19th Australasian Fluid Mechanics Conference
Melbourne, Australia
8-11 December 2014

The Use of Porosity to Simulate the Presence of Aerofoils on a Gas Turbine Shaft under Natural Cooling

E.O. Smith and A.J. Neely

School of Engineering and Information Technology
University of New South Wales, Canberra 2612, Australia

Abstract

After a gas turbine is shut down, internal buoyant convection during the cool down process can result in an asymmetric thermal condition on the compressor shaft. The resultant differential thermal expansion will cause the shaft to bow. In previous studies, the authors have used a combination of 3D conjugate heat transfer (CHT) computational fluid dynamics (CFD) and finite element analysis (FEA) to model the onset, duration, severity, and geometric distribution of the shaft thermal bow using simplified analogue models. This study investigates the use of porosity to simulate the presence of gas turbine compressor aerofoils for the purposes of modelling the onset of shaft thermal bow during the gas turbine static cooling process. The study then outlines a process for appropriate selection of the necessary parameters to describe a porous domain for use in a simplified gas turbine 3D CHT CFD model.

Introduction

During the cooldown process after a gas turbine is shut down, buoyant convection, both within the compressor shaft and the gas path, results in the development of an asymmetric thermal condition on the compressor shaft. The buoyant flow results in the upper parts of the shaft being hotter than the lower parts of the shaft at a given point in time. As the shaft cools and contracts, the upper parts of the shaft contract more slowly than the lower parts of the shaft, causing the shaft to bow. If the gas turbine is started while the shaft is in a sufficiently bowed state, rotor-to-stator contact can cause further isolated contact heating, leading to further bow, in a process known as the Newkirk Effect [2, 4-7].

In previous studies, the authors have used a combination of 3D conjugate heat transfer (CHT) computational fluid dynamics (CFD) and finite element analysis (FEA) to model the onset, duration, severity, and geometric distribution of the shaft thermal bow using simplified analogue models [6, 7]. Smith and Neely [6] established that the presence of compressor aerofoils does not play a significant role in the structural response of the shaft, but their role in the behaviour of the buoyant plume and the rate of heat transfer on the surface of the shaft is likely to be more important.

In a later study, Smith and Neely [7] established that while the inclusion of rotor and stator aerofoils would be too expensive computationally, the use of a porous domain in the CHT CFD simulation could account for the effect of the aerofoils on the behaviour of the buoyant plume and the transient thermal response of the shaft. The application of porous CFD domains to replace turbomachinery was explored by Ford, et al. [3], who used this approach to simulate the influence of injector swirl vanes, reporting a significant reduction in computation time, with only a slight reduction in the accuracy of the solution. The application of a porous domain in this way requires careful selection of parameters such as interfacial area density, heat transfer coefficient, as well as the actual value of porosity itself, as the result is highly sensitive to these inputs. Other methods and applications of porosity to conjugate heat transfer problems are discussed by Betchen, et al. [1].

Method

Previous studies by the authors have identified that the use of simplified models, such as the baseline model shown in figure 1, provides a useful starting point from which the relative contribution of various design elements of both the engine itself, and its installation to the airframe, can be assessed for their contribution to the shaft thermal bow phenomenon. One such design element, initially considered at Smith and Neely [7], is the presence of aerofoils in the gas path. Choosing to fully model the rotor and stator aerofoils would drastically increase the element cell count.

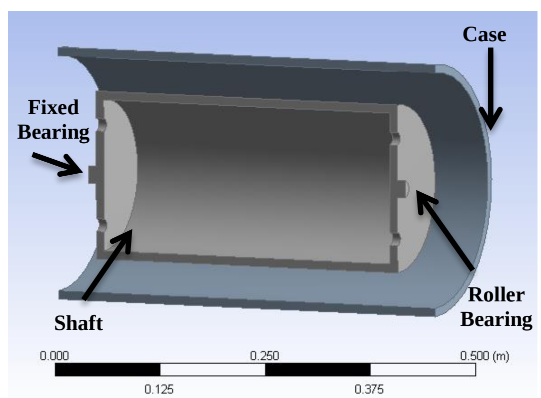

Figure 1. Baseline shaft and case model [7]

Figure 2 shows the baseline model with rotor and stator blades modelled in close to full detail. As the flow through the model is driven purely by buoyancy, the blades do not need to be aerodynamically accurate, and as such, are represented by flat plates. From a gas path blockage and heat transfer perspective, however, this is considered sufficiently realistic. For ongoing parametric studies, the computational cost of meshing and solving the blade flow paths is considered excessive. The use of a porous domain offers a low cost viable alternative.

