A Contribution to the Coanda Effects*

By Akiharu Mitunaga**

Tatuzo Hirose***

Digital simulation of a discontinuous diffusing flow with particular reference to the Coanda effects is carried out. We give an initial flow pattern arbitrarily and modify it step by step so that it satisfies the Navier-Stokes equations of motion. Reynolds number is assumed to be low enough to keep the flow within the laminar region. A symmetrical initial pattern results in a symmetrical solution anyway in all cases but it is supposedly an unstable flow pattern for Reynolds numbers higher than a critical value. We suppose so because: (1) an unsymmetrical initial pattern results in an asymmetrical solution in which the jet flow attaches to either wall, (2) the symmetrical solution is deformed by a small disturbance imposed and eventually settles in the above mentioned unsymmetrical pattern, only for a Reynolds number higher than the critical value. It may thus be said that the Coanda effects occur also in the laminar region and, when they occur, solution of the Navier-Stokes equations is not unique.

1. Introduction

Many authors have studied a discontinuous diffusing flow with the Coanda effects in relation to the fluid amplifier. They investigated the distance from the entrance of a jet flow to the reattachment point and the curvature of the jet flow axis.\(^{10,22}\) As for theoretical works, Hung et al. performed a calculation assuming symmetrical flow\(^{6}\) and from et al. obtained a numerical solution of an unsteady turbulent flow in a diffuser\(^{6}\). In spite of these existing contributions the analysis on the flow with the Coanda effects still remains uncompleted\(^{11,15}\). Above all, it is not yet certain when or under what condition the flow axis bends to one side. From the engineer's point of view this criterion is really important.

We will carry out a digital simulation study on the flow pattern in a discontinuous diffuser with particular interest in the stability of a symmetrical and an unsymmetrical flow. It is true that fluid amplifiers are usually operated in the turbulent region and most investigations on the Coanda effects were made in that region accordingly. In turbulent region, however, mathematical analysis on the flow stress and mean velocity are very difficult and the result unreliable. This is the reason why we limited our analysis to within the laminar region. According to our analysis, Coanda effects do occur also in the laminar region and this result will hopefully lead us to a qualitative explanation of the Coanda effects in the turbulent region.

Incidentally the study on the laminar flow pattern in a diffuser reminds us of the behavior of the flow behind a body. Toman et al.\(^{10}\) and Hirota et al.\(^{8}\) obtained a computational solution of the Navier-Stokes equations for the laminar flow behind a circular cylinder. They showed that there can be always a pair of symmetrical vortices behind a cylinder, which, by a small disturbance, become unsteady and eventually change into two rows of Kármán vortices. They stated also that if the velocity of the main stream is low enough, the symmetrical vortices are stable and Kármán vortices do not come out after a disturbance. Fromm\(^{9}\) obtained the similar result for the flow behind a rectangular cylinder.

Behavior of the flow we obtained in the symmetrical two-dimensional conduit with abrupt expansion is rather similar to this; a symmetrical flow can be obtained as the result of computation, which, by a small disturbance, becomes unsteady and changes into an unsymmetrical flow in which the jet attaches to one side. This is, of course, a flow pattern with the Coanda effect. When the main flow velocity is low enough, our calculations indicate that any unsymmetrical flow can not occur. This again resembles the above mentioned behavior of the flow behind a circular cylinder.

Such behavior of the flow in a discontinuous diffuser and also the one behind a cylinder is, in our view, of a great theoretical interest concerning the uniqueness of the solution of Navier-Stokes

---

* Received 10th November, 1972.
** Assistant, Faculty of Engineering Science, Osaka Univ., Toyonaka.
*** Professor, Faculty of Engineering Science, Osaka Univ., Toyonaka.
equations; apart from its practical interest, we should pay attention to this aspect of the Coanda effects.

