

CALCULATION OF FLOW CHARACTERISTICS IN HEAT TURBOMACHINERY TURBINE STAGE WITH DIFFERENT THREE DIMENSIONAL SHAPE OF THE STATOR BLADE WITH ANSYS CFX SOFTWARE

A. Yangyozov^{*}, R. Willinger^{}**

^{*} Department of Marine Machinery,
Technical University-Varna, Bulgaria, e-mail: naskosan@abv.bg

^{**} Institute for Thermodynamics and Energy Conversion,
Vienna University of Technology, Austria, e-mail: reinhard.willinger@tuwien.ac.at

Abstract: Main purpose of the presented work was the investigation of influence of three dimensional blades over flow characteristics. The paper presents comparison between experimental and CFD (Computational Fluid Dynamics) data so called verification test, impulse turbine stage with Frozen Rotor and Transient Rotor Interface modeling. Static pressure distribution was measured at mid span of the blade. The paper discusses secondary flow phenomena.

Keywords: CFD, heat turbomachinery, Compound Lean Nozzle, Controlled Flow Nozzle, efficiency.

I. INTRODUCTION

Before start with turbine stage modeling, especially important is for user to be sure in software capability that it can solve the problem. The most of commercial software use finite volume method and like every numerical method it also has some deviation from real parameters of the flow (errors), the same is in force for experiment. It is important this deviation to come to zero. This depends on designer experience. That's why first step is verification test making. In TU-Vienna, Institute for Thermodynamics and Energy Conversion is build wind tunnel with linear cascade with cylindrical blades. The latter are manufactured by use of coordinates from [2]. On fig.1 are presented some geometrical characteristics of the cascade. All of these data are used to model verification test blade. On fig.2 is shown mesh. This is unstructured tetrahedral mesh. To solve this problem is necessary to apply boundary conditions. There are as follows:

Inlet: velocity distribution in boundary layer is approximated by 1/7-th power law. Mid span velocity is about 18 m/s. The measured inlet turbulence intensity is $Tu=4\%$. Turbulent length scale is 0.1% of the pitch.

Outlet: At outlet plane static pressure is applied 96950 Pa.
In laboratory cascade there are nine blades, in CFD simulation is modeled one, it is used also symmetry boundary condition at mid span, and periodic boundary conditions.

To make sure of mesh quality, engineer must make preliminary calculation and check y^+ value. If the latter is not satisfying it is necessary to generate new mesh. In this simulation is used incompressible flow ($M < 0.2$) [1], and different turbulence models-Shear Stress Transport, Standard k-epsilon and RNG k-epsilon.

Results are presented on the next figures. Fig.4 shows secondary surface flow at hub wall comparisons between experiment with paint [2] and CFD simulation, but this is only qualitative picture of the flow in blade-to-blade channel. On this picture can be seen growing place of horseshoe vortex (only lower half part of the blade is shown). On fig.3 is visible downstream development and size decreasing of secondary vortex.

For engineer is important to have quantitative comparison. In turbomachinery experiment practice is used pneumatic probe to make static pressure measurements on blade surface. This data was collected from wind tunnel. Comparisons between experiment and CFD are shown on fig.5,a. For user is interesting to check sensitivity of this software and influence of inlet boundary layer. It is not good practice to ignore velocity change in boundary layer (it is possible for initial guess). Fig.5,b proves this. Inlet velocity distribution is obtained by experiment and this data follows 1/7-th power law [1].

Results show a good agreement between experiment and calculation (fig.5, (a)).

II. TURBINE STAGE CALCULATIONS

1. CFD simulation with Frozen Rotor Interface

After verification test start real turbine stage modeling. It is a part of unit which works in power station in Bulgaria. From fig.6 can be seen that this is impulse stage (degree of reaction is about 20%) and some parameters calculated with one dimension theory (at mid span). With this information starts three dimensional simulation. Fig. 7 presents work steps.

On next fig.8 are shown different modifications of the nozzle blades-common view of geometry [6]. The full geometry contains 30 stator blades and 124 rotor blades. The CFD turbine stage model contains four rotor blades and one stator blade. Periodic boundary conditions are also used. The geometry of the blades can be made in CATIA or SolidWorks. The rotor rotates about the Z-axis while the stator is stationary. Periodic boundaries are used to allow only a small section of the full geometry to be modeled. In this simulation is used Frozen Rotor interface, after that is made new transient simulation with a transient rotor-stator interface model. The Frozen Rotor solution is used as an initial guess for the transient rotor-stator simulation [3].

The geometry for three stator modifications and rotor are shown on fig.8. Into stator fluid domain is generated about 250,000 cells and for one rotor passage, $4 \times 62,500 = 250,000$ cells. ICEM-CFD can be used. Turbine stage is with 500,000

hexahedral cells. In CFX-Pre are set turbulence model- Shear Stress Transport [4], working fluid- superheated steam. For latter modelling is used the IAPWS-IF97 (International Association for the Properties of water and Steam-Industrial Formulation 1997) database. The boundary conditions are as follow:

Inlet: velocity distribution in boundary layer is inhomogeneous with boundary layers. Mid span velocity is about 40 m/s. Total inlet temperature 470°C. The inlet turbulence intensity is set to $Tu=5\%$.

Outlet: At outlet plane static pressure is applied 6.78 MPa.

CFX-Post is used to treat calculation results. Fast, without user effort can be checked pressure blade load at different span sections defined by engineer. In CFX-Post can be used mass averaged values of pressure and temperature to calculate isentropic efficiency. Some results are shown on fig.9 to fig.13.

2. CFD simulation with Transient Rotor Interface

In this part of paper is described data preparation for transient simulation [3]. It is necessary to define time interval such that the rotor blades pass through one stator pitch (12°) using 10 time steps. The time step size is calculated as follows:

Rotational Speed = 314.16 rad/s ($n=3000$ rev/min),

Rotor Pitch Modeled = $4 \cdot (2\pi/124) = 0.2027$ rad,

Time to pass through one pitch = $0.2027/314.16 = 6.4516 \cdot 10^{-4}$ s.

