

MODELLING OF LOW-REYNOLDS-NUMBER CENTRIFUGAL COMPRESSORS: CHALLENGES IN PERFORMANCE AND FLOW CONTROL PREDICTION

JONNA TIAINEN, AKI GRÖNMAN*, AHTI JAATINEN-VÄRRI AND
JARI BACKMAN

Lappeenranta University of Technology
P.O. Box 20, 53851 Lappeenranta, Finland
*gronman@lut.fi, www.lut.fi

Key words: Drag reduction, RANS, Reynolds number effect

Abstract. A low-Reynolds-number causes a performance deterioration in centrifugal compressors due to the relative increase in boundary layer thickness, tip clearance, and surface roughness. The degradation of the efficiency can be estimated by empirical correlations, and they provide a good foundation for evaluating the accuracy of the numerical models. In order to improve the compressor performance, four passive flow control methods, grooves, riblets, squealers and winglets have been proposed to be the most promising ones. However, to accurately simulate their influence on low-Reynolds-number machines, the specific modelling requirements should be understood. This study compares the accuracy of numerical predictions relative to an empirical correlation with a changing Reynolds number for two centrifugal compressors. In addition, challenges and possibilities in the modelling of the most promising flow control methods are analysed and discussed.

1 INTRODUCTION

A micro-scale gas turbine is a promising option for distributed energy generation. It also has potential in a number of other applications requiring high power density, such as batteries and unmanned aerial vehicles fighting wildfires, to name but a few. The micro-scale gas turbines are comprised of centrifugal compressors and radial turbines. Especially, the performance of the compressor has an essential role for the overall efficiency of the cycle. Small-sized centrifugal compressors have a lower performance than their larger counterparts. The performance deterioration originates in the relative increase in boundary layer thickness, tip clearance, and surface roughness. The reduction in compressor efficiency can be estimated using empirical correction equations in the design process [1]. The empirical correlations are especially valuable if there is no experimental data to validate the numerical simulations, and they are commonly used in a design process.

The performance deterioration with the decreasing Reynolds number could be tackled with different flow control methods, such as grooves [2–4], riblets [5, 6], squealers and

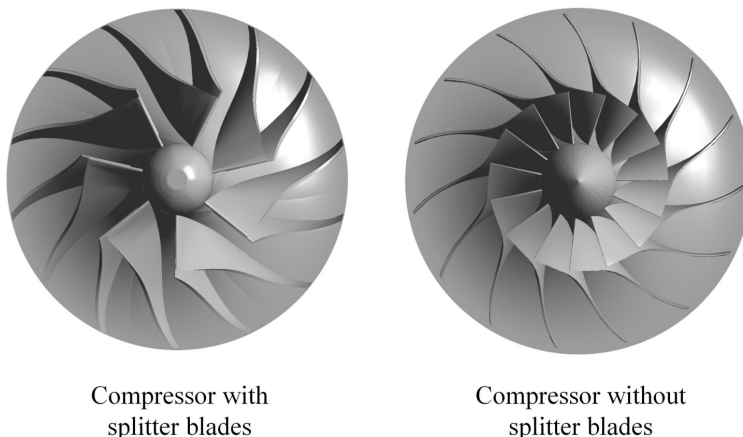


Figure 1: Compressor geometries (not to scale)

winglets [7], as was suggested by Tiainen et al. [8]. The mentioned flow control methods have shown their effectiveness in delaying stall [4], reducing drag [3, 6], and reducing tip leakage [7] in a number of applications, e.g., in transportation, energy, sports engineering, and in nature [9]. The preliminary investigation of the applicability of different flow control methods in a low-Reynolds-number centrifugal compressor requires numerical modelling, and therefore the specific requirement of each method should be understood from the modelling point of view.

From the presented background, the first objective of this study is to compare the numerical stage performance results against the empirical correction equation of Dietmann and Casey [1] in order to evaluate the achievable accuracy of the used modelling approach. The second objective is to analyse challenges and possibilities in the modelling of the most promising flow control methods in low-Reynolds-number centrifugal compressors.

