Numerical Analysis of Combustor Burn-Through Representative Jet

Z. Hussain*[1], D. Bacci[2] and L. Pinelli[3]

1: Rolls-Royce plc, Thermofluids Dept, Moor Lane, PO Box 31, Derby UK, DE24 8BJ
2: Oxford Thermofluids Institute, University of Oxford, Oxford, United Kingdom, OX2 0ES
3. Department of Industrial Engineering, University of Florence, Via S. Marta 3, 50139, Florence, Italy

* Correspondent author: zahid.hussain@rolls-royce.com

Abstract

Aircraft safety has always been of paramount importance to the civil aviation industry. The continuous increase in the number of flying hours has caused the necessity to reassess those safety issues that have historically been rare, as such, though an event may be statically improbable, the total number of occurrences can increase with an increase in the total number of operational flying hours. One such aircraft safety issue is combustor burn-through, which is a recognised cause of fire originating from within the aircraft engine casing. The nature of the jet associated with combustor burn-through is that of a highly under-expanded jet, due to the internal combustor-to-ambient pressure ratio. Nevertheless, the nature of the flow can be considerably changed in the presence of a solid boundary placed downstream of the nozzle. In such conditions, a strong shock is formed, and the nature of the jet is varied. Outside of the core flow, recompression to the ambient pressure is achieved through compression waves that coalesce to form a shock structure known as a barrel shock. The objective of this study was to gain further insight into the characteristics of such flows, using a numerical approach. A geometry representing an under-expanded free jet impinging on a perpendicular planar surface was modelled using a density-based formulation of OpenFOAM CFD solver, adopting the improved delayed detached-eddy simulation (IDDES) turbulence model. Assuming the flow starting from a reservoir with a pressure ratio of 40, two geometries were simulated, having a nozzle-to-plate spacing of 3 and 5 nozzle diameters respectively. According to the distance of the nozzle to the plate, a different flow pattern was developed. Moreover, as already evidenced in previous experiments, it was confirmed that the highest heat transfer on the impingement plate is largely determined by vortex flow and possible total temperature separation resulting from shearing flows. Finally, the result analysis indicated that the shock system was not stable but experienced an oscillatory behavior locked at exact frequencies, which varied with plate distance.

Keyword: *Underexpanded jet, impingement, OpenFOAM, CFD, burn-through*

1. Introduction

Aircraft safety is of paramount importance to the civil aviation industry. An increase in the number of flying hours seen in the international civil aviation fleet over the last decades has caused the necessity to reassess those safety issues that have historically been statically improbable.

One such aircraft safety issue is combustor burn-through, which is a recognized cause of fire originating from within the *aircraft engine casing*. Regulation of this safety event is defined within both the EASA and the FAA as general management of critical flight safety issues. The means compliance with this regulation is advisory in nature [1]. This recommends a *means of compliance* of conditions of 3000-3500°F (1922-2200K) and 350-550psi (24.1-37.9bar) using a circular nozzle or aperture of 1 inch (25.4mm) diameter.

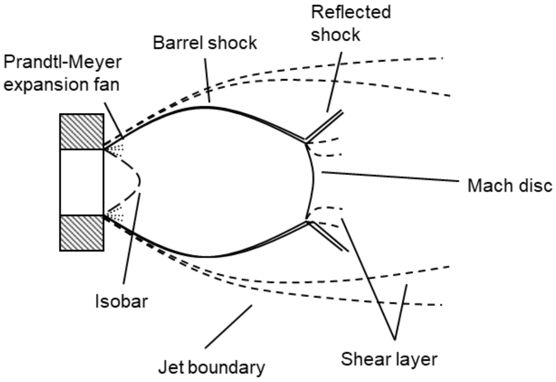

Figure 1. Configuration of a highly under-expanded free jet with a viscous flow structure.

The critical condition for such a failure and in line with the regulatory material, is aircraft take-off. This results in the creation of a highly under-expanded jet outside the engine casing. A common flow distribution is shown in Figure 1.