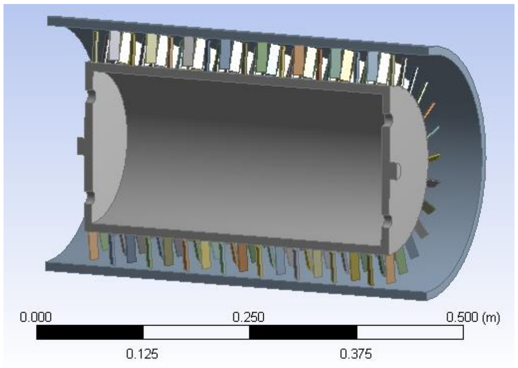

Figure 2. Fully bladed baseline model

Two key questions arise from this observation:

a) Do the blades contribute to the shaft thermal bow phenomenon sufficiently as to warrant their inclusion in the model?

b) Can the contribution of the blades be captured through a simplified porous domain?

The first question has been partly answered by Smith and Neely [6], who showed that the presence of blades on a shaft model under an unbalanced thermal load had a negligible effect on the structural response of the shaft. Considering the effect on the heat transfer and buoyant plume behaviour, Smith and Neely [7] performed a simple sensitivity analysis to show that the use of porosity to capture the effect of the blades could have a large effect on the overall shaft thermal bow time history, depending on how the porous domain was setup. This study aims to expand on that initial study, to better understand the application of porous domains to capture this aspect on the problem.

In order to measure the effect of aerofoils on the behaviour of a buoyant plume, a simple cascade calibration model was developed, shown in figure 3. This approach was chosen, rather than starting with the fully bladed baseline model shown in figure 2, primarily because of the difficulty in meshing the fully bladed baseline model, and because the computational cost of running that model would be high for the intended use in parametric studies. The simpler cascade calibration model was able to be run relatively quickly.

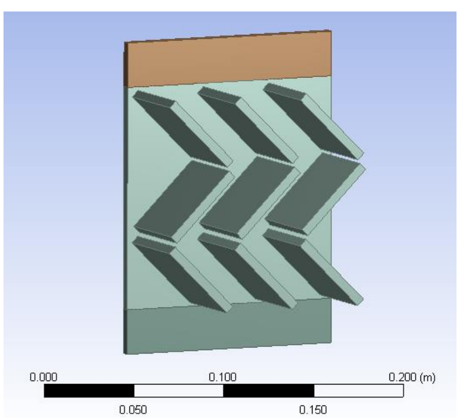

Figure 3. Cascade calibration model

Figure 3 shows the cascade calibration model, designed to match the basic porosity characteristics of the fully bladed baseline model, on a much smaller scale. The model is a crude representation of a series of rotor and stator blades in a compressor. For the purposes of calibrating the porous domain, and demonstrating that the application is viable, this model is considered adequate. More detailed aerofoils would give an answer that is closer to reality, but given the simplicity of the baseline shaft and case model, the level of fidelity in the cascade calibration model is considered sufficient. Porosity and interfacial area density, two of the key parameters necessary for the definition of a porous domain, were calculated as follows:

$$Porosity = 1 - \frac{Total\ Blade\ Volume}{Total\ Annulus\ Volume} \tag{1}$$

$$Interfacial\ Area\ Density = \frac{Total\ Blade\ Surface\ Area}{Total\ Annulus\ Volume} \tag{2}$$

Material	Ti-6Al-4V
Porosity	0.856
Interfacial Area Density	69.234 m^{-1}

Table 1. Cascade calibration model details

Table 1 shows the calculated values of interfacial area density and porosity, based on the cascade calibration model shown at figure 3. These values are considered broadly in line with the actual values found in parts of a gas turbine compressor. The degree of porosity matches estimates of the number of compressor blades compared to the volume in a compressor annulus, assuming the annulus is of constant area. Future studies will apply the technique demonstrated in this study to more closely match the values that would be expected in a real compressor.

