1. Introduction

In the field of fluid dynamics, the prediction of flow information basically depends on two kinds of approaches. One is the experimental investigation. The other is the computational fluid dynamics (CFD)\textsuperscript{11}.

The most realistic information concerning fluid flow is, in principle, given by direct measurements. However, the experimental tests are usually very expensive, and some measurements are difficult to obtain in many situations. These drawbacks have heavily limited many practical applications of the experimental investigation.

During the last two decades, there has been a rapid development in computer science. Besides, numerical algorithms for solving physical problems on the computer have also been developed to be much more accurate. Owing to the advancement in these two areas, the complex fluid behavior is frequently modeled and predicted by numerical techniques. These techniques involve solving equations that approximate the physical laws governing a flow.

The physical aspects of any fluid flow are governed by three fundamental principles: (1) mass conservation; (2) Newton's second law; and (3) energy conservation. These fundamental physical principles can be expressed in terms of basic mathematical equations, which in their most general form are either integral equations or partial differential equations such as,

- Continuity equation
  \[
  \frac{D \rho}{Dt} + \rho \nabla \cdot \mathbf{V} = 0
  \]

- Navier-Stokes equation
  \[
  \frac{D \mathbf{V}}{Dt} = \mathbf{F} - \frac{1}{\rho} \mathbf{\nabla} P + \nu \nabla^2 \mathbf{V} + \frac{1}{3} \mathbf{\nabla} (\mathbf{\nabla} \cdot \mathbf{V})
  \]

- Energy equation
  \[
  \frac{D T}{Dt} = \frac{1}{\rho} \mathbf{\nabla} \cdot \mathbf{\nabla} T
  \]

where

\[
\mathbf{\nabla} = i \frac{\partial}{\partial x} + j \frac{\partial}{\partial y} + k \frac{\partial}{\partial z}
\]

\[
\mathbf{\nabla}^2 = \frac{\partial^2}{\partial x^2} + j \frac{\partial^2}{\partial y^2} + k \frac{\partial^2}{\partial z^2}
\]

\[
\frac{D}{Dt} = \frac{\partial}{\partial t} + u \frac{\partial}{\partial x} + v \frac{\partial}{\partial y} + w \frac{\partial}{\partial z}
\]

Here

- \(V\) : velocity of the fluid
- \(P\) : pressure
- \(T\) : temperature
- \(F\) : external force
- \(x,y,z\) : axes of Cartesian coordinate system
- \(u,v,w\) : velocity components in \(x,y,z\) directions
- \(t\) : time
- \(Pr\) : Prandtl number
- \(\nu\) : kinematic viscosity
- \(\rho\) : density

The so-called CFD approach is the art of replacing the integrals or the partial derivatives (as the case may be) in these equations with discretized algebraic forms, which in turn are solved to obtain numbers for the flow field values at discrete points in time and/or space.

CFD solutions generally require the repetitive manipulation of many thousands, even millions, of numbers, a task that is humanly impossible without

\[\text{Vol.9 No.5}\]
the aid of a computer. Storage capacity and execution speed of a computer are the limiting factors for CFD when dealing with more detailed and sophisticated fluid problems. In today’s CFD, computational simulations of various fluid phenomena are still time-consuming even with using the high-speed digital computers.

In CFD methodology, grid generators, solution methods, and algorithms have streamlined the computational part of obtaining a solution. However, much of the reasoning involved in geometry definition, discretization, parameter adjustment, data format, code execution, graphical display, and intermediate solution assessment, is sensitive to error and still requires some experience or expertise [3-4]. Therefore, it has become a rate-limiting step in the process of obtaining a solution. Indeed, doing a CFD simulation is generally a complex job even for a well-trained engineer. To successfully carry out CFD calculations, a great deal of human and computer resources are still needed.

Due to the complexities of CFD theory and of the numerical techniques, there has been a great demand for developing an easy-to-use flow prediction tool for engineering application purposes. In the past years artificial neural network has been extensively investigated in various areas because of their favorable features, i.e., nonlinearity and learning ability [5-8]. However, applying neural network to CFD problems has not yet become widespread. This is mainly due to the difficulty of effective use of this method and to the inherently high nonlinearity of flow problems.

