Numerical Modelling of an Impinging Jet

Matthew S. Mason, Graeme S. Wood
Wind Engineering Services, Department of Civil Engineering, The University of Sydney
David F. Fletcher
Department of Chemical Engineering, The University of Sydney

1.0 Introduction

Impinging flow occurs when a fluid impacts a comparatively solid boundary upon which divergence occurs. A perfect example of an impinging flow is the impact and divergence of air at ground level during a thunderstorm outflow. The importance of modelling thunderstorm outflows, and in particular the downburst is now well-known to the wind engineering community and research into many of its characteristics is underway throughout the world. The reader is directed to the text by Fujita [1] for an introduction to downburst concepts and theory.

Thunderstorm downbursts can be simply (at least initially) modelled as an impinging air jet. To date this has been done primarily physically, but has recently been studied numerically [2]-[7]. The numerical simulation of downbursts as impinging jets seems a logical step when considering wind profiling, because it allows simple and efficient variation of parameters that are difficult to produce within a physical experiment, e.g. transient and variable initial conditions. Numerical simulations also allow the researcher to use a full-scale domain negating the possible scaling errors that may occur from working physically at model scale.

Numerically modelling impinging jet flow is not as simple as modelling a typical shear flow. Difficulties occur because flow in the stagnation region is almost "irrotational normal straining" and contains significant streamline curvature, unlike simple shearing for which standard CFD turbulence models have been developed [8]. This difference often leads to an over prediction of turbulence in the stagnation region. Therefore the first step in any study of impinging flows must be the determination of an appropriate turbulence model. This is the primary aim of this paper. A comparison between experimental data obtained for a fully developed impinging pipe flow [8] and numerically produced velocity and turbulence kinetic energy (TKE) profiles has been performed to determine the most appropriate closure strategy for the numerical transport equations. A turbulence model that leads to velocity and turbulence characteristics that compare well with experimentation gives confidence for future extrapolation of experiments which cannot be validated physically. This paper presents results obtained using the commercially-available CFD code ANSYS CFX-10.0 and compares the turbulence models given in Table 1. All specifics concerning the mathematical formulation of each turbulence model can be found in [9]. In this paper outflow comparisons are made for a single Reynolds number; in future this will be expanded to verify if the outflow is Reynolds number dependent.

Table1: Summation of turbulence models.

Turbulence Model	Eddy Viscosity Model (EVM) Reynolds Stress Model (RSM)	Resolved to wall
Shear Stress Transport (SST)	EVM	Yes
k-ε	EVM	No
RNG k-ε	EVM	No
k-ω	EVM	Yes
Baseline (BSL) k-ω	RSM	Yes
SSG	RSM	No

This paper represents the first steps in a PhD project: therefore results presented should not be considered final as not all numerical checks have been carried out to confirm these results.

2.0 Model Setup

The flow being modelled is a, stationary, steady flow, impinging air jet (25°C) with a pipe Reynolds number of approximately 24 000. The computational domain is shown in Fig.1 and is similar to an impinging jet validation report [10]. The domain itself is 13D (jet diameters)× 13D based on a jet diameter of 0.0402 m. Since the flow is steady and axisymmetric it was possible to run a 2D simulation with a 1° rotation of the geometry. The jet outlet is set at 2D above the impingement surface so that results can be compared directly with the experimental result of [8]. Although the numerical diameter is slightly different to that in [8] the difference is not expected to affect the results significantly.

The velocity, *k* (TKE) and ε (turbulent eddy dissipation) inlet conditions were set for flow at the top end of the inlet pipe and were determined from separate fully-developed pipe flow simulations. The top and side boundaries were setup as openings with the front and back faces set as symmetry boundaries. The ground was treated as a no-slip smooth wall.