2. Formulation

Assuming a flow in a two-dimensional conduit with discontinuous expansions for incompressible fluid, we take the Navier-Stokes equations for incompressible fluid. Every length is nondimensionalized by dividing by the width of the opening of the jet, velocity by the mean velocity \( U \) at that section, pressure by \( fU^3 \), time by \( U/D \), stream function by \( UD \) and vorticity by \( U/D \). Reynolds number \( Re=UD/\nu \) is also introduced where \( \nu \) is the kinematic viscosity. Hereafter we will use these nondimensional quantities throughout. Equation (1) results from the relation between stream function and vorticity, and upon the elimination of the pressure terms from the Navier-Stokes equations the vorticity transport equation (2) is obtained, as follows:

\[
\psi = -\varphi_x + \psi_n \quad \quad \quad (1)
\]

\[
\varphi + \nabla \cdot \varphi = \frac{1}{\rho} \nabla p + \mathbf{f}\quad \quad \quad (2)
\]

where suffixes denote the differentiation in respect to the corresponding parameters, e.g., \( \varphi_n = \psi / \Delta x \) and \( \varphi_{nn} = \psi / \Delta x^2 \) and so on.

The steady state solution for certain boundary conditions means the solution of Eq. (2) without the term \( \varphi_{nn} \). In spite of the fact above mentioned, however, the digital simulation approach may diverge if

\[
\psi_{n+1} = \left( \frac{1}{\Delta x^2} + \frac{1}{\rho} \right) \left[ \frac{1}{\Delta x} \psi_{n+1}^{x+1} + \frac{1}{\Delta x} \psi_{n-1}^{x+1} + \frac{1}{\Delta x} \psi_{n+1}^{x-1} + \frac{1}{\Delta x} \psi_{n-1}^{x-1} \right] + \frac{1}{\rho} [ \psi_{n+1}^{x+1} - \psi_{n-1}^{x-1} ] \quad \quad \quad (3)
\]

where the suffix \( n \) denotes \( t/\Delta t \), i.e., \( t = n \Delta t \).

The other is called A.D.I. method, developed by D.W. Peaceman et al. (12). It was employed to solve the flow around a circular cylinder by J.S. Seo et al. (13). In case of this method, a pair of equations are needed as the time interval \( \Delta t \) is divided into halves and one step of time proceeds by two successive calculations, i.e.,

\[
\psi_{n+1/2} = \left( \frac{1}{\Delta x^2} + \frac{1}{\rho} \right) \left[ \frac{1}{\Delta x} \psi_{n+1}^{x+1} + \frac{1}{\Delta x} \psi_{n-1}^{x+1} + \frac{1}{\Delta x} \psi_{n+1}^{x-1} + \frac{1}{\Delta x} \psi_{n-1}^{x-1} \right] + \frac{1}{\rho} [ \psi_{n+1}^{x+1} - \psi_{n-1}^{x-1} ] \quad \quad \quad (4-a)
\]

\[
\psi_{n+1} = \left( \frac{1}{\Delta x^2} + \frac{1}{\rho} \right) \left[ \frac{1}{\Delta x} \psi_{n+1}^{x+1} + \frac{1}{\Delta x} \psi_{n-1}^{x+1} + \frac{1}{\Delta x} \psi_{n+1}^{x-1} + \frac{1}{\Delta x} \psi_{n-1}^{x-1} \right] + \frac{1}{\rho} [ \psi_{n+1}^{x+1} - \psi_{n-1}^{x-1} ] \quad \quad \quad (4-b)
\]

A.D.I. method proved quicker in convergence than the first for some cases so we used this conveniently in such cases.

