Mansoura Bulletin Vol. 11, No. 2, December 1986 M. 1

PREDICTION OF LAMINAR CONVECTIVE HEAT TRANSFER IN A NON-CIRCULAR DUCT

By

A. El-Hadik, M. Mahgoub, and M. Shalaby
Faculty of Engineering
El-Mansoura University

ABSTRACT

This paper is concerned with the investigation of heat and fluid flow in a straight non-circular duct, with special interest to prediction of heat transfer from the fuel rods in a pressurized water reactor during a postulted loss of coolant accident. Theoretical solution for the fully developed laminar flow in cusped ducts is described. Variables to be solved for are the temperature and velocity distribution in the plane of the duct cross-section. A standard finite difference numerical procedure is used and integration is effected using a point to point iteration method. The theoretical solution is found to provide realistic predections when compared with experimentally obtained results.

INTRODUCTION

Fully developed flow and heat transfer in a straight non-circular duct of constant cross sectional area has important practical applications in the field of thermal engineering. As a result of the complexity and non-linearity of the momentum equations, no general solutions are known which simultaneously include proper allowences for all the physical effects included therein. Two basically different techniques have been used in exploiting the physical information succinctly contained in the Navier-Stokes equations. One method, the one most recently employed, is direct numerical analysis in the manner used in (1,2). The other technique involves separating flow processes into kinds or regimes in which some simpler form of Navier-Stockes equations applies (3).

In many cases, the laminar flow problems are amenable to analytical solutions (4). The turbulent flow case is considerably more complex and necessitates modelling of the turbulence driven secondary flows which exist in the plane of the duct cross-section.

During a postulated loss of coolant accident, the cladding material, is subjected to high temperatures and pressure difference (5,6). The combined effect of such high temperature and pressure difference may lead to local ballooning or swelling of the cladding, and in the extreme case the caldding of the adjacent fuel rods comes in contact to each other. In such case the cooling process will not be efficient when the emergency cooling system is put in operation.

The above mentioned geometry shape is idealized as shown in Fig. 2. The case of turbulent flow in such a 4 cusp duct has been studied in reference (7) where attention is focussed on the development of the numerical technique and the modelling of the turbulence characteristics and secondary flows.

In this work a theoritical model is proposed to solve the problem of laminar flow in such 4-cusp duct. To examine the validation of the theoritical code, a test rig was constructed to simulate the situation in a nuclear reactor (partly).

EXPERIMENTAL APPARATUS

The experiments for studying the fluid flow and heat transfer for water flowing in an idealized geometry of a blocked channel of a pressurized water reactor utilized the open-loop system are shown in Fig. 1. City water enters the system through a needle flow meter (1) and is ducted to a large upstream plenum (3) through the control valve (2). The 4-cusp test section (6) spans between the plenum chamber and a smaller downstream plenum (9). The purpose of the plenum chambers was to provide a well defined abrupt inlet and exit for the test section. Two plastic tubes (4) are used to connect the test section duct in the system. The two plastic tubes are used to prevent the back conduction heat transfer at the test section ends. The water leaves the downstream plenum through the riser (10) and empties into the drain (12).

Test Section:

The duct is formed by four stainless steel tubes, which shown in Fig. 2. Each tube is 12mm outside diameter and 1mm wall thickness and 4m long. Heating of the duct wall was effected using a welding rectifier unit type MCRA 900 with maximum current of 900 amperes and maximum voltage of 65 volts. The heat flux in all tests was less than 6000 w/m^2 which corresponds to the rate of energy release several hours after reactor shutdown (5). The electrical power was supplied to the test duct through two busbar connections (5). Digital multi-meter was used to sense the voltage drop along the test duct with devision of 0.01 volte and a D.C. ameter is used to measure the current passing through the duct cross-section.

All tests were performed at atmospheric pressure. The test section was insulated with glass wool (7) mating to minimize the external surface thermal losses. The test duct as well as its thermal insulation and electrical connections were contained in a wooden box (8). In order to asses the magnitude of heat losses, the apparatus was operated at low power without coolant flow and the wall temperatures were measured. These data were then used to correct the value of input power in normal operation.

