Improved Model for Meanline Analysis of Centrifugal Compressors with a Large Tip Clearance

Andrey Sherbina[1], Ivan Klimov[2] and Leonid Moroz[3]
SoftInWay Inc., 1500 District Avenue, Burlington, MA, 01803, USA

For small fast centrifugal wheels, the relative tip clearance can be large even for tenths of millimeters absolute clearance. Some of these impellers inherently have large relative clearance due to boundary conditions, geometry limitations, and mechanical design of the compressor. In the presented work, a centrifugal impeller large tip clearance model (LTCM) is developed. LTCM improves existing state-of-the-art compressor loss models by effecting flow kinematics and introducing impeller discharge recirculation. LTCM model is validated against experimental results and has good agreement with 3D CFD results.

Nomenclature

U – blade speed
Cu – flow absolute tangential velocity
Cm – flow meridional velocity
r – radius
P – pressure
ρ – fluid density
t – blade pitch
γ – streamline slope angle with respect to axis of rotation
β – blade angle with respect to tangent
δ – tip clearance size
b – hub-to-shroud passage width
σ – slip factor, $Cu2/Cu2\infty$
d – diameter
m – mass flow
N – rotational speed

Subscripts

0 – basic or reference value
$1t$ – impeller inlet tip
2 – impeller outlet condition
$CORR$ – corrected parameter
BF – back flow
rec – recirculation
∞ - condition for infinite number of blades

[1] Senior Aerodynamic Engineer, A.Sherbina@softinway.com
[2] Application Engineer, Ivan.Klimov@softinway.com
[3] President and CEO, L.Moroz@SoftInWay.com.

1. Introduction

The subject of this paper is the flow in high-efficient open centrifugal impellers. The state-of-the art models for centrifugal compressor impellers account for tip clearance as a hydraulic loss caused by clearance flows. Some of these impellers inherently have a large relative clearance due to boundary conditions, geometry limitations, and mechanical design of the compressor. The increased leakage across the blade tip due to large clearance at the impeller outlet results in additional flow deviation and flow recirculation. The aim of this work is to investigate the 3D flow particularities at large tip clearance in the impeller and correct the simulating models for 1D analysis. Improved loss model accounts for these two phenomena and forms the impeller large tip clearance model (LTCM). In the presented work, a centrifugal impeller large tip clearance model (LTCM) is developed, validated against experimental results and has good agreement with 3D CFD results.

2. Technical background

Tip clearance is a source of loss. Typically, compressor designers minimize tip clearance and losses associated with clearance flow from pressure to suction side of the impeller. Impeller clearance is selected based on mechanical design, structural analysis and dynamic response of the rotor-bearing system.

Relative tip clearance is the ratio of absolute tip clearance to blade exit height. For small fast centrifugal wheels, the relative tip clearance can be large even for tenths of millimeters absolute clearance. Depending on bearing selection and axial forces, safe operational clearance can vary. For example, high speed and small rotor weight of a centrifugal compressor predetermines the selection of gas dynamic bearings. The gas dynamic bearing clearance is greater than the clearance of thrust axial-radial ball bearings or oil lubricated plain bearings. The later bearings have some limitations. Thus, the utilization of gas bearings becomes preferable at high rotational speeds. The model is not limited to such rotor system – it's where the author discovered such physical phenomenon.

Various sources revealed that most of the current published work on this topic is devoted to experimental research [5, 7] or 3D CFD analysis [4, 6]. The state-of-the-art models provide the turbomachinery community with a tip clearance calculating model for 1D analysis which only takes into account hydraulic losses caused by leakage flow across the blade tip [2]. Such a model type is applicable with relatively small tip clearance – less than 0.11 – 0.12 according to authors' estimation – when almost whole flow is under influence from the blade. The leakage in the relatively small tip clearance does not distort flow dramatically and the classic 1D approach with an advanced loss model system [2] can be successfully applied for design and analysis of the centrifugal compressors. It is clear from general consideration that the flow scarcely can be assumed as uniform and as a result, can't be treated with conventional 1D methods when the tip clearance is relatively high. The results shown below confirm this conclusion.

