Proceedings of
10th European Conference on Turbomachinery Fluid dynamics & Thermodynamics
ETC10, April 15-19, 2013, Lappeenranta, Finland

A GENERIC STEAM TURBINE EXHAUST DIFFUSER WITH TIP LEAKAGE MODELLING AND NON-UNIFORM HOOD OUTLET

Zoe Burton, Dr Grant Ingram and Dr Simon Hogg

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

ABSTRACT

This paper advances the work of Burton et al. (2012) to highlight the significance of the last stage blade tip leakage jet on the flow structure in the low pressure (LP) steam turbine exhaust hood, widely regarded as being significant in other published research. The flow structure in the exhaust hood is distinctly asymmetric when the leakage jet is included due to the increased swirl. This paper offers the new contribution of the effect of the condenser on the flow structure within the exhaust hood. It has been shown that the pressure gradient at the condenser inlet due to the heating of the cooling water reduces the asymmetry caused by the tip leakage jet. Improvements have been made to Durham University's public domain LP exhaust diffuser with accompanying last stage blades geometry to include the flare of the casing.

NOMENCLATURE

$C_p = \frac{P_2 - P_1}{P_{t1} - P_1}$	Pressure Recovery Factor
P	Pressure [Pa]
P_t	Total Pressure [Pa]
α	Swirl Angle [º]
β	Pitch Angle [º]
V	Absolute Velocity [m/s]

Subscripts

R	Radial
T	Tangential
X	Axial
1	Hood/Diffuser Inlet
2	Hood Outlet

INTRODUCTION AND LITERATURE REVIEW

This paper build on the numerical simulations carried out by Burton et al. (2012) on a generic steam turbine last stage blade and accompanying exhaust geometry to include tip leakage, an aspect widely considered to be important by other researchers, (Finzel et al. 2011, Benim et a. 1995, Tajc et al. 2006) and the effects of the condenser cooling water flow. The exhaust hood of steam turbine is 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 (Liu and Hynes, 2002, Fu and Liu, 2012), which at the higher end of this range gives a lower turbine exit pressure than that in the condenser. Keller (1986) equates a 0.7 increase in C_p from -0.4 with a 5MW gain for a 1000MW plant.

Pressure recovery is lower when the condenser is positioned beneath the turbine, as the flow is required to turn sharply through 90° from the axial to the radial direction in a relatively short axial distance, generating a highly vortical flow. 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. The non-axisymmetric hood geometry results in the flow downstream of the turbine developing a circumferentially non-uniform distribution which is thought by Benim et al. (1995) to facilitate the formation of vortices within the diffuser, shown in Figure 1.

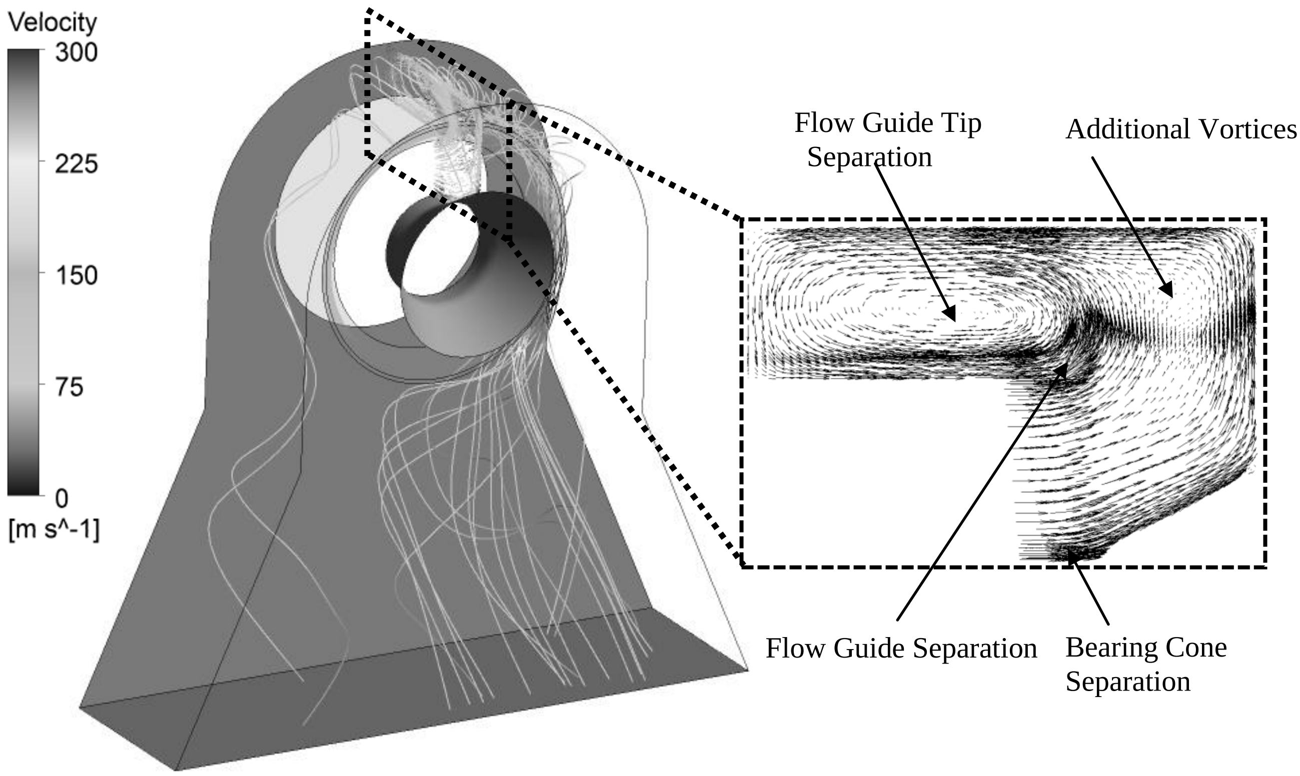

Figure 1: Flow Structure in the Exhaust Hood & Vortex Formation in the Diffuser (Burton et al., 2012)

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 unsteady rotational flow field of the turbine coupled to the accompanying exhaust hood with full internal furniture is 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.