In this study, we investigated the potential of using artificial neural network for predicting flow patterns. By combining neural networks with fundamental theory of fluid dynamics and discrete idea in CFD approach, a reasoning procedure was constructed. Owing to the characteristic of the problem in this study, three structured neural networks [6-8] were employed to complete the flow pattern estimation. Each network is responsible for estimating the flow patterns on one region of the flow field. The three networks worked as the action side of a qualitative rule in the reasoning procedure for predicting the whole flow patterns. By conducting the reasoning procedure in this study, it was found that the calculation is less time consuming and the estimation accuracy is very encouraging. The advantage of the proposed reasoning procedure is very useful in engineering application, especially in the cases where a rough but readily available result will be more appreciated than an accurate but time-consuming CFD solution.

2. The test problem

In this paper, we examined the phenomenon called Kármán vortex street. The typical flow patterns of the phenomenon are shown in Fig.1.

![Fig.1 Typical flow patterns of Kármán vortex street (from Homann's experiment in 1936)](https://example.com/fig1)

The behavior of Kármán vortex street can be summarized as: when a cylinder or a prism is placed in a uniform moving fluid, two staggered parallel rows of counterrotating vortices are formed behind it. Each row contains an infinite number of line vortices. Any other vortex in the same row moves with the same velocity so that the vortex configuration remains unaltered at all times.

Fig.1 shows that the Reynolds number (Re) gives a great influence on the Kármán vortex street. The Reynolds number is an important dimensionless parameter in fluid dynamics, which is defined as:

\[ Re = \frac{U_\infty L}{\nu} \]  

(7)

where \( U_\infty \) is the freestream velocity, \( L \) is the reference length, and \( \nu \) is the kinematic viscosity of the fluid.

Currently, the flow is assumed to be two-dimensional. The geometry and coordinate system of this problem are shown in Fig.2. A prism with elongated rectangular cross section is fixed in the flow field between two parallel walls. For the present objective, the Reynolds number is expressed as,

\[ Re = \frac{U_{\text{side}} H}{\nu} \]  

(8)
where $U_{inlet}$ is the flow velocity in inlet and $H$ is the height of the prism. Since the flow is incompressible, the kinematic viscosity $\nu$ is a constant in eq.(8). Therefore, $U_{inlet}$ is decided by the Reynolds number. For a two-dimensional flow, the flow velocity at any point over the flow field is represented by two velocity components $u$ and $v$, where $u$ is the horizontal velocity component in $X$-direction, and $v$ is the vertical velocity component in $Y$-direction.

3. The reasoning procedure

Like in CFD methodology, we used a structured mesh to discretize the flow field into many cells, as shown in Fig.3. Each cell, which stands for a control volume in the flow field, is made very small. Therefore, the vector sum of the four velocity components relating to each cell represents the velocity of the cell. According to the mass conservation law, the net mass flow out of any control volume through its surface should equal the time rate of decrease of mass inside that control volume. On the assumption that the fluid is incompressible, we obtain the following equation for each cell.

$$\sum_{i=1}^{m}(u_{i,j} - u_{i,j+1}) dy + (v_{i-1,j} - v_{i,j}) dx = 0$$

where $dy$ is the width of the cell in $Y$-direction, and $dx$ is the width of the cell in $X$-direction.

From eq.(9) it is clear that any one of the four velocity components in each cell can be quantitatively expressed by the rest three components in the same cell. In other words, to calculate the velocity of each cell, only three velocity components need to be decided. Owing to the structured mesh in Fig.3, it is easy to understand that two out of four velocity components have already been known in each cell. Therefore, to calculate the velocity of each cell, one more component needs to be decided. Fig.3 also shows that two adjacent cells share one velocity component except $u_{i,1}$ ($i=1, \ldots, m$). On account of this, it is possible to construct a quantitative velocity relationship among all cells from inlet to outlet.

In this study, artificial neural networks are used to supplement estimated velocity component for each cell. These neural networks are used to memorize the nonlinearity relationship between horizontal velocity profiles on two adjacent cross sections. The reasoning procedure of this method is shown in Fig.4.

In order to get the perception of the horizontal velocity distributions on cross sections in the flow field, a sensitivity study was conducted as shown in Fig.5. From Fig.5 it is noted that variation among these profiles in different areas over the flow field is significant. In addition, these profiles show a similar trend of changing under different Reynolds numbers.