Since 10 time steps are used over this interval each time step should be $6.4516 \cdot 10^{-5}$ s.

Results can be seen on fig. 14.

CONCLUSIONS

1. Verification tests with ANSYS CFX software, TU-Vienna experiment description, comparison between both has been made.
2. A CFD Design Methodology has been presented. Real physical object has been modeled.
3. Comparison between turbine stages with different stator blades modifications (Lean, Inverse lean, Compound Flow Nozzle and Control Flow Nozzle) has been made.
5. Turbine stage unsteady transient simulation has been made.
6. Computational Fluid Dynamics shows that is possible to increase efficiency by 0.6%.

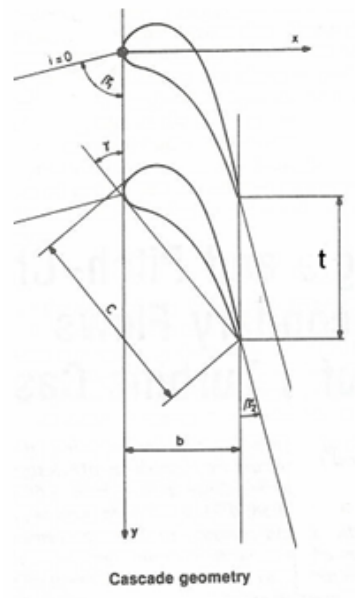
ACKNOWLEDGMENTS

The work described in this paper formed part of a Ph. D. Thesis at the Technical University – Varna, Bulgaria. The author wishes to thank for the teaching and is particularly grateful to his supervisor Ao. Univ. Prof. Dr. Reinhard Willinger (Austria). The work was carried out with the support of the Austrian Ministry of Education, Science and Culture with Scholarship ERNST MACH for period 03.03.2008-30.06.2008 and the staff of Vienna University of Technology, Institute for Thermodynamics and Energy Conversion.

REFERENCES

1. Beer W., Willinger R. Numerical Optimization of a HP Steam Turbine Blade Using a Genetic Algorithm. The 13th International Conference on Fluid Flow Technologies. Budapest, Hungary, September 6-9, 2006.
2. Perdichizzi A., Dossena V. Incidence Angle and Pitch-Chord Effects on Secondary Flows Downstream of a Turbine Cascade, Journal of Turbomachinery, Vol. 115. Transactions of the ASME, July 1993.
3. ANSYS CFX. Tutorials. ANSYS CFX Release 11.0, December 2006.
4. Хитрых Д. Проектирование турбомашин: обзор моделей турбулентности. ANSYS Solutions. Русская редакция, осень, 2005.
5. Kaiser J. ANSYS Workbench Makes Simulating FSI Easier. Enhancements in version 11.0 streamline setup of fluid structure interaction problems. ANSYS Solutions, Volume 6, Issue 5, 2006.
6. ALSTOM Project No. 303: Summary Report: March 2005.

FIGURES



CASCADE GEOMETRIC DATA

Height, $h=150$ [mm]

Pitch, $t=73$ [mm]

Chord length, $c=100$ [mm]

Axial chord length, $b=61.85$ [mm]

Aspect ratio, $AR=h/c=1.5$

Inlet blade angle (tangential), $\beta_1'=76.1^\circ$

Outlet blade angle (tangential), $\beta_2'=14.5^\circ$

Stagger angle, $\gamma=39.9^\circ$

INLET BOUNDARY LAYER

Thickness, $\delta=22.9$ [mm]

Displacement thickness, $\delta^*=2$ [mm]

Momentum thickness, $\delta^{**}=1.5$ [mm]

Form factor, $H=\delta^*/\delta^{**}=1.34$

Mach number, $M<0.2$

Reynolds number, $Re_2 \times 10^6=0.5$

Solidity $c/t=1.37$

Fig. 1. Linear cascade characteristics description [1].

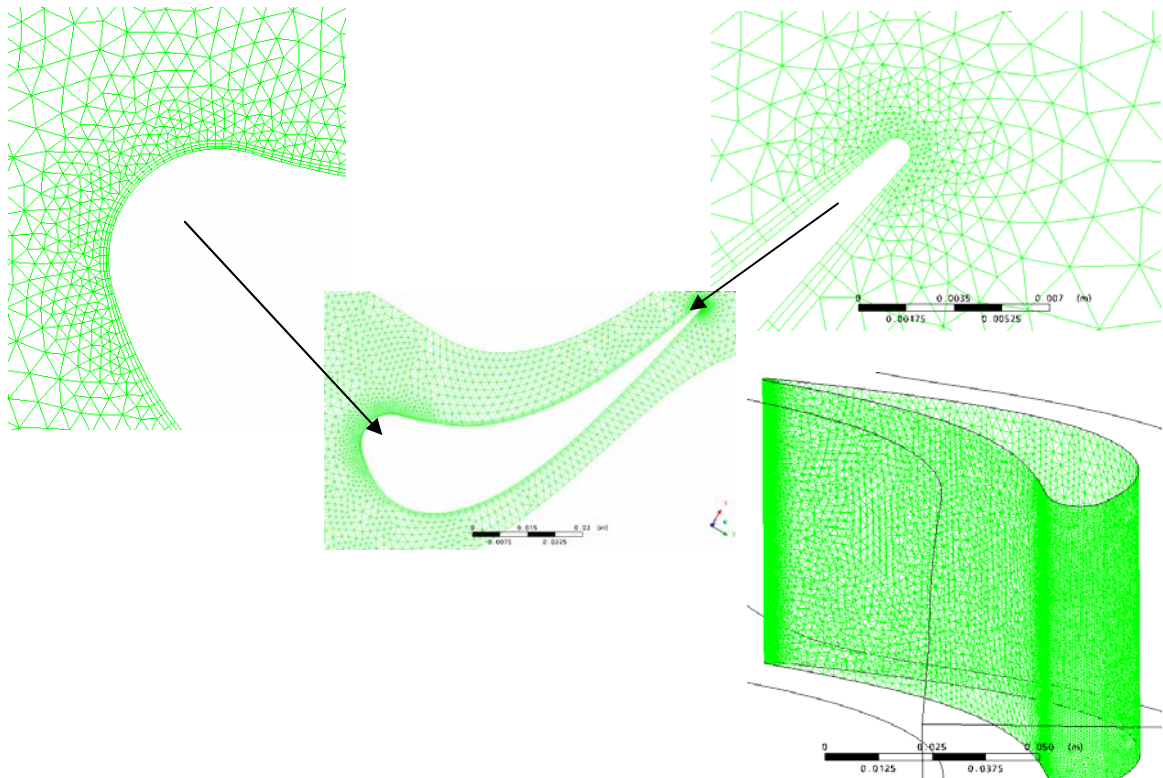
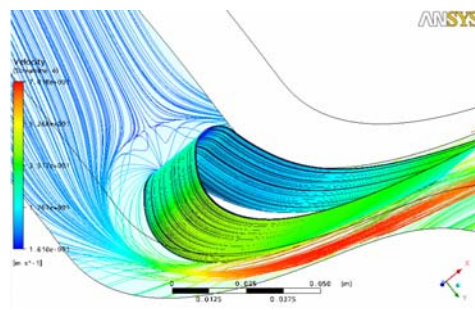
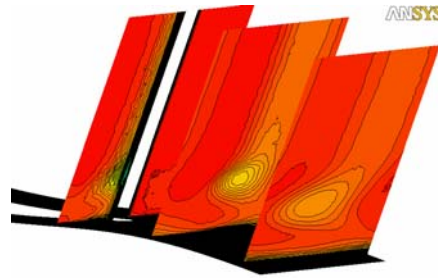