The simulation was performed on a structured inflation grid up to a height of approximately 0.075D above which a tetrahedral mesh was used. The y^+ value at the ground was below 2.0 for all turbulence models that resolved the flow characteristics to the boundary, and approximately 15 for the turbulence models that implemented a scalable wall function (not resolved to the wall). An initial mesh independence study was undertaken based on the *k*-ε and SST models, however some of the presented results show signs of a problem with the switch from the inflation layer to tetrahedral mesh which needs to be addressed.

3.0 Results

Fig.2 displays the normalised mean velocity profiles along the wall at radial distances of r/D = 0.5, 1.0 and 2.5. Fig.2a shows good comparison between all turbulence models except the SSG Reynolds stress model. The SST and RNG *k*-ε models follow the experimental results most accurately in the lower boundary layer region ($z/D < 0.02$), but after which point they slightly (<10%) over-predict the velocities. Fig.2b shows the normalised velocities at r/D=1.0, which is close to the point of maximum velocity and thus extremely important for structural design. The SST model is shown to perform very well for the region below z/D=0.1 (perhaps up to 50 m to 150 m at full-scale). All other turbulence models under-predict the magnitude of the maximum velocity. For the region $0.1 \leq z/D \leq 0.2$ it is seen that all models, except the *k*-ε under predict the velocity magnitude. This result is in contrast to that reported by [10], who showed a very good agreement over this region using the SST model. This area will be further inspected when the grid refinement is reassessed. In Fig.2c, at r/D=2.5, the SST and RNG *k*-ε are seen to over predict the peak velocity while all other models under predict its value. Above z/D=0.1 most of the turbulence models over-predict the velocity magnitude while the RNG *k*-ε predicts the values very well, and the SST model slightly under predicts the values. From these results it is deduced that no turbulence model predicts mean velocity profiles exactly with arguably the best model, the SST being out by just over 10% in the worst case.

Fig.3 shows a plot of normalised turbulent kinetic energy (TKE) against height. The experimental results presented in these figures are not true TKE values as they incorporate only u' and w' (Fig.1) components instead of u', v' and w' components as is standard practice. The v' component is the r.m.s. velocity fluctuations (respectively the same for u and v directions) parallel to the surface but normal to the mean flow direction (this would be in and out of the page in Fig.1) and has not been reported in literature. This component is not expected to be particularly large but would serve to increase the experimental values. Therefore the experimental TKE values should be taken as an estimation of shape and approximate magnitude (though always on the low side). At r/D=0.5, Fig.3a, the SST model shows the best agreement, in shape and magnitude to the experimental results presented. The *k*-ε model, as expected from previous studies [8], over-predicts the turbulence energy in this near stagnation region. Even though this figure shows exaggerated turbulence levels the velocities shown in Fig.2 at r/D=0.5 are not too bad; as velocities in this region are driven primarily by pressure [8], the heat transfer or turbulent mixing would be a different matter. It can be seen that some seemingly unphysical behaviour occurs in both the BSL and *k-ω* TKE profiles at approximately z/D=0.075; this point is associated with the transfer from a structured inflation layer to a tetrahedral mesh and it is hoped that these problems shall be remedied with further grid refinement. Looking at Fig.3b the BSL, RNG *k-ε* and SST models all show profiles of comparable shape and magnitude to the experimental results. The peak turbulence observed with all models occurs between z/D of 0.1 and 0.15 as this is where the wall jet is mixing with the surrounding air. A similar shape (to Fig.3b) experimental TKE approximation is observed in Fig.3c, but with a peak magnitude approximately twice the magnitude but occurring at a marginally lower height; this is what is expected with the thinning of the wall jet. Again the SST and RNG *k-ε* models produce the best looking results with all other models over-predicting the turbulence levels.

4.0 Conclusions

From the results in Fig.2 and Fig.3 it is believed that for the steady jet case the SST model produces the most realistic results and would be recommended for use. This selection may however become problematic when considering a non-stationary impinging flow as the SST model, being based on the *k-ε* and the *k-ω* [9] models, makes the assumption of an isotropic eddy viscosity (analogues to assuming isotropic turbulence) which is not realistic in this type of flow and may cause problems when trying to resolve any vortex motion. This however is the subject of future work and may require a shift to either a Reynolds stress model (from presented results the BSL looks the most promising), or a much more computationally expensive Large Eddy Simulation (LES) based model.

