IMPROVEMENT OF A TURBINE EXHAUST HOOD AND DIFFUSER PERFORMANCE WITHIN SPATIAL LIMITATIONS

N.I. Kuklina[1], M.V. Smirnov [2], A.A. Sebelev [1], E.A. Volkov, N.A.Zabelin [2]

[1] Engineering Center "Center of Computer-Aided Engineering" of Peter the Great St. Petersburg Polytechnic University (SPbPU), St. Petersburg, Russia, kuklina@compmechlab.ru

[2] Peter the Great St. Petersburg Polytechnic University (SPbPU), St. Petersburg, Russia, m.smirnov.turbo@gmail.com

ABSTRACT

Gas turbine diffusers and exhaust hoods for power generation units are widely studied nowadays. Numerous designs exist as well as empirically derived ratios for optimal performance. Unfortunately, very few of them are appropriate while facing marine gas turbine exhaust hood design problem. The main restriction in their application is tightly limited space for diffuser and exhaust hood. Classical volute-type hoods, which are satisfactory for land power generation units with almost no space limitations, show poor performance in maritime power systems application. The main challenges are: 1) high pressure loss in hood; 2) high static pressure nonuniformity at the turbine outlet which may cause instability and failures; 3) low pressure recovery coefficient. Bend-type (elbow) hoods within spatial restrictions are prone to generate flow nonuniformity and unsteadiness. On this account, advanced exhaust hood design was proposed to address the abovementioned issues in space limited conditions. RANS CFD simulations were used for the hood enhancement. The first part of the numerical investigation in the current paper is based on isolated exhaust system (diffuser and hood). Then in order to capture the interaction effect between rotor and exhaust system and provide reliable results the two final best exhaust hood models were tested coupled with rotor. The novel exhaust hood design significantly increased pressure recovery coefficient. The paper summarizes recommendations on designing diffusers and exhaust hoods in spatial restrictions on the base of wide related literature overview and deliverables obtained by the authors.

KEYWORDS

TURBINE DIFFUSERS AND EXHAUST HOODS, COMPACT EXHAUST SYSTEMS

NOMENCLATURE

c_p	pressure recovery coefficient
D_{et}	ship exhaust tower diameter
h_c	height of a collector
l_{ax}	axial length of an exhaust system
l_c	axial length of a collector
M	Mach number
s_c	maximum width of a collector
S_{ESO}	exhaust system outlet area
S_{Din}	diffuser inlet area
S_{HO}	hood outlet area
S_{HJ}	half joint plane area
p	pressure
R_{max}	maximum radial size of diffuser

Subscripts and abbreviations

*	total parameter
ESO	Exhaust system outlet
Din	Diffuser inlet
HO	Hood outlet
HJ	Half Joint Plane
n/a	not applicable
n/d	not defined

INTRODUCTION

The performance of diffusers and exhaust hoods is of critical importance for overall turbine unit efficiency and power output. Pressure recovery coefficient c_p, equation (1), indicates the ability of exhaust systems to transform the turbine discharge flow's pressure head into the static pressure. In case of a consideration of several exhaust systems with fixed inlet-to-outlet area ratio, which have different design, c_p can be a measure of a system's efficiency in providing static pressure recovery. In cases when pressure recovery is not provided, c_p remains negative and total pressure loss in an exhaust system is often used as indicator of its performance.

$$c_p = \frac{p_{ESO} - p_{Din}}{p^*_{Din} - p_{Din}}. \quad (1)$$

In accordance with the evaluations carried out by Keller (1986), increasing of the power output by 0.5% corresponded to the enhancement of c_p from -0.4 to 0.3. The breakdown of losses in typical steam turbine presented by Tanuma et al. (2011) shows that exhaust hood losses are comparable with blade losses and losses due to wetness. Xingsu et al. (1981) provided data for turbine power drop vs. pressure loss diagram for different gas turbine types. In general, each 1000 Pa of pressure loss cause overall power drop from 0.8 to 1.5%.

Diffuser and exhaust hood operation is defined by their design and area ratios in particular. Studies of exhaust system behavior carried out by scientists from University of Stuttgart (Finzel et al. 2011) show that decreasing of Half Joint Plane area has adverse effect on overall exhaust system performance. This is in line with the results obtained by Liu et al. (2002). Sensitivity of a diffuser performance from Half Joint Plane area poses challenges while developing effective exhaust systems for maritime applications due to tight space limitations.

MARITIME EXHAUST SYSTEMS: COMMON LIMITATIONS AND TYPICAL SOLUTIONS

Maritime exhaust systems have to meet different requirements in terms of its effectiveness, sizing and interaction with components up and downstream. In order to provide a maximum plant power output higher c_p value is strongly desirable. Meanwhile, interaction between turbine and diffuser requires circumferentially uniform flow field at diffuser inlet in order to provide safe system operation. Fu, J.L. et al. (2012) corroborated that unsteady static pressure field defined by the diffuser and exhaust hood is a crucial for the last stage blade safety. Moreover, exhaust gases flow is often used to drive ejector which provide turbine under-hood area ventilation, like described by Sun et al. (2016). On this account, hood outlet flow uniformity is also of particular importance. Discharge flow uniformity is also required to reduce losses in ship exhaust tower and provide proper and long-life operation of silencers. To sum up, a reasonable balance of exhaust system efficiency, expressed in c_p value, and discharge flow's steadiness and uniformity is to be achieved.

Basically, maritime application imposes strict limitations on all the system dimensions, as shown in figure 1: radial size R_{max}, axial length l_{ax}, collector width s_c, length l_c and height h_c, ship exhaust tower diameter D_{et}.

Lu Xingsu and scientists from Aerodynamic research Laboratory in Harbin (Xingsu et al. 1981) carried out comprehensive overview of gas turbine exhaust hoods especially for maritime application. All the possible designs of devices can be classified as volutes with axial-radial diffusers, bend-type and box type hoods. Each type has specific spatial demands: volutes with axial-radial diffusers require significant radial size and collector width, while bend-type ones are space-consuming in axial dimension. Box type exhaust hoods are often used in case of limitations in both axial and radial directions.

