

Room Air Motion; Effect of Buoyant Air Jets

**Vahab Hassani
Staff Engineer**

&

**Terry Penney
Branch Manager**

**Thermal Engineering & Sciences Branch
National renewable Energy Laboratory
1617 Cole Boulevard
Golden, Co 80401**

ABSTRACT

The velocity and flow pattern characteristics of a small model room were studied under various ventilation arrangements using Fluent. The simulations were performed for three-dimensional cases where the inlet width was varied compared to the width of the room. The computational results were compared to the experimental data gathered from various sources. Comparisons between the velocity and turbulence intensity profiles at various locations inside the room indicate that the predicted results are precise enough for design purposes.

The behavior of buoyant wall jets and their separation from ceiling inside ventilated rooms were also studied. At very high Archimedes numbers buoyancy forces overcome the inertial forces and lead the buoyant wall jet to separate from the ceiling. The separation point of buoyant wall jets with various Archimedes numbers were calculated using Fluent. The separation point predictions were then compared to the experimental results of several researchers and good agreements were observed.

Introduction

The demand for economical use of energy in buildings in conjunction with providing comfortable environment for the occupants has introduced new challenges into the area of heating, ventilating and air conditioning of buildings. Design engineers are facing optimization problems that can not be easily addressed by simple experimental techniques. As a result, increasing attention is being paid into modelling and predicting room air motion and occupants comfort conditions by using computational fluid dynamic techniques.

As an example, cold air distribution systems have been proposed as a method to be used in conjunction with thermal storage systems for commercial building air conditioning. With cold air distribution system, both the supply air temperature and flow rate are lower than the conventional systems for the same cooling load. The cold air is supplied to room at 38 °F to 49 °F instead of the conventional value of 55 °F. Cold air distribution systems can offer greater energy, load and space savings relative to competing systems due to smaller duct and fan size requirements. Although cold air distribution systems show several advantages over the conventional systems, there are some concerns about the impact of cold air jets on thermal comfort of the occupants in a room. Because of very high temperature differences, the cold wall air jet introduced into a room can separate from ceiling and enter the occupied space before adequate mixing has occurred. This can cause cold temperatures as well as high local velocities which in turn can cause discomfort of the occupants. Using CFD techniques, one can readily simulate the behavior of buoyant wall jets in rooms under various conditions and predict their effect on human comfort.

It is estimated that up to 90% of a typical person's time is spent indoors mainly in a commercial or residential building. Increasing awareness in recent years about the indoor air pollutants and their effect on human health has directed a lot of attention towards better understanding of the ventilation systems in buildings. The performance of ventilating systems and the kind of indoor air motion that they create is also important for predicting the level of thermal comfort of the occupants. Specification of the occupants comfort under a given ventilating system requires detailed information about the velocity and temperature distribution prevailing in a building. In most cases it is not possible to conduct experimental studies of the performance of ventilating systems because of the cost and time factors. However, with the recent development in computational fluid dynamics it is possible to carry out numerical simulations of the performance of ventilating systems in short time. Several researchers have applied CFD codes using k- ϵ models to room air motion problems and have obtained encouraging results. The computational techniques have been applied to study the velocity, temperature and turbulence intensity distributions in rooms. However, recently, more attention is being paid to contaminant concentrations and air quality in rooms.

This paper is aimed at testing the capability of the CFD code FLUENT in conducting room air motion simulations. Our main interest is to verify if the precision of the predicted results are adequate for design purposes. In this paper, a set of experimental data obtained by Nielsen et al.[1] will be used to validate the simulation results of this work. The data obtained by Nielsen has been recommended by International Energy Agency [2] for validating different numerical models. Since Nielsen's tests were conducted under isothermal conditions, this work will first carry out simulations under conditions identical to that of Nielsen's. In later sections, the effect of very high temperature difference at discharge will be investigated and simulations will be performed to study the behavior of buoyant wall jets inside a model room. The separation distance of buoyant wall jets will be determined from computational data and the results will be compared to the existing experimental data and analytical predictions.

Room Configuration and Boundary Conditions

The test room arrangement used by Nielsen is depicted in Figure 1. The height of the model test room is 89.3 mm and other dimensions are listed below:

$$\begin{aligned} L/H &= 3.0 \\ W/H &= 1.0 \\ h/H &= 0.056 \\ t/H &= 0.16 \\ w/W &= 0.5 \text{ and } 1.0. \end{aligned}$$

In his tests, Nielsen used two values of the width of the inlet slot to determine the

extend of the influence of a three-dimensional inlet flow on the velocity distribution of the room. The inlet velocity profile is shown in Figure 2. This profile is symmetric about $y=0.5h$ and closely corresponds to a developed turbulent channel flow profile. The inlet velocity profile used by Nielsen is almost uniform in z -direction. This study assumes a uniform velocity and turbulence intensity profiles in z -direction. The inlet turbulence intensity is 0.04 which results in a turbulence kinetic energy of about 0.54. The analysis in this work is mainly conducted with Reynolds numbers of about 5000 and 300. Low Reynolds numbers were selected, so that the behavior of the cold air jets (buoyant ventilating wall jets), characterized by a non-dimensional parameter called Archimedes number (Archimedes number is the ratio of buoyancy to inertia forces), could be simulated. The Archimedes number in this study covered a range from zero for isothermal jets to 0.007.

For isothermal case, adiabatic wall boundary conditions were specified and in non-isothermal case, a uniform heat flux on the floor was specified. The inlet and outlet temperatures, heat flux, inlet average velocity, and inlet Reynolds and Archimedes numbers for different tests are listed in Table 1.

Computational Results

Isothermal Studies

Present computations were carried out using $29 \times 17 \times 12$ grid nodes. The influence of the number of grid nodes on the converged solution was tested by using $55 \times 17 \times 12$ and $29 \times 29 \times 12$ grid nodes. Although in each case a different converged solution was obtained, the discrepancies were less than 3%. This order of uncertainty is likely to persist with any number of grid nodes.

To fulfill the objectives of this work in applying Fluent to room air motion simulation problems, a two step procedure was adopted. In first step, simulation of configurations for which experimental data was available were carried out and comparisons between the experimental data and predictions reported. In the second step, problems of interest to industry were considered for which reliable and accurate experimental data have not been reported.

The first set of computations were carried out for isothermal air jet entering the room at $Re=5000$ with turbulence intensity of 0.04, as listed under Tests #1 in Table 1. The velocity profiles at distances $x/H=1.0$ and 2.0 are presented in Figures 3 and 4. The computational results are compared to the experimental data of Nielsen [1] at different locations inside the room. The comparisons for $x/H=1.0$ and $w/W=1.0$ have been presented in Figure 3 which shows that the agreement between the experiments and predictions is very good. Nielsen [1] presented turbulence intensities only at $x/H=1.0$ and his data are shown in Figure 3. Fluent does not provide turbulence intensities directly but a simple procedure

for calculating them is to use the kinetic energy profiles and assume homogeneous turbulence intensity in all directions. This assumption results in

$$u' = \sqrt{\frac{2}{3}kE} \quad (1)$$

where velocity fluctuations u' are calculated by post processing kinetic energy profiles at different x/H locations. Figure 3 shows the comparison between the results of this work and the measured turbulence intensities by Nielsen (1978). The results seem to be acceptable to within the experimental errors. The results for $x/H=2.0$ is shown in Figure 4. The experimental and computational results compare reasonably well. The data for $z/W=0.4$ is also shown in this figure. The overall velocity distribution for Test #1 is shown in Figure 5.