For under-expanded flow, the pressure gradient at the nozzle exit is great enough to accelerate the boundary layer flow to a Mach number of unity. The sonic line at the nozzle (isobar) intersecting the lip isolates the flow from any external influences. In order for the gas to expand to the ambient pressure, the flow has to be deflected through a sufficient Prandtl-Meyer fan angle which is only possible close to the nozzle wall. For other streamlines the fan overexpands the fluid and the pressure falls below the ambient condition. The maximum overexpansion acts along the center-line with recompression of the core flow occurring across a normal shock, known as the Mach disc. A barrel shock is created from the nozzle lip, this shock structure being of an oblique shock nature and is initially swept away from the nozzle centre-line axis by the radial component of velocity from the jet expansion. However, further downstream the ambient pressure is sufficiently above the flow in the region of this shock to push the barrel shock back towards the nozzle axis. There is sufficient pressure gradient across the shock, caused by the fluid’s acceleration, to force it to follow the barrel curvature. It has been noted that a repeat of the above-described shock cell structure can occur up to pressure ratios of 7. Shear instabilities create mixing layers around the jet and with pressure loss occurring through the shock structures, cause a damping effect on the pressure variation along the jet centerline. This eventually affects the dissipation of the potential core and pressure in the jet to reach equilibrium with the ambient condition.

For certain conditions, the free jet can produce crossing oblique shocks formed in a classical shock diamond structure downstream of the shock cell/s. Within the jet boundary, it has been concluded that there exists the presence of streamwise flow features of a vortical nature. These features exist in the supersonic region of the jet boundary and are observed to be subject to a process of vortex merging. The presence of these vortices originates from the concave curvature of streamlines at the jet boundary due to the jet core’s pressure variation. This results in the well-documented creation of Taylor-Görtler vortices. The assumption is often made that the jet impingement is axisymmetric. However repeatable evidence has been obtained from impingement research on engine operating flow.

Previous experimental research conducted by the author showed that the central area of the impingement region on the plate was at a lower temperature than the immediate surrounding region. The increase in temperature immediately adjacent to this region was associated with the formation of Taylor-Görtler vortices which produced high heat transfer coefficients. These results confirmed that the highest heat flux was not located in the central impingement region, but in an outside ring whose position was determined by the formation of the above-mentioned vortices. In impingement cases the shock structure, local to the nozzle exit, is being influenced by the body force of the impingent surface.

At pressure ratios (N) higher than 11, it has been noted that for the free-jet case, sufficient pressure is lost through the first shock cell and the mixing layer losses are such, that no repeating shock cell structure is observed as would be the case at lower pressure ratios [2].

Within the jet boundary it has been concluded that there exists the presence of streamwise flow features of a vortical nature [3]. These features exist in the supersonic region of the jet boundary and are observed to be subject to a process of vortex merging. The presence of these vortices originates from the concave curvature of streamlines at the jet boundary due to the jet core’s pressure variation. This results in the well documented creation of Taylor-Görtler vortices [4].

The assumption is often made that the jet impingement is axisymmetric in nature. However repeatable evidence has been obtained from impingement research at engine operating flow conditions that show the flow to be non-axisymmetric producing component failure remote from the conventional stagnation region. An example of this phenomenon [5] was the cause for the present research; experimental assessment was made and the body that is the focus of the paper is the ability of Computational Fluid Dynamics (CFD) to replicate the flow field and impingement phenomena.

2. Computational Set-up

2.1 Simulation Domain

A simplified geometry of a convergent nozzle fed from an upstream plenum chamber was generated from a 2D axisymmetric profile, Figure 2a. The radial extent of the volume was chosen to replicate the work of Li et al [6]. The geometry was configured to simulate two nozzle-to-wall dimensions (L/D) of either 3 or 5.

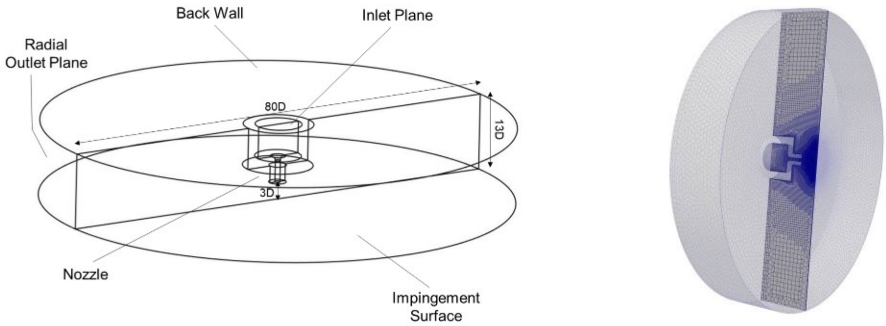

Figure 2a.(left) Configuration of a highly under-expanded free jet with a viscous flow structure for the L/D=3 case.
Figure 2b.(right) The initial computational mesh generated via the ICEM.