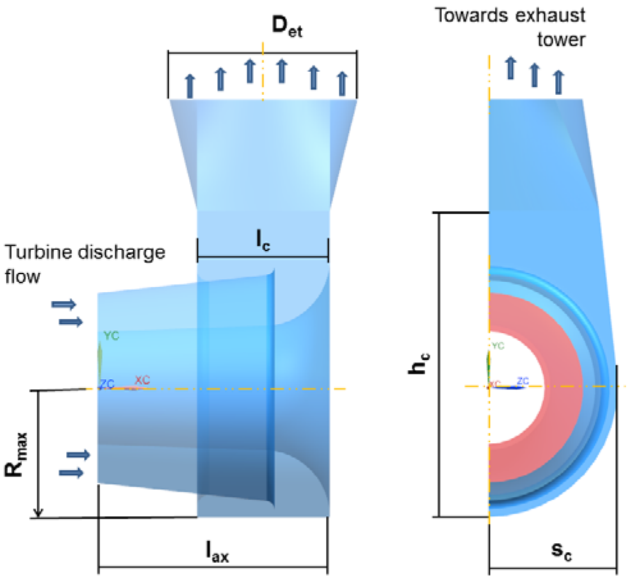

Figure 1: Major spatial restrictions

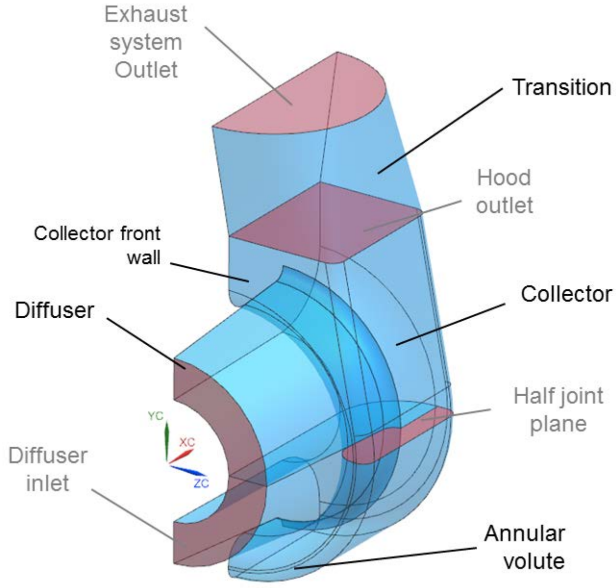

Figure 2: Exhaust system sections and components

STUDY METHODOLOGY

A methodology of study of exhaust hoods and diffusers has been widely developed over the last 50 years. In 1956 Kondak et al. demonstrated the interaction between diffuser and exhaust hood and established the additional pressure loss in diffuser due to exhaust hood. Gray, L et al. (1989), Zaryankin et al. (1969) and Chernikov et al. (1984) were the first researchers to point out the necessity of taking into account last stage-diffuser interaction to replicate the real exhaust system operating conditions.

At present a significant number of studies (Kluss et al. (2009), Fu et al. (2012), Burton et al. (2012)) have corroborated the importance of replicating interactions between last stage blades, diffuser and exhaust hood to securely predict exhaust system performance. Gardzilewicz et al. (2009) have quantified advantages of last stage and exhaust system joint simulation in comparison with transmitting of up/downstream boundary conditions in coupling approach. To sum up, the most accurate prediction is achieved when exhaust system components and last turbine stage in experimental or numerical model are joined.

Nevertheless joint simulation has significant shortcoming expressed as high cost of experimental and high computational cost of numerical simulations. On this account simplified models can be applied which, however, have to consider key factors influencing an exhaust system performance. These factors are summarized by Burton et al. (2013a) and Gogolev et al. (1995) as follows:

- swirl angle at diffusor inlet;
- inlet flow nonuniformity in circumferential and radial directions;
- turbulent intensity;
- last turbine stage tip leakage jet.

With an eye to reducing computational power demand two steps have been carried out in the current study. First, the proposed design modifications were studied within isolated exhaust hood model. In the current case swirl angle and flow nonuniformity in their range do not affect significantly an exhaust system performance and therefore are skipped. Tip leakage jet is not taken into account since it requires fine grids.

Then, joint simulation approach was implemented for the baseline RA-0 and final models in order to evaluate the stage influence on the exhaust system performance and ensure that the proposed design is effective upon the real operating conditions. In doing so, the tip gap was included into the rotor domain in order to take into account tip leakage jet effects.

There are several numerical approaches to capture interaction between turbine stage and exhaust system: mixing plane approach, Frozen Rotor approach, nonlinear harmonic method described by Burton et.al (2013b).

In the current research mixing plane (Stage) approach was used since the static pressure circumferential distribution at diffusor inlet was not considered. Actually, pressure circumferential non-uniformity may be significant for nonaxisymmetric hood configurations as shown by Gao et al. (2017) causing blade force amplitude bigger than the one due to rotor-struts interaction. Blade forcing study however requires extensive transient simulations and is a subject of a special study.

INVESTIGATED OBJECTS

In the current paper several classical exhaust systems are considered and then novel box type hood is proposed in order to enhance the efficiency and the flow uniformity. The process of box type hood topology modification is described in detail so as major factors which affect performance can be clearly identified. All the investigated models have the same diffuser aperture angle and were designed within the same spatial restrictions, R_{max}=1000 mm and l_{ax}=2250 mm. The diffuser inlet hub and shroud diameters are kept the same as well as the ship exhaust tower diameter. Exhaust system major sections and components are depicted in figure 2. Major geometrical parameters of the investigated models are given in table 1. General appearance of all the models is shown in figure 3.