The configuration used for Test #2 was the same as that used for Test #1, except that the ratio w/W was set equal to 0.5 instead of 1.0. In other word, for Test #2, the inlet was half as wide as that used for Test #1. The inlet velocity profile was as shown in Figure 2 with no variations in z -direction.

The velocity profile inside the room are compared to the experimental data of Nielsen in Figures 6 and 7. These figures show that there are some discrepancies between the measured and predicted data. The differences can be attributed to the coarser mesh that was chosen for this particular test. The authors are convinced that by increasing the number of grids in y and z directions, better agreements can be achieved because Figures 6 and 7 indicate that the overall trend of the predicted and experimental results are very close, and Figures 3 and 4 of Test #1 present an indication of this fact where larger number of grid points were used.

Test #4 for $w/W=1.0$ configuration was carried out with an average inlet velocity of 1 m/s. The inlet velocity profile was kept similar to the inlet velocity profile of Test #1, as shown in Figure 2. This test was conducted for later comparison with identical non-isothermal test the results of which will be reported and discussed in the next section.

Non-isothermal studies

In the second step of this study, non-isothermal ventilating air jets were studied. Air at very low temperatures were introduced into a room with uniform floor heating and the effect of the buoyant ventilating air jets on the overall room air motion and the possibility of detecting any early separation of the buoyant wall jets from the ceiling were studied.

Separation of buoyant wall jets occur when the buoyancy forces are slightly larger than the inertia forces. Finding the separation point of buoyant wall jets is

important because it determines if the performance of a given diffuser is acceptable under 38 °F to 49 °F discharge temperatures. It is therefore the objective of this section to find the separation point of buoyant wall jets and relate that to jet's discharge conditions. As a result of this study, the impact of cold air diffusion on room air motion and temperature distribution can be evaluated.

The important forces effecting a buoyant air jet are buoyancy forces caused by density differences and momentum or inertia forces. The relationship between these forces is well characterized by a non-dimensional number called Archimedes number which is the ratio of buoyancy to inertia forces as given below:

$$Ar = \frac{gh\beta\Delta T_o}{u_o^2} = \frac{Gr}{Re^2} \quad (2)$$

where h is the height of the inlet and ΔT_o is the temperature difference between the inlet and outlet air.

Test #3 was conducted to study the effect of discharge temperature on room air flow pattern. The inlet velocity profile was kept similar to Test #1 and a uniform heat flux of $q=10,000 \text{ W/m}^2$ for the floor and $q=0$ for the ceiling and the remaining walls were specified. A uniform inlet temperature $T_i=278 \text{ K}$ was specified for this case and the resulting room outlet temperature after convergence was $T_o=305 \text{ K}$. The velocity profile for this case at $x/H=1.0$ is presented in Figure 3. In general, for a $\Delta T=27^\circ\text{C}$, there does not seem to be a noticeable difference between the velocity profiles of an isothermal and non-isothermal case as shown in Figure 3. Note that Reynolds number for both cases was equal to 5000 and the Archimedes number for Test #3 was 2×10^{-5} . The comparison between Tests #1 and #3 show that the inertia forces in Test #3 are strong enough that the buoyancy forces have little or no effect on the velocity distribution of the room.

To introduce and study the effect of much stronger buoyancy forces on the flow pattern and possible separation of the jet from ceiling, test #5 was carried out where the average inlet velocity was reduced to 1 m/s while the inlet velocity profile was kept similar to the profile shown in Figure 2. Test #5 was conducted under conditions similar to that of Test #4 except that test #5 was non-isothermal. Reducing the inlet velocity and increasing the temperature difference resulted in lower Reynolds numbers and higher Archimedes numbers meaning that the effect of buoyancy forces were increased. A uniform temperature of 278 K. was specified for inlet air, and a uniform heat flux of 1000 W/m^2 was assigned to the floor with the ceiling and all the other walls set at $q=0$. For more information about the test conditions refer to Table 1. Figures 8 and 9 show the non-dimensional velocity and turbulence intensity for tests #4 and #5. Comparison between velocity profiles for these two tests at $x/H=1.0$ and $x/H=2.0$ show a significant change in the air flow pattern especially in the bottom half of the room (this region is referred to as "occupied region"). However, the turbulence intensities for both cases are very close. A detailed analysis of velocity

distribution for this case shows that a flow reversal occurs at $x=0.225$ and that is where the cold air jet separates from the ceiling.

The mean velocity profiles presented in previous figures indicate the wall-jet behavior of the flow down-stream of the inlet. To further verify this behavior, the growth rate of the wall jets tested in this study was compared to the idealized wall jet configuration obtained by Scharz and Cosart [3]. Figure 10 shows the maximum velocity decay for several configurations used in this study. The isothermal and non-isothermal case with $Re=5000$ agree very well with the corresponding predictions of Schwarz and Cosart [3] for that Reynolds number. Figure 10 shows a decay coefficient $K=10$ for jets with $Re=5000$ and $K=5$ for jets with $Re=300$. The test cases with $Re=300$ show faster decay because of lower initial momentum. The isothermal case of test #4 shows even faster decay compared to the non-isothermal case of Test #5. It is expected that a non-isothermal jet should show faster decay than a similar isothermal-jet because of larger entrainment of warmer room air into the cold air jet. However, in the simulations presented here, the non-isothermal jet shows lower decay and higher peaks compared to a similar isothermal jet. One major reason for this observation is that the mass continuity is satisfied in all control volumes in the solution domain. Therefore in non-isothermal case because of lower density levels, higher velocities are obtained to fulfill the continuity conditions.

The Archimedes number for Test #5 was $Ar=0.0067$. Tests similar to Test #5 but at different discharge velocity and temperature difference were conducted, the results of which are not presented here. However, for each case the separation point of the buoyant wall jet was specified and the Archimedes number at those conditions were calculated. Having the decay coefficient K and air inlet width h , the separation point data were plotted vs. Archimedes number in Figure 11. This figure also shows the experimental data gathered from several sources where the researchers have reported the separation point results for tests that were conducted in actual size rooms. The experimental data of Kubota [4] were conducted in a small scale test facility and his data were gathered for a separation point of a heated floor jet instead of a cooled ceiling jet. Presented in Figure 11 is also analytical prediction of separation point that was carried out by Hassani and Kirkpatrick [5]. The agreement between the analytical and computational predictions is very good and they both compare very well with the experimental data gathered from other sources.

Conclusions

The most important conclusion of this study is that by proper choice of number of grids in all three directions, $k-\epsilon$ turbulent model is capable of predicting the velocity characteristics of a ventilated room. Comparison between the experimental data and the predictions show that the precision of the results is adequate for design purposes.