3. Experimental results that gave grounds to develop the model

3.1. Compressor flow path description and approach

The subject of this investigation is a centrifugal compressor which consists of centrifugal impeller, vaneless space, vaned diffuser and volute (Figure 1). The exact value of the tip clearance on the impeller was imposed before beginning the design procedure. This constraint resulted in large relative tip clearance being equal to 0.215 (21.5%). The comparison of experiment and meanline analysis using state-of-the-art loss models is presented in Figure 2.

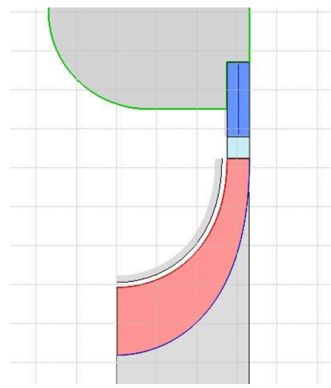

Figure 1 - Compressor flowpath meridional view

All iterations of the present study are accomplished utilizing the commercial software platform, AxSTREAM®, for turbomachinery design, analysis and optimization [1]. Although many loss models exist in the industry, Aungier's loss mechanisms are utilized here for the purposes of modeling losses in the impeller and diffuser [2]. The utilized deviation model for the impeller is the Wiesner formula with slip factor defined by Aungier [2]. The Howell deviation model is adopted for the vaned diffuser with radial flow direction. Volute losses are defined via Ris formulation [3]. All 1D calculation models are summarized in Table 1.

Table 1 Loss models

Flowpath component	Loss model	Deviation model
Centrifugal impeller	Aungier	Wiesner
Radial diffuser	Aungier	Howell
Volute	Ris	N/A

The compressor performance is shown in non-dimensional coordinates. Mass flow is normalized to the flow at best efficiency point on maximum speed line (eqn. 1). Rotational speed is normalized to maximum rotational speed (eqn. 2).

$$n\_norm = N / Nmax \quad (1)$$

$$m\_norm = m / m(Nmax, BEP) \quad (2)$$

where n_norm = normalized rotational speed;
m_norm = normalized mass flow rate;
Nmax = maximal rotational speed at which the compressor was tested;
BEP = best efficiency point.

3.2. Comparison of experimental and initial 1D analysis results

The author designed a compressor utilizing the approach described in paragraph 3.1. The designed compressor was built and tested. Experimental data was obtained and consisted of integral characteristic for the compressor performance: pressure ratio versus mass flow rate and efficiency versus mass flow rate for four speedlines. The test efficiency is the hydraulic total-to-total calculated for measured suction and discharge thermodynamic conditions. This data is normalized using eqn. 1 and eqn. 2.

The performance comparison is shown in the Figure 2. The simulated speedline is truncated intentionally to emphasize the difference in compressor work input (pressure ratio) rather than stall and surge margins. The pressure ratio is simulated with state-of-the-art models and is much higher than in the test data. 1D analysis overpredicts the pressure ratio by up to 11% at maximum rotational speed. Compressor efficiency is also largely overpredicted utilizing models described in Paragraph 3.1. At maximum speed n_norm = 1.0, the total-to-total efficiency discrepancy between model and experiment is about 8 % whereas at slower speeds, this difference reduces to 5 % (Figure 3). The experimental and model shape of the efficiency curve have the same trend (Figure 3).