One common method of simplification is to consider the exhaust hood isolated from the last stage blades, but it is vital to ensure the inlet boundary conditions are accurate to give a representative flow structure. In the mid-1990s several authors demonstrated that applying uniform flow properties at inlet produced inaccurate flow structures (Liu and Hynes, 2002b, Tindell et al., 1996). Applying empirically determined radial distributions of pressure, temperature, swirl and pitch angles, taken downstream of the rotor trailing edge, were found to give indicative flow-fields at a reduced computational cost (Liu et al., 2003, Gardzilewicz et al., 2003, Kreitmeier and Greim, 2002). However, in the last 15 years it has become increasingly apparent that the interaction between the last stage blades and the exhaust hood is important, and subsequently the most accurate numerical simulations are when the two systems are coupled.

Denton's mixing plane approach has been adopted by multiple researchers, including Fan (2007), to couple the two systems. Although representative flow-field results were achieved, the method circumferentially averages over the hood inlet plane, which removes the inlet asymmetry due to the non-axisymmetric hood geometry, a feature which was attributed by Benim et al (1995) to play an "important role" in the formation of vortices within the exhaust hood. Zhou et al. (2007) advised modelling all blade passages to capture the asymmetry of the inlet flow, but this significantly increases the size of the computation. Application of the Frozen Rotor approach has become an increasingly popular method (Zhou et al., 2007, Verstraete et al., 2012, Li et al., 2012) in recent years to capture the circumferential inlet variation as all blade passages are modelled but the rotor remains in a fixed position, reducing the computational demand compared to the full annulus unsteady calculations. However computation times remain long, over 250 hours, due to the high cell count in excess of 50M, meaning the Frozen Rotor methodology is only practical as a "research and validation tool, rather than a design tool." (Verstrete et al., 2007)

The first comprehensive coupled model was developed by Benim et al. (1995), and is still in use by a leading commercial company today (Beevers et al., 2010, Yoon et al., 2011). For an assumed turbine back pressure, the flow field through the turbine is calculated with this flow field then applied to the hood inlet. The hood calculation then yields a new back pressure for the turbine, for which the flow field is recalculated and the process repeats until convergence is achieved at a constant back pressure condition. Stanciu et al. (2011) and Gardzilewicz et al. (2003) have both compared the flow field results from different methodologies but at present, no single `best practice' approach has come to the forefront which is thought to consistently achieve accurate results.

Despite the division of opinion over the best simplification 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 et al. (1995). More recent experimental work by Tajc et al. (2006) 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. Investigations by Finzel et al. (2011) showed the separation point moved downstream until it was completely suppressed with increasing jet strength.

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 (Fan et al., 2007, Fu et al., 2012). 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. Sometimes this boundary is extended, as in the work of Li et al. (2012), to ensure the reverse flow structure in the hood is not impacted by the static pressure applied at this boundary. In reality, the outlet boundary condition is inherently linked to the condenser, as the exhaust hood inlet is to the last stage blades. With the majority of power plants using surface condensers, the flow of the cooling water across the condenser means there is a pressure gradient across the exhaust hood outlet due to the heating of the cooling water flow. Senoo et al. (2004) is one of the only researchers to investigate the effect of the condenser on the exhaust hood flow structure. Numerically coupling the condenser and the hood revealed that the pressure distribution in the condenser neck is significantly influenced by the change in cooling water temperature across the condenser.

This paper aims to highlight the impact the tip leakage jet has on the performance of a generic exhaust hood geometry and outline how far 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.

TIP LEAKAGE MODELLING

Stage Calculations

For this study, the generic last stage blade geometries from Burton et al. (2012) were modified to include the rotor tip gap. This was set at 4.2mm (approximately 0.5% of the blade height) based on the tip gap size supplied by a leading turbine manufacturer, upon whose blade the generic last stage blade geometry is based.

Numerical Set-Up

It is widely recognised that one of the major problems in the computational modelling of steam turbine exhaust hoods is producing a representative flow structure at a practical computational demand. Many approaches to simplification have yielded successful results but at present no single method has become ‘best practice’ The approach adopted by Liu et al. (2002b) of modelling the exhaust hood isolated from the stage has been selected as it has been shown the yield results comparable with field data at a significantly reduced computational cost. This requires the stage calculations to be carried out in isolation and flow profiles, taken from downstream of the trailing edge of the last stage rotor blades, to be applied at the inlet boundary of the exhaust hood calculation.

Hence, in this study, the stage calculations were carried out isolated from the exhaust hood. The stator mesh from Burton et al. (2012) was carried over to this work. The rotor mesh required modification to include the tip leakage gap. The rotor was re-meshed in Pointwise V16.04 in order to produce a higher quality grid to successfully capture the high velocity tip jet,. A structured grid was generated for the blade span with the complex tip gap geometry modelled using an unstructured block. Although it is desirable to have a fully structured grid to more accurately capture the boundary layer flows, the highly twisted blade with the large flare to the annulus prevented a good quality structured mesh from being generated in the tip region. The approach of using an unstructured grid in the tip gap was adopted by Verstraete et al. (2012) yielding successful results. The stator mesh has 900k cells with the rotor mesh around 1.2M cells.

