Characteristics of the Turbulent Flow Field
in a Wall Attachment Device*
(In the Case of No Control Port, One Output Port Closed)

By Yasuki NAKAYAMA**, Hirochiko OGINO***

On a simplified model with no control port and one output port closed, some numerical computations and visualizing experiments have been performed. In this calculation method, the flow field was divided according to local Reynolds number into three regions, that is, turbulent, transient and laminar regions, and the calculation method matched to each region was applied. Then, the finite difference method with $k-e$ model and Up-wind method were used for the turbulent region.

In the experiments, the velocity measurement using a 3-port Pitot tube and the flow visualization by tuft and tracer methods were performed.

The calculation results indicated good agreement with the experimental results, and the characteristics of the turbulent flow field in a wall attachment device and the validity of this calculation method were able to be clarified.

Key Words: Numerical Analysis, Finite Difference Method, Turbulence Fluidics, Wall Attachment Device, Flow Visualization Flow Measurement

1. Introduction

Fluidic devices do not have any mechanical moving part, and all the controlling actions are achieved by fluid motion. Therefore, the fluidic device have many advantages that they do not wear down and have excellent endurance, and they can stably act under severe conditions with electric noise, high temperature, gas, etc. Accordingly, the fluidic devices are used in many fields utilizing respective features.

Studies on the flow in a wall attachment device which is a basic component of fluidic devices have been performed experimentally and theoretically by many investigators. However, these studies are not sufficient because the flow in a wall attachment device is so complex that the theoretical investigations are very difficult.

At present, the theoretical studies on the flow in a wall attachment device are performed by theoretical analysis and numerical calculation. The report of Horikoshi** is a typical theoretical analysis, treating the flow in a wall attachment device as a non-viscous potential flow and analyzing it by using the transformation. As numerical calculations, there are the reports of Nakayama et al., Itoh et al., and Horikoshi et al., using the finite difference method, and the report of Sano* using the finite element method. However, these calculations were performed under the condition that the flow in wall attachment devices was laminar. A theoretical study which treated the flow in a wall attachment device as turbulent has not been undertaken.

On the other hand, there are many experimental studies on the flow in wall attachment devices. Itoh et al. paid attention to a load sensitive oscillator, and modeled the oscillating action using the relation between the switching motion of main jet and the pressure change in output ports. Hayashi et al. paid attention to a sonic oscillator, and modeled the oscillator characteristics using the relation between the control flow rate and control pressure. Abe* observed the dynamic switching motion of a wall attachment device by the flow visualization technique using dry ice smoke. However, the subject of these experimental studies was the global characteristics of a wall attachment device, and the state of inner flow of a device has not been sufficiently clarified.

The authors** had carried out the visualization of a flow in a wall attachment device using the hydrogen bubble method, but the flow was laminar, and the Reynolds number was not practical.

The purpose of this study is an analysis of the inner turbulent flow with practical Reynolds number in a wall attachment device with same form as an actual one by numerical calculation and
experiment, in the first stage of the study, the numerical calculation and experiment were performed for the wall attachment device of a simplified model with no control port and one output port closed, under the condition that Reynolds number was $10^4$. In this calculation method, the flow field was divided by the ratio of viscous force and inertia force. The experiments were the velocity measurement using a 3-port Pitot tube and the flow visualization using the tuft and tracer methods. By comparing the theoretical and experimental results, the reliability of this calculation method was confirmed, and from the results obtained, the state of the inner flow of the device was clarified.

