

# The Influence of Condenser Pressure Variation and Tip Leakage on LP Steam Turbine Exhaust Hood Flows

**Zoe Burton\***

School of Engineering and Computing Sciences  
Durham University  
South Road, Durham  
DH1 3LE  
Email: zoe.burton@durham.ac.uk

**Grant L. Ingram**

Senior Lecturer  
School of Engineering and Computing Sciences  
Durham University  
South Road, Durham  
DH1 3LE  
Email: g.l.ingram@durham.ac.uk

**Simon Hogg**

Reader  
School of Engineering and Computing Sciences  
Durham University  
South Road, Durham  
DH1 3LE  
Email: simon.hogg@durham.ac.uk

*This paper aims to highlight the importance of the accurate computational modelling of both the inlet and outlet exhaust hood boundary conditions. The computations presented are calculated using the public domain LP exhaust diffuser test case proposed by Burton in 2012. The original test case did not include the effect of tip leakage on diffuser flows, this paper describes the inclusion of tip leakage and the results are shown to be in-line with the outputs produced by other authors. The key advance in this paper is that calculations were conducted with a representative condenser pressure gradient caused by the temperature variation inside the condenser tube nest. It is shown that accurately modelling the exit boundary calculation has a large influence on the flow structure and a smaller influence on the pressure recovery inside the exhaust diffuser. This influence is smaller than that seen by other authors when including unsteady effects or accounting for the circumferential non-uniformity of the turbine exit flow but will need to be included in design calculations as diffuser design advances.*

## Nomenclature

$C_p = \frac{P_2 - P_1}{P_{T1} - P_1}$  Static Pressure Recovery  
 $c_p$  Heat Capacity at Constant Pressure  
P Pressure  
V Relative Velocity  
 $\alpha$  Swirl Angle

$\Gamma$  Ratio of Specific Heats

## Subscripts

T Total/Stagnation  
t Tangential  
u Upstream  
x,y,z Velocity Components  
1 Hood/Diffuser Inlet  
2 Hood Outlet

## Abbreviations

CFD Computational Fluid Dynamics  
IP Intellectual Property  
LSB Last Stage Blades

## 1 Introduction

The low pressure (LP) steam turbine exhaust diffuser and hood are a vital area of power plant design. The performance of the exhaust hood strongly influences the efficiency and ultimately the power output of the generating unit. The flow exiting from the last stage steam turbine blades is decelerated by the exhaust diffuser, converting the kinetic energy into pressure recovery. This creates a lower pressure downstream of the turbine and, for a given condenser pressure, generates a greater power output.

The performance of the exhaust hood is typically quantified by its pressure recovery factor,  $C_p$ . In the majority of cases this value is low, ranging from -0.25 to +0.5 [1, 2]

---

\*Address all correspondence to this author.

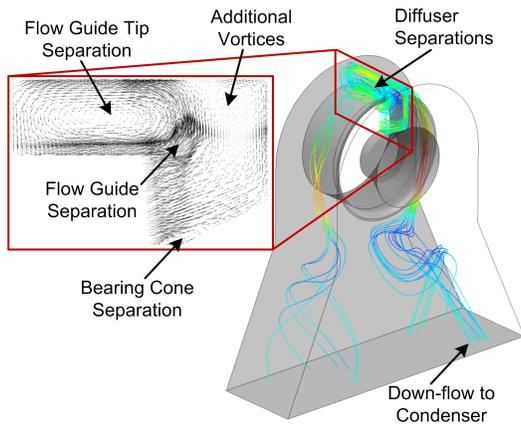


Fig. 1. Vortices and complex exhaust hood flow structure

which at the higher end of this range gives a lower turbine exit pressure than that in the condenser and subsequently a higher power output.

Pressure recovery is lower when the condenser is positioned beneath the turbine, as the flow is required to turn sharply through  $90^\circ$  from the axial to the radial direction in a relatively short axial distance, generating a highly vortical flow, Figure 1. In addition, the structural internal furniture within the hood contributes additional blockage, hindering the pressure recovery potential. Further to this, the flow exiting from the last stage blades is highly non-uniform due to the strong interaction between the turbine and the hood. The pressure recovery of the exhaust hood is governed by the turbine, but conversely the operating point of the turbine is dictated by the performance of the exhaust hood.

In recent years, computational fluid dynamics (CFD) has come to the forefront of steam turbine exhaust diffuser research, enabling accurate flow field representations to be generated at a fraction of the time and expense of experimental testing. However, the unsteady, transonic, wet steam flows within the complex exhaust hood geometry pose a severe challenge for even modern computing power, with long convergence times and sensitive solutions. To model the full 3D,  $360^\circ$  unsteady rotational flow field of the turbine coupled to the accompanying exhaust hood with full internal furniture is currently too computationally expensive for routine design calculations. This has led to the development a range of simplified methodologies over the last 10 years to reduce the computational demand to more manageable levels, although at present no single ‘best-practice’ approach has come to the forefront. A comprehensive review of current steam turbine exhaust hood CFD modelling practice is included in work from Burton in 2013 [3].

One of the most computationally efficient methods of generating representative exhaust hood flow fields is by the so-called ‘sequential approach.’ This widely adopted [4,5,6] method of simplification involves solving the exhaust hood flow field separate from that of the last stage blades. Empirically determined or circumferentially averaged computational flow variables taken downstream of the rotor trailing edge are then applied as the inlet boundary condition

to a separate exhaust hood calculation. Provided the mass flow rate through the exhaust hood the same as that through the stage, the calculation is coupled in the streamwise direction. The exhaust hood “sees” the influence of the last stage blades, but information only travels in the flow direction so the last stage blades are unaffected by the exhaust hood. This method was first used in the mid-1990’s with uniform flow properties at inlet [1, 7] but was shown to produce inaccurate flow structures. Ensuring the accuracy of the inlet boundary condition by incorporating radial variations of flow properties enables representative flow structures to be generated in modern flow calculations [8].

