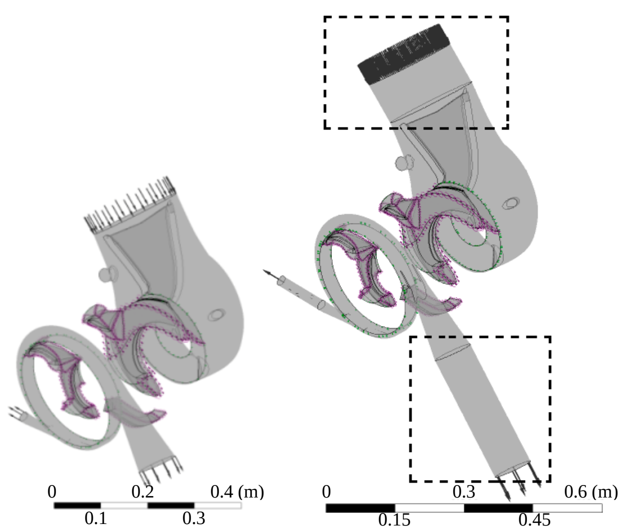

Fig. 2: Improving pump model: a) first model and b)second model

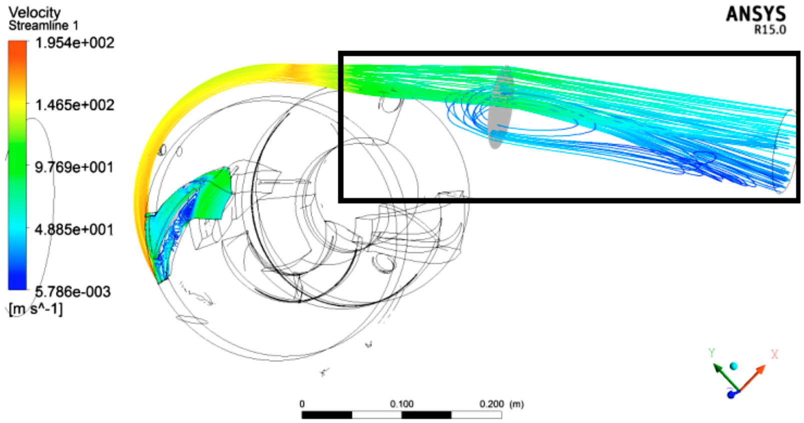

Fig. 3: Vortex in cone diffuser

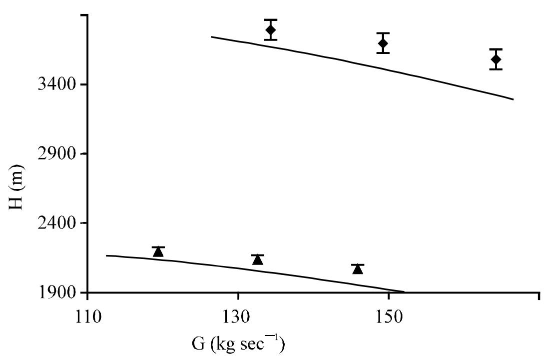

Fig. 4: Mass flow performance comparison

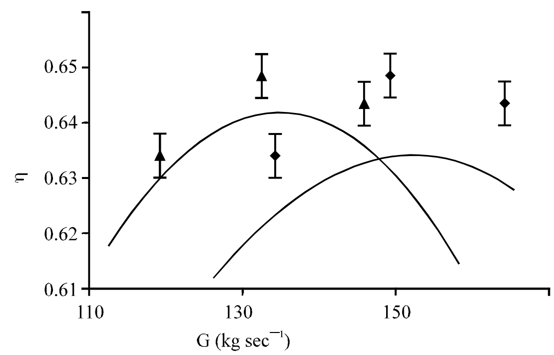

Fig. 5: Efficiency performance comparison

allow to consider the output parameters uniform. Using boundary type "open" instead "static" in the first model allowed to obtain the same flow pattern and parameters but data for the modernization of design becomes extremely small.