2. Nomenclature

$b$: Width of supply port
$c_{1},c_{2},c_{3}$: Turbulent constants
$k$: Turbulent energy
$p$: Pressure
$Re$: Reynolds number ($Re=\frac{u_{m}b}{\nu}$)
$R$: Local Reynolds number ($R=\frac{Vb}{\nu}$)
$u, v$: Local velocity components
$\bar{u}$: Mean velocity at the inlet of supply port
$V$: Local velocity at any grid point
$x, y$: Coordinates
$\phi$: Vorticity
$\psi$: Stream function
$\rho$: Density
$\nu$: Kinematic viscosity
$\varepsilon$: Dissipation rate
$\delta_{x},\delta_{y}$: Turbulent constants
$\tau$: Turbulent flow
$m$: Mean
$\varepsilon$: Dissipation rate
$\Phi$: Vorticity
$t$: Time mean component
$s$: Turbulent component
$t$: Dimensionless symbol

3. Numerical Calculation

3.1 Basic equation

It was assumed that the flow in a wall attachment device was two-dimensional, isothermal and steady; the fluid was viscous and incompressible; and an external force did not exist. Under these conditions, the Reynolds equation which is a time-averaged momentum equation and the equation of continuity in turbulent flow are given by Eqs.(1) and (2).

$$\frac{\partial(G_{u})}{\partial x} + \frac{\partial(G_{v})}{\partial y} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x} + \frac{1}{\rho} \left[ 2\nu \frac{\partial \bar{u}}{\partial x} - \bar{u} \bar{u} \right] + \frac{\partial}{\partial y} \left[ \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) \bar{u} \bar{v} \right]$$

$$\rho \frac{d \bar{u}}{d \bar{t}} = -\frac{\partial \bar{p}}{\partial x} + \frac{\varepsilon}{\varepsilon + C_{1}} - \frac{1}{\rho} \frac{\partial}{\partial y} \left[ \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) \bar{u} \bar{v} \right]$$

(time-averaged)

where $\bar{p}$ is Reynolds stress terms $\bar{u} \bar{u}, \bar{v} \bar{v}, \bar{u} \bar{v}$ in Reynolds equation Eq.(1). Therefore, in order to obtain information on the time-averaged motion using Eqs.(1) and (2), the relation between the Reynolds stress and time averaged motion must be assumed.

3.2 $k-\varepsilon$ turbulent model

For the treatment of Reynolds stress, there are various methods, and respective corresponding turbulent model exist. In this study, the $k-\varepsilon$ turbulent model is adopted, because it is considered that the physical meaning, the generality of turbulent constants appearing in the model, precision and applicability are most excellent, and generally used.

The $k-\varepsilon$ turbulent model describes a turbulent flow by turbulent energy $k$ and dissipation rate $\varepsilon$, which are defined by Eqs.(3) and (4).

$$k = \frac{1}{\nu} \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right)^{2}$$

$$\varepsilon = \frac{1}{\nu} \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right)$$

Concretely, the following expression is given between Reynolds stress term and time-averaged motion by utilizing a turbulent viscous model of Boussinesq,

$$-\bar{u} \bar{v} = 2 \nu \frac{\partial \bar{u}}{\partial x}$$

$$-\nu v = \nu \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right)$$

$$\varepsilon = \frac{1}{\rho} \frac{\partial \bar{v}}{\partial y}$$

Where, $\nu$ is turbulent kinematic viscosity, and $\nu$ is turbulent constant, when $k$ and $\varepsilon$ are obtained from respective transport equations (7) and (8), Reynolds equation systems (1) and (2) can be closed using Eqs.(5) and (6).

$$\frac{\partial k}{\partial x} + \frac{\partial (k \varepsilon)}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{1}{\rho} \left( \frac{\partial \bar{u}}{\partial x} \right) \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right)$$

$$S_1 = \nu \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial x} + \frac{\partial (k \varepsilon)}{\partial y} = \frac{1}{\rho} \frac{\partial p}{\partial x} \left( \frac{\partial \bar{u}}{\partial x} \right) \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right)$$

$$S_2 = \nu \left( \frac{\partial \bar{u}}{\partial x} \right) \left( \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} \right) - \varepsilon$$

NII-Electronic Library Service
The symbols $C_u$, $C_1$, $C_2$, $C_4$, $C_6$ in Eqs. (6) and (8) are called turbulent constants. Many researchers proposed peculiar values regarding these constants. For example, Kawamura applied a two-equation turbulent model to a turbulent gas flow in a circular pipe which was severely heated, and determined the turbulent constants with which the solution satisfying a number of physical conditions was obtained. In the end, the method of carrying out calculation on the flow of as wide range as possible and determining the optimum value by trial and error is found the most general method. In this study, the values shown in Table 1 were cited from the report of Jones et al. as the reference for independently determining turbulent constants.

