MODELLING HEAT TRANSFER IN ROTATING DISC CAVITIES AND APPLICATION TO THE T56 ENGINE

John Faragher and Andrew Ooi.
Airframes and Engines Division, Defence Science and Technology Organisation,
506 Lorimer Street, Fishermans Bend, Victoria, 3207, Australia.

The Defence Science and Technology Organisation provides support to the Royal Australian Air Force in the area of aircraft engine life management. The low cycle fatigue life and creep life of turbine and compressor discs depends on the mechanical and thermal stresses they experience and on the disc material properties, which vary with temperature. The mechanical stresses depend on engine speed and in general they are larger than the thermal stresses, which depend on the temperature gradients. Even though the thermal stresses are smaller than the mechanical stresses they do significantly affect the life of the discs. The ability to predict the temperature gradients requires accurate predictions of both the flow in the cavities between the discs and the heat transfer in the complex boundary layers that are formed. A numerical method has been validated[1] with available experimental data for the flow and heat transfer in a generic cavity between two co-rotating discs with axial inflow and radial outflow of fluid, a configuration common in gas turbine engines. Predictions are currently being obtained from a numerical model of a turbine assembly in an actual engine.

The first validation case was simulated with a two-dimensional axisymmetric model of laminar isothermal flow. The configuration shown in Figure 1(a) was used by Owen and Pincombe[2] to determine the flow structure using flow visualisation and to measure the velocity distribution using laser-Doppler anemometry. Figure 2(a) shows a comparison between laser-Doppler anemometry measurements performed by Owen and Pincombe[1] and computed radial velocity, for $Re_\phi = 25{,}000$, along a line across the cavity at $r = 6a$, as shown in Figure 2(b). The streamlines and velocities calculated using the numerical CFD code closely matched the experimental data.

In the second case, numerical predictions using a two-dimensional axisymmetric model of turbulent flow with heat transfer in a cavity with the same configuration shown in Figure 1 were compared with experimental data of Northrop and Owen[3]. The standard k - ε turbulence model with the two-layer zonal model for near wall treatment[4] was used. The rotational speed of the discs was 255.7 rad/s (2442 rpm), and $Re_\phi = 3.2 \times 10^6$. Figure 3 shows a plot of Nusselt number vs. radial coordinate for the upstream disc. The results from the two-dimensional axisymmetric model are within 20 percent of the experimental data. The shape of the curve is correct but the peak is occurring a little too close to the bore of the disc.

Three-dimensional models were then used to determine whether modelling discrete holes at the outlet, as opposed to a continuous slot, which is the approximation that is inherent in the two-dimensional axisymmetric model, would improve the agreement between the computed and experimental results. It is sufficient to model an 11.25° sector of the disc assembly, as shown in Figure 1(c), because the perforated shroud used by Northrop and Owen[3], which joined the two discs at their rims, had 32 holes equally spaced around the circumference. The results in Figure 3 show that the two different three-dimensional models produced almost identical Nusselt numbers across the entire radius of the disc. These results show that the approximation of the holes by a slot makes little difference to the predicted results and hence the simpler two-dimensional approach is being used for the T56 engine predictions. Further work is being performed to determine the reason for the difference between the computed and experimental Nusselt numbers.

