

NUMERICAL INVESTIGATIONS OF THE TURBINE LAST STAGE – EXHAUST HOOD FLOW

A. Gardzilewicz

Turbine Aerodynamics Department,
Institute of Fluid Flow Machinery, Fiszer 14, 80-952 Gdańsk, Poland, gar@imp.gda.pl

J. Badur

Thermo-Chemical Power Department,
Institute of Fluid Flow Machinery, Fiszer 14, 80-952 Gdańsk, Poland, jb@imp.gda.pl

M. Karcz

Thermo-Chemical Power Department,
Institute of Fluid Flow Machinery, Fiszer 14, 80-952 Gdańsk, Poland, mkarcz@imp.gda.pl

J. Swirydczuk

Turbine Aerodynamics Department,
Institute of Fluid Flow Machinery, Fiszer 14, 80-952 Gdańsk, Poland, jsk@imp.gda.pl

ABSTRACT

The paper compares two different approaches, based on CFD calculations, that were employed for the problem of studying flow phenomena in the area covering the LP turbine last stage and exhaust hood. The first approach makes use of a so-called coupling method, in which the flow calculations in the last stage and in the exhaust hood are carried out separately in an iterative manner until a proper correspondence is obtained between the total and static pressures at the coupling plane between the two areas of interest.

The second approach is a direct flow calculation in the entire region of interest. This approach models the overall performance of the last stage/exhaust hood set, losing however, in accuracy in the last stage stator and rotor blading areas, in which numerous unfavorable flow effects, like flow separations and shock waves, can be observed. Therefore its application to the detailed studies of the flow in this region is rather limited. Nevertheless, the comparison of these two methods allows better assessment of the results obtained in each of them individually.

INTRODUCTION

The flow of steam at exit of the large-power turbine is highly non-uniform so it becomes one of the most challenging problems in fluid dynamics (Perycz, 1961; Traupel, 1967). The exhaust hood has always a complex geometry making the flow change direction by 90°. When studying the flow in this area a complex set of flow properties and processes have to be taken into account, including: viscosity, compressibility and shock waves, heat transfer with phase transition, changing thermodynamic properties of the steam at the presence of water, and unsteady flow effects connected with vibrations of rotating blades and stiffening elements (Łuniewicz et al., 1997; Gardzilewicz et al., 2002; Gardzilewicz et al., 2003b).

That was why designers of this turbine part make wide use of model studies, most often performed on air stands (Stastny and Feistauer, 1978; Ratliff et al., 1993; Biswas and Stetter, 1994;

Zariankin, 1995). Some measurements of flow parameters at turbine exit were also performed on real turbine sets in power stations (Gardzilewicz et al., 2003a). All of those studies were performed using, as a rule, numerous simplifying assumptions, nevertheless they provided an opportunity to make a rough estimation of flow losses and validation data for computations.

The schematic view of the LP turbine/exhaust hood system with the locations of measuring probes is shown in Fig.1, while the selected results of pressure distribution measured at turbine exit are given in Fig. 2. In this diagram only six circumferential positions of probe are marked.

