Developing 2D Navier Stokes solver on intel MIC platform:A Survey

Akshay S. Pungle
Department of Computer Engineering,
Pune Institute of Computer Technology,
Pune, India.

Prof. Arati Deshpande
Department of Computer Engineering,
Pune Institute of Computer Technology,
Pune, India.

Dr. Vikas Kumar
CAE Group,
CDAC,
Pune, India

***Abstract*- In the field of Computational fluid dynamics (CFD) fluid flows are analyzed by simulating them with the help of high speed computers. To analyze these fluid flows we require some mathematical equation which governs the fluid flow, these are the Navier-Stokes (NS) equations. Near about every problem in CFD is related to NS equation, so solving these equation is necessary in order to analyze fluid flow. These NS equations are partial differential equations so different numerical methods are used to solve these equations. Solving these equations using numerical schemes requires huge amount of calculations to arrive at desired result. We can solve these equations by writing a parallel program which can then run on powerful hardware to get the results. High speed supercomputer will provide us very good performance in terms of reduction in execution time. In paper focus will be on finite volume as a numerical method. We will also see what Intel MIC (*Many Integrated* Core Architecture) is and how we are taking its advantages to solve CFD problems.**

Keywords-*High Performance computing(HPC), Computational Fluid Dynamics(CFD), NS(Navier Stokes),intel MIC(Many Integrated* Core Architecture)

I. INTRODUCTION

Computational Fluid dynamics deals with motion of fluid flows by using NS equations. High speed computers have been used to solve the problems in different fields. Computational fluid dynamics is like such field which can utilize those high speed computers to solve their problems. NS equations are the set of equations which governs the motion of fluid flow with properties like temperature, pressure, viscosity. The NS equations governs the motion of compressible and/or incompressible fluid flows. It is used to simulate the real life phenomenon like wind flow.

Thus solving the NS equations requires large computational power and capacity. Therefore such a large and complex problem is divided into the sub-problems that is interrelated. Previously the NS equation was solved using schemes like finite difference method (FDM), finite element method (FEM), finite volume method (FVM) [12]. With the advent in the computer system, the accelerator device like Intel Xeon Phi enable to analyze such complex problem rapidly and economically. Computational fluid dynamics (CFD), help us to explore and deal with a wide range of problems related with heat and mass transfer (HMT) using numerical solution of a governing equations.

For solving a large and complex problems in CFD we are using powerful digital computers to solve these equations numerically and simulate engineering problems with different phenomena. We can aim at developing a program that will allow the detailed numerical simulation using parallel computing resources.HPC cab be used to solve this problem.
In this we can use Intel Xeon Phi accelerator and parallel programming to solve those problems and reduce the time required. Intel Xeon-Phi is an accelerator based architecture targeted for high performance computing (HPC). The accelerator typically can execute 244 parallel threads on its 60 cores and houses a small RAM. The small RAM is often inadequate for typical code executions. This often requires intelligent data management across the host and accelerator.

II. LITERATURE SURVEY

In [1] authors have used finite difference method for simulating incompressible fluid flows based on parallel computing platform. They used explicit scheme for pressure and an implicit scheme for velocities. Original calculation is decomposed into several blocks and they execute in parallel.MPI is used for message passing. Execution time and speedup are calculated and performance is evaluated.

NS equation is solved using artificial compressibility method and its implemented using GPU.Finite difference numerical scheme is used in the work.Discretizations have been carried out on a MAC grid. Calculation of pressure and velocity is carried on GPU. Two kernels were used for the communication between GPU grids. Speedup is calculated and comparison is done between CPU and GPU based implementation [2].

Parallelism is achieved within PETSc framework using single program-multiple-data (SPMD) message passing model. In implementation, interface library layer for MPI and the sparse matrix solver in PETSc are used, i.e., both message Passing and solver operations are performed by PETSc library Subroutines [3].

Lattice Boltzmann method is used in the work to solve two dimensional incompressible Navier-Stokes equation. Numerical method is implemented in sequential and CUDA program. Results are then compared with the analytical solution that exists. Computational performance is increased using CUDA.Speedup is almost 10 times than that of sequential program [4].

FFTW (it is a C subroutine library for computing the discrete Fourier transform (DFT) in one or more dimensions) and CUFFT (CUDA Fast Fourier Transform library) is used to solve the two dimension incompressible Navier-Stokes equations with pseudo-spectral method. Results shows performance on GPU is much better than CPU [5].

Sine collocation method is used to driven cavity problem. The Navier Stokes equations were solved using two dimensional sine collocation using the primitive variables method. Simulation were done using finite difference method and results are compared. Simulations were also done using the commercial CFD code FLUENT [6].

Multigrid solver is implemented in the work. Coral, a Beowulf-class system based on a high performance GigaNet CLAN interconnect and Dual Pentium III nodes is used as a computing platform. The solver attained good convergence rates and also achieves satisfactory efficiencies on the target cluster [7].

