J. of Thermal Science Vol.13, No.1

A Numerical Analysis of Turbulent Compressible Radial Channel Flow with Particular Reference to Pneumatic Controllers

Gilles Roy Dinh Vo-Ngoc Vadim Bravine
Faculty of Engineering, Université de Moncton, Moncton, NB, Canada, E1A 3E9

A numerical analysis of turbulent axisymmetric radial compressible channel flow between a nozzle and a flat plate is presented in this paper. In particular, the application of this type of flow situation in pneumatic dimensional control systems is considered. The Spalart-Allmaras one-equation turbulence model is used. The resulting highly coupled PDE system has been solved using the control volume based numerical approach where the power-law scheme was used extensively to compute the diffusive and convective fluxes of momentum. Results show that local Mach numbers can easily achieve and surpass unity for typical industrial configurations. Also, in the case of a standard industrial nozzle geometry, the presence of a toroïdal recirculation zone that moves radially outward is clearly identified in most cases. Separated flow areas are of particular concern as it has been shown previously that they can cause nozzle fouling in industrial applications. It has been shown that the size of this region is dependant on feed pressure. Considerable differences between results obtained using the Spalart-Allmaras and standard k- ε turbulence models have also been noticed.

Keywords: compressible radial flow, numerical simulation, pneumatic controllers, impinging jet with confinement.

Introduction

The study of radial flow, either as flow between disks or between a nozzle and a flat plate, has been the subject of numerous research projects over the past half century[1~3], etc. However, research work of such flows for applications in dimensional pneumatic control is very limited. Furthermore, turbulent, compressible radial flows do not seem to have been considered extensively as well. The radial flow between a nozzle and a flat plate for specific applications in pneumatic metrology has, to our knowledge, only been considered in the past 10 years by a group of researchers at the Université de Valenciennes in France[4,5]. These studies have been mainly experimental in nature. Pressure distributions on a flat plate were measured and seemed to indicate the presence of a low pressure separation area for cases using the standard nozzle. Various nozzle geometries were studied and, in some cases, the low pressure area was eliminated. No experimental flow visualization techniques were used (or velocity fields measured – mainly due to the very limited space between the flat plate and the nozzle) and therefore no information on the entire flow field was found. Furthermore, cases studying impinging jets for very small distances separating the nozzle and the flat plate ($\delta/D<0.5$, where δ is the distance separating the nozzle and the flat plate and D is the nozzle diameter), are quite sparse and are also mainly experimental[6]. This range is actually of interest in industrial metrology applications. However, the nozzles used in the studies are thin-walled and therefore do not consider the effects of confinement. Only recently[7], have numerical studies been made in the case of an impinging jet at distances in this range ($\delta/D<0.5$) with an interest in the effects of confinement. However, only standard type nozzles were considered in the aforementioned paper. One can note that because of the small distance "δ" between the nozzle and the flat plate (100 μm ≤ δ ≤ 200 μm compared to a nozzle external diameter of 4 mm), the only practically measurable quantity is the wall pressure distribution on the surface of the flat plate. Information on the entire radial flow field between the nozzle and the plate can therefore be more easily obtained by numerical simulation, which is presented in this paper.

Received 2003

Pneumatic Dimensional Control Basics

Pneumatic controllers can be used in high precision applications (order of a micron), no physical contact is made between the nozzle and the surface of the machined part, part cleaning is done as the process is on-going, they can measure multiple dimensions simultaneously, they are robust and internal dimensions can easily be measured (i.e. holes/bores). One can find a few different variations of the basic pneumatic dimensional control apparatus, Fig.1. Most of these are based on the detection of a pressure differential inside the apparatus. This pressure differential is highly dependent on the distance "δ" between the nozzle and the surface of the machined part (controlled surface). Quite a large variety of operating pressures can be found with these various methods (as low as 1.5 kPa or as high as 400 kPa). The basic pneumatic control setup consists of a compressed air source regulated by a pressure regulator upstream of the controller, Fig.1(a). The compressed air then enters a chamber consisting of an orifice plate "A" and an injection nozzle "B" placed a small distance away from the surface to be controlled. The pressure inside the chamber is very sensitive to changes in the distance between the nozzle and the controlled surface. This characteristic is used to advantage in industrial metrology as calibration can permit the determination of the distance "δ" as a function of the pressure inside the chamber. However, this type of setup requires a carefully controlled feed pressure. This dependence can be avoided by using a differential apparatus as illustrated in Fig.1(b)[8]. In the differential apparatus, two branches are used, one is used as a reference and the other as the measuring branch with the Δp measured between both chambers.

