Study on Three-Dimensional Flow and Heat Transfer in Miter-Bend

(1st Report. Analysis of Flow in Laminar Region)*

By Hiroshi YAMASHITA**, Genichiro KUSHIDA***

and Ryotaro IZUMI****

Three-dimensional flow and heat transfer in a rectangular miter-bend, with a 90° turn, was studied for the purpose of examining the effect of centrifugal force due to bending, in connection with a corrugated wall channel as a means of augmenting a forced-convection heat transfer with a single-phase flow in a heat exchanger. Analysis of results is reported for the flow in a laminar region with aspect ratio α=1 and Reynolds number Re=300. Complicated aspects of the flow in terms of velocity vectors, path lines, pressure contours, and wall shear stress contours are shown with detailed examination of flow mechanism.

Key Word: Fluid Mechanics, Numerical Analysis, F.D.M., Miter-Bend, Three-Dimensional Flow, Laminar Flow, Heat Exchanger

1. Introduction

The authors(1)-(5) have studied by a two-dimensional treatment a corrugated wall channel systematically as a means of augmenting a forced-convection heat transfer with a single-phase flow in a heat exchanger. In those studies, a theoretical analysis was made by considering a model of a parallel channel, and a corrugated rectangular channel with a very small aspect ratio was examined experimentally. However, results of the visual observation experiment of flow were not in perfect agreement with those of theoretical analysis, because under the effect of centrifugal force due to bending, the main flow of the central part with large velocity was pushed to the outer wall of bending and took a roundabout way towards the both side walls, and the flow in the separating zones tended to turn spirally from the central part to the both side walls. Effects of the three-dimensional flow on pressure loss and heat transfer offer no problems when the aspect ratio is small. In an actual heat exchanger, however, a miter-bend with a flow section of a fairly large aspect ratio is considered, and therefore a study by three-dimensional treatment becomes necessary. Accordingly, in this study, we deal with the three-dimensional flow and heat transfer in a rectangular miter-bend, with a 90° turn, for the purpose of examining the effect of centrifugal force due to bending, in connection with a corrugated wall channel. This paper presents results of an analysis of the flow in a laminar region.

A miter-bend is widely used as a fundamental element of piping channel in the industrial field, and so there are many technical data(6) on the hydraulic losses; but, the study of its flow mechanism is made relatively recently. Experimental studies include the following: Suu et al.(7) showed a complicated three-dimensional flow similar to ours for the rectangular miter-bend by a visual observation experiment, and examined the detailed flow mechanism especially in the separating zones. Goldstein et al.(8) showed the existence of a secondary flow similar to Görler vortex for a corrugated wall channel bent three times with a fairly small aspect ratio by naphthalene sublimation technique. Tunstall et al.(9) showed in a turbulent region the existence of a backflow zone and the unsteadiness of flow downstream of an inner bending corner for miter-bend with a round flow section. Shimizu et al.(10) showed complicated aspects of secondary flow for a corrugated wall channel bent several times. As for the secondary flow of a bend with similar centrifugal force due to a smooth curvature, there are available many experimental data. On the other hand, some theoretical analyses were made as follows: such as the two-dimensional analyses made by the present authors(1),(3),(4), Suu et al.(11), Hur(12), Orland(13) and others.
As for three-dimensional analysis similar to this paper, only Fukunaga studied (14) a miter-bend itself, where the unsteady flow for a square miter-bend was calculated by convection finite difference method and the secondary flow at a small Reynolds number with no separation and reattachment was shown in terms of velocity vectors. On the other hand, there are many studies on a bend with a smooth curvature. As for the developed flow, there are many studies including analyses by Mori et al. (15), (16) and Uchida et al. (17) who modelled the secondary flow, where it was separated into the central part of channel and the boundary layer along channel walls. Especially in Cheng's (18) and Joseph's (19) studies, the existence of an additional vortex was shown together with the ordinary secondary flow at a certain Dean number. As for an entrance flow, there are Akiyama's analyses (20), (21) by a parabolic equation, and Yee's one (22) by a perfect elliptic equation where it was pointed out that a parabolic or a semi-elliptic method was not adequate for the channel with large curvature. Humphrey et al. dealt with a square-duct bend with fairly large curvature, showed (23) the existence of a backflow zone in the neighborhood of the outer wall beside both side walls in a laminar region and made (24) a theoretical analysis in a turbulent region comparing with its experimental data. Moore et al. (25), (26) studied a bend in an accelerated region comparing with its experimental data.