Temperature Measurements:

Local temperatures of the wall surface were sensed in five points by copper-constantan thermocouples (0.15 mm diameter) as shown in Fig. 3. Two thermocouples were also soldered to the inner surface of the cusped duct at two locations 3.0 m and 3.5 m downstream from the channel entrance to determine out the axial temperature gradient dT/dz. All the used thermocouples were connected to a 24-point self switching temperature recorder, having a full scale of 200 C. No significant effect was noted indicating that natural convection effects were negligable. This fact was also confirmed by using the criterion

$$(Gr/Re^3) < 0.002$$

proposed by Heiber (8).

It is estimated that the total experimental error in measuring the heat transfer rate may rise to about 7%.

THEORETICAL ANALYSIS

In the analysis of the considered cusped duct, the flow and heat transfer are assumed to be fully developed. Due to symmetry, the analysis is restricted to the two dimensional region AOB shown in Fig. 4, in which the axial momentum and energy equations are solved under the following assumptions:

- i- The flow is fully developed laminar, unidirectional, and steady,
- ii- The flow is two-dimensional recirculating, and
- iii- The fluid is incompressible and has constant viscosity

Using the polar cylinderical cooerdinates (r, θ, z) as shown in Fig. 4, it is possible to describe the steady state flow by the generalized form of Gossman et al. 1 which is given by

$$a \frac{\partial}{\partial r} \left(\phi . \frac{\partial \psi}{\partial \theta} \right) - \frac{\partial}{\partial \theta} \left(\phi . \frac{\partial \psi}{\partial r} \right) - \frac{\partial}{\partial r} b.r. \frac{\partial}{\partial r} (c\, \phi)$$

$$- \frac{\partial}{\partial \theta} (b/r). \frac{\partial}{\partial \theta} (c. \phi) + d.r = 0 \qquad (1),$$

where ϕ is a parameter which stands for T and U, and Ψ is the stream function, while the coefficients a,b,c and d are given in the following table:

ϕ \ Coefficient	a	b	c	Source term "d"
U	0	ν	1	$(1/\rho)(\partial p/\partial z)$
T	0	ν/σ_h	1	$(U/\rho c_p).(\partial T/\partial z)$

Substituting for the values in the table one gets the following basic equations:

$$(r/\rho)(\partial p/\partial z) - \frac{\partial}{\partial r}\,\nu . r\,(\partial U/\partial r) - (\partial/\partial\theta).\,(\frac{\nu}{r}\,.\,\frac{\partial U}{\partial\theta}) = 0 \quad (2)$$

Energy equation:

$$ru(\partial T/\partial z) - \frac{\partial}{\partial r}\,\alpha\, r(\partial T/\partial r) - \frac{\partial}{\partial\theta}\;\frac{\alpha}{r}\,(\partial T/\partial\theta) = 0 \quad (3)$$

where $\partial T/\partial z$ = constant, obtained from the overall heat balance over the axial length dz or experimentally.

The boundary conditions of the problem can be stated as follows (Fig. 5);

At the symmetry line OA $\partial U/\partial\theta = 0$ and $\partial T/\partial\theta = 0$

At the symmetry line OB, $\partial U/\partial\theta = 0$ and $\partial T/\partial\theta = 0$

At the duct wall $U = 0$ and $\partial T/\partial n = -q''/k$ (4)

where n is normal to the wall surface.

To determine the temperature gradient $\partial T/\partial n$ at the wall surface, one has the relation

$$dT/dn = (dT/dr)\,(dr/dn) + (dT/d\theta)(d\theta/dn) \quad (5)$$

To get the values of dr/dn and d /dn consider the triangle OPC shown in Fig. 5 and apply the sine-rule one obtains

$$\sin\,(180-\beta)/(\sqrt{2}\,R_o) = \sin\theta\;/R_o$$

That is, $\sin\beta = \sqrt{2}\,\sin\theta$

Again, from Fig. 5 one has

$$dr/dn = \sin\beta = \sqrt{2}\;\sin \quad , \quad \text{and}$$

$$dr/dn = \cos\beta/r = \sqrt{1 - \sin^2\beta}/r = \sqrt{1 - 2\sin^2\theta}/r$$

$$= \sqrt{\cos\,(2\theta)}\;/r \quad (6)$$

Substituting for dr/dn and dθ/dn from equations (6) in equation (5) one gets the discretization equation at the wall as follows

$$T_{i,j}\left[\frac{\sqrt{2}\ \sin\theta}{r_i - r_{i-1}} - \frac{\sqrt{\cos(2\theta)}}{r_i(\theta_{j+1}-\theta_j)}\right] = -\bar{q}/K + \frac{T_{j,i-1}(\sqrt{2}\ \sin\theta)}{r_i - r_{i-1}} - \frac{T_{j+1,i}\sqrt{\cos(2\theta)}}{r_i(\theta_{j+1}-\theta_j)} \quad (7)$$