Fig. 2. Unstructured mesh.

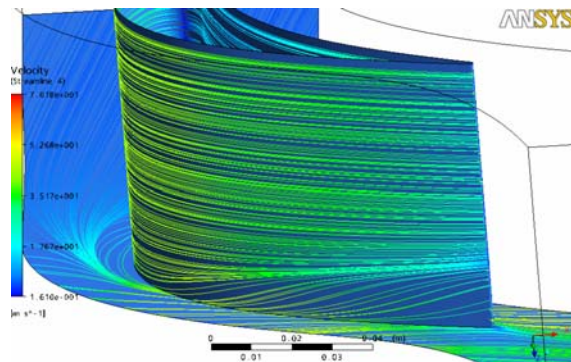
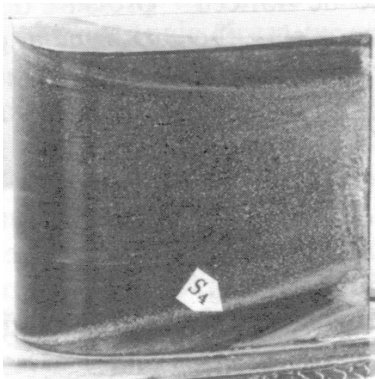
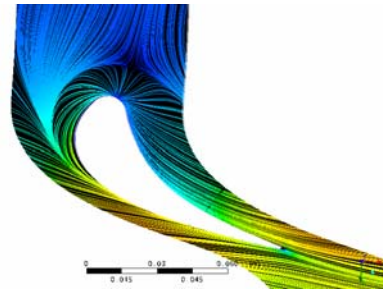
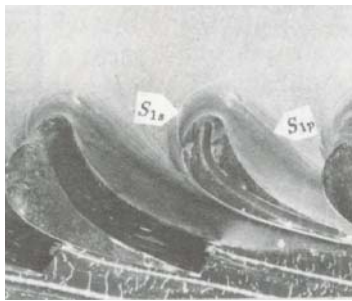


a)



b)

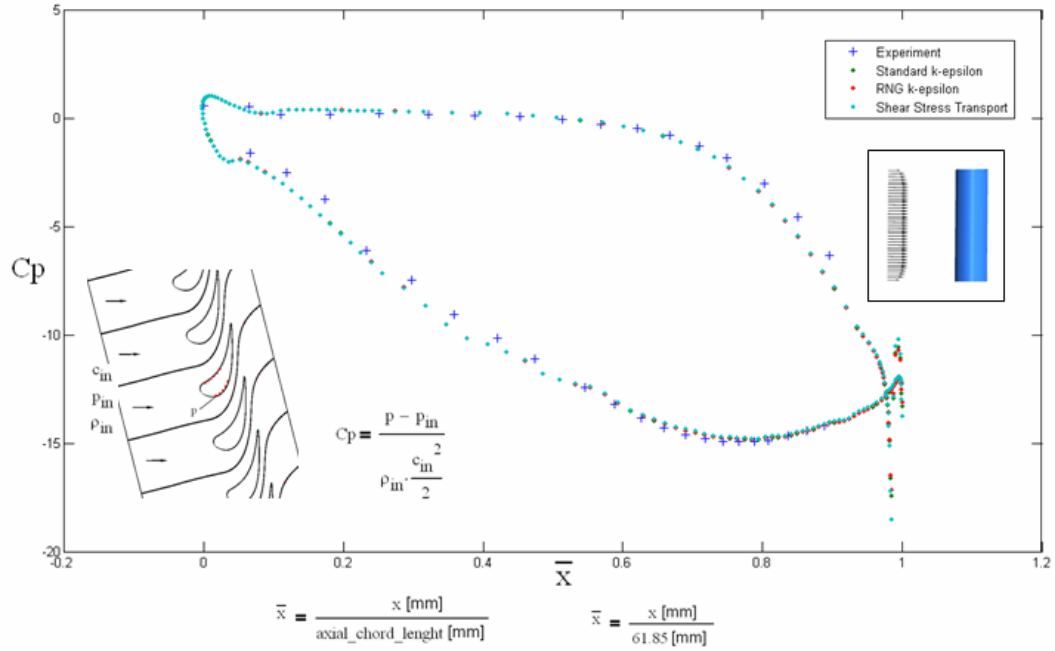
Fig. 3. Some results from CFX-Post: (a) streamlines surface and three dimensional secondary flow at hub, (b) total pressure contour lines (yellow color is vortex core, only lower half blade part is shown).