Although there is debate over the best computational modelling strategy, researchers are united in acknowledging the importance of modelling the tip leakage jet in generating representative flow structure in the exhaust hood, particularly near the flow guide. The high adverse pressure gradient in this region can result in a separation forming along the flow guide. The tip leakage adds momentum to the boundary layer and can suppress this separation by means of a suction effect. Hence the most accurate vortex structures and losses are predicted when the leakage is modelled. The significance of the leakage jet was first acknowledged by Benim in the mid 1990s [9]. More recent experimental work [10] found that using a synthetic jet tangentially blowing in steam can reduce hood losses by 20% as the separation along the flow guide is suppressed. Experimental investigations [11] have showed the separation point can be moved downstream until it was completely suppressed with increasing jet strength. Computational results by Li showed that the suppression of the separation along the flow guide can yield significant increases in static pressure recovery, from 0.01 to 0.398 [12]. However this increase in  $C_p$  does not necessarily lead directly to an increase in turbine efficiency as the benefits of tip leakage flows on exhaust hood must be balanced against the increased leakage losses in the turbine [11]. Although of course in reality the turbine will always have leakage effects.

At present, the majority of research focuses on the importance of the inlet boundary conditions and little attention is paid to ensuring representative conditions at hood outlet. Most researchers model the exhaust hood outlet as a constant static pressure boundary condition, set to give the correct mass flow rate through the exhaust hood system. The absolute value of this static pressure is governed by the condenser heat sink, which varies seasonally with the atmospheric temperature and/or sea water temperature (depending on the method of cooling). This annual variation has been shown to be as large as 0.29 bar (from 0.23 bar to 0.52 bar) for a sea water cooled plant in Finland [13]. An EDF study noted that large, potentially unsafe vibrations in the turbine shaft bearings are present at critical condenser pressures due to a shift in exhaust hood re-circulations which drive water films down the bearing cone causing a thermal imbalance in the shaft [14]. This study also showed that exhaust hood flows can be clearly categorised depending on the condenser operating point. At nominal conditions, the turbine is most efficient and both sub and supersonic flows occur. At lower pressures the re-circulations move upstream and the stage is

choked. At higher pressures the re-circulations move further upstream until reaching the LSB. All these studies were carried out at uniform outlet pressures and no account was taken of the large pressure gradient which in reality occurs in water cooled steam turbine condensers.

This paper describes the addition of the rotor tip leakage jet to the public domain LP exhaust diffuser test case proposed by Burton in 2012 [8] but the key advance is to explore how the variation in exhaust hood outlet static pressure due to the flow of the condenser cooling water impacts the pressure recovery and flow structure in the exhaust hood.

## 2 LP Exhaust Diffuser Test Case

The Durham Stage and Exhaust Diffuser Test Case consists of a last stage stator and rotor design that is 0.89m long with a design speed of 3000 rpm. The blades are designed to be representative of modern aerodynamic practise but no mechanical design has been carried out on them.

The exhaust hood and diffuser has been generated from an amalgamation of previously published designs. The diffuser has a straight bearing cone geometry, as in the work of Fan [15] and an axial length of approximately 1.1 times the blade length, typical of current industrial practice for straight bearing cones. The flow guide turns from the axial to the radial direction in three equally spaced steps of  $30^\circ$ , from the work of Yoon [16]. The diffuser area ratio is 1.4, recommended by [17] for optimum diffuser performance. The expansion continues into the exhaust hood, recommended by Liu [1], with an divergence ratio from hood inlet to the half joint plane of 1.4 advised by Finzel's experimental study [11]. The other major geometry dimensions are extrapolated and scaled from the blade height when compared with those published in existing literature. Further details of the exhaust hood design process can be found in the work of Burton et. al. [8] and the CFD calculation procedure are found in a School Technical Report [https://www.dur.ac.uk/ecs/ecs\\_research/technical\\_reports/](https://www.dur.ac.uk/ecs/ecs_research/technical_reports/).

*TODO: Give this a reference and mark it "forthcoming" in the publication date. By the time the paper has finished reviewing then we should have a more precise review done.*

## 3 Computational Modelling

This calculation used a 'sequential coupling approach' to calculate the flow structure in the exhaust hood. Studies will be carried out with and without the effect of the tip leakage jet to investigate its influence. The hood outlet boundary condition will be set as a uniform static pressure and incorporating a gradient, to study the effect of the condenser on the exhaust hood flows.

### 3.1 Stage Calculations

For this study, the generic last stage blade geometries from the work of Burton in 2012 [8] were modified to include the rotor tip gap. This was set at 4.2mm (approximately 0.5%

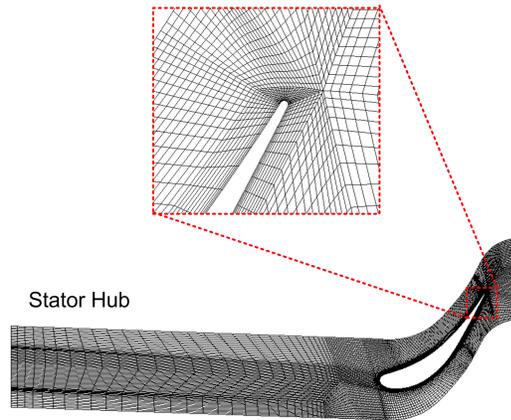


Fig. 2. Stator mesh images

of the blade height) based on discussions with a leading turbine manufacturer.

#### 3.1.1 Mesh Generation

The stage was subdivided into two domains, a single stator passage and a single rotor passage. The stator mesh generated from the work of Burton 2012 [8] was carried over for this work, Figure 2. The original grid was generated in ANSYS ICEM-CFD grid generation software. The multi-block structured grid consisted of 0.97 million cells. The wall cell width gave a  $y_+$  between 30 and 200 for use of wall functions to resolve the boundary layer.

The modified rotor blade from the original work of Burton was re-meshed in Pointwise V16-04 in order to produce a higher quality grid to successfully capture the high velocity tip jet, Figure 3. A multi-block structured grid covered the blade span with the complex tip gap modelled by an unstructured block. The use of an unstructured block in the tip region has previously yielded successful results by Verstraete in 2012 [18]. The final grid comprised of approximately 1.2 million cells with a  $y_+$  between 30 and 200.

