Univerza v Ljubljani
Fakulteta za strojništvo

Numerical Simulation of the Turbulent Flow around a Sphere

Laboratorij za nelinearno mehaniko LANEM

Project is dealing with massive numerical simulation of turbulent fluid flow around a sphere with Reynolds number 50,000. For numerical simulations finite volumes numerical method was used. Comparation between results using of K- ε, K-Ω and SST turbulence models and direct numerical simulations with filters LES and DES were performed. The simulations were provided on the mesh with one million and refined one with seven millions of finite volumes.

With numerical simulations it were found that only DES and LES satisfactory simulate separation wakes around sphere in comparations with experimental data. Moreover LES and DES provided wakes behind separation area with burst of overshoot flow total energy. This very interesting phenomenon has to be independently verified by experiments in low turbulence wind channel.

Simulation domain is decomposed and calculated on BCX parallel supercomputer. It was found that communications between processor inside node are good but latency in communications between nodes is difficult although the best available network connections Infiniband were used. This proves that massive calculations with a lot of communications between nodes are possible to obtain only on high performance supercomputers with extremely fast networks.

Introduction

Flow around the sphere is a challenging case for turbulence modelling due to the relatively complex physical phenomena. In the sub-critical regime till Reynolds number 200,000 downstream of the stagnation point, a very thin laminar boundary layer is formed which separates just before the equator and becomes fully turbulent behind sphere. As the flow separates from a smooth surface, the separation line is not fixed. The wake is highly turbulent and includes a large recirculation region. It is known that most RANS-models are not capable of handling this kind of flows with satisfactory accuracy; in the case of sphere, the predicted drag coefficient is often less than half the measured value. Also it was found that quality of numerical grid in area of separation is crucial for proper simulation of separation wake.

To obtain numerical simulation of flow around the sphere with satisfactory accuracy grid with more than 1,000,000 finite volumes has to be used. With finer mesh in separation region number of finite volumes exceeds 5,000,000. For numerical simulations of unsteady flow with such number of finite volumes an enormous numerical power has to be used. In simulation of unsteady flows a lot of flow variables information travel trough numerical domain. Consequence of this behaviour is a high exchange of information between domains in a case of domain decomposition usage and demand for less as possible latency for network, which connect parallel computer nodes. A classic cluster made of workstations with standard network is not capable to fulfil demands for high level of communications between nodes. Just a dedicated supercomputer cluster with low latency network is suitable for such operations. Trough HPC Europa project we got possibility to use supercomputer BCX which has demanded performances.

Results of simulations

We made massive numerical simulations of fluid flow around a sphere for sub-critical Reynolds number of 50,000. For this Reynolds number the flow over sphere is far from equilibrium, unsteady, and experiences transition to turbulence.

The numerical domain is rectangular, 355mm wide, 407mm high, and 1m long. A sphere with diameter D = 61.4 mm is supported by a 5 D long cantilever with diameter 0.125D. The free stream velocity was U0 = 12.66 m/s, which corresponds to Re = 50,000. The problem geometry and physical conditions were chose that results of simulations can be directly compared with measurement in Low Turbulence Wind Channel on a Faculty for Mechanical Engineering, University of Ljubljana.

Polyhedral numerical grid around a sphere with 7,000,000 finite volumes is shown on Fig.1. On the detail it can be seen finer mesh in the separation region. The numerical code used is a general finite volume Navier-Stokes solver called Star ccm+, CD-adapco. The solver is based on unstructured grids with arbitrary shape of control volumes. The collocated variable arrangement with the SIMPLE algorithm for pressure-velocity coupling is used. The code is parallelized using grid decomposition and the message passing interface system MPI. For the advancement in time, the implicit three-time level scheme is used. Second order discretisation methods are used, except in regions far from the sphere and wake where the central-differencing scheme (CDS) for convection is combined with upwind-differencing scheme (UDS) to avoid possible oscillations.