As mentioned above, as for a miter-bend, analytical studies are few as compared with a bend and its phenomenon has been unexplained theoretically. Although experimentally its complicated aspects of flow were made relatively clear. From this point of view, in this analysis, we wish to analyze theoretically by using perfect elliptic equation a square miter-bend with aspect ratio \( \alpha = 1 \) and relative large Reynolds number \( R_e = 300 \). Where the three-dimensional flow due to centrifugal force is considered to be remarkable, and separation and reattachment occur. We show especially path lines to know complicated aspects of the flow clearly as well as velocity vectors, pressure contours and wall shear stress contours with detailed examination of the mechanism. The analysis is made numerically by solving a three-dimensional continuity equation and Navier-Stokes equation by using Patankar's finite difference method (27) on velocity and pressure for a steady incompressible viscous laminar flow. We obtain paths by calculating the values at optimal position by interpolating values at grid points, and integrating a path line equation.

**Nomenclature**

- \( D_2, D_3 \): channel widths
- \( d_c \): characteristic diameter
- \( \alpha \): aspect ratio
- \( \mu \): viscosity
- \( \nu \): kinematic viscosity
- \( \rho \): density
- \( \tau \): shear stress
- \( \varphi \): independent variable
- \( x, y, z \): coordinates
- \( n \): number of iterations
- \( s \): starting point
- \( \text{ref} \): reference

**2. Calculation Method**

**2.1 Calculation of velocity and pressure fields**

Analytical model is shown in Fig. 1. We divide the channel into two zones \( \square \) and \( \square \) with overlapping part, and take three-dimensional Cartesian coordinates in each zone: the origin is at the side end of the inner corner. \( x_1 \) is in the flow direction. \( x_2 \) and \( x_3 \) are in a cross section perpendicular to it. \( D_2 \) and \( D_3 \) are the channel widths in \( x_2 \) and \( x_3 \) directions respectively. Inlet and outlet of the channel are respectively at \( 1.5D_2 \) upstream and \( 1.8D_3 \) downstream of the inner bending corner. Aspect ratio \( \alpha \) and characteristic diameter \( d_c \) are defined as follows:

\[
\alpha = \frac{D_2}{D_3}, \quad d_c = \frac{2D_2D_3}{D_2^2 + D_3^2}
\]

Fig. 1 Analytical model
Considering a steady incompressible viscous laminar flow in the channel and constant physical properties of fluid, the basic equations are continuity and Navier-Stokes equations:

\[ \nabla \cdot \mathbf{u} = 0, \quad \frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho} \nabla p + \nu \Delta \mathbf{u} \]

The boundary conditions for velocity and pressure are as follows. The flow at the inlet is assumed to be hydrodynamically fully-developed, whose velocity profile apply Purday's approximation distribution. At the outlet the flow is assumed such that variables have no change in the flow direction. At the walls, velocity and pressure corrections are zero. The values at the boundaries are given by the values at the corresponding inner points in overlapping parts, and so the two zones are connected.

On the basis of these equations and boundary conditions, we calculate numerically by using Patankar's finite difference method(27) for each zone of the channel. Using the control volume of rectangular face as shown in Fig.2, we define pressure \( p \) at its central point and normal velocity components \( u_1, u_2, u_3 \) at the middle of each face respectively. So we derive a differential equation and a pressure correction equation from the basic equations, and solve these steady equations by A.D.I. method. Then, upwind-difference method is applied to the convection terms. In calculations, we take 0.4 as the values for the relaxation parameter, and set \( \max(\nabla_1, \nabla_2, \nabla_3)/S_{ref} < 10^{-6} \) for the convergence criterion, where \( n \) is variable, \( n \) is the number of iterations, and \( \nabla_1, \nabla_2, \nabla_3 \) are mean velocity \( u_1 \) for velocity and \( p_{vel}/2 \) for pressure correction, respectively. Then, the error is at most only three per cent. when calculating the flow quantities at each flow section by integrating numerically, using spline-method(29), and so the results seem to be good enough for this grid resolution.