The domain of interest has been devided into 11x21 grids. The grids have been examined and were satisfactory in all inspections. The non-uniform step length were employed in both directions r and . The grid size is increased gradually away from the wall and corners. The solution procesure is employed with the boundary conditions in equations (4) and (7). The algorithm of the computer code employed in the present study is described in dersils in reference (1). Briefly, the procedure is to cast the differential equations into descritization equations which relates the variables at a point in the flow field to that at the surroundings. A numerical solution is then obtained by iterative solution of the descritization equations for a specified array of nodes throughout the flow field by Gaus-Siedel point to point method (2).

It will be noticed that in equations (2) and (3) convective terms representing effects of the cross-sectional velocities do not appear. This is because, in this analysis the effects of extremely small (typically of the order of 0.5% of the mean axial velocity) but finite secondary flows are ignored. The secondary flows play an important role in the manner in which the axial velocity is distributed in the sub-channel and therefore significantly affect the distribution of the temperature at the wall which in turn is so important from the point of view of examining the cooling process of the fuel rods.

RESULTS AND DISCUSSION

While the investigation is primarly concerned with heat transfer, temperature distribution along the length of the duct was measured to help the prediction model. Fig. 6 illusterates the behaviour of the wall temperature along the length of the duct for two different azimuthal positions. It is clear that the axial wall temperature gradient dT_w/dz is constant. This result agrees with that obtained from the overall heat balance over an axial element dz as given by the following relation

$$T_w = T_{in} + \ddot{q} \left[(1/h) + \left(\frac{\rho}{\dot{m}\ c_p}\right) z \right],$$

which applies to fully established flow with uniform heat flux.

The fully developed nature of heat transfer conditions at two sections 90 and 120 equivelent diameters downstream are shown in Fig. 7, where the pheripheral wall temperatures profiles exhibit identical shapes as obtained theoretically and experimentally.

Most noticable is the temperature behaviour near the channel corner, where the predicted values are lower than those obtained experimentally. This difference may be explained as due to the non-uniform nature of the heat flux, which is not considered in the theoretical code.

Also, the temperature near the corner is higher than the other pheripheral temperatures which is due to bad cooling near the corner where the fluid velocity is low. The same is concluded from figures 8 and 9 where a comparison between the predicted and experimentally obtained values of heat transfer coefficient and Nusselt number is shown. The curves indicate that the rate of heat transfer near the corners is low which is physically acceptable. According to the curves, the experimentally obtained values of heat transfer coefficients and Nusselt numbers are higher than that obtained theoretically for different Reynolds numbers. This deviation may be explained as due to surface roughness which increases heat transfer with increased friction losses as indicated by Dippery and Sabersky (9).

Finally, as the Prandtle number of water under atmospheric pressure and at temperatures up to 90°C is considerably higher than one ($\simeq 2$), it is expected that the velocity boundary layer is thicker than the thermal boundary layer. This is turn leads to bad cooling near the corners of the cusped duct as indicated by the high values of temperature.

CONCLUSION

From the previous discussion it is concluded that the proposed theoretical code is a good tool for the prediction of the laminer convection problem in a non-circular duct as applied to the 4-cusped geometry. When the roughness effects and the non-uniform nature of the heat flux are taken into consideration realistic results may be obtained.