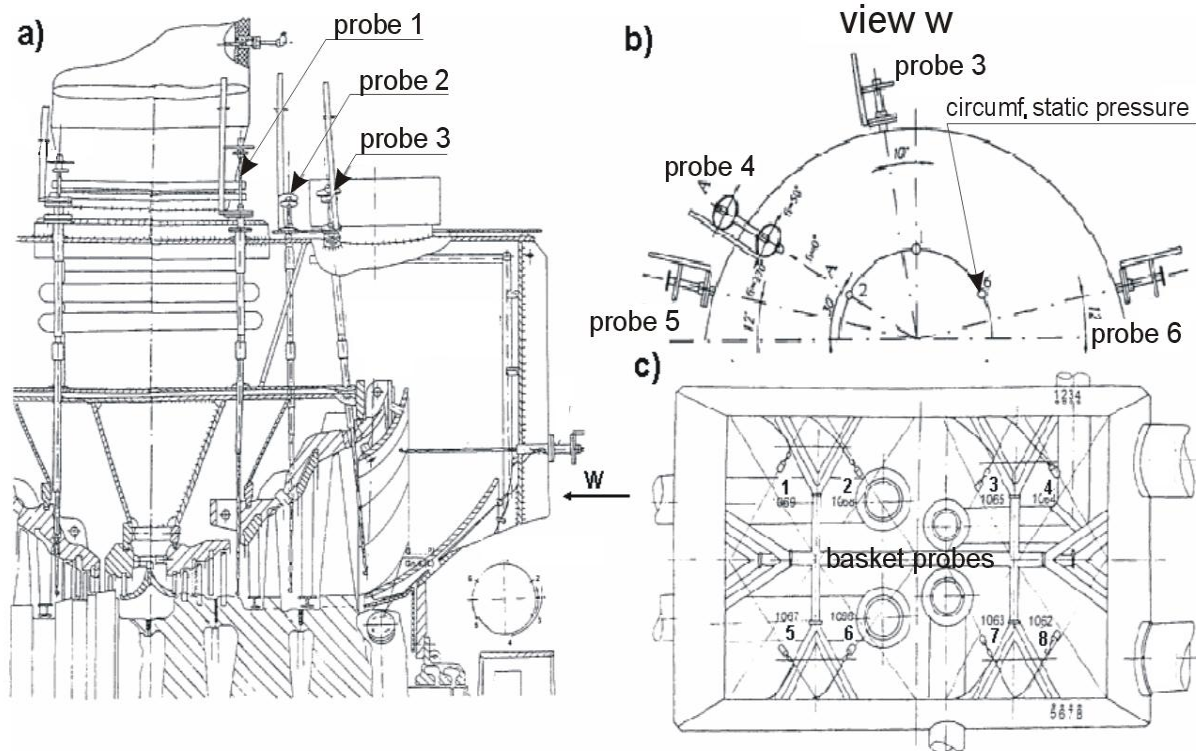


Fig. 1. Measuring instrumentation in LP part of 18K-360 turbine:
 a) turbine axial cross-section, b) exhaust hood inlet circumferential cross-section,
 c) exhaust hood exit cross-section – in front of condenser

Additionally, Fig. 3 presents radial distributions of velocity and flow angles at the turbine last stage exit.

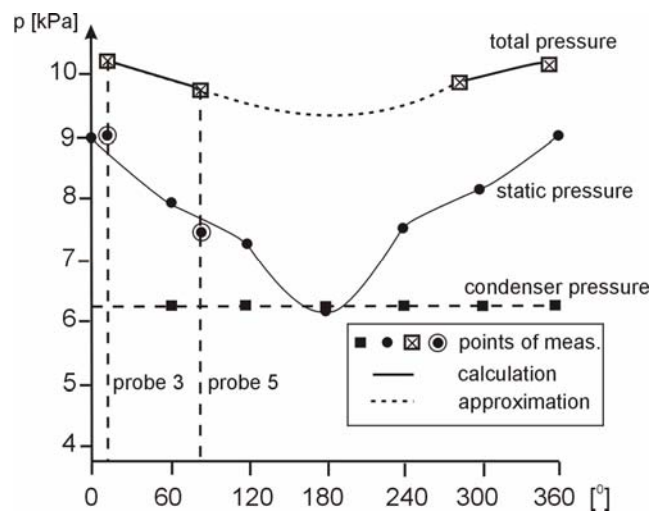


Fig. 2. Circumferential distributions of static and total pressure at the last stage exit

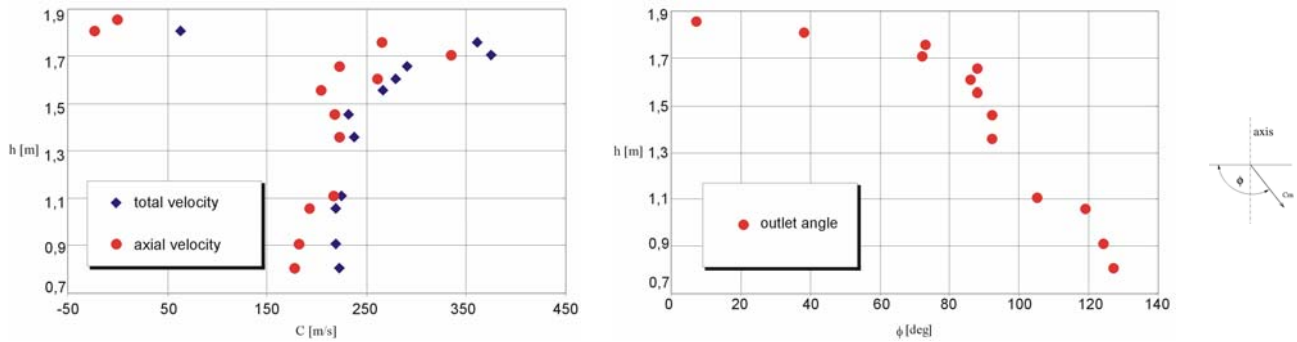


Fig. 3. Radial distributions of velocity, and flow angles at the 360 MW turbine last stage exit

The operation of an exhaust hood is mainly based on the proper estimation of the kinetic energy loss coefficient, which depends both on the geometry and flow parameters, especially velocity, changing with the turbine set load change (Gardzilewicz, 2003b).