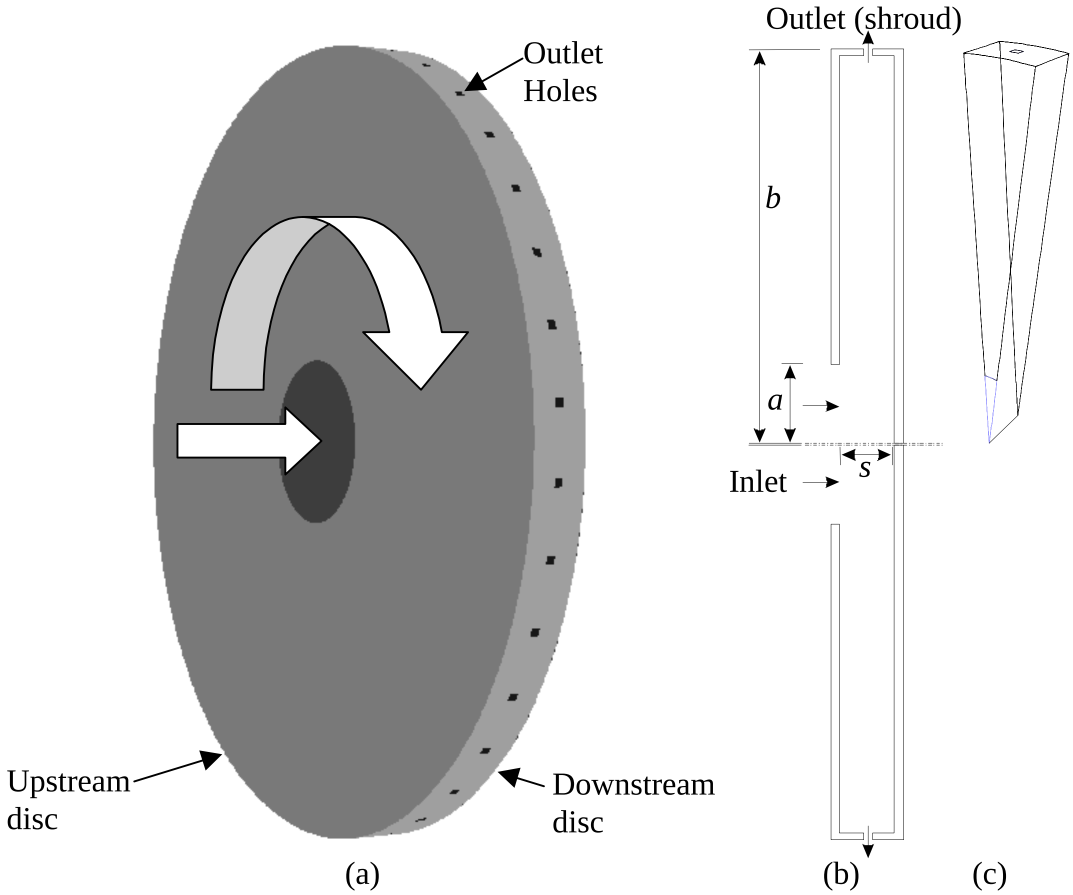

Figure 1. (a) The configuration of the system, (b) cross-section; (c) 3D model.

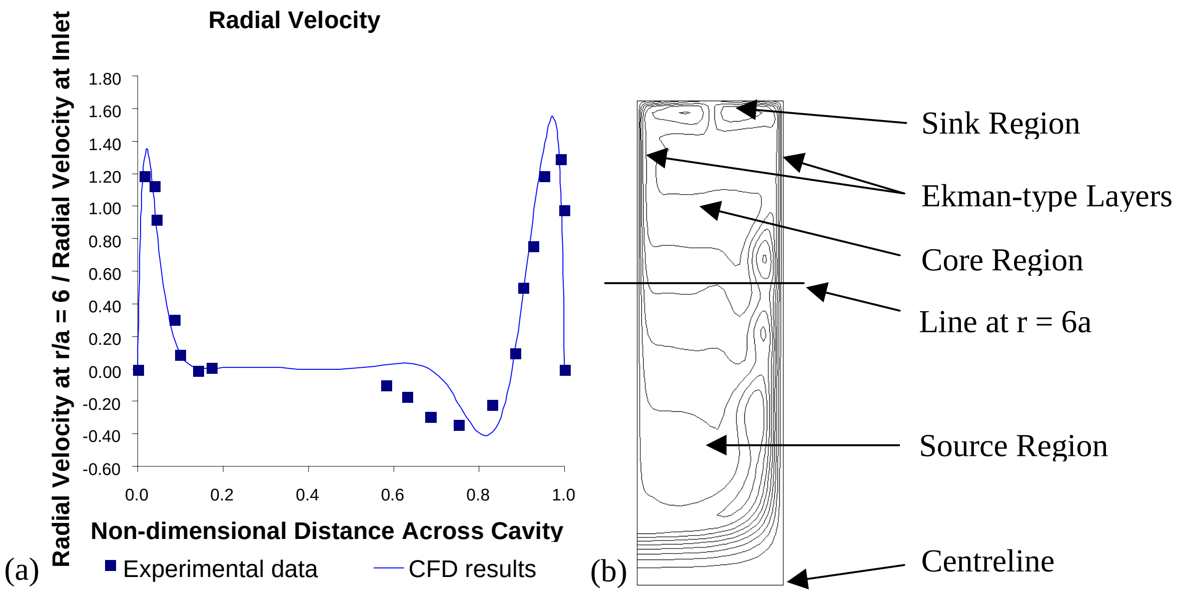

Figure 2.(a) Radial velocity for laminar isothermal flow case at r = 6a. (b) Streamlines predicted by the axisymmetric CFD analysis: $Re_\phi = 25000$.