Other solution techniques and modeling of room air turbulence might be necessary if the flow and thermal boundary conditions are very complex. There are concerns about proper modeling and simulation of complex inlet diffusers. This study did not devote any time to address this issue but our future work should investigate Fluent's capability in simulating complex inlet diffusers.

Our predictions of general room air motion and separation of cold air jets from ceiling compared reasonably well with the experimental data that were available to us. However, for more accurate results, the number of grid nodes should be increased by at least twice as much as those used in this study. The number of grid nodes become especially important when the local heat loads that are the source of buoyant plumes are included in the solution domain.

Our future study will apply Fluent to real life problems in heating, ventilating and air-conditioning. The temperature and velocity fields for various configurations will be generated from which comfort data in the occupied zone will be obtained. The effect of local loads on the overall room air motion will be specified.

Table 1. Test Conditions For Room Air Motion Simulations

Test#	u_o , m/s	T_i , °K	T_o , °K	floor heat flux, W/m ²	Re	Ar
1	15	isoth.	isoth.	0.0	5000	0
2	15	isoth.	isoth.	0.0	5000	0
3	15	278	305	10^4	5000	2×10^{-5}
4	1	isoth.	isoth.	0.0	300	0
5	1	278	322	10^3	300	0.0067

REFERENCES

1. P.V. Nielsen, A. Restivo, and J.H. Whitelaw, The Velocity Characteristics of Ventilated Rooms, *J. Fluids Engineering*, vol. 100, pp. 291-298, 1978
2. International Energy Agency, Air Flow Pattern within Buildings. Flow Flash, No. 1., Int. Energy Agency Annex 20, 1989.
3. W.H. Schwarz, and W.P. Cosart, The Two-Dimensional Turbulent Wall Jet, *Journal of Fluid Mechanics*, Vol.10, pp.481, 1961.
4. H. Kubota, A basic Study on The Room Air Distribution, personal communication, 1991.
5. A.V. Hassani, and A. Kirkpatrick, Cold Air Diffuser Selection and Design Guideline, Report prepared for Electric Power Research Institute, 1992.
6. P.V. Nielsen, and A.T.A. Moller, Measurements on Buoyant Jet Flows from a Ceiling-Mounted Slot Diffuser, Proceedings of the 3rd Seminar on Application of Fluid Mechanics in Environmental Protection, Gliwice, Poland, Silesian Technical University, 1988.