Computational fluid dynamics provides new opportunities in exhaust hood investigation and design processes. Practical solution of the set of the conservation equations applied to fluid-flow machines has been the motivation for implementing modern CFD procedures to the design of steam turbine exhaust hoods in the nineties of the last century (Ratliff et al., 1993; Dejean et al., 1999; Kardaś and Gardzilewicz 1999; Gardzilewicz and Solodov, 1999). However, some numerical methods used in the exhaust hood design should be verified and tested.

COUPLED TURBINE STAGE / EXHAUST HOOD FLOW CALCULATIONS

High cost of experimental investigations, together with numerical problems concerning simultaneous calculations of the flow through the last stage and exhaust hood diffuser have been the main motivation for implementing a procedure of separate flow calculations in these two areas. This procedure, known as a coupling method, offers high range of flexibility in creating computational meshes which better take into account significant differences in dimensions of basic elements in the turbine stage and in the exhaust hood. Its disadvantage is connected with high number of calculations to be carried out before a final solution is obtained. The accuracy of multiple passing flow parameters between the last stage exit and the exhaust hood inlet is also an open question.

The calculations in the turbine last stages and in the exhaust hood are performed in an external iteration process, shown schematically in Fig.4., following the principles worked out in ALSTOM (2002; Gardzilewicz et al., 2003b).

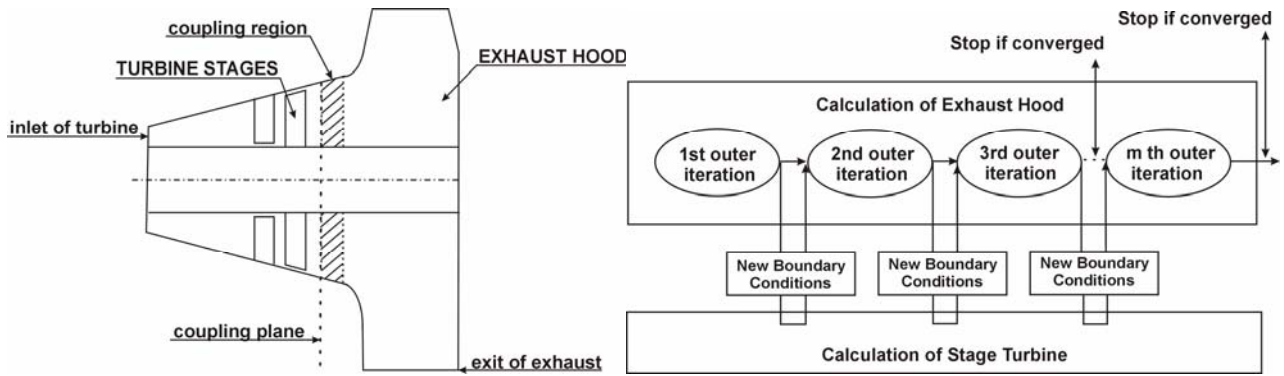


Fig. 5. Algorithm of coupled calculations of the turbine last stage/exhaust hood flow.

Stage calculations are performed, independently of the exhaust hood computations, for a number of variants of circumferentially symmetrical exit distributions of the static pressure, corresponding to total pressure at selected points along the circumference of the exhaust hood inlet. The results of the stage calculations make the data base for correcting circumferentially asymmetrical conditions at the exhaust hood inlet. Such an approach significantly reduces the number of computing variants and the resulting computing time.

This methodology has been checked in the present study on the design outlet of a 200 MW turbine set. In this case the calculations were performed using two CFD codes, namely FlowER and Fluent, the characteristics of which have been given earlier (Gardzilewicz et al., 2003b).

The comparison of pressure distributions on the coupling plane, as obtained in two consecutive iterations, is presented in Fig. 5. The first iteration of the exhaust hood calculation is performed for constant pressure and temperature distribution along the exhaust hood inlet circumference. The convergence between turbine exit and exhaust hood inlet parameters is reached, in practice, after three iterations, which is shown in Fig. 6 presenting the distributions of parameters obtained after iteration I, II, III and IV, after averaging them in radial direction.

