Proceedings of COBEM 2005
18th International Congress of Mechanical Engineering
November 6-11, 2005, Ouro Preto, MG

SIMULATION OF A SEMI-CONFINED IMPINGING JET

Fábio Alfaia da Cunha
Mechanical Engineering Departament (CT/ DEM/ UFPA), Belém – PA – Brasil, CEP: 66075-110
alfaia@ufpa.br

Pedro Andrey Cavalcante Sampaio
Mechanical Engineering Departament (CT/ DEM/ UFPA), Belém – PA – Brasil, CEP: 66075-110
asampaio@ufpa.br

Danielle R. S. Guerra
Mechanical Engineering Departament (CT/ DEM/ UFPA), Belém – PA – Brasil, CEP: 66075-110
daguerra@ufpa.br

Abstract. *Impinging jet flows are frequently used in industrial practice because of the high rates of heat exchange. Typical applications include heating, cooling and drying processes such as the tempering and shaping of glass, the drying of textile and paper products. The objective of the present work is to report numerical modeling of jets using the FLUENT 6.0 commercial software. This has been used to investigate the effects of nozzle-to-plate spacing and turbulence models on the flow field of jet in the free, impinging and wall regions. A semi-confined configuration was used. In this problem, fluid is injected from a round nozzle with 43,5mm inner diameter, at nozzle-to-plate spacing of H/D=2 and Reynolds number of 35,000. Four models of turbulence are adopted in this work and the results are compared with experimental data of velocity and temperature field to determine the applicability of the model in the study of an impinging turbulent jet.*

Keywords*: Impinging jet, temperature field , velocity field.*

1. Introduction

The flow produced by an impinging jet provides a high heat transfer rates between the flow and the impingement surface. Due to this fact, impinging jets have been used in a variety of practical engineering applications, such as drying, cooling and heating processes. The applications include anneling of metal sheets, tempering glass, cooling of the turbine blades and mechanical devices. The fluid dynamic structure of the impinging jet is complex and this kind of flow is very unsteady. The turbulent flows such as impinging jets are modeling through turbulent models, such as, Reynolds number kappa-epsilon models, Reynolds stress models and the kappa-omega model.

With the advanced in turbulence modeling and high performance computers, numerical simulations of turbulent impinging jets have been carried out actively. Many authors have been examined numerically this type of flow. Ashforth-Frost and Jambunatham (1996) carried out numerical and experimental study using the standard κ-ε eddy viscosity model of turbulence in conjunction with the logarithmic law of the wall to the prediction of a fully developed turbulent axisymmetric jet impinging within a semi-confined space. The authors have been obtained the velocity, turbulence and heat transfer data using a laser-Doppler anemometry and liquid crystal thermograph respectively. As results, in the wall jet the heat transfer is predicted to within 20% where the turbulent kinetic energy is reasonable predicted by κ-ε eddy viscosity model, however, the stagnation point Nusselt number is over predicted by about 300%.

Behnia et al. (1999) has been used an elliptic relaxation turbulence model ($v^2 - f$) to simulate the flow and heat transfer in circular confined and unconfined impinging jet. The model has been validated against available experimental data. Results have been obtained for a range of jet Reynolds number and jet-to-target distances. The effects of confinement on the local heat transfer behavior have been determined. It has been shown that confinement leads to a decrease in the average heat transfer rates, but the local stagnation heat transfer coefficient is unchanged. The effect of confinement is only significant in very low nozzle-to-plate distances.

Shi et al. (2002) presented simulation results that have been studied through the commercial finite volume code FLUENT 5.0. Two turbulence models, the standard κ-ε model and the RSM were used to calculate the heat transfer under a semi-confined impinging slot jet for various flow and geometric parameters. The authors observed that both the models slightly over predicted the Nusselt number distributions under some conditions, but the qualitative trends compared very well with the experimental trend in most cases. Their simulated results compare favorably with experimental results for large nozzle-to-plate spacing.

Some recent numerical studies of turbulent impinging jet have been used the κ-ω models. The standard κ-ω model in FLUENT is based on the Wilcox κ-ω model (1998), which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The Wilcox model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets, and is thus applicable to wall-bounded flows and free shear flows.

Park et al. (2003) have been used the κ-ω model proposed by Wilcox (1998) to predict the flow and heat transfer characteristics of two-dimensional confined impinging slot jets. The results of their study were that for turbulent impinging jets, the x-component of the mean velocity is shown to be in excellent agreement with the experimental data where the wall jet is developing; the calculated Nu distribution is also shown to be in good agreement with the experimental data for low Reynolds number range, but as the Reynolds number increases the Nu distribution disagree more with the experimental data.