The flow calculations were carried out using the commercial CFD software Fluent 12.1. The stage was subdivided into two domains, stator and rotor, with the domains coupled by a mass-averaged mixing plane situated equidistant between the stator trailing edge and rotor leading edge, as shown in Figure 2B. Representative profiles of total pressure and three components of velocity were applied at inlet to the stator, shown in Figure 2A, along with a representative total temperature profile. The rotor outlet static pressure was set at 8800Pa to give a stage mass flow rate of 86.6 kg/s, matching some preliminary calculations carried out with the OEM partner from whom the stage geometry was obtained. The turbulence was modelled using the standard k-ε turbulence model. Standard wall functions were used to give a manageable cell count. Wet steam was set as the working fluid in Fluent with flow properties corresponding to wet steam modelled as an ideal gas. Convergence was achieved in around 6000 iterations.

Calculation Results

Flow profiles (taken along the trailing edge of the rotor at the location indicated by the broken line in Figure 2B) considered important in other published research (Fu and Liu, 2010) are compared with profiles from other studies in Figure 3 and show good agreement. The profiles compared (except Burton and Durham tip leakage) are for blades of a different geometries, and are included to illustrate that the generic blade geometry used in the present study produces profiles that are consistent with the flow structure of the blades used by some other workers.

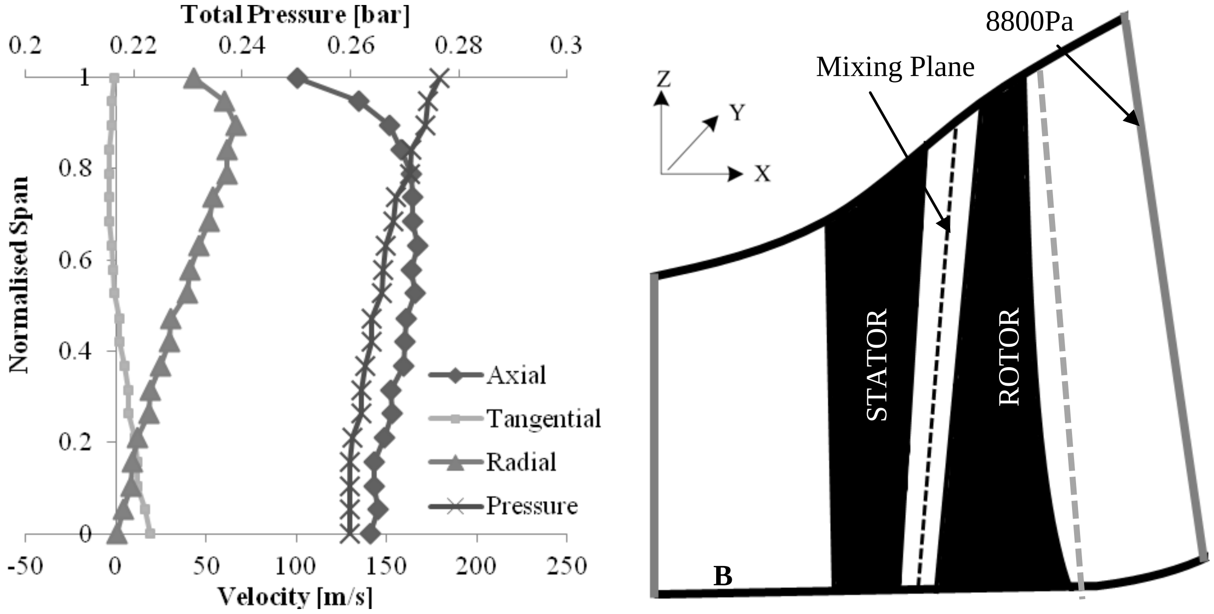

Figure 2: A) Flow Profiles Applied at Stator Inlet Plane B) Schematic of Stage Domains

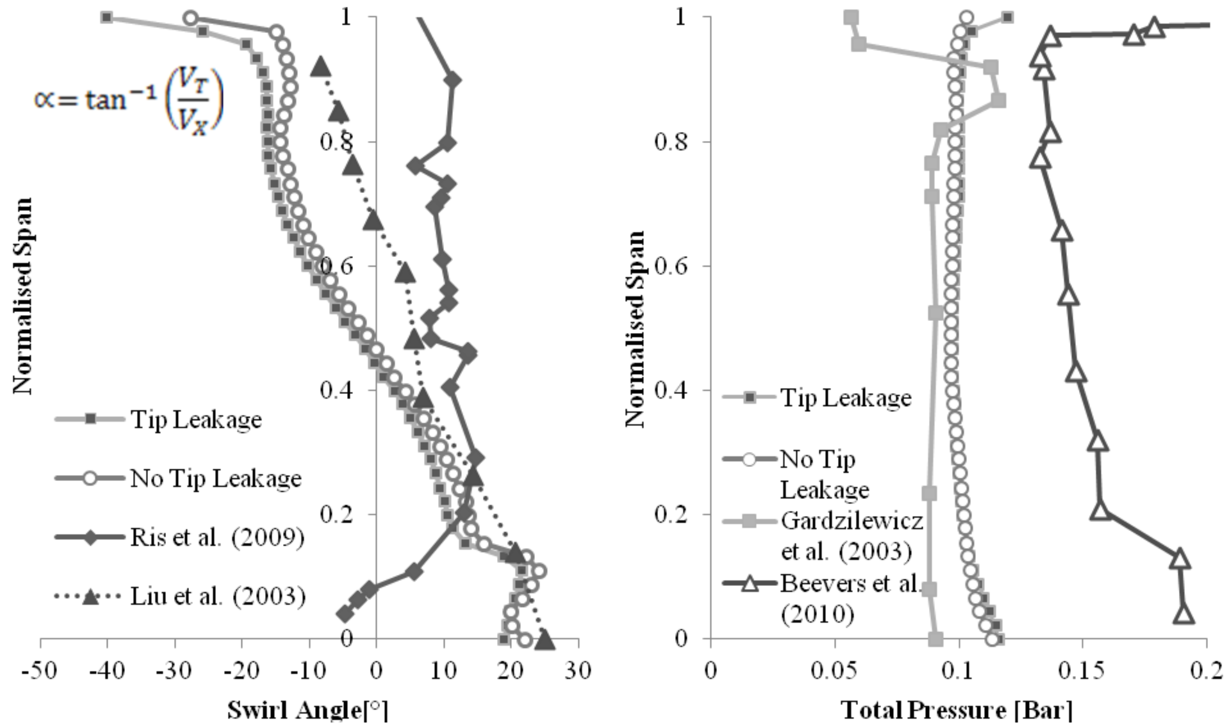

Figure 3: A) Comparison of Swirl Angle Profiles with Literature B) Comparison of Total Pressure Profiles with Literature

