Numerical Wind Tunnel (NWT) Project

Toshiyuki IWAMIYA,*) Takashi NAKAMURA and Masahiro YOSHIDA

National Aerospace Laboratory
7-44-1 Jindaiji-Higashimachi, Chofu 182, Japan

§ 1. Introduction

Computational Fluid Dynamics (CFD) has been one of the important application fields promoting the development of high speed computers. Turbulence simulation was carried out on a monumental parallel computer ILLIAC IV. At National Aerospace Laboratory (NAL), researchers also started research on CFD in 1960's. Since then we have introduced high-end commercial machines including the so-called supercomputers and have been contributing to the progress of CFD. In the meanwhile, the physical models have been changed from the potential equations to the Navier-Stokes equations and the applicability has been drastically extended. But we realized that the performance of commercial supercomputer would be not enough to apply the CFD technology to the design of airplanes or spaceplanes and to enhance the accuracy of simulation results for turbulence-dominated flows.

Hence, we have been pursuing the development of a CFD-oriented high speed computer since 1986 to promote CFD research as well as its application to real world problems especially in the field of aeronautics and space technology. Our approach is based on the experience of vector computers and models derived from typical computational algorithm used in CFD. We also lay a stress on the sustained performance for the CFD calculations as well as the usability and manageability in the establishment of the NWT concept. The NWT was realized through a joint research and development with Fujitsu Co. and put into operation in 1993.

In this paper, we discuss our approach to figure out the NWT concept and describe the system configuration and some features of the NWT FORTRAN. Then we show some performance evaluation results and finally we also display some examples of CFD research activities done at NAL with the NWT.

§ 2. Motive and the target of NWT project

In February 1987, we introduced a supercomputer Fujitsu VP-400. Since then, the Lab's CFD researchers rushed into three-dimensional viscous flow simulation. The VP400 displays the peak performance of 1.14 GFLOPS and has main memory of 1GB. The objects are airplanes, spaceplanes and their propulsion systems. Most of

*) Speaker.

Table I. Grid points and CPU time run on VP400.

Grids (Kpts)	CPU time (hours)	
2500	30	STOL"ASKA", Transonic Flow
1300	15	Civil Transport, Transonic Flow
400	10	"HOPE", Hypersonic Flow, Real Gas Effect
180	30	"OREX", Hypersonic Flow, Real Gas Effect
500	5	Spaceplane, Supersonic Flow
500	10	Turbine Cascade with Tip Clearance
200	50	SCRAM Jet Engine, Hydrogen Fuel

such flow calculations are based on Reynolds-averaged Navier-Stokes equations with a turbulence model. These simulations are referred to in the following as RANS simulation.

In 1986, NAL started an in-house investigation for the desirable high-speed computer system for the development of CFD technology.

Table I sums up a number of grid points and CPU time on the VP400 for some examples of RANS simulation.

CFD simulation results for rather simple configurations display a good agreement with experimental data as far as a cruising speed is concerned and we can say CFD reached a stage where they could be utilized for design. However, there remain many problems to settle in terms of the accuracy for complex flow fields and the data productivity. They are all caused by the insufficiency of computer performance and main memory capacity.

2.1. *Required performance for CFD research*

We made the following rough estimates:

1. The RANS simulations with clean aircraft configurations consisting of fuselage, main wing and tail wings need about 1 million grid points on the average. It takes about 10 hours of CPU time on the VP400. Accordingly, parametric studies by using RANS simulation seem almost impossible, which are key to the development of aircraft. It is said that around 10 minutes is reasonable amount of time to allow CFD to be used for parametric studies for aircraft design.
2. 5 million to 15 million grid points are required for the RANS simulation with complete aircraft configurations with engine nacelles and control surfaces. Once that simulation capability dealing with such complex configurations within a practical amount of time is available, CFD could become an effective tool or a replacement of wind tunnels for the technological development.
3. CFD at present stage cannot yet produce reliable data for flow where turbulence phenomena are dominant. For these purposes, progresses in turbulence modeling should be made. Large Eddy Simulation or Direct Simulation are believed to be a candidate to treat the turbulence more accurately to establish turbulence

models. They require calculations with at least 100 million to 150 million grid points.

We intended to develop a computer one or two hundred times as fast as the VP400 to meet these requirements by early 1990's because this machine was to replace the VP400.

