Computational fluid dynamics: research activities and services at NRC
National Research Council of Canada. Transportation Technology Program

For the publisher's version, please access the DOI link below./ Pour consulter la version de l'éditeur, utilisez le lien DOI ci-dessous.

Publisher's version / Version de l'éditeur:
https://doi.org/10.4224/21274668
Report (Transportation Technology Program); no. TTP-02-92, 1992-09

National Research Council Canada
Conseil national de recherches Canada

Transportation Technology Program
Programme de technologie des transports

NRC · CNRC

Computational Fluid Dynamics
Research Activities and Services at NRC

September 1992

TTP-02-92

NRC 32441

Canada

Contents

Foreword 1

IME Coastal Zone Engineering Program
Hydraulics Laboratory 2

IRC Building Performance Laboratory 3

IMD Numerical Wave Tank Program
Computational Hydrodynamics Laboratory 5

IMD Ship Dynamics Simulation Program
Computational Hydrodynamics Laboratory 6

IME Combustion and Fluids Engineering Program 7

IMD Arctic Vessel Research Laboratory 8

IAR Aerodynamic Applications and Developments
High Speed Aerodynamics Laboratory 9

IAR Unsteady Aerodynamics Program
Applied Aerodynamics Laboratory 10

IME Cold Regions Engineering Program 11

IMI Modelling of Casting Processes
Computer Integrated Materials Processing 12

Foreword:

Computational Fluid Dynamics (CFD), or the numerical simulation of aero and hydrodynamic processes, is playing an increasingly important role in design and production processes in Canadian industries. Among many other applications, the design of ships and aircraft, the simulation of flow of molten metals in casting processes, the computation of combustion and of air flow, weather, tidal and wave-forecasting, atomic research, environmental assessments and coastal and river engineering are only a few examples where numerical methods are used as indispensable design aids. Within NRC's Transportation Technology Program CFD contributes substantially in the areas of waterways (flow and vessel behaviour), road, rail and air traffic (air flow around vehicles and aircraft) to improve reliability, efficiency and safety in transportation. In combination with powerful computers and sophisticated visualization methods, the use of mathematical models has become a daily necessity in many areas of production and research.

While some of the processes involved are still not fully understood physically, approximations and empirical expressions help to bridge that gap. The researcher uses CFD parallel to experimental work to quickly assess the feasibility of certain routes to obtain the goal. The access to experimental facilities at NRC together with the confirmation through analytical projection puts the researcher into a strong position to verify and improve numerical codes.

Moreover, the continuous work on the development, validation and improvement of numerical simulation systems in cooperation with companies, universities and other laboratories nationally and internationally takes NRC's engineering institutes to the leading edge of technology. This also creates the necessary base to assist Canadian industries in their quest for excellence and support them in international competitions.

The following brochure provides an overview of the major CFD activities in NRC. It lists the software being developed or used, the computer facilities and research activities in various areas where CFD is being applied. Further information can be obtained from the contacts indicated with names, phone and fax numbers.

Volker Barthel
Chairman, NRC CFD Group
Tel: (613) 993-6649
Fax: (613) 952-7679

Terry Maloney
Coordinator
Transportation Technology Program
Tel: (613) 993-9899
Fax: (613) 954-1473

Institute for Mechanical Engineering
Coastal Zone Engineering Program
Hydraulics Laboratory

Objective:

Simulation of hydrodynamic processes in the coastal zone of oceans and lakes, as well as in rivers and estuaries.

These processes include:

- wave propagation and wave disturbance in shallow and intermediate waters,
- tidal flow and circulation, salinity,
- sediment transport under waves and currents,
- pollutant spreading.

This work helps Canadian engineering firms to compete internationally. It supports government departments and agencies mandated with establishing and maintaining safe waterways, harbours and coastal structures and provides a deeper understanding of hydrodynamic processes for coastal researchers.

Numerical Tools:

- 2D-finite difference model for tidal and short wave propagation (St. Venant equations).
- multi-layered 2D-finite element code based on Navier-Stokes equations with advection-dispersion capabilities and sediment transport module.
- 3D-beach evolution model (under development).
- interactive grid generators for regular and irregular FD and FE meshes.
- extensive pre- and post-processing software, including colour plots and animation.

Hardware:

- SGI 4D80GTB Supergraphics Workstation
- SGI 4D35 32 MHz Power Server
- SGI indigo Workstations
- QPS 100 Postscript Colour Plotter
- RGB Scan Converter with High-Res Videomachine.

Research Activities:

Continuous verification and improvement of the existing modelling systems through comparison with experimental and field data; development of a common database and pre- and post-processing "expert" shell for all models.

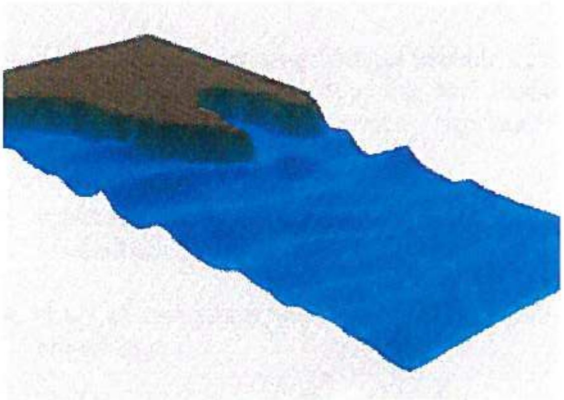
Active collaboration between NRC Coastal Zone Engineering Program and: The Danish Hydraulic Institute, Hydraulics Research Ltd. Wallingford, the University of Mississippi, and many other Canadian and international universities and institutes.