The swirl angle profiles in Figure 3A are also consistent with profiles published by other workers in the open literature, and particularly comparable the work of Liu et al. (2003). The swirl angle is relatively small along the length of the blade, indicating that the blades are designed for minimum leaving kinetic energy losses. The magnitude of the total pressure profiles in Figure 3B cannot be compared due to the different operating conditions, but it is useful to compare the distribution shapes. The data for Beevers et al. (1995) is taken from an assumed zero as the published data does not include an x-axis scale. 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 (No Tip

Leakage vs. 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 (Figure 1) in the diffuser (Burton et al., 2012, Fu and Liu, 2008). Overall, it can be seen that the tip jet has been successfully captured and its effect on the flow structure in the exhaust hood can be investigated.

NON-UNIFORM EXHAUST HOOD OUTLET BOUNDARY

At present, most researchers 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 condenser static pressure variation has been developed, free of IP restrictions to facilitate more work in this area.

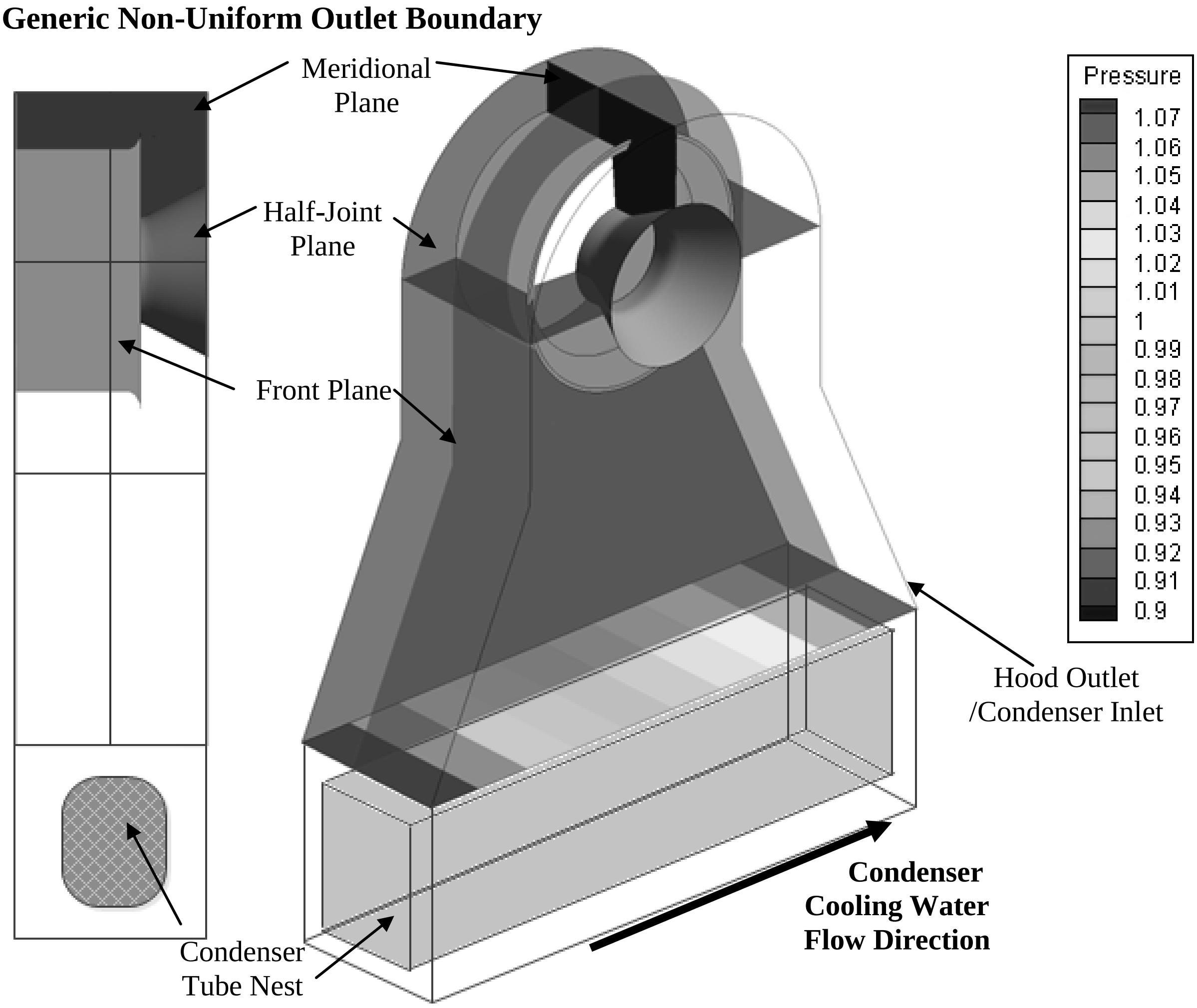

Figure 4: Schematic of the Hood and Condenser with Pressure Variation due to Cooling Water Flow (contours show dimensionless pressure level relative to the mean value over the hood exit plane).

The `generic 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, the generic outlet pressure field, shown in Figure 4, 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 average level at the cooling water outlet.