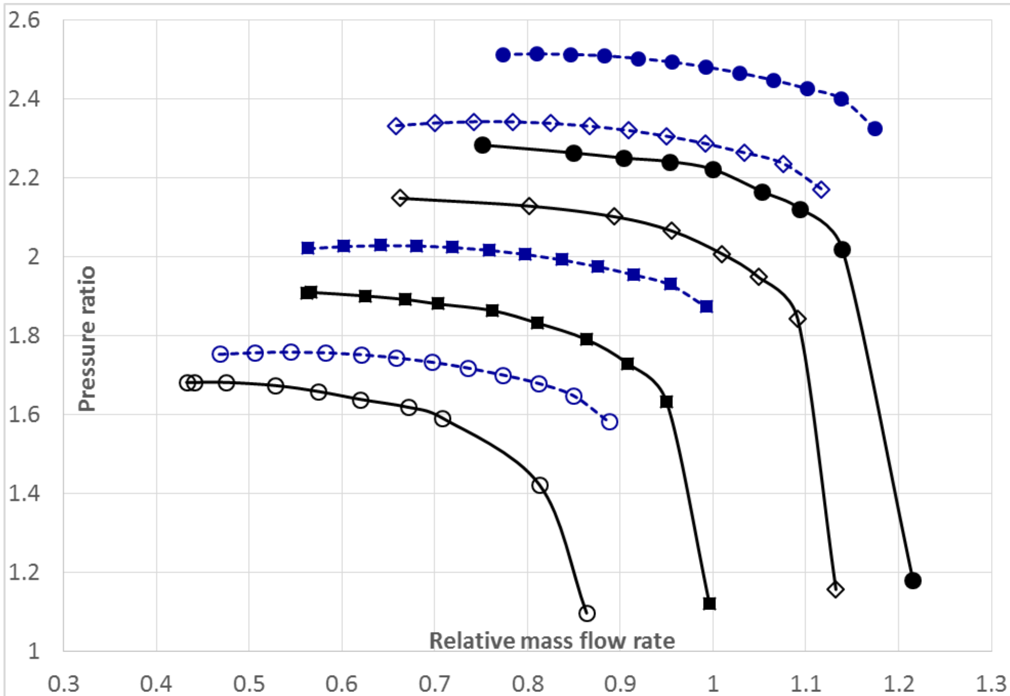

Figure 2 –Initial model simulation versus experiment. Pressure ratio versus normalized mass flow characteristic comparison.
(Performance: ── experiment, - - - 1D meanline; normalized speed: ● 1.0, ◇ 0.95, ■ 0.86, ○ 0.76)

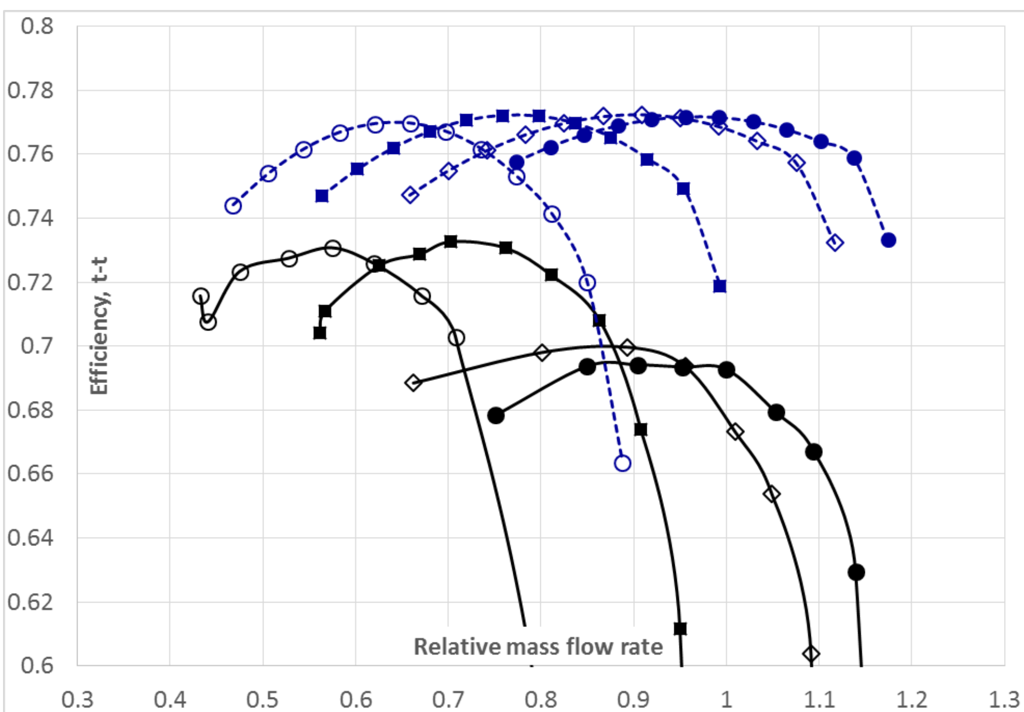

Figure 3 - Initial model simulation versus experiment. Total-to-total hydraulic efficiency ratio versus normalized mass flow characteristic comparison.
(Performance: ── experiment, - - - 1D meanline; normalized speed: ● 1.0, ◇ 0.95, ■ 0.86, ○ 0.76)