#### 3.1.2 Calculation Set-Up

Stage flow calculations were carried out with the CFD software Fluent 12.1. The stator and rotor domains were coupled by a mass-averaged mixing plane situated equidistant between the stator trailing edge and rotor leading edge, shown in Figure 4. Representative profiles of total pressure, total temperature, swirl and pitch angle were applied at the total pressure stator inlet boundary, Figure 5, **obtained from the in the blade design process in conjunction with a leading turbine manufacturer.** The rotor outlet static pressure was set at 8800Pa to give a stage mass flow rate of 86.6 kg/s. These predictions match those carried out by the turbine manufacturer with whom the stage geometry was developed with. The turbulence was modelled using the standard  $k-\epsilon$  turbulence model with standard wall functions. The working fluid was set to wet steam with flow properties corresponding to wet steam modelled as an ideal gas, ( $C_p = 4153 \text{ J/kgK}$ ,  $\Gamma = 1.12$ , **Thermal Conductivity = 0.061 W/mK and Dynamic**

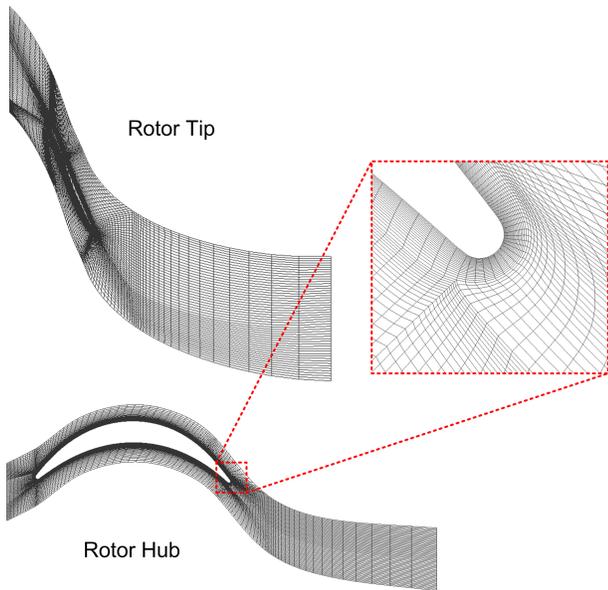


Fig. 3. Rotor mesh images

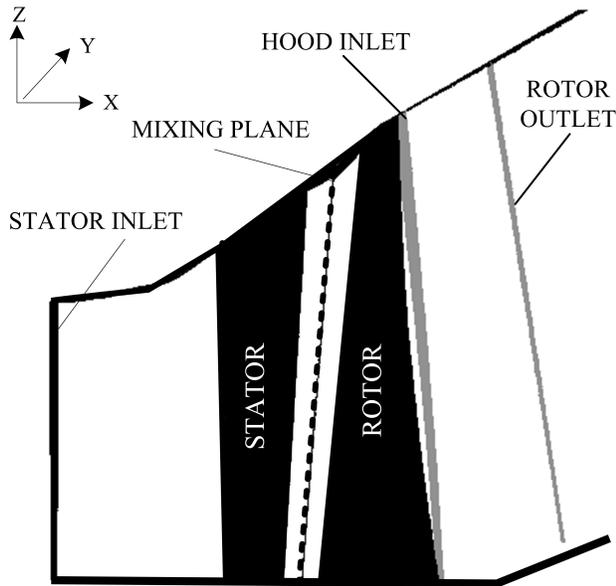


Fig. 4. Schematic of stator and rotor calculation domains

Viscosity =  $1.032 \times 10^{-5}$ ). Convergence was achieved in approximately 6000 iterations.

### 3.1.3 Stage Results

As the generic geometry aims to produce representative flow properties, the results are compared with those existing in the literature. Research from Fu and Liu in 2010 [19] indicated that total pressure and swirl angle profiles are the most significant flow properties influencing the formation of vortices within the diffuser. The Durham Last Stage Test Case with and without tip leakage jet is compared with a selection of flow profiles (with different blade geometries)

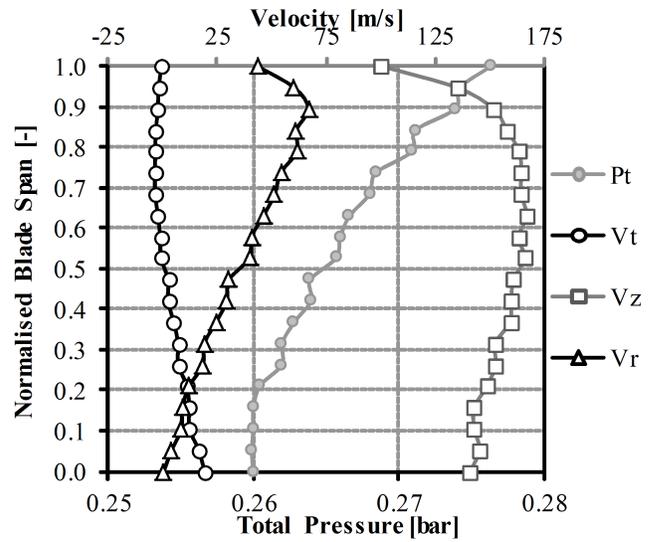


Fig. 5. Flow properties applied at stator inlet plane

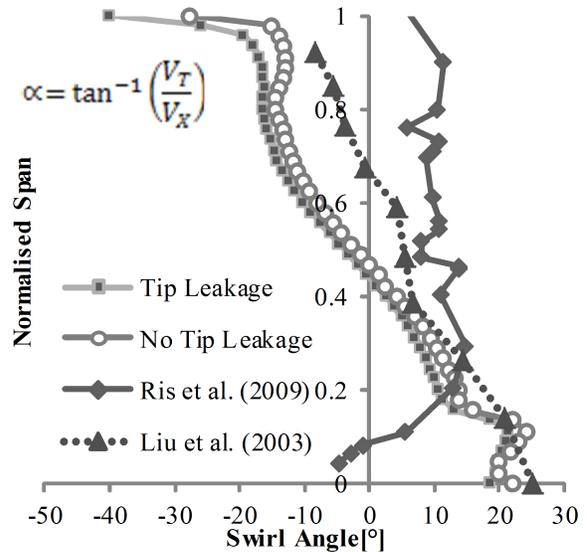


Fig. 6. Swirl angle at rotor outlet

available in the literature in Figures 6 and 7.

