# Simulations of flows around vehicles with Linux clusters

Siniša Krajnović

Division of Fluid Dynamics, Department of Applied Mechanics,
Chalmers University of Technology, SE-412 96 Gothenbur, Sweden
sinisa@chalmers.se

**Abstract.** Time-dependent simulations of flows around vehicles require computations on large number of processors during long time. Such simulations are made possible with introduction of cost-efficient Linux clusters. Here we present a review of the work in this field from the pioneering simulations on expensive sheared memory parallel computers to computations on Opteron Linux clusters used today. Examples of simulations varying from flow around a bus and a car to a flow around a high-speed train are presented. The use of the computer resources for these simulations and comparison between different machines are discussed.

## 1 Introduction

Simulations of flows around bluff bodies (such as cars) have in past suffered from inadequacy of turbulence modeling to deal with regions with separations. In particular the flow region behind the body, called wake, with a wide range of different turbulent scales from the size of the body to microscopic ones could not be predicted. Beside the absence of turbulence models that can deal with such complex flows, the failure of the CFD simulations was due to simplified equations called Reynolds-Averaged Navier Stokes (RANS) equations. As the computer resources were unavailable for solution of time-dependent Navier-Stokes equation, the time-averaged RANS equations were solved. Bluff-body flows are very time-dependent and due to the time-averaging in RANS equations all instantaneous information is lost making the modeling task important and difficult. Furthermore, the inherently time-dependent flow processes such as cross-wind stability or aeroacoustics are prevented with such an approach.

Relatively old CFD technique called Large Eddy Simulation (LES) received a renaissance in 1990's with better availability of parallel computers. LES solves time-dependent Navier-Stokes equations for all turbulent vortices that are larger than the size of computational cells, and only the influences of the smallest flow structures are approximated with some turbulence model. In practice around 80-90 % of all turbulence in the flow is computed with this technique. Computing so much of the turbulent structures requires very fine computational grids and short time steps (i.e. a large number of numerical time steps for simulating physical time that is required for study of the flow).

In the second half of 1990's the author has used this technique to compute the flow around a sharp-edged surface mounted cube [1] shown in Fig. 1a. These simulations used between 200 000 and 1.6 million computational cells and were run on small number of processors on an ORIGIN 2000 shared memory parallel computer. An efficient single-block finite volume code was used for the computations. It uses multigrid for the acceleration of convergence when solving the pressure equation. The computational cost for a typical simulation was around 60 CPU hours on an SGI R10000. Although

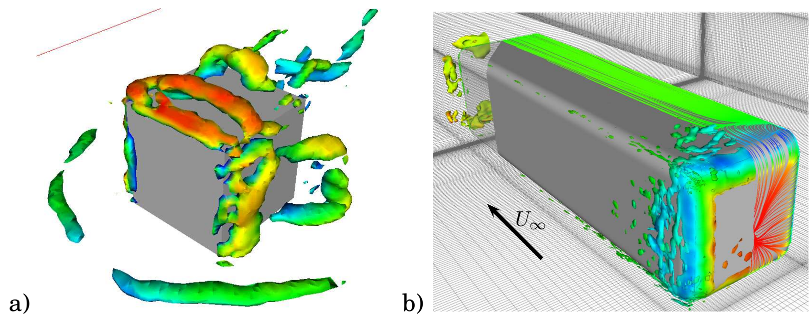


**Fig. 1.** Flow around a) a surface mounted cube; b) a simplified bus.

these initial simulations were encouraging, simulation of flow around vehicles with complex geometry require a code that uses domain decomposition . First LES simulation of vehicle flow was that of a simplified bus [2] shown in Fig. 1b. This simulation used a computational grid containing 4.5 million computational cells and around eighty thousand time steps. Forty processors on an ORIGIN 2000 were required during four months (wall time) for resolving the necessary turbulent scales. Having in mind the high cost of this machine, this was an expensive simulation. The long running time is partly due to the time it takes for the flow to develop, and partly due to the time-averaging and low frequent motions of the flow.