5.0 References

[1] Fujita. T. T., "The Downburst – Microburst and Macroburst", Univ. Chicago Press, Chicago, 1985

[2] Holmes, J.D., "Physical modeling of Thunderstorm downdrafts by wind-tunnel jet," *AWES, in 2nd Workshop on Wind Engineering, Melbourne*, February 20-21, 1992

[3] C.W. Letchford, M.T. Chay, Pressure distributions on a cube in a simulated thunderstorm downburst, Part B: moving downburst observations, J. Wind Eng. Ind. Aerodyn. 90 (2002) 733–753.

[4] Mason, M., Letchford, C., James, D., "Pulsed wall jet simulation of a stationary thunderstorm downburst – Part A" J. Wind Eng. Ind. Aero. Vol. 93, pp 557-580, 2005

[5] Selvam, R.P., Holmes, J.D., "Numerical Simulation of Thunderstorm Downdrafts", J. Wind Eng. Ind. Aero., 41-44, pp 2817-2825, 1992

[6] Wood G., Kwok, K., Motteram, N., Fletcher, D.F., "Physical and numerical modelling of thunderstorm downbursts", J. Wind Eng. Ind. Aerodynamics, 89, pp 535-552, 2001

[7] Hangan, H., Roberts, D., Xu, Z., Kim, J-D. "Downburst simulations. Experimental and numerical challenges", ICWE, in 11th International Conference on Wind Engineering, Lubbock, Texas, Part 2, pp 2241-2248, 2003

[8] Craft, T.J., Graham, L.J.W., Launder, B.E. "Impinging jet studies for turbulence model assessment – 2. An examination of the performance of four turbulence models." Int. J. Heat Mass Transfer, Vol. 36, No. 10, pp. 2685-2697, 1993

[9]ANSYS CFX-10.0 User Manual, 2005

[10] Vieser, W., Esch, T., Menter, F., "Heat transfer predictions using advanced two-equation turbulence models", ANSYS CFX Validation Report, CRX-VAL10/0404, 2004