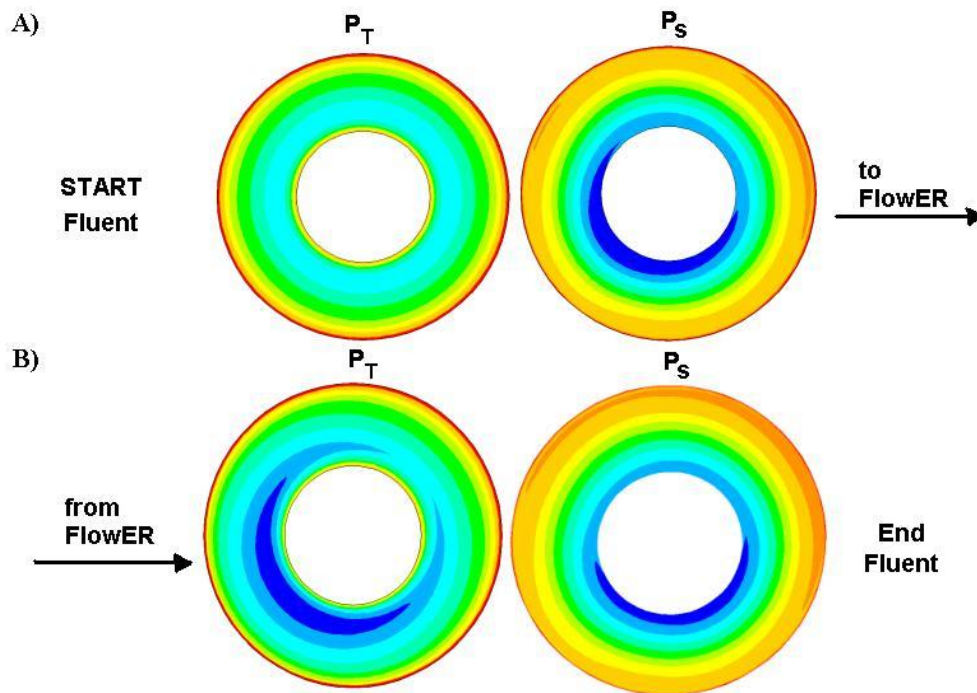


Fig. 5. Distributions of total pressure (left) and corresponding static pressure (right) at diffuser inlet, for iteration I (A) and II (B).

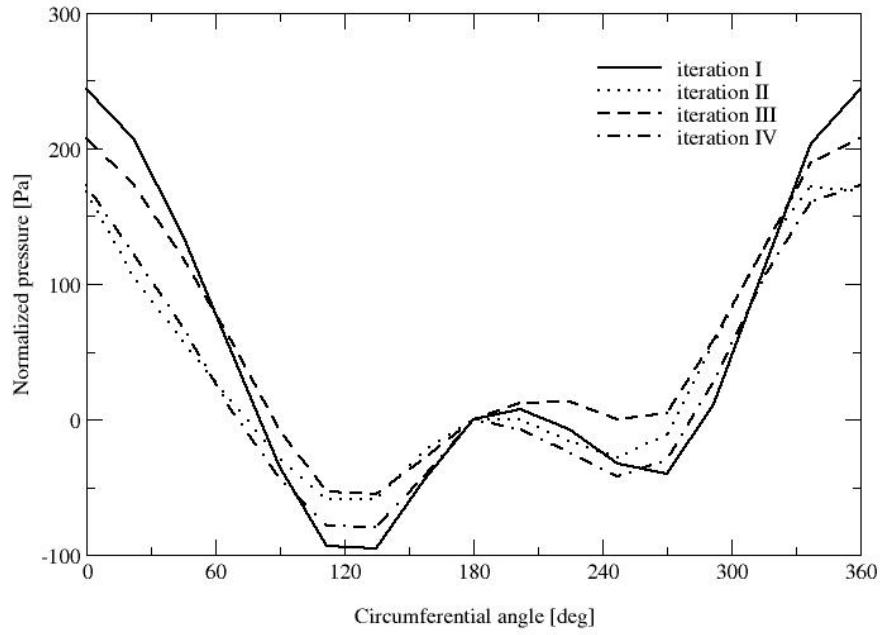


Fig. 6. Comparison of circumferential static pressure distributions in iteration I, II, III and IV (mid-span section).

Quantitative comparison of flow parameters obtained after iteration IV with those passed from FlowER as initial data is given in Fig. 7. In this diagram, the points stand for the data taken from FlowER, while the lines represent the results obtained in Fluent calculations. The presented Fluent distributions are close to their counterparts generated by FlowER, with rather limited improvement observed in successive iterations.