REFERENCES

1. Gosman, A.D.; Pun, W.M.; Runchal, A.K.; Spalding, D.B.; Wolfstein, M. "Heat and mass transfer in recirculating flows" Accademic Press, London and New York, 1969.
2. Patankar, S.V. "Numerical heat transfer and fluid flow" Hemisphere Publishing Corporation, 1980.
3. Schlichting, H. "Boundary layer theory" Mc Graw-Hill Book Company. 1960.
4. Holman, J.P. "Heat transfer", Mc Graw-Hill Book Comp. 1981.
5. Cohen, B.I. "Physocs of the reactor meltdown accident" Nuclear Science and Engineering, Vol. 80, pp. 47-60, 1982.
6. Barrow, H. and Hassan, A.K.A., "Azimuthal temperature variation in the cladding of P.W.R. fuel elements" Nuclear Engineering and Design Journal, 1983.
7. El-Hadik, A.A.B. "A sudy of the effect of the secondary flows on the longitudinal velocity distribution in cusped ducts" 3rd Int. Conference in Numerical Analysis on Thermal Problems, Seattle, USA, 1983.
8. Heiber, C.A. "Mixed convection above a heated horizontal surface" Int. J. Heat Mass Transfer, Vol. 16, pp. 769-785, 1973.
9. Dippray, D.F. and Sabersky, R.H. "Heat and momentum transfer in smooth and rough tubes at various Prandtl Numbers" Int. J. Heat Mass transfer, Vol. C, pp. 329, 1963.

MOMENCLATURE

a,b,c,d	Occasional constants, coefficients
c_p	Specfic heat, KJ/Kg. grad
k	Thermal conductivity, w/m grad
h	Heat transfer coefficient, w/m^2 grad
q''	Heat flux, w/m^2
R_o	Rod outside radius, m
U	Fluid velocity
ν	Kinematic vescosity, m^2/sec
α	Thermal difusivity, k/ c_p
ρ	Density, Kg/m^3
T_{in}	Inlet temperature, C
T_w	Wall temperature ,C
P	Wetted perimeter
p	Pressure
$\dot{m}$	Mass flow rate
x	Peripheral distance measured from the corner
Gr	Grashof number
Nu	Nusselt number
Pr	Prandtle number