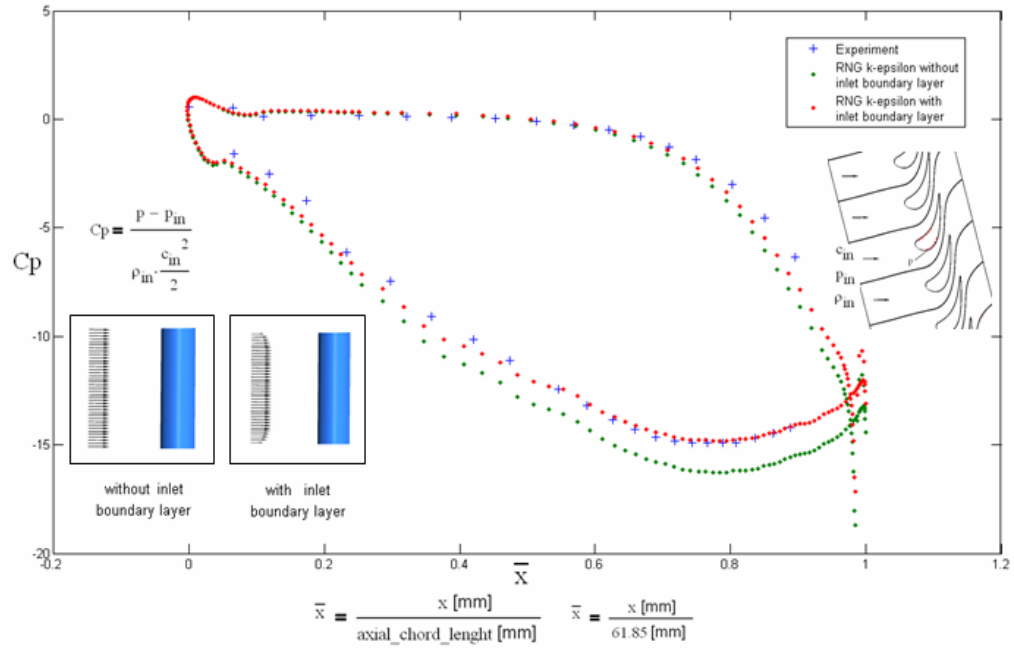
a)

b)

Fig.4 . Comparisons between experiment (a) and CFD calculation (b) (only lower half blade part is shown).

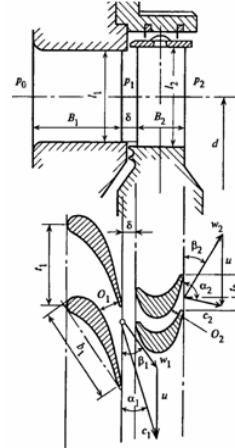


a)



b)

Fig. 5. Static pressure coefficient distribution on blade surface at mean section: (a) comparison between experimental data (+) and CFD calculations with different turbulence models, (b) comparison between experimental data (+) and two CFD calculations with RNG k-epsilon turbulence model-with inlet boundary layer and without inlet boundary layer.

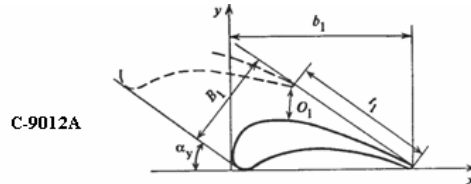
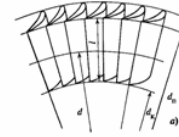


Flow path of axial turbine stage and cross section at middle diameter.
O1-stator throat. O2-rotor throat

	Pressure, [MPa]	Temperature, [C]	Specific Volume, [m ³ /kg]	Enthalpy, [kJ/kg]	Entropy, [kJ/(kg·K)]
point 0	7.5720	466.80	0.04199	3321.67	6.6485
point 0'	7.5900	467.20	0.04191	3322.50	6.6485
point 1	6.9130	452.55	0.04504	3295.60	6.6512
point 1s	6.9130	451.75	0.04498	3293.60	6.6485
point 2s	6.7780	449.15	0.04584	3289.10	6.6512
point 2	6.7780	450.00	0.04594	3292.00	6.6552
point 2'	6.7980	450.80	0.04582	3292.85	6.6552

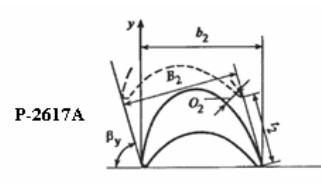
Annular rows geometrical parameters

Middle diameter, $d_{mid}=0.877$ [m]
Hub diameter rotor, $d_{hub}=0.810$ [m]
Shroud diameter rotor, $d_{shr}=0.944$ [m]
Stator blade height, $l_1=0.064$ [m]
Rotor blade height, $l_2=0.067$ [m]
Degree of reaction at root, $\rho_{root}=5.77\%$
Degree of reaction at middle section, $\rho_{mid}=18.3\%$
Degree of reaction at shroud, $\rho_{shr}=28.43\%$
Axial Width Stator Blade $B1=80$ [mm]
Axial Width Rotor Blade $B1=30$ [mm]
Axial gap between Stator and Rotor blades, $\delta=9.2$ [mm]



Stator profile geometry at mean diameter

Throat = 16.0864 [mm]
LE Pitch = 91.7345 [mm]
TE Pitch (S) = 91.7345 [mm]
Camber Length = 159.209 [mm]
Cord Length (C) = 142.689 [mm]
Stagger Angle = -55.9°
Solidity (C/S) = 1.55546
Pitch Cord Ratio (S/C) = 0.642896