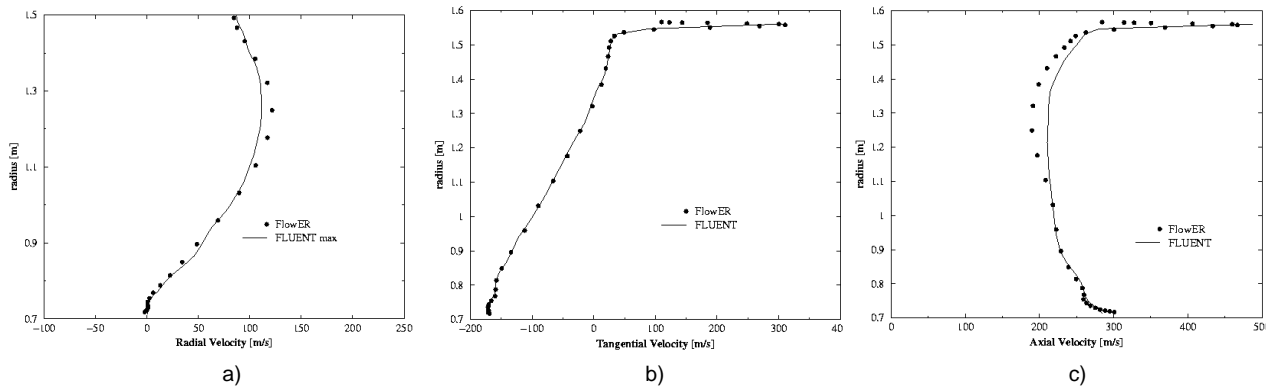


Fig. 7. Comparison of velocity distributions from Flower and Fluent. Velocity component: a) radial, b) tangential, c) axial.

DIRECT CALCULATIONS OF THE FLOW THROUGH TURBINE AND EXHAUST HOOD

Problems with preparing compatible boundary conditions at the coupling plane between the last stage exit and the exhaust hood diffuser inlet make the process of coupling calculations difficult and time consuming. The scale of difficulties is enlarged by other effects, like shock waves, condensation, and flow unsteadiness, with possible flow separation, recorded in the assumed area of the coupling plane location. That was why an attempt was made to perform a direct calculations in the area covering both the low pressure part of the turbine and the exhaust hood. However, if we take into consideration that a standard LP turbine comprises of 8 to 10 rows, each consisting of several dozens of stator and rotor blades, and, at the same time, its exhaust hood has a set of reinforcing ribs and expansion pipes, preparing a mesh which would follow the geometry of such a complicated system with a satisfactory accuracy would require millions of cells. Even for modern multi-processor computers such a task would be a serious challenge.

An example of direct calculations of the flow through turbine and exhaust hood is presented below. This is part of wide investigations concerning unsteady aerodynamic forces acting on rotor in low pressure part of the turbine of 200 MW output (Badur et al., 2003).

Multi-block hybrid meshes were prepared by means of the Gambit grid generator for further flow calculation using Fluent. The geometry modules of the last stage and exhaust hood are presented in Fig.8. The details of numerical procedure and grids have been reported by Badur et al. (2003) and Gardzilewicz et al. (2003b).

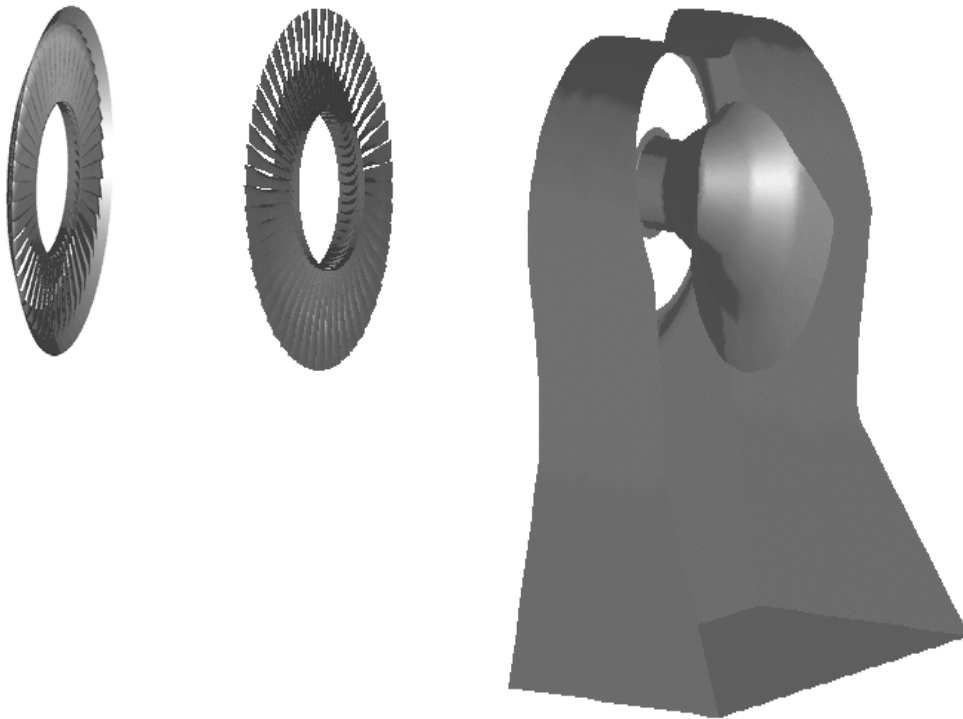


Fig. 8. Geometry of the turbine last stage and exhaust hood area

As a necessary simplification, only inviscid flow of steam was considered, so no turbulence model was employed. The calculations were performed using a sliding mesh technique, in which sliding interfaces are set between last-stage stator and rotor blades, as well as between last-stage rotor blades and the exhaust hood. Unlike the coupling method case, this technique requires an unsteady mode of computations. Initial data for the calculations were taken from the results obtained by FlowER and selected measurements. The measurement data were especially helpful for