BA-G0944901

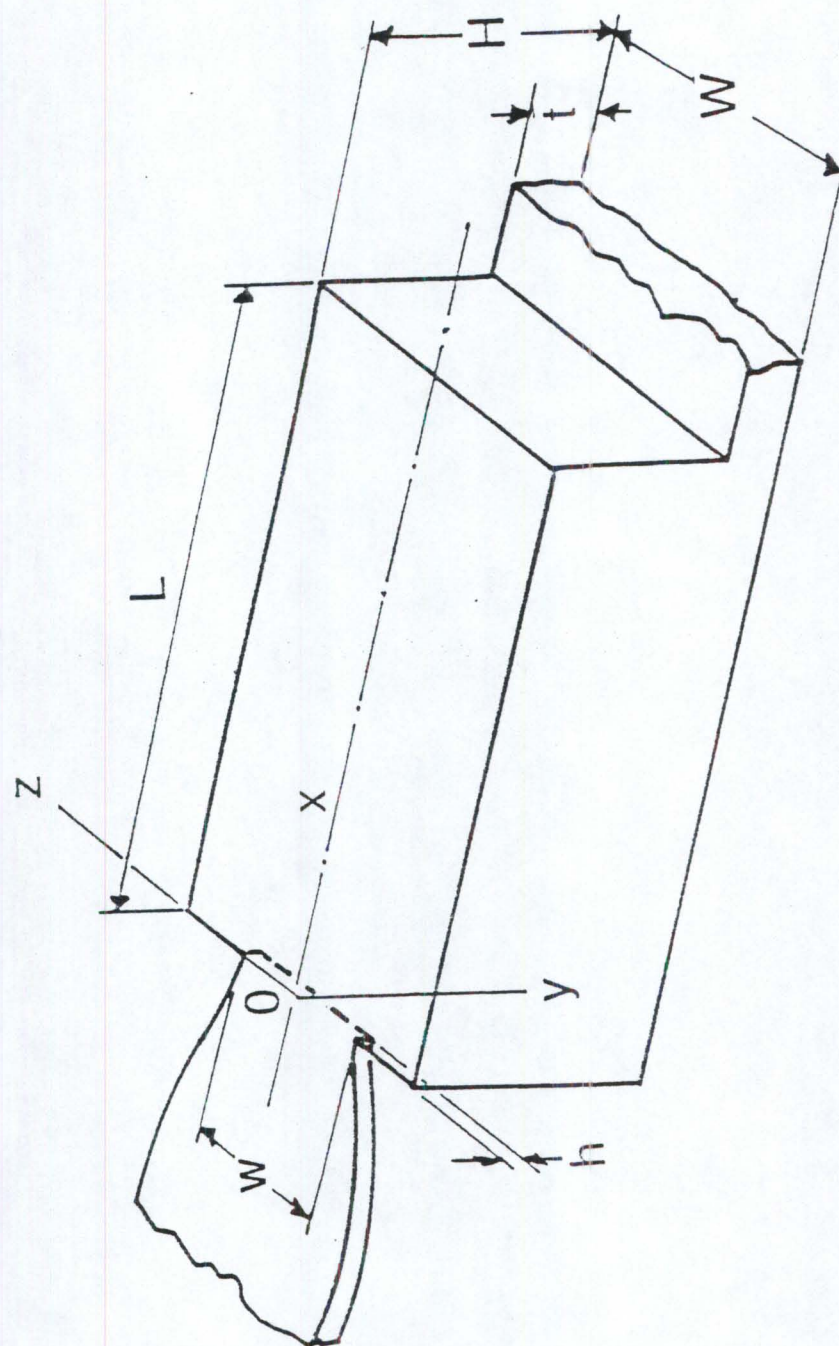


Fig.1 Test room configuration and coordinate system

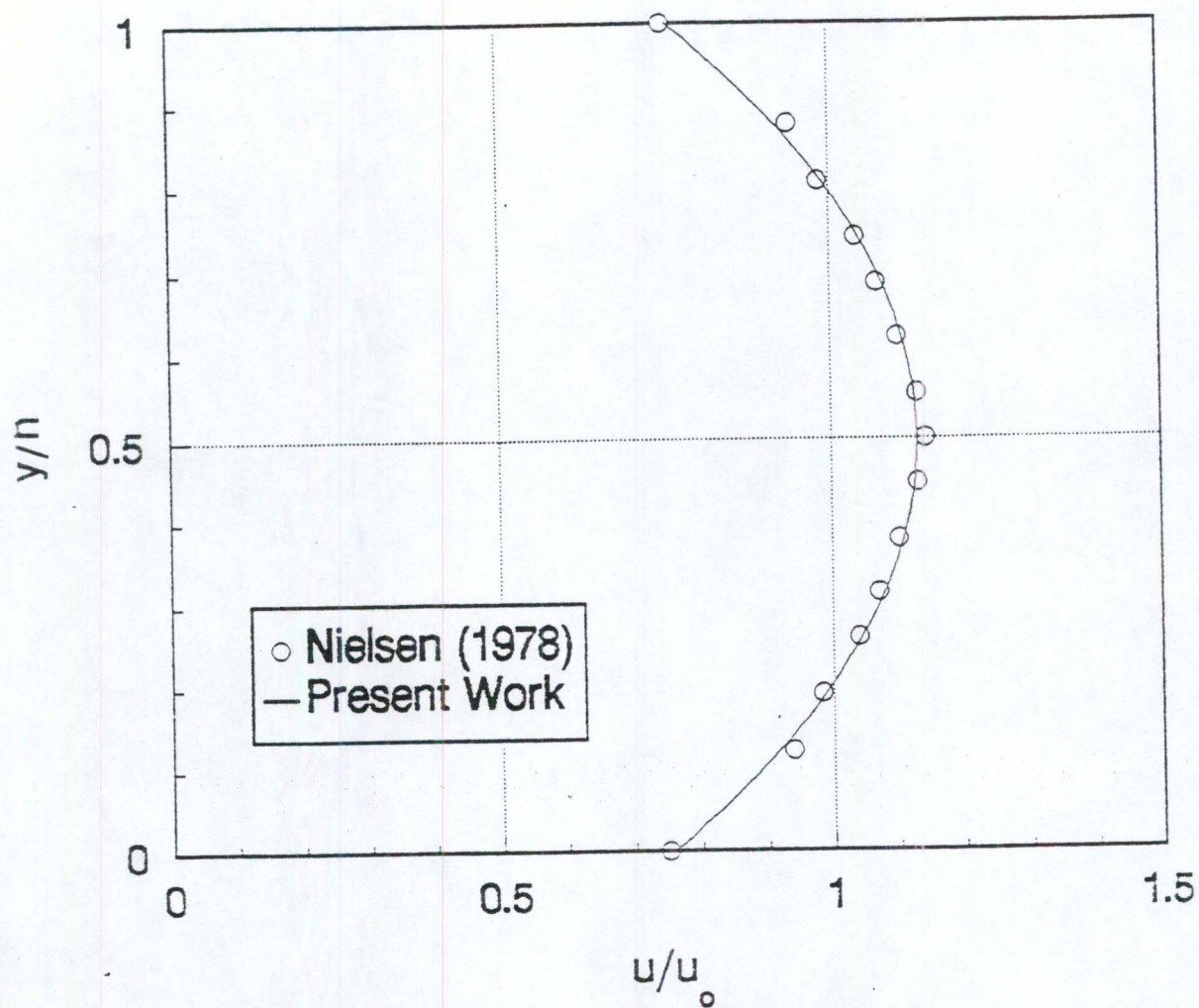


Fig.2 Inlet velocity profile

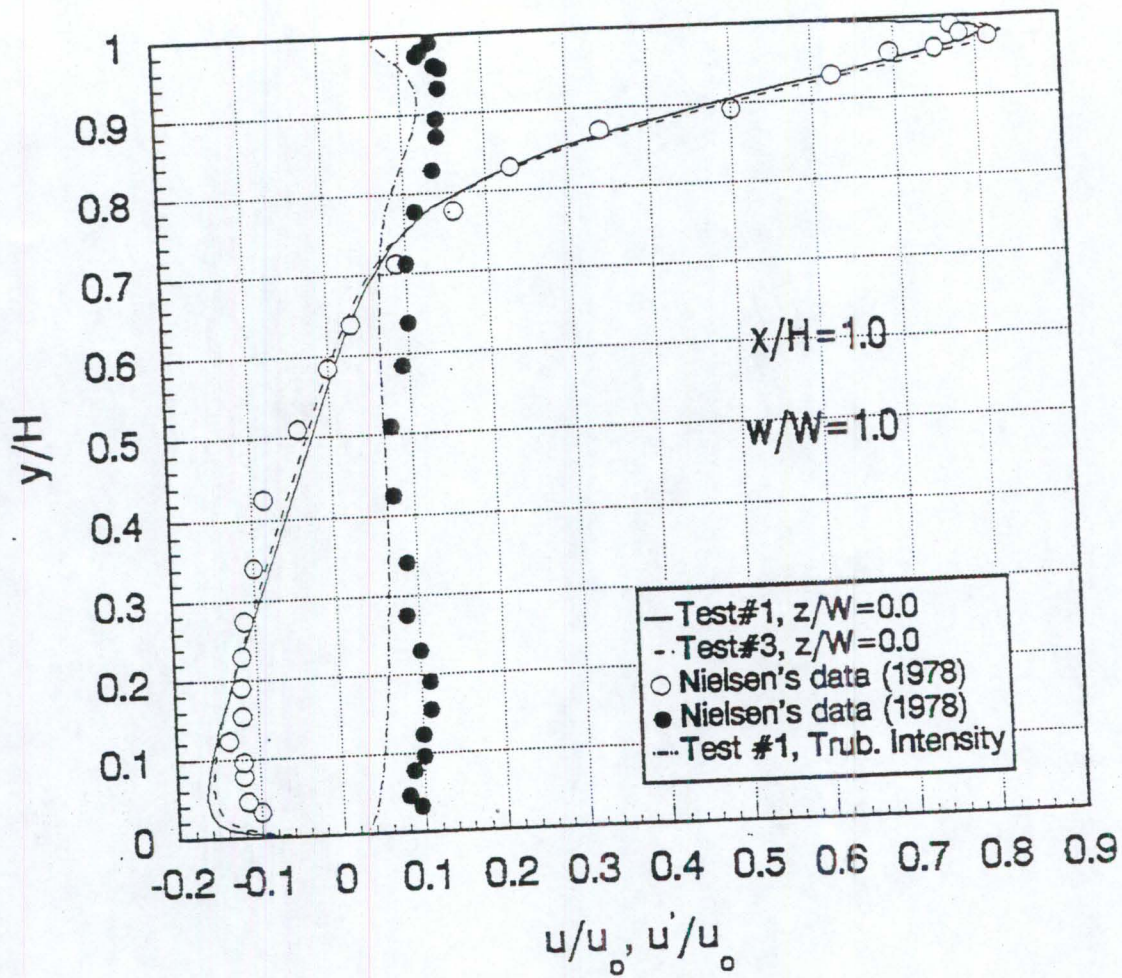


Fig.3 Measured and computed velocity and turbulence intensity profiles for $x/H=1.0$, $w/W=1.0$ and $Re=5000$