The baseline model RA-0 has so-called volute-type hood. It comprises axial-radial diffuser, annular volute, collector and transition. The volute has constant circumferential cross-section. The collector’s width is a maximum at Half Joint Plane and then diminishes in order to fit the diameter of the exhaust tower. The collector has rectangular cross-section in order to provide the maximum flow path area within the given width and length. Then it is followed by the transitional section aimed to connect rectangular collector and circular exhaust tower.

The following design RA-X is shaped so as to replicate the configuration that was shown to be efficient by Xingsu et al. (1981). The model incorporates axial-radial diffuser and a collector. The bottom part of the collector designed with variable cross section so as to meet the streamwise increasing of the flow volumetric flow. The maximum collector width is reached at Half Joint Plane and then is kept at the certain distance downstream. The lower part of the collector is cut to eliminate excessive space. The collector maximum width and length are kept the same as in the Baseline model.

Bend-type exhaust system is also considered and its outline is shown in figure 3. Circular turbine outlet section and exhaust tower inlet sections are connected via curved tube. Cylindrical through cut is arranged in order to place the drive shaft inside.

Analysis of the numerical results provided below showed that the abovementioned exhaust systems are not able to provide positive c_p values. The main reason for this is strict limitation of a collector width resulted in small Half joint plane area. Finzel et al. (2011) have shown that c_p drop by a factor of 2.5 corresponded to decreasing of S_{HJ}/S_{Din} ratio from 1.4 to 0.6. In case of RA-0 and RA-X models the said ratio is about 0.3 which is evidently out of its optimal range. In this terms a box type exhaust hood was developed via gradual topology modification from V1 to V6 as shown in figure 3. Discussion chapter provides detailed justifications of the design evolution.

Basic model V1 comprises axial diffuser and collector. Diffuser has the same aperture angle as models RA-0 and RA-X. Collector cross-section increases from the bottom to Half joint plane so as to meet the flow volumetric flow rate rise. The collector maximum width s_c is set the same as in the Baseline model. Box type design suggests that both diffuser and hood are placed sequentially in meridional plane, while in volute type a hood surrounds a diffuser. As a result, within the same axial length l_{ax} both diffuser and collector occur to be shorter for the box hood design. In order to increase the diffuser length, the front wall of models V1-V6 is moved towards the rear wall which leads to a decrease of the collector width l_c right after the upward bend. In models V1-V4 l_c remains constant downstream the flow turn, as a result, not all the available axial distance is exploited by the collector and the hood outlet area S_{HO} decreases. This shortcoming is eliminated in models V5 and V6 by introducing the front wall incline.

Models V2-V6 comprises consistent modifications as follows: in V2 collector and diffuser are connected with a face blend (1) of a variable radius (figure 3). In order to diminish the bottom area cross-section the backward incline (2) of the rear wall of the collector is incorporated in V3 model. V4 model has additional oblique cutout (3) in the bottom part in order to eliminate the area with low-speed flow and support the flow turn. In model V5 the front wall (4) of the collector has forward incline aimed at increasing of the hood outlet area. Model V6 has more intensive front wall incline, and in contrast with models V1-V5, has the same hood outlet area as RA models. For models V4-V6 the following equation may be applied for l_c right after the axial radial deflection: $l_{cV4} < l_{cV5} < l_{cV6}$. In V5 and V6 l_c keeps increasing till the S_{HO} plane due to the wall incline.