estimating real turbulence properties. The level of turbulence Tu in real turbosets at the last stage/exhaust hood interface can even be as high as 10÷20%. Those values have been obtained by Gardzilewicz et al. (2003a) on the basis of measured pressure fluctuations at different turbine loads: 50, 75 and 100% of nominal load (Fig. 9). The magnitude of pressure fluctuations is strongly related to the flow velocity. The high values of pressure pulsations were measured for higher loads. The maximum of pulsation intensity was observed at the blade tips, which can be seen in Fig. 9.

The results of flow calculations of the full geometry by means of stream lines are shown in Fig. 10. The presented flow patterns reveal a complicated structure inside the exhaust hood – similar results were presented earlier (Gardzilewicz and Solodov, 1999; Gardzilewicz et al., 2003b). The full unsteadiness of flow between the last stage and exhaust hood was confirmed by measurements (Gardzilewicz et al. 2003a).

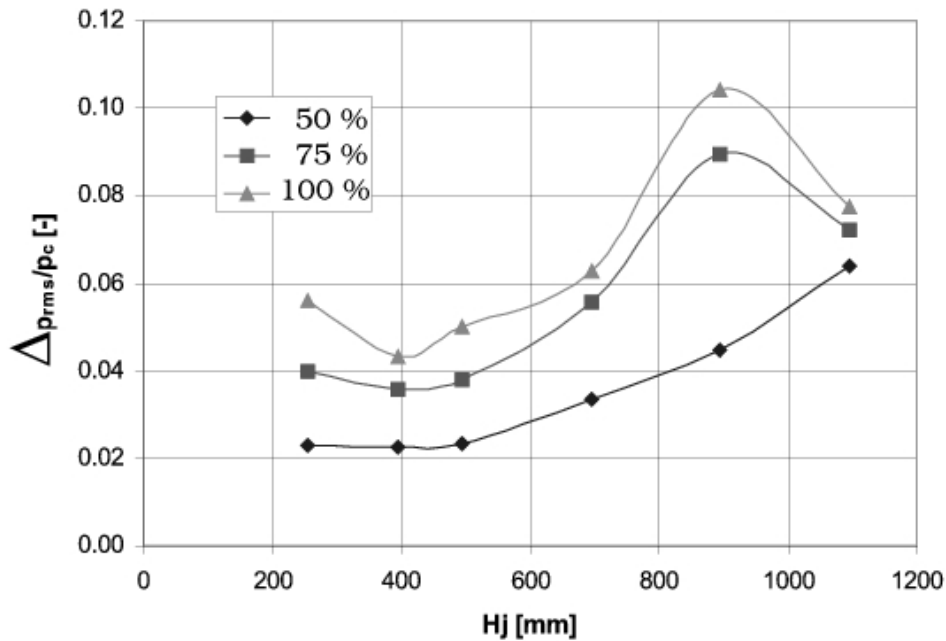


Fig. 9. RMS value of the total pressure pulsation (Δp_{rms}) related to the total pressure averaged value (p_c) at different heights of the last stage blades (H_j)