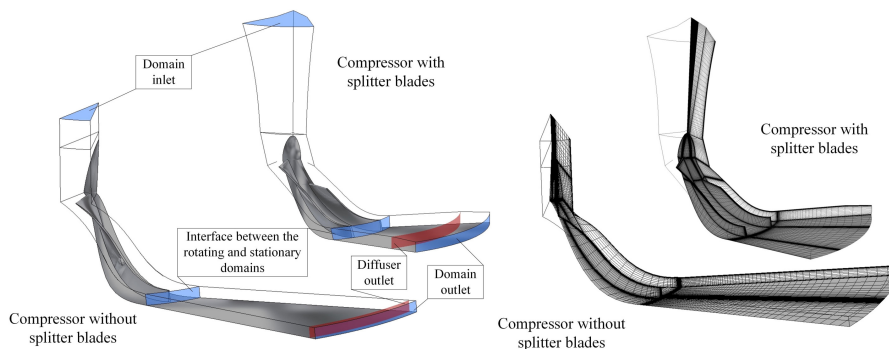
2 METHODS AND NUMERICAL MODEL

The changes in compressor size, operating conditions, or medium result in a change in the Reynolds number. In the present study, the Reynolds number was changed by varying the compressor size. The modelled two compressors were unshrouded centrifugal compressors with vaneless diffusers. One compressor had splitter blades and the other had only full blades, as shown in Fig. 1. The compressor with splitter blades was studied experimentally and numerically at Lappeenranta University of Technology, Finland [10], and the compressor without splitter blades is the test case Radiver, in which the measurements were carried out at the Institute of Jet Propulsion and Turbomachinery at RWTH Aachen, Germany [11]. The compressor with splitter blades was studied at the design point and the compressor without splitter blades at the peak efficiency point at a reduced speed. Technical data of the compressors is shown in Table 1.

The commercial software ANSYS CFX 17.0 was used for steady-state numerical calculations. In the computational domain (Fig. 2), the volute was neglected, rotational periodicity was assumed, and the frozen rotor approach was used between the rotating

Table 1: Technical data of the compressors

		With splitter blades	Without splitter blades
Number of blades	Z	7 + 7	15
Relative blade height	b_2/D_2	0.058	0.041
Relative tip clearance	t/b_2	0.052	0.045
Chord Reynolds number	$Re_{c,\text{ref}} = (w_1 c)/\nu_1$	$17 \cdot 10^5$	$16 \cdot 10^5$
Flow coefficient	$\phi = q_v/(U_2 D_2^2)$	0.065	0.051
Pressure coefficient	$\psi = \Delta h_s/U_2^2$	0.520	0.450
Specific speed	$N_s = (\omega \sqrt{q_v})/\Delta h_s^{0.75}$	0.830	0.830
Tip speed Mach number	$Ma_U = U_2/a_1$	0.920	1.170


Figure 2: Computational domains and structured grid (not to scale)

and stationary parts. The total pressure and total temperature were specified at the inlet of the computational domain, and the mass flow rate at the outlet. Adiabatic and no-slip conditions were specified at the solid walls. Turbulence was modelled by using the $k - \omega$ Shear Stress Transport (SST) model, which is widely used and has been validated for turbomachinery applications [12]. Regarding the grid quality, the y^+ values were below unity on most of the surfaces, with the most challenging region for meshing being the stagnation point at the blade leading edge. At the inlet of the computational domain, the turbulence intensity of 5% was specified. The target values for numerical convergence were the efficiency and mass imbalance between the inlet and outlet. The simulation was considered converged when the change in the target values was below 0.1% and the change in the normalised residuals of energy, mass, momentum, and turbulence parameters was stabilised.