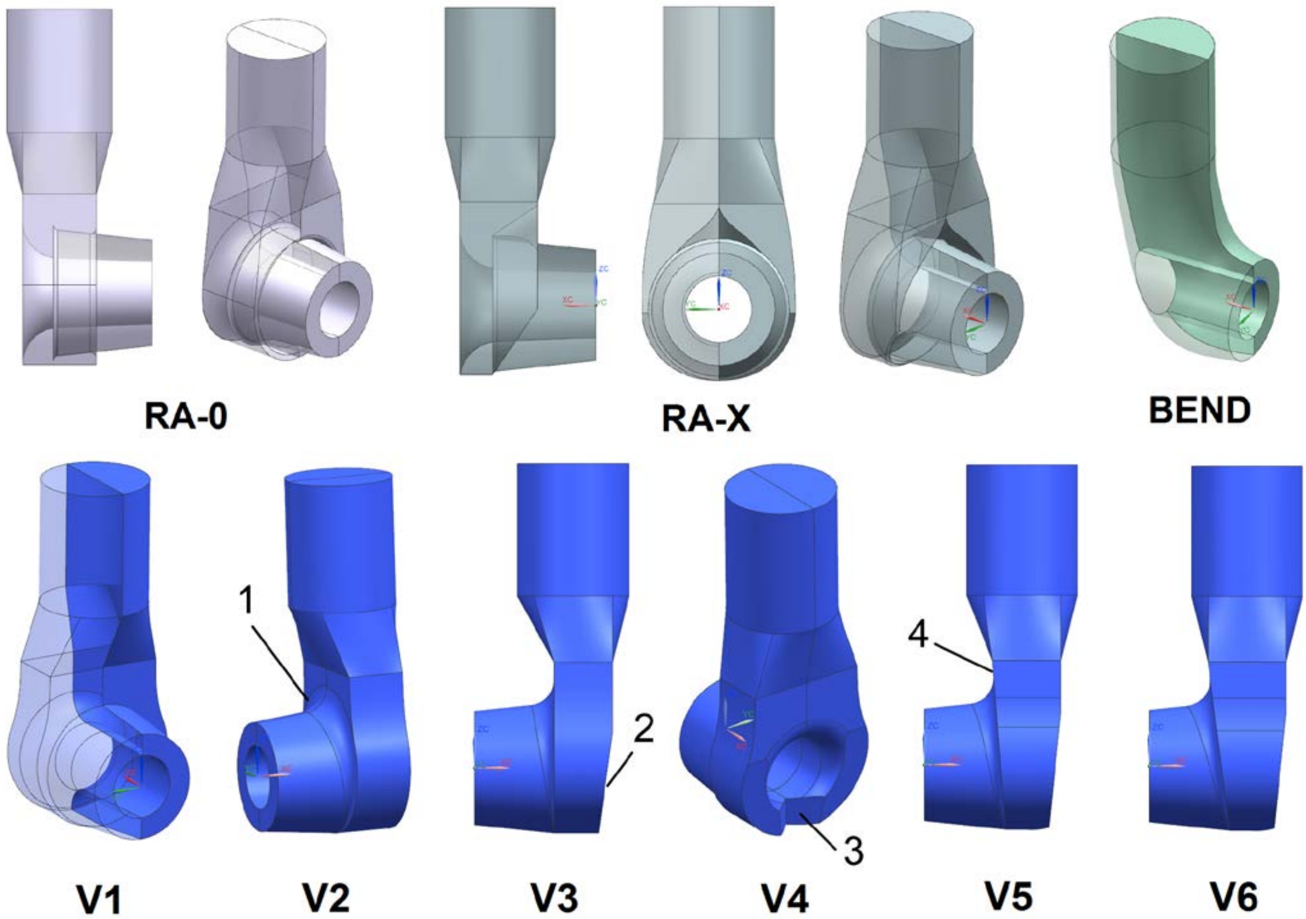

Figure 3: General appearance of the investigated models

NUMERICAL ANALYSIS

Numerical Simulation Method

ANSYS CFX was used to carry out the numerical simulation. Pressure and temperature range in the case in question enabled to consider the working fluid (gas turbine flue gases) as a viscid compressible gas obeying to ideal gas equation of state. The specific heat capacity of the fluid was set as polynomial. Total energy of the fluid inside the exhaust system was kept constant, while the heat transfer to the environment was suppressed. Since the pressure recovery was to be evaluated, mass flow at the inlet and static pressure and the outlet boundary were set.

In case of isolated diffuser simulation uniform flow was set at the inlet boundary. Whereas in case of exhaust hood-rotor last stage joint simulation the flow field taken from the stator of the last stage was applied to the rotor inlet. Stage interface was chosen to provide a connection of the rotor with the exhaust system. The choice of the domain interface is justified in “Study methodology” section.

Turbulence intensity and length scale are proven to affect the flow behavior, but since their particular values into the real operating conditions were unknown, the default ones for the inlet boundary in Ansys CFX were set.

k-ω SST turbulence model was used in steady state approach. The maximum element size of unstructured tetra mesh was 20 mm for the exhaust system and 4 mm for the rotor part corresponding to 15 mln and 2.7 mln total number of nodes in the relevant domains. The surface element size was equal to the maximum volume element size. In the exhaust system domain the grid discretization was chosen on the base of the grid sensitivity study, the results are provided below. y+ parameter was kept below 2 in the exhaust system which was justified by the grid independence study as well. Since the rotor characteristics themselves are not a subject of the current study, a grid sensitivity study has not been carried out for the rotor domain. Wall functions were used on the blade surface, y+ over 30; y+ below 2 was set on the blade’s shroud surface and on the top endwall. The maximum grid size was set so as to provide at least 20 nodes in a blade passage pitch-wise (not taking into account the prism layers).

Monitoring of the RMS residuals, imbalances and pressure recovery coefficient was used to control the convergence of steady-state solution process. The convergence criteria for the steady-state solutions were:

- drop of RMS residuals more than 10^2;
- imbalances less than 0.5%;
- stable values or periodic fluctuation of c_p.

Some considered exhaust system configurations exhibit unsteady behavior and therefore oscillating c_p values. For these occasions transient simulations are preferable but require extensive computational efforts. In order to verify steady state results transient simulation was performed for V1 reference model only and is extended to the others due to the similarity of a flow behavior. Comparison of the steady state and transient results revealed the discrepancy of averaged c_p value within the 1.5% limit which may be deemed satisfactory.

Application of equation 1 requires an averaging of pressure in the system inlet and outlet sections. In these terms, the area averaging method was applied for the static pressure, whereas for calculation of total pressure the mass flow averaging method was used.

The boundary conditions and simulation assumptions are summarized in table 1.

Table 1: CFD boundary conditions and assumptions

Parameter	Dimensions	Value
Turbulence modeling	-	RANS, SST
Time discretization	-	Steady state
Working fluid	-	Ideal gas, compressible, viscid
Joint or isolated rotor-hood simulation	-	Both
Domain interface in joint simulation	-	Stage
Inlet total temperature	K	below 800
Outlet static pressure	Pa	101325
Inlet average Mach number	-	~0.19
Last stage tip flow Mach number	-	~0.37
Average inlet Reynolds number	-	$\sim 1.9 \cdot 10^6$

Grid sensitivity study

In order to ensure the validity of the results the mesh independence study has been carried out. For the exhaust system domain the grid element sizes from 70 mm to 10 mm were considered. The surface maximum element size was equal to the global maximum size. The study was implemented for y+ values over 30 (with using the wall functions) and y+ below 2. Inlet-to-outlet total pressure drop was chosen as the control value to assess the mesh-to-mesh convergence. The results are provided in figure 4.

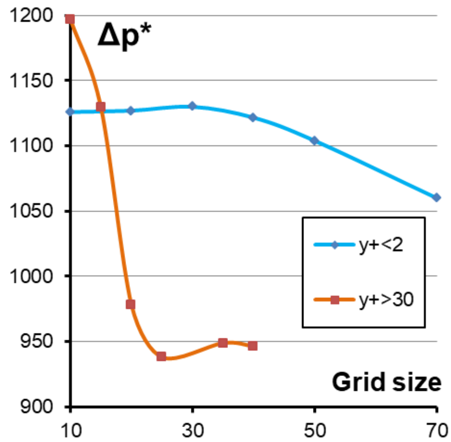

Figure 4: Grid sensitivity results

It should be noted that the wall functions (y+>30) fail to provide grid-to-grid convergence within the element size decrease. The total pressure drop remains about 950 Pa from 40 mm to 25 mm of the grid element size and then surges to almost 1200 Pa within the further size decrease. On the contrary, the results obtained with using y+ below 2 exhibit fine grid-to-grid convergence. Inherently, the pressure drop values for the grid size range from 10 to 40 mm are all within 1% margin of deviation and therefore indicate the results' independence from the grid. However, in order to provide an acceptable grid aspect ratio 20 mm maximum global element size was applied.