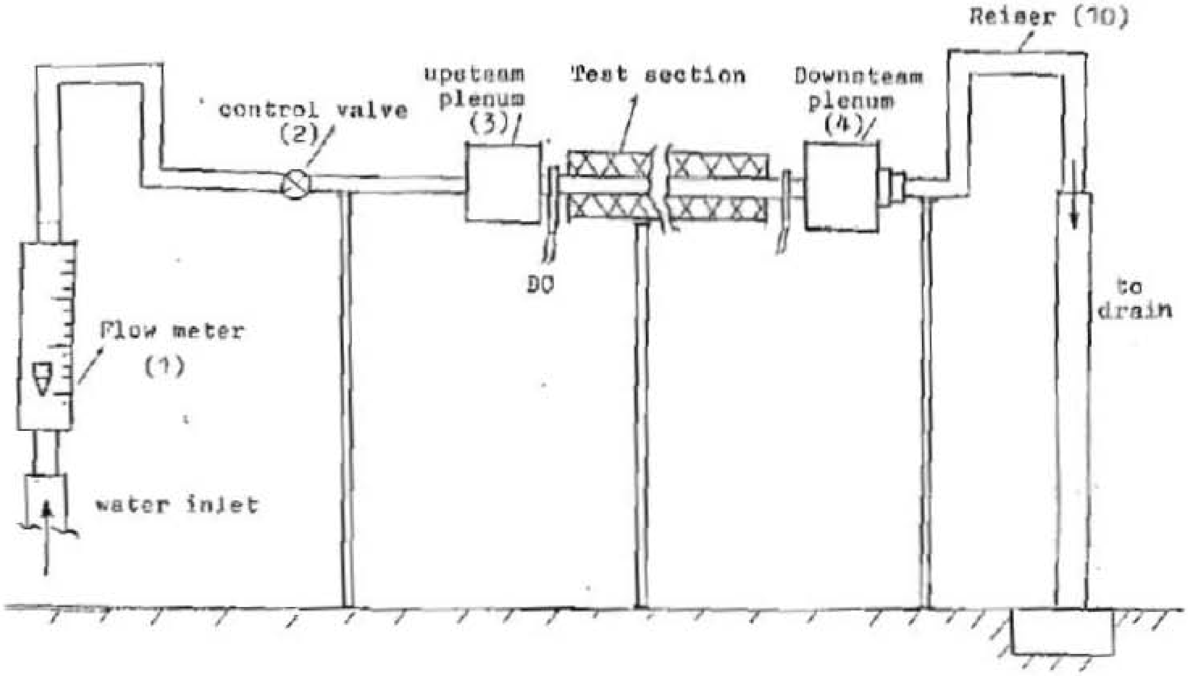


Fig. 1 Experimental Test Rig

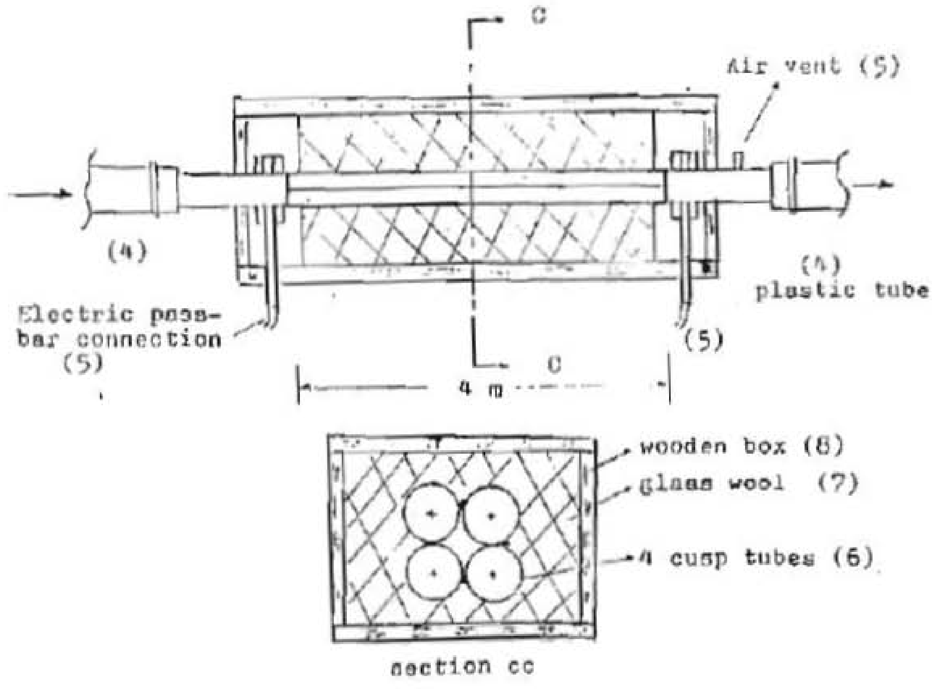


Fig.2 Test Section

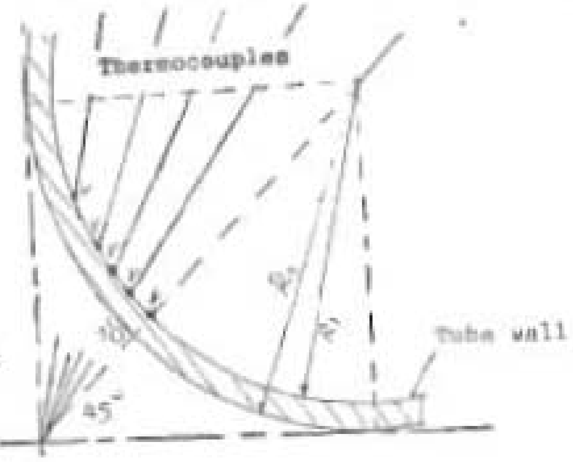
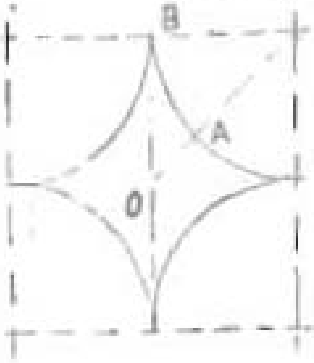