2.2. *Establishment of NWT concept*

First of all, there seemed to be a limit of improvement of a vector supercomputer machine cycle, probably about 1 GHz. Hence a parallel processing is inevitable. But since automatic parallelization technology is far from mature, we cannot expect to develop a high speed general-purpose parallel computer. Accordingly, it is critical to analyze the algorithms in the application fields concerned and to establish a basic computation model.

Since 95% or more of the numerical simulations at NAL was focused on the CFD technology, we paid the most attention on CFD. Through investigation into CFD algorithms, we made computation models on the ground that the target computer runs efficiently even for implicit methods especially the Implicit Approximate Factored (IAF) method which is widely appreciated. Essentially the IAF method solve iteratively the following equations:

$$L_x L_y L_z \Delta Q^n = RHS_n ,$$

$$Q^{n+1} = Q^n + \Delta Q^n ,$$

where Q is a vector of physical quantities and L_x, L_y and L_z are difference operators restricted to x, y and z-directions respectively. Figure 1 shows a typical flow of 3 dimensional viscous flow simulations with the IAF method.

First, we considered that peak performance of single processors would be at most 10 GFLOPS. Hence if one tries to make the target supercomputer, one needs 20 or more processing elements

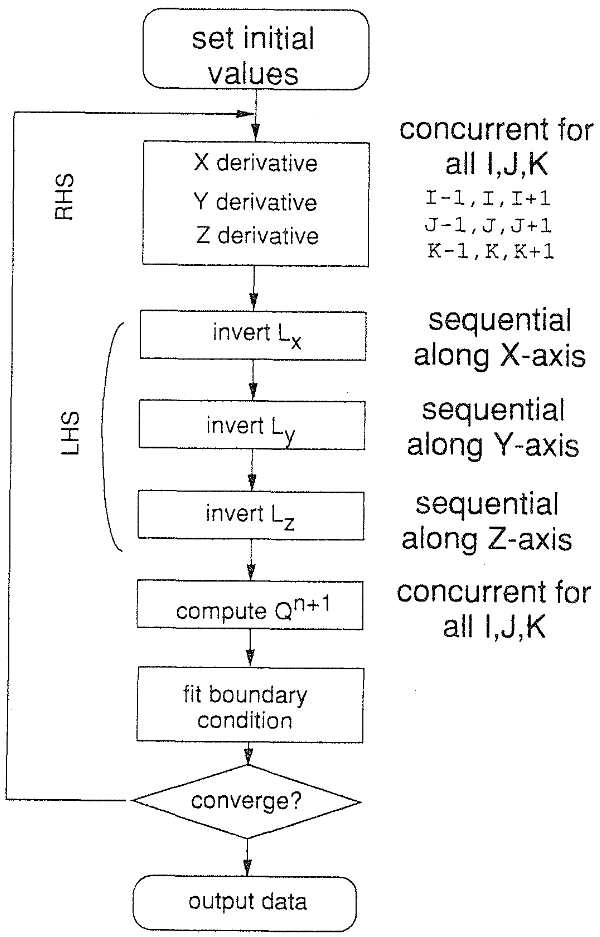

Fig. 1. Flow of IAF method.

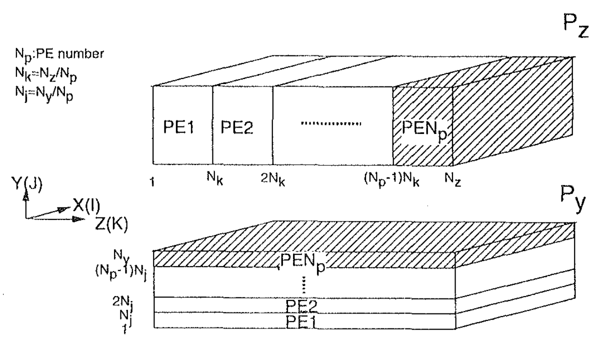

Fig. 2. Decomposition of data along z-direction P_z.