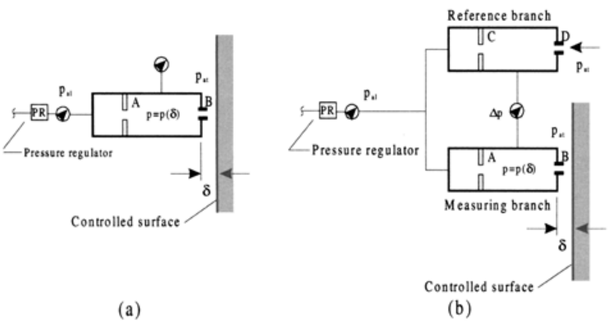

Fig.1 Basic pneumatic controller

Although industrial pneumatic controllers are a well-established and credible way of performing dimensional control in an industrial environment, as in any other type of controller some minor problems are found to exist. Nozzle fouling is one of them. Annular regions of dirt and oil deposits are often found on the frontal surfaces of the injection nozzles, Fig.2. The resulting fouling requires that the controllers be cleaned and calibrated at shorter than desired intervals. In the present paper, we are specifically concerned with the study of the behaviour of the flow between a nozzle and a flat plate with geometries and operating conditions in typical industrial application ranges.

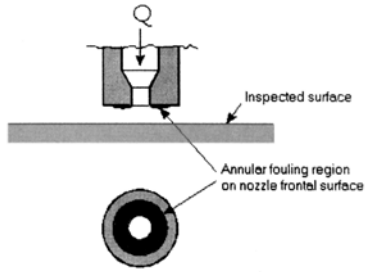

Fig.2 Nozzle fouling

Modeling and Simulation

Problem configuration

The problem configuration considered in the present paper is illustrated in Fig.3. The nozzle and flat plate are stationary and a constant flow rate $\dot{m}$ is axially injected into the domain of interest. The nozzle, which has an interior radius of "R_i" and an exterior radius of "R_e", is separated from the flat plate by a distance of "δ". Usual configurations have a clearance space "h" with the body of the nozzle (generally $\approx 3\ R_i$). The body of the nozzle extends to a radius of R_d which is approximately 5 R_i. Typical industrial dimensions are: R_i =1 mm, R_e =2 mm, δ = 0.15 mm, R_d=5 mm, h = 3 mm.

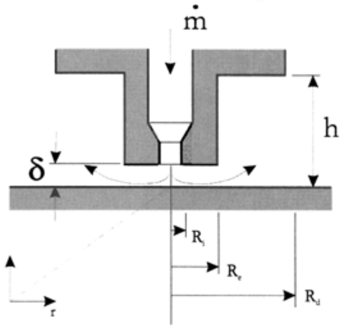

Fig.3 Problem configuration (not to scale)

Governing equations

The turbulent axisymmetric flow of a viscous compressible fluid between a nozzle and a flat plate is considered in this paper. All external forces are considered negligible. The Spalart-Allmaras model is used for turbulence modeling. This recently developed, relatively simple one-equation model solves a modeled transport equation for the kinematic eddy turbulent viscosity[9]. It has been developed for applications in wall-bounded flows and has been shown to give good results in various applications, including 2D U-duct flows[10] and turbomachinery[11]. To our knowledge, its applicability has never been tested in compressible radial flow problems. Under these conditions, the aerodynamic field is governed by a set of six equations (mass conservation, 2D axisymmetric momentum, energy, perfect gas equation of state and the modelled transport equation for the kinematic eddy turbulent viscosity). This set of governing equations is a non-linear and highly coupled PDE system. The mean flow equations, written in the cylindrical coordinate system, are the Reynolds-averaged equations for conservation of mass and momentum. Due to limited space, these standard equations are not presented here as they are readily available in literature. The inlet Reynolds number is defined as $Re_d = 2\dot{m}/\pi\mu R_i$, where $\dot{m}$ is the mass flow rate at the inlet of the nozzle.