Turbulence models present deficiency near the wall region. Generally, in these cases, wall functions are used to calculated flow field, turbulent kinetic energy and temperature field. A version of the law of the wall proposed by Launder and Spalding is used in industrial flows. In the FLUENT code this version is known as *Standard Wall Function*. In this function the rates of production and dissipate turbulence energy are equal in the computational cell adjacent to the wall. The logarithmic law is employed when y+ > 11.225.

Shi et al (2004) have been predicted the heat transfer under a semi-confined turbulent round impinging jet through the FLUENT code. The authors used the *Standard Wall Function and Non-equilibrium Wall Function* (1995). They concluded that the Standard Wall Function predict values higher than the Non-equilibrium Wall Function for both standard κ-ε and RSM turbulence models. Some near-wall treatments for wall-bounded turbulent flows can be founded in the code, like as the Enhanced wall treatment, that is a near-wall modeling method that combines a two-layer model with enhanced wall functions. The description of the models is shown in the manual of the code FLUENT (2003).

In this work a CFD, FLUENT 6.0 code, is used to predict the velocity and temperature field of a semi-confined turbulent round impinging jet. The purpose of the work is to present the results that have been obtained using four turbulence models: RNG κ-ε (1986), Realizable κ-ε (1995), SST κ-ω (1994) and Reynolds Stress models (1975). The simulation results were compared with experimental data available in Guerra et al. (2005).

2. Description of the Problem and Experimental Apparatus.

The test section of the experimental apparatus is shown in Figure 1. Air at 18.5 ^oC is pumped through a centrifugal blower connected to a 1350 mm long pipe with 43.5 mm internal diameter. Inside the pipe, a honeycomb is fitted, constructed from drinking straws glued together; screens are also set in place. The jet is set to emerge from the circular nozzle with a bulk velocity of 12 m/s.

The impingement flat surface is made of a 3.7 mm thick aluminum circular sheet. This sheet has 840 mm in diameter and is laid over a plenum chamber as shown in Figure 1. The plenum chamber is 20 mm height and 815 mm in diameter. At the bottom of the chamber a series of electrical resistances are placed. The walls of the plenum were completely insulated from the ambient.

The controlled parameters in the experiments are the nozzle-to-plate spacing, the resistance heat flux and the stagnation pressure. At each test, the centerline of the jet is lined up with the center of the impingement surface. The temperature of the aluminum sheet was monitored through thermocouples. The readings of the thermocouples were routed to an AMD Athlon +2000 MHz personal computer via a Picolog acquisition system model TC-08.

Mean velocity profiles and turbulence intensity levels were obtained using a DANTEC hot-wire system series 56N. The boundary layer probe was of the type 55P15. A Pitot tube, an inclined manometer, and a computer controlled traverse gear system were also used. In getting the data, 10,000 samples were considered. The profiles were constructed from about 100 points. The mean temperature profiles were obtained through a chromel-constantan micro-thermocouple mounted on the same traverse gear system used for the hot-wire probe. The traverse gear system has 0.02 mm sensitivity.

To develop the numerical study a schematic of the problem was drawing, and then the simulation domain was defined as shown in Figure 2.

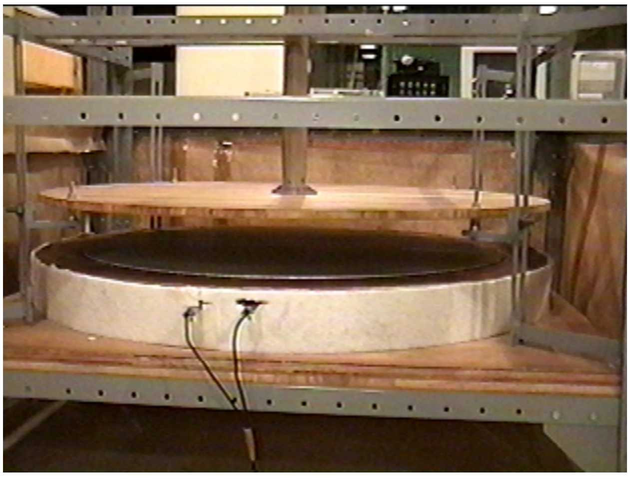

Figure 1. The test section of the experimental apparatus.

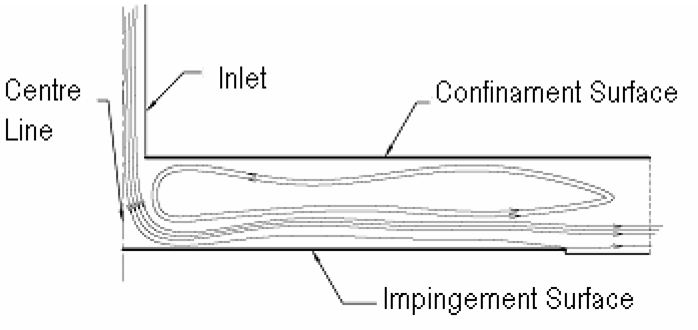

Figure 2. The simulation domain.