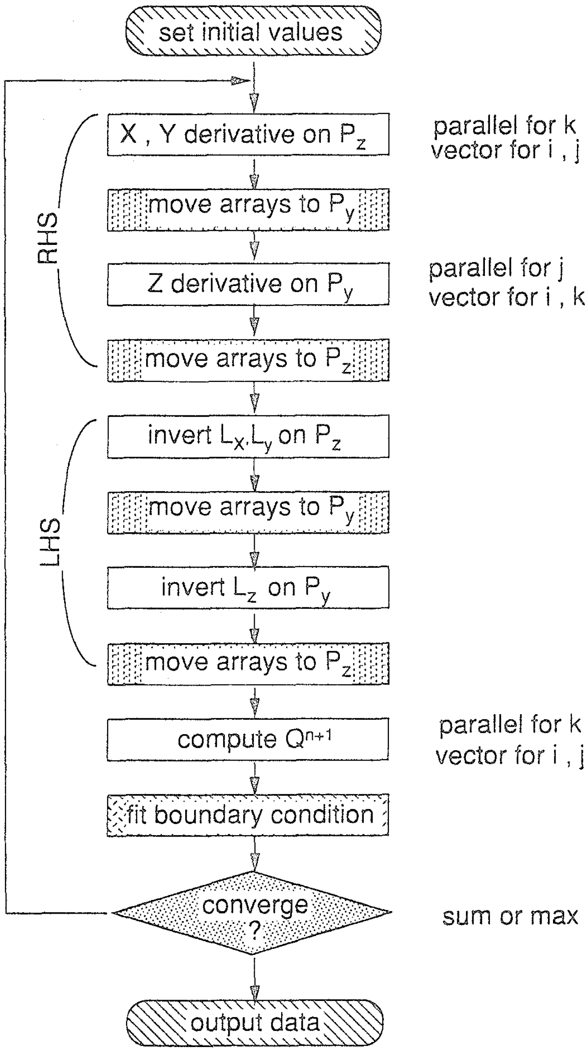

Fig. 3. Parallel version of IAF method.

(PEs). Since it is difficult to connect such number of high speed computer with shared memory and to ensure the necessary data transfer rate, distributed memory should be adopted.

Then the efficient parallel processing for three inversions in the IAF method is critical. It should be noted that 3 dimensional simulation with structured grid allows concurrent computations along at least 2 directions. Hence one direction can be used for vectorization and another for parallelization. For example, the inversion of L_x can be done concurrently along y- and z-directions. Hence we divide array data along z-direction and assign each part to each processor as depicted in Fig. 2. We refer to this partition as P_z. Then the inversion of L_x can be done on each processor independently and vectorization can be done in y-direction. The inversion of L_y also can be executed in parallel on the partition P_z. Then we transpose the data in advance into the partition P_y divided along y-direction so that the inversion of L_z could be done in parallel. The parallel version of the IAF method is shown in Fig. 3.

Next problem was what performance each processing element could have. Each processing element should deliver performance as high as possible. The lower the performance of each PE is, the more the number of PEs becomes, and hence interconnection network is limited and not flexible. This feature is restrictive to obtain high sustained performance for various CFD methods.

On the other hand, vector computers show a very effective performance for CFD calculations because CFD algorithms usually have a long vector length. Through the investigation of CFD programs used at NAL and the analysis with a performance estimation software of vector computers, we concluded that in case of vector computers with large vector register even pipelines with the multiplicity of 8 could show high performance. Moreover, the peak performance of a processing element should be more than 1 GFLOPS, almost the same as VP400, because it allows CFD researchers to continue their works of the moment as well as to attack a new field.

If the interconnection network is equidistant such as fully connected or crossbar, some degrees of freedom are ensured in operation. For example, in case that several users employ parallel processors simultaneously, PEs are allocated flexibly without interference.

Consideration as above leads to a conclusion that distributed memory parallel processors with about 200 PEs having the same or more performance than that of VP400 and with strong interconnection network could meet the NAL's requirements. That is an approach of parallel vector computer with distributed memory. Since we do not have computer manufacturing technology such as LSI and mounting technology, we decided to extend an investigation with computer companies. Feasibility studies started in 1989 with some manufacturers and NWT is realized through the joint research with Fujitsu starting in 1991.

§ 3. The numerical wind tunnel

3.1. *Configuration of the NWT*

Figure 4 shows the configuration of the NWT based on the architecture as mentioned above. It consists of 140 processing elements, vector computers assuming computation processing, and 2 control processors. These processors are connected with a crossbar network.

The configuration of the processing element is similar to that of the VP400. The machine cycle is 9.5 nanoseconds, and multiplicity of pipeline is 8. Each processing element has 256 MB of main memory and 1.7 GFLOPS of peak performance. The system's overall peak performance is 236 GFLOPS, and its overall main memory capacity is 35 GB. The data transfer rate of the cross-bar network is 421 megabytes per second for read and write, respectively. The two control processors administer the NWT as a system. Researchers do not directly access these control processors. The VP-2600 supercomputer serves as front end processor. The capacity of the system storage unit is 16 GB, its transfer rate is 1200 MB per second for read and is 800 MB for write.