For this study numerical simulations were conducted using OpenFOAM v2306, using the density-based "rhoCentralFoam" routine coupled with IDDES turbulence model, already validated for nozzle flows [7]. The mesh was of poly-hexa typology with 44.6e+6 elements for the L/D=3 case (Figure 2b) and 56.9e+6 elements for the L/D=5 case.

2.2 Boundary Conditions

The inlet boundary was set to a pressure of 20bar and an outlet boundary condition of 0.5bar. The temperature of the inlet flow was set to 600K and the outlet was set to 288K as an ambient temperature condition. The impingement surface was set at a constant temperature of 288K. These boundary conditions replicated experimentation undertaken in the Oxford University Gun Tunnel (OUGT) which is a short duration facility with a run time >100ms.

2.3 OpenFOAM Configuration

The turbulence model used was the IDDES k-ω based formulation with the working fluid treated as a thermally perfect gas. Gradients were computed using a cell-limited, least square formulations. Fluxes, alternatively, were discretized using a 2nd order vanLeer scheme, whilst unsteady terms adopted an implicit second order scheme. The time-advancement of the solution was constrained to a max. CFL number of 0.2.

3. Results and Discussion

The computation was run for several months to generate a solution for each one of the cases, L/D=3 & L/D=5, with the convergence criteria 1e-8 for residuals of key flow parameters. Values for y+ on the impingement surface for the solutions peaked at 5.6 in small, localized regions with the majority <1, allowing the boundary layer to be resolved. The main objectives were the recreation and capture the general aerodynamic structure of the jet and heat transfer coefficient based on adiabatic wall temperature (the jet impingement surface being defined as an adiabatic wall). A total run time for this analysis for both cases was ~ 2ms.

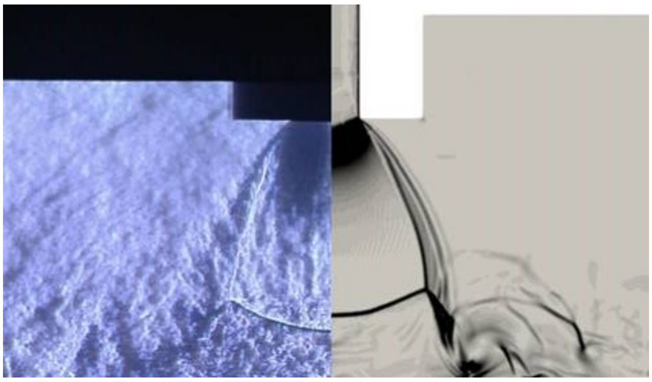

Figure 3. Schlieren comparison between the OUGT and OpenFOAM result shown left and right respectively for L/D=3.

An assessment of Schlieren comparison between experimentation undertaken at the Oxford University Gun Tunnel (OUGT) and the CFD, as a 2D slice through the jet, indicated a good general definition of the shock structure. The Mach disk shows some deviation from the schlieren, but time dependent capture of this phenomena shows that the Mach disc and the barrel shock does move around. The image presented in Figure 3 shows the average location of the shocks in the jet to allow for comparison.

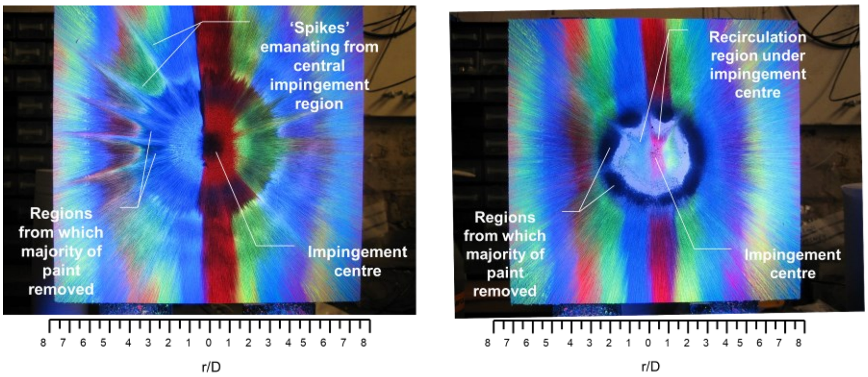

Figure 4a. (left) The OUGT flow visualization for L/D=3 clearly indicating a mechanism of pressure relief from the center of the jet impingement, through the region where the jet boundary impinges on the plate. 4b. (right) Flow visualization or L/D=5 showing clear indication of the presence of a recirculation bubble in the center of the impingement; also of note is the elevated temperature (bleached paint) in a region inside of the jet boundary shear layer.

