Evaluation of CFD simulation on boundary between meshes of different types

Tomáš Radnic[1,*], Pavel Šafařík[1]

[1] ČVUT v Praze, Fakulta strojní, Ústav Mechaniky Tekutin a termomechaniky, Technická 4, 166 07 Praha 6,

Abstract

The paper deals with the feasibility of usage of the structured and unstructured mesh with a boundary between the two types of the meshes for a simulation of a transonic flow past a mid-section cascade of the last stage rotor blade of a turbine of large power output with a non-prismatic stabilization device. The stabilization device does not allow creation of a structured mesh and therefore the possibilities of boundary between two types of meshes must be evaluated. The meshes were generated in Ansys ICEM CFD 17.2, simulations were performed in Ansys CFX 17.2.

Keywords: CFD, transonic flow, structured mesh

1. Introduction

The authors dissertation thesis deals with evaluation of a transonic flow in the interblade channel. Investigated blade cascade is a mid-section of the Doosan Škoda power module 7, 1220 mm long blade. Because of the length of the blade, the aspect ratio of the blade is very high. The high aspect ratio makes the blade prone to deformation. Therefore, it has been fitted with a stabilization device called tie-boss, see Fig. 1.

***Fig. 1** – Tie-boss mounted on a 1220 mm long blade*

Investigation of the flow field consisted of an experimental investigation and a CFD simulation. The experimental investigation was performed in the laboratories of the Institute of Thermomechanics of the Czech Academy of Sciences in Nový Knín. The CFD simulation was performed on the Institute of technical mathematics of the Faculty of Mechanical engineering on Karlovo Náměstí [1].

The experimental investigation has shown that there is a strong separation on the intersection of the tie-boss body and the suction side of the blade and that there are strong vorticial structures on the intersection of the body of the tie-boss and pressure side of the profile, see Fig. 2.

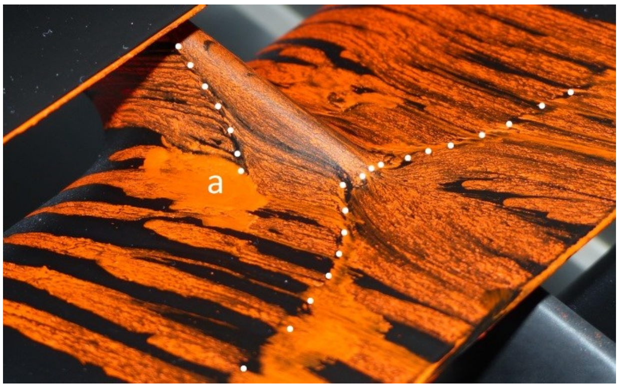

***Fig. 2** - Separation bubble at a) and traces of shockwaves marked by dotted lines*

The simulations were performed on a structured mesh consisting with a y^+ value of 40 – 50. The flow was modeled using the system of time-averaged Navier-Stokes equations for compressible flows combined with the ideal gas law and with the two-equation SST turbulence model [2]. Relatively high y^+ value meant that the model could have had fewer elements, however it did not describe the interactions within the proximity to the blade. For better understanding of the phenomena in the vicinity of the tie-boss body and the blade, a model with y^+ lower than 1 had to be done.

The two possible approaches to the modelling of the mesh present considerable problems for the simulation. The structured mesh will have fewer elements with a better accuracy of the results than the unstructured mesh. The unstructured mesh will accurately mesh surfaces of very complicated shapes such as the tie-boss, where the structured mesh could not be done. Therefore, a combination of meshes was seemed the most viable. The unstructured mesh would be used to generate a mesh around the body of the tie-boss while the structured mesh in the rest of the interblade channel and the area in front and behind it.

An analysis of the influence of the boundary between the two types of mesh had to be made. The analysis was completed by a sensitivity analysis of both types of mesh and SST and BSL EARSM turbulence models [3].

*radnic@it.cas.cz

2. Used model

The model used for the simulation is a mid-section of a very long last stage blade that belongs to a steam turbine of large output made by Doosan Škoda power, see Fig. 3.