The swirl angle profiles in Figure 6 are consistent with profiles published by other work in the open literature, and particularly comparable the work of Liu [4]. Data from Liu et al. is extracted from a *farc115* scale air test facility where a screen and guide vane are used to simulate the swirl and pitch angle of a 300/600MW Westinghouse turbine from the 1980s. Profiles from Ris et al. are taken from a CFD simulation of an exhaust hood including baffles, operating at 53 kg/s with a rotor outlet static pressure of 3.52 kPa, the results of which were verified by experimental data. In all cases, the swirl angle is relatively small along the length of the blade,

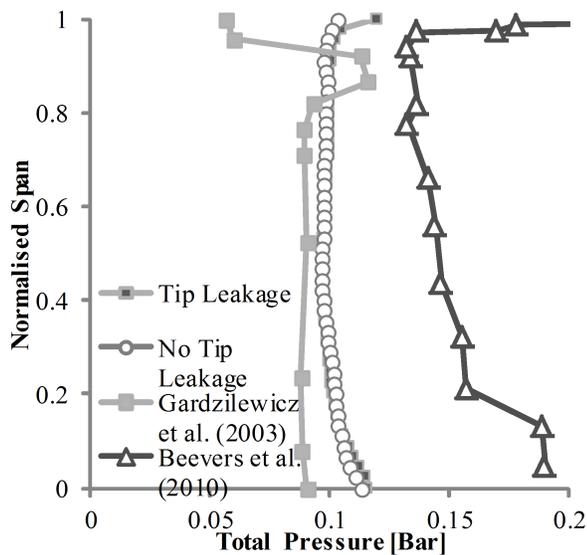


Fig. 7. Total pressure at rotor outlet

indicating that the blades are designed for minimum leaving kinetic energy losses. The magnitude of the total pressure profiles in Figure 7 cannot be compared due to the different operating conditions, but it is useful to compare the distribution shapes. The data from Gardzilewicz et al. are experimental test measurements taken from a 360MW industrial turbine. Details of the operation point are not published. The data for Beevers [20] is taken from an 10% scale test facility at Alstom Power operating at comparable Mach number to field data. As the published data does not include an x-axis scale, a zero point is assumed for comparison. The total pressure profile is again shown to be in good agreement with the literature.

Comparing results from the same blade with and without the tip leakage the high total pressure in the tip region due to the tip leakage jet has been successfully captured, and is of similar magnitude to that found in other published work. The profile along the blade span remains relatively uniform. The elevated total pressure at the hub of the blade is due to the tangential lean applied to the fixed blade in the blade design process, a feature which has been shown to have a positive effect on the bearing cone separation in the diffuser [21]. Overall, it can be seen that the tip jet has been successfully captured and the blade design can be used with confidence in the exhaust hood calculations.

## 3.2 Isolated Exhaust Hood Calculation

### 3.2.1 Geometry Modifications

Burton et al. in 2012 [8] successfully produced a generic steam turbine exhaust diffuser geometry, free from commercial restrictions, which generated a representative flow structure in the exhaust hood to facilitate research in this field. As no experimental or field data exists for this geometry, the computed flow field has been bench-marked against existing published work and has been shown to be comparable. For

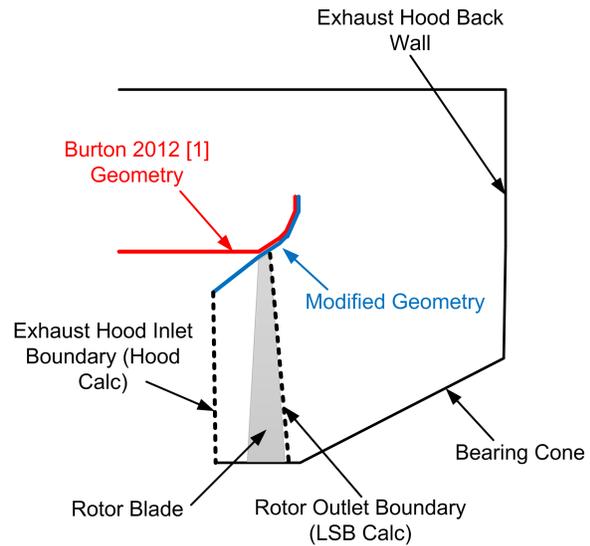


Fig. 8. Modifications to the 2012 exhaust hood geometry [8]

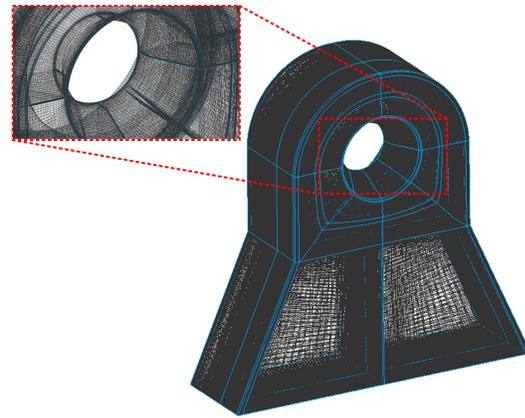


Fig. 9. Exhaust Hood Mesh

this work, the geometry has been modified and improved, as shown in Figure 8 to include the flare of flow guide so the radial velocity component could be applied at inlet to the hood in order to achieve representative pitch angles.

### 3.2.2 Calculation Set-Up

The grid generation package Pointwise V16.04 was also employed to mesh the exhaust hood, Figure 9. A structured, hexahedral mesh was applied, with a cell count of around 2.6 million. The hood outer casing, bearing cone and flow guide were specified as non-slip walls and the outer casing behind the hood inlet was set as a symmetry plane as only one LP flow was simulated. The wall cell width was set to give a  $y_+$  of between 30 and 200.

To aid convergence, the inlet boundary in the exhaust hood calculation was positioned one rotor axial chord upstream of the actual rotor trailing edge location, shown in Figure 8, a method applied by Liu [4].

The commercial flow solver, ANSYS Fluent 12.1 was used for the numerical calculations. Second order discretiza-

tion was applied and the turbulence was modelled with the standard k- $\epsilon$  turbulence model with standard wall functions. This turbulence model was selected as it is widely adopted by other researchers [22, 20, 12]. Ris et al. (2009) showed a small difference in the calculated pressure recovery (approximately 5%) with varying turbulence models [23]. With water vapour as the working fluid, flow properties were set corresponding to wet steam modelled as an ideal gas a simplification widely adopted by other workers (Stanciu et al, 2011, Ris et al, 2009). Convergence was achieved in around 10000 iterations.

Circumferentially averaged profiles of total pressure, total temperature, swirl and pitch angle, consisting of 45 points taken along the rotor trailing edge (location shown by 'Hood Inlet' in Figure 4 in the stage calculation) were applied at the hood calculation total pressure inlet boundary. As the height of the exhaust hood inlet is smaller than the height of the rotor trailing edge because of its upstream location, shown in Figure 8, profile was scaled before application at the hood inlet.