It was hypothesised that the two primary contributions of the aerofoils in the gas path would be through restricting the flow of the buoyant plume, and changing the heat transfer rate, either through a change to the heat transfer coefficient itself, or simply through the increased solid surface area.

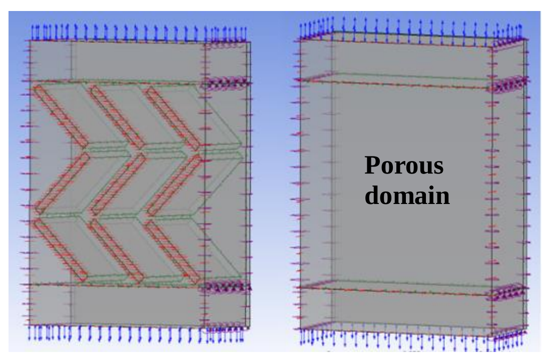

Figure 4. Cascade and porous model boundary conditions (boundary condition arrows: blue = opening, purple = periodicity, red = symmetry)

Figure 4 shows the boundary conditions for the cascade and porous calibration models. Both models use translational periodic interfaces on the left and right sides, openings at the back, top and bottom, and symmetry on the forward face of the model.

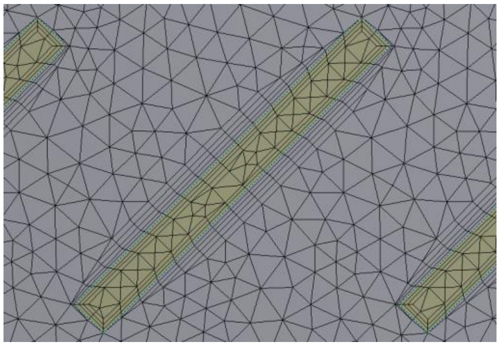

Figure 5. Cascade model mesh at fluid solid interface

Figure 5 shows the near wall mesh resolution for the cascade model. In order to accurately capture the heat transfer between the solid and fluid domains, the mesh was refined to reduce the y+ to <1, using the SST turbulence model. All solutions were also run with adaptive time stepping such that the RMS Courant number was 1.0, also in an effort to improve the accuracy of the heat transfer predictions.

Momentum Loss Model

For the presence of the aerofoils in the gas path to affect the behaviour of the buoyant plume, there must be a pressure drop as the flow passes through the blade rows. A pressure drop would indicate that the cascade is restricting the flow. To categorise the test cascade pressure loss, a cold steady state flow was established using a fixed input velocity. This method was chosen after it was found that the heat transfer driven buoyant flow resulted in little or no measureable pressure loss across the cascade.

Using a fixed input velocity of 1.0 $m.s^{-1}$ (approximately equal to the flow velocity due to natural convection), a pressure loss of 7.3 Pa was measured from the bottom to the top of the cascade. As the cascade is 0.125m tall, this represents a dP/dx of 63.12 $Pa.m^{-1}$. Figure 6 shows the total pressure and flow vectors through the cascade under a steady state input velocity of 1.0 $m.s^{-1}$.

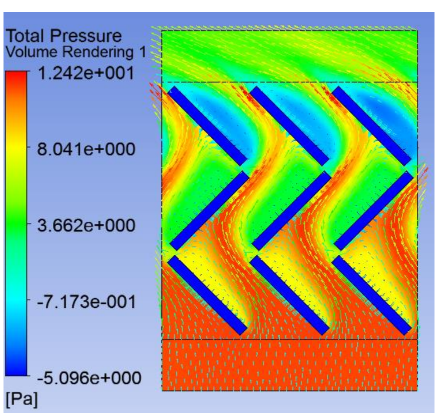

Figure 6. Total pressure and velocity vectors across test cascade at 1.0m/s y-axis input velocity

To replicate this pressure loss per unit length in the porous domain in CFX, the quadratic resistance coefficient is calculated as follows:

$$k_q = \frac{dP}{dx} \times \frac{1}{v^2} \quad (3)$$

where v is the flow input superficial velocity.

Using equation (3), k_q is calculated as 63.12 $kg.m^{-4}$. Repeating this test with an input velocity of 1.5 $m.s^{-1}$ and 0.5 $m.s^{-1}$ was found to yield a very similar value of k_q.