3. The Computational Conditions

3.1. Grid and Geometry

The geometry of the plate used in the simulation is shown in Figure 3. The model is two-dimensional and axysimmetric to the centre of the axis. The geometry has been divided in four parts to facilitate the discritization process of the continuous domain. The cells have been constructed in two ways: triangles and quadrangles. The quadrangles cells were used adjacent to the boundary. The grid of the areas 1, 3 and 4 were constructed by quadrangles cells. The area 2 was constructed of triangles and quadrangles cells, this type of grid helps in the transition of the elements size inner the domain. The cells that have been used in the area 1 were done to have the smaller value of aspect ratio (< 1.3), except near the wall where the grid refinement was done (<5), 13086 cells was used in this region. The areas 3 and had 15405 (79×195) and 13921 (79×176) quadrangles cells, respectively. In the area 2 have been used 12846 cells as shown in Figure 6. The maximum cells aspect ratio was 10 in the regions 2, 3 and 4.

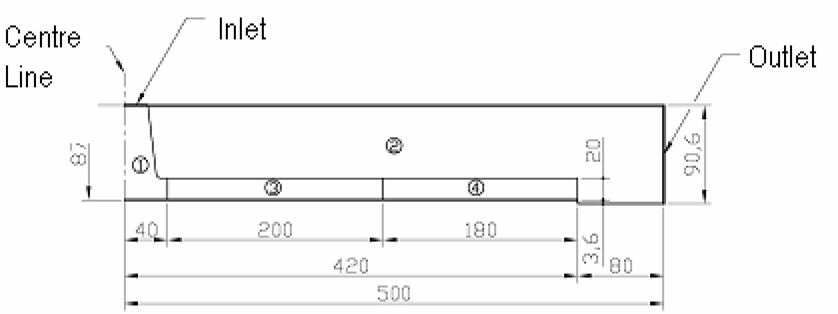

Figure 3. Flow geometry of the impinging jet: definition of different regions.

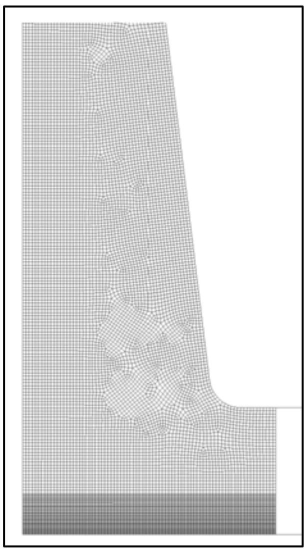

Figure 4. Grid: area 1

The plate grid was done in steps. First an initial grid was constructed and adapted for the gradients of the flow; next to the wall was necessary to do refinements. These refinements have been done with the condition of y+ about 1. This value has been recommended by the code to use the *enhanced wall treatment* and the SST k-ω model. The dimensions of the elements near the impact region were 100μm, but with refinements the elements dimensions decreases to 25μm in some positions.