As the stage calculations were considered isolated from the exhaust hood, the exhaust hood outlet static pressure was determined by repeating the hood calculations for different values of pressure, iterating until the calculated stage mass flow rate of 86 kg/s was also achieved through the exhaust hood. This methodology is described in greater detail in Burton's 2012 work [8]. It was found that a constant exhaust hood exit static pressure of 8000Pa was required to achieve this balance in the calculations.

### 3.3 Generic Condenser Pressure Gradient

At present, most researchers [1, 14] assume a uniform outlet static pressure boundary condition at exit from the exhaust hood. However, the presence of the condenser at hood outlet means that in reality this is not the case. Variations in outlet static pressure at hood outlet are strongly dictated by the individual exhaust hood geometry and the condenser, so for this work a generic outlet condenser static pressure variation has been developed.

The outlet conditions developed in this paper are based on field data taken from a 700MW steam power plant. Field data typically features a large variation in static pressure due to "exhaust-specific" features, such as separations from the condenser neck walls, blockages due to internal furniture such as the bled steam pipework and the cores of the pair of counter-rotating vortices which form from the separations generated in the exhaust diffuser that progress down through the condenser neck.

Removing the hood specific pressure variations, a "generic outlet" was generated as a percentage pressure variation between left hand and right hand sides of the exhaust hood relative to the average exhaust hood outlet static pressure. This variation is purely as a result of the condenser cooling water flow. As the cooling water flows from inlet to outlet in the condenser tube nests, its temperature gradually increases giving the pressure gradient shown, from 10% below average at the cooling water inlet, to 7% above av-

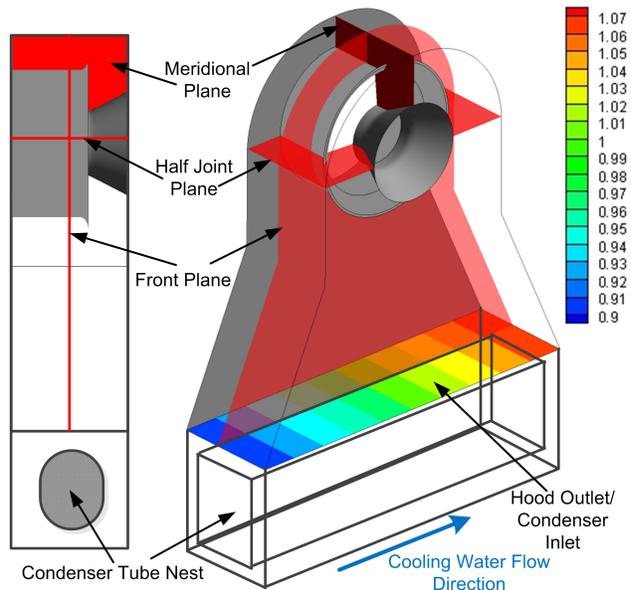


Fig. 10. Diagram of condenser pressure variation

erage level at the cooling water outlet, shown in Figure 10 which also shows the three planes (meridional plane, half joint plane and front plane) used to present the results.

## 4 Results

### 4.1 Tip Leakage Modelling

The tip gap was set at 4.2mm (approximately 0.5% of the blade height) tip gap. The addition of the tip leakage jet significantly influences the flow structure in the exhaust hood calculation. Contours of static pressure at the meridional plane are shown in Figure 11 the corresponding tangential velocity contours are shown in 13. Pressure contours for the half joint plane are shown in Figure 12.

#### 4.1.1 Pressure Recovery

There is a significant rise in the predicted value of  $C_p$  when the tip leakage jet is included in the simulations, from -0.035 without the tip leakage jet to 0.236. This is a similar magnitude (0.01 to 0.398) to that observed by Li [12] in a comprehensive CFD study of tip leakage flows, giving confidence in the reliability of the results presented here.

The increase in pressure recovery is due to the high velocity jet energizing the flow along the flow guide, reducing the size of the low pressure region, shown in Figure 11. The magnitude of the low pressure region behind the flow guide has also reduced and the strength of the low pressure vortex core has decreased.

#### 4.1.2 Flow Asymmetry

The flow is significantly more asymmetric with the addition of tip leakage jet, as shown in Figures 12. The magnitude of the low pressure regions on both the left and right hand sides of the exhaust hood has decreased with the addition of the tip leakage jet due to the reduced vortex strength