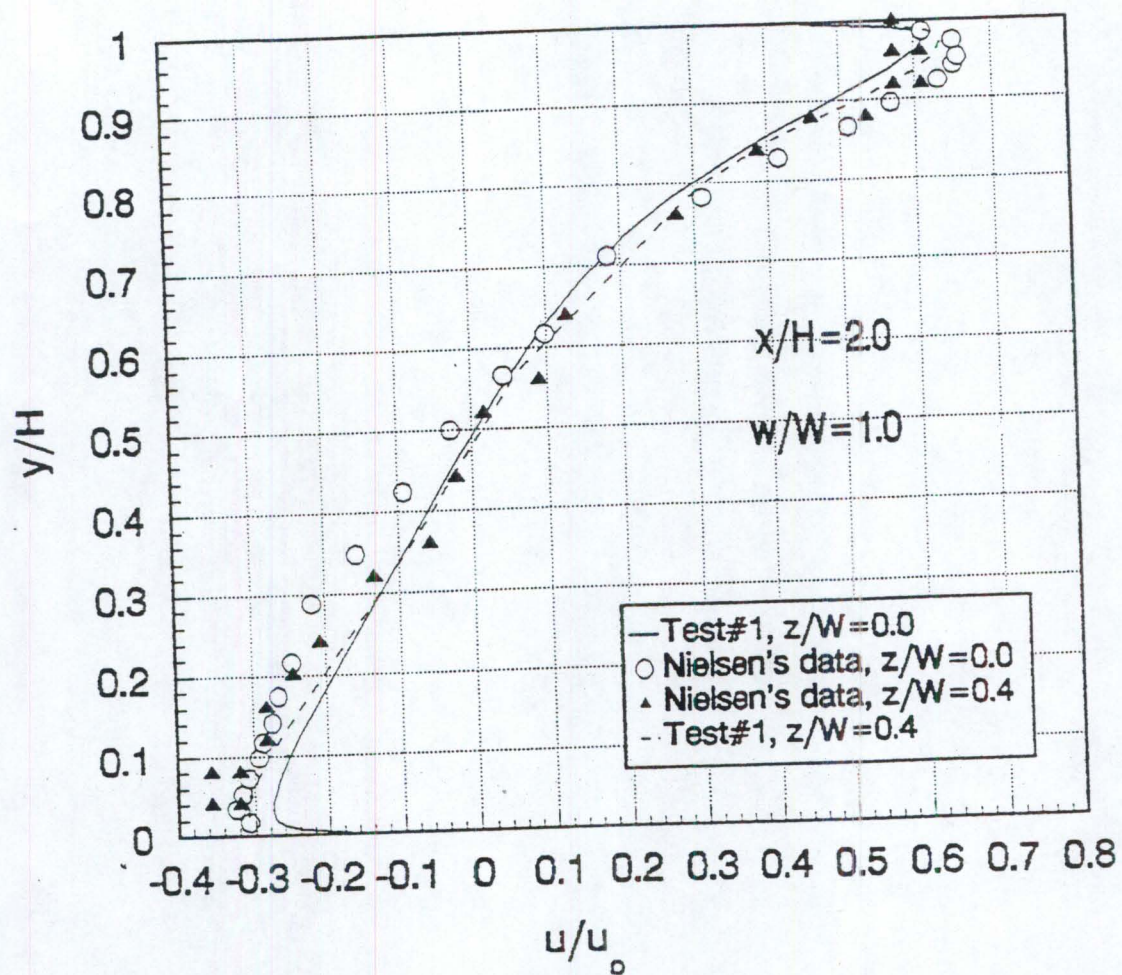
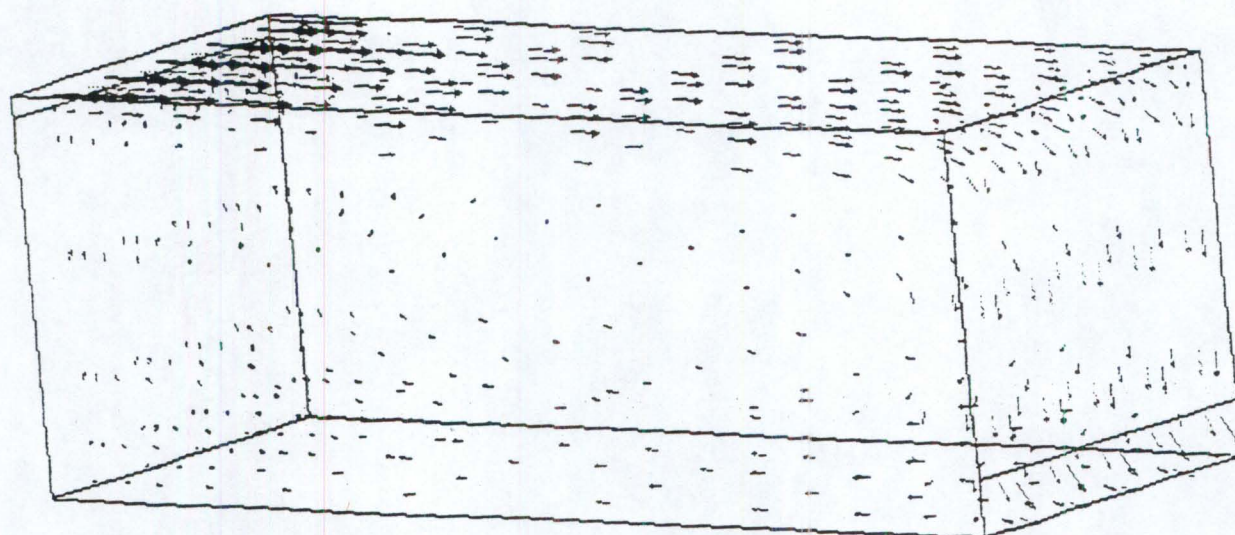


Fig.4 Measured and computed velocity profiles for $x/H=2.0$, $w/W=1.0$ and $Re=5000$

01
01
01
01
01
01-01
-01
-01
-01-00
-00
-00-00
-00
-00-00
-00
-00+00
+00
+00+00
+00
+00+00
+00
-01
-01

x



VELOCITY VECTORS (METRES/SEC)