DISCUSSION OF THE RESULTS

Exhaust hood topology modification

In the first part of the study presented models were simulated without the last turbine stage. Averaged pressure recovery coefficients are presented in table 2. Pressure recovery coefficients stability is illustrated via c_p iteration evolution in figure 5. Velocity streamlines and contours are depicted in figure 6 to represent the flow structure and particularities.

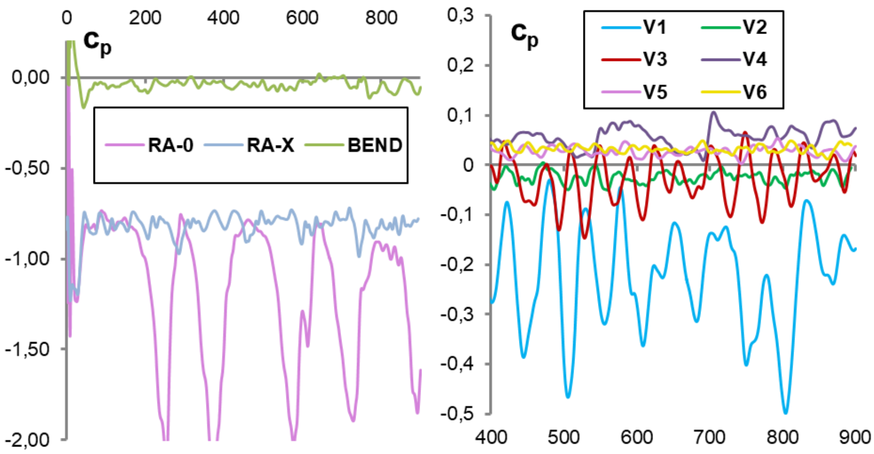

Figure 5: c_p iteration evolution

Table 2: Geometrical parameters of the models and c_p values

<table>
<tr><th></th><th>$\frac{S_{ESO}}{S_{Din}}$</th><th>$\frac{S_{HO}}{S_{Din}}$</th><th>$\frac{S_{HJ}}{S_{Din}}$</th><th>C_p</th><th>C_p +stage</th></tr>
<tr><td>RA-0</td><td rowspan="9">1.51</td><td>1.60</td><td>0.37</td><td>-1.24</td><td>-1.30</td></tr>
<tr><td>RA-X</td><td>1.60</td><td>0.32</td><td>-0.81</td><td rowspan="5">n/d</td></tr>
<tr><td>Bend</td><td>n/a</td><td rowspan="7">n/a</td><td>-0.04</td></tr>
<tr><td>V1</td><td rowspan="4">1.19</td><td>-0.228</td></tr>
<tr><td>V2</td><td>-0.025</td></tr>
<tr><td>V3</td><td>-0.027</td></tr>
<tr><td>V4</td><td>0.055</td><td>0.23</td></tr>
<tr><td>V5</td><td>1.41</td><td>0.029</td><td>n/d</td></tr>
<tr><td>V6</td><td>1.60</td><td>0.033</td><td>0.21</td></tr>
</table>