Rotor profile geometry at mean diameter

Throat = 6.49982 [mm]
LE Pitch = 22.1938 [mm]
TE Pitch (S) = 22.1938 [mm]
Camber Length = 46.2428 [mm]
Cord Length (C) = 30.9168 [mm]
Stagger Angle = 14.0°
Solidity (C/S) = 1.39304
Pitch Cord Ratio (S/C) = 0.717856

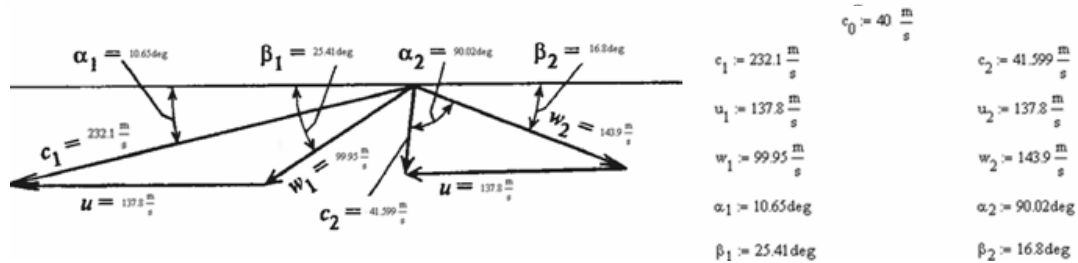


Fig. 6. Some thermodynamical, geometrical and kinematical characteristics of the turbine stage.

CFX-PRE
CFX-SOLVER

CFX-POST **MATLAB** **MATHCAD**

Fig. 7. Work flow in ANSYS CFX.

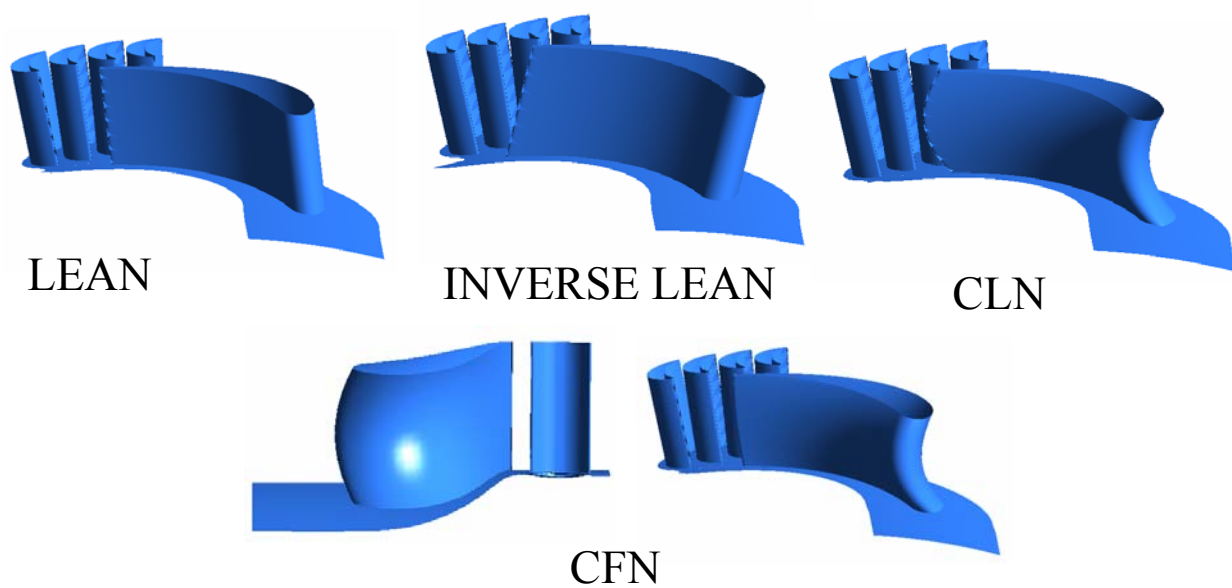


Fig. 8. Overview of some of modifications.