This hood exit pressure boundary condition was applied to the generic hood geometry with tip leakage, to investigate the impact of representative hood outlet conditions on the flow structure within the exhaust hood.

EXHAUST HOOD CALCULATION RESULTS

Generic Geometry Modification

Burton et al. (2012) 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 benchmarked against existing published work and has been shown to be comparable. For this work, the geometry has been modified and improved, as shown in Figure 5 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.

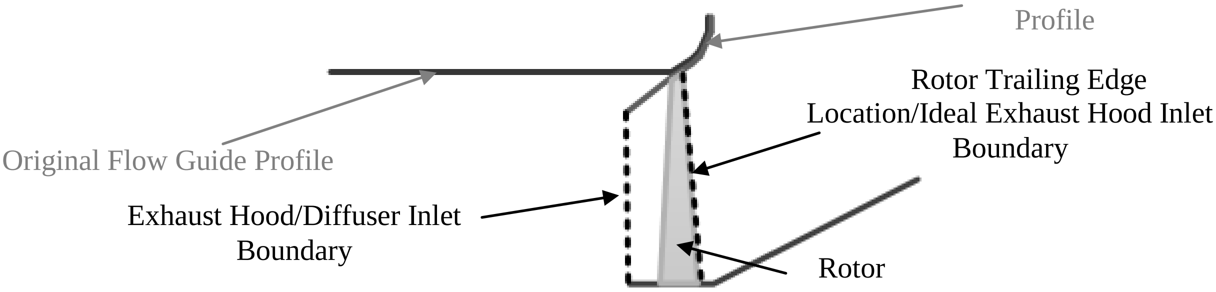

Figure 5: Original (Burton et al., 2012) and Modified Generic Exhaust Diffuser Geometries

Numerical Set-Up

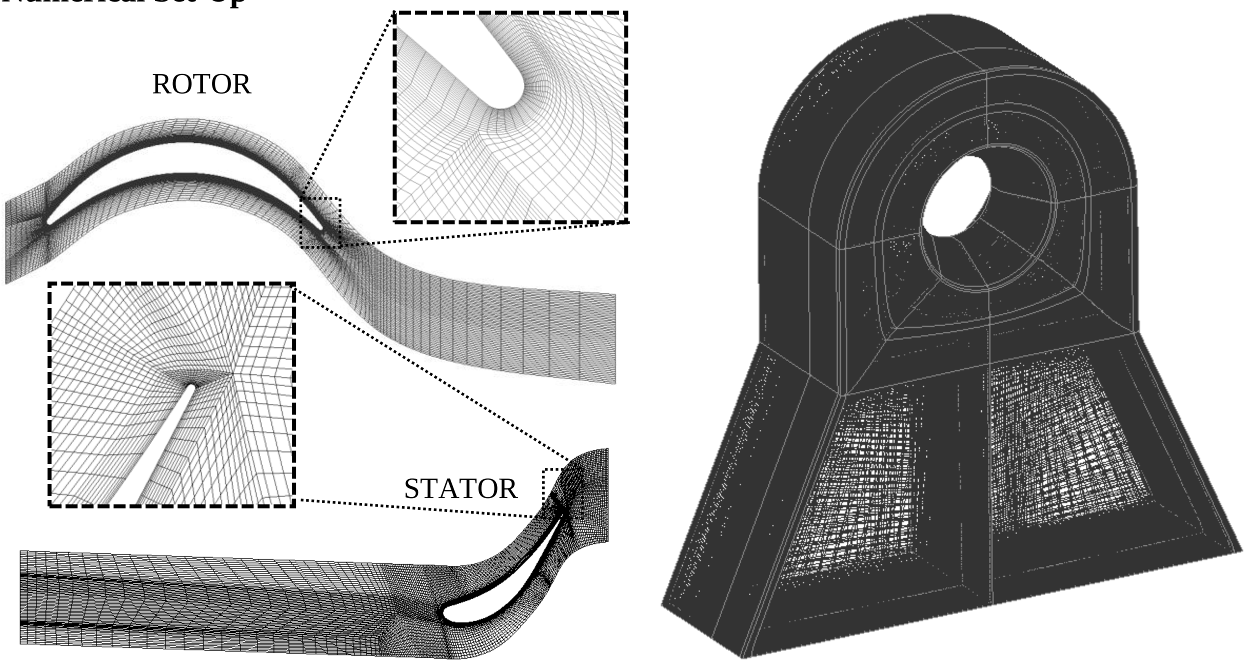

Figure 6: Stator, Rotor and Exhaust Hood Meshes

The grid generation package Pointwise V16.04 was employed to mesh the exhaust hood. A structured, hexahedral mesh was applied, with a cell count of around 2.6M, Fig. 6. 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.

To aid convergence, the inlet boundary in the exhaust hood calculation was positioned one rotor axial chord upstream of the rotor trailing edge location, shown in Figure 5, a method applied by Liu et al. (2003). The commercial flow solver, Fluent 12.1 was used for the numerical calculations. Second order discretization was applied and the turbulence was modelled with the standard k-ε turbulence model with standard wall functions. This turbulence model was selected as it is widely adopted by other researchers. Ris et al. (2009) showed little difference in the calculated pressure recovery with varying turbulence models. 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 average profiles of total pressure, total temperature, swirl and pitch angle, consisting of 45 points taken along the rotor trailing edge in the stage calculation were applied at the hood calculation inlet boundary. As the height of the exhaust hood inlet is smaller than the height of the rotor trailing edge, Figure 5, the 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 et al. (2012). It was found that a constant exhaust hood exit static pressure of 8000Pa was required to achieve this balance in the calculations.

Impact of Leakage

As has been shown by many other researchers, the addition of the tip leakage jet significantly influences the flow structure in the exhaust hood calculation.