Numerical approach

Although the region of interest is limited in the space between the nozzle and the plate, the calculation domain is extended over a large space adjacent to the nozzle, up to R_d, in order to obtain good results in the area of interest (region 2, Fig.4). The calculation domain is divided into several regions with different grid generations in accordance to geometry and flow conditions of each region. Several non-uniform meshings were tested and a 20, 391–node quadrilateral-cell grid has been chosen for subsequent simulations. The grid points are highly packed near the radial flow channel entrance, as well as adjacent to all walls. The set of PDEs previously discussed has been solved using the control volume based numerical approach where the power-law scheme was used extensively to compute the diffusive and convective fluxes of momentum[12].

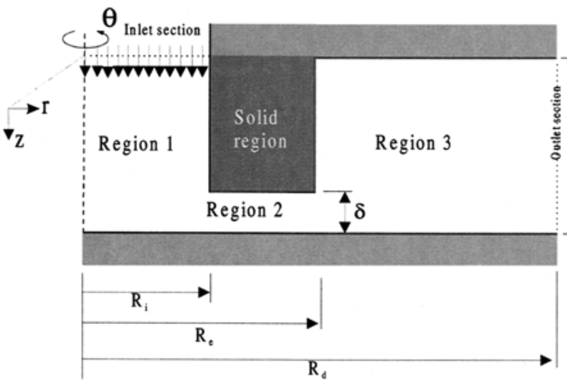

Fig.4 Considered domain for calculations with used dimensions (not to scale)

The boundary conditions used were the following:

- No-slip condition on all walls,
- Total pressure and total temperature at inlet section (with an initial static pressure),
- Static pressure at outlet (i.e. standard atmospheric pressure),
- Inlet modified turbulent viscosity.

The convergence criteria for all computations presented in this paper were based on the degree of mass conservation obtained over each control volume in the computational domain. These values gradually decreased with the number of iterations and were generally in the order of 0.0001% for all considered cases. This criterion generally corresponds to a precision of approximately 10^{-8} for all independent variables (velocity components, pressure, density and temperature).

Results and Discussions

Standard nozzle typical flow behavior

Flow behaviour for the standard nozzle is shown in Figs.5(a)～(d). The entire pressure field at the inlet section (region 1) is essentially of constant pressure as the velocity in this area is quite small, Fig.5(a). The development of a separated flow area is found on the frontal surface of the injection nozzle, as clearly seen in the general flow contour lines illustrated in Fig.5(b). For the considered parameters (inlet total pressure P_o = 110 kPa, outlet static pressure P_s = 101.3 kPa R_i = 1.0 mm, R_e = 2.0 mm, δ = 0.15 mm, h = 3.0 mm, R_d = 5.0 mm), the separated flow recirculation zone practically covers the entire frontal area of the nozzle. One can also see from Fig.5(c) that the local Mach number for this particular case remains relatively small. As can be noticed in the wall pressure distribution on the flat plate, Fig.5(d), an important pressure drop is found to exist. This can be attributed to the fact that since the radial area at the entrance of the space between the nozzle and the flat plate is smaller than the area of the inlet tube ($A_{space} = 2R_i\pi\delta < A_{tube} = \pi R_i^2$) and therefore the airflow is accelerated. This is true as long as $2\delta < R_i$, which is the case for all results presented here. Furthermore, the presence of a separated flow area considerably reduces the "effective" flow area. This basically translates into a vena contracta effect at the entrance of the space between the nozzle and the flat plate. This general behaviour corresponds with the results found in the experimental work by [5]. Clearly, this separated flow area is the probable cause of fouling in industrial applications.

Effects of inlet total pressure on flow behavior

Fig.6(a) illustrates the effects of inlet total pressure on flow contour lines and Mach number. One can note that all other parameters are the same as for results presented in the typical flow behaviour (i.e. outlet static pressure P_s =101.3 kPa, R_i =1.0 mm, R_e =2.0 mm, δ = 0.15 mm, h = 3.0 mm, R_d =5.0 mm). In these figures, the flow contour lines are superimposed on the Mach number floods. Although the general flow patterns seem quite plausible, one can notice that the local Mach numbers can easily achieve and surpass unity for the considered inlet pressure range, even at relatively low inlet total pressures (i.e. approximately $p_o = 1.6\times10^5$ Pa).