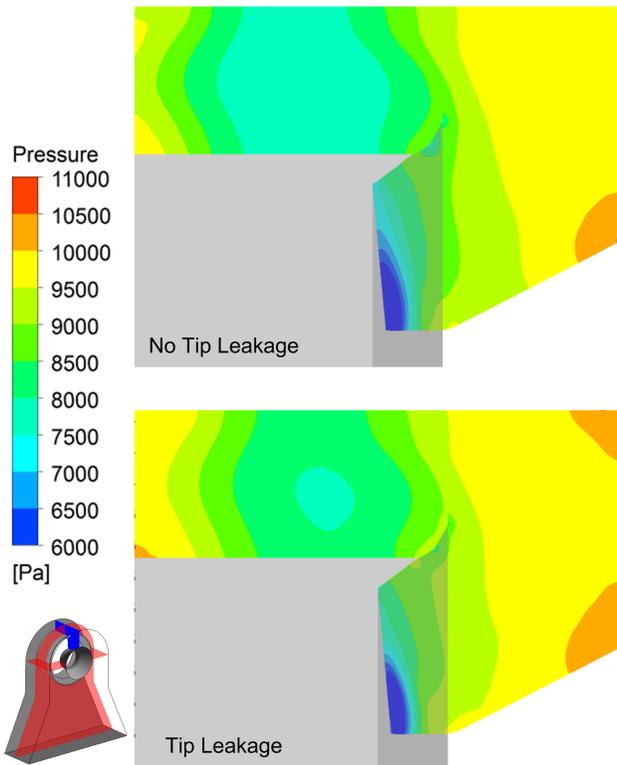


Fig. 11. Static pressure contours at the meridional plane

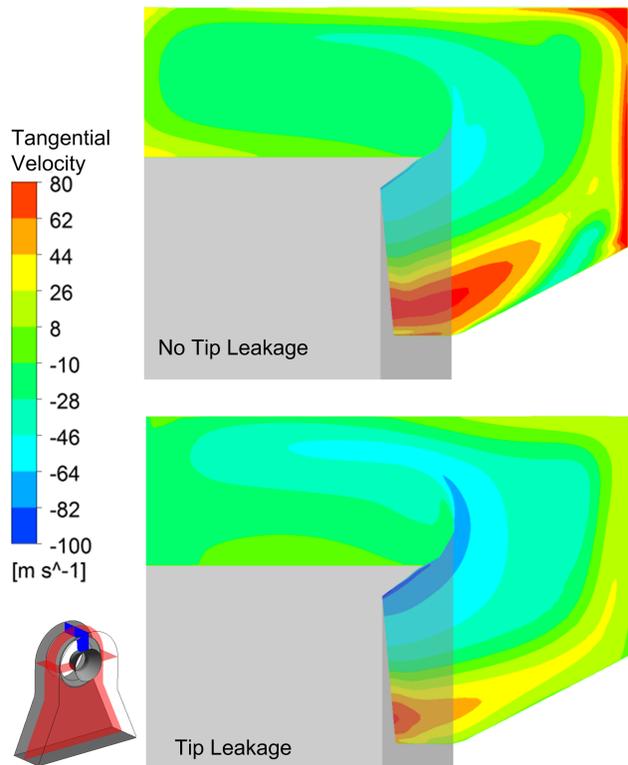


Fig. 13. Tangential velocity contours at the meridional plane

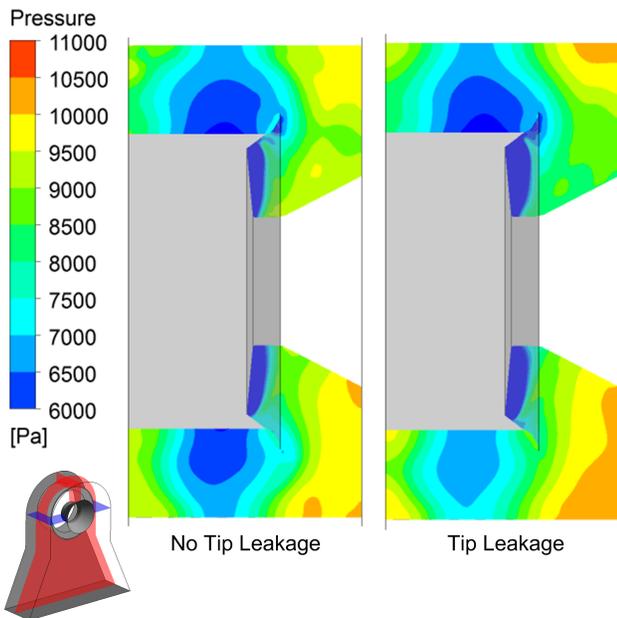


Fig. 12. Static pressure contours at the half joint plane

formed in the upper exhaust hood. However, the magnitude of this reduction is different for the two sides. This is due to the change in inlet circumferential velocity, as the high velocity jet adds more flow at a higher radius, increasing the swirl at the tip and driving more fluid to the left hand side of the exhaust hood, as shown in Figure 13.

## 4.2 Condenser Pressure Gradient

The effect of non-uniform hood outlet pressure distribution was investigated by taking the tip leakage simulation solution and applying the non-uniform hood outlet boundary condition and continuing the calculation until convergence, in around 2000 additional iterations. This was an iterative process, as the average outlet static pressure of the pressure profile was set to give the same mass flow rate as in the non-tip/tip leakage calculations. The required average hood exit pressure was found to be 7800Pa.

### 4.2.1 Flow Asymmetry

The application of a non-uniform hood outlet boundary condition was shown to have an attenuating effect on the asymmetry of the flow with the tip leakage jet included, as shown in Figure 12. Due to the direction of the condenser cooling water flow, the pressure gradient works against the flow asymmetry, decreasing the magnitude of the asymmetry between the left and right hand sides of the exhaust hood, see Figure 14.

However, the magnitude of the low pressure vortex core has increased due to the large low pressure volume in the condenser neck, giving an overall poorer pressure recovery with the predicted  $C_p$  value decreasing to 0.1839 from the previous value of 0.236 calculated before the addition of the non-uniform outlet condition.

The improvement in the asymmetry characteristic of the exhaust hood noticed in Figure 14 is highly dependent on the direction of the condenser cooling water flow (or conversely the direction of rotation of the moving blades), which