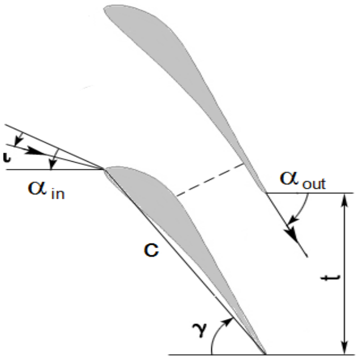

Fig. 3 *- Geometry of the interblade channel*

There were two computational domains modelled. First one was for the testing of individual turbulence models and had long inlet and outlet areas. The other was much smaller for the purpose of testing of the boundary only. The cell number budget had to be restricted because of the computational power available to the author.

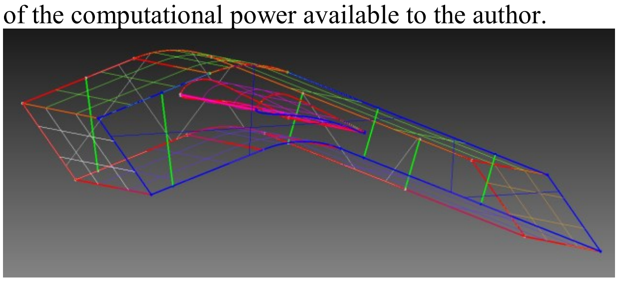

Fig. 4 *- Long computational domain*

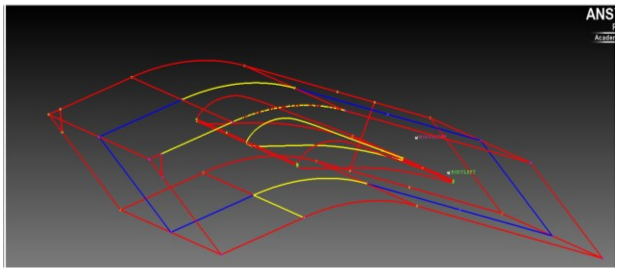

Fig. 5 *- Short computational domain*

The domains were shaped to roughly copy the direction of the flow in the interblade channel. Only the design regime of the cascade would be simulated, so the angle of incidence $\iota = 0°$. This means that only one shape of the inlet part was necessary.

3. Modelling software and solver

The 3D models of the computational domain were made in Solidworks CAD software. The mesh was prepared in Ansys ICEM CFD version 17.2. Unstructured mesh was generated by Octree algorithm, the prismatic layer was generated by post inflation. The solver used for the simulation is Ansys CFX 17.2.

The boundary conditions were uniform for all the tested cases. The inlet was set to total pressure $p_{tot} = 100\ kPa$ and $T_{tot} = 298\ K$. Inlet angle was set to be $\alpha_{in} = 30{,}9°$. Fluid was modelled as an ideal gas. Outlet was set to average static pressure of $p_{out} = 30\ kPa$. Side walls placement was not consistent trough the cases, but were always set as adiabatic no-slip walls as well as the surface of the blade. Inlet Mach number was approximately 0,3, outlet 1,4.

4. Mesh sensitivity analysis

For mesh sensitivity analysis total of four meshes have been prepared. Structured meshes with 4,5 and 6,3 million elements and unstructured meshes with 1,7 and 5 million elements.