4. 3D flow structure analysis and grounds to model development

Aungier loss mechanisms and calculation models are fit for a wide range of compressors and the author has confidence in utilizing them. However, in the case of large tip clearance on the impeller, this loss model system gives significant discrepancy compared to test data as shown in Figure 2 and Figure 3. To better understand flow physics in the compressor wheel with such an abnormal tip clearance, the 3D CFD analysis was performed for the whole compressor stage.

4.1. CFD settings

To find the deficiency of 1D model system, Reynolds Averaged Navier-Stokes equations (RANS) 3D CFD code is employed to deal with compressible fluids. The K-epsilon turbulence model was chosen in this calculation. The computational grid was generated for the single impeller and vaned diffuser passage by using the structured H-type grid. Volute mesh type – tetra in full volume and prismatic boundary layer. Number of mesh elements for impeller – $1.64*10^6$, vaned diffuser – $1.17*10^6$ and volute –$1.17*10^6$. Average y+ was kept near a value of 4.06, 3.72 and 5.84 for the impeller, vaned diffuser and volute, respectively. This mesh is acceptable for the turbulence model being used. As the boundary conditions, the total pressure and the total temperature were applied uniformly at the inlet boundary. Meanwhile, the mass flow was specified at the volute outlet. Since the flow in the near clearance region was the main subject of consideration, special attention was given to the mesh in this zone. The mesh of quite fine quality in blade tip clearance is shown in Figure 4.

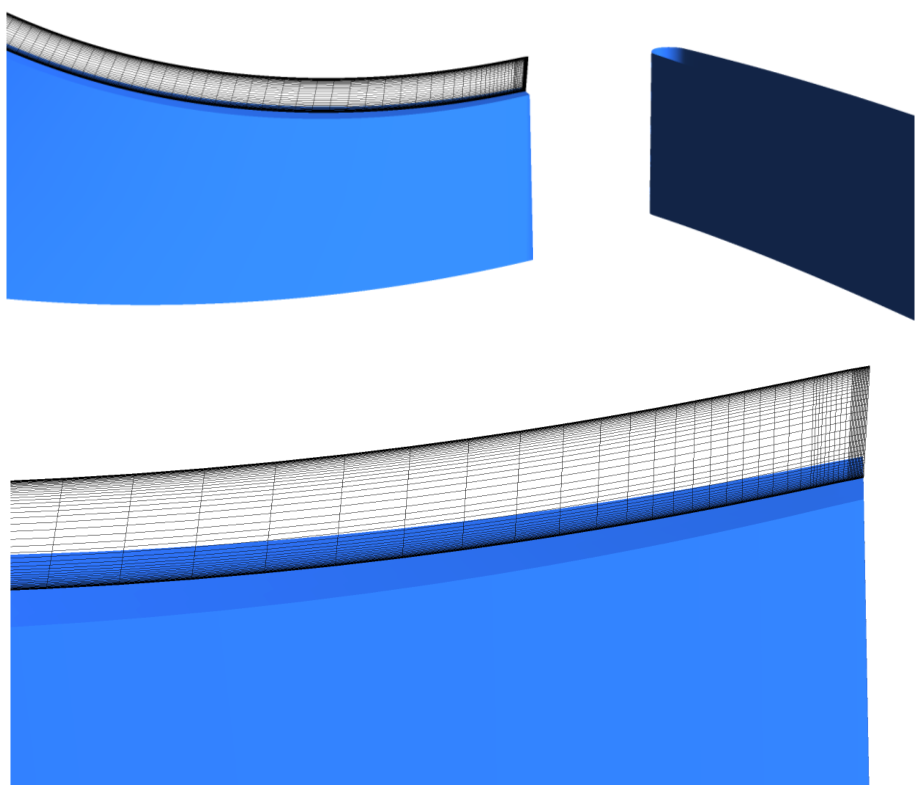

Figure 4 – Mesh in blade tip clearance

4.2. 3D flow analysis

The results of the 3D CFD simulation are presented below which demonstrates the particularities of the flow pattern near the tip clearance at the impeller outlet. One can observe on the 3D view the back flow zone which occupies the significant part of the impeller discharge area (Figure 5) and recirculation at the impeller discharge in the meridional plane (Figure 6).