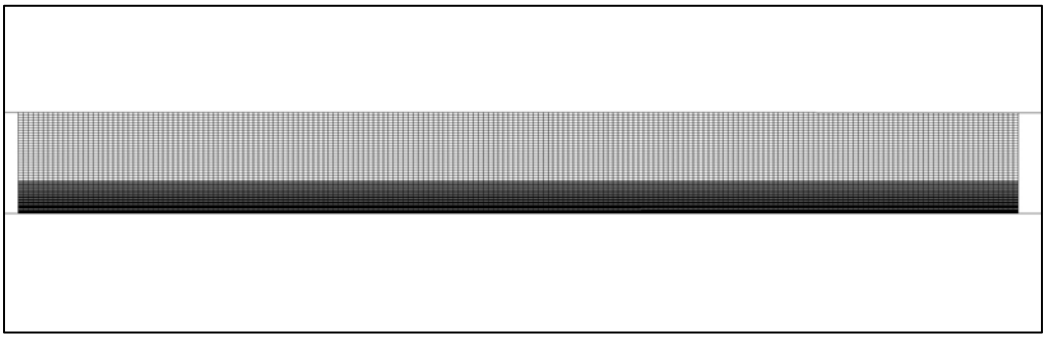

Figure 5. Grid: area 3

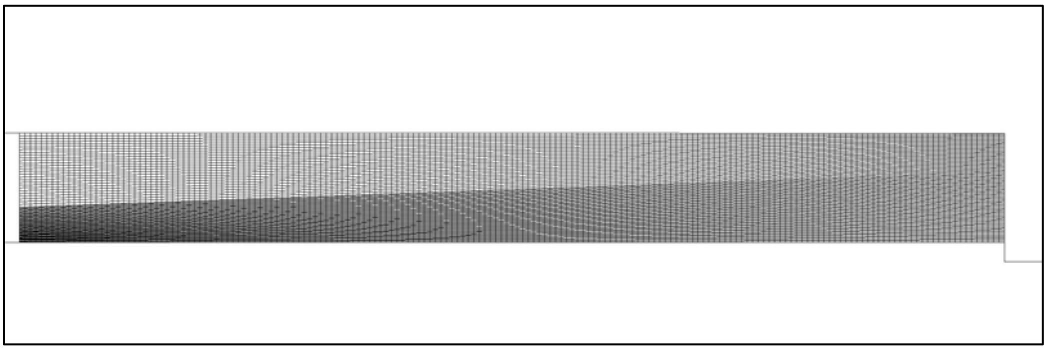

Figure 6. Grid: area 4

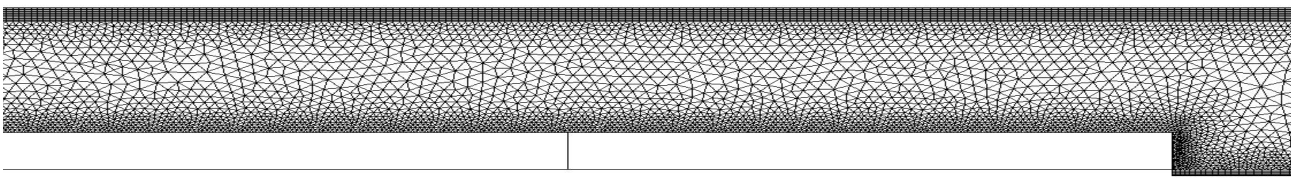

Figure 7. Grid: area 2

3.2. Numerical simulation

The numerical simulations were performed with a computational fluid dynamic code, the FLUENT 6.0, which has models that can predict the velocity and temperature flow field. In this code the governing equations are solved by the finite volume methods in a non-structure grid system. In the finite volume method the differential governing equations are integrated in a conservative form, in control volumes generating an algebric equations system. This system of equation is solved by the code through iterative form.

The procedure to begin the simulation is to provide an initial value different to zero with a not refinement grid. The results have been interpolated in a refinement grid. Park *et al*. (2003) noted that the structure of an impinging jet is so complex that its numerical prediction could be severely affected by false diffusion (artificial diffusion) of an upwind type numerical method. Based in this observation, a second order upwind scheme has been used in the momentum and turbulence equations. The second order scheme is necessary in these equations to minimize the called numerical diffusion intrinsic to the used scheme. The numerical diffusion lies due the upwind scheme which tends to soothe higher gradients (Maliska, 2004). In this work a second order scheme has been used in the pressure equation. The algorithmic SIMPLE has been used for associate pressure and velocity field.

The boundary conditions that have been used in this work are described as following. At the nozzle exit a velocity profile was applied with a temperature of 23.2ºC, that is has been obtained in a previous simulation. The simulation has been conducted for a bulk velocity of 12 m/s at the centre of the jet. A turbulence intensity of 5% and inner nozzle diameter of 0.0435 m have been fixed. A heat flux of 284.16 W/m^2 was applied to predict the temperature field. The confinement surface was considered adiabatic.

A suitable grid has been generating but some measurements were necessary to the convergence of the calculus. In all cases the solver has been adjusted to run with double precision. For this calculus, it was necessary more computational memory, therefore the calculus has been presented more stability.

4. Results

The results that have been obtained through the simulations are presented for some positions in the plate for various turbulence models. The radial positions are: r = 80, 105, 110, 120, 130, 140 and 150 mm measured from the centre of the nozzle. Four turbulence models have been applied in this work and compared with experimental data of velocity and temperature flow field. Figures 8 – 13 show the temperature profiles and figures 14 – 19 show the velocity profiles.

