TASK QUARTERLY 2 No 2 (1998), 199-204

SHORT REVIEW OF THE CFD ACTIVITIES IN POLAND

ZBYSZKO KAZIMIERSKI

Institute of Turbomachinery
Technical University of Lodz
Stefanowskiego 1/15, 90-924 Lodz

The Computational Fluid Dynamics is being developed at several universities and scientific centres in Poland. Below, these centres are presented and a general description of the CFD activities carried out is provided.

It should be emphasised that the centres mentioned here are only the best known ones. The information provided has been prepared in co-operation with the responsible scientists and is authorised by them. Therefore the form of topics description is not uniform.

1. Warsaw University of Technology

a) Institute of Aeronautics and Applied Mechanics

Prof. Andrzej Styczek

1. Stochastic vortex blob methods for simulation of flows at high Reynolds numbers
2. Numerical investigation of the turbulence properties in the flows behind bluff bodies
3. Study of intermittency phenomena in small scale turbulence
4. Vortex particle method for simulation of flows in 3D configurations
5. Investigation of the drag and lift forces acting on moving and deformable bodies
6. The method of algebraic moments of vorticity for the viscous fluid flow

7. Investigation of dust spreading in turbulent atmosphere using random walk method

Dr. Jacek Rokicki

1. Overlapping grid technique for simulation of compressible/incompressible flows in complex geometries
2. Unstructured Domain Decomposition for the Navier-Stokes equations
3. Parallelization of the CFD codes
4. Compact high order methods for the Navier-Stokes equations
5. Investigation of fluid motion in regions with controlled and free boundaries
6. Numerical prediction of the creeping flow in the Face Seals
7. Symbolic algebra techniques for automatic generation of discretisation formulas

b) Institute of Heat Engineering

Dr. Jerzy Banaszek

FEM simulation of transient multidimensional conductive and convective heat transfer in incompressible fluids and solids with a solid-liquid phase transition includes:

1. Theoretical analysis of conditions for a proper discrete representation of fundamental physical features of the convection-diffusion transport phenomenon — "behavioural" error analysis where the question is addressed of how to fulfil the conservation and maximum principles, as well as how to reduce the mass balance error, wiggles, spurious numerical dissipation and dispersion on a coarse finite element grid.
2. Development of enhanced FEM algorithms based on:
   - the fractional step method of Chorin to decompose the coupling between the continuity and momentum equations;
   - the operator splitting technique, where operators of discrete balance equations for momentum and energy are split according to physical processes and the contribution from each of these processes is calculated separately;
   - the enthalpy-porosity model or the enthalpy-variable viscosity model to simulate isothermal and temperature-range solid-liquid phase transitions and to mimic the behaviour of fluid in the two-phase zone ("mushy region").
3. Implementation of the above mentioned theory and algorithms to calculate incompressible fluid flow and heat transfer in binary solid-liquid materials used in foundry technology and energy storage systems.

2. Cracow University of Technology

Division of Applied Mathematics at the University Computer Centre

Dr. Andrzej Karafiat, Dr. Krzysztof Banaś et al.

The section of Applied Mathematics was created at Cracow University of Technology by Prof. Leszek Demkowicz (currently at Texas Institute for Computational and Applied Mathematics) in 1991. The members of the group took part in the development of the theory and first computer codes for hp finite element and boundary element methods. Currently their efforts concentrate on the application of the adaptive finite element methods in acoustics and simulations of laminar flows of inviscid and viscous compressible fluids. The research includes investigations of theoretical problems, as well as development of efficient computer codes and their implementation in modern computational environment. The State Committee for Scientific Research remains the main sponsor of the research, currently under the grant "Algorithms and computer codes for parallel flow simulations by the adaptive finite element method".

3. Silesian Technical University

Institute of Power Engineering

Prof. Tadeusz Chmielniak, Dr. Włodzimierz Wróblewski et al.

1. Algorithms and computer programs for non-stationary Euler equations. Transonic flows in 3-D and 2-D formulation. The flow through the turbomachinery stages, mutual interference of rotating and non-rotating annular blade cascades.
2. Solutions of 2-D viscous flow problems with implementation of Baldwin-Lomax and k-e turbulence models. Stationary and non-stationary flows.
3. Algorithms and programs for the fluids complicated thermodynamically. Water steam flows with very high pressures and temperatures and for, low parameters, with water condensation.
4. An attempt to introduce the parallelization of the computational techniques using several work stations.

4. Radom Polytechnic High School

Institute of Basic Sciences

Prof. Zbigniew Kosma

1. Different problems of conformal mapping of the multi-domain contours with an application to the 2-D grid generation.

2. Viscous flow determination using analytical and numerical methods with decomposition of the velocity field and the method of artificial compressibility.
3. Algorithms for gas flows in the turbulent boundary layer in the domain of interaction between the shock-wave and boundary layer.
4. Development of general methods of the CFD, based on the application of spline functions.

5. Technical University of Wroclaw

Institute of Heat Engineering and Fluid Mechanics

Dr. Adam Wanik, Dr. Henryk Kudela

1. The algorithm and computer programs for the 3-D turbulent flow with combustion. The k-e turbulence model and different models for combustion processes have been introduced in the programs (CHAMBER).
2. Numerical simulation of thermal and hydraulic processes in the shell and tube-shaped heat exchangers. The theoretical models and practical computer codes have been elaborated.
3. Numerical modelling of the hydrodynamic phenomena by the vortex methods.