Due to the characteristic of variation of horizontal velocity profiles, in order to do the neural computing easily, we divided the flow field into three portions, and used three structured neural networks for these areas, respectively. The three areas are (a) the upstream region of the prism, (c) the downstream region of the prism and (b)
4. The neural networks model

Each network in Fig.6 is a three-layer feedforward neural network. The learning algorithm for these networks is backpropagation (BP) algorithm. In this study, the active function used in hidden layer of each network is the hyperbolic tangent function, then

\[
f(x) = \frac{1 - e^{-\xi x}}{1 + e^{-\xi x}}
\]

where \( \xi \) specifies the steepness of the curve. To avoid the compression of extreme values, the identity function is used in output layer of each network. The fundamental architecture of each network is

Fig.5 The sensitivity study about \( u \) profiles in different areas over the flow field

Fig.6 The structured neural networks
the same as each other, which is shown in Fig 7. For the network (a), its index \( j \) is defined from 1 to 25 to correspond to the region (a). Similarly, the index \( j \) for the network (b) is defined from 25 to 86 to correspond to the region (b), and the index \( j \) for the network (c) is defined from 86 to 150 to correspond to the region (c).

As shown in Fig.7, in input layer of each network, 79 out of 82 neurons represent the \( u_{ij} (i = 1, \ldots, 79) \) velocity components along the jth cross section. The rest three neurons correspond to the Reynolds number, the index of the jth cross section and the phase of vortex generation \( \theta \), respectively. Here the phase of vortex generation \( \theta \) is defined as the time series within one vortex-creating cycle. In this paper, the vortex-creating cycle means the time used for generating one vortex from one side of the prism. In the output layer of each network, all the neurons correspond to the \( u_{ij+1} (i = 1, \ldots, 79) \) velocity components along \((j+1)\) downstream cross section. The number of neurons in the hidden layer of each neural network is decided through preliminary simulations. Table 1 shows the number of hidden layer nodes and the parameters of BP algorithm in each network.

In this study, the number of grids along each cross section is 79. However, for other flow problems, the grids used may greatly exceed this number and thus result in a huge amount of time for network training. To avoid this problem, we can loosen the mesh over the flow field while holding the flow characteristics simultaneously by satisfying the mass conservation law.

5. Computational results

In this research, four sets of precalculated CFD solutions for different Reynolds numbers \((Re = 250, 340, 590, 800)\) are used to train our networks. From our experience, other Reynolds number groups which differ slightly from the present one will not influence the estimation accuracy significantly. Within each set of these CFD solutions, there are nine subsets of results representing nine different phases of vortex generation. The CFD solutions for Reynolds number 470 are used as evaluating data in this study.

The calculated results of horizontal and vertical velocity distributions obtained by the proposed reasoning procedure are compared with those by CFD calculations, as shown in Fig.8 and Fig.9. In the upstream region of the prism, due to the impediment of the prism, the running flow must divide into two side streams when the flow reaches to the vicinity of the front edge of the prism. Here \( u \) velocity components are nearly zero, which in turn results in the increment of \( v \) velocity components. This can be seen clearly in either the CFD solutions or the estimation results shown in Fig.8(a) and Fig.9(a). For the area where the prism locates, since the width of the flow path is reduced, flow velocity is strengthened here according to the mass conservation law. In Fig.8(b), a strong increment of \( u \) gradient \((\partial u/\partial y)\) in the region adjacent to the upper and lower surface of the prism, is observed. This is due to the strong viscous effect in the liquid-solid boundary. The estimation results in Fig.8(b) and Fig.9(b) show these characteristics clearly, and the estimation errors are tolerable comparing with the CFD solutions. In the downstream region of the prism, due to the recovery of the width of the flow path, two side streams will interfere with each other in this region, causing the generation of vortex there. This behavior, expressed by local peaks in the velocity profiles, is captured by both the CFD solutions and the estimation results, as shown in Fig.8(c) and Fig.9(c).

The flow patterns in form of the velocity vector, obtained by both the CFD simulation and the

![Fig.7 The architecture of each neural network](image-url)

**Table 1** Parameters of BP algorithm and the number of hidden layer nodes in each network

<table>
<thead>
<tr>
<th>Networks</th>
<th>Number of nodes</th>
<th>Learning rate</th>
<th>Momentum rate</th>
<th>Slope tanh</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>19</td>
<td>0.0012</td>
<td>0.33</td>
<td>1</td>
</tr>
<tr>
<td>(b)</td>
<td>19</td>
<td>0.0011</td>
<td>0.33</td>
<td>1</td>
</tr>
<tr>
<td>(c)</td>
<td>19</td>
<td>0.0012</td>
<td>0.34</td>
<td>1</td>
</tr>
</tbody>
</table>
Application of Structured Artificial Neural Networks to Computational Fluid Dynamical Problems