## 2 Introduction of Linux clusters in vehicle aerodynamics

With introduction of Linux clusters, the performance/cost ratio increased drastically. An example of simulation made on a Linux cluster is that of the flow around a simplified Volkswagen Golf car shown in Fig. 2a. This simulation was made on the cluster Monolith at the National Supercomputer Centre in Linköping (NSC). The entire cluster contains around 200 computers with 2 processors each. The peak performance is 4.8 Gflops/CPU and MPI runs over two different networks: SCI -based network using ScaMPI and Fast Ethernet, using MPICH or LAM. 32 processors and SCI-based network were used for this simulation. The simulation using a grid of 16.5 million computational cells and 110, 000 time steps required around three and a half months (wall time). However in the same time two smaller simulations using 7.5 million cells were run both on this machine and on a similar machine (with the same type of dual computers) where Fast Ethernet using LAM is the only network. The computational time was decreased with only some 15 % when the SCI-based network was used. This simulation is good

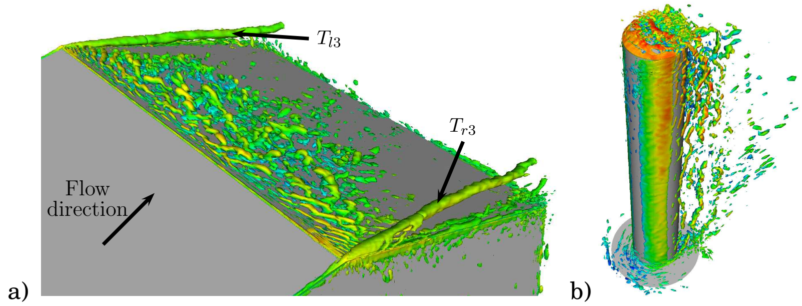


**Fig. 2.** a) Flow on the rear window of a simplified Golf car. b) Flow around a surface mounted finite cylinder.

example of how Linux clusters had an impact on understanding of vehicle flows. It was very successful and helped us to solve a mystery that has puzzled researchers for more than two decades. Interested reader is refereed to [3] for more details about these simulations.

## 3 Greater confidence in CFD with Linux clusters

One of the conclusions we drew in our work on the Golf-like car was that prediction of the flow from our simulation was in excellent agreement with the experimental data. Furthermore, much more data were obtained and we were able to explain this flow for the very first time. This has led to greater confidence in numerical simulations of flows around vehicles and other bluff bodies. Thus we are today using Linux clusters not only to explain different flows but also to make extensive database that can be used by other researchers who are studying such flows. Normally such databases are produced with experimental tests in wind tunnels. An example of a flow that was studied with Linux clusters for the purpose of making a database is the flow around a finite cylinder such as a chimney. It is difficult to investigate this flow due to complex vortex interaction and unsteadiness. Our

LES simulation has discovered an extremely complex flow picture, shown in Fig. 2b.

## 4 Use of 64-bit processors

Introduction of AMD Opteron computers have decreased the computational time of our simulations when single precision is used. Besides, it is now possible to allocate much more of internal memory than on 32-bit processors. This is beneficial both for our single-block and multi-block codes. An example of simulation where our single-block code was run on an Opteron cluster is that of a flow around a three-dimensional hill where approximately 15 million computational cells were used on a single Opteron processor with 10 GB of internal memory. Thus using an efficient solver and small number of Opteron processors we can today run very large simulations. However only relatively simple geometries can be handled with a single-block code. For complex geometries we use our multi-block code on Opteron clusters. Flow around a high-speed train is an example that is simulated both on Opteron cluster Sarek at High Performance Computing Center North (HPC2N) and the Swegrid cluster. Sarek has dual AMD Opteron 248 (2.2 GHz) processors, 8 GB memory per node and uses Myrinet 2000 high speed interconnection. The Swegrid cluster has a 2.8 GHz Intel P4 processors, 2 GB memory per processor and gigabit Ethernet. It should be mentioned that the internal memory was not a limiter in these computations as they used approximately 600 MB per processor. This computation uses 38 processors to compute the flow in 17 million computational cells. It was found that the simulation on Sarek was by 40 % faster from the one on Swegrid.

## 5 Conclusions

CFD is capable of predicting and explaining flow around vehicles and other bluff bodies if the unsteady Navier-Stokes equations are solved. This approach became feasible with the introduction of Linux clusters. An optimal tool for such simulations is a large Linux cluster with 64-bit processors and high speed interconnection. However, the key word in this research is availability of the resources. Simulations on a homemade Linux cluster with Intel P4 or AMD Opteron processors and Fast Ethernet have proved to be very efficient tools in vehicle aerodynamics research.

## References

1. Krajnović, S., Davidson, L: Large Eddy Simulation of the Flow Around a Bluff Body AIAA Journal **40/5** (2002) 927–936
2. Krajnović, S., Davidson, L.: Numerical Study of the Flow Around the Bus-Shaped Body. ASME: Journal of Fluids Engineering **125** (2003) 500–509
3. Krajnović, S.: Discovering the flow around a Golf car with cluster Monolith. NSC Newsletters **25** March (2006)