The target values regarding grid independence were the total-to-total isentropic efficiency and total-to-total pressure ratio between the computational domain inlet and diffuser outlet. The grid independence study is presented in the latest study by the present authors [13], and its main results are repeated here for the readers' convenience.

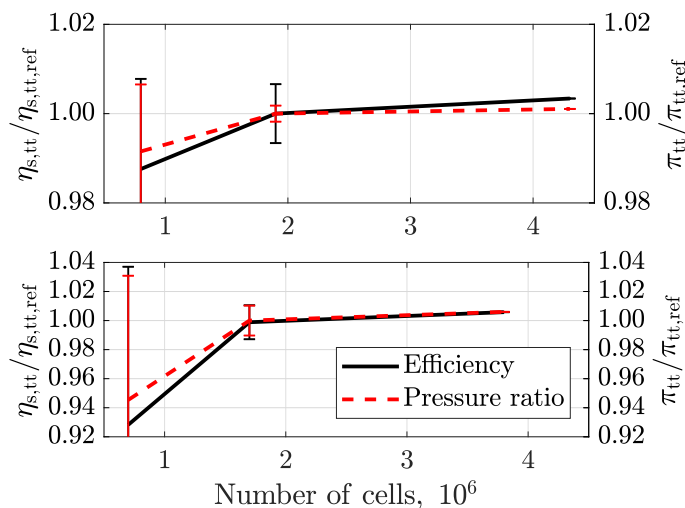


Figure 3: Grid independence of the compressors with splitter blades (top) and without splitter blades (bottom) with discretisation error bars.

As a result of the grid independence study, the grids with 1.9 and 1.7 million cells (Fig. 2) were chosen for the compressors with and without splitter blades, respectively. To reduce the discretisation error, the grid was made denser in the areas of larger gradients, and second order discretisation scheme was used. The discretisation error estimated by using the procedure presented by Celik et al. [14] is shown with error bars in Fig. 3.

The accuracy of the models was evaluated with the baseline compressors at different operating conditions against the experimental data in Tiainen et al. [13], and the results are represented for the reader's convenience in Fig. 4. It can be noticed that the numerical model over-predicts the pressure ratio and efficiency, but the trends are still captured. The over-prediction results from the neglect of the volute and exit cone in the numerical model. Based on the experimental results of Hagelstein et al. [15] for the total pressure loss coefficient of 0.4-0.85, the volute and exit cone would result in an additional loss of 1.5-6% in the total-to-total pressure ratio and total-to-total isentropic efficiency. In the computational models, the losses due to disk friction, leakage flow through the backside cavity, and surface roughness were also neglected. Leakage flow through the backside cavity can result in an additional loss of 1% in the pressure ratio and efficiency [16].

Despite the over-prediction of the pressure ratio and efficiency, the flow field was predicted fairly accurately as shown in Fig. 5 for the compressor without splitter blades. The relative difference between the area-averaged experimental and numerical results of the absolute velocity, relative velocity, and flow angle (from the radial direction) at $r/r_2 = 0.99$ were -2.5%, -1.7%, and +1.0%, respectively.

The numerical results with the decreasing Reynolds number were compared to the performance prediction of the well-known empirical correction equation published by Dietmann and Casey [1] (Fig. 6) in order to evaluate the achievable accuracy of the models. The trend of the numerical results follows very closely the trend of the correction equation, and the differences are mostly inside the discretisation error bars.