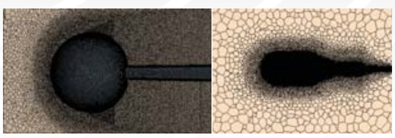

Figure 1 *Polyhedral numerical grid around the sphere supported by cantilever in the longitudinal cross-section.*

Numerical simulations were performed on IBM BCX supercomputer, CINECA simultaneously on maximum 128 Opteron, AMD processors with 2GB RAM per processor. Four processors are connected in one node connected with HyperTransport link. For communication between nodes Infiniband network is used. In the process of numerical calculations 85% utilisation of processors was achieved. Communications between processors inside one node has very low latency because of usage HyperTransport bi-directional parallel high-bandwidth point-to-point link between processors. For the communications between nodes high-performance switched fabric topology Infiniband was used with less effectiveness. It is important to notice that on BCX LSF batch job system was installed and many times chose nodes for calculations on different communication branches.

Numerical simulations were performed for unsteady flow with RANS models: K-ε, K-Ω and SST. It was found that although flow was calculated as unsteady and fine mesh was used all RANS simulations tend to incorrect stationary solution. Because of this the results of simulations with RANS models were rejected. Results obtained with LES are better than with DES but the penalty is almost doubled computational time. Because of report volume we will present just result of LES simulations on fine grid. For calculation of unsteady flow time step was gradually lowered with simulation process down to 50 microseconds. After unobstructed flow makes ten length of domain, fluid flow was considered as developed. Time averaging of flow variables was done in time need for unobstructed flow to make next three length of domain. There are calculated values of mean and instant values of velocity and pressure. Variance and covariance of velocity components and pressure are also calculated, but because of size of report just the most interested will be presented.

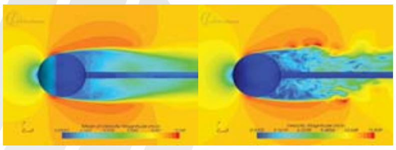

Figure 2 *Mean and instant velocity field around sphere supported by cantilever obtained by LES on fine mesh in the longitudinal cross-section.*

On the fig. 2 mean and instant value of velocity fields are presented. Mean velocity field on the left show symmetry and the maximum velocity near the sphere before equator. Recirculation point is approximately 1.5D after sphere. On the left side time averaged velocity field is shown. The maximal velocity is not near sphere but in the wake, which travel on the separate boundary layer.

On the fig. 3 mean and instant pressure filed around sphere in longitudinal cross-section was presented. Mean values of the flow on the right sight show minimal unsimetry, which can be suppressed by increasing of averaging time. Difference between mean and instant value of pressure in stagnation point is less than 0.2 percent corresponding to theoretical value, which strongly argues validity of results. In the case of mean flow area of the minimum pressure is on the sphere before equator. But on the instant pressure field the minimum is located in the separated boundary layer and is moving downward with wakes. In the time of simulation calculations time sequence of instant of fluid flow values were produced with time step 20 microseconds. They are integrated in the films in which are possible to observe phenomenon of pressure field and velocity extremes moving with the wake.

In the time of HPC-Europa visit we start to check flow energy balance also although fluid was accepted as incompressible and energy equation is not solved directly. We found that LES and DES provided wakes behind separation area with flow parts consisting of higher total energy than inlet - burst of overshot energy. Sphere represents a passive object, which takes energy from fluid flow by viscosity effects. This means that in the field around sphere total energy in the flow not affected by sphere will be equal and in the flow affected by sphere lower than inlet total energy.