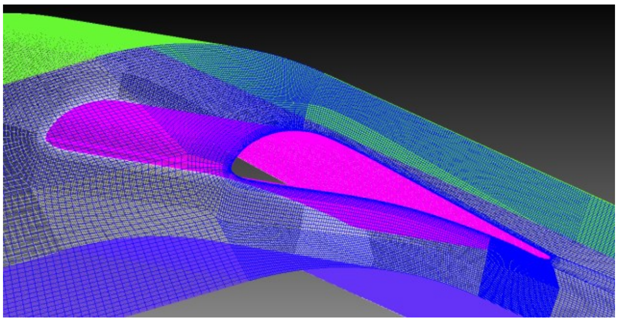

Fig. 6 *- Sturctured mesh, 4,5 mil. of elements*

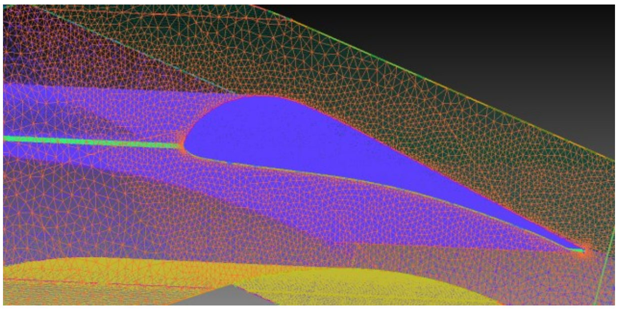

Fig. 7 *- Unstructured mesh, 5 mil. of elements*

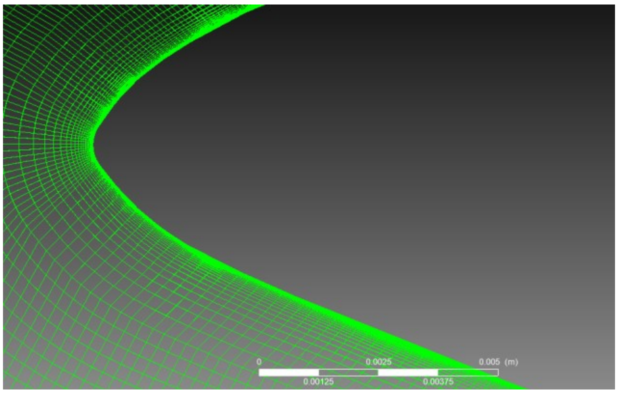

Fig. 8 *- Detail of leading edge mesh, structured mesh, 6,2 mil. of elements*

The cell budged did not allow a creation of the prismatic layer on the surface of the blade for the unstructured meshes of 1,7 and 4,5 million cells. The mesh generated for the boundary evaluation was smaller and the cell budged allowed to have refined enough mesh for achieving the y^+ values around 1.

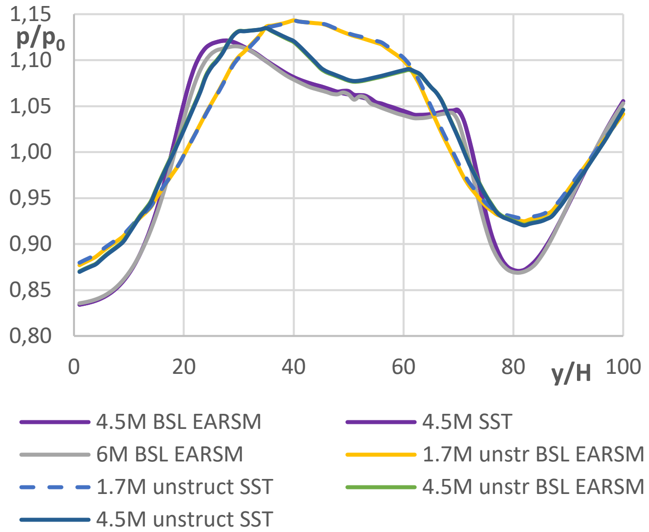

Fig. 9 *- Relative pressure distribution along midline of the domain 20 mm behind the trailing edge*

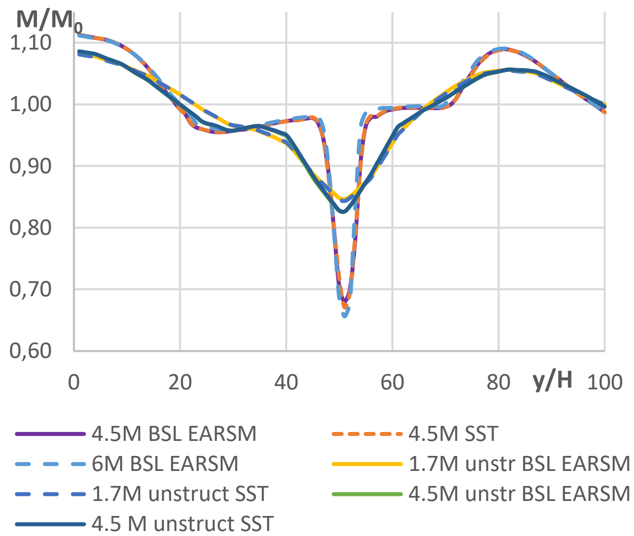

Fig. 10 *- Relative Mach number distribution along midline of the computational domain 20 mm behind the trailing edge*

On Fig. 10 there is a clearly visible disparity between the structured meshes and unstructured meshes in the middle of the height of the channel and at about 80% of the height. The dependence on the turbulence model seems very marginal, therefore for comparison purposes, only BSL EARSM model of turbulence was used. The visual inspection of the data can be performed on Fig. 11 and Fig. 12. It is clearly visible that the simulation performed on the structured mesh described the shockwaves much more clearly. It is also clear that the simulation on unstructured mesh does blur the wake of the blade very close to the blade. The quality of the results is much better for the structured mesh. Also, in comparison with the Fig. 13, the simulation on the structured mesh captured the shock wave reflection with much better accuracy.