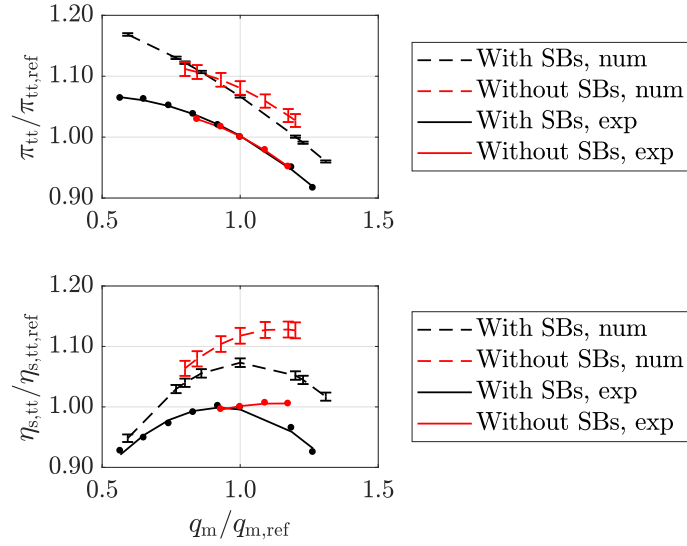


Figure 4: Validation of numerical results of pressure ratio (top) and efficiency (bottom) against the experimental data

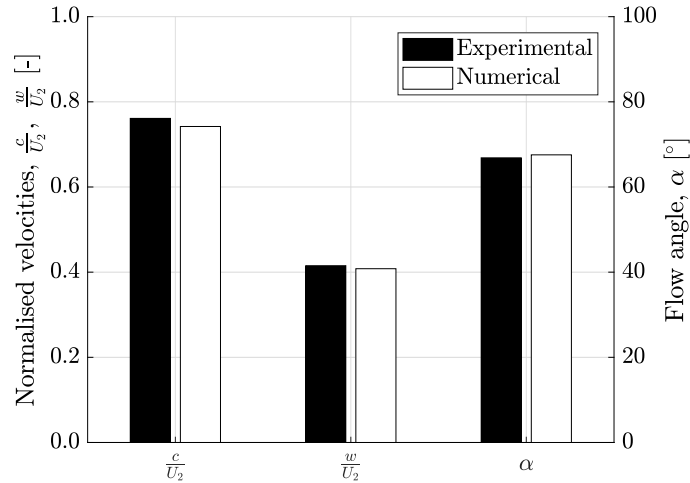


Figure 5: Validation of numerical results of normalised absolute velocity, normalised relative velocity, and flow angle (from the radial direction) in the compressor without splitter blades at $r/r_2 = 0.99$ against the experimental data

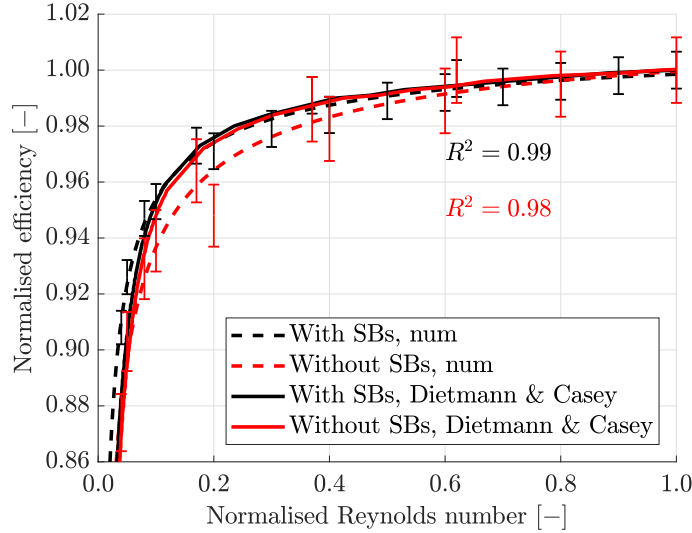


Figure 6: Validation of numerical results against experimental correction equation

The decrease in Reynolds number results in thicker boundary layers and increased friction losses. Generally, the losses due to blade boundary layers, endwall boundary layers, and tip leakage are of the same order of magnitude in compressors [17]. The latest study by the present authors [13] indicated that tip leakage strengthened and the boundary layer losses increased relatively more near the impeller hub and diffuser surfaces with the decreased Reynolds number. Therefore, the potential flow control methods for tip leakage and boundary layer control in centrifugal compressors will be discussed next.