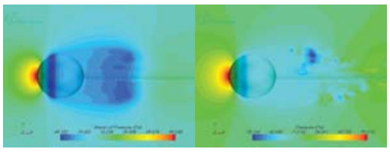

Figure 3 *Mean and instant pressure field around sphere supported by cantilever obtained by LES on fine mesh in the longitudinal cross-section.*

On Fig.4 it is presented mean value of total energy in meaning of the pressure, which proves statement. Mean and instant value on stagnation point differs from theoretical one by 0.5 percent. On the right side instant value of total energy in meaning of the pressure is presented. In the wake it can be seen that instant total energy is higher than inlet. These areas are travelling downstream with the wakes. The overshooting of energy is more than 20 percent and cannot be accepted as numerical error concerning a good agreement of stagnation point pressures. Although time averaging of total energy gives very good results showing that exactly with sphere affected flow loses energy. Our presumption is that numerical simulation found mechanism that can momentarily concentrate energy in appointed area and consecutivenessly increase instability of the flow. This instability has crucial influence on wake development. Although is very rare to finding phenomena with numerical simulations earlier than with experiments it is important to observe that it is difficult to build a sensitive pressure gage with fast response to observe fast pressure change in the field. Although this very interesting phenomenon, it has to be independently verified by experiments in low turbulence wind channel to be widely accepted.

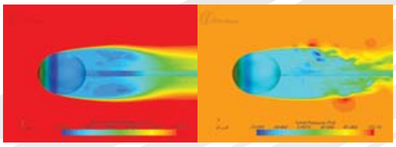

Figure 4 *Mean and instant energy field around sphere supported by cantilever obtained by LES on fine mesh in the longitudinal cross-section*

The last proves that massive parallel simulations are possible to obtain only on High Perform Computers, which are offered within HPC Europa project. It is almost impossible to perform such calculations on ordinary clusters made of single working stations connected with average networks, because even high performance Infiniband has represent major part of latency in data exchange between nodes.

Conclusions

The numerical simulations performed in CINECA support by HPC-Europa program fulfil expectations completely. Even more we found a new phenomena which we can verify after we develop fast response sensitive pressure measuring device. Idea for analysing of total energy was developed in the time of HPC-Europa visit. The productions of papers with new disclosures are in the process and soon they will be submitted to review.

For numerical simulating a high frequency separation on sphere, the number of finite volumes has to be rapidly increased to more than 50.000.000. which with new time step smaller than 10 microseconds will demand usage of more than 2000 processors. That is almost the whole power of BCX top500 supercomputer.

Although is very rare to find phenomena numerically before experimentally it has to be mentioned that first has to be invented new more accurate measurement device for the dynamic low pressure measurement. After that it will be possible to validate results of numerical simulations.

Viktor Šajn
viktor.sajn@fs.uni-lj.si, www.fs.uni-lj.si

Bibliography

Viktor Šajn has finished study of Faculty of Mechanical Engineering, University of Ljubljana, Slovenia 1991 with diploma title "Stress analysis of turbo compressor rotor" and finished Master study 1996 with the title "Numerical modelling of solid-fluid interaction. He finished his doctor study 2003 with the title: "Optimisation of deformable aircraft wing". From 1992 is employed on Faculty of Mechanical Engineering. Viktor Sajn designed and built Extreme Low Turbulence Wind Tunnel for laminar flow visualisation and measurements. For measurement of wind flow he developed and constructed high precision hot-wire and LDA measurement devices.

Acknowledgment

The work has been performed under the Project HPC-EUROPA (RII3-CT-2003-506079), with the support of the European Community - Research Infrastructure Action under the FP6 "Structuring the European Research Area" Programme. The program code used for numerical simulations was kindly provided by CD-Adapco

ŠAJN, Viktor. Numerical simulation of the turbulent flow around a sphere. V: ALBERGIO, Paola (ur.), ERBACCI, Giovanni (ur.), GAROFALO, Francesca (ur.), MONFARDINI, Silvia (ur.). Science and supercomputing in Europe : report 2007, (European Commission - 6th Framework Programme). Bologna: CINECA Consorzio Interuniversitario, cop. 2007, str. 677-682. [COBISS.SI-ID 10636059]