3.2. *NWT FORTRAN*

CFD researchers have a lot of program resources that are described in conventional FORTRAN. It is also very important to take advantage of these resources. NWT FORTRAN was so designed as to be based on the FORTRAN77 enhanced with compiler directives. Hence the parallelized version of a program is fundamentally able to run on conventional supercomputer.

User should only describe data decomposition and corresponding process decomposition to effectively perform parallel processing.

The SPREAD/BARRIER execution method is used for the parallel execution and synchronization.

The NWT is a distributed memory computer and main memories are physically distributed. To ease a programming, the operating system together with

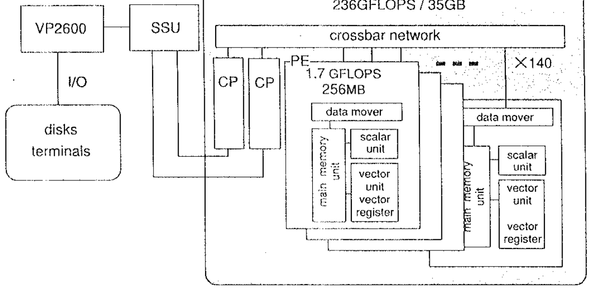

Fig. 4. Configuration of the NWT.

the compiler provides a logical global space. Of course, the access speed to the global space is much slower. To speed up the access to the global space which is physically allocated in the local memory, a function to regard the global space as a local space is realized by using EQUIVALENCE statement.

The function to effectively process the indices plus minus 1 or whatever emerging frequently in finite difference methods is implemented by overlapped partition which is called WING.

§ 4. Performance evaluation

In this section, some measured performance of the NWT are shown.

4.1. *LINPACK*

First one is LINPACK highly parallel benchmark test. The highest performance listed on MP-LINPACK was far below 100 GFLOPS until July in 1993. The NWT first opened the door of 100 GFLOPS and presented a fine target. The INTEL PARAGON at Sandia exceeded the performance in May, 1994. It is Fujitsu VPP500 and CRAY T3D that enter into the arena of 100 GFLOPS up to now after that. At present, after some modification, the NWT with 140 PEs attained 170 GFLOPS.

4.2. *NS3D*

NS3D is a CFD program which has been used for performance evaluation in the course of the development of the NWT. It solves three-dimensional Navier-Stokes equation without turbulence model. The scheme utilized is TVD scheme and implicit approximate factored method. It solves directly block tridiagonal matrices arising from this scheme, that is the same as BT in the NAS parallel benchmark test. As a programming technique, it uses only 20 of three dimensional arrays. This means that it computes every time such values as pressure, acoustic speed and so on appear.

Table II. Parallel efficiency with NS3D.

No. of PE	2	4	8	16	32	64	128
Speed-up	1.89	3.72	7.49	14.96	29.77	57.65	101.7
Efficiency	0.95	0.93	0.94	0.93	0.93	0.90	0.79

Table III. Measured performance with TVDSD.

No. of PE	50	100	130
grid points	$1001\times100\times50$ (5 M points)	$1001\times200\times100$ (20 M points)	$1001\times260\times130$ (34 M points)
floating ops (Gops)	13.5	55.2	93.8
performance (GFLOPS)	23.6	50	71
time	18	37	44

Accordingly, total numbers of floating point operations are more than ten thousands per grid point. It is very large compared to other usual CFD programs. If you want to simulate large scale CFD problems, you must trade off computing time with memory to some extent.

To evaluate the parallel efficiency, we calculated with a grid system of $64 \times 128 \times 128$ totaling about 1 million points which can be treated on a single PE. The parallel efficiency is shown in Table II.

To see the maximum active performance, we calculated with maximum size of grid points of about 60 millions using 140 PEs. The measured performance is 116 GFLOPS.

4.3. *TVDSD*

TVDSD is a typical CFD program using algebraic turbulence model. It uses 88 of three dimensional arrays including work area. The biggest difference between TVDSD and NS3D is operation count per grid point, about 2300 compared to more than ten thousands.

Table III summarizes the performance of TVDSD on the NWT. The last column shows the computing time in minutes per 2000 iterations. This shows that flow fields around complex geometry can be simulated with such a large number of grid points only in an hour or two.

4.4. *BIGCUBE*

The last example of measured performance is a direct numerical simulation of 3 dimensional isotropic turbulence using a spectral method. The program is called BIGCUBE.