We assume at first an initial flow pattern and the stream function at \( t = 0 \) accordingly. Then we use either Eq. (3) or (4.a,b) to obtain \( \psi \) at a time one step further. The stream function at that time is calculated using the following equation which is reduced from Eq. (1) by the relaxation method, i.e.,

\[
\psi_{n+1} = (1-\alpha) \psi_n + \frac{\alpha}{2\Delta x^2 + \Delta t} \left[ \psi_{n+1,x}^{+} + \psi_{n+1,x}^{-} + \psi_{n+1,y}^{+} + \psi_{n+1,y}^{-} - \psi_{n,y}^{+} - \psi_{n,y}^{-} - \psi_{n,x}^{+} - \psi_{n,x}^{-} \right] \quad \quad \quad (5)
\]

\( m \) is the number of iterations, and \( \alpha \) is a relaxation parameter which is determined by the equation

\[
\alpha = \frac{1}{1 + (t/\Delta t)^{1/2}} \quad \quad \quad (6)
\]

where \( I \) and \( J \) are the numbers of meshes in \( x \) and \( y \) directions respectively.

Contrary to the stream function \( \psi \), \( \varphi \) is not given directly from either Eq. (3) or Eq. (4.a,b) on the walls. It is calculated by Eq. (7) (refer to Fig. 1) after the inner values of \( \varphi \) are determined either by Eq. (3) or by Eq. (4).
\[ \xi_s = \frac{3}{4} \left( \frac{\rho_0}{\rho} \right) \left( \phi_0 - \phi_2 \right) - \frac{1}{2} \psi_2, \]  \hspace{1cm} (7)

We take \( k^2 \) when a wall is parallel to the x axis, and \( k^2 \) when the wall is parallel to the y axis. Eq. (7) is obtained by expanding \( \psi \) and \( \xi \) near the wall into a power series. To be accurate with the expansion, incidentally, the term \( k^2/8 \) \( \frac{S_0 + S_0}{\rho} + 0(k^2) \) should be added to the right hand side of Eq. (7). In this term, \( \frac{S_0 + S_0}{\rho} \) on a wall becomes zero for steady states, and \( \frac{S_0}{\rho} \) does for unsteady states. It is possible to calculate \( \frac{S_0}{\rho} \) using the values of \( \xi \) for \( n=1 \) at \( n=1 \), but it is not necessary since this term is of \( 0(k^2) \) while Eq. (3) for inner region contains the same order error. Eq. (7) can thus be used safely.

In carrying out the calculation there is a computational problem concerning a singularity of the flow at the nozzle mouth. That is, fluid separation obviously occurs at the mouth corner, so that the vorticity there can not be defined by the laminar analysis. Hung et al. computed the vorticity at this point by Eq. (7) under the assumption that the vorticity there depends only upon the vorticity distribution in \( y \)-direction at the mouth position. The validity of this assumption is not clear enough, in our view, and after a comparative survey on a few different approaches we decided to assume a parabolic velocity distribution along the \( y \)-axis at the mouth. The boundary values of \( \psi \) and \( \xi \) at this section are thus given "apriori". We chose this way because we felt that the least logical ambiguity should be involved in the present analysis on the uniqueness of the flow pattern. There could be a better approach to give a better agreement with observed velocity distribution incidentally.

It is impracticable to continue calculations infinitely to downstream and we should end the calculation at an adequate downstream section. \( \psi \) and \( \xi \) at the extreme downstream section are calculated by the equations,

\[ \psi_s = \psi_{s-1} - 2\psi_{s-1} + 2\psi_{s-2}, \]  \hspace{1cm} (8)

\[ \xi_s = \xi_{s-1} - 2\xi_{s-1} + 2\xi_{s-2}. \]  \hspace{1cm} (9)

Calculations are carried out to the symmetrical order in \( y \) direction, namely \( \psi_s = \psi_{s-1} = \psi_{s-2} \) and then \( \psi_s = \psi_{s-1} \) for both of \( \psi \) and \( \xi \). As for \( x \) direction we calculate from upstream to downstream. We take \( k = 1/8 \), \( h = 3/16 \) and \( k = 1/16 \), \( h = 3/32 \). The time interval is \( t = 0.05 \). At every time step, after values of \( \xi \) have been obtained, new values of \( \psi \) are calculated applying an iteration process and if \( \psi \) converges enough, then the process proceeds by one step. This process is repeated until values of \( \xi \) do not vary with time. We regard \( \psi \) to have converged enough when the condition \( \psi_s = \psi_{s-1} \) is satisfied. \( \xi_s \) is the largest value of the stream function at the outlet of the nozzle. Similarly as for \( \xi \), we set the condition of convergence as follows:

\[ (\xi_{s+1} - \xi_s)/\xi_s < 0.0001 \]  \hspace{1cm} (10)

where \( S_0 \) is the largest value of vorticity at the outlet of the nozzle.