Who can use our numerical tools?

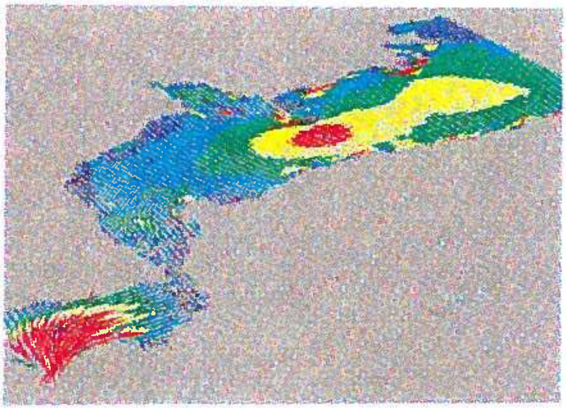
Any institution in the private or public sector which needs numerical tools in support of design tasks or for the solution of environmental problems will be accommodated within the capabilities of the simulation systems available. The Hydraulics Laboratory will either run the simulation for the client or provide the necessary support to enable the client to operate the system on the dedicated computer system.

Contact:

Dr. Volker Barthel
Tel: (613) 993-6649
Fax: (613) 952-7679



Surface Structure of Waves in a Harbour



Velocity Vector Fields in an Estuary

Institute for Research in Construction
Building Performance Laboratory

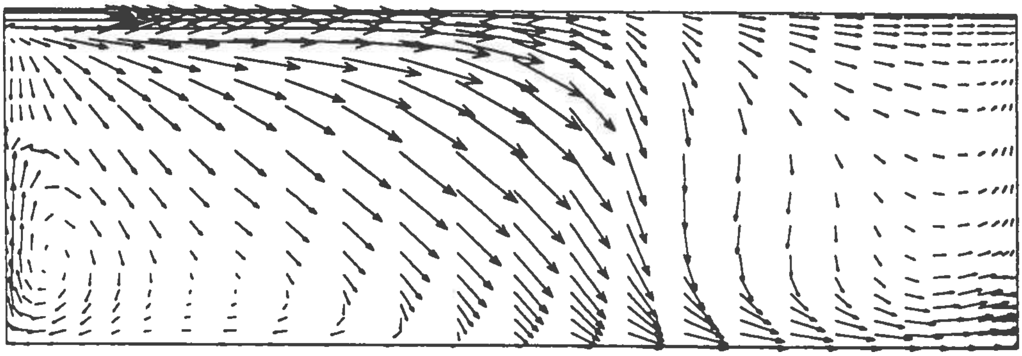
Objectives:

Apply CFD techniques and tools to:

- study airflow patterns and the distribution of air temperature and contaminants within buildings with respect to thermal comfort and indoor air quality.
- evaluate and control air, heat and moisture movement through building envelope components and systems.
- development of design guidelines for short and long term hygrothermal performance of construction assemblies.

Numerical Tools:

- 3-D code, a public domain program developed for room air flow applications.
- 2-D general application code.
- 3-D code (TWIST - Turbulent WInd Simulation Techniques) for wind flow applications.
- 2-D hygrothermal porous flow code.



Indoor Air Flow Patterns

Current Research Activities:

- study of ventilation effectiveness in a typical office space equipped with work stations.
- evaluation of CFD codes by comparing with laboratory experimental data.
- development of design guidelines to control rain penetration into wall assemblies and curtain wall systems using pressure equalization principle.
- simulation of wind driven rain patterns and wind effects on buildings.
- study of two phase moisture flow in building materials.

Hardware:

- 2, HP Apollo work stations (Model 720,730 CRX)
- 1, Silicon graphics workstation (Model IRIS INDIGO SGI)
- 1, Colour Graphic PaintJet XL Printer

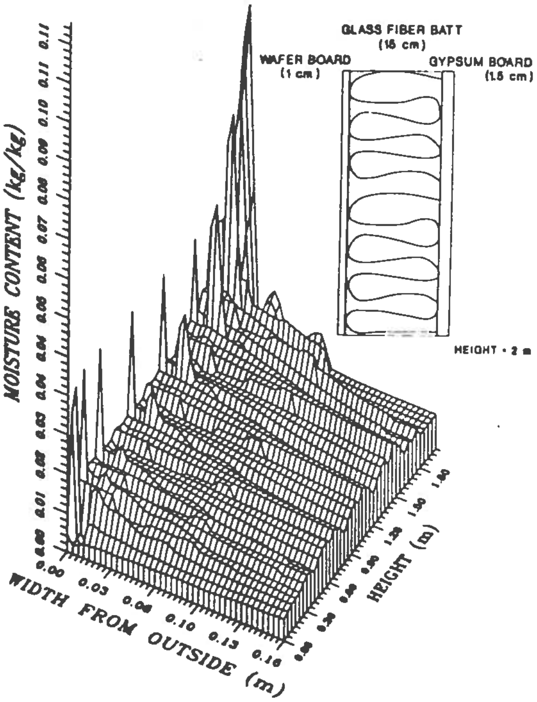
Contact:

Dr. Bas A. Baskaran
Tel: (613) 990-3616