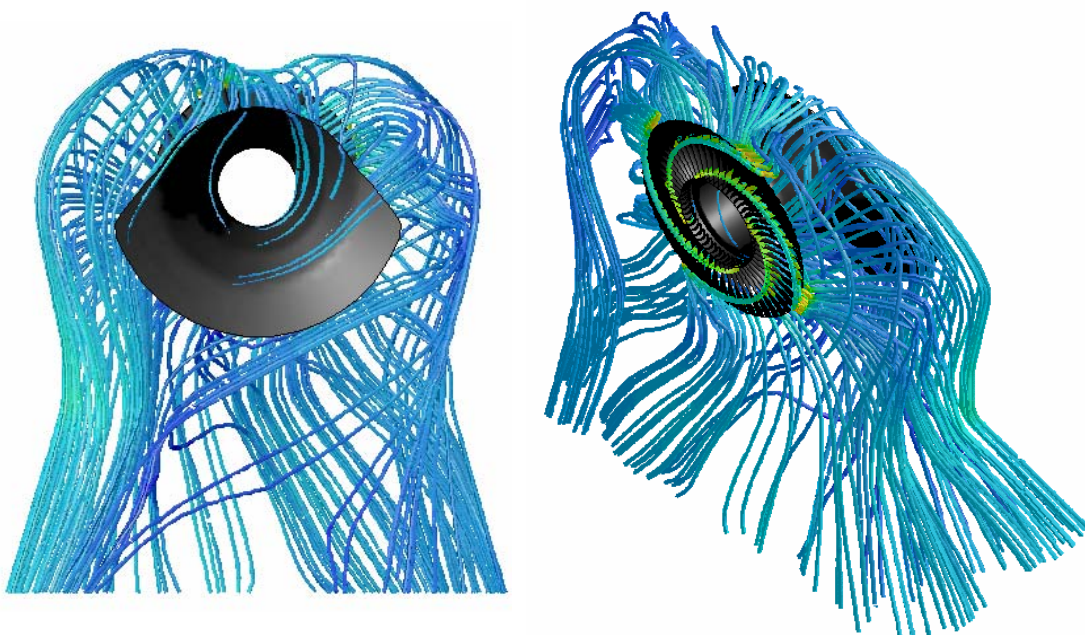


Fig. 10. Streamline pattern inside the exhaust hood and last stage

The corresponding static and total pressure distributions at the interface between the rotor blades and exhaust hood are shown in Fig. 11. The comparison of Fig. 5 and Fig. 11 reveals some differences between the direct and coupling methods of calculation of pressure fields behind the turbine last stage. For the direct method, the non-uniformity of the pressure field due to the presence of rotor blades is clearly visible. The coupling method could produce similar results only in the case of a much larger number of points for data transferred between FlowER and Fluent solvers.

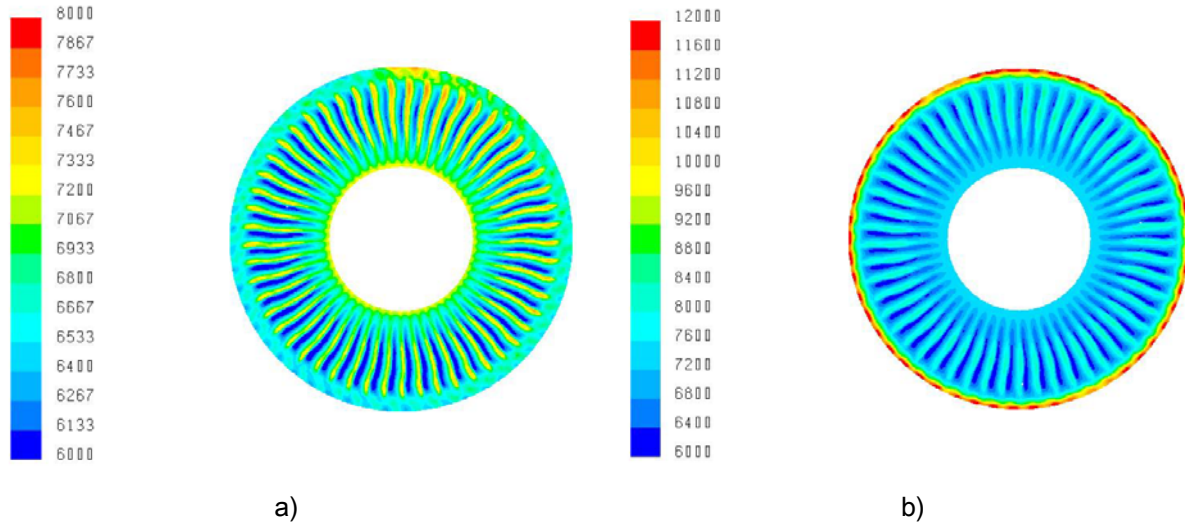


Fig. 11. Pressure distribution on the surface located between the last stage and exhaust hood: a) static pressure, b) total pressure

Pressure distribution along the circumference of the last stage exit/ exhaust hood inlet section, obtained for direct and coupling calculations (iteration IV), is shown in Fig.12. It is noteworthy that these two methods give qualitatively similar results. Especially the ranges of normalized pressure changes correspond very well. The shape of curves for both methods reveals that the pressure distribution behind the last stage is not constant and some non-uniformities in the form of those shown in Fig.12 should be taken into account when a computations are performed in this area.