3. Cases for Numerical Calculation

Numerical calculation is performed for a flow in a conduit whose width is abruptly expanded by the ratio 1:3. We made the calculations for the following four cases:

(1) Symmetrical initial pattern \( R = 100 \), no disturbance; give a symmetrical initial pattern. Reynolds number \( R = 100 \). Proceed with the step by step calculation to obtain the final solution. \( k = 1/16 \) and \( h = 3/32 \). The flow pattern thus obtained is exactly symmetrical. Eq. (3) is used as the vorticity equation.

(2) Unsymmetrical initial pattern \( R = 100 \), no disturbance; next give an unsymmetrical pattern and proceed with the similar calculation. This time the flow pattern does not approach the above mentioned symmetrical one. Instead, another quite different pattern appears as the final solution; the flow axis bends to one side and attaches to the wall. Here it should be noted that there can be two entirely different steady flow patterns under the same boundary condition; a symmetrical pattern of the case (1) and an unsymmetrical one of this case. Eq. (3) is used as the vorticity equation. \( k = 1/16 \) and \( h = 3/32 \).

(3) Symmetrical initial pattern \( R = 100 \), with a disturbance; take the final flow pattern of the case (1) as the initial pattern in this case. It is, of course, symmetrical and satisfies Navier-Stokes equations. Then assume a small opening on one side wall and inject a small quantity of fluid for a short time. Close the opening right away and start the step by step calculation as before, but using Eqs. (4a,4b). The manner of flow injection is to increase the flow rate with the time up to 4% of the main flow rate, and then to decrease it at the same pace. When it reaches nil, the opening is closed. This procedure is to see how the symmetrical flow suffers from a small disturbance and thus to examine the stability of the flow. The quantity of injection is very small and the injection time is quite short, so the disturbance is small enough. The flow pattern does not hold any more symmetrical shape, however, and gradually bends opposite to the injection side. Eventually it approaches the pattern in which the jet attaches to the side wall. This result suggests that the symmetrical flow pattern is not stable. Eq. (4) was employed to calculate \( S \) and \( k = 1/8 \), \( h = 3/16 \) were used.

(4) Symmetrical initial pattern \( R = 45 \) / with a disturbance; we tried this low Reynolds number case in connection with the analogy with the flow behind a plate, as was already stated in the present introduction. Firstly the same procedure as the case (1) but with \( R = 45 \) instead of 100 is taken. The final flow pattern turns
out to be symmetrical. Then we introduce a disturbance the same as the case (3), i.e., injecting a small quantity of fluid for a short time. Details of injection are the same as the case (3). Unlike the case (3) with \( R=100 \), the flow quickly recovers its symmetrical pattern after the injection is finished. Any steady unsymmetrical pattern does not come out. This suggests that the symmetrical pattern is stable at \( R=45 \) while it is unstable at \( R=100 \). Eq. (4) was used for calculating vorticity and \( k=1/8, h=3/16 \) was taken.

4. Discussion on the Results

Among the results of calculation for the cases mentioned in the previous section, figures of the stream function are shown as follows, and details and special features of them are explained. The vorticity was also calculated and distribution of velocities or other parameters can be deduced easily. Since we have no direct concern with them in this investigation, however, explanation of them is omitted.

(1) In Fig. 2 a symmetrical flow pattern is shown; case (1).
(2) In Fig. 3 an unsymmetrical flow pattern with Coanda effect is shown; case (2). Initial values of the stream function were decided arbitrarily only to make seemingly initial flow unsymmetrical, so they did not satisfy even the continuity equation. A vortex occurs at downstream of the reattachment point. As the reason for this, it is considered that the separation occurs since direction of the jet flow changes at that point and stream tube expand.