The differential grid of whole channel is shown in Fig.3. The numbers of \( x_1 \) grid lines are \( I=1 \) to \( I=15 \) in region 1 and \( I=17 \) to \( I=59 \) in region 2, and the numbers of \( x_2 \) and \( x_3 \) grid lines are \( J=K=1 \) to 11, respectively. The grid distances are not equal, that is, they are small in the neighborhood of bending region and walls. The coordinates of standard position in \( J \) and \( K \) directions are shown in Table 1. and other positions are similar to them. Here, we call the wall of \( K=1 \) and \( K=11 \) the inner and outer walls and the wall of \( J=1 \) and \( J=11 \) the side wall.

### 2.2 Interpolation and path lines

Since, in the above calculation by finite difference method, the values of variables are given at only grid points and the grids are staggered, it is necessary to interpolate the values at grid point for the values of variables at an optional position especially to solve the path line equation and show velocity vectors. Here, we interpolate three-dimensionally by Lagrange's second-ordered interpolation formula(30) in terms of the values at 3×3×27 grid points about the nearest point to the position considered. This is two-dimensionally shown in Fig.4 for convenience: that is, the values at the position of the shadowed portion are interpolated in terms of the values at 3×3×9 grid points as shown in Fig.4.

![Fig.2 Control volume](image)

![Fig.3 Differential grid of miter-bend](image)

![Fig.4 Grid for interpolation](image)
Path line equation is given in the following form:

\[
\frac{dx_1}{u_1} = \frac{dx_2}{u_2} = \frac{dx_3}{u_3} = dt
\]

We obtain path lines by integrating this equation numerically while interpolating velocity components from the channel inlet. Where \( x_1, x_2 \) and \( x_3 \) are dependent variables and \( t \) is an independent variable. Integration is made by Runge-Kutta-Verner method to an accuracy of \( 10^{-10} \).

3. Calculation Results and Examination

In this paper, for the flow in a laminar region for aspect ratio \( a=1 \) and Reynolds number \( R_e = 300 \), velocity vectors, path lines, pressure contours and wall shear stress contours are shown with detailed examination of flow mechanism.

Further, we show in the below figures the aspects from the channel inlet to \( 300 \) downstream of the inner bending corner in the neighborhood of bending region where the flow is complicated.

3.1 Expression by velocity vectors

The projection on each channel section of the three-dimensional velocity vectors in a channel that is the resultant vector of the velocity components at its section are shown in Figs. 5 to 7. Figures 5(a), (b) and (c) correspond to the sections of J=2, 4 and 6, respectively. At the section of J=2, the stream from upstream goes against the opposite outer wall downstream of the outer bending corner as if impinging there. At the section of J=4 it is considerably curved downstream, and at the section of J=6 it is sucked in downstream of the inner bending corner after joining the flow coming out of the opposite outer wall. As for the velocity vector downstream of bending region, at the section of J=6, the vectors point to the
outer wall except those at the vortex region in the neighborhood of the outer bending corner and the backflow region downstream of the inner bending corner, although at the section of J=4 they point parallel to the wall or slightly to the inner wall, and at the section of J=2 they point perfectly to the inner wall and are fairly large. Figures 6(a), (b) and (c) correspond to the sections of K=2, 6 and 10 upstream of bending region, and Figs. 6(d) and (e) to the sections of K=2, 6 and 10 downstream of bending region, respectively. In the neighborhood of the inner wall, the stream from the inlet to the bending corner moves to the middle gradually while flowing down and is sucked into the central part forming a backflow region downstream of the corner. On the other hand, in the neighborhood of the outer wall, the stream extends radially about the impinging point. So the backflow forms a vortex region and the flow going aside springs towards the side walls. And at the middle section, the stream from upstream forms a secondary flow taking a roundabout route towards both side walls by impinging, and downstream it flows parallel to the side wall and the magnitudes of the middle vectors are small because of the secondary flow. Figures 7(a) and (b) correspond to the sections of I=10 and 30 upstream and downstream of bending region, respectively. It is shown that upstream of bending region the vectors point to the middle of the inner wall, and downstream there is a typical secondary flow due to the centrifugal force. However, the secondary flow such as this can not be seen in the neighborhood of the inlet and at the section of 30 downstream of the inner bending corner.