### 3.3 Flow field division

At first the authors tried the calculation of the total region of a wall attachment device using the basic equations in perfect form and the $k$-$\varepsilon$ turbulent model i.e. Eqs.(1) to (8). However, the unreal phenomena, that is, the negative values of $k$ and $\varepsilon$ in separation region, the vortex and low velocity regions in a closed output port, occurred, so that the calculation did not converge.

Then, a calculation method of dividing a flow field according to the relation of viscous force and inertia force was worked out. It was known by several examinations that the division of a flow field according to local Reynolds number $Re=Vb/\nu$ was most opportune for calculation. Figure 1 shows the conception of flow field division into the three regions named (a) turbulent region, (b) transient region and (c) laminar region. The transient region was so devised as to shift to the laminar region by weighting convection terms. The boundary values of $R$ determining each region were decided such that the calculation results of the width of main jet from a supply port, the width of separation vortex region, the position of reattachment point, etc qualitatively agreed with the experimental results obtained by the velocity measurement using a Pitot tube and flow visualization using the tuft and tracer methods.

(a) Turbulent region

In this region, the value of $R$ was greater than 1600. The inner flow of a supply port, main jet in diffuser region and the development region in an output port were assumed to be in the turbulent region, to which, the complete $k$-$\varepsilon$ turbulent model was able to be applied.

(b) Transient region

This is a region with the value of $R$ greater than 160 and smaller than 1600. In spite of the existence of disturbances, the flow velocity was very small as compared with main jet, and it was difficult to use the $k$-$\varepsilon$ turbulent model for the transient region. The separation region due to Coanda effect arising from a diffuser to an output port and the vortex region formed between main jet and a closed output port were assumed to be transient region. The convection terms of transport equations using the $k$-$\varepsilon$ turbulent model cannot estimate correctly the convection flux because the velocity is very small in this region.

Therefore, by using the local Reynolds number and boundary values $Rtu$ and $Rla$ for dividing the regions (a) and (b), and (b) and (c), the transported amounts $M_k$ and $M_\varepsilon$ of turbulent energy $k$ and turbulent dissipation $\varepsilon$ by convection were calculated by Eqs.(9) and (10), and used as the convection terms in Eqs.(7) and (8).

$$M_k=\beta \left( \frac{u_{av}}{\alpha} \right) \frac{\partial u_{av}}{\partial y}$$

$$M_\varepsilon=\beta \left( \frac{u_{av}}{\alpha} \right) \frac{\partial u_{av}}{\partial y}$$

where,

$$\beta = \frac{1}{R_{tu} - R_{la}}$$

(c) Laminar region

This is a region with the value of $R$ smaller than 160. The effect of molecular viscosity is greater than the effect of turbulent diffusion in the laminar region, where the fluid flow gradually stops in a closed output port. The $k$-$\varepsilon$ turbulent model can not be applied to this region, accordingly, the Reynolds equations (1) in which Reynolds stress terms are neglected are applied.

### 3.4 Change of basic equations to dimensionless form and finite difference form

The calculation was performed by upwind finite difference method which had been developed by Gosman et al. After Eq.(5) was substituted into Reynolds equation (1), the pressure terms

![Fig. 1 Flow field division](image_url)
were eliminated. Then, the stream function \( \psi \) and vorticity \( \zeta \) defined by Eqs.(12) and (13) were introduced.

\[
\begin{align*}
\psi &= \frac{\partial \alpha}{\partial x} - \frac{\partial \beta}{\partial y} \quad \text{--- (12)} \\
\zeta &= -\frac{\partial \beta}{\partial x} - \frac{\partial \alpha}{\partial y} \\n\end{align*}
\]