3 FLOW CONTROL METHODS

The applicable flow control method in the low-Reynolds-number centrifugal compressor depends on whether the compressor is designed to operate at a single, maximum efficiency point or in a wide operating range. If a passive flow control method is implemented in a compressor designed to operate in a wide range, the method can deteriorate the compressor performance in off-design conditions. For example, riblets are designed and manufactured on a surface in a streamwise direction of the flow, but if the flow direction is markedly changed, the drag reduction changes to a drag increase. Contrary to passive methods, active methods can be switched on and off, based on the operating conditions. Therefore, they will not deteriorate the performance when not needed. However, the disadvantage of the active methods in turbomachinery applications is their complex structure. Based on the aerodynamic and structural demands, the potential flow control methods in centrifugal compressors are riblets for boundary layer control, and squealers, winglets and grooves for tip leakage control [8] (Fig. 7). In this section, the challenges and possibilities in the modelling of these control methods are analysed.

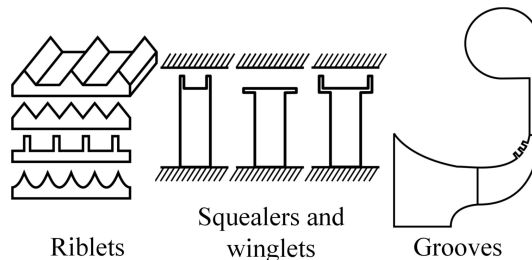


Figure 7: Riblets, squealers, winglets and grooves

3.1 Riblets

The riblets are small streamwise aligned grooves that shift turbulent vortices farther away from the surface, resulting in decreased momentum transfer and wall shear stress [18, 19]. As the results presented above indicate that the numerical results follow very closely the trend of the empirical correction equation of Dietmann and Casey [1], the same correction equation is modified to account for the drag reduction capability of riblets in order to estimate the effect of riblets on compressor performance. The effect of riblets on compressor efficiency is estimated based on the drag reduction capability of riblets, which is proportional to the reduction in the friction factor. The reduction in the friction factor results in the increase in efficiency. If the maximum drag reduction of 13% [20] is assumed, the efficiency improvement varies in the range of 1-3% with a decreasing Reynolds number (Fig. 8).

The challenge in modelling the riblets in turbomachinery applications is due to the size of the riblets, which is of the same order of magnitude as the thickness of the viscous sublayer. To account the riblets directly in the computational domain of a compressor geometry would result in increased computational effort due to the extremely dense mesh. Martin and Bhushan [6], Boomsma and Sotiropoulos [21], and Peet et al. [22] used Large Eddy Simulation to solve flow field over a flat plate with riblet surface, whereas Duan and Choudhari [23], García-Mayoral and Jiménez [24], and Strand and Goldstein [25] used Direct Numerical Simulation to solve flow field over a flat plate with riblet surface.

However, computationally less expensive methods are also available. According to Spalart and McLean [26], RANS simulations are generally used to predict the effect of riblets in the aeroplane applications, with the smooth surface assumption and velocity profile adjustment. To account for the riblets in the modelling of turbomachinery, Köpplin et al. [5] published a correlation-based riblet model. The model was applied to the $k - \omega$ turbulence model, and it damps the specific dissipation rate near the wall.

3.2 Squealers and winglets

The purpose of the squealers and winglets is to reduce the losses associated with the tip leakage flow. Squealers are vertical protrusions on a blade tip that point towards the casing. If squealers are applied on both the pressure and suction sides of the blade tip, a cavity is formed between them. Winglets are protrusions on a blade tip, perpendicular