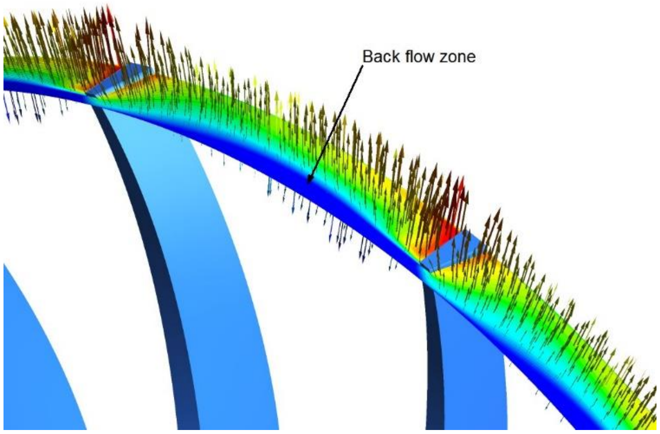

Figure 5 - Backflow zone at the impeller discharge

The main conclusions which can be made drawn on 3D flow study are as follows. When relative tip clearance is very large, δ / b2 > 0.11 – 0.12, flow is featured by the increased leakage across the blade in clearance and the back flow in meridional direction near shroud (Figure 6). The increased leakage across the blade is due to large clearance at the impeller outlet which leads to additional flow deviation from direction of blade trailing edge. It can be said that part of the blade near tip loses deflecting ability to some degree. The second phenomenon is flow recirculation in meridional direction and additional work input which results in the additional enthalpy and entropy raise.