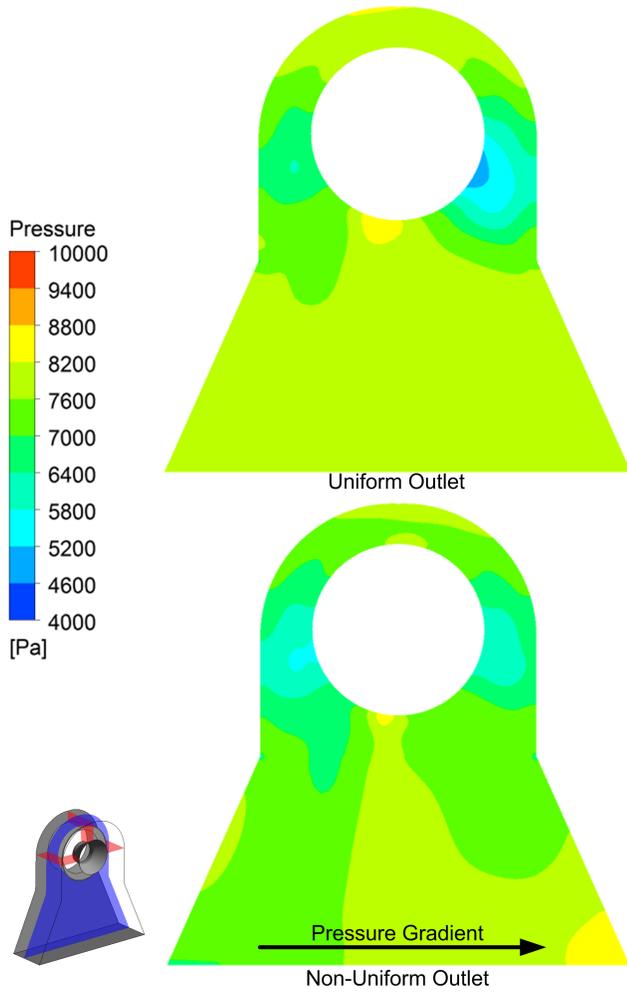


Fig. 14. Diagram of condenser pressure variation

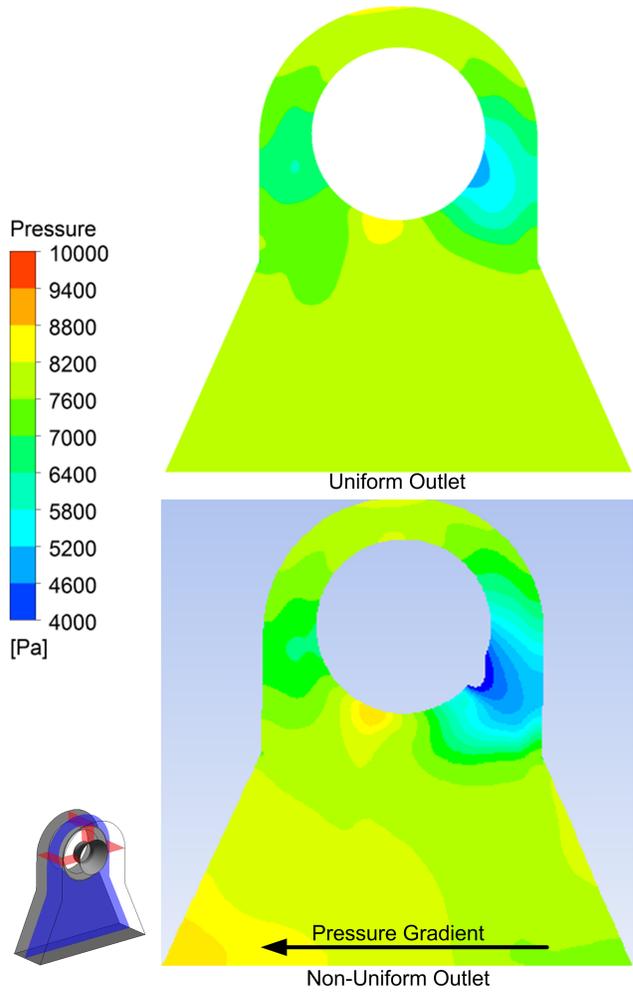


Fig. 15. Diagram of reversed condenser pressure variation

is not standard between power plant designs. Reversing the direction of the pressure gradient sees significantly increased asymmetry, Figure 15 and a subsequently reduced pressure recovery potential of 0.167.

## 5 Discussion

The public domain LP exhaust diffuser test case proposed by Burton in 2012 has been modified to include the effect of tip leakage on diffuser flows. The inclusion tip leakage jet has shown to significantly increase the static pressure recovery of the exhaust hood, consistent with findings in other published research. This is due to the additional momentum of the jet reducing the low pressure region near the hood flow guide. However, the asymmetry of the flow increases due to the additional swirl in the tip leakage flow. The changes in  $C_p$  are shown in Table 1.

The key advance in this paper is that calculations were conducted with a representative condenser pressure gradient. It is shown that accurately modelling the exit boundary calculation has a large influence on the flow structure and to a lesser extent the pressure loss recovery inside the exhaust diffuser.

Table 1.  $C_p$  for Different Configurations

Tip Leakage	Outlet Pressure	$C_p$	$\Delta C_p$
No Tip Leakage	Uniform	-0.035	-
Tip Leakage	Uniform	0.236	+0.271
Tip Leakage	Gradient	0.186	-0.05
Tip Leakage	Reversed Gradient	0.167	-0.02

In order to assess the significance of the changes caused by using the outlet pressure gradient on diffuser performance a short survey of other author's work in changing aspects such as turbine exit conditions, the inclusion of internal reinforcements, bearing cone and other optimisations and the inclusion of circumferential non-uniformities were examined.

Liu's experimental study in 2003 [4] showed a  $C_p$  of 0.350 when using an uniform inlet boundary condition but a  $C_p$  of -0.218 when using non-uniform inlet boundary with a swirl and total pressure representative of a turbine exit. This leads to an  $\Delta C_p$  of around 0.57.

Typically, an exhaust diffuser flow guide and bearing cone optimisation studies yield improved  $C_p$  of around 0.3, ranging from 0.26 [24] to 0.38 [16].

Tajc studied the effect of internal reinforcements on  $C_p$  on a range of hood geometries and found an average increase in loss coefficient of 0.138 when the exhaust hood furniture was modelled [10].

Burton et al. [25] study compared the mixing plane method of coupling the exhaust hood to the turbine with the **non-linear harmonic (NLH) approach**. The accurate modelling of the circumferential asymmetry at the exhaust hood inlet with the latter approach was found to yield  $\Delta C_p$  of 0.025 in generous axial length diffusers. For more compact diffusers this was to increase to 0.095.

The results of this short survey are summarised in Table 2 also included is an estimate of the  $\Delta C_p$  introduced by changing the turbulence model estimated from Ris's [23] data. This suggests that the key variable in modelling exhaust diffusers is to ensure that representative boundary conditions are applied from the turbine exit. This is perhaps obvious in hindsight as in any CFD solution selection of the correct boundary conditions is essential to obtaining a correct solution.

It is also clear that the influence of a representative pressure gradient at exit from the condenser is small, this means that if one wishes to improve diffuser performance the largest gains will come from geometrical optimisations of the flow path or the exhaust furniture. However as the state of the art of diffuser design advances to make further gains consideration of the influence of exhaust pressure gradient will become necessary.

## 6 Conclusions

1. Computations were carried out on the "IP Free" Durham Stage and Exhaust Diffuser Test Case using the a 3D RANS solver. A sequential approach was taken to the calculations.
2. Tip leakage modelling was included in the stage calculations. As expected this had a significant influence on the pressure recovery coefficient increasing it by 0.27.
3. The asymmetry of the condenser cooling water flow pressure gradient was applied to exhaust hood calculations for the first time.
4. Significant flow field variations were observed when this asymmetry was applied. The asymmetry of the flow in the upper exhaust hood is significantly reduced as condenser pressure gradient works against the hood asymmetry. However the changes in pressure recovery coefficient were not as significant with a maximum change of around 0.05.
5. This change is smaller than other authors have observed when adding exhaust reinforcements or conducting optimisation studies of bearing cones and flow guides. The current state of the art in diffuser design does not therefore justify the use of an asymmetric condenser cooling water boundary condition as more significant effects must be accounted for first.

6. As the state of the art in diffuser design advances and smaller gains are achievable this boundary condition will need to be included to extract the best performance.