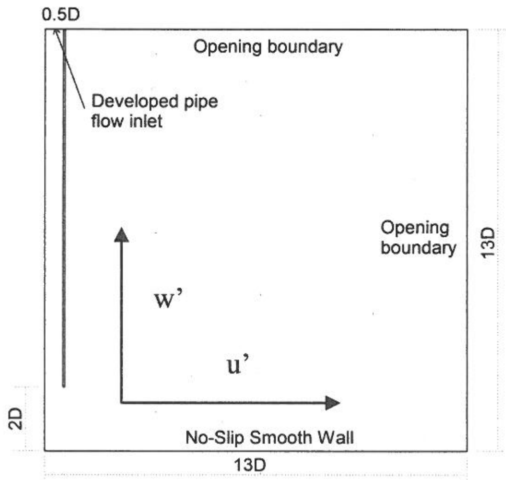

Fig.1: Impinging jet geometry

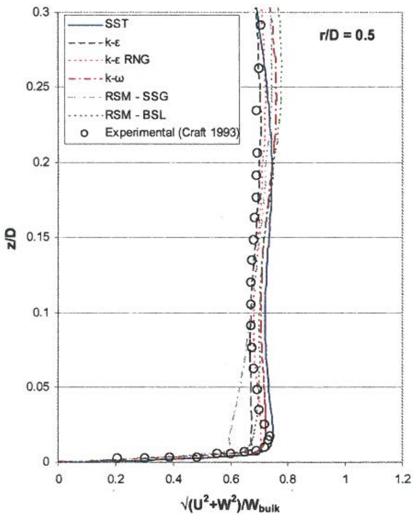

Fig.2a: Velocity profile, r/D = 0.5

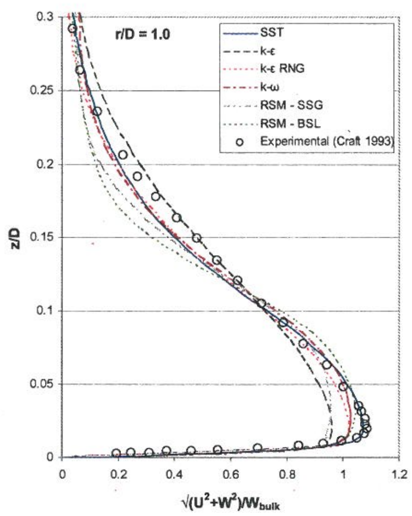

Fig.2b: Velocity profile, r/D = 1.0

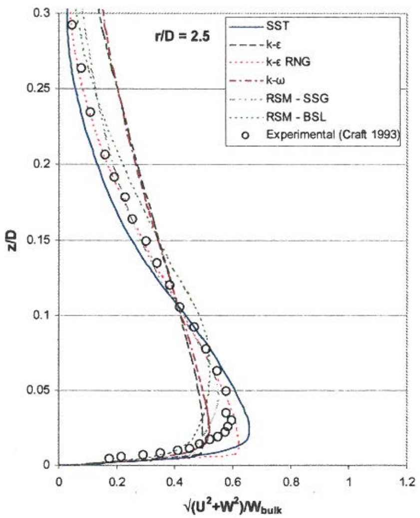

Fig.2c: Velocity profile, r/D = 2.5

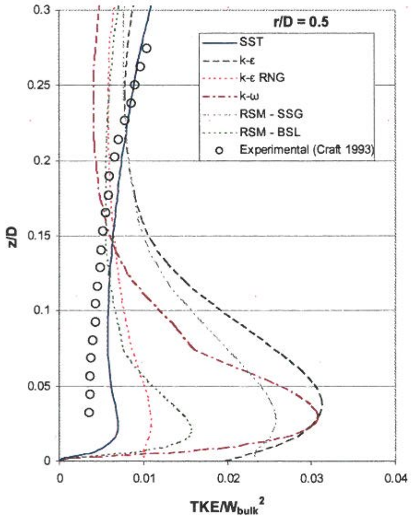

Fig.3a: TKE profile, r/D = 0.5

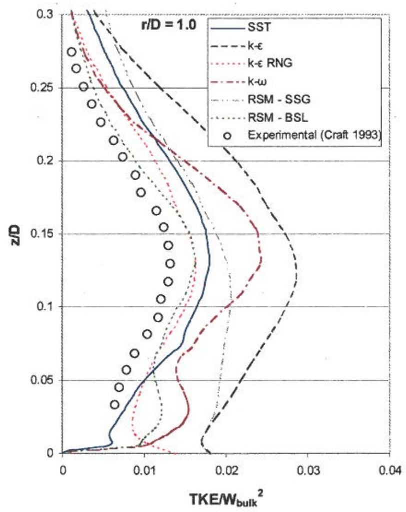

Fig.3b: TKE profile, r/D = 1.0

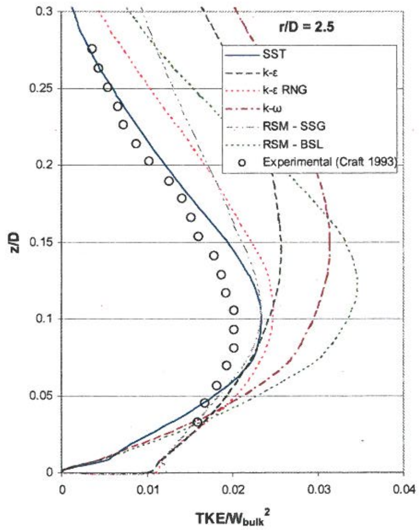

Fig.3c: TKE profile, r/D = 2.5