Finite volume method is used to solve the incompressible flow NS equations. Solver is implemented using CUDA on to GeForce GTX480.CPU based and GPU based solvers are compared. Two strategies were used firstly computations associated with the SIMPLE algorithm have been transferred to GPU and then all computations are carried out on GPU.Speedup of up to 40x is observed via GPU [8].

One dimensional Burgers equation is solved and implemented using implicit method. Intel Xeon Phi Coprocessor is used as a computing facility. MAGMA MIC library is used which is an open source high performance library for solving a systems of non-linear equations. For high performance computation offload mode is considered as the primary mode of operation for Intel Xeon phi coprocessor. Implicit scheme is then compared with the exact values and it's seen that the results obtained are approximate and reliable. System achieved higher performance on Intel MIC platform [9].

In the paper written by Inan et al, they have used a new scheme where they first smooth the equation using LU decomposition and later applied the implicit exponential finite difference method and fully implicit exponential finite difference method [10].

In solving this particular partial differential equations selection of appropriate numerical method leads to greater convergent solution so proper numerical scheme is essential. Kutluay et al. have presented explicit and exact-explicit finite difference methods for solving these equations. They have discretized the diffusion equation from Hopf-Cole transformation and gave an explicit expression for the exact solution of it. But they used severe stability condition which requires small size of time steps. Discretization scheme used by them was explicit [11].

III. Methodologies To Solve NS Equation

A. 2D NS equation

Two-dimensional steady laminar viscous incompressible flows are governed by Navier- Stokes equations [16]. In primitive variables (u, v, p) they are given by,

Continuity equation,

$$\frac{\delta u}{\delta x} + \frac{\delta v}{\delta y} = 0 \qquad (1.1)$$

x - Momentum equation,

$$u\frac{\delta u}{\delta x} + v\frac{\delta u}{\delta y} = -\frac{1}{\rho}\frac{\delta p}{\delta x} + \frac{\mu}{\rho}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \qquad (1.2)$$

y - Momentum equation,

$$u\frac{\delta v}{\delta x} + v\frac{\delta v}{\delta y} = -\frac{1}{\rho}\frac{\delta p}{\delta y} + \frac{\mu}{\rho}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \qquad (1.3)$$

Where,
u, v are the velocity components of the two dimensional flow,
ρ is the density
μ is the coefficient of viscosity
p is the pressure.

B. Finite Volume Method

Different methods are there to solve the partial differential Equation like finite difference, finite element etc. Here we are using finite volume method. As shown in figure 1 It represents the partial differential equation as algebraic equation then those equations are solved iteratively to get the solution. Finite volume method [16] is most widely used in CFD problems. It allows us to formulate unstructured meshes as well. As shown in the fig.1 firstly domain is divided into finite set of sub domains. This process is also called as grid generation. Then the NS equation is integrated over each control volume applying conservation of mass and momentum equations. This gives us the algebraic set of equation from set of partial differential equation. Finally this algebraic set of equation are solved iteratively till solution converges.

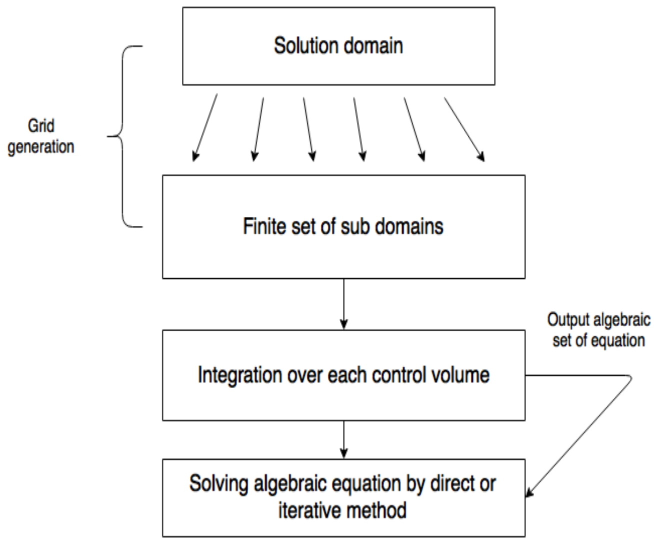

Fig. 1. Finite Volume method

C. Porting NS solver to Intel MIC(Xeon Phi)

The use of heterogeneous architecture clusters for solving CFD problem are currently in widespread use. A total of 62 supercomputers in the November 2012 Top500 List [12] are using accelerator/Intel Xeon Phi co-processor technology. Intel published an Intel MIC Platform Software Stack (MPSS), Which consists of an embedded Linux, a minimally modified GCC, and driver software [13] [14]. For porting the NS solver On Intel MIC platform we are going to apply a very basic approach. Firstly we are going to develop the serial code for solving the NS equation which is then divided into the submodules. And this sub-modules is then executed parallel using Intel MPI library. The NS equations are a complex set of partial differential equation that requires high computational power for obtaining solution and hence we are using accelerator.