The vorticity transport equation (14) was obtained by the above operation.

\[
\begin{align*}
\frac{\partial}{\partial t} \left( \zeta + \nu \frac{\partial \zeta}{\partial y} \right) - \frac{\partial}{\partial y} \left( \frac{\partial \zeta}{\partial x} \right) &= \frac{\partial^2 \psi}{\partial x \partial y} \left( \frac{\partial^2 \psi}{\partial x^2} \right) + S_{\psi} \\
S_{\psi} &= -\frac{\partial^2 \psi}{\partial x \partial y} \left( \frac{\partial^2 \psi}{\partial x^2} \right) \quad \text{--- (14)} \\
\end{align*}
\]

When Eq.(12) was substituted into Eq.(13), Eq.(15) was obtained;

\[
O = \frac{\partial^2 \psi}{\partial x \partial y} + \zeta \quad \text{--- (15)}
\]

After substituting Eq.(12) into the transport equations (7) and (8) using the \( k - \varepsilon \) turbulent model, all the equations including Eqs.(14) and (15) were changed to dimensionless form. By using the width of a supply port \( b \) and mean velocity \( U_\infty \) as the base quantities, the dimensionless parameters are shown by Eq.(16).

\[
\begin{align*}
x^* &= \frac{x}{b} \quad y^* = \frac{y}{b} \quad u^* = \frac{u}{U_\infty} \\
\psi^* &= \frac{\psi}{b \nu U_\infty} \quad \zeta^* = \frac{\zeta}{\nu} \quad \xi^* = \frac{\xi}{\nu} \\
\end{align*}
\]

Reynolds number \( Re \) and turbulent Reynolds number \( Rt \) are defined by Eq.(17), using the width of a supply port \( b \) and the mean flow velocity \( U_\infty \).

\[
R_e = \frac{U_\infty b}{\nu} \quad R_t = \frac{U_\infty b}{\nu} \quad \text{--- (17)}
\]

In the above operation, the basic equations to be solved are Eqs.(18), (19), (20) and (21). However, symbol * showing dimensionless form is omitted hereinafter.

\[
\begin{align*}
\frac{\partial}{\partial t} \left( \frac{\partial \psi^*}{\partial y^*} \right) - \frac{\partial}{\partial x^*} \left( \frac{\partial \psi^*}{\partial x^*} \right) &= \frac{\partial^2 \psi^*}{\partial x^* \partial y^*} \left( \frac{\partial^2 \psi^*}{\partial x^*^2} \right) + S_{\psi^*} \\
S_{\psi^*} &= -\frac{\partial^2 \psi^*}{\partial x^* \partial y^*} \left( \frac{\partial^2 \psi^*}{\partial x^*^2} \right) + S_{\psi^*} \\
O &= \frac{\partial^2 \psi^*}{\partial x^* \partial y^*} + \zeta^* \quad \text{--- (18)} \\
\end{align*}
\]

\[
\begin{align*}
\frac{\partial}{\partial t} \left( \frac{\partial \zeta^*}{\partial y^*} \right) - \frac{\partial}{\partial x^*} \left( \frac{\partial \zeta^*}{\partial x^*} \right) &= \frac{\partial^2 \psi^*}{\partial x^* \partial y^*} \left( \frac{\partial^2 \psi^*}{\partial x^*^2} \right) + S_{\psi^*} \\
\end{align*}
\]

In these equations, the unknown parameters are \( \psi^*, \zeta^*, k^* \) and \( \epsilon^* \). The turbulent Reynolds number \( Rt \) is obtained by using Eq.(22).

\[
\frac{1}{R_t} = \frac{k^*}{\epsilon^*} \quad \text{--- (22)}
\]

The up-wind finite difference method which was developed by Gorman et al. was applied to Eqs.(18) to (21). Namely, each equation was area-integrated in a minute area \( n-e-s-w \) which was shown in Fig.2, and the up-wind method was applied to change the convection terms to difference form. The points \( n, e, s, w \) in Fig.2 are the central points between the grid points \( P-N, P-E, P-S \) and \( P-W \). When the variables represented by small letters are approximated by using the values of grid points shown by capital letters, finite difference equations are obtained.