Figure 5 shows a simulation with 512 cubed of spatial resolution. The intense velocity of high Reynolds number turbulence concentrates into intermittent narrow region in space. The regions are composed of local short vortex tubes and the so-called warm structure is observed.

The performance of BIGCUBE is shown in Table IV along with the performance of another similar simulation for comparison.

We do not apply any vector tuning for CRAY. But it seems reasonable because the active performance of CFD programs with YMP is reported to be about half the peak performance. Jimenez and others[3] reported 4 GFLOPS on the Touchstone Delta with 512 nodes at Caltech in a similar calculation.

As a result, the NWT displays the actual performance 23 times more than the Touchstone Delta, 514 times more than the CRAY YMP. This implies that it would take one year with the YMP and two weeks with Delta to finish the same simulation.

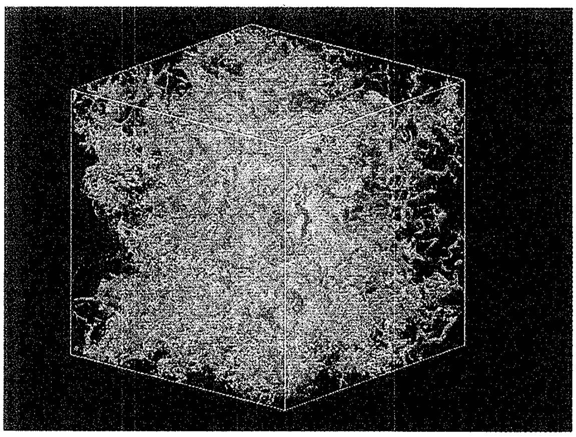

Fig. 5. Direct simulation of isotropic turbulence.

Table IV. Measured performance with BIGCUBE.

	YMP M92/1	Delta/512	NWT/128
grid points	256^3	512^3	512^3
performance (GFLOPS)	0.175	4	90
Ratio	1	23	514

§ 5. Some examples of CFD simulation on the NWT

After the installation of the NWT, NAL's researchers have been attacking both basic research on fluid dynamics and application to aircraft, spacecraft and engines. In this section, we show some examples of simulation done at NAL.

Figure 6 shows numerical simulation of compressible mixing layers. Direct numerical simulation of a spatially evolving compressible shear layer are performed using a high-resolution finite difference method with initial perturbations specified. Instantaneous flow fields are visualized in the figure in terms of isovelocity surface. As the convective Mach number increases the vortical structures drastically change and the spatial spreading of the shear layer is suppressed.

Figure 7 shows numerical simulation of transonic wind tunnel flows about a fully configured model of aircraft. This computation is an attempt to solve whole flow fields within the transonic wind tunnel at NAL where a fully configured model of aircraft named ONERA M-5 is settled. In order to solve flow fields around a complicated configuration, whole computational region is covered with several overlapping grids, and flows are solved by a TVD scheme.

Figure 8 shows CFD analysis for aerothermodynamic characteristics of HOPE (an unmanned space shuttle Japan is planning) for practical design. Numerical study of hypersonic flow around HOPE is conducted using an upwind TVD Navier-Stokes

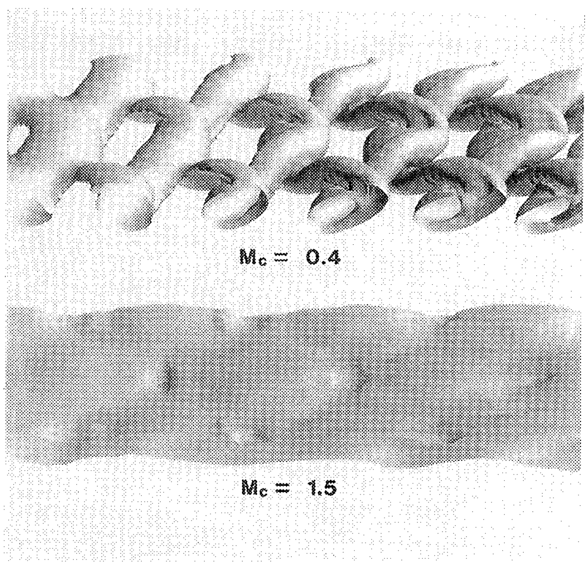

Fig. 6. Compressible mixing layers.