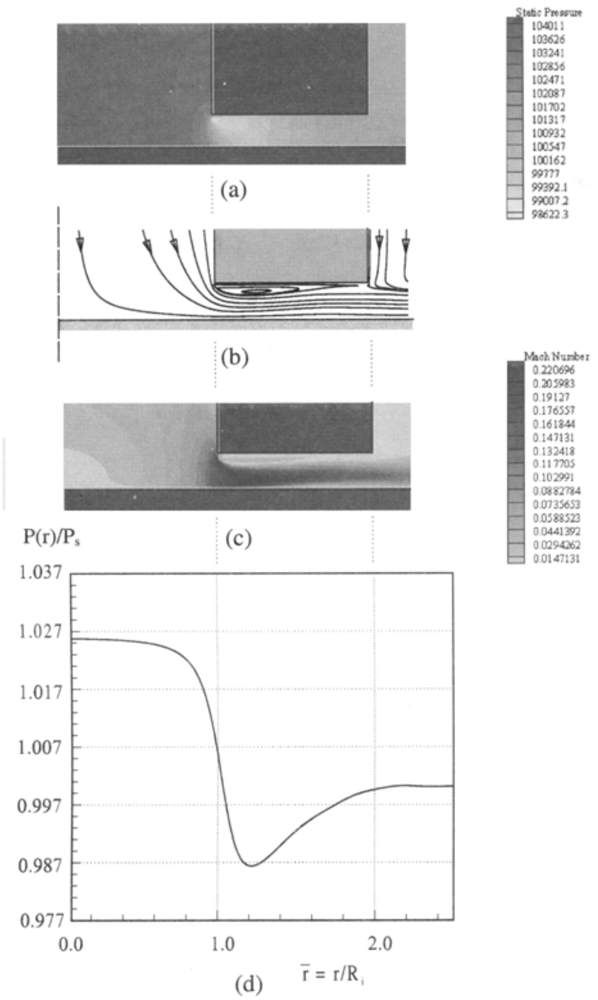

Fig.5 Typical flow behaviour (P_o = 110 kPa)

These results may not be as one may expect as a simple calculation for the extreme case of a one-dimensional isentropic flow would indicate a choked flow situation for approximately $p_o = 1.9 \times 10^5$ Pa. Also, previous work on compressible, laminar flow for similar configurations gave results closer to 2 bars for local Mach numbers to achieve unity for similar configurations[13]. These differences could possibly be attributed to the turbulence model used. Certain authors have stated that the Spalart-Allmaras turbulence model does not seem to be well suited for applications in which high Reynolds numbers are found, although this probably isn't the case here as inlet Reynolds numbers for the presented cases cover a range of Re_d = 2000 to 12000. Fig.6(b) illustrates the inlet total pressure effects on the wall pressure distribution. We can notice that an increase in inlet pressure increases the importance of the negative pressure area as well.