Some of above results indicate the aspects similar to the observation by Suu et al. (7), and so it is proved that an analysis of the steady laminar flow can considerably explain the complicated three-dimensional flow in a miter-bend. Although the expression by velocity vectors shows a changing of the flow at each local position of channel very well, and observation of the vectors at all sections as a whole helps us imagine the aspects of the flow, it is considerably difficult to grasp the complicated behaviors of fluid particles accurately. Consequently, the expression by path line will be made below.

(a) Oblique projection
(b) Orthogonal projection
Fig.8 Path lines (J-6)

(a) Oblique projection
(b) Orthogonal projection
Fig.9 Path lines (J=5, 6 and 7)

(a) Oblique projection
(b) Orthogonal projection
Fig.10 Path lines (J=4)
3.2 Expression by path line
The path lines are distinguished by the values of \( (J, K) \) at the inlet section of \( I=1 \), and are shown in terms of three-dimensional oblique projection and its orthogonal projection to each channel wall in Figs. 8 to 11. Figures 8(a) and (b) show the oblique projection and the orthogonal projection of path lines at the middle sections \( (J=6, K=2, 4, 6, 8 \) and \( 10) \). The path lines go along the section of \( J=6 \) by symmetry, and the three path lines near the inner wall \( (J=6, K=2, 4 \) and \( 6) \) go downstream while being pushed to the outer wall by the centripetal force. Meanwhile the two path lines near the outer wall \( (J=6, K=8 \) and \( 10) \) whirl in the neighborhood of the outer bending corner and are curled towards a certain central point of vortex. Although these path lines ought to stay at this point naturally, they diverge in a normal direction of the vortex surface through a small numerical error, and go downstream again. This small numerical error corresponds to a minute fluctuation in the actual stream. In this calculation they diverge in the direction of the smaller value of \( J \) by accident. Figures 9(a) and (b) show the path lines of \( (J=5, 6 \) and \( 7, K=6) \) and \( (J=5, 6 \) and \( 7, K=10) \) about those of \( (J=6, K=6 \) and \( 10) \) as shown above. The path lines for \( J=5 \) and \( 7 \) are perfectly symmetric. The path line \( (J=6, K=10) \) goes downstream and curls about the path line \( (J=6, K=6) \) spirally, whereas, the path lines \( (J=5 \) and \( 7, K=6) \) diverge to a position far away from the path line \( (J=6, K=6) \). Figures 10(a) and (b) correspond to the path lines \( (J=4, K=2, 4, 6, 8 \) and \( 10) \). These path lines are gradually curved from upstream depending on the velocity under the effect of the secondary flow, and downstream they show considerably complicated aspects as they move spirally. Figures 11(a) and (b) show the path lines in the neighborhood of the side wall \( (J=2, K=2, 4, 6, 8 \) and \( 10) \). These path lines except the path line \( (J=2, K=10) \) move to the middle gradually as they approach the bending region, and go downstream after they are sucked into the central part downstream of the corner. The path line \( (J=2, K=10) \), which is on the grid nearest to the side wall and outer wall at the inlet, moves towards the inner wall gradually during movement along the side wall from upstream, and then goes across the corner. After it flows into the central part during backflowing along the inner wall downstream of the corner, it goes downstream together with the main flow. In particular, at these two separation zones in the neighborhood of bending region, a new fluid from the main flow goes in continuously, and back to the main flow again after curling and backflowing there though it has a slow velocity. This indicates clearly that the separation zones are not isolated unlike the result of the two-dimensional analysis in the previous paper. As mentioned above, the complicated aspects of flow which cannot be considered by only velocity vectors, have been revealed.

3.3 Pressure contours
We show the pressure contours with solid lines at each section of the channel in Figs. 12 to 14. and the numerals written alongside the curved lines indicate the pressure coefficients \( (p-p_0)\sqrt{\rho g}/2 \), where \( p_0 \) is the standard pressure at a
position which is at the center of the cross section at the inlet. Figures 12(a) and (b) show the results for the sections of J=1 and J=3. Figs. 13(a) and (b) for the sections of K=1 and 11 upstream of the bending corner. Figs. 13(c) and (d) for the sections of K=1 and 11 downstream of it. and Figs. 14(a) and (b) for the sections of I=10 and 30 upstream and downstream of it.