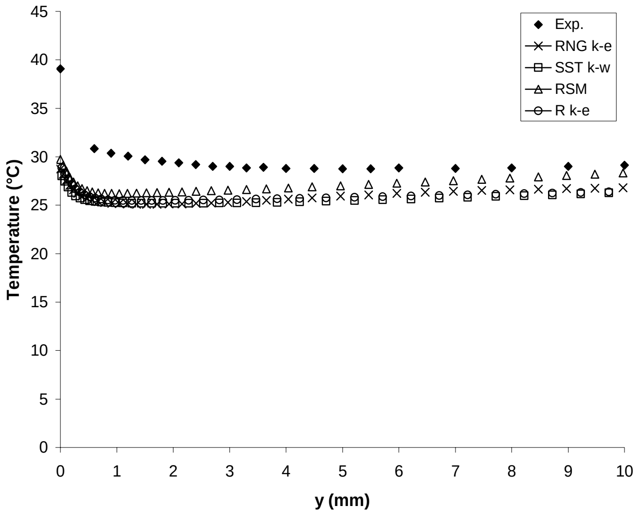

Figure 8. Position r= 80mm

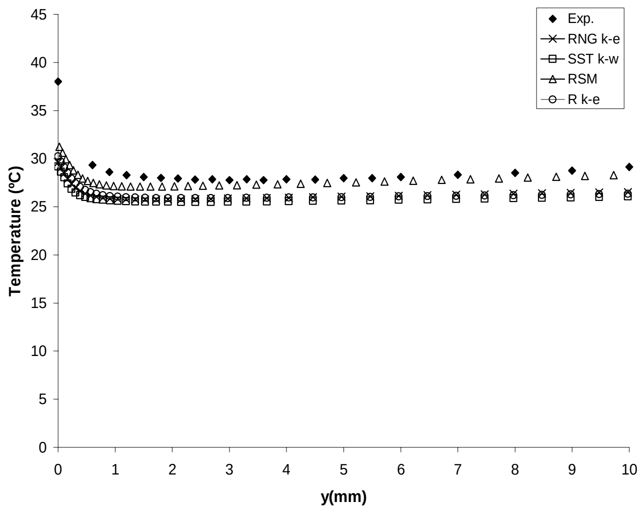

Figure 9. Position r= 105mm

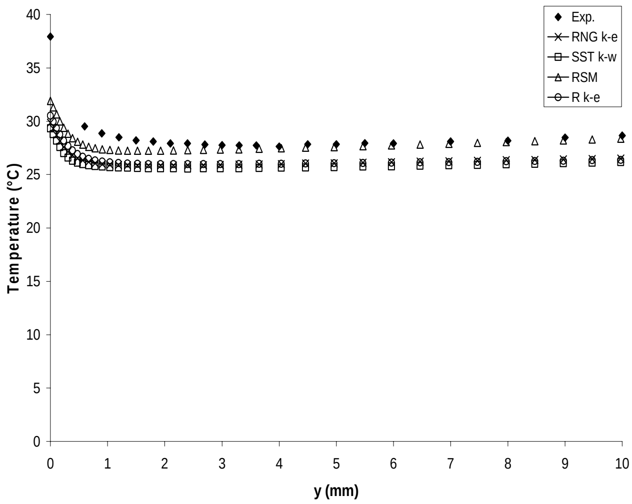

Figure 10. Position r= 110mm

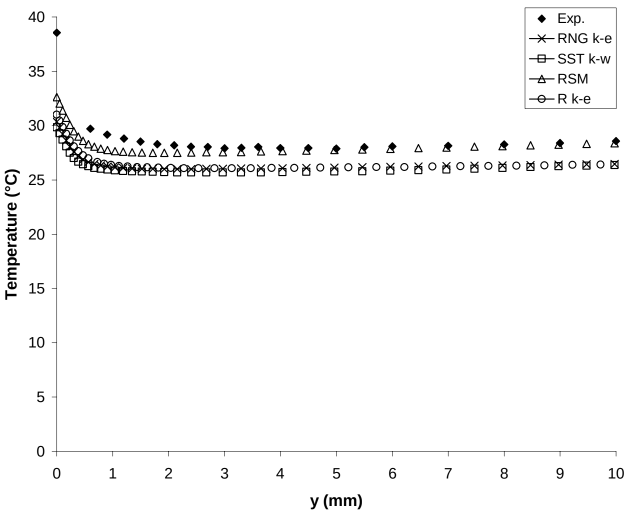

Figure 11. Position r= 120mm

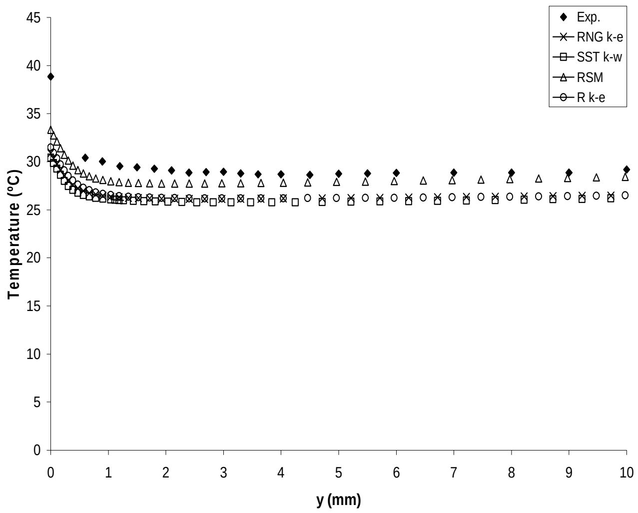

Figure 12. Position r= 130mm

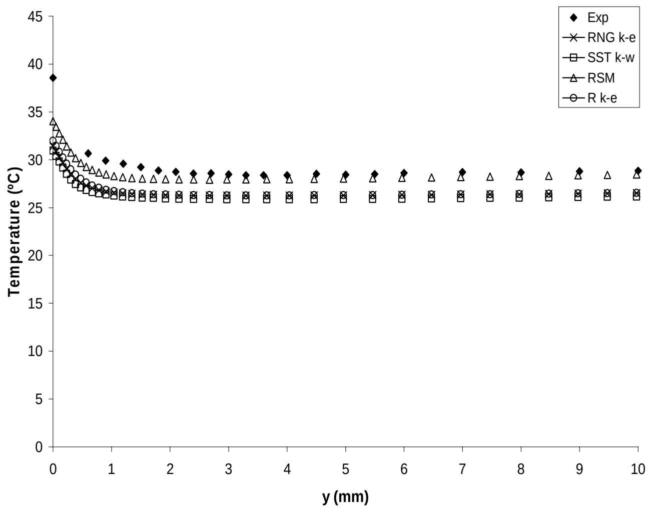

Figure 13. Position r= 140mm

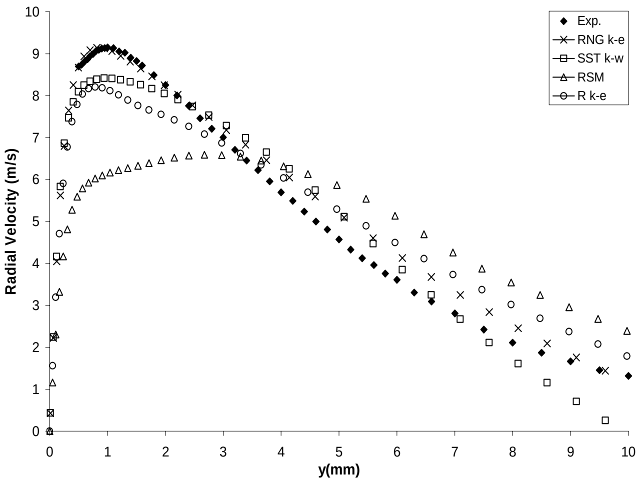

Figure 14. Position r= 80mm

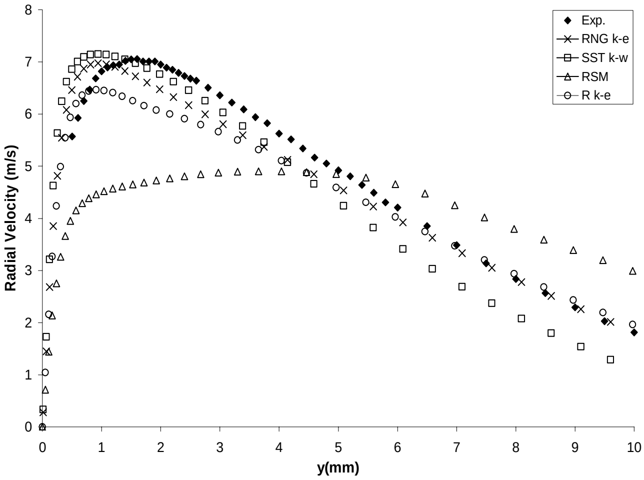

Figure 15. Position r= 105mm

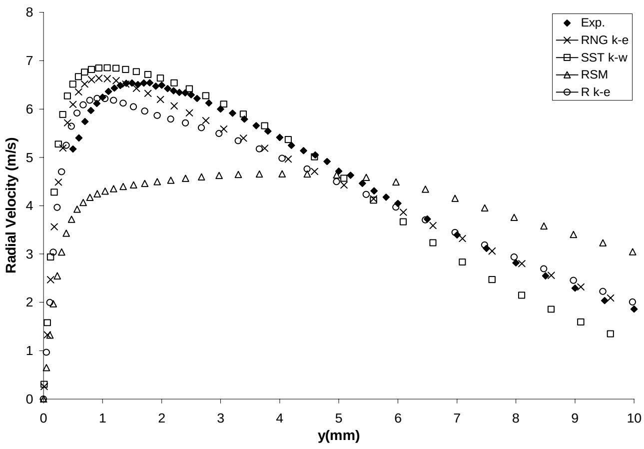

Figure 16. Position r= 110mm

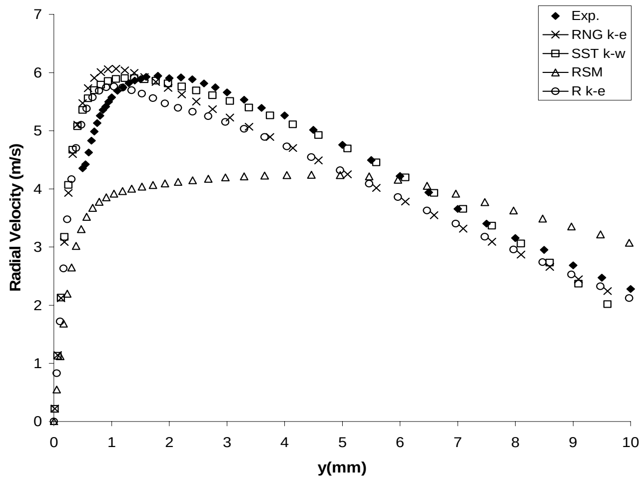

Figure 17. Position r= 120mm

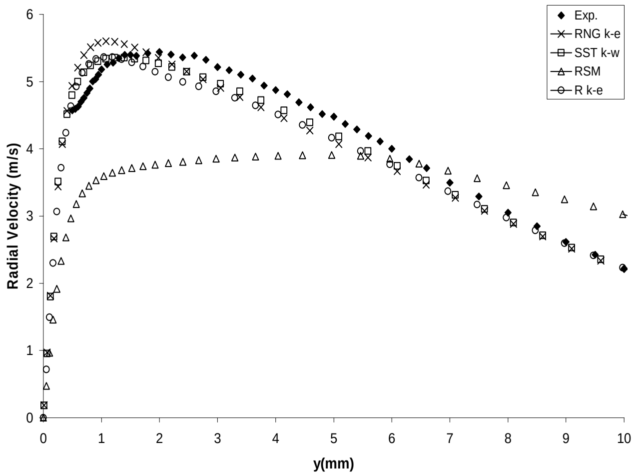

Figure 18. Position r= 130mm

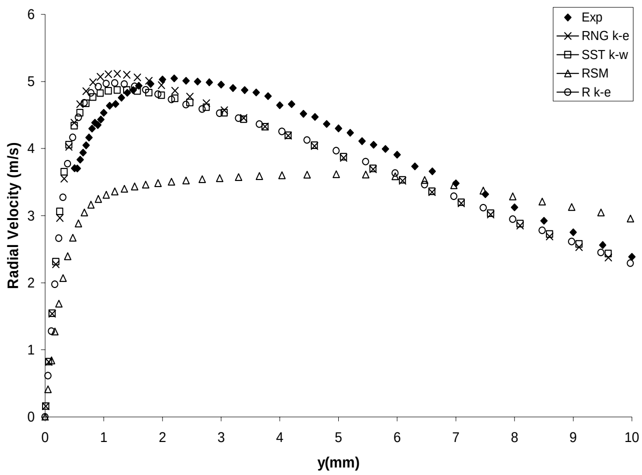