Results of CFD-modeling was compared with experimental data and are consistent with ones within the experimental error: head-capacity curve (Fig. 4) coincides with an accuracy of 5.2%, coefficient efficiency curve (Fig. 5) 1.3%. Experimental data of water are represented bytriangular points, CFD-data by curve of the circular points.

Assessing the impact of types of boundaries on the results of numerical simulations showed little dependence within the error.

Thus, the verification of powerful screw centrifugal kerosene pump was conducted in ANSYS CFX on the water component. Comparison of simulation results with experimental data allows to recognize used CFD Model adequate within the experimental error. CFD Model can be used for further research of this type of pumps.

CALCULATION WITH KEROSENE

Using adequate CFD Model calculation was performed on kerosene component with the modified data:

- Low pressure rotor speed 4750 rpm, high-pressure rotor speed 17500 rpm
- Temperature of kerosene at the pump inlet 258 K
- Kerosene properties: density 855 kg/m^3, molar mass 167.3 kg kmol^{-1}, isobaric heat capacity 1880 J/(kg.K), dynamic viscosity 1.5.10^{-3} kg (m.sec^{-1}), thermal conductivity 0.14 W/(m.K)

Data for comparison were obtained by re-calculation of the experimental data on the basis of hydrodynamic similarity of kerosene and water.

Results of CFD-modeling was compared with experimental data and are consistent with ones within the experimental error: head-capacity curve (Fig. 4) coincides with an accuracy of 6.9%, coefficient efficiency curve (Fig. 5) 2.3%. Experimental data are represented by rhomboids points, CFD-data by dashed curve. Vortex in cone diffuser was present at all operation modes.

Table 2: Distribution of the total pressure, static pressure and absolute velocity in the analytical calculation and CFD-modeling

	Total pressure (MPa)			Static pressure (MPa)			Absolute velocity (m sec^{-1})		
Boundaries	1D	CFD 3D	Δ (%)	1D	CFD 3D	Δ (%)	1D	CFD 3D	Δ (%)
Inlet	0.405	0.405	0.05	0.392	0.389	0.96	5.41	5.99	10.67
Inlet duct/LPS	0.379	0.373	1.65	0.360	0.350	2.86	6.67	7.90	18.38
LPS/LPI	0.782	0.767	1.95	0.686	0.653	4.83	14.95	16.69	11.67
LPI/TP	1.530	1.730	13.43	1.130	1.310	16.21	30.60	30.85	0.83
TP/HPS	1.420	1.360	3.63	1.310	1.170	10.87	15.90	26.18	64.65
HPS/HPI	4.500	4.450	0.97	3.910	3.540	9.33	36.93	44.46	20.40
HPI/Outlet duct	34.630	34.000	1.81	19.640	20.020	1.95	188.40	182.27	3.25
Outlet	30.940	29.970	3.15	30.560	28.670	6.18	29.30	54.01	84.35

Nominal mode corresponds to the flow at the outlet equal 145.4 kg sec^{-1}. Results of CFD-modeling at nominal mode were compared with analytical (Matveev *et al*., 2006) by the parameters of total and static pressure, the absolute velocity. The results of compare are shown in Table 2. Due to differences in location interfaces of CFD Models and analytical calculation the parameters at the boundaries were averaged.

Allowable error between CFD and analytical calculation of 10% (Gulich, 2010). Analysis of data from Table 2 shows a significant difference in the parameters CFD-modeling and analytical calculation on the borders of the transferring passage. Likely, this is due to the inability to correctly analytically identify the parameters on such complex curved geometry. In general, the analytical calculation is well suited for 1D-modeling powerful screw centrifugal kerosene pump.

CONCLUSION

The powerful screw centrifugal kerosene pump has been simulated successfully using CFD-program ANSYS CFX. The verification of CFD Model of pump was conducted on the water component. The results of verification allow to recognize the CFD Model adequate within the experimental error. CFD-modeling on component kerosene was performed, results agree well with the analytical calculation which suitable for 1D-modeling of pump.