Observation of the profiles of the jet flow onto the impingement surface at the OUGT, key understanding of the impingement surface flow, and consequently the jet flow above this, can be derived. Clear indication of the difference in flow from the two nozzle-to-plate distances can be observed in Figure 4a & 4b.

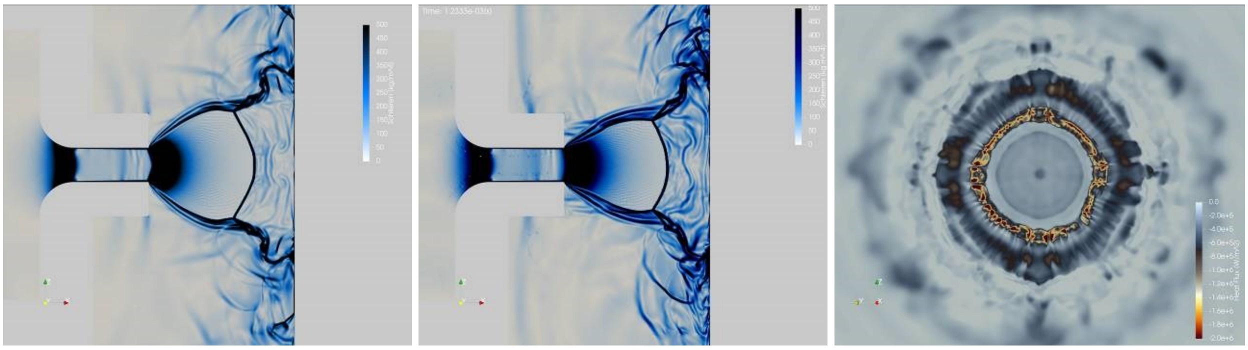

Figure 5a. (left) & Figure 5b. (middle) CFD schlieren image of the L/D=3 jet at the jet's maximum width for the time dependent solution (a) and minimum width (b) as the jet boundary 'flaps' to release pressure from the fluid downstream of the Mach disc; Figure 5c. (right) The impingement heat flux profile showing the peak values in a coherent ring of points for L/D=3.

While the general jet shock structure up to the Mach disc appears to be correctly generated, the streamwise vortical structures in the jet boundary, widely observed for free-jet cases [8,9], are not generated and consequently have a significant effect on the nature of the jet impingement. In the case of the experimental observations, the structures in the jet boundary in the vase of L/D=3 do not form a coherent 'curtain' isolating the flow under the Mach disc as in the case of the CFD solution, but rather permitting pressure and mass built up in the impingement center to be relieved between the vortices. As the CFD has not been able to capture this type of flow structure, the 'curtain' generated by the jet boundary must be periodically lifted to allow such pressure relief resulting in 'flapping' of this region of the flow. This can be seen as generating a movement in the overall jet structure from Figure 5a to 5b.

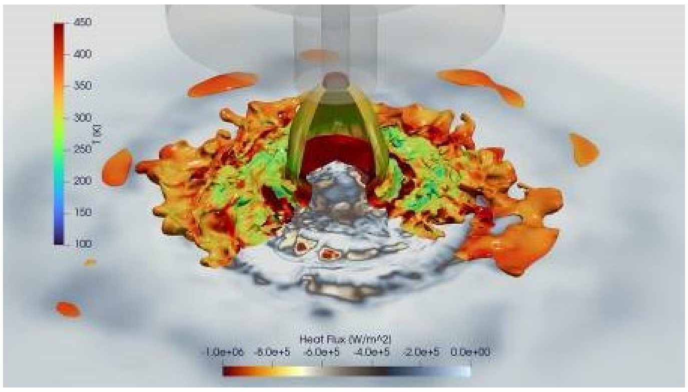

Figure 6. L/D=5 CFD solution showing peaks of heatflux in multiple coherent rings of localized points, but the solution has no indication of a recirculation bubble or the experimentally observed flow configuration. The iso-surface for the jet is for Mach=1and colored with total temperature contours.