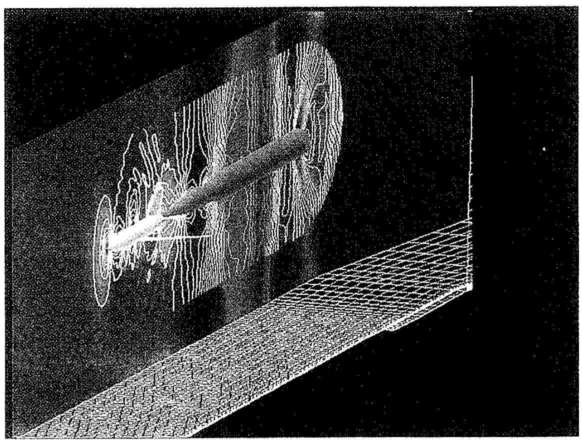

Fig. 7. Transonic wind tunnel flows about a fully configured model of aircraft.

Fig. 8. Aerothermodynamic characteristics of HOPE.

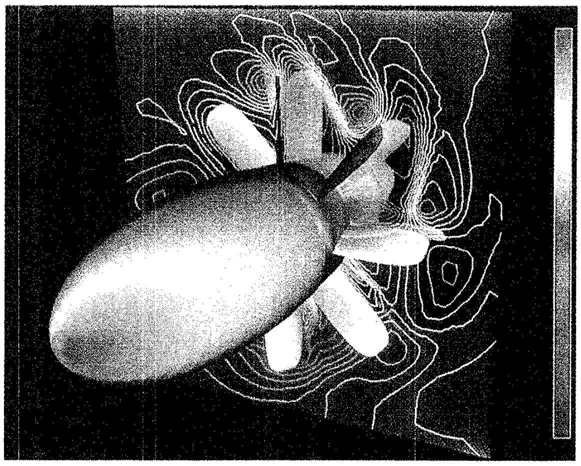

Fig. 9. Unsteady flow through a counter-rotating propeller.

CFD code. Figure 8 shows three-dimensional stream lines around double delta type and power delta type HOPE basic configuration.

Figure 9 shows numerical simulation of unsteady flow through a counter-rotating propeller at angle of attack. This figure shows simulated instantaneous pressure contours. The flow field is non-periodic in the circumferential direction. Hence, the whole area around the propeller is solved on the NWT using a domain decomposition algorithm.

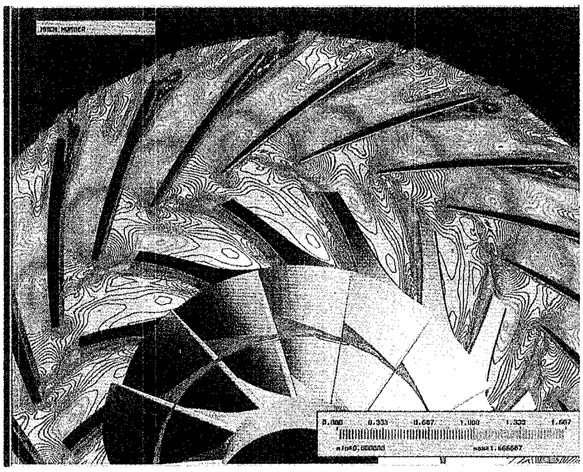

Fig. 10. Interaction of flow fields through a centrifugal impeller/diffuser.

Figure 10 shows an unsteady simulation of a centrifugal impeller/diffuser interaction flow fields. The effects upon vaned diffuser flow fields due to the wake which is ejected from a centrifugal impeller have been analyzed by an unsteady 3-dimensional viscous calculation.

§ 6. Conclusion

We designed and developed the NWT with Fujitsu Co. exclusively to contribute to the development of CFD research. The NWT displays good performance we have expected at least for solving problems in fluid dynamics. Recently, we obtained an opportunity of doing a joint research work with the Yamagata University to extend the application field of the NWT to QCD problems and saw that the NWT is also useful there. We are pleased to see the machine we developed could contribute to the development of various research fields.

Acknowledgements

We thank to Professor A. Nakamura in the Yamagata University for giving us an opportunity to introduce the NWT and our research activities.

References

1) M. Fukuda, T. Iwamiya and H. Miyoshi, Notes on Numerical Fluid Mechanics, Vol. 37 (1993), p. 157.
2) T. Iwamiya, M. Fukuda, T. Nakamura and M. Yoshida, *Parallel Computational Fluid Dynamics, New Trends and Advances* (Elsevier Science B. V., 1995), p. 51.
3) J. Jimenez et al., Journal of Fluid Mechanics, 1993, vol. 255, p. 65.
4) H. Miyoshi et al., *Proceedings of Supercomputing '94* (1994), p. 685.