Figure 3 shows the mesh system used. The calculation and experiment are performed under the condition that the Reynolds number is 10'.
3.5 Boundary condition
3.5.1 Inlet condition

The inlet conditions of all variables were given by the experimental data which had been obtained by the experiment using a two-dimensional channel of 100mm height, 500mm width and aspect ratio 5 shown in Fig.4. The measurement was carried out at 2700mm downstream of an inlet nozzle of which the contraction rate was 1/10, where the flow developed sufficiently, setting the Reynolds number at $10^7$. The mean velocity distribution, turbulent energy distribution and Reynolds stress distribution were calculated from the values obtained by using an isothermal type hot wire anemometer with an X type probe.

The turbulent kinematic viscosity was calculated by the second equation of Eq.(5) using the mean velocity gradient and Reynolds stress obtained by experiment. This time, since Reynolds stress $-u'v'$ and $-vv'$ were smaller than $-uu'$ in the case of a two-dimensional channel flow, Reynolds stress $-uu'$ was determined by experiment.

The dissipation rate was calculated by Eq.(6) using the turbulent energy obtained from experiment and the turbulent kinematic viscosity determined above.

Velocity distribution was determined by the 1/7th power law of Kármán and Prandtl, and stream function $\phi$ and vorticity $\zeta$ were calculated by Eqs.(12) and (13) using this velocity distribution. However, since the velocity distribution did not become smooth when the 1/7th power law was applied to the center point, the velocity in the vicinity of the center was calculated by a quadratic equation obtained by the least squares method using eight experimental data around the center. Figures 5(a), (b) and (c) show the inlet distributions of turbulent energy $k$, dissipation rates $\epsilon$ and velocities.

3.5.2 Wall condition

It was assumed that the velocity, turbulent energy and dissipation rate on the wall surface were zero. As the turbulent kinematic viscosity, the molecular kinematic viscosity was taken. The vorticity on the wall surface was extrapolated using the values of inner grid points.

As shown in Fig.6(a), when the grid point on the wall surface was denoted by $P$, and the point of one section inside was NP, by applying Taylor development to the stream function $\phi_{NP}$, $T$ became Eq.(23).

$$T_{NP} = \frac{1}{1 + \frac{1}{4} \left( \frac{T_{NP}}{T_{NP}} - \frac{T_{NP}}{T_{NP}} \right)}$$

On an oblique wall surface, Eq.(23) was used by neglecting the change in $x$ direction. On the wall surface in parallel with $y$ direction, the point NP was taken as Fig.6(b) shows, and the equation in which $x$ was substituted for $y$ in Eq.(23) was used.

3.5.3 Outlet condition

It was assumed that there was not a large change in the state of flow near an outlet port, and the first order approximation was made by using Eq.(24) from upstream grid points in figure 6(c).

$$\phi_{NP} = 2\phi_{NP} - \phi_{NP}$$

Equation (24) was applied to stream function, vorticity, turbulent energy and dissipation rate as the outlet equation.
3.6 Calculation procedure

The calculation was performed by the iteration method using successive relaxation method. That is, the value at \((i,j)\) grid point at nth time iteration was calculated by Eq.(25).

\[
\phi^{n+1} = \phi^{n} + \alpha \left( \phi^{n} - \phi^{n-1} \right) \quad \cdots (25)
\]

By changing the symbol \(\epsilon\) to \(\varphi\), \(\xi\), \(k\) and \(e\), each parameter was able to be calculated. The symbol \(\varphi^m\) indicates a value obtained by finite difference equation. The values shown in Table 2 were used as the relaxation parameters \(\alpha\). Convergence was decided by Eq.(26). This equation means that when the total sum of errors at all grid points is smaller than 0.15, the calculation converges.

\[
\sum |\varphi^m - \varphi^{m-1}| < 0.15 \quad \cdots (26)
\]

When all parameters satisfied Eq.(26), the calculation was judged as converged.