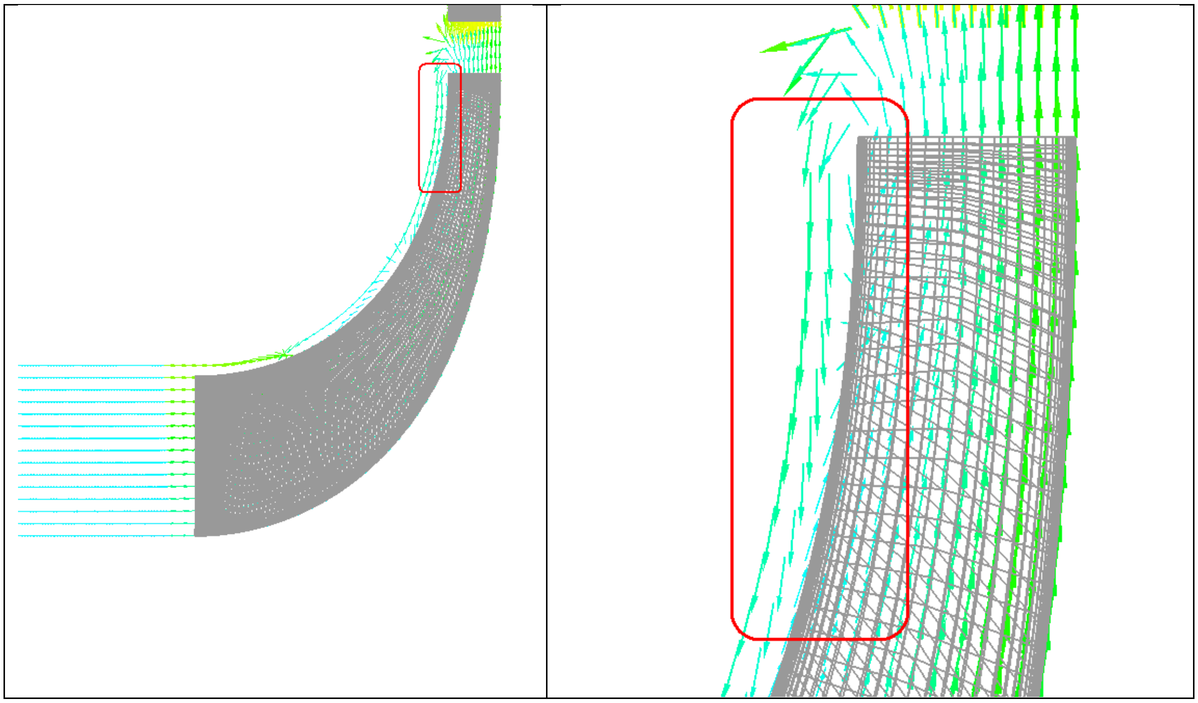

Figure 6 – Velocity vector field in meridional view: left – impeller; right - impeller discharge

Upon completion of the 3D CFD analysis and post-processing of the results, it was concluded that the work input into compressor was overpredicted. The strong leakage across blade tip and large back flow zones cause significant deviation from Wiesner model and additional recirculation work input. With this in mind, it was decided to conceptualize this phenomenon into a general model applicable for 1D analysis.

5. LTCM model development

5.1. The need to develop LTCM

The existing models account for the loss in tip clearance. The initial simulation results show existing models overpredict the discharge pressure. From here, detailed CFD analysis is suggested to modify existing loss models by correcting the slip and introducing discharge recirculation due to large tip clearance.

5.2. Deviation / Slip factor effects

Momentum equation in circumferential direction is the basic governing law:

$$\frac{Cm}{U}\frac{d(U\,Cu)}{dr} = \frac{dP}{\rho t} \tag{3}$$

After transformation, the expression for pressure gradient at impeller discharge can be written as follows:

$$\frac{\Delta P}{\rho_2 t_2} = \frac{Cm}{r_2}\left(2U_2 - \frac{Cm_2}{tg\beta_2}\right) sin\gamma_2 \tag{4}$$

Velocity of leakage flow in clearance [2]:

$$U_{CL} = 0.816\sqrt{\frac{2\Delta P}{\rho_2}} \tag{5}$$

Now, let's introduce an additional slip velocity component due to flow leakage in large clearance:

$$\Delta Cu = a\,U_{CL}\left(\frac{\delta}{b_2} - \bar{\delta}_0\right) \tag{6}$$

where a – empirical coefficient;

$\bar{\delta}_0$ – relative tip clearance below which leakage in the clearance does not deform main flow significantly and additional deviation does not appear.

Correction for slip factor is expressed via eqn. 7:

$$\Delta\sigma = \frac{\Delta Cu}{Cu_{2\infty}} \tag{7}$$

Corrected slip factor

$$\sigma_{CORR} = \sigma_0 - \Delta\sigma \tag{8}$$

where σ_0 – value defined by basic slip factor model.

5.3. Recirculation effects

Averaged meridional velocity of the back flow:

$$C_{BF} = \sqrt{2\,\Delta Cu\,U_2} \tag{9}$$

Back mass flow:

$$m_{BF} = C_{BF}\,\rho_2\,\pi\,d_2\,\delta \tag{10}$$

Recirculation heat input

$$H_{rec2} = f\, U_2\, \Delta Cu\, \frac{m_{BF}}{m} \tag{11}$$

where f – correction factor which depends on diameter ratio d_2/d_{1t};
m – mass flow through the impeller.

It is necessary to give some explanation concerning correction factor f. As it was shown above, the flow pattern can be divided into two zones: main flow, which is featured by work input from the blades, and back flow near the shroud. It is obvious that part of the fluid with high enthalpy from main flow drops down near the shroud region to a lower radius. Then, enthalpy is transferred to this part of recirculating flow repeatedly on the way from the lower radius to impeller outlet. Therefore, the following assumption can be made: heat (enthalpy) input due to flow recirculation is proportional to some expression of radius/diameter ratio d_2/d_{1t}.

6. Model validation against experimental results

Compressor performance map simulated with large tip clearance model (LTCM) and experimental curves are compared on Figure 7. It can be noted that the new model simulation results correlate well with the experiment. The two speedlines simulated with CFD (Fig.7 and 8) confirm that loss model system complemented with LTCM results in the correct trend of performance curves. Also, AxSTREAM® 1D results better predict choke margin than 3D CFD.