In a gas turbine annulus, as shown in figure 2, the buoyant plume will flow upwards along the y-axis until it reaches the compressor case, and then will flow axially along the x-axis. Accordingly, the quadratic resistance coefficient must be calculated in both the x- and y- directions. Following the same process as described earlier, $(k_q)_{x\text{-}axis}$ was calculated to be much higher, at 390 $kg.m^{-4}$. The crudeness of the calibration geometry is likely to have exaggerated the pressure loss in the x-axis due to the small, potentially unrepresentative rotor-to-stator gaps, as well as the use of a worst case blade alignment. Further studies using more realistic geometries and different clocking arrangements need to be performed in future work.

Heat Transfer Coefficient

To determine the heat transfer coefficient (HTC) required as an input to the porous model, a transient CFD model of the cascade test case was run for 5 minutes, and the area average HTC over the fluid-solid interface between the blades and fluid was taken. The model had initial temperatures of 600K for the solid domains, and 500K for the fluid (air) domains. All fluid domain boundaries were set to 300K. Using this method, the HTC was found to 4.36 $W.m^{-2}.K^{-1}$, and was confirmed to be stable over the 5 minute solution.

Validation of Porosity Model Settings using CFD

To validate that the porous domain settings had been accurately determined from the cascade test model, transient CHT CFD simulations were run with both the bladed cascade and porous models, to compare the results. Figure 7 shows the temperature time history for a point embedded in the wall of both the cascade and porous models. The blue and red lines represent the cascade and porous model results, and indicate that the heat transfer rate in the wall has been well matched between the porous and cascade models, using the approach defined above. For comparison, the green line shows the temperature time history of the same point with pure natural convection. This result indicates that the presence of blades in the flow significantly slows the heat transfer rate of the wall, and confirms that the result can be matched using the porous approach. While the cascade model has a greater surface area, the decrease in heat transfer rate is due to the reduction in flow velocity associated with the pressure loss, which is the dominant effect.

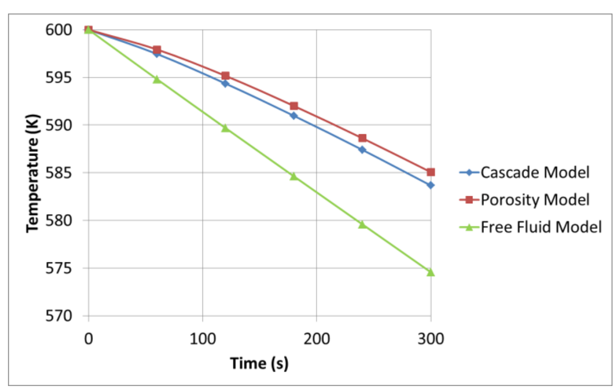

Figure 7. Temperature time history for cascade and porous models

The benefit of employing the porous domain in place of the full cascade model is that the porous solution was approximately 12% faster to solve using the same convergence criteria, and the same adaptive time-stepping criteria maintaining an RMS Courant number of 1.0. The difference in the two results was approximately 0.24%, which was likely due to error in solver when establishing the resistance coefficients, and then further error in the two separate 5 minute solutions. It is expected that the use of porosity would represent a greater benefit as the problem is scaled up, and the difference between running the fully bladed baseline model versus a porous baseline model is expected to be greater than the 12% demonstrated through comparison of the small calibration models.

Application to Shaft Bow Research

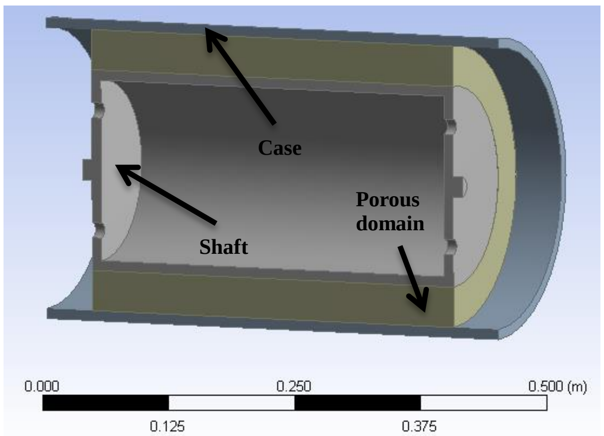

Figure 8. Porous domain applied to shaft and case model

Taking the axial and transverse porosity settings from the cascade test case and applying them to the baseline geometry as shown in figure 8, the effect of the rotor and stator blades on the shaft thermal bow can be explored. This study has not investigated the pressure loss in the radial direction as it is expected to be significantly less than the axial or circumferential pressure losses.