Pressure Recovery

In line with other published work, there is a significant rise in the predicted value for C_p when tip leakage is included in the simulations. C_p changes from -0.035 to 0.236. As shown in Figure 7B, the high velocity jet has energized the flow along the flow guide, reducing the size of the low pressure region compared with Figure 7A. The magnitude of the low pressure region behind the flow guide has also reduced and the strength of the low pressure vortex core is decreased.

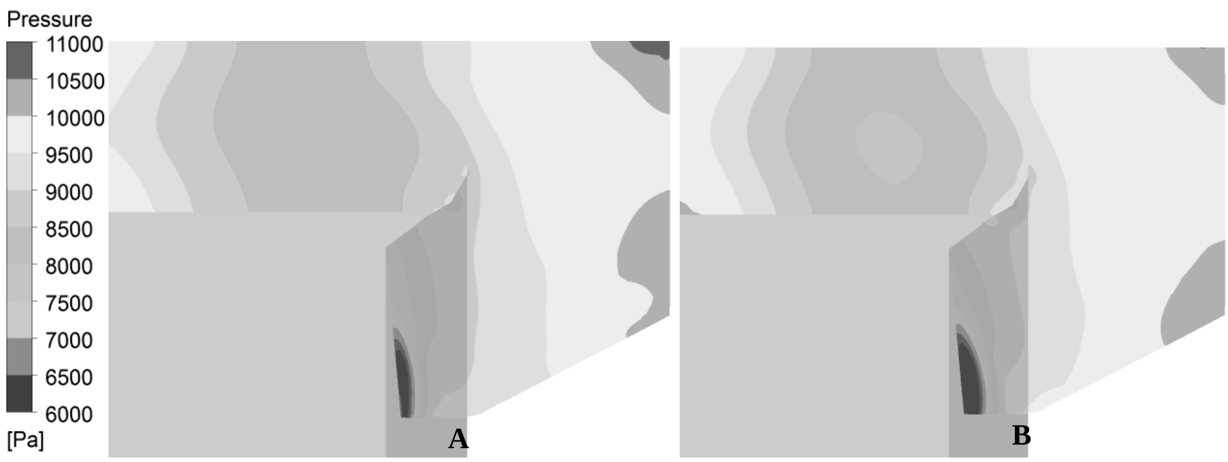

Figure 7: Pressure Contours for Meridional Plane for A) No Tip Leakage and B) Tip Leakage

Flow Asymmetry

The flow is significantly more asymmetric with the addition of tip leakage jet, as shown in Figure 8A and B. 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 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 9A and B.

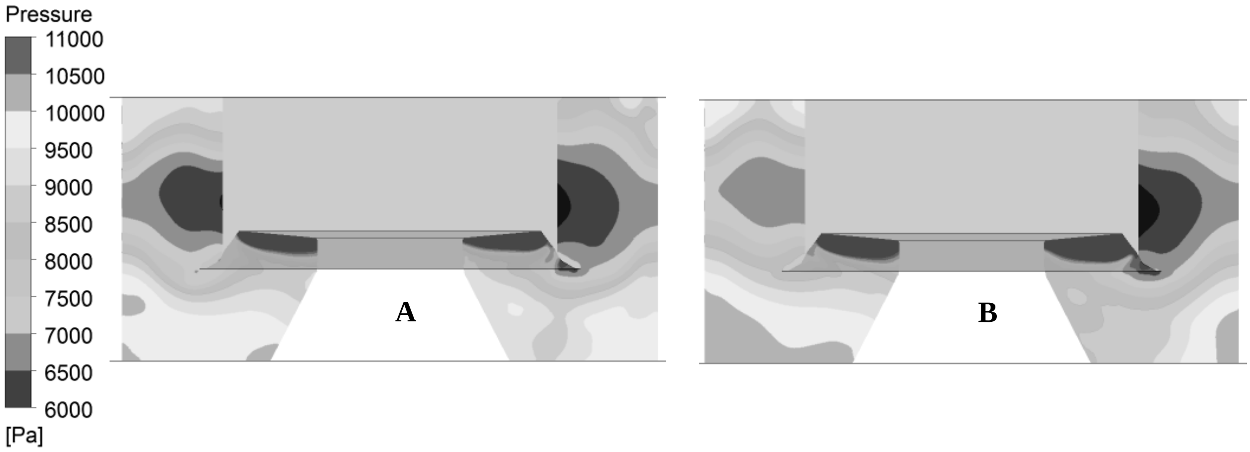

Figure 8: Pressure Contours at Half Joint Plane for A) No Tip Leakage B) Tip Leakage

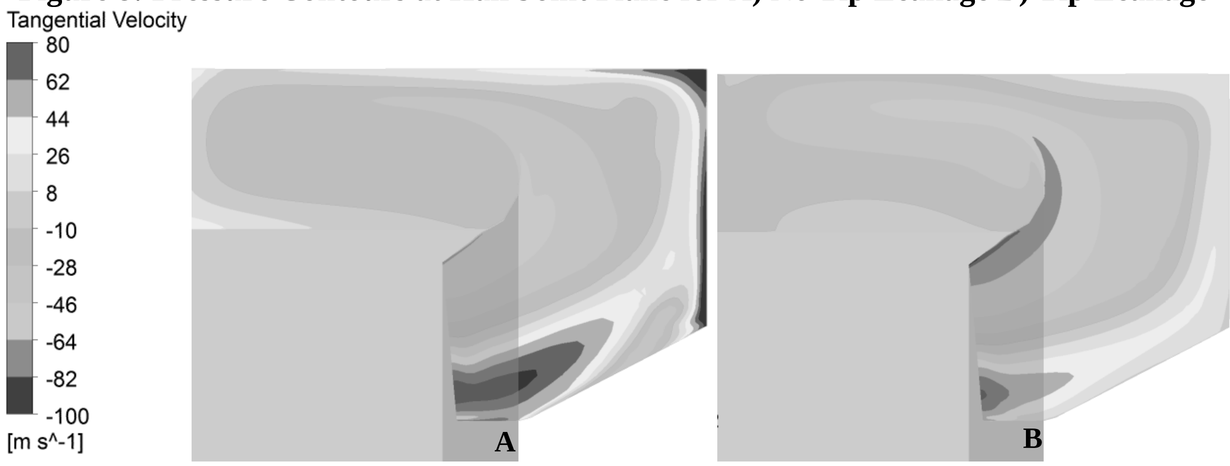

Figure 9: Contours of Absolute Tangential Velocity at Meridional Plane A) No Tip Leakage B) Tip Leakage

Non-Uniform Outlet

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.