U = 1.69352E+01

ORIENT = X

3-D DOMAIN

FLUENT
CREARE.X. INC.Fig.5 Velocity vectors for $w/W=1.0$ and $Re=5000$

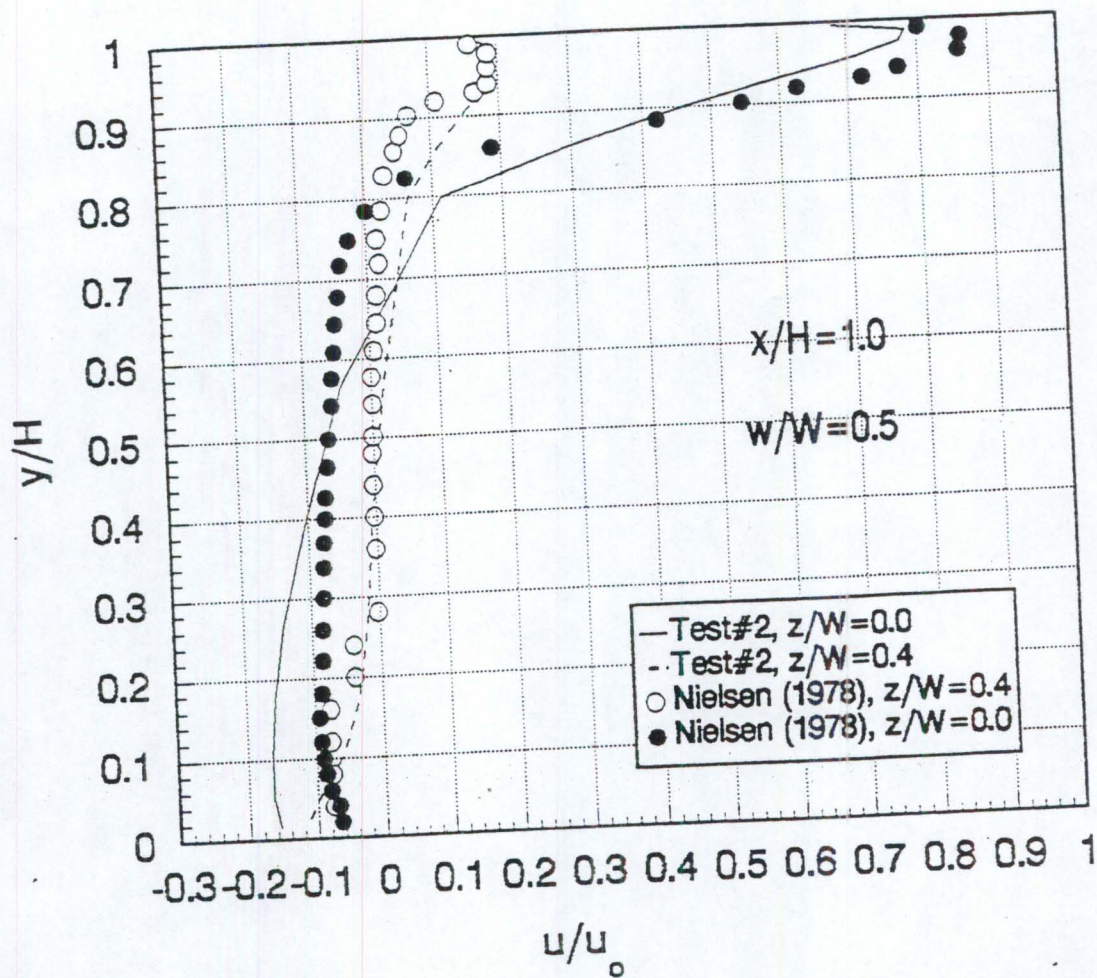


Fig.6 Measured and computed velocity profiles for $x/H=1.0$, $w/W=0.5$ and $Re=5000$

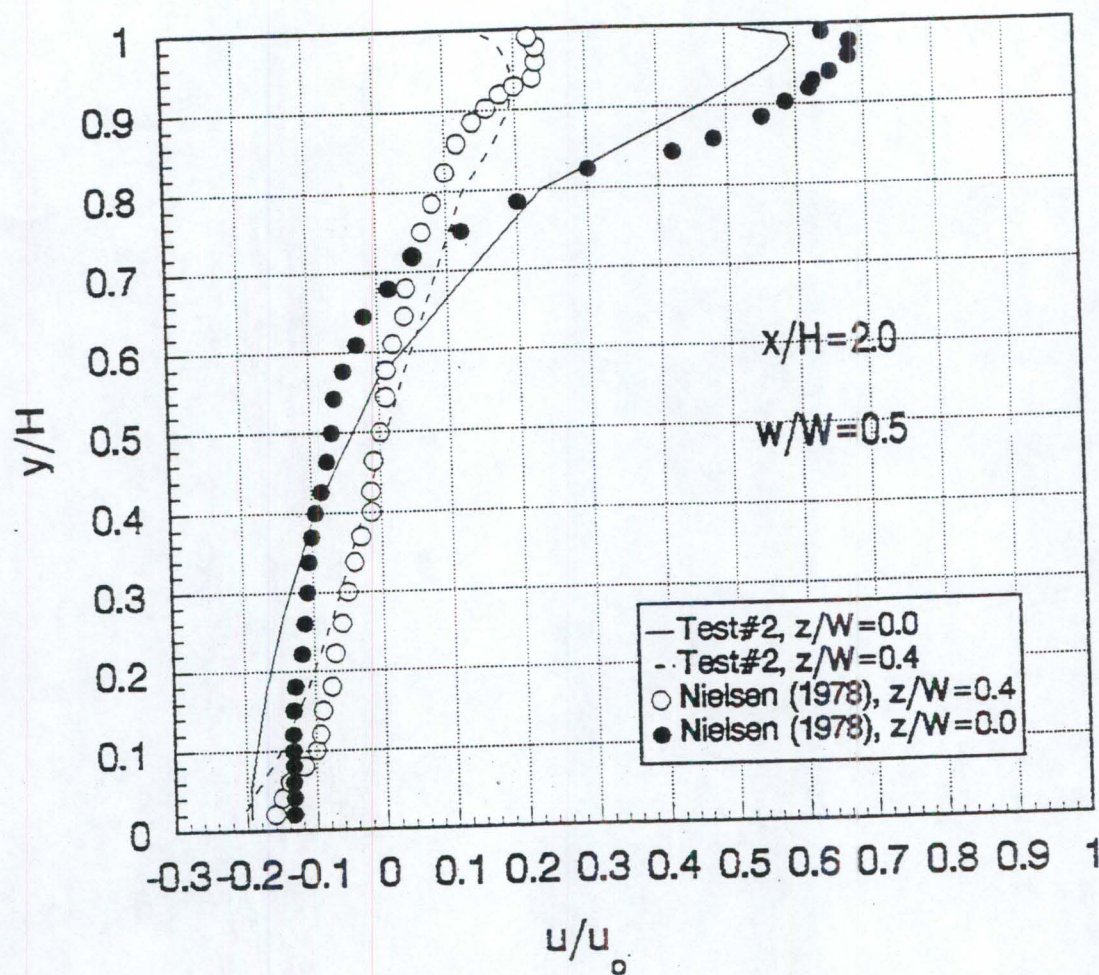


Fig.7 Measured and computed velocity profiles for $x/H=2.0$, $w/W=1.0$ and $Re=5000$