The pressure contours are complicated in the neighborhood of bending region. The pressure drops abruptly at the inner bending corner, and rises abruptly reaching a great maximum at the impinging point. It also has a maximum at the side wall near the outer wall where the impinging flow goes aside and impinges again. Furthermore, at the inner wall against which its flow runs along the side wall. Except for these positions which have a pressure maximum, the pressure is almost uniformly distributed in the spanwise or x3 direction in spite of the bending region. And it is proved that there is a considerable pressure loss in the bending region.

3.4 Wall shear stress contours
In the three-dimensional flow, for example, the shear stresses which are on the surface normal to x3 direction and in x1 and x2 directions. are 
\[ \tau_{13} = \mu \left( \frac{\partial u_1}{\partial x_3} + \frac{\partial u_3}{\partial x_1} \right) \]
and 
\[ \tau_{23} = \mu \left( \frac{\partial u_2}{\partial x_3} + \frac{\partial u_3}{\partial x_2} \right), \]
and in particular at the wall 
\[ \tau_{13} = \mu \left( \frac{\partial u_1}{\partial x_3} \right) \]
and 
\[ \tau_{23} = \mu \left( \frac{\partial u_2}{\partial x_3} \right). \]
We define the magnitude of this resultant force \( \tau \) as 
\[ \frac{\bar{\tau}_{13} + \bar{\tau}_{23}}{\sqrt{2}} \]
and the corresponding wall shear stress coefficient as 
\[ \frac{\bar{\tau}}{\rho u_t^2/2}. \]
The contours of these wall shear stress coefficients at each wall are shown in Fig. 15. Figure 15(a) corresponds to the side wall of J=1. Figs. 15(b) and (c) to the inner wall of K=1 upstream and downstream of the corner. Figs. 15(d) and (e) to the outer wall of K=1. The magnitude and direction of velocity vector in the neighborhood of wall as shown in Fig. 15, are similar to those of this \( \tau \).

The wall shear stress has a minimum at the impinging point of the flow from upstream. Since its impinging flow runs in all directions and accelerates the fluid in the neighborhood of the wall, it has a very large value of maximum downstream of the impinging point and at the side wall near the outer wall, and also has a maximum at the inner wall near the side walls downstream and upstream of the outer bending corner. And it increases abruptly upstream of the inner bending corner where the flow is accelerated by inclining towards the inner wall. Furthermore, it has a minimum upstream of the outer bending corner and downstream of the inner bending corner which is the boundary between the separation zones and the main

NII-Electronic Library Service
flow. Comparing the wall shear stress distributions and the pressure distributions at the wall for the positions of the maximum, it is proved that there is a maximum position of the wall shear stress immediately downstream of the pressure maximum position. This shows that the pressure increases due to impinging with the wall and interception, and then the flow goes downstream while being accelerated along the wall, shows a maximum velocity and is reduced again. It is proved that the values of the wall shear stress are considerably large and the hydraulic loss due to friction is large.

4. Conclusions

With respect to the three-dimensional flow and heat transfer in a rectangular miter-bend, with 90° turn, an analysis of the flow in a laminar region for aspect ratio a=1 and Reynolds number Re=300 is made numerically by finite difference method. The results are summarized as follows:

(1) We divide the channel into two zones with overlapping parts for a miter-bend, and calculate numerically the whole channel by applying Patankar’s finite difference method to each zone and linking the values in the overlapping part.

(2) The effect of weight due to bending is made clear by expressing clearly the aspects of the flow in a miter-bend in terms of velocity vectors, path lines, pressure contours and wall shear stress contours.

(3) The complicated behaviors of the three-dimensional flow in the neighborhood of the bending region are made clear by showing in particular the aspects of the flow impinging with the opposite outer wall, the flow in the separation zones and the secondary flow at the section normal to the main flow.

(4) We show the changing of the pressure and the wall shear stress in the neighborhood of the bending region and in particular locate these maximum positions. We show that there are considerable pressure losses and large hydraulic losses in the bending region as a whole.

Acknowledgements

The authors used the computer FACOM M-200 of Nagoya University Computer Center to calculate the numerical values in this research and would like to express their sincere gratitude to the staff of Computer Center. They are also grateful to Mr. M. Hayakawa for his help in drawing the figures.

References