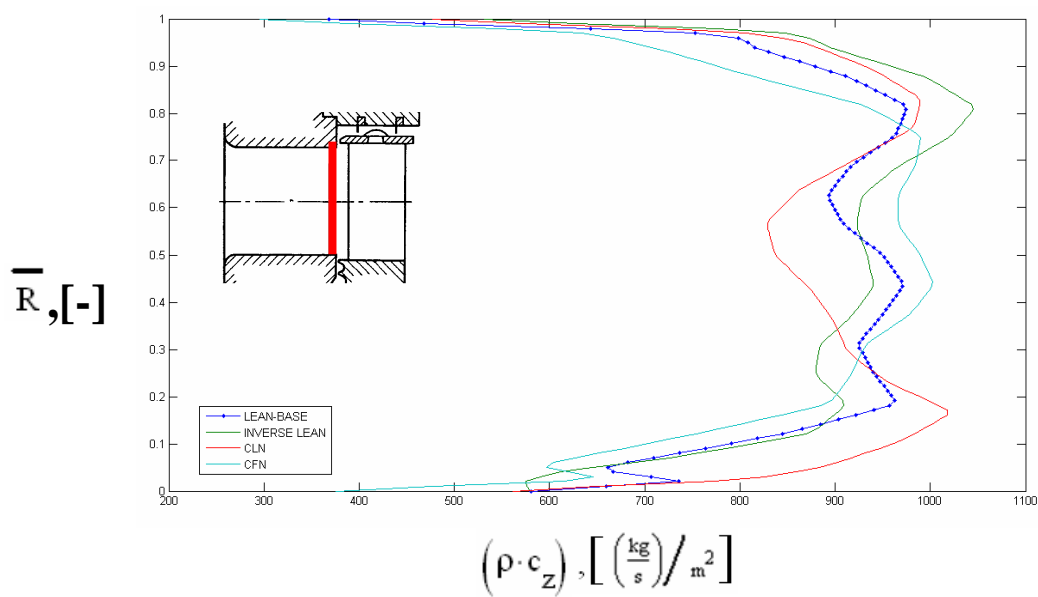
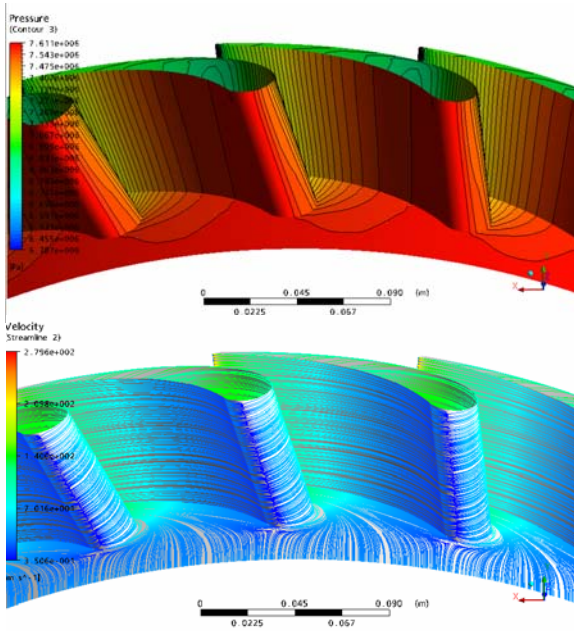
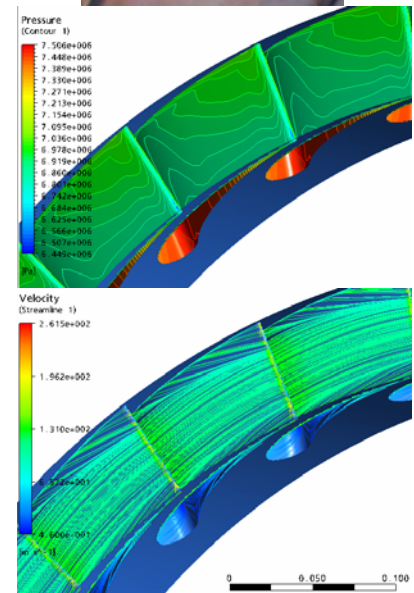


Fig. 9. Specific mass flow at Stator Outlet (Streamwise=0.98). Comparison between three modifications and base construction.



a)



b)

Fig.10. Some CFX-Post results: (a) Stator blade Leading Edge (LE) view- prediction of pressure contours and surface streamlines, (b) Stator blade Trailing Edge (TE) view- prediction of pressure contours and surface streamlines.

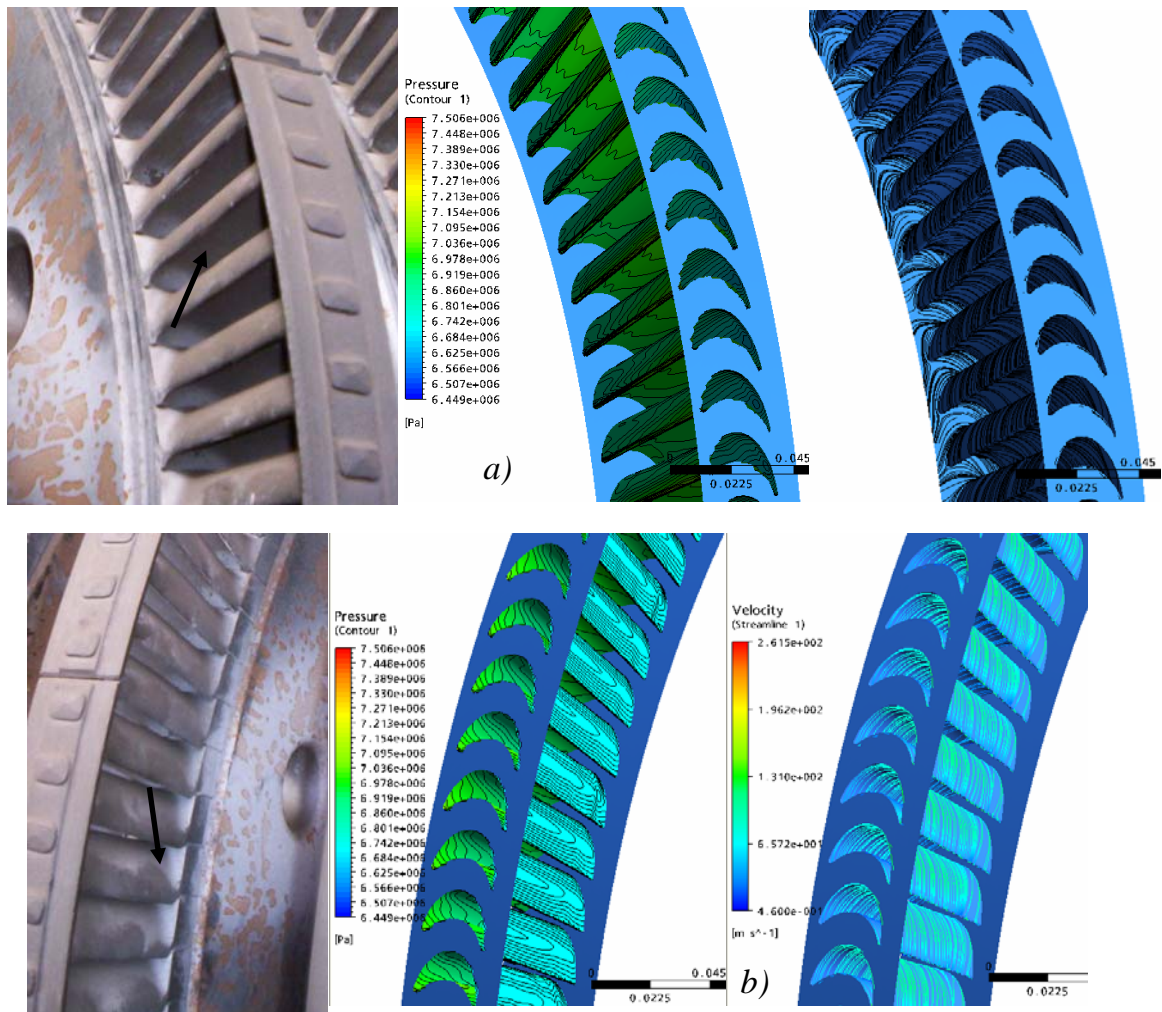


Fig. 11. Some CFX-Post results: (a) Rotor blade Leading Edge (LE) view- prediction of pressure contours and surface streamlines, (b) Rotor blade Trailing Edge (TE) view- prediction of pressure contours and surface streamlines.