In the case of L/D=5, the CFD provides no indication of a recirculation bubble on the impingement surface as has been observed with the OUGT experimentation. Once again, the CFD generates a ‘flapping’ flow structure similar to that of L/D=3. Experimentation at both nozzle-to-plate spaces in both cases are indicative of a stable impingement structure, rather than one ‘flapping’ in nature, especially in the case of L/D=5 where the paint has been bleached due to short duration focused local heating. In the case of the CFD solution at L/D=3, Figure 5c does show localized point of high heatflux, but these are temporal in nature due to the flapping of the jet. Consequently, the peak heatflux measurements made within the OUGT are an order of magnitude lower that generated via the CFD.

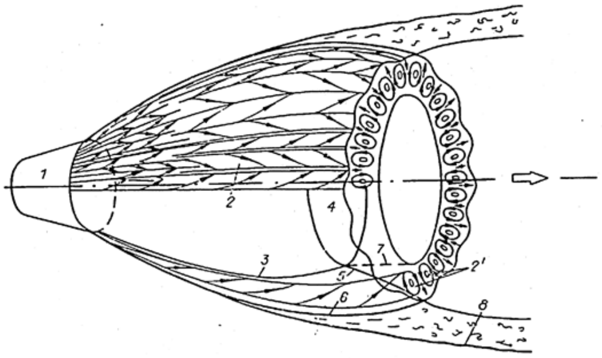

Figure 7. Streamwise vortical structures observed by Zapryagaev et al, in the jet boundary that are likely make their way to the impingement surface [8]. (1) Nozzle barrel; (2) geared vortex pairs, (3) barrel shock, (4) Mach disc, (5) oblique shock, (6) jet boundary, (7) shear plane, (8) mixing region.

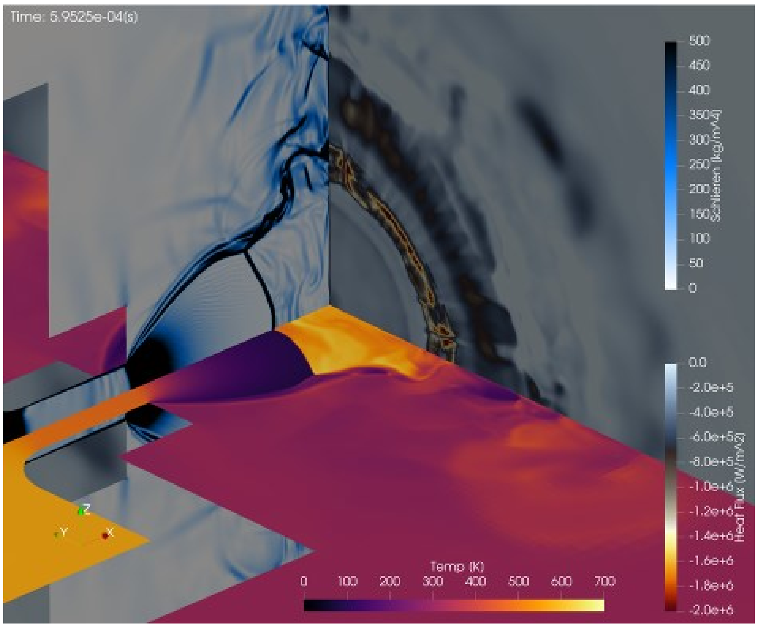

Fig. 8. Instantaneous visualization of flow characteristics for L/D=3 nozzle case.

Also, of note when observing Figure 5c in comparison to Figure 4a is that the experimental flow visualization, where paint on the surface has been removed, aligns with the anticipated number of streamwise vortices observed for free jets [8] being between 12-16, while the localized points in the CFD far exceed this number. Figure 8 shows the combined schlieren, heatflux and temperature variation of the jet, but the structure is best appreciated as an animation to understand the locations of the heatflux peak and its migration radially outward with the jet boundary as this region of the jet ‘flaps’. Consequently, the CFD does not generate any part of the jet structure that is stable, subjecting it to significant variation.

4 Conclusion

What can be observed from this analysis is that the basic premise that the peak of heat transfer for an underexpanded jet impingement existing in the center of the impingement, does not hold true. It is of interest that this analysis method has simulated the flow structure theories by Goldsmith et al [10] which promoted the creation of rolling, toroidal flow structures over the impingement plate (Figure 7) rather than those postulated by Yokobori et al [11] as streamwise vortical structures (Taylor-Görtler vortices) and observed from the flow visualization experimentation undertaken in the Oxford University Gun Tunnel.