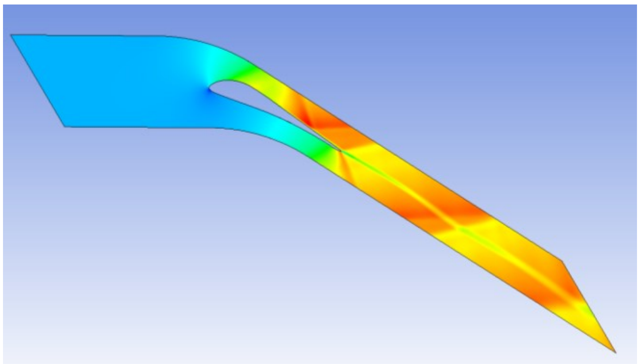

Fig. 11 *- Mach number distribution on midplane of computational domain – stuctured mesh 6 million, BSL EARSM*

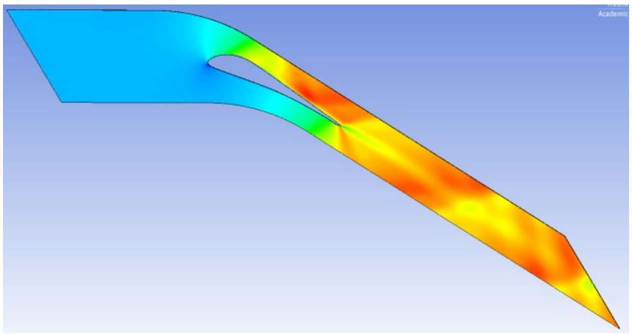

Fig. 12 *- Mach number distribution on midplane of computational domain – unstructured mesh 4.5 million, BSL EARSM*

Fig. 13 *- Schlieren photography of the flow field past the cascade in the design conditions*

5. Mesh interface behaviour

The two meshes prepared for the boundary evaluation are composed of 560 000 elements of structured mesh and 4,5 million elements of unstructured for the fine one and 279 000 elements of structured mesh and 900 000 elements of unstructured mesh for the coarse one. The selected images of individual meshes are shown below.

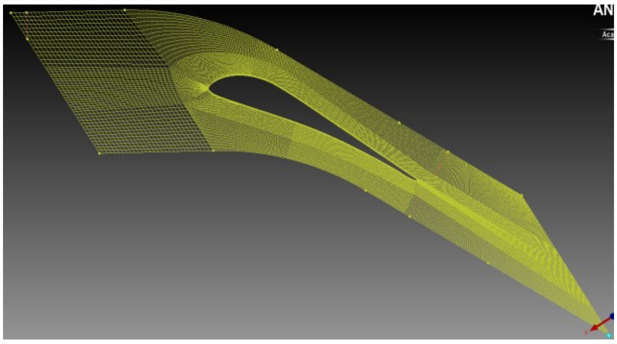

***Fig. 14** - Fine mesh - structured mesh profile*

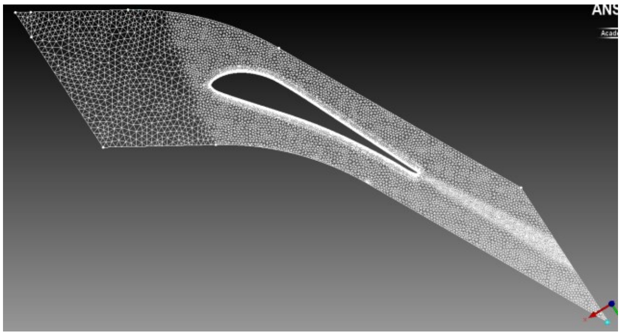

***Fig. 15** - Fine mesh - unstructured mesh profile*

The meshes were not joined at the boundary. A mesh interface was created instead in CFX pre-processing. Joining meshes was seemed to be unviable approach since the robust algorithm cannot be controlled precisely. Therefore large disparities in the mesh density were present at the boundary which created negatively oriented elements and penetrating faces in trials.