4. Experimental Method

4.1 Wall attachment device used for experiment

Figure 7 shows the wall attachment device used for the experiment. The shape was same as that for the numerical calculation shown in Fig.3, and the various particulars are as follows.

Wall attachment device
  (without control ports)
  Offset length : 2.5mm
  Splitter distance : 88.6mm
  Angle of side wall : 15°
  Width of supply port : 10mm
  Aspect ratio : 30

Fluid enters a supply port through a tank and a bellmouth with contraction ratio of 1/10. The length of the supply port is 40cm, so that the device is given a flow with completely developed velocity distribution. The one side output port is closed.

The device is a sandwich structure made of aluminum and acrylic plates. It can be used for either measurement or flow visualization. There are holes for insertion of the probe on the side wall surface as shown in Fig.7.

<table>
<thead>
<tr>
<th>Table 2. Relaxation factors</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\alpha_v)</td>
</tr>
<tr>
<td>1.0</td>
</tr>
</tbody>
</table>

4.2 Experimental setup

4.2.1 Experiment with water

Figure 8 shows the system of experimental setup. Water pumped up from a lower tank by a vortex pump was supplied into the wall attachment device through the orifice for mixing tracer and flow regulating valves. Overflowing water from the device returned to the lower tank through the flux controlling tank. Flow rate was set by the measurement of discharge from the flux controlling tank using the weight method.

4.2.2 Experiment with air

Figure 9 shows the system of experimental setup. Compressed air enters a tank, then, it is supplied to the device. Flow rate is set by the Pitot tube in the device, and controlled by two valves in a pipeline.
4.3 Experimental procedure

4.3.1 Pitot tube

The velocity measurement was performed by using a 3-port Pitot tube. The 3-port Pitot tube was of circular type with 6mm diameter. The pressure holes were provided 10mm above the bottom of the probe. Pressure was measured using a Göttingen type manometer. It was confirmed prior to the use that the coefficient of the Pitot tube did not change with the variation of Reynolds number Re. The Pitot tube was inserted into the device through the slit plate installed on the top of the device.

4.3.2 Tuft method

The tuft was made of a cotton thread of 5mm length, and the support was a tungsten wire of 0.5mm diameter, to which beads of 2mm diameter were attached free to rotate. The tuft was inserted into the device through a probe hole on the side wall.

4.3.3 Tracer method

Tracer was polyethylene-latex particles, of which the mean diameter was 200μm. Tracer was put into tracer tank, and when the valve to an orifice was opened, the tracer was absorbed into the flow by the negative pressure effect of the orifice.

5. Results and Examination

Figure 10 shows the division of turbulent, transient and laminar flow fields.

In this calculation method, the convection terms in the transport equations of k-ε turbulent model were weighted in the transient region, but since flow velocity is small there, the method of putting the convection terms equal to zero is conceivable. On the other hand, when the authors had employed the latter calculation method, the reattachment point of separated vortices due to Coanda effect shifted farther upstream side than the experimental results, and about 240 times of repetitions were needed up to the convergence. However, by the weighted method, it was known that the calculation results agreed well with the experimental results regarding the reattachment point and calculation converged after about 100 times of repetitions.

In the following calculated and experimental results, the Reynolds number is 10^7.

5.1 Stream lines and flow pattern

Figure 11 shows the stream lines obtained by numerical calculation and the flow pattern obtained by tuft method. It is known from this figure that there are the stream lines with the same value of the stream function as that on wall surface at the boundary of main jet and closed output port side, the boundary of main jet and separation vortices, and inside a closed output port, and the closed regions are formed, respectively.

Figure 12 shows the flow pattern obtained by tracer method. The triangle symbol in this figure indicates the reattachment point.

It is known from these figures that the calculated results of the width of separation vortices, the position of a reattachment point, the scale of large vortices arising in a closed output port agree with the experimental results.