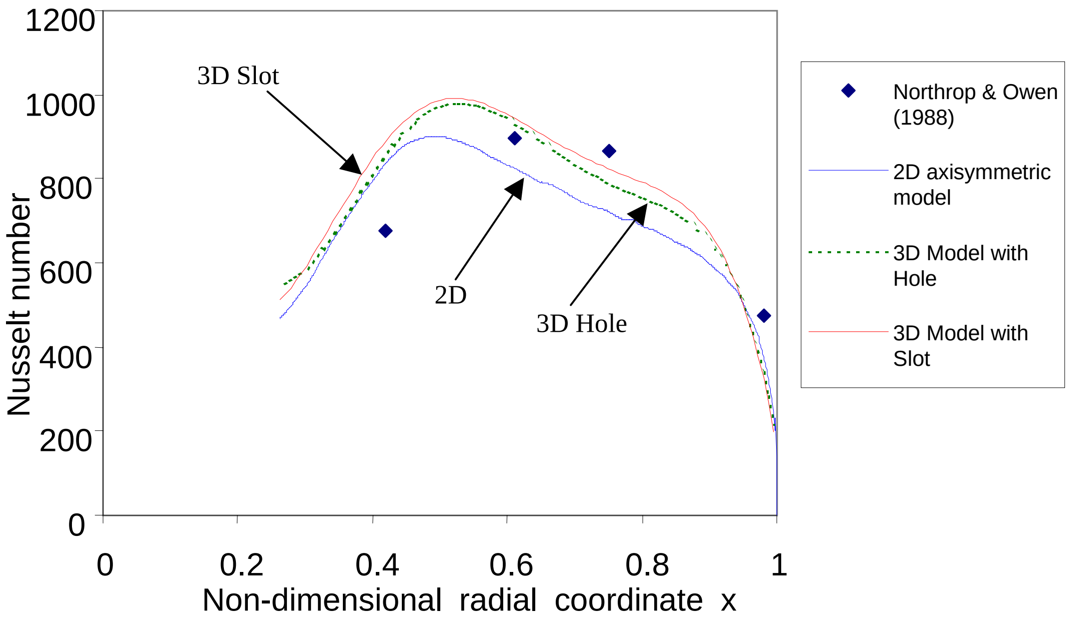

Figure 3. Nusselt numbers for the rotating disc cavity for turbulent flow and heat transfer with Re = 3.2e6.

A numerical model is currently being applied to simulate the flow and heat transfer inside the turbine rotor assembly of the Allison T56 engine shown in Figure 4. The results of this analysis will be presented.

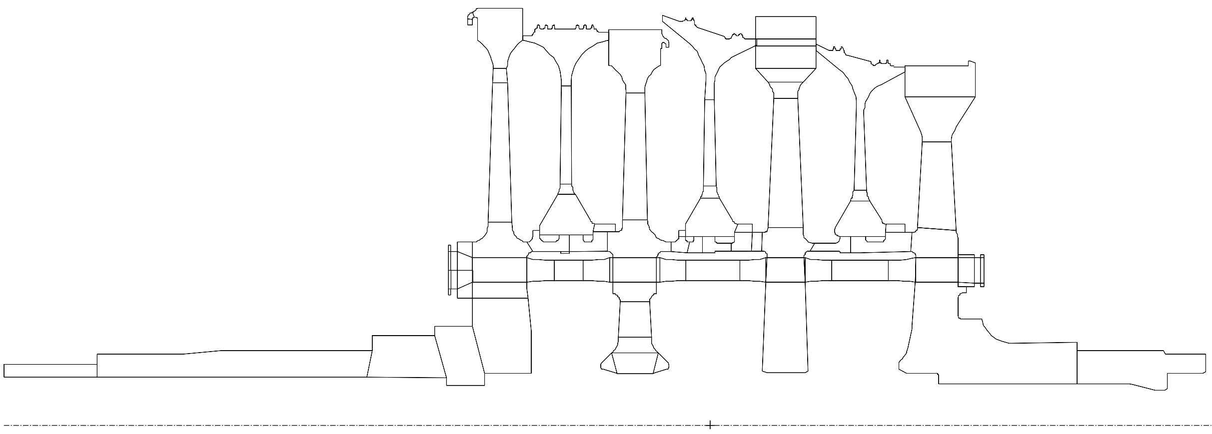

Figure 4. The turbine rotor assembly of the Allison T56 engine.

References

1. Faragher, J.S. & Ooi, A., 2000, Proc. 7th Australasian Heat & Mass Transfer Conf., Townsville, Australia.
2. Owen, J.M. & Pincombe, J.R., 1980, J. Fluid Mech., 99, pp.111-127.
3. Northrop, A. & Owen, J., 1988, Int. J. Heat & Fluid Flow, 9(1), pp27-36.
4. Chen, H.C. and Patel, V.C., 1988, AIAA Journal, 26(6), pp641-648.