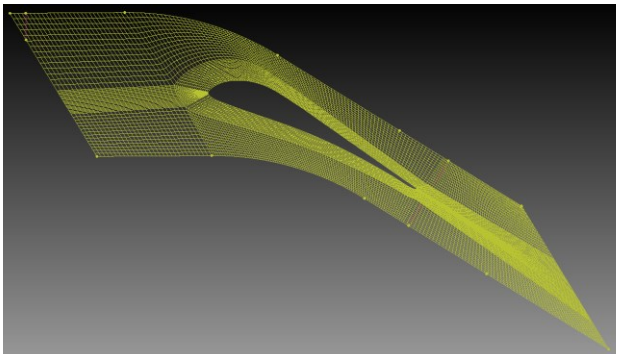

***Fig. 16** - Coarse mesh - structured mesh profile*

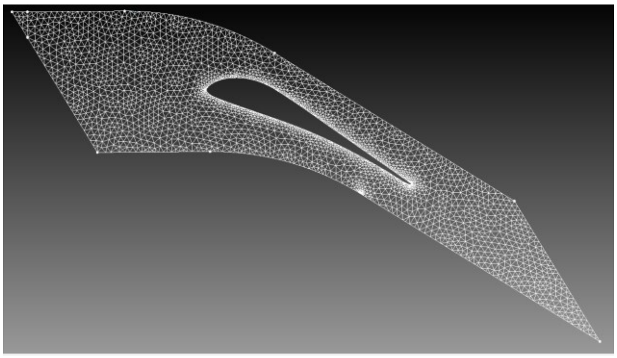

***Fig. 17** - Coarse mesh - unstructured mesh profile*

The two boundaries will be evaluated separately for better understanding of the results. The fine mesh cell budged allowed for modelling the boundary layer with y+ values below 1 to precisely describe the interactions of the boundary layer. On the Fig. 18 there is visualization of distribution of the y^+ on the surface of the blade.

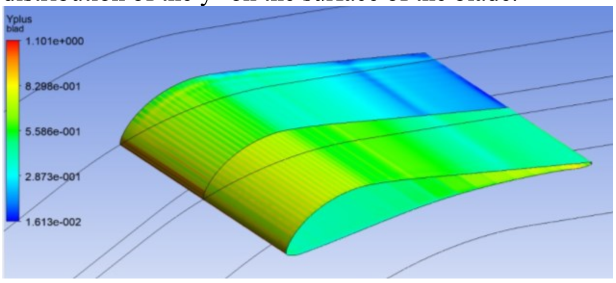

***Fig. 18** - y^+ distribution on the surface of the blade*

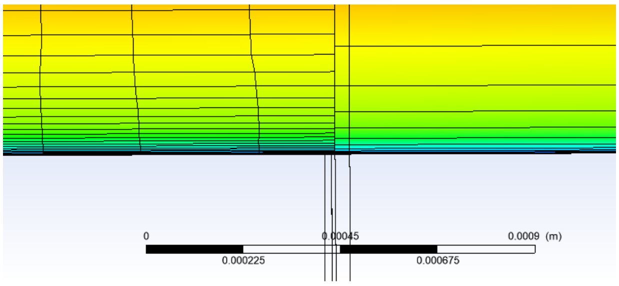

***Fig. 19** - Distribution of Mach number in the boundary layer on the mesh interface*

It is clear that the boundary does not have any significant negative influence on the quality of the results near the blade. The Fig. 19 also shows that the structured mesh, however coarser, does provide same results as the unstructured mesh with multiple times more elements.

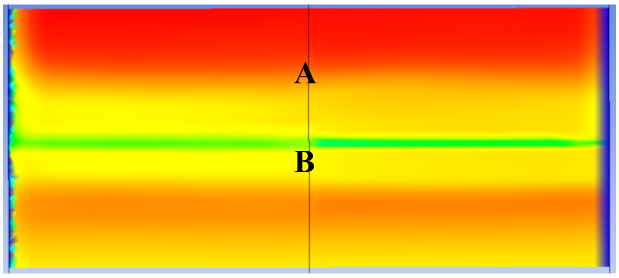

***Fig. 20** - Distribution of Mach number on a plane 6 mm behind the trailing edge, unstructured mesh is on the left*

On Fig. 20 there have been marked a trace of the outer branch of the blade exit shockwave by the letter A and blade wake by the letter B. It is apparent that the boundary between two types of mesh does not seem to have any influence on either the shock wave or the wake beside the influence of different type of mesh.

The coarse mesh however shows a different behaviour. From Fig. 21 it is apparent that the mesh interface does allows sort of sharp change of parameters on the boundary between the two meshes. This phenomena should not be a problem while modelling of the tie-boss will be done because the mesh will be very fine in the vicinity of the blade.