IV. Conclusion

In this paper based on some literature studies conducted by researchers we presented pro and cons of different numerical schemes for solving NS equation. And also suggested to use the finite volume method to solve NS equation as it provides a good convergent rate. It is hoped that this survey paper will help to stimulate more research and application of Navier-Stokes equation in computational fluid dynamics (CFD).

Acknowledgment

I am indeed thankful to my internal guide Prof. Arati Deshpande and external guide Dr. Vikas Kumar for their guidance and assistance to complete this paper. I am grateful for their valued support and faith on me. I extend my special thanks to Head of Department of Computer Engineering, Prof. Girish P. Potdar who extended the preparatory steps of this paper-work.

References

[1] De Angeli, Joo Paulo, et al. ”Finite difference simulations of the Navier-Stokes equations using parallel distributed computing.” Computer Architecture and High Performance Computing, 2003. Proceedings. 15th Symposium on. IEEE, 2003.

[2] Itu, Lucian Mihai, et al. ”Optimized GPU based simulation of the incompressible Navier-Stokes equations on a MAC grid.” Roedunet International Conference (RoEduNet), 2011 10th. IEEE, 2011.

[3] Karimian, S., and Anthony G. Straatman. ”Benchmarking of a 3D, unstructured, finite volume code for incompressible Navier-Stokes equation on a cluster of distributed-memory computers.” High Performance Computing Systems and Applications, 2005. HPCS 2005. 19th International Symposium on. IEEE, 2005.

[4] Kosriporn, Treenath, Worasait Suwannik, and Montri Maleewong. ”Lattice Boltzmann method for two-dimensional incompressible Navier-stokes equation with CUDA.” TENCON 2014-2014 IEEE Region 10 Conference. IEEE, 2014.

[5] Lou, Kaiyuan, and Zhaohua Yin. ”A CUDA pseudo-spectral solver for two-dimensional Navier-Stokes equation.” Distributed Computing and Applications to Business, Engineering Science (DCABES), 2012 11th International Symposium on. IEEE, 2012.

[6] Narasimhan, Susheela, Kuan Chen, and Frank Stenger. ”The solution of incompressible Navier Stokes equations using the sine collocation method.” Thermal and Thermomechanical Phenomena in Electronic Systems, 2000. ITHERM 2000. The Seventh Intersociety Conference on. Vol. 1. IEEE, 2000.

[7] Prieto, Manuel, et al. ”A multigrid solver for the incompressible Navier-Stokes equations on a Beowulf-class system.” Parallel Processing, 2001. International Conference on. IEEE, 2001.

[8] Niksiar, Pooya, et al. ”Implementation of a GPU-based CFD Code.” Computational Science and Computational Intelligence (CSCI), 2014 International Conference on. Vol. 1. IEEE, 2014.

[9] Hitesh P. Kahale,Rekha Kulkarni,Vikas Kumar. ”Developing and Porting multi-grid solver for CFD application on Intel-MIC platform” International Journal of Computing Algorithm,Volume: 04, Issue: 01, June 2015

[10] Inan, Bilge, and Ahmet Refik Bahadir. ”Numerical solution of the onedimensional Burgers equation: Implicit and fully implicit exponential finite difference methods.” Pramana 81.4 (2013): 547-556.

[11] Kutluay S., Kadaoir M.B. and Zde A. ”Stable numerical solution of one dimensional Burgers equation: explicit and exact-explicit finite difference methods.” Journal of Computational and Applied Mathematics, vol. 103, no. 2, pp. 251-261, 1999.

[12] Top500 supercomputing sites, http://www.top500.org”

[13] James Reinders,Knights corner: Open source software stack.” http://software.intel.com/en-us/blogs/2012/06/05/knights-orneropensource- software-stack/.

[14] Knights corner micro-architecture support,” http://software.intel.com/enus/blogs/2012/06/05/knights-corner-icroarchitecture- support/

[15] B. Hadri. Application Experiences on a Cluster Supercomputer Equipped with Intel Xeon Phi Coprocessors. In Supercomputing Lab, 2013.r ”

[16] Versteeg, Henk Kaarle, and Weeratunge Malalasekera,An introduction to computational fluid dynamics: the finite volume method,Pearson Education, 2007.

Akshay S. Pungle received Bachelor's degree in Computer Engineering from Savitribai Phule Pune University in 2013 (email:akshay.pungle@gmail.com). He is currently pursuing Master of Engineering from Savitribai Phule Pune University. He is also an Intern at CAE Group, CDAC, Pune.

Arati Deshpande is Professor of the Department of Computer Technology, PICT, Pune for almost 16 years.

Dr.Vikas Kumar has been leading CFD group of CDAC since 2002. He worked as a Research Engineer in Computer Aided Engineering Department at Thapar Centre for Industrial Research Development, Patiala from 1996 to 2002.He has got his Ph.D. in 2005 in Mechanical Engineering with specialization in Heat Transfer from Thapar University., Patiala. He has carried out his Post-Doctoral research work at University of Nebraska, Lincoln, Nebraska, USA during 2007-2009.His areas of interest is Computational Fluid Dynamics Simulation in Heat Transfer Fluid Flow, and applications of High Performance Computing.