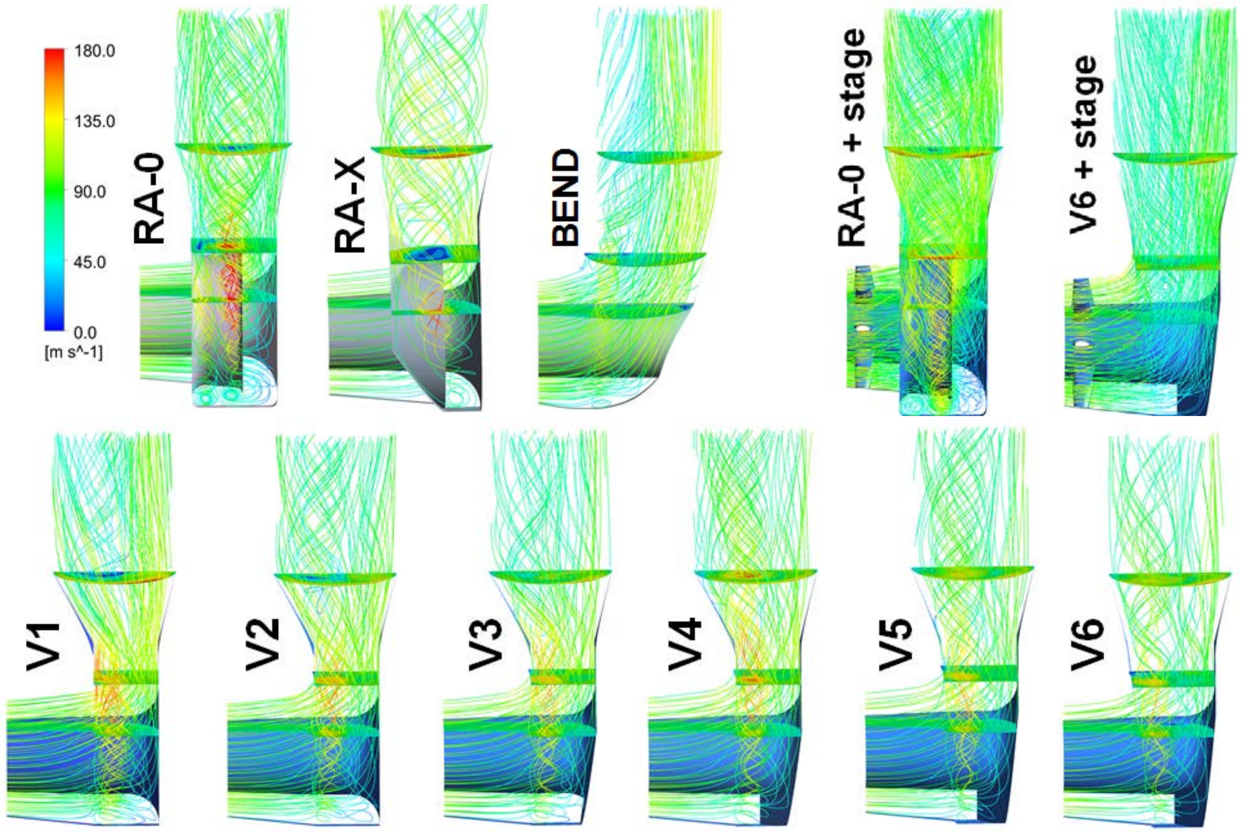

Figure 6: Velocity streamlines

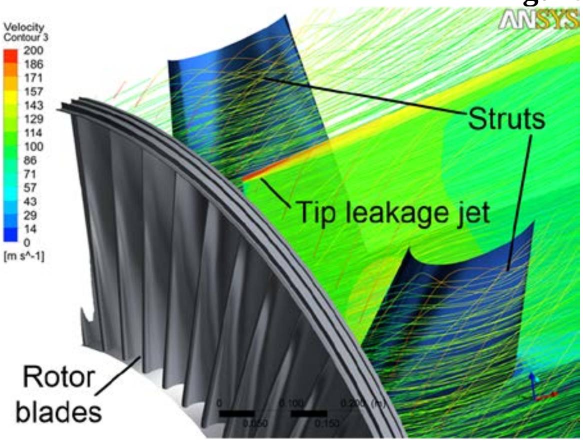

Figure 7: Velocity streamline at tip region

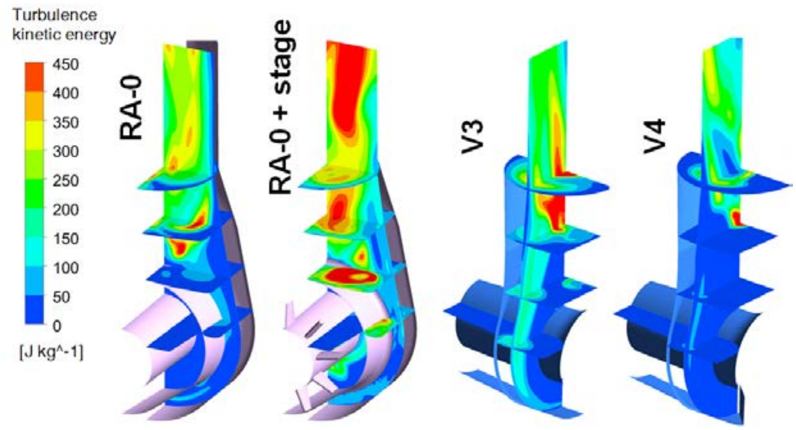

Figure 8: Turbulent kinetic energy

<u>Models RA-0, RA-X, Bend</u>

All three designs RA-0, RA-X and Bend are prone to generate strong vortex structures. Models with axial-radial diffusers have highly vortical flow as a result of 90 degrees turn is short axial distance as it was highlighted by Burton et al. (2013a). A more detailed analysis of a vortex formation process was presented by Jayatuna in his doctoral thesis (2005).

Area in the bottom part of a hood in RA-0 and RA-X models is a starting point of a vortex core as can be seen in figure 6. The vortex structures occupy the hood area from the bottom to the very top resulting in high flow non-uniformity and low efficiency.

Cylindrical throughout cut in Bend model replicates the conditions for von Karman vortex street formation, which cause significant temporal flow instability in addition to vortices.

The baseline model RA-0 has negative c_p value -1.24. RA-X model shows c_p -0.81 in contrast to high efficiency (c_p=0.57) obtained for the same topology by Xingsu et al. (1981). Bend type hood has higher c_p value, but it still remains negative and, moreover, temporally unstable.

Small Half joint plane area is likely to be another reason of dramatic efficiency drop in RA-0, RA-X and Bend cases. Decreasing of a cross section from Hood Inlet area to Half joint plane area causes flow acceleration which raises friction losses and supports vortex formation mentioned above.

Box type models V1-V6

Due to strict spatial limitations box type configuration is proposed in order to improve exhaust system performance by eliminating vortex formation. In this case a box type solution is helpful since the bottom hood area, where the vortex onset occurs, is eliminated. Moreover, an approach of a variable cross section in circumferential direction instead of the constant one is likely to support the proposed concept.

The said measures in model V1 resulted in mitigation of the vortex flow velocity as can be seen in figure 6. It should be noted that the size of the vortex also decreased and more uniform flow in Hood outlet section is achieved. However, V1 model is not able to propose satisfactory performance due to significant mixing losses after the flow turn towards vertical direction. Face blend introduced in model V2 is helpful to support the flow turn which results in decreasing of the vortex flow velocity. As can be seen in figure 6, for models V1 and V2 the equal aerodynamic blockage effect of the vortex upstream the HO section can be pointed out because of the constant vortex size. However, a decrease of the vortex intensity (velocity) has led to diminishing of the blockage effect between HO and ESO sections. As a result, c_p value has increased. Nevertheless, V1 and V2 efficiency remains low since vortex structures formation is not completely suppressed. While in RA-0 model vortices are aided by the model topology itself, in box type hood design the reason of its onset is different. A volume in the bottom part of a collector is still excessive, causing low velocity zones. An interaction of high speed flow of a middle collector part and low speed flow in the bottom one causes flow twisting and, therefore, vortices. V1 model, furthermore, is prone to generate highly instable flow as illustrated by oscillating c_p.

The excessive hood area is eliminated by rear wall's backward incline introduced in model V3. As a result, vortex intensity is diminished in the vortex region which can be seen in figure 6. V3 design also facilitates the flow uniformity in Exhaust system outlet section. However, having remained of the same value, c_p exhibits significant temporal oscillations as depicted in figure 5.