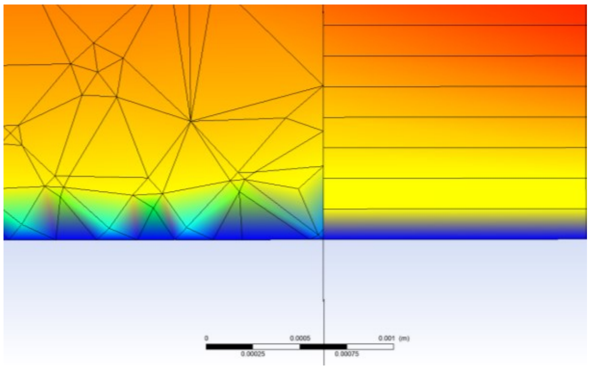

Fig. 21 *- Distribution of the Mach number in the vicinity of the blade on coarse mesh*

On Fig. 22 there are clearly visible differences between the solution of the unstructured mesh (left) and structured mesh (right). It is also apparent that the interface interpolates parameters of the flow field on both sides of the mesh. The influence is particularly visible on the trace of the outer branch of the exit shockwave of the blade. It is also clear that this coarse unstructured mesh provides compromised results of the simulation, blurring both the wake and the shockwaves present in the flow field while still having thrice the element count.

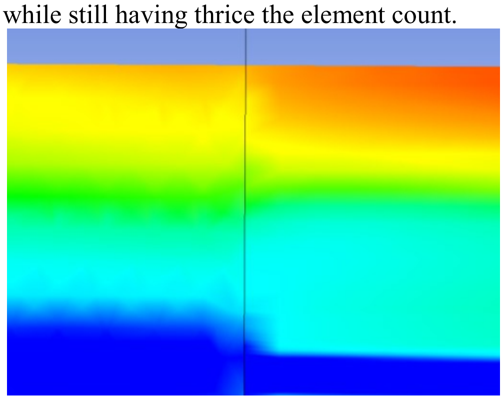

Fig. 22 *- distribution of Mach number on a plane 6 mm behind the trailing edge*

6. Conclusions

The testing of different meshes allowed for several conclusions to be drawn. Firstly it is clear that the quality vs element count is much better for the structured meshes. Secondly it is apparent that the difference between SST and BSL EARSM turbulence model is rather marginal. The real strength of the BSL EARSM should be apparent in a case with vortices in the boundary layer as it is capable of more accurate transition between boundary layer and free flow and of more accurate capturing of the secondary flows.

The testing on the boundary revealed that it does not pose any problems for the simulation. Unless there is a high disparity in the cell sizes there is no sharp change in the flow field parameters. It was also proven that the simulation of the boundary layer is not affected by the presence of the boundary between the meshes.

Future analysis must be performed on the vorticial structures that will emerge in the corners of the model. Also a boundary that will not be parallel to the flow must be assessed.

7. Acknowledgements

Acknowledgements of the authors belong to the Technology Agency of the Czech Republic, which supported previous research under grants no. TA03020277 and no. TH02020057. The acknowledgements also belong to the Faculty of mechanical engineering which supported the numerical investigation trough grant SGS17/074/OHK2/1T/12.

8. Nomenclature

M	Mach number, 1
T	temperature, K
Tu	turbulence intensity, 1
c	chord length, mm
p	pressure, kPa
t	pitch, mm
α	pitch angle of the flow, °
β	yaw angle of the flow, °
ι	angle of incidence, °
y^+	dimensionless wall distance, 1

Subscripts or superscripts

in	inlet
out	outlet
ref	chosen reference
tot	total

9. Literature

[1] T. Radnic, D. Šimurda, J. Hála, M. Luxa, J. Fürst, J. Hasnedl a J. Kellner, „Aerodynamic effects of tie-boss in extremaly long turbine blades,“ *ASME Journal of Gas power and Turbines,* 2018.

[2] F. R. Menter, „Two-equation eddy-viscosity turbulence models for engineering applications,“ *AIAAJ,* sv. 32, pp. 1598-1605, Aug. 1994.

[3] S. a. J. A. Wallin, „A complete explicit algebraic Reynolds stress model for incompressible and compressible flows,“ *Journal of Fluid Mechanics, 403,* pp. 89-132, 2000.