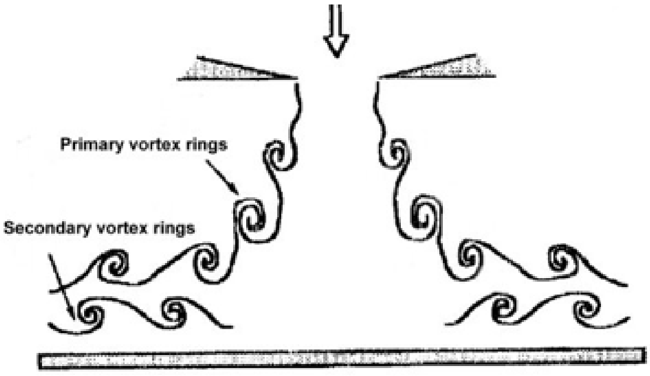

Figure 7. Toroidal structure in a sub-sonic jet boundary postulated by Goldstein et al [10] that shows similar structure to that seen in the CFD undertaken across the impingement surface.

This poses an interesting dilemma as to whether the flow field being replicated is able to switch between the two theorized formats or the CFD is unable to capture the flow physics correctly. It could be that there are critical parameters that need to be over emphasized (e.g., viscosity) within the computational flowfield. The computational complexity and the time taken to generate a solution precluded the ability to run sensitivity assessments to determine which parameters could cause the solution to better replicate the experimental conclusions. As such if future work is undertaken in this area, consideration needs to be applied to the approach possibly including alternative modelling techniques, e.g., alternative turbulence models or significantly higher mesh definition. For the case of freejets, success had been achieved with the use of Monte-Carlo type solvers [9] that could also be trialed for this jet configuration.

5. References

[1] European Aviation Safety Agency AMC 25.903 (d)(1), and Federal Aviation Administration AC 20-135, Section 8, Part C, both current.

[2] Cain, T. M., Morris, N. A., Roberts, T. P., and Jones, T. V., "A Comparison of Numerical Predictions and Experiments on Under-Expanded Jets," 1992. https://doi.org/10.2514/6.1992-4027

[3] Krothapalli, A., Buzyna, G., and Lourenco, L., "Streamwise Vortices in an Underexpanded Axisymmetric Jet," *Physics of Fluids A*, Vol. 3, No. 8, 1991. https://doi.org/10.1063/1.857914

[4] Schlichting, H., and Gersten, K., "Boundary-Layer Theory," 2016. https://doi.org/10.1007/978-3-662-52919-5

[5] Hussain, Z. M., Coney, M. H., Moss, J. B., Ireland, P. T., and Jagger, S., "Characteristics of an Underexpanded Jet and Its Surface Impingement for Combustion Burnthrough," Vol. 3, 2009.

[6] Li, M., Karami, S., Sandberg, R., Soria, J., Ooi, A., Heat Transfer Investigation On Under-expanded Supersonic Impinging Jets, International Journal of Heat and Fluid Flow, Volume 101, 2023.

[7] Giaccherini, S., Pinelli, L., Marconcini, M., Pacciani, R., Arnone, A., Validation of an Analytical Model for the Acoustic Impedance Education of Multi-Cavity Resonance Liners by a High Fidelity LES Approach. ASME Journal of Turbomachinary 145(8):081002,2023

[8] Zapryagaev, V.I. and Solotchin, A.V., "Three-dimensional Structure of Flow in a Supersonic Underexpanded Jet", Journal of Applied Mechanics and Technical Physics, Vol.32, Pt. 4, pp. 503-507, 1991.

[9] Usami, M. and Teshima, K., "Numerical Simulation of Instability in Supersonic Jet Boundary", Computational Fluid Dynamics Journal, 13(2):33, pp. 241-246, 2004.

[10] Goldstein, R.J., Behbahani, A.I., Kieger Heppelmann, K., Streamwise distribution of the recovery factor and the local heat transfer coefficient to an impinging circular jet, International Journal of Heat and Mass Transfer, v29, n8, pp1227-1235, 1986

[11] Yokobori, S., Kasagi, N., Hirata, M., Nakamaru, M., Haramura, Y., Characteristic behaviour of turbulence and transport phenomena at the stagnation region of an axi-symmetrical impinging jet; Symposium on Turbulent Shear Flows, 2nd, London, England, July 2-4, 1979, Proceedings. (A80-49226 21-34) London, Imperial College of Science and Technology, 1979, p. 4.12-4.17. Ministry of Education of Japan