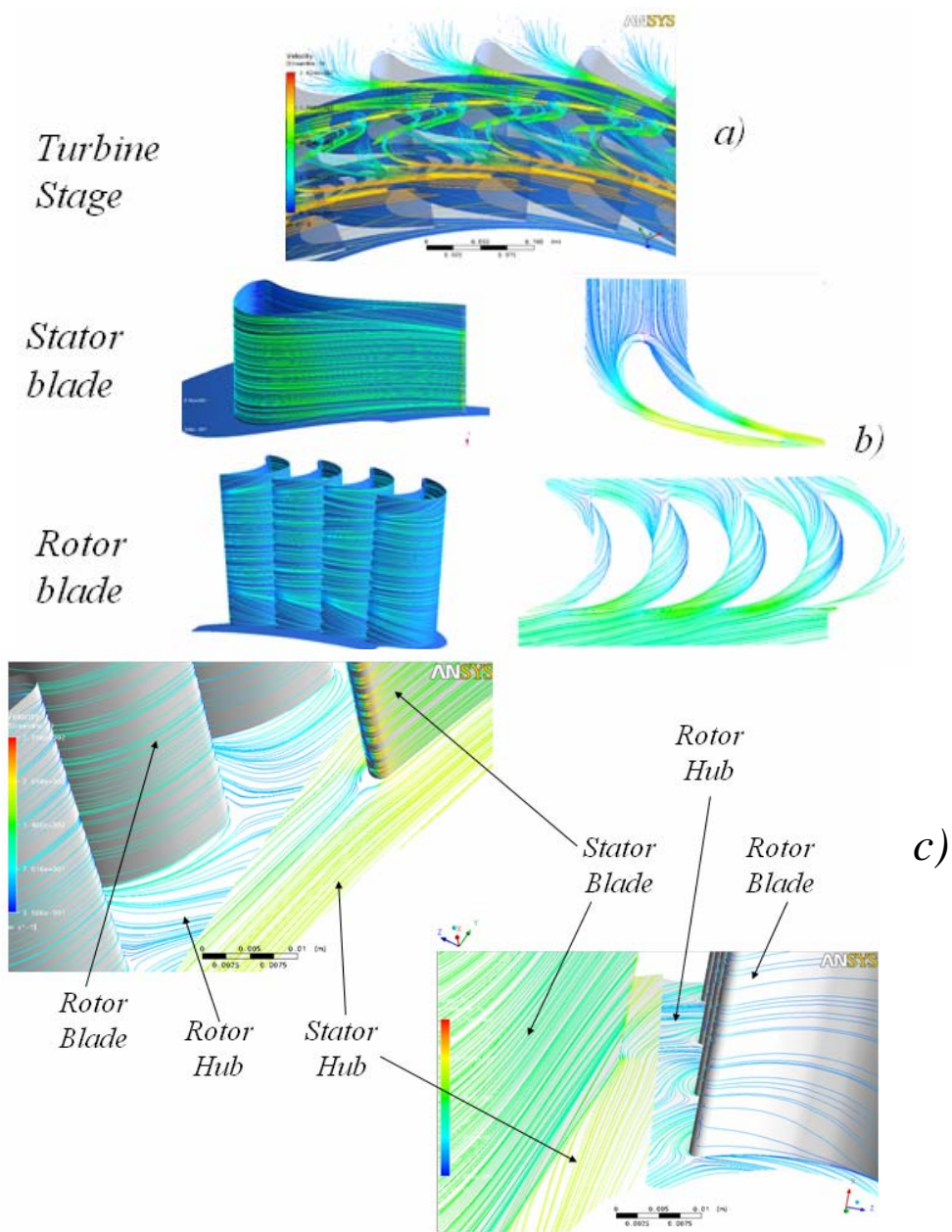


Fig. 12. Secondary flow visualization near the shroud and hub turbine blades: (a) Three-dimensional streamlines, (b) Surface streamlines stator and rotor blades, stator and rotor hub, (c) Surface streamlines between blades at interface region.