Fig.3 Thermocouples arrangement



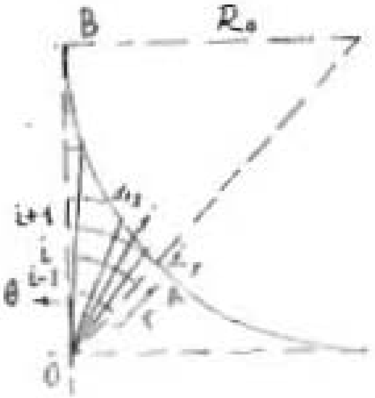
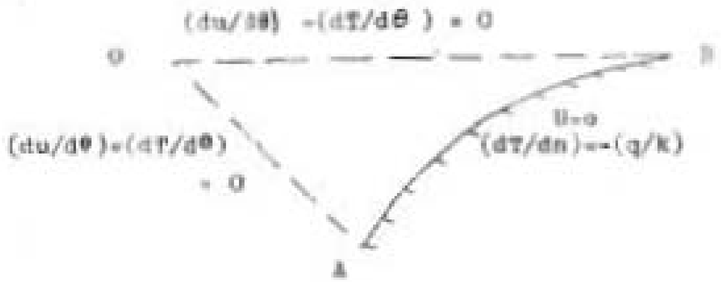
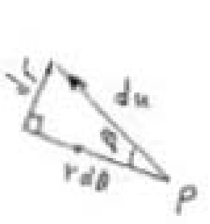