## References

- [1] Liu, J., and Hynes, T. P. "The investigation of turbine and exhaust interactions in asymmetric flows: Part 2 - turbine-diffuser-collector interactions". In ASME Turbo Expo, Amsterdam, Netherlands, GT2002-30343, pp. 1–10.
- [2] Fu, J.-L., Liu, J.-J., and Zhou, S.-J., 2012. "Unsteady interactions between axial turbine and nonaxisymmetric exhaust hood under different operational conditions". *Journal of Turbomachinery*, **134**(4), p. 041002.
- [3] Burton, Z., Ingram, G. L., and Hogg, S., 2013. "A literature review of low pressure steam turbine exhaust hood and diffuser studies". *Journal of Engineering for Gas Turbines and Power*, **135**(6), pp. 1–10.
- [4] 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". *Journal of Engineering for Gas Turbines and Power*, **125**(1), pp. 292–299.
- [5] Gardzilewicz, A., Swirydczuk, J., Badur, J., Karcz, M., Werner, R., and Szyrejko, C., 2003. "Methodology of CFD computations applied for analysing flow through steam turbine exhaust hoods". *Transactions of the Institute of Fluid-Flow Machinery*, **113**, pp. 157–168.
- [6] Kreitmeier, F., and Greim, R., 2003. "Optimization of blade-diffuser interaction for improved turbine performance". *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, **217**(4), pp. 443–451.
- [7] Tindell, R., Alston, T., Sarro, C., Stegmann, G., Gray, L., and Davids, J., 1996. "Computational fluid dynamics analysis of a steam power plant low-pressure turbine downward exhaust hood". *Journal of Engineering for Gas Turbines and Power*, **118**, pp. 214–224.
- [8] Burton, Z., Ingram, G. L., and Hogg, S. "A generic low pressure exhaust diffuser for steam turbine research". In ASME Turbo Expo, Copenhagen, Denmark, GT2012-68485.
- [9] Benim, A., Geiger, M., Doehler, S., Schoenenberger, M., and Roemer, H., 1995. "Modelling the flow in the exhaust hood of steam turbines under consideration of turbine-exhaust hood interaction". *VDI BERICHTE*, **1185**, pp. 343–343.
- [10] Tajc, L., Bednar, L., Sikova, I., Feldberg, L. A., and Goudkov, E. I. "The experimental investigation of the influence of the flow swirl and tip clearance jet on aerodynamic characteristics of exhaust hoods". In Engineering Mechanics, Svatka, Czech Republic, SKODA Power, pp. 1–10.
- [11] Finzel, C., Schatz, M., Casey, M. V., and Gloss, D. "Experimental investigation of geometrical parameters on the pressure recovery of low pressure steam turbine

Table 2.  $\Delta C_p$  for Different Changes

Author	Change Made	Typical $ \Delta C_p $
Liu	Diffuser boundary conditions	0.57
Wang or Yoon	Geometrical optimisation	0.30
Tajc	Exhaust reinforcements modeled	0.14
Burton	Circumferential asymmetry	0.10
Ris	Change of turbulence model	0.01
Present	Condenser pressure gradient	0.05

- exhaust hoods”. In ASME Turbo Expo, Vancouver, Canada, GT2011-45302.
- [12] Li, J., Li, Z., and Feng, Z. “Effects of the last stage rotor tip leakage flow on the aerodynamic performance of the exhaust hood for steam turbines”. In ASME Turbo Expo, San Antonio, Texas, GT2013-94377.
- [13] Punnonen, P., Kaikko, J., Backman, J., and Savolainen, S. “Modelling of nuclear power plant steam turbines”. In 10th European Turbomachinery Conference, Lappeenranta, Finland.
- [14] Stanciu, M., Marcelet, M., and Dorey, J.-M. “Numerical investigation of condenser pressure effect on last stage operation of low pressure wet steam turbines”. In ASME Turbo Expo, San Antonio, Texas, GT2013-94070.
- [15] Fan, T., Xie, Y., Zhang, D., and Sun, B. “A combined numerical model and optimization for low pressure exhaust system in steam turbine”. In ASME Power Conference, San Antonio, Texas, POWER2007-22147.
- [16] Yoon, S., Stanislaus, F. J., Mokulys, T., Singh, G., and M., C. “A three-dimensional diffuser desing for the retrofit of a low pressure turbine using in-house exhaust design system”. In ASME Turbo Expo, Vancouver, Canada, GT2011-45366.
- [17] Hoznedl, M., Tajc, L., Krejcik, J., Bednar, L., Sedlak, K., and Linhart, J., 2009. “Exhaust hood for steam turbines-single-flow arrangement”. *Frontiers of Energy and Power Engineering in China*, **3**(3), pp. 321–329.
- [18] Verstraete, T., Prinsier, J., Di Sante, A., Della Gatta, S., and Cosi, L. “Design optimization of a low pressure steam turbine radial diffuser usding an evolutionary algorithm and 3d cfd”. In ASME Turbo Expo, Copenhagen, Denmark, GT2012-69515.
- [19] Fu, J., and Liu, J. “Investigation of influential factors on the aerodynamic performance of a steam turbine exhaust system”. In ASME Turbo Expo, Glasgow, UK, GT2010-22316.
- [20] Beevers, A., Congiu, F., Pengue, F., and Mokulys, T. “An analysis of the merits of cfd for the performances prediction of a low pressure steam turbine radial diffuser”. In ASME Turbo Expo, Glasgow, UK, GT2010-22107.
- [21] Fu, J.-L., and Liu, J.-J., 2008. “Influences of in-flow condition on non-axisymmetric flows in turbine exhaust hoods”. *Journal of Thermal Science*, **17**(4), pp. 305–313.
- [22] Zhou, S., Liu, J., and Fu, J., 2007. “Experimental and numerical investigation of interaction between turbine stage and exhaust hood”. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, **221**(7), pp. 991–999.
- [23] Ris, V. V., Simoyu, L. L., Galaev, S. A., Gudkov, N. N., Kirillov, V. I., Smirnov, E. M., Kirillov, A. I., and Ermolaev, V. V., 2009. “Numerical simulation of flow in a steam turbine exhaust hood: Comparison results of calculations and data from a full-scale experiment”. *Thermal Engineering*, **56**(4), pp. 277–283.
- [24] Wang, H., Zhu, X., and Du, Z., 2010. “Aerodynamic optimization for low pressure turbine exhaust hood using kriging surrogate model”. *International Communications in Heat and Mass Transfer*, **37**(8), pp. 998–1003.
- [25] Burton, Z., Ingram, G. L., and Hogg, S. “A novel method of coupling the steam turbine exhaust hood and the last stage blades using the non-linear harmonic method”. In ASME Turbo Expo, San Antonio, Texas, GT2013-94184.