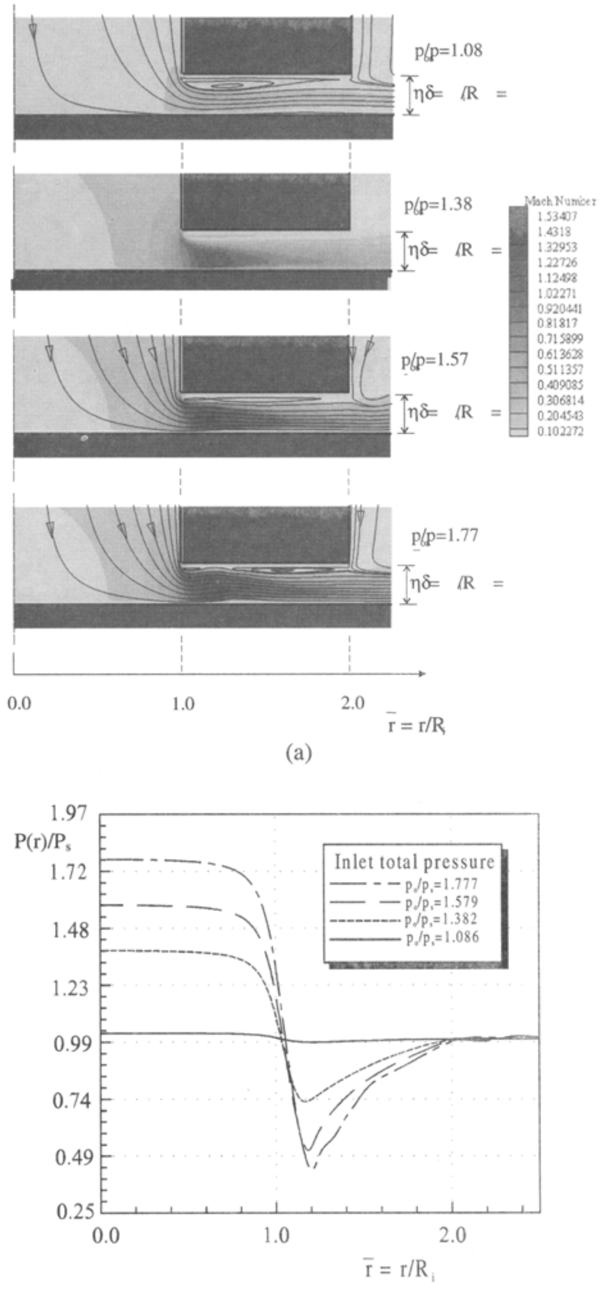

Fig.6 Influence of inlet total pressure on flow streamlines, local Mach number and wall pressure distributions

Result comparison with standard k-ε turbulence model

Fig.7 show comparisons in the general flow contour lines and constant local Mach number floods for two different inlet total pressures between the Spalart-Allmaras (S-A) model and the traditional two-equation k-ε turbulence model. As could be expected, differences exist in the results obtained by these two models. A few general comments can be made. The S-A model indicates areas of separated flow on the nozzle surface while no such areas were found with the standard k-ε model, and this, for both total inlet pressures considered. The

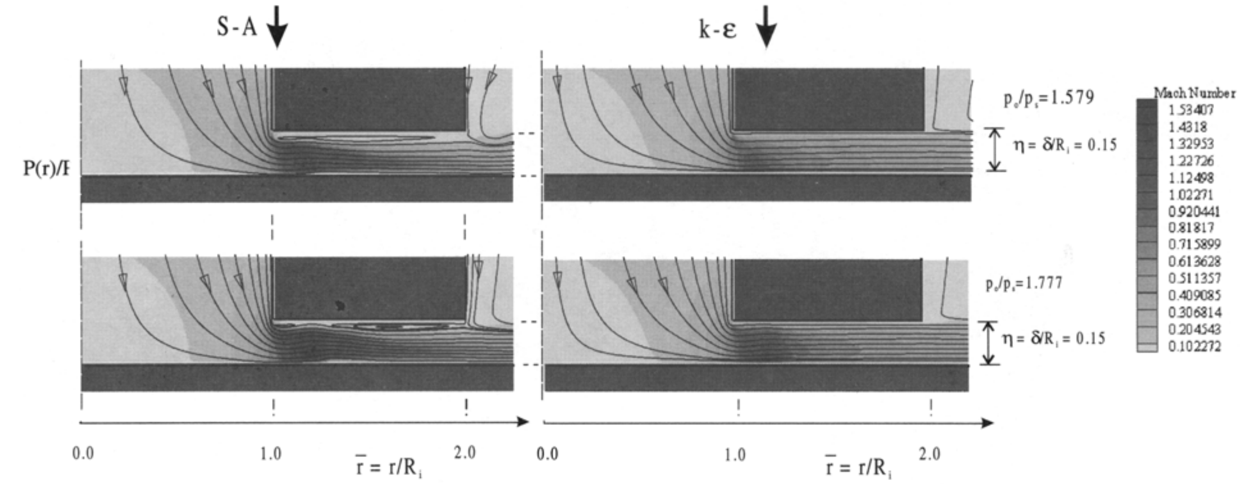

Fig.7 Influence of turbulence model on flow streamlines and local Mach number

accuracy of this model in impinging jet/radial flow between disks applications has also been subject to numerous discussions as this type of flow is not strictly parallel to a wall and has complex features due to entrainment, stagnation and high streamline curvature ([7], [14], for example). Also notable, the local Mach number achieves unity at considerably larger inlet pressures for the k-ε model when compared to the S-A model. Differences are also found in the wall pressure distributions, Fig.8. These clearly show that the minimum pressure is more important with the SA model, thus indicating a greater flow acceleration in the area between the nozzle and the flat plate. Furthermore, the mass flow rates obtained with the SA model were greater than those found with the k-ε model (for the same pressure conditions at the inlet and outlet sections). No concrete comparisons with experimental results were possible at the present time, further work on turbulence modelling in this type of application is required.