The vortex methods were used in the modelling of the hydrodynamic phenomena. The numerical results that were obtained by the vortex methods for the Rayleigh-Taylor instability and for the motion of the bubble in the gravity field were presented. The numerical results for the viscous flow in the channel with a complex geometry were also presented. For the Rayleigh-Taylor instability, it was assumed that the flow was potential and the fluid was incompressible and inviscid. The interface can be regarded as a vortex sheet. The study of the motion could be reduced to the solution of the initial value problem for the vortex sheets. From the initially elliptic bubble, a thread emerged. The evolution of the bubble was sensitive to initial conditions. The thread structure was observed in the experiments. For the study of the viscous flow by the vortex blob method, a general program was developed. To calculate the auxiliary potential flow, which was needed to satisfy the boundary condition on the wall, the fast elliptic solver with the capacitance matrix technique was used.

6. Military Academy of Technology, Warsaw

Institute of Aerodynamics

Prof. Zbigniew Dżygadło, Dr. Stanisław Wrzesień

The subject of the numerical investigations are supersonic flows around profiles and axial-symmetrical bodies. The solution takes into account a viscous, heat conductive gas. The numerical algorithm is based on the Navier-Stokes equation, the equation of mass and energy conservation. The shock waves appearing in these flows are properly determined. The flow structures, depending on the body geometry, angles of attack, Ma and Re numbers, can be analysed by means of the developed computer programs.

7. Technical University of Lodz

Institute of Turbomachinery

Prof. Zbyszko Kazimierski, Dr. Marek Rabiega, Dr. Pawel Wiewiórski

1. The subject is numerical simulation of subsonic 3D compressible and incompressible viscous (turbulent) and inviscid flows through rotating and nonrotating turbomachinery channels.

   The computer programs allow to solve 3D parabolic and 3D elliptic flows. The boundary conditions are typical for turbomachienery flows, including a moving inlet, periodicity, mutual movement of the channel walls, leakages between channels, etc. The results are presented by means of own graphical codes using the GRAPHER program as a tool and the TECPLOT postprocessing codes. The algebraic grid generation is performed under control of AUTO-CAD. Recently the main effort has been paid to parabollization of the calculation with a fluent exchange of the boundary conditions. The investigation of the numerical viscosity influence on the results of calculations have been initiated.

2. The stability of the uniform state of a gas fluidized bed is analysed numerically. The uniform fluidization state has been analysed in the frame of the two-fluid model of stability. The 2-D bed has been taken into consideration. The results of numerical calculations for particular cases of boundary conditions are presented. The calculations have shown that a simulation of a linear, as well as of a nonlinear, stage of the fluidization process is possible. The development of the process may lead in a short time to the formation of a certain type of heterogeneous state — the slug flow. The numerical results can be used in comparisons with the images of structures of a real system.

8. Institute of Fluid Flow Machines

Polish Academy of Sciences in Gdansk

a) Numerical Fluid Mechanics Department

Prof. Wlodzimierz Prosnak, Prof. Ewa Klonowska -Prosnakowa

1. The general method of conformal mapping of the arbitrary set of multi-domain contours have been elaborated and presented in the form of computer codes.

The method is widely used for 2-D grid generation and solution of 2D inviscid, incompressible flows.

2. General methods of the Navier-Stokes equations are in progress. The first one is based on the application of spline functions and the second one belongs to the class of spectral methods. The spectral method introduces the holomorphic functions containing the rational functional series with unknown complex coefficients. Several linearised Navier-Stokes problems have been solved recently by means of the above mentioned spectral methods. The computer programs for these calculations have been developed.

b) Gas Dynamics Department

1. Numerical simulations of transonic flows (*Prof. P. Doerffer, Dr. J. Kaczyński, MSc. J. Czerwińska*)
   — shock wave – boundary layer interaction in internal and external flows (nozzles, curved channels, wings, helicopter rotors),
   — shock induced separation (incipient separation, numerical recognition of 3-D flow structures in separated flow),
   — turbulence modelling in transonic flows.
2. Numerical modelling of multiphase flow (*Prof. Z. Bilicki, Prof. J. Badur, Dr. D. Kardaś, Dr. J. Pozorski*)
   — flashing flow,
   — condensation flow,
   — unsteady flow of water – air mixture,
   — turbulent combustion of gases,
   — Lagrangian PDF (probability density function) for turbulence description,
3. iscous, compressible, 3-D simulations of flow through a multistage turbine, design of curved blades (*Dr. A. Gardzilewicz, Dr. P. Lampart*)

c) Ship Propeller Department

Prof. J. Szantyr, Prof. T. Koronowicz, Dr. T. Bugalski

1. CFD methods for prediction of potential and viscous ship hull flows, including free surface and hull-propeller interaction effects, leading to resistance and propulsion analysis of ships.
2. CFD methods for determination of the acoustic and hydrodynamic pressure field connected with ship hull flow and operation of a cavitating propeller.
3. Unsteady lifting surface and surface panel methods for the analysis of screw propellers operation, including cavitation prediction and unsteady hydrodynamic foces calculation.