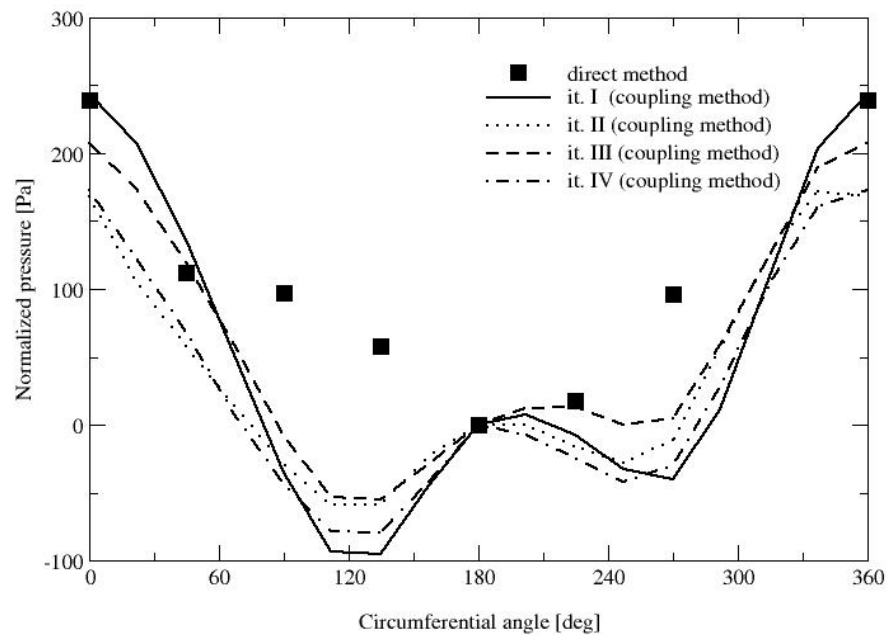


Fig. 12. Comparison of circumferential static pressure distributions obtained from coupled and direct methods (mid-span section).

CONCLUSIONS

- The complicated nature of strongly asymmetrical and whirled flow forces performing the last stage/exhaust hood calculations in a simplified manner, with respect to both the geometry and physical properties of the flow.
- In this context both presented methods are complementary to each other, allowing the user gaining deeper insight into the nature of examined flow.
- Despite the imposed simplifications, CFD calculations offer designers a practical advantages in a form of output power gains, elimination of separation zones, and vibrations of stiffening elements. Those advantages are confirmed by experimental investigations and measurements, both on model stands and real turbines in operation.

ACKNOWLEDGEMENTS

The authors would like to express their gratitude to Alstom Power Ltd., Elblag, Poland for agreement to publish some results of investigations.

REFERENCES

1. Badur J., Solodov V.G., Karcz M., Kucharski R., (2003), The results of investigations on unsteady aerodynamic forces acting on rotor in low pressure part of the 13K235 turbine, Internal reports of IFFM PAS-ci, 3345/03,
2. Biswas R, Stetter H, (1994), Experimental investigation on axial exhaust-duct models of steam turbine, Proc. of 10th Conf. Steam and Gas Turbine, ed. M. Stastny, Karlove Vary, 19-28
3. Dejean F., Mart C., Hartman O., (1999), Three-dimensional coupled flow calculation in the low pressure steam turbine last stage and exhaust hood for nominal and partial load, Proc. European Conf. of Turbomachinery, London,
4. Gardzilewicz A., Badur J., Karcz M., Swiryczuk J., (2002), Unsymmetrical flow in the last stage due to interaction with an exhaust hood of steam turbine, 16-th *Seminar on Turbomachinery*, Dresden'2002,

5. Gardzilewicz A., Marcinkowski S., Adamkowski A., Janicki W., Sobera H., (2003a), Measuring of unsteady pressures at the steam turbine exit, *Transactions of the IFFM PAS-ci*, **113**, 169-178
6. Gardzilewicz A., Solodov V.G., (1999), Spatial unsteady flow pattern in an old-design 200MW turbine exhaust hood and ways of its improvement, *Cieplne Maszyny Przepływowe - Turbomachinery*, **115**, 139-148
7. Gardzilewicz A. Swirydczuk J., Badur J., Karcz M., Werner R., Szyrejko Cz., (2003b), Methodology of CFD computations applied for analyzing flows through steam turbine exhaust hoods, *Transactions of the IFFM PAS-ci*, **113**, 157-168
8. Kardaś D., Gardzilewicz A., (1999), CFD calculations of the steam turbine hood, *Cieplne Maszyny Przepływowe - Turbomachinery*, **115**, 185-190
9. Łuniewicz B., Gardzilewicz A., Zariankin A., Sołodow W., Marcinkowski S., Sowa R., Karbowski M. (1997), Modernisation of the 200 MW turbine exhaust hood, Proc. of Conf. Modeling and Design Fluid Flow Machinery, eds. J. Badur et al., Gdańsk,
10. Perycz S., (1961), The effect of flow efficiency in diffuser on exit energy losses in steam turbine, BWK 13 Heft 9, (in German),
11. Ratliff P. et al., (1993), Retrofit to days technology to operating steam turbines, Report of Westinghouse,
12. Stastny M., Feistauer M., (1978), Flow in annular axial radial diffuser, ASME Paper 78 GT 133,
13. Traupel W., (1967), Thermal turbomachinery, Band 3, Ed. Springer, Berlin, (in German),
14. Zariankin A., (1995), Analiz konstrukcji wychłopnych patrubkow parowych turbin i wazmoznost snizenija ich soprotiwlenija Tieploenergetika, **3**, (in Russian),
15. Internal Reports of ALSTOM Power, 2002