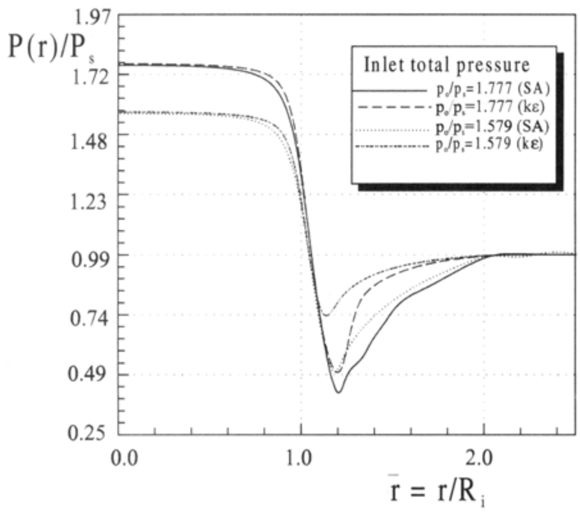

Fig.8 Turbulence model comparisons on wall pressure

Conclusion and Related Future Work

Preliminary results of an investigation of the turbulent radial compressible flow between an injection nozzle and a flat plate has been presented in this paper. Results have confirmed the existence of a separated flow area on the surface of the frontal area of the nozzle when using the Spalart-Allmaras turbulence model. This recirculation zone is the probable cause of injection nozzle fouling found in industrial applications. Further research is required to determine the use of turbulence models in these types of applications.

Acknowledgements

The authors of the present paper wish to thank the Université de Moncton for their financial support.

References

[1] Moller, P S. Radial Flow Without Swirl Between Parallel Discs. The Aeronautical Quaterly, 1963. 163—186

[2] Ishizawa, S, Watanabe, T, Takahashi, K. Unsteady Viscous Flow Between Parallel Disks With a Time-Varying Gap Width and a Central Fluid Source. J. Fluids Eng., 1987, 109: 395—402

[3] Prakash, C, Powle, U S, Suryanarayana, N V. Analysis of Laminar Radially Outward Flow and Heat Transfer Between a Stationary and a Rotating Disk. AIAA, 1984

[4] Bettahar, A. Application des écoulements radiaux à la métrologie pneumatique dimensionnelle: [Ph.D. Thesis]. France: Université de Valenciennes, 1993

[5] Crnojevic, C, Roy, G, Florent, P, et al. Influence of Regulator Diameter and Injection Nozzle Geometry on Flow Structure in Pneumatic Dimensional Control Systems. J. of Fluids Eng., 1997, 119(3)

[6] Lytle, D, Webb, B W. Air Jet Impingement Heat Transfer at Low Nozzle-Plate Spacings. Int. J. Heat Mass Transfer,

1994, 17(12): 1687—1697

[7] Behnia, M, Parneix, S, Shabany, Y, et al. Numerical Study of Turbulent Heat Transfer in Confined and Unconfined Impinging Jets. Int. J. of Heat and Fluid Flow, 1999, 20(1): 1—9

[8] Wattebot, L. L'amplification Pneumatique. Principe. Théorie. Journal de Mécanique, 1937. 70—72

[9] Spalart, P R, Allmaras, S R. A One-equation Turbulence Model for Aerodynamic Flows. La Recherché Aérospatiale, 1994, (1): 5—21

[10] Kumar, V, Pavithran, S. Turbulent Flow in Two-dimensional U-Ducts: A Computational Study. PVP-Vol. 396, Emerging Technologies in Fluids, Structures and Fluid/Structure Interactions, ASME, 1999. 65—78

[11] Houzeaux, G, Codina, R. A Finite Element Method for the Solution of Incompressible Flows in Rotodynamic Machines. PVP-Vol.397-2, Computational Technologies for Fluid / Thermal / Structural / Chemical Systems with Industrial Applications-Vol.II, ASME, 1999. 165—172

[12] Patankar, S V. Numerical Heat Transfer and Fluid Flow. McGraw-Hill, New York, 1980

[13] Roy, G, Vo-Ngoc, D. Étude de L'écoulement Radial Compressible Entre une Buse de Soufflage et une Plaque Plane. Proc. 18th CANCAM, 2001, 1: 125—126

[14] Craft, T, Graham, L, Launder, B. Impinjing Jet Studies for Turbulence Model Assessment – II. An Examination of the Performance of Four Turbulence Models. Int. J. Heat Mass Transfer, 1993, 36: 2685—2697