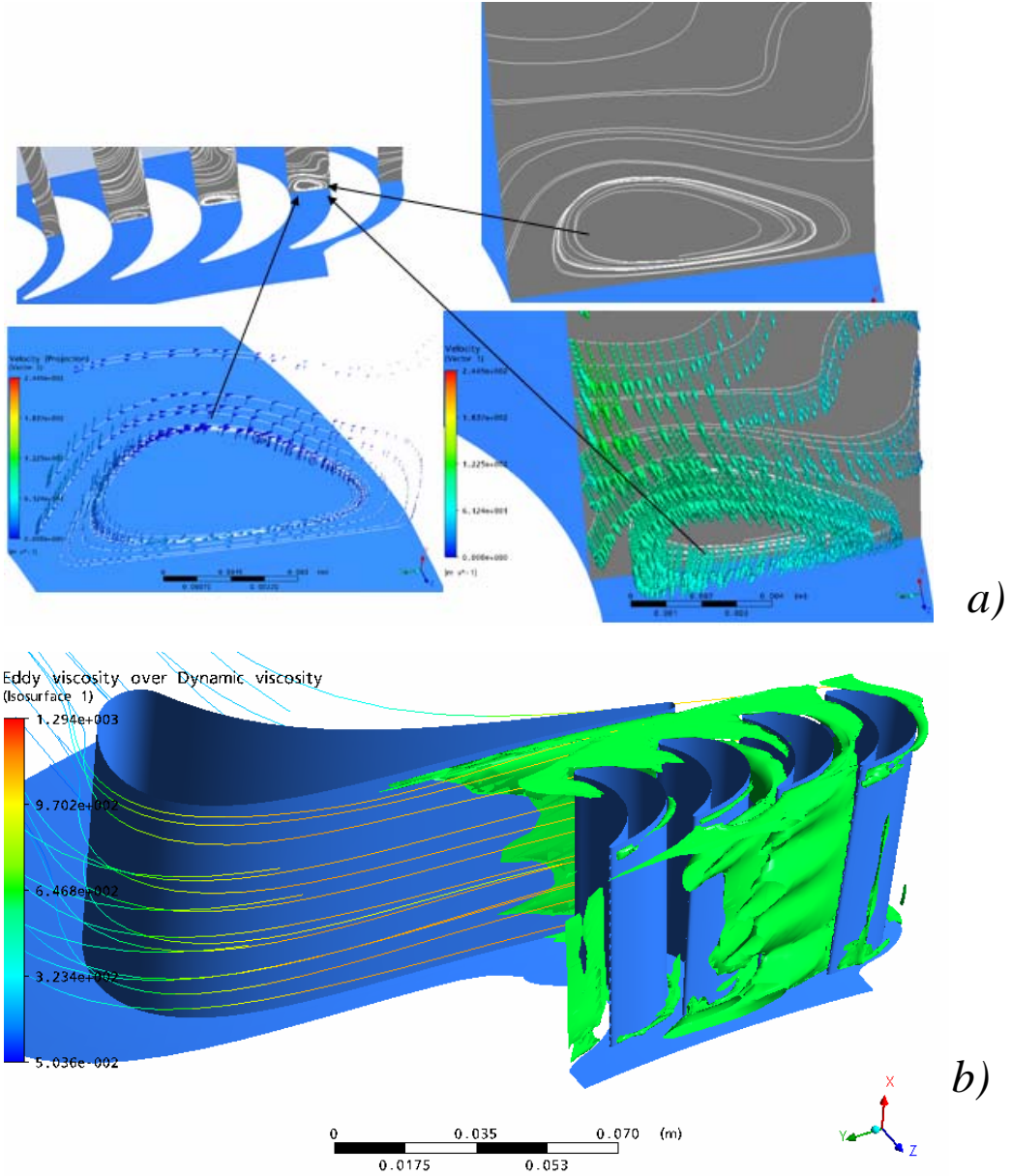


Fig. 13. Secondary vortices at rotor fluid domain: (a)-Surface streamlines at plane, defined in rotor fluid domain. Secondary vortex, and velocity on this plane with vortex at hub, (b)-Ratio eddy viscosity/dynamic viscosity isosurface.

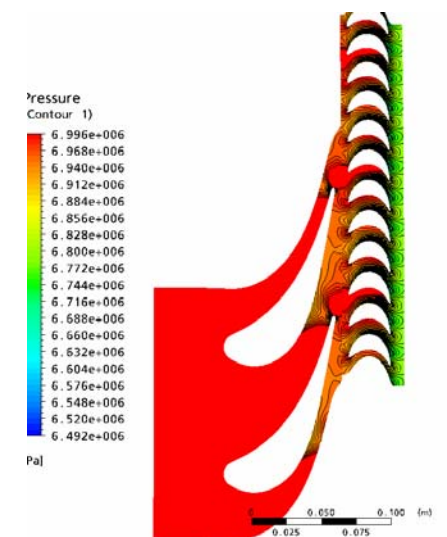
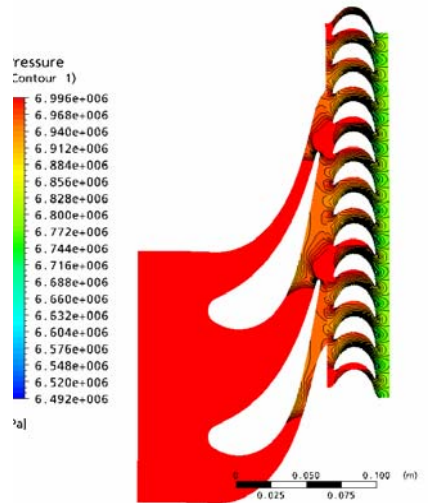
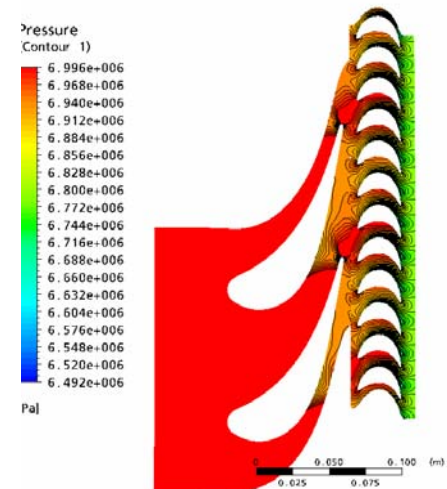
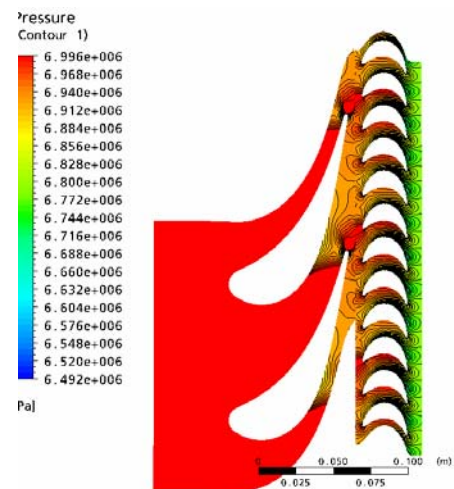


Fig. 14. Pressure contour change with the time.