To further improve the adequacy of the model is necessary to assess the effect of grid quality and turbulence models on the accuracy of the numerical method of hydraulic parameters (Zubanov *et al*., 2012).

This adequate CFD Model will be used to upgrade the design of the pump in order to increase its efficiency and predict loads to rotor, bearings (Novikov, 2014), discharge and damper devices (Falaleev and Vinogradov, 2006; Falaleev *et al*., 2014). CFD Model can also be used for further research workflow in the screw and/or centrifugal pumps kerosene to reduce the amount of development tests.

ACKNOWLEDGEMENTS

This research was financially supported by the Government of the Russian Federation (Ministry of education and science) based on the Government of the Russian Federation Decree of 09.04.2010 No. 218 (theme code 2013-218-04-4777).

REFERENCES

Bellary, S.A.I. and A. Samad, 2014. Improvement of efficiency by design optimization of a centrifugal pump impeller. Proceedings of the ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, June 16-20, 2014, Dusseldorf, Germany.

Bolotov, M.A., E.A. Kapenkina, N.V. Rusanov and V.A. Pechenin, 2014. Choice of production measuring instruments based on techno-economic analysis, taking into account the type I error and type II error. ARPN J. Eng. Applied Sci., 9: 1834-1841.

Falaleev, S. and A. Vinogradov, 2006. Concept of combined gas-dynamic mechanical seal and discharge device of aircraft engine rotor support. ARPN J. Eng. Applied Sci., 9: 1842-1848.

Falaleev, S.V., K.N. Chaadaev and D.S. Diligenskiy, 2014. Selection of the hydrodynamic damper type for the turbomachine rotor. Life Sci. J., 11: 502-505.

Gonzalez, J., J. Fernandez, E. Blanco and C. Santolaria, 2002. Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump. J. Fluids Eng., 124: 348-355.

Gulich, J.F., 2010. Centrifugal Pumps. 2nd Edn., Springer, New York.

Kraeva, E.M., 2010. Calculation of energy parameters in high-speed centrifugal pumps of low specific speed. Russian Aeronautics, 53: 73-76.

Limbach, P., M. Kimoto, C. Deimel and R. Skoda, 2014. Numerical 3D simulation of the cavitating flow in a centrifugal pump with low specific speed and evaluation of the suction head. Proceedings of the ASME Turbo Expo 2014: Turbine Technical Conference and Exposition, June 16-20, 2014, Dusseldorf, Germany.

Matveev, V.N., N.F. Musatkin and V.M. Rad'ko, 2006. Design of Screw Centrifugal Pump, Tutorial. SSAU Publication, Samara, Pages: 64.

Novikov, D.K., 2014. Development of the unified range of friction couples of gas-dynamic face seals. Life Sci. J., 11: 354-356.

Popov, G.M., A.O. Shklovets, A.I. Ermakov and D.A. Kolmakova, 2014. Methods to reduce the resonant stresses level of gas turbine engines compressor rotor wheels. Proceedings of the 4th International Conference on Simulation and Modeling Methodologies, Technologies and Applications, August 28-30, 2014, Vienna, Austria, pp: 619-624.

Shabliy, L. and A. Cherniaev, 2014. Optimization of gas turbine compressor blade parameters for gas-dynamic efficiency under strength constraints. Proceedings of the 4th International Conference on Simulation and Modeling Methodologies, Technologies and Applications, August 28-30, 2014, Vienna, Austria, pp: 523-528.

Shabliy, L.S. and I.B. Dmitrieva, 2006. Conversion of the blade geometrical data from points cloud to the parametric format for optimization problems. ARPN J. Eng. Applied Sci., 9: 1849-1853.

Shabliy, L.S., G.M. Popov and D.A. Kolmakova, 2012. Approaches to parametric models of blade machines formation. Vestnik SSAU, 3: 192-196.

Zubanov, V.M., A.V. Krivcov and A.A. Shtraub, 2012. The parameters of turbulence models and grids investigation on CFD results of plane turbine cascade. Vestnik SSAU, 2: 185-191.