Figure 19. Position r= 140mm

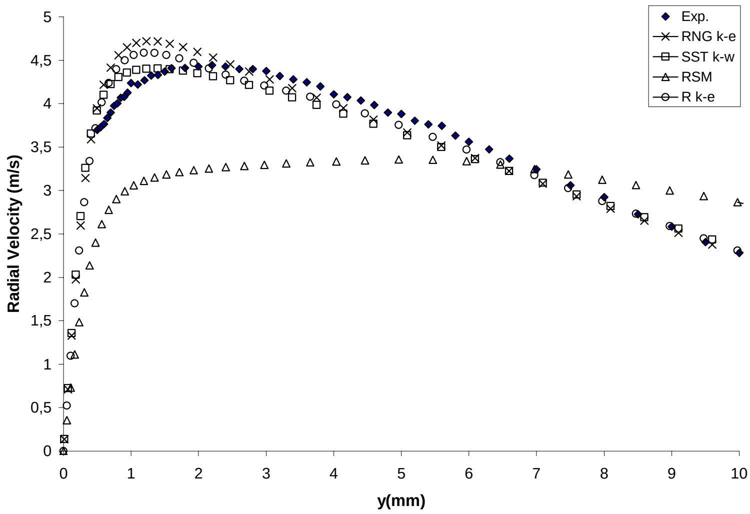

Figure 20. Position r= 150mm.

The present results is for a semi-confined turbulent impinging jet, for a nozzle-to-plate spacing of H/D= 2.0 and Re= 35.000. Numerical and experimental profile of radial velocity and temperature are compared for a various positions. Figures 8 – 13 show the temperature field. The temperature prediction using RNG k-ε, Realizable k-ε and SST k-ω disagree from the experimental data in all positions, however the RSM model has been showed a prediction near the experimental data.

The simulation results for prediction the velocity field have been shown that near the wall, in the inner region, all turbulence models disagree of the experimental data, they achieved very poor agreement with experiments. Far from the wall at the outer region of the profile, the SST k-ω has been provided best results.

About the grid some configurations have been done in which the aspect ratio cells were smaller than 10 and this condition provided a more stable calculus. All variable have been monitoring and checking during the calculus to determine the solution convergence. The convergence occurs when the convergence criterion is satisfied. The criterion consists that each residue was smaller than 10^{-4}. The calculus convergence for RNG k-ε, Realizable k-ε and SST k-ω models have been reached around 2000 interaction from a convergence solution with Standard k-ε model. The RSM model the convergence has been reached around 1000 and 1500 interactions from a solution obtained through the RNG k-ε model.