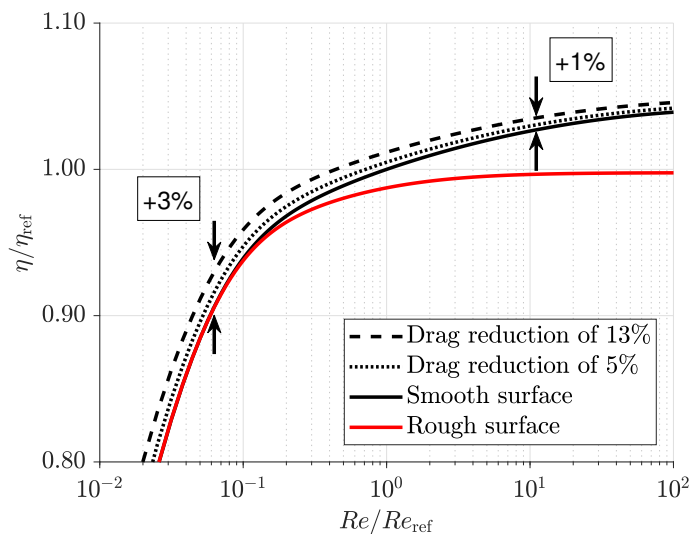


Figure 8: Estimated effect of riblets on the performance of the compressor with splitter blades

to the blade surface, pointing towards adjacent blades. They can be applied on pressure, suction, or both sides of the blade, separately or together.

In the numerical simulations, the structures of the squealers and winglets are accounted for directly in the mesh generation. The turbulence is modelled either using the Spalart-Allmaras model [27–29] or the $k - \omega$ SST model [7, 30, 31].

In general, good agreement between the numerical and experimental results has been reported. The validation of Han and Zhong [27] showed that the modelled and measured interaction between the shock wave and tip leakage vortex agreed well. The validation of Caloni et al. [7] showed that the modelled and measured interaction between the shock and boundary layer equals accurately. Also the tip leakage vortex is well-captured with the model when the modelled and measured total pressure loss contours are compared.

3.3 Grooves

Circumferential grooves at the shroud surface reduce the boundary layer thickness near the grooves. The grooves transfer low momentum fluid from the pressure to the suction side while energising it by decreasing the radial velocity and increasing the tangential velocity. From four to eight grooves should be located near the separation point with a spacing of one-half groove width. It is recommended that the groove depth and width are approximately of the order of the boundary layer displacement thickness [32].

Overall, grooves could be beneficial in low-Reynolds-number centrifugal compressors in which they increase the pressure-ratio and weaken the wake. Micro-scale compressors with relatively large tip clearances in particular could benefit from the modified tip leakage flow and more uniform flow field at the impeller outlet. However, the drawback of the grooves is that the increased wall surface results in additional friction losses so that the efficiency is not improved.

Likewise the squealers and winglets, grooves are modelled as geometrical modifications

in the computational mesh [4]. The $k - \omega$ SST model [2, 4, 33] is generally used. The $k - \omega$ SST model was chosen by Cevik et al. [2], as it predicts the tip leakage vortices better than other turbulence models.

4 CONCLUSIONS

All in all, the accuracy of the chosen numerical approaches based on the two equation turbulence model and RANS equations is on an acceptable level when the effect of the Reynolds number is studied. To get more accurate validation, an experimental campaign should be made for comparison.

From the flow control modelling point of view, the passive methods for tip leakage control, i.e. squealers, winglets and grooves, can be applied in the numerical model as modifications in a computational model of a compressor geometry. However, as the modelling of a centrifugal compressor is still practically limited to steady or unsteady RANS models, the small-scale turbulence in the vicinity of the riblet structure is recommended to be modelled with the correlation-based model.

As the riblets are predicted to improve the compressor efficiency even by three per cent, it is recommended as a future work that the correlation-based riblet model should be implemented in a centrifugal compressor with different Reynolds numbers. After comparing the results with the correlation-predicted performance improvement, a deeper preliminary understanding of riblet's feasibility in this application can be achieved.