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 10. 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 10.

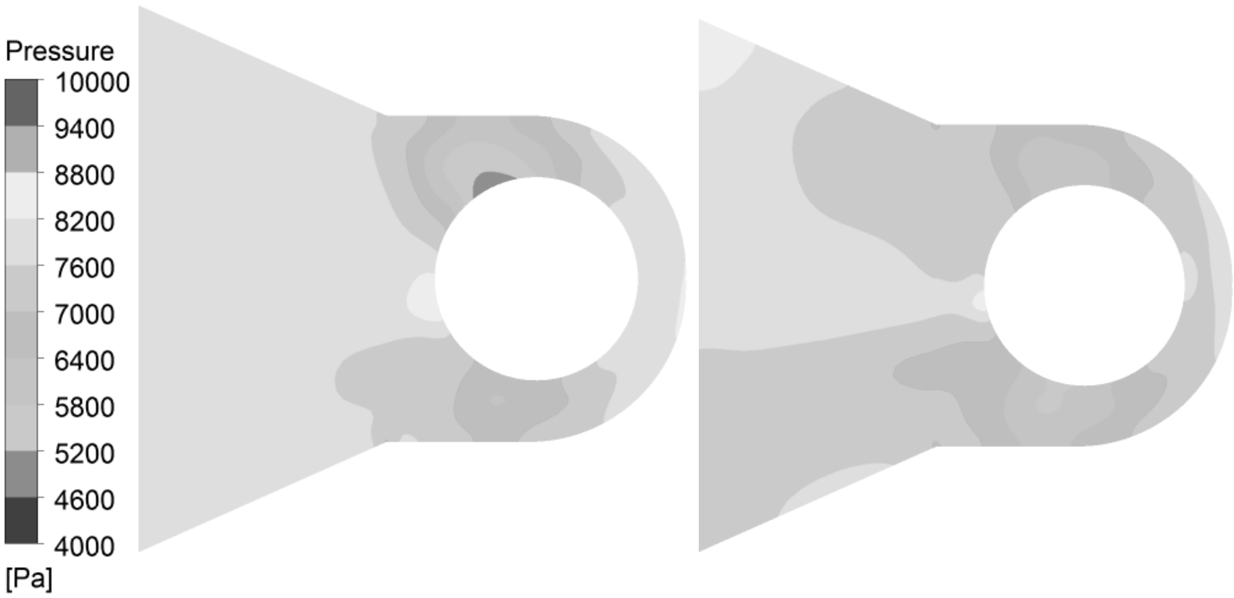

Figure 10: Pressure Contours on the Front Plane of A) Uniform Outlet B) Non-Uniform Outlet

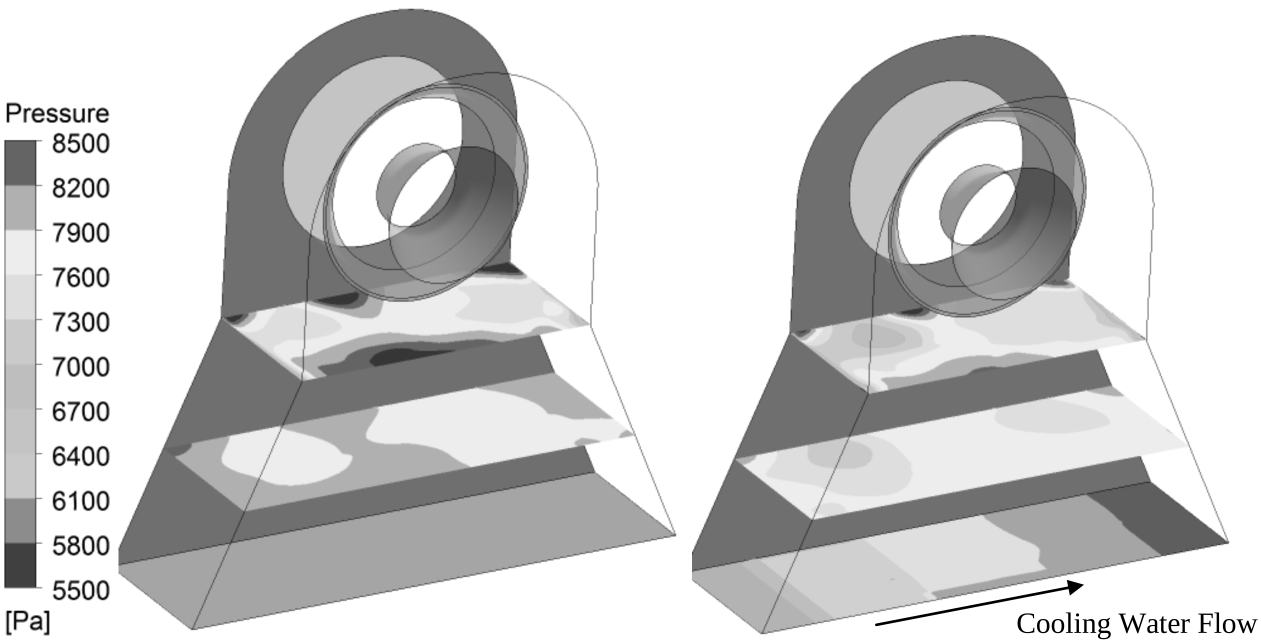

Figure 11: Pressure Contour Progression Through the Condenser Neck A) Uniform Outlet B) Non-Uniform Outlet

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.