It is known from Figs.11 and 12 that the main jet from a supply port divides the flow areas into the separation vortices which begin at the step of diffuser inlet and the vortex region arising on closed output port side. Moreover, from Fig.12, it is known that there is a large difference of velocity between main jet and vortices on closed output port side, separation vortices, a closed output port and so on.
5.2 Equi-vorticity line

Figure 13 shows the equi-vorticity lines obtained by calculation. It is known from this figure that the equi-vorticity lines are distributed closely at the step of diffuser entrance region and the tip of a splitter. Therefore, it is likely that the shearing force will be large at these portions.

5.3 Turbulent energy and dissipation rate distribution

Figure 14 shows the turbulent energy distribution, and Fig.15 shows the dissipation rate distribution, each obtained by calculation. It is known from Fig.14 that the turbulent energy is large in diffuser region as compared with supply port and output port regions. Therefore, it is shown that the disturbance in diffuser region is large.

The dissipation rate of turbulent flow is the loss of turbulent energy as heat per unit time due to viscosity, and it is defined by Eq.(27).

$$\epsilon = -\frac{dk}{dt}$$  \hspace{1cm} (27)

It is known from Fig.15 that the dissipation rate distribution is similar to the turbulent energy distribution, and it is large also in the diffuser region as compared with the other regions.

It follows from these figures that main jet and its neighboring fluid affect each other in diffuser region, and are largely disturbed, therefore, the transport of heat and momentum is carried out actively.

5.4 Velocity distribution

Figure 16 shows the velocity distribution. Solid lines show the synthesized velocity distribution obtained by the calculation results, and the circle show the velocity obtained by the measurement using a 3-port Pitot tube.

It is known from this figure that the numerical calculation results agree well with the experimental results. However, velocity is not able to be measured in the separation vortex region and closed output port side because the velocity is very small in these regions.

5.5 Reynolds stress distribution

Figure 17 shows the distribution of Reynolds stresses $\langle u'v' \rangle$ obtained by calculation. Reynolds stress was calculated by the 2nd equation of Eq.(5) using turbulent viscosity and velocity gradient, each obtained by calculation.

It is known from this figure that the Reynolds stress distribution is similar to the turbulent energy distribution and dissipation rate distribution. From the

Fig. 13 Equi-vorticity line

Fig. 14 Turbulent energy distribution

Fig. 15 Dissipation rate distribution

( $U$ : Local velocity at any point )

Fig. 16 Velocity distribution

Fig. 17 Reynolds stress distribution
fact that the Reynolds stress is large in diffuser region as compared with other regions, supposing that the Reynolds stress gets energy from main flow and supplies it to a turbulent flow, a large disturbance ought to exist in diffuser region.

6. Conclusions

The numerical calculation, experiment and flow visualization of turbulent flow in a wall attachment device which had no control port and closed one side output port were performed.

As the result, the following conclusion is obtained.

(1) A calculation method has been developed, in which the flow field is divided according to the ratio of viscous and inertia forces, i.e. local Reynolds number. The turbulent region is calculated by the $k\varepsilon$ turbulent model. The transient region is calculated by the transport equation of the $k\varepsilon$ turbulent model, in which the convection terms are weighted so as to transfer to laminar region. The laminar region is calculated by the Reynolds equation, from which Reynolds stress terms are eliminated. It is known that this calculation method is able to be applied to several complex turbulent flows with good convergence, and the flow in wall attachment devices under various boundary conditions.

(2) The results of numerical analysis agreed well with the experimental results obtained by the velocity measurement using a 3-port Pitot tube and the flow visualization by the tuft method using air and the tracer method using water, and the validity of this calculation method was confirmed.

(3) The stream lines, the distributions of velocities, turbulent energy and dissipation rates, equi-vorticity lines and Reynolds stress distribution at practically used Reynolds number were able to be obtained from the results of this analysis. In this way, the characteristics of wall attachment devices have been clarified.

The numerical calculation was performed with the UNIVAC 1100/61E system in Tokai Electronic Computer Center, and the iteration counts of about 100 times were required for the convergence of the calculation.

References