REFERENCES

- [1] F. Dietmann and M.V. Casey. The Effects of Reynolds Number and Roughness on Compressor Performance. In J. Backman, G. Bois, and O. Leonard, editors, *Proceedings of the 10th European Conference on Turbomachinery: Fluid Dynamics and Thermodynamics*, pages 532 – 542, 2013. April 15–19, 2013, Lappeenranta, Finland.
- [2] M. Cevik, H.D. Vo, and H. Yu. Casing Treatment for Desensitization of Compressor Performance and Stability to Tip Clearance. *Journal of Turbomachinery*, 138(12):Article ID 121008, 16 pages, 2016.
- [3] C.T. DeGroot, C. Wang, and J.M. Floryan. Drag Reduction Due to Streamwise Grooves in Turbulent Channel Flow. *Journal of Fluids Engineering, Transactions of the ASME*, 138(12), 2016.
- [4] S. Bareiß, D. M. Vogt, and E. Chebli. Investigation on the Impact of Circumferential Grooves on Aerodynamic Centrifugal Compressor Performance. In *Proceedings of ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, 2015. Paper No. GT2015-42211, p. 11. June 15–19, 2015, Montreal, Canada.
- [5] V. Köpplin, F. Herbst, and J. Seume. Correlation-Based Riblet Model for Turbomachinery Applications. *Journal of Turbomachinery*, 139(7), 2017. Article ID 071006, 10 pages.

- [6] S. Martin and B. Bhushan. Modeling and Optimization of Shark-Inspired Riblet Geometries for Low Drag Applications. *Journal of Colloid and Interface Science*, 474:206–215, 2016.
- [7] S. Caloni, S. Shahpar, and J.D. Coull. Numerical Investigations of Different Tip Designs for Shroudless Turbine Blades. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 230(7):709–720, 2016.
- [8] J. Tiainen, A. Grönman, A. Jaatinen-Värri, and J. Backman. Flow Control Methods and Their Applicability in Low-Reynolds-Number Centrifugal Compressors—A Review. *International Journal of Turbomachinery, Propulsion and Power*, 3(1), 2018. 2.
- [9] B. Dean and B. Bhushan. Shark-Skin Surfaces for Fluid-Drag Reduction in Turbulent Flow: A Review. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 368(1929):4775–4806, 2010.
- [10] Ahti Jaatinen-Värri, Pekka Rönttö, Teemu Turunen-Saaresti, and Aki Grönman. Experimental Study of Centrifugal Compressor Vaneless Diffuser Width. *Journal of Mechanical Science and Technology*, 27(4):1011–1020, 2013.
- [11] K.U. Ziegler, H.E. Gallus, and R. Niehuis. A Study on Impeller-Diffuser Interaction - Part I: Influence on the Performance. *Journal of Turbomachinery*, 125(1):173–182, 2003.
- [12] F.R. Menter. Review of the Shear-Stress Transport Turbulence Model Experience from an Industrial Perspective. *International Journal of Computational Fluid Dynamics*, 23(4):305–316, 2009.
- [13] J. Tiainen, A. Jaatinen-Värri, A. Grönman, T. Fischer, and J. Backman. Loss Development Analysis of a Micro-Scale Centrifugal Compressor. *Energy Conversion and Management*, 2018. Submitted.
- [14] I.B. Celik, U. Ghia, P.J. Roache, C.J. Freitas, H. Coleman, and P.E. Raad. Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications. *Journal of Fluids Engineering*, 130(7):Article ID 078001, 4 pages, 2008.
- [15] D. Hagelstein, K. Hillewaert, R.A. Van Den Braembussche, A. Engeda, R. Keiper, and M. Rautenberg. Experimental and Numerical Investigation of the Flow in a Centrifugal Compressor Volute. *Journal of Turbomachinery*, 122(1):22–31, 2000.
- [16] Z. Sun, C. Tan, and D. Zhang. Flow Field Structures of the Impeller Backside Cavity and Its Influences on the Centrifugal Compressor. In *Proceedings of ASME Turbo Expo 2009: Power for Land, Sea and Air*, volume 7, pages 1349–1360, 2009. Paper No. GT2009-59879. June 8-12, 2009, Orlando, Florida, USA.

