Development of CFD Software Package for the Simulation of Clean Rooms using Sequential and Parallel Computing Algorithms

Researchers

Asst.Prof.Dr.Ekachai Juntasaro — Suranaree University of Technology
Asst.Prof.Dr.Varangrat Juntasaro — Kasetsart University
Asst.Prof.Dr.Putchong Uthayopas — Kasetsart University

Introduction

At present, nanotechnology has played an important role for a variety of industries, especially electronics. Many equipments tend to be very small and delicate so that advanced and high-precision technology has to be used for production. The clean room is a representative sample of that technology. It is necessary to control the behavior of environment (velocities, temperature, etc) inside the clean room for appropriate conditions. Due to the complexity of the domain, huge memory storage and large computational time give rise to the need of high performance computing and sophisticated computing technique. The PC cluster is adopted for such purposes to parallelize the computation.

Objective

To develop a CFD software package for the simulation of clean rooms by using sequential and parallel computing in order to reduce the cost and time needed to design, analyze, develop and examine the clean rooms and to increase the efficiency of the electronic manufacturing process.

Method

The finite volume method is adopted to discretize the governing equations. The SIMPLE algorithm is employed to numerically solve the discretized equations of RANS and continuity and to subsequently solve all remaining discretized equations of scalar field: equations of turbulence model, temperature, water vapor mass fraction, and particle volume fraction. The domain is divided into several sub-domains or blocks and then computed simultaneously by parallel computation: one block assigned to one processor. To exchange the information between adjacent blocks, the MPI library is adopted in the present work.

Results

The software is developed from two main parts. One part is to develop and validate the turbulence and physical modeling. The other part concentrates on the numerical and computational techniques. Fig. 1 shows the CAD model of the present clean room and Fig. 2 displays the simulated velocity vector inside such a clean room in which a single grid is used. To cope with the complexity of the domain, instead of using the multiblock technique, the block-off technique is firstly adopted for the sake of simplicity. Parallel computation in conjunction with the multiblock technique are later used to cope with the problems of large memory storage, long intensive computation time and domain complexity. Fig. 3-5 are the results of the simulation of flow in a T-junction. The code is firstly validated as shown in Fig. 3. Fig. 5 exhibits the performance of parallel computing. The domain is divided into 15 sub-domains or blocks and each block is assigned to one processor.

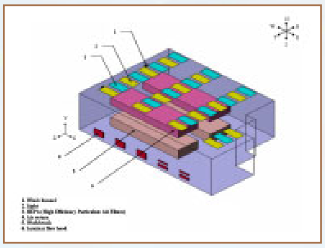

Fig. 1 Model of the SQE C/R clean room installed at Seagate Technology (Thailand) Ltd. in Nakhon Ratchasima

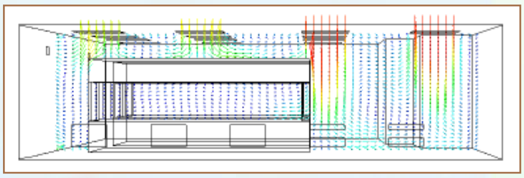

Fig. 2 Velocity vector field (side view in the negative z-direction)

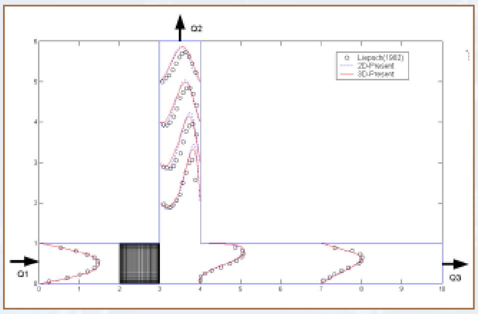

Fig. 3 The T-junction flow problem, grid configuration and validation of the computed results

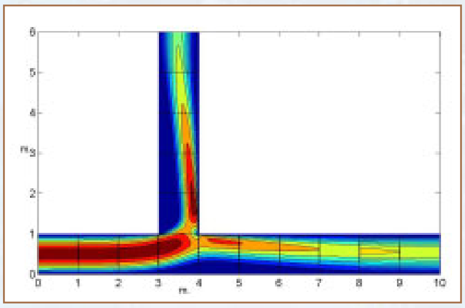

Fig. 4 Contour plot of velocity field

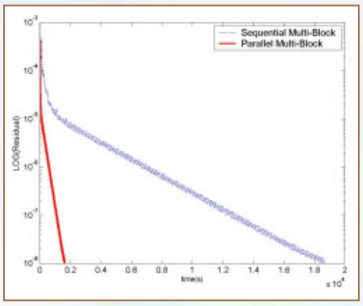

Fig. 5 Comparison of residual reduction versus time used for parallel and non-parallel computing

Conclusion

Up to now the development of the CFD code in the part of turbulence and physical modeling is somewhat more advanced than in the part of numerical and computational techniques. To develop the former part, the validation of the CFD code is performed continuously with the standard benchmark problems.

Even though the clean room can already be simulated, but it takes so much computing time and involves the problems of internal boundary and complicated domain which are not resolved well with the use of the block-off technique. The development of numerical and computational techniques is proceeding toward the coordination of parallel computing, multiblock technique and multigrid method to reach a state of high performance computing software.

References

Liepsch, D., Moravic, S., Rastogi, A. K., and Vlachos, N. S. (1982) *"Measurements and Calculations of Laminar Flow in A Ninety-Degree Bifurcation"*, J. Biomechanics, Vol. 15, pp. 473.

Versteeg, H. K., and Malalasekera, W. (1995) *"An Introduction to Computational Fluid Dynamics: The Finite Volume Method"*, Longman Scientific & Technical.

http://www.cleanroom.com

โครงการนี้ได้รับทุนอุดหนุนการวิจัยและพัฒนา จาก
ฝ่ายเครือข่ายการวิจัยและพัฒนา (RDD)
ศูนย์เทคโนโลยีอิเล็กทรอนิกส์และคอมพิวเตอร์แห่งชาติ
112 อุทยานวิทยาศาสตร์ประเทศไทย ถนนพหลโยธิน
ตำบลคลองหนึ่ง อำเภอคลองหลวง จังหวัดปทุมธานี 12120
โทรศัพท์ 02-564-6900 ต่อ 2501-10 โทรสาร 02-564-6901..2

http://www.nectec.or.th/

NSTDA ECTI-21 NECTEC