For the further explanations turbulence parameters have to be considered. Figure 8 illustrates turbulent kinetic energy distribution along the exhaust system. It is clear that maximum value of the said parameter is in vortex core region. It is assumed that significant kinetic of the turbulence energy is caused by interaction of high and low speed flows in the very bottom part of a collector. In this terms, model V4 is designed with oblique cutout introduced in the bottom part area aimed at elimination of excessive volume and accelerating the flow. In the same time, the cutout is likely to be helpful in supporting a flow turn. The results obtained for V4 support the assumption. The vortex regions in V3 and V4 have comparable sizes and velocities, but turbulent kinetic energies are significantly different. Plummeting of the said parameter has adverse effect on the flow uniformity in V4 hood outlet section, but in terms of pressure recovery it is clearly favorable: c_p reaches positive values. A clear correlation between losses and turbulent kinetic energy has also been noted by Jayatuna in the doctoral thesis (2005).

As it has already been mentioned in "INVESTIGATED OBJECTS" chapter, in box type configurations within the same axial length l_{ax} both diffuser and collector are shorter in comparison with volute type systems. In models V1-V4 the collector length l_c is reduced in favor of the diffuser length. However, as a result not all the available axial size l_c is exploited. On this account two models with forward incline of a front wall are introduced. Model V6 has the same hood outlet area as models RA-0 and RA-X. For V5 and V6 the blockage effect of the vortex is diminished due to an increase of HO area, since the vortex size itself has not changed. Despite the reduction of the adverse blockage effect, both designs provide slightly lower c_p in comparison with V4. Nevertheless, c_p stability is enhanced, and a more stable behavior in V6 corresponds to larger HO area of this model. V5 and V6 models provide also the best flow uniformity. The latter one is of particular importance in order to provide a proper operation of a discharge ejector, reduce losses in a ship exhaust tower and ensure long-term operation of a silencer. V6 design provides the best combination of c_p and flow stability and therefore may be considered as the targeted outcome.

Exhaust hood- last stage joint simulation

In order to provide reliable c_p values joint simulations of the last stage and the exhaust systems are carried out. The flow field taken from the stator vane of last stage was applied as the inlet boundary condition of joint simulation. Baseline RA-0 model, V4 and V6 models are paired with the shrouded rotor. The tip gap width is 2.6 mm, which amounts to 0.76% of the blade height. In addition, the struts of the bearing cone are included into the investigated models in order to introduce the real operating conditions of the exhaust system. Simulation results are presented via velocity streamlines and contours in figure 6 and vortex core regions by lambda2-criterion method in figure 9. A comparison between isolated and joint diffuser simulation results can be carried out using the said figures. As one can see the flow non-uniformity, swirl angle turbulence intensification caused by the turbine tip leakage jet alter the flow structure in exhaust system and has a favorable effect on its performance. V4 and V6 models experience a surge of c_p up to 0.21-0.23 due to the suppression of the flow vortex structures by high-speed tip leakage flow (Mach number is two times as high as in the flow core, see table 1). This phenomenon was also quantitatively assessed by Institute of Thermal Turbomachinery in Stuttgart (Finzel et al. 2011): increasing of the tip flow Mach number from 0.4 to 1.2 resulted in a twofold increase of c_p for some investigated configurations. The latter indicates some reserve of a further enhancement of the exhaust system by means of increasing of the tip Mach number, but such measures have to be additionally justified.

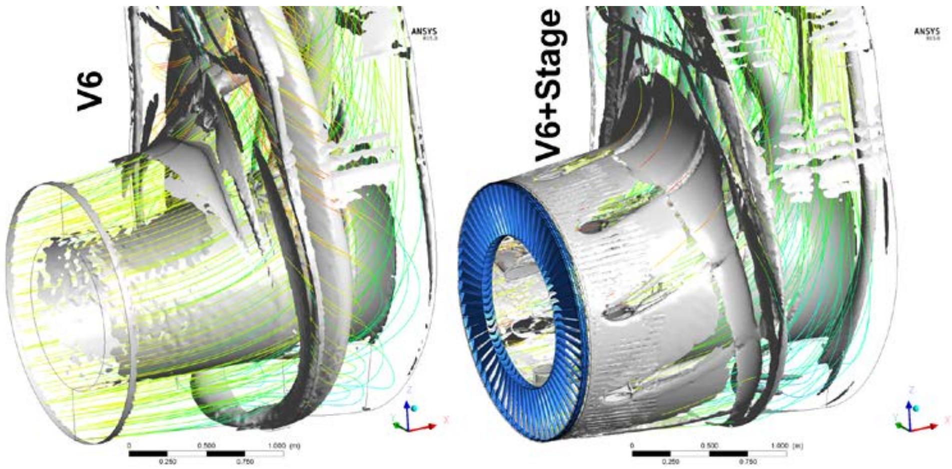

Figure 9: Comparison between isolated and joint diffuser simulation

In contrast, the diffuser inlet turbulence intensification has adverse effect on system performance for the baseline model RA-0. Due to the design topology vortex structures are stable and are not distorted by the turbine discharge flow perturbations. Higher initial turbulence caused by the turbine stage increases the turbulent kinetic energy magnitude along the whole exhaust system which is reflected in c_p drop. While simulating turbine exit flow by screen and vane Liu, J. J et al. (2003) also reported the similar adverse effect on pressure recovery capability for a similar hood design.

It has to be outlined particularly, that despite the same S_{ESO}/S_{Din} ratio, all the considered models exhibited completely different performance and behavior. This suggests that in case of strict spatial limitations a hood design and a cross section evolution are of particular importance along with area ratios.