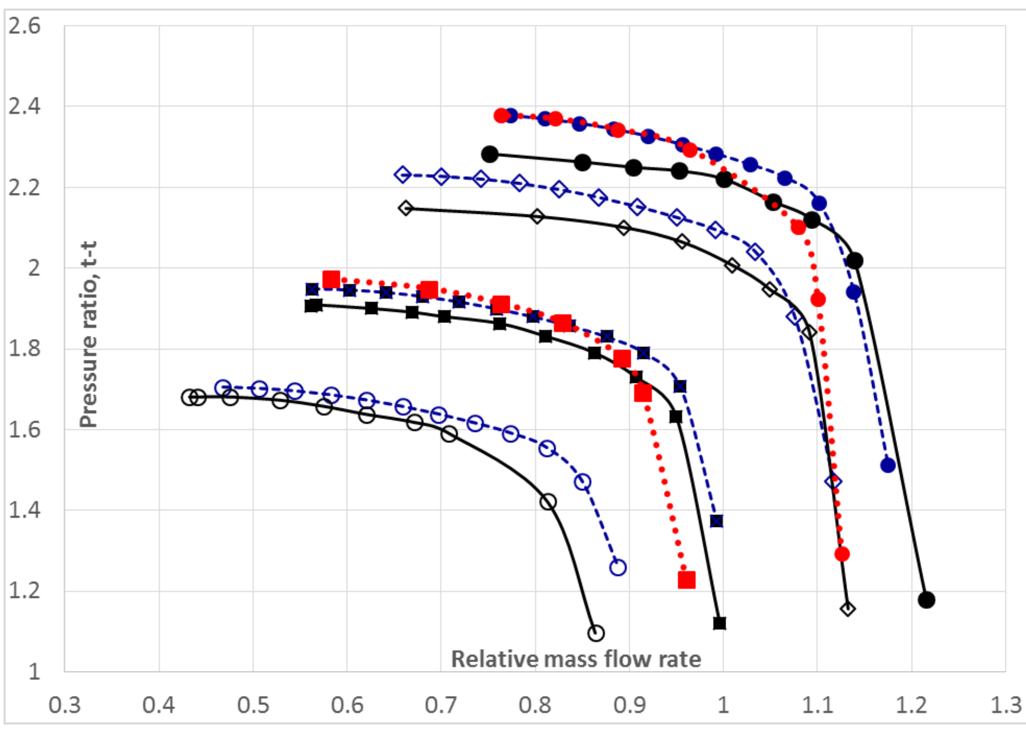

Figure 7 - LTCM simulation versus experiment. Pressure ratio versus normalized mass flow characteristic comparison (Performance: —— experiment, - - - 1D meanline, · · · 3D CFD; normalized speed: ● 1.0, ◇0.95, ■ 0.86, ○ 0.76)

The simulated compressor efficiency is shown in Figure 8. The discrepancy in the simulated results and experimental data has been reduced due to new model development (maximum difference is 3%).