- [17] J. D. Denton. Loss Mechanisms in Turbomachines. *Journal of Turbomachinery*, 115:621–656, 1993.
- [18] D.W. Bechert, M. Bruse, and W. Hage. Experiments with Three-Dimensional Riblets as an Idealized Model of Shark Skin. *Experiments in Fluids*, 28(5):403–412, 2000.
- [19] S.-J. Lee and S.-H. Lee. Flow Field Analysis of a Turbulent Boundary Layer over a Riblet Surface. *Experiments in Fluids*, 30(2):153–166, 2001.
- [20] J.R. Debisschop and F.T.M. Nieuwstadt. Turbulent Boundary Layer in an Adverse Pressure Gradient: Effectiveness of Riblets. *AIAA Journal*, 34(5):932–937, 1996.
- [21] A. Boomsma and F. Sotiropoulos. Riblet Drag Reduction in Mild Adverse Pressure Gradients: A Numerical Investigation. *International Journal of Heat and Fluid Flow*, 56:251–260, 2015.
- [22] Y. Peet, P. Sagaut, and Y. Charron. Turbulent Drag Reduction using Sinusoidal Riblets with Triangular Cross-Section. In *38th AIAA Fluid Dynamics Conference and Exhibit*, 2008. Paper No. AIAA-2008-3745, p. 9. June 23 - 26 2008, Seattle, WA.
- [23] L. Duan and M. M. Choudhari. Effects of Riblets on Skin Friction in High Speed Turbulent Boundary Layers. In *50th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, 2012. January 9–12, 2012, Nashville, Tennessee.
- [24] R. García-Mayoral and J. Jiménez. Hydrodynamic Stability and Breakdown of the Viscous Regime over Riblets. *Journal of Fluid Mechanics*, 678:317–347, 2011.
- [25] J.S. Strand and D.B. Goldstein. Direct Numerical Simulations of Riblets to Constrain the Growth of Turbulent Spots. *Journal of Fluid Mechanics*, 668:267–292, 2011.
- [26] P.R. Spalart and J.D. Mclean. Drag Reduction: Enticing Turbulence, and Then an Industry. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*, 369(1940):1556–1569, 2011.
- [27] S. Han and J. Zhong. Effect of Blade Tip Winglet on the Performance of a Highly Loaded Transonic Compressor Rotor. *Chinese Journal of Aeronautics*, 29(3):653–661, 2016.
- [28] Z. Schabowski and H. Hodson. The Reduction of Over Tip Leakage Loss in Unshrouded Axial Turbines Using Winglets and Squealers. *Journal of Turbomachinery*, 136(4):Article ID 041001, 11 pages, 2014.
- [29] Z. Schabowski, H. Hodson, D. Giacche, B. Power, and M.R. Stokes. Aeromechanical Optimization of a Winglet-Squealer Tip for an Axial Turbine. *Journal of Turbomachinery*, 136(7), 2014.

- [30] R. Da Soghe, C. Bianchini, L. Toni, and D.T. Rubino. Effects of Impeller Squealer Tip on Centrifugal Compressor Performance. volume 2D-2016, 2016.
- [31] J.-S. Jung, O. Kwon, and C. Son. An Investigation on Aerodynamics Loss Mechanism of Squealer Tips of a High Pressure Turbine Blade Using URANS. volume 2D-2016, 2016.
- [32] A. J. Burton. Centrifugal Compressor Boundary Layer Control, July 1975. US, 3,893,787. United Aircraft Corporation. 8.7.1975.
- [33] J. Du and J. R. Seume. Design of Casing Treatment on a Mixed-Flow Compressor. In *Proceedings of ASME Turbo Expo 2017: Turbomachinery Technical Conference and Exposition*, 2017. Paper No. GT2017-65226, p. 10. June 26 -30, 2017, Charlotte, NC, USA.