Fig.4 Grid design





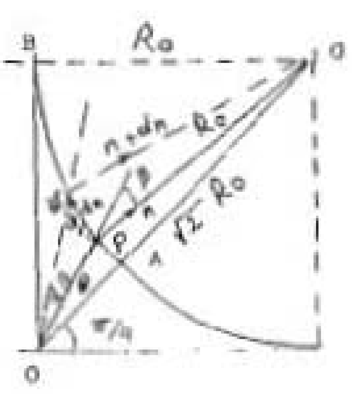


Fig.5 Boundary conditions

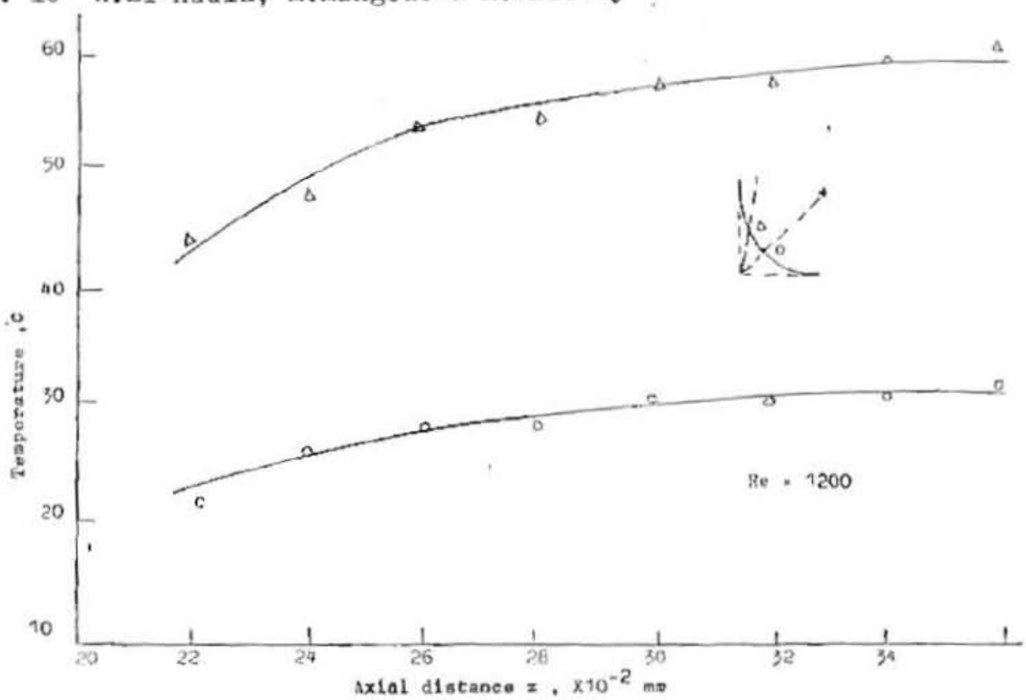


Fig.6 Wall temperature against the axial distance along the duct

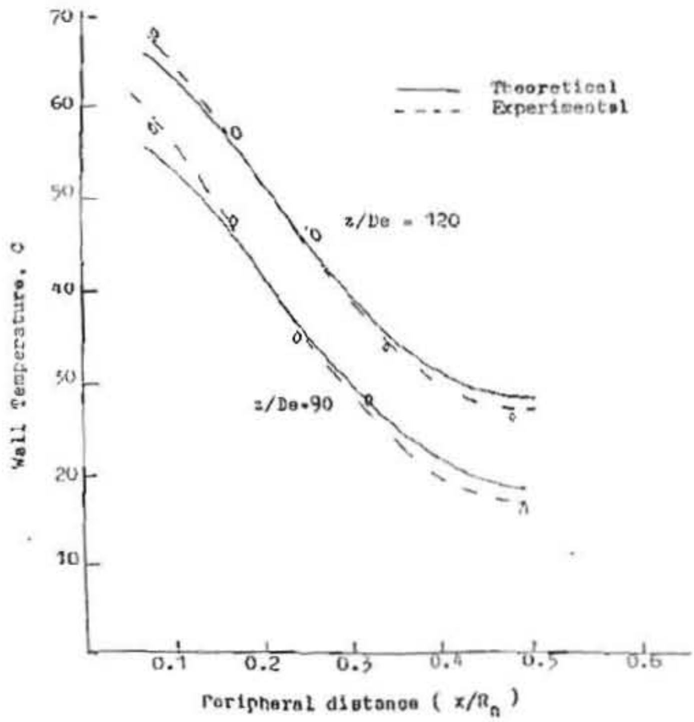


Fig.7 Variation of the peripheral wall temperature

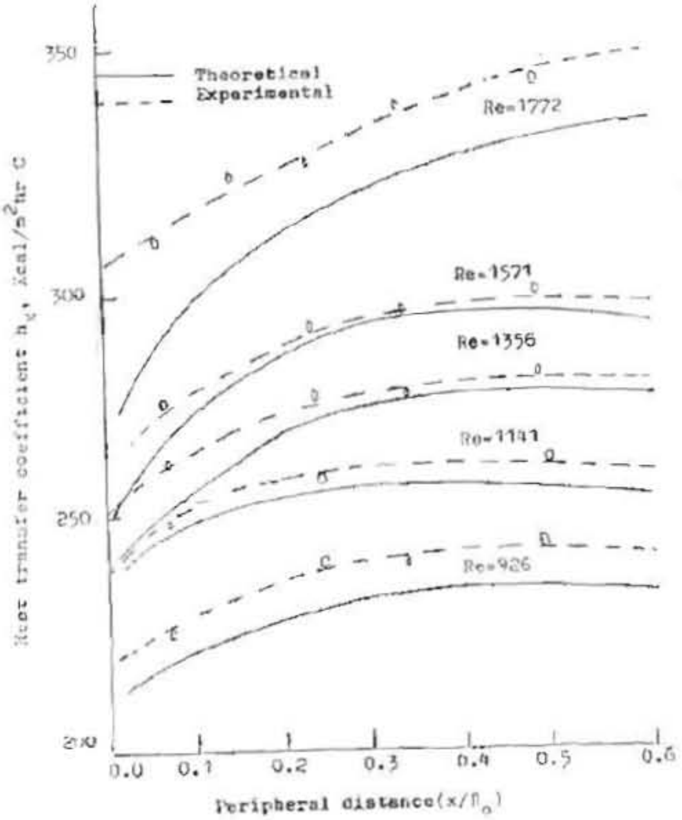


Fig.8 Variation of the heat transfer coefficient around the wall

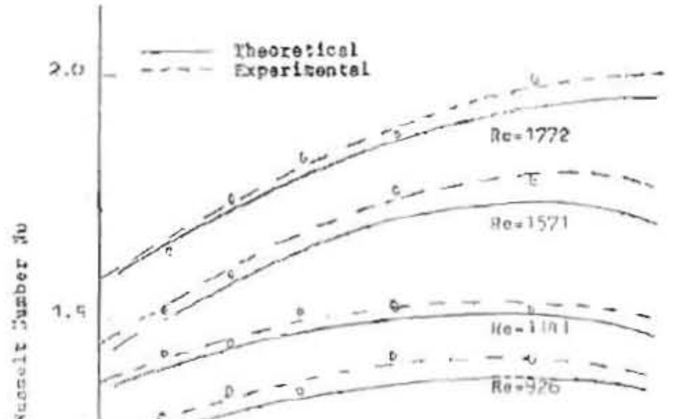


Fig.9 Variation of local Nusselt Number around the wall