Figure 9 shows a short time history of the temperature difference between the top and bottom of the compressor shaft, with and without porosity, when allowed to cool naturally from 600K. This result highlights that the presence of the porous domain, representing the flow restriction and change in heat transfer rate of the shaft wall, has a significant contribution to the overall shaft thermal history result.

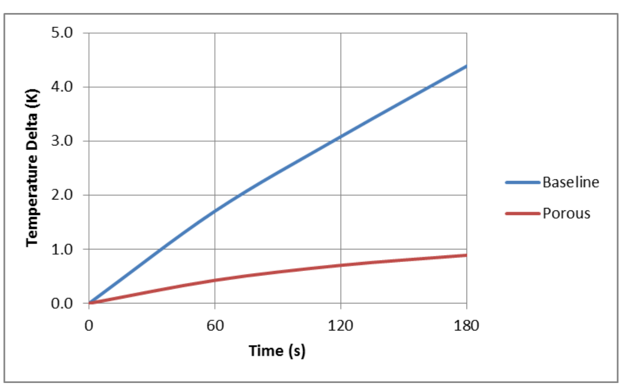

Figure 9. Shaft temperature delta for baseline and porous models

This significant change in the temperature time history of the compressor shaft indicates that the presence of the aerofoils, simulated by the porous domain, would have a significant effect on the thermal bow behaviour of the shaft, and therefore, should be included in future studies.

Conclusions

From this study it can be concluded that the contribution of compressor rotor and stator aerofoils to the natural cooling shaft thermal bow phenomenon is significant, and should be included in future modelling. Furthermore, this study has also shown that the contribution of the compressor aerofoils can be accurately represented using a porous domain, following the outlined method, thus saving time and computational resources.

This study has also found that strict convergence criteria must be adhered to when employing porous domains for conjugate heat transfer problems. In particular, a convergence criterion of at least 1×10^{-5} was required to avoid significant thermal domain imbalances.

Future Work

Future studies will improve the accuracy of axial and circumferential pressure loss predictions using more accurate geometries, and will also investigate axially variable porous domain parameters to more accurately represent actual gas turbine compressor design.

Acknowledgments

The authors would like to thank the Royal Australian Air Force for their continued support of this research. This research was undertaken with the assistance of resources provided at the NCI National Facility systems at the Australian National University through the National Computational Merit Allocation Scheme supported by the Australian Government.

References

[1] Betchen, L., A.G. Straatman, and B.E. Thompson, *A Nonequilibrium Finite-Volume Model for Conjugate Fluid/Porous/Solid Domains.* Numerical Heat Transfer, Part A: Applications, 2006. **49**(6): p. 543-565.

[2] Dimarogonas, A.D., *A study of the Newkirk effect in turbomachinery.* Wear, 1974. **28**(3): p. 369-382.

[3] Ford, C.L., J.F. Carrotte, and A.D. Walker, *The application of porous media to simulate the upstream effects of gas turbine injector swirl vanes.* Computers & Fluids, 2013. **77**(0): p. 143-151.

[4] Marinescu, G. and A. Ehrsam. *Experimental Investigation Into Thermal Behavior of Steam Turbine Components. Part 2 – Natural Cooling of Steam Turbines and the Impact On LCF Life*. in *ASME Turbo Expo 2012*. 2012. Copenhagen, Denmark.

[5] Newkirk, B.L., *Shaft Rubbing.* Mechanical Engineering, 1926. **48**(8): p. 830-832.

[6] Smith, E.O. and A.J. Neely. *Shaft Thermal Bow Modelling in Gas Turbines - An Initial Study*. in *21st International Symposium on Air Breathing Engines*. 2013. Busan, South Korea.

[7] Smith, E.O. and A.J. Neely. *The Effect Of Aircraft Integration Design On Gas Turbine Shaft Thermal Bow And The Newkirk Effect*. in *ASME Turbo Expo 2014: Turbine Technical Conference and Exposition*. 2014. Düsseldorf, Germany.