5. Conclusions

The present work has described the behavior of a semi-confined turbulent round impinging jet over a flat surface. Results of numerical simulations using four turbulence models, RNG κ-ε, Realizable κ-ε, (SST) κ-ω and Reynolds Stress models have been compared with experimental data for the velocity and temperature flow field. The analysis of the velocity profiles have been shown that the RSM model gave poor results, this turbulence model wasn't able to predict the velocity data at the inner and outer region of the field. Near the jet nozzle, at positions 80 and 105 mm, for the inner region of the flow, the RNG κ-ε is in a good agreement with the experimental data as shown in Figure 14 and 15, but far from the jet nozzle the model the results wasn't good. However, the RNG κ-ε model gave best results when compared with the three other models.

In the case of the temperature profiles, the RSM model gave the best results. Spite of the results obtained at the position r= 80 mm, the model was able to predict the temperature in the inner and outer region of the field. It's important to note that the difficult to measure and to predict numerically the velocity and temperature near the jet nozzle, is due the existence of the stagnation region.

The results found at this preliminary investigation indicate that the inner region of the velocity profile wasn't predicted well by the turbulence models. The same occurs in the temperature results. The research will continue to predict the skin friction coefficient and turbulence intensity.

Acknowledgements. D.R.S.G. is grateful to the Brazilian National Research Council (CNPq) for the award of a research fellowship (Grant No 150403/2004-6). P.A.C.S. is grateful to the FUNTEC for the award of a scholarship.

6.References

Ashforth-Frost, S. and Jambunathan, K., 1996, "Numerical Prediction of Semi-Confined Jet Impingement and Comparison with Experimental Data", International Journal for Numerical Methods in Fluids, Vol.23, pp. 295-306.

Behnia, M. and Parneix, S., 1998, "Prediction of Heat Transfer in an Axisymmetric Turbulent Jet Impinging on a Flat Plate", Int. J. Mass Transfer, Vol.41. No. 12. pp. 1845 – 1855.

Behnia, M., Parneix, S., Shabany,Y. and Durbin, P. A., 1999, "Numerical Study of Turbulent Heat Transfer in Confined and Unconfined Impinging Jets", International Journal of Heat Fluid Flow, Vol.20. pp. 1 – 9.

FLUENT Inc., 2003, "FLUENT 6.1, User's Guide Volume", Vol. 1-4, Lebanon, USA.

Guerra, D. R. S. and Silva Freire, A. P., 2003, "An experimental heat transfer study of a cold jet impinging onto a hot surface", Congresso Brasileiro de Engenharia Mecânica, São Paulo, December.

Guerra, D. R. S. and Silva Freire, A. P., 2004, "A study of the heat transfer behaviour for a cold jet impinging upon a hot surface", Congresso Nacional de Engenharia Mecânica, Belém, August.

Guerra, D.R.S, Su, J. and Silva Freire, A. P., 2005, The near wall behavior of an impinging jet, International J. Heat and Mass Transfer.

Hinze, J.O., 1975, "Turbulence", McGraw-Hill Publishing Co., New York.

Kim, S.E. and Choudhury, D., 1995, "A Near-Wall Treatment Using Wall Functions Sensitized to Pressure Gradient", In ASME FED, Vol.217, Separated and Complex Flows, ASME.

Maliska, C. R., 2004, "Transferência de Calor e Mecânica dos Fluidos Computacional".

Menter, F.R., 1994, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, Vol.32, No 8, pp.1598-1605.

Park , T. H., Choi, H. G. and Yoo, J. Y. and Kim, S. J., 2003, "Streamline Upwind Numerical Simulation of Two-Dimensional confined Impinging Slot Jets", International Journal of Heat and Mass Transfer, Vol.46, pp. 251 – 262.

Shih, T.H., Liou, W.W., Shabbir, A. and Zhu, J.,1995, "A New k-ε Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation", Computers Fluids, Vol.24, No 3, pp.227-238.

Shi, Y., Mujumdar, A. S. and Ray M. R., 2004, "Effect of Large Temperature Difference on Impinging Heat Transfer Under a Round Turbulent Jet", Vol.31, No. 2 , pp. 251 – 260.

Shi, Y., Ray, M. B. and Mujumdar A. S., 2002, "Computational Study of Impinging Heat Transfer Under a Turbulent Slot Jet", Ind. Eng. Chem. Res, Vol.41, pp. 4643 – 4651.

Wilcox, D.C., 1998, "Turbulence Modeling for CFD, DCW Industries", Inc, La Canada, CA, pp. 84-87.

Yakhot, V. and Orszag, S.A.,1986, "Renormalization Group Analysis of Turbulence: I. Basic Theory", Journal of Scientific Computing, Vol.1, No 1, pp.1-51.