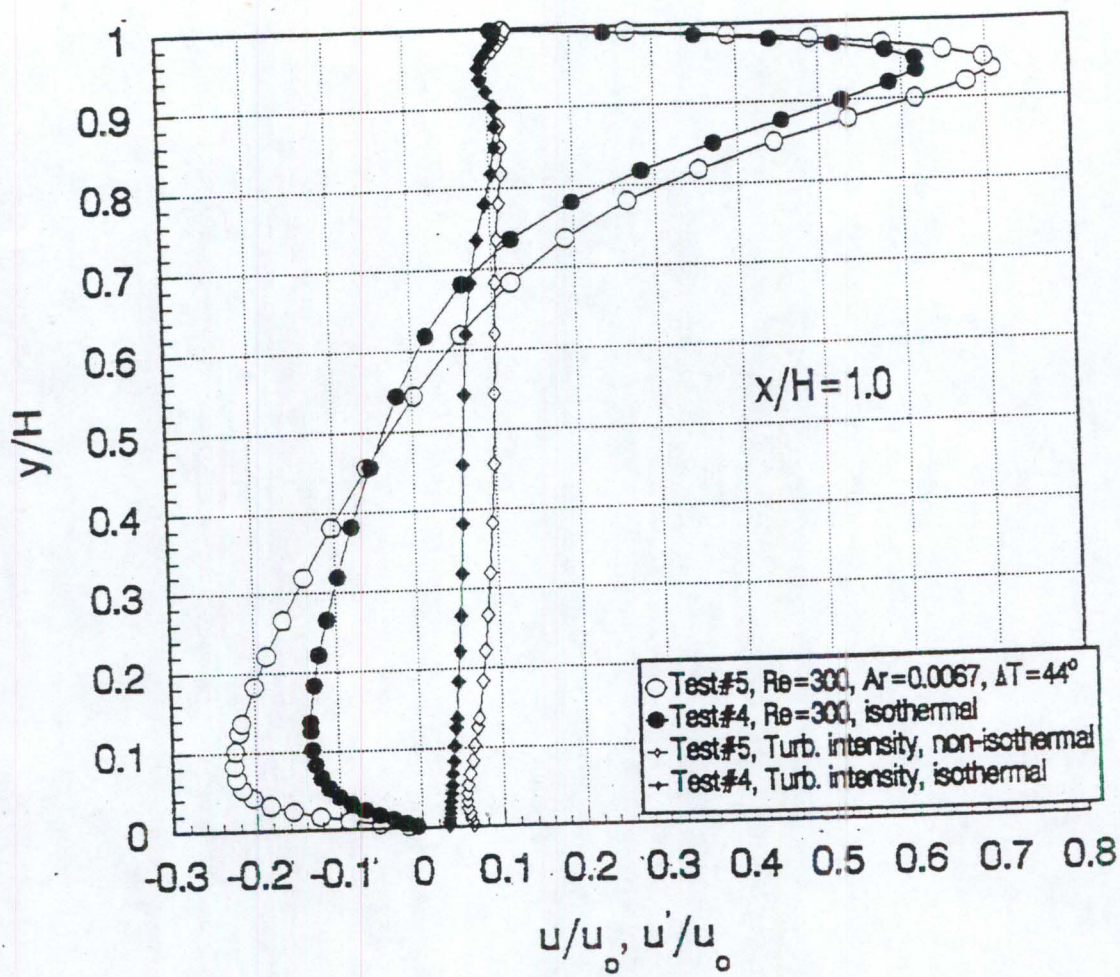


Fig.8 Computed velocity and turbulence intensity profiles for both isothermal and non-isothermal tests at $x/H=1.0$, and $Re=300$

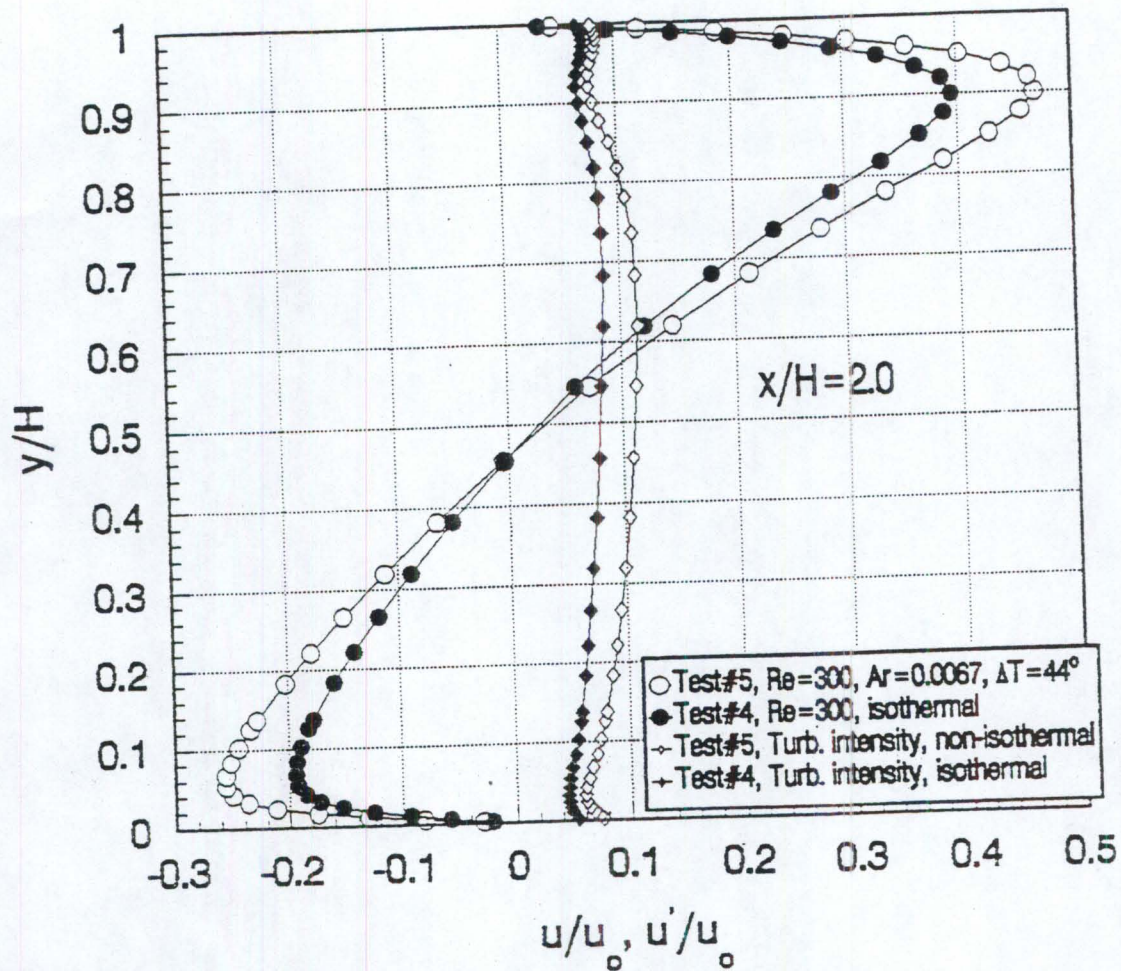


Fig.9 Computed velocity and turbulence intensity profiles for both isothermal and non-isothermal tests at $x/H=2.0$, and $Re=300$