CONCLUSIONS

This work highlights that the generic exhaust hood geometry, developed by Burton et al. (2012) can be successfully developed to include the important tip leakage jet. The work reported in this paper is consistent with other published work, highlighting strong coupling between the tip-leakage jet and the flow structure in the exhaust hood. The pressure recovery performance of the hood improves by around 0.25 due to the additional momentum of the jet reducing the low pressure region near the hood flow guide. The asymmetry of the flow increases due to the additional swirl in the tip leakage jet.

	C_p
No Tip Leakage	-0.035
Tip Leakage/Uniform Outlet	0.236
Tip Leakage/Non-Uniform Outlet	0.184

The calculations have also demonstrated the new findings of the importance of introducing representative outlet boundary conditions to CFD simulations. The asymmetry of the hood is decreased and improved due to the pressure gradient induced by the cooling water flow working against the asymmetry and reducing its magnitude. Although the pressure gradient is detrimental to performance due to the increased low pressure region in the condenser neck, the flow structure is markedly changed, even in the upper exhaust hood. This highlights the importance of establishing representative hood exit boundary conditions.

ACKNOWLEDGEMENTS

The authors like to extend my thanks to Peter Millington and Brian Haller from Alstom Power, Rugby, for their contribution to this project.

REFERENCES

[1] Beevers, A., Congiu, F., Pengue, F., and Mokulys, T., 2010, ``An Analysis of the Merits of CFD for the Performances Prediction of a Low Pressure Steam Turbine Radial Diffuser," Proc. ASME Turbo Expo, Glasgow, UK, GT2010-22107.

[2] 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."

[3] Burton, Z., Ingram, G. L., and Hogg, S., 2012, ``A Generic Low Pressure Exhaust Diffuser for Steam Turbine Research," Proc. ASME Turbo Expo, Copenhagen, Denmark, GT2012-68485.

[4] Fan, T., Xie, Y., Zhang, D., and Sun, B., 2007, ``A Combined Numerical Model and Optimization for Low Pressure Exhaust System in Steam Turbine," Proc. Proceedings for the ASME Power Conference, San Antonio, Texas, pp. 349-358, POWER2007-22147.

[5] 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," Proc. ASME Turbo Expo, Vancouver, Canada, GT2011-45302.

[6] Fu, J., and Liu, J., 2010, ``Investigation of Influential Factors on the Aerodynamic Performance of a Steam Turbine Exhaust System," Proc. ASME Turbo Expo, Glasgow, UK, GT2010-22316.

[7] Fu, J.-L., and Liu, J.-J., 2008, ``Influences of Inflow Condition on Non-Axisymmetric Flows in Turbine Exhaust Hoods," Journal of Thermal Science, 17(4), pp. 305-313.

[8] 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," Journal of Turbomachinery, 134(4).

[9] 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.

[10] Keller, H., 1986, ``Aerothermodynamics of Low Pressure Steam Turbines and Condensers," Hemisphere, pp. 119-216.

[11] 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.

[12] Li, Z., Li, J., Yan, X., Feng, Z., Ohyama, H., and Zhang, M., 2012, ``Investigations on the Flow Pattern and Aerodynamic Performance of Last Stage and Exhaust Hood for Large Power Steam Turbines," Proc. ASME Turbo Expo, Copenhagen, Denmark, GT2012-69291.

[13] Liu, J., and Hynes, T. P., 2002, ``The Investigation of Turbine and Exhaust Interactions in Asymmetric Flows: Part 1 - Blade Row Models Applied," Proc. ASME Turbo Expo, Amsterdam, Netherlands, GT-2002-30342.
[14] Liu, J., and Hynes, T. P., 2002, ``The Investigation of Turbine and Exhaust Interactions in Asymmetric Flows: Part 2 - Turbine-Diffuser-Collector Interactions," Proc. ASME Turbo Expo, Amsterdam, Netherlands, GT-2002-30343.
[15] 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.
[16] 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," Journal of Thermal Engineering, 56(4), pp. 277-283.
[17] Senoo, S., Takahashi, F., Shikano, Y., and Kimura, T., 2004, ``The Computational Technique for Compressible Fluid Based on Steam Properties and Performance Improvements on Steam Turbines," 14th International Conference on the Properties of Water and Steam, Kyoto, Japan.
[18] Stanciu, M., Fendler, Y., and Dorey, J.-M., 2011, ``Unsteady Stator-Rotor Interaction Coupled with Exhaust Hood Effect for Last Stage Steam Turbines," Proc. 9th European Turbomachinery Conference, Istanbul.
[19] Tajc, L., Bednar, L., Sikova, I., Feldberg, L. A., and Goudkov, E. I., 2006, ``The Experimental Investigation of the Influence of the Flow Swirl and Tip Clearance Jet on Aerodynamic Characteristics of Exhaust Hoods," SKODA Power, Svratka, Czech Republic.
[20] 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.
[21] Verstraete, T., Prinsier, J., Di Sante, A., Della Gatta, S., and Cosi, L., 2012, ``Design Optimization of a Low Pressure Steam Turbine Radial Diffuser Using an Evolutionary Algorithm and 3D CFD," Proc. ASME Turbo Expo, Copenhagen, Denmark, GT2012-69515.
[22] Yoon, S., Stanislaus, F. J., Mokulys, T., Singh, G., and M., C., 2011, ``A Three-Dimensional Diffuser Design for the Retrofit of a Low Pressure Turbine Using in-House Exhaust Design System," Proc. ASME Turbo Expo, Vancouver, Canada, GT2011-45366.
[23] 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.