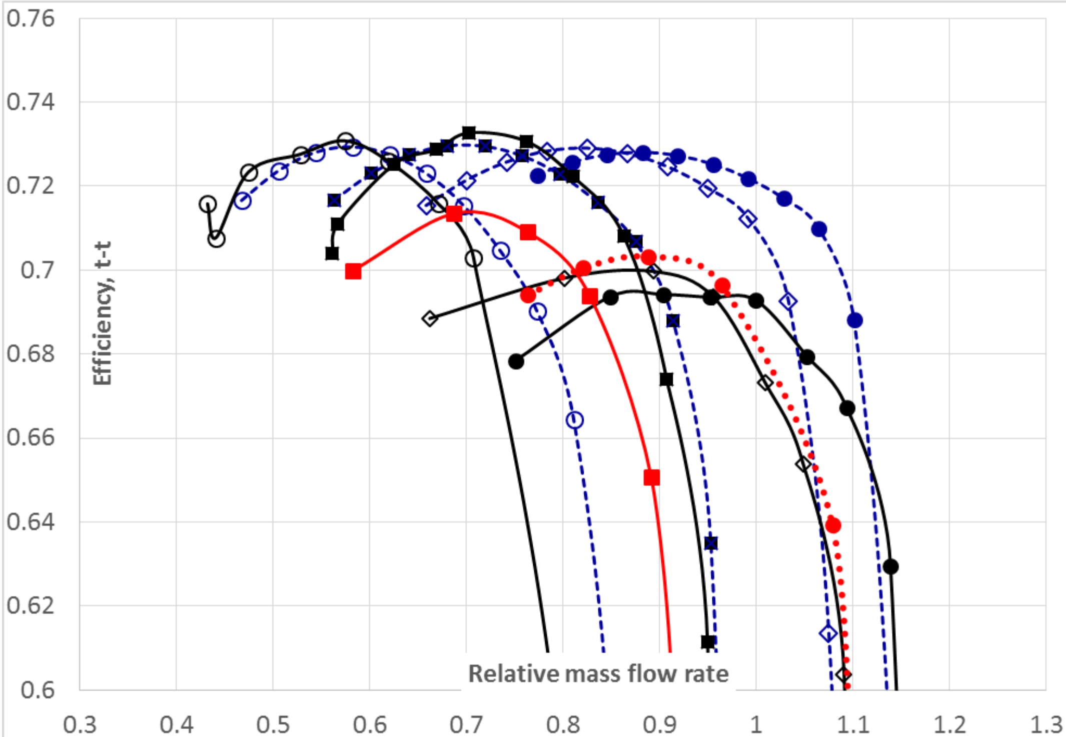

Figure 8 - LTCM simulation versus experiment. Hydraulic total-to-total efficiency versus normalized mass flow characteristic comparison. (Performance: ── experiment, - - - 1D meanline, • • • 3D CFD; normalized speed: ● 1.0, ◇ 0.95, ■ 0.86, ○ 0.76)

7. Conclusions

The new large tip clearance model is developed to improve turbomachinery design and analysis tools. In particular, at the preliminary design stage, accounting for this phenomenon gives the ability to design a compressor in a way that has better correlation with experiment and CFD. This saves tremendous amounts of design time.

AxSTREAM® curves overpredict the experimental characteristic. Possibly, there are some other losses external to the flow path or experimental inaccuracy to flow path that introduce such error.

The good agreement between AxSTREAM® 1D and 3D CFD signals that the gas dynamic simulation is precise and provides enough grounds for validity and accuracy of the large tip clearance model.

Correction of the deviation angle (slip factor) was applied for the impeller with large tip clearance (δ/b2>0.11–0.12). Now, the slip factor for such impellers is less than calculated by Wiesner formula and consequently, deviation is higher.

Additional heat input due to flow recirculation in meridional direction at the impeller outlet near shroud is simulated in 1D meanline analysis.

References

[1] Moroz, L., Govoruschenko, Y., Pagur, P., and Romanenko, L., 2008, "Integrated Conceptual Design Environment for Centrifugal Compressors Flow Path Design," ASME International Mechanical Engineering Congress and Exposition Paper No. IMECE2008-69122

[2] Aungier, R. H., 2000, “Centrifugal Compressors: a strategy for aerodynamic design and analysis”, ISBN 0-7918-0093-8

[3] Ris, V. F., 1981. “Centrifugal compression machines”. L.: Mashinostroenie. 1981. 351 p.

[4] Danish, S. N., and others 2006 “The Influence of Tip Clearance on Centrifugal Compressor Stage of a Turbocharger,” 4th WSEAS International Conference on Fluid Mechanics and Aerodynamics, Elounda, Greece, August 21-23, 2006 (pp6-11).

[5] Schleer, Matthias, and others, 2008, “Clearance Effects on the Onset of Instability in a Centrifugal Compressor,” Journal of Turbomachinery JULY 2008, Vol. 130.

[6] Tang, J., 2006 “Numerical investigation of the effects of tip clearance to the performance of a small centrifugal compressor, ” GT2006-90893

[7] Backman, Jari L. H., 2007, “Effects of impeller tip clearance on centrifugal compressor efficiency” Paper No. GT2007-28200