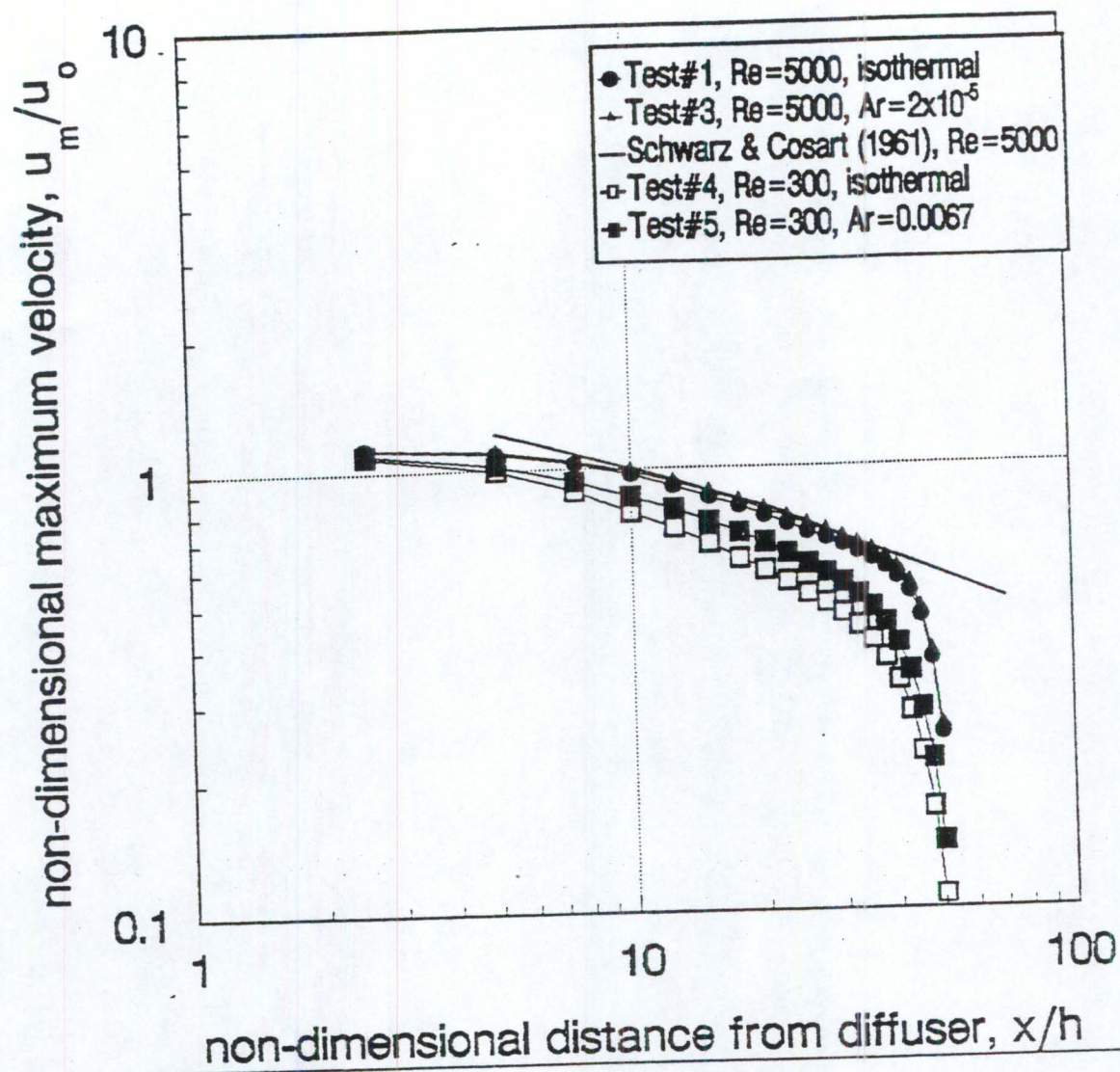


Fig.10 Computed and predicted peak velocities in jet, for Re=5000 and Re=300

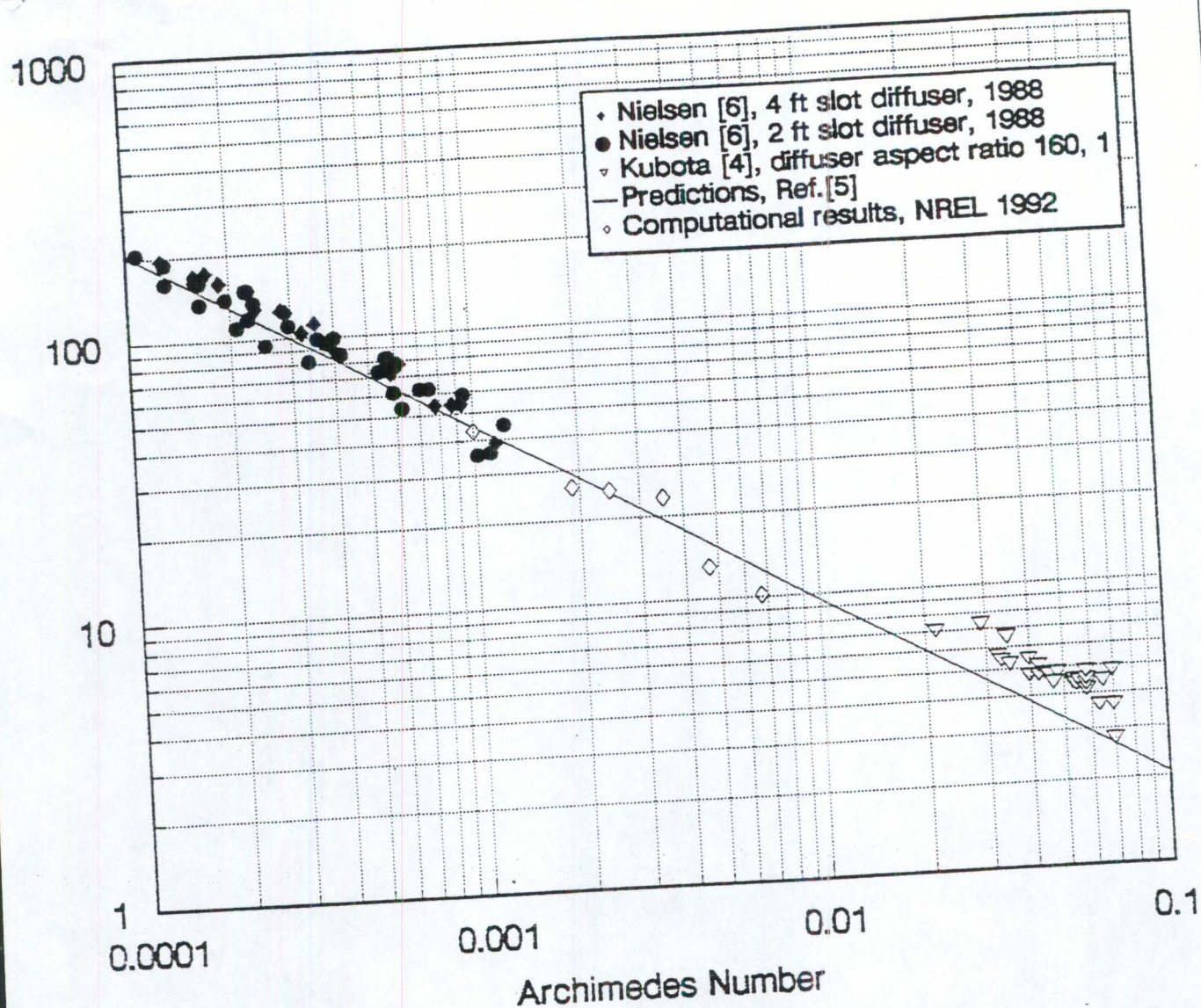


Fig.11 Distance between the separation point and discharge vs. discharge Archimedes number