Fig. 8 The comparison of the horizontal velocity distributions on different cross sections between the proposed reasoning procedure and CFD calculation (Re=470, θ=π)

Fig. 9 The comparison of the vertical velocity distributions on different cross sections between the proposed reasoning procedure and CFD calculation (Re=470, θ=π)

Fig. 10 The flow pattern obtained by the proposed reasoning procedure (CPU time: 10 s)

Fig. 11 The flow pattern obtained by CFD simulation (CPU time: 480 h)

The proposed procedure, are shown in Fig. 10 and Fig. 11, respectively. Comparing with the CFD solution, the reasoning procedure successfully predicts the dramatic change of the flow in the region directly behind the prism. Furthermore, the far wake development of the Kármán vortex street in the downstream region obtained by the proposed procedure is also in good agreement with the CFD solution.

So far we have discussed the results at θ=π. To further explore the capability of the present method in terms of predicting time-dependent flow behavior, we also examined the case at θ=3π/2 and found that the reasoning result is consistent with CFD solution. Due to the limitation of space, the result at θ=3π/2 is not shown in this paper. When we carried out the calculation of this problem on a
DEC Alpha workstation (233 Hz), about 480 hours of CPU time are required by CFD simulation. However, by using the reasoning procedure proposed in this study, only 10 seconds are needed to get a relatively correct result.

6. Conclusion
This study demonstrated the potential of using structured artificial neural networks for flow pattern estimation. It inspired us with confidence that the fluid dynamic problems may be easily solved by using neural network when the mass conservation law (continuity equation) is considered, thus the usual bottleneck in CFD—dealing with the complex high nonlinear Navier-Stokes equation can be avoided. For realizing the reasoning procedure in this investigation, the sensitivity study of the horizontal velocity profiles on several cross sections over the flow field was conducted. On account of the characteristic of variation of these profiles, to conduct the neural computing easily, three structured neural networks were employed to carry out the flow pattern estimation. They worked as the action side of a qualitative rule in the reasoning procedure. Since some fundamental rules of the fluid dynamics are used, the proposed reasoning procedure is in principle applicable to other kinds of flow problems. However, in those cases some modifications in the neural network model must be considered to account for the new characteristics of those flow patterns. Comparing with the CFD solutions in this study, the reasoning accuracy is satisfactory. Furthermore, the proposed reasoning procedure can give a prompt answer compared with the time-consuming CFD simulation. It was found that this method is an easy-to-use and engineering-acceptable tool.

REFERENCE

(1996年12月18日 受 付)
(1997年 7月18日 再受付)

[問い合わせ先]
〒321 栃木県宇都宮市石町2753
宇都宮大学工学部都市理工学研究室
張 蕾 (Lei ZHANG, ちょうらい)
TEL：028-689-6032
FAX：028-689-6043
E-mail：zhang@mech.utsunomiya-u.ac.jp
Application of Structured Artificial Neural Networks to Computational Fluid Dynamical Problems

by

Lei ZHANG, Mitsunobu AKIYAMA, Hitoshi SUGIYAMA and Nao NINOMIYA

Abstract:
Experimental and computational fluid dynamics (CFD) approaches are now orthodox means in flow prediction. However, in some engineering applications where these approaches are used as a design tool, their expensive and time-consuming nature may hamper the process of reaching specialists' final goal. In this study, we investigated the potential of applying structured artificial neural networks to fluid dynamical problems. A typical hydraulic flow phenomenon, the Kármán vortex street was examined here. For realizing the reasoning procedure in this investigation, the sensitivity study of the horizontal velocity profiles on several cross sections over the flow field was...
conducted. Based on the sensitivity study, three structured neural networks were employed to carry out the flow pattern estimation. They were modeled as the action side of a qualitative rule to work in the reasoning procedure. Compared with the computational fluid dynamical solutions, the estimation accuracy is very encouraging. Furthermore, the proposed reasoning procedure can give a prompt answer compared with those time-consuming conventional approaches.

**Keywords**: Structured Artificial Neural Network, Computational Fluid Dynamics, Sensitivity Study, Kármán Vortex Street

**Contact Address**: Lei ZHANG

*Dept. of Production and Information Science, Graduate School of Engineering, Utsunomiya University*

2753 Ishii-Machi, Utsunomiya 321, Japan

TEL : 028-689-6032
FAX : 028-689-6043
E-mail : zhang@mech.utsunomiya-u.ac.jp