CONCLUSIONS

Aerodynamic modification of a gas turbine exhaust system installed within strict space limitations is provided in the current paper. Box type exhaust hood design is developed in order to achieve better performance and flow uniformity. The design modifications are based on 3D CFD simulations and aimed at sequential elimination of the factors that negatively influence the performance. The following major findings can be outlined:

1. The classical type hoods satisfying for no space-restricted applications typically provide a poor performance within spatial limitations. Compact hoods require individual design approach.
2. The greatest possible value of outlet/inlet area ratio is not a principal criterion for achieving the highest exhaust system performance and stable operation when significant spatial limitations are imposed.
3. A box type hood design is able to provide satisfactory c_p values in case of strict spatial restrictions while classical design solutions (volute hood design, bend type hood) fail to provide positive c_p.
4. Diminishing of a cross section in a hood bottom part is favorable for box type design. In this terms variable hood cross section (from the bottom to Half joint plane), backward wall incline and additional cutouts may be used. The abovementioned measures are helpful for decreasing a vortex formation and turbulent kinetic energy, finally resulting in better performance.

ACKNOWLEDGEMENTS

The resources of Supercomputer center of Peter the Great Polytechnic University were used to perform high computational cost simulations. The authors also appreciate technical support provided by Supercomputer center team.

REFERENCES

Burton, Z., Ingram, G. L., and Hogg, S., (2012). *A Generic Low Pressure Exhaust Diffuser for Steam Turbine Research.* Proceedings of ASME Turbo Expo, Copenhagen, Denmark, June 11–15, ASME Paper No. GT2012-68485.

Burton, Z., Ingram, G. L., and Hogg, S., (2013a). *A Literature Review of Low Pressure Steam Turbine Exhaust Hood and Diffuser Studies*, ASME J. Eng.Gas Turbines Power, 135(6), p. 062001.

Burton, Z., Ingram, G. L., and Hogg, S., (2013b). *A Novel Method of Coupling the Steam Turbine Exhaust Hood and the Last Stage Blades Using the Non-Linear Harmonic Method*. Proceedings of ASME Turbo Expo, San Antonio, TX, June 3–7, ASME Paper No. GT2013-94184.

Chernikov V.A., Yeremin A.C., Mokravtsov F.V., (1984). *Experimental characteristics of the transition of the diffuser of the turbine of the gas generator with a step largest-circulation GTU GPA*. NIIE informenergomash. Energeticheskoe mashinostroenie. N 4, 1984, pp. 18 – 24. (*in Russian*).

Finzel, C., Schatz, M., Casey, M. V., and Gloss, D., (2011). *Experimental Investigation of Geometrical Parameters on the Pressure Recovery of Low Pressure Steam Turbine Exhaust Hoods*. Proceedings of ASME Turbo Expo, Vancouver, Canada, June 6–10, ASME Paper No. GT2011-45302.

Fu, J.L., Liu, J.J., and Zhou, S.J., (2012). *Unsteady Interactions between Axial Turbine and Non-axisymmetric Exhaust Hood under Different Operational Conditions*. ASME. J. Turbomach. 2011;134(4):041002-041002-11.

Gao J, Lin F, Niu X, et al. (2017). *Flow Interactions Between Shrouded Power Turbine and Nonaxisymmetric Exhaust Volute for Marine Gas Turbines*. Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Volume 1: Aircraft Engine; Fans and Blowers; Marine; Honors and Awards ():V001T25A001.

Gardzilewicz, A., Badue, J., Karcz, M., Swirydczuk, J., (2009). *Numerical Investigation of the turbine last stage-exhaust hood flow.* https://www.researchgate.net/publication/267560268

Gogolev, I., Drokonov, A., (1995). *Aerodynamics of turbine stages and exhaust hoods*. Bryansk, Grani, 258p.

Gray, L., Sandhu, S., Davids, J., and Southall, L., (1989). *Technical Considerations in Optimizing Blade-Exhaust Hood Performance for Low Pressure Steam Turbines*. Latest Advances in Steam Turbine Design, Blading Repairs, Condition Assessment and Condenser Interactions, (ASME PWR), Vol. 7, ASME, New York, pp. 89–97.

Jayatunga C., (2005). *An aerodynamic study of industrial gas turbine exhaust systems*. Doctoral Thesis, Loughborough University.

Keller, H., (1986). *Aerothermodynamics of Low Pressure Steam Turbines and Condensers.* Hemisphere, Washington, DC.

Kluß, D.; Stoff, H.; Wiedermann, A. (2009). *Effect of Wakes and Secondary Flow on Re-attachment of Turbine Exit Annular Diffuser Flow.* Journal of Turbomachinery, Vol. 131, 041012 pp.1-12, 2009.

Kondak H. M. (1956). *Application of circular turns in the turbine exhaust hoods*. Trudy instituta teploenergetiki AN USSR, No 13, 1956, pp. 57 - 63.

Liu, J. J., and Hynes, T. P., (2002). *The Investigation of Turbine and Exhaust Interactions in Asymmetric Flows: Part 2—Turbine-Diffuser-Collector Interaction*. Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Volume 5: Turbo Expo 2002, Parts A and B ():179-188.

Liu, J. J., Cui, Y. Q., and Jiang, H. D., (2003). *Investigation of Flow in a Steam Turbine Exhaust Hood With/Without Turbine Exit Conditions Simulated*. ASME J. Eng. Gas Turbines Power, 125(1), pp. 292–299.

Sun T., Sun L., Luan Y., Sun P., (2016). *Research on Characteristics of a New Marine Gas Turbine Exhaust Ejector Device*. ASME. Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Volume 1: Aircraft Engine; Fans and Blowers; Marine ():V001T22A006. doi:10.1115/GT2016-57214.

Xingsu, L., Kunyuan, P., Zuomin, W., (1981). *Aerodynamic design and experimental study of marine gas turbine exhaust volutes*. Proceedings of ASME Turbo Expo: Power for Land, Sea, and Air, Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery ():V001T02A004.

Zaryankin, A. E., and Myslitskii, E. N., (1969). *Study of Exhaust Hood in Combination With a Wheel of a Radial-Axial Turbine*. MEI, Moscow, 1969.