(3) Figs. 4 through 9 show the transient patterns after the symmetrical steady flow at \( R=100 \) is disturbed by a fluid injection; case (3). The initial pattern is the one shown in Fig. 2. The fluid injection starts at \( t=0 \). The position of the injection port is shown in Fig. 4; located between \( x=5.8D \sim 7.3D \) where D denotes the nozzle dimension. We use a nondimensional time \( t=(\text{real time})/(1/\nu) \). From \( t=0 \) to \( t=4 \), the injection flow rate increases at a constant pace; it reaches \( 4\% \) of the main flow rate at \( t=4 \). The stream lines at this time are shown in Fig. 4. From \( t=4 \) to \( t=8 \).
the injection rate decreases at the same pace and it becomes nil at \( t=8 \) and at the same time the port is closed. Thereafter calculation is carried out under the same boundary conditions as those in the case of Fig. 2. Stream lines obtained are shown successively in Fig. 5 at \( t=30 \), in Fig. 6 at \( t=50 \) and so on. In Fig. 5 the flow pattern is similar to the symmetrical one i.e., Fig. 2. In Fig. 7 a vortex being to split into two parts and an unsymmetrical pattern is forming. After \( t=90 \) (Fig. 8), the unsymmetrical flow pattern which is almost steady has little difference from the one in Fig. 3.

(4) Figs. 10 through 13 show the transient patterns after disturbance but with \( R=45 \); case (4). At first the symmetrical flow at \( R=45 \) was calculated. Then a small amount of fluid was injected in the same way as the case (3). The flow proved to be hardly influenced. So we next tried to double the injection flow rate; 8% of the main flow rate at the peak at \( t=4 \). Fig. 10 show the stream lines at this time. In spite of the relatively large amount of injected fluid the unsymmetry is not prominent. Fig. 11 shows stream lines when the injection rate decreases to 2%. Then stream lines at \( t=22 \) are shown in Fig. 12 and those at \( t=28 \) in Fig. 13 which proves to be perfectly symmetrical. This result tells us that the symmetrical flow is stable at Reynolds number as low as 45, while it is unstable at \( R=100 \).

5. Conclusions

A step by step digital simulation of two dimensional flows in a symmetrical conduit with abrupt expansion is carried out. The results obtained are:

(1) Starting from a rigorously symmetrical initial flow pattern a symmetrical pattern is obtained as the final steady state solution.

(2) When the Reynolds number exceeds a certain value, an unsymmetrical initial flow pattern settles in an unsymmetrical pattern in which the main flow bends and touches one side wall.

(3) When the Reynolds number exceeds a certain value, the symmetrical steady pattern is perturbed by a small disturbance and eventually settles in another shape of flow pattern in which the main flow bends to and touches one side wall, an unsymmetrical one; this tells us that the symmetrical flow pattern is not stable at relatively high Reynolds number, and

(4) When the Reynolds number is small, the symmetrical flow pattern is stable and recovers its shape after a small disturbance.

From these computational results it may be said that the Coanda effects occur in laminar region. At the same time these results are an example of the cases in which the solution of the Navier-Stokes equations is not unique; one solution is stable and the other unstable. This reminds us of the stability of vortices behind a circular cylinder. In the present case of Coanda effects, however, the stable solution can also be steady. This differs from the case of the Kármán vortices, and presumably it is a simpler case than Kármán vortices.

Concerning the numerical computation procedure the fact that the vorticity at the nozzle mouth corner can not be defined
introduces a difficulty. This is left for future study. Beside this problem, theoretical examination on the accuracy of the finite difference method should be encouraged. At the same time comparison of such numerical computation with experimental result is important.

References

(9) Fromm, J. E. and Harlow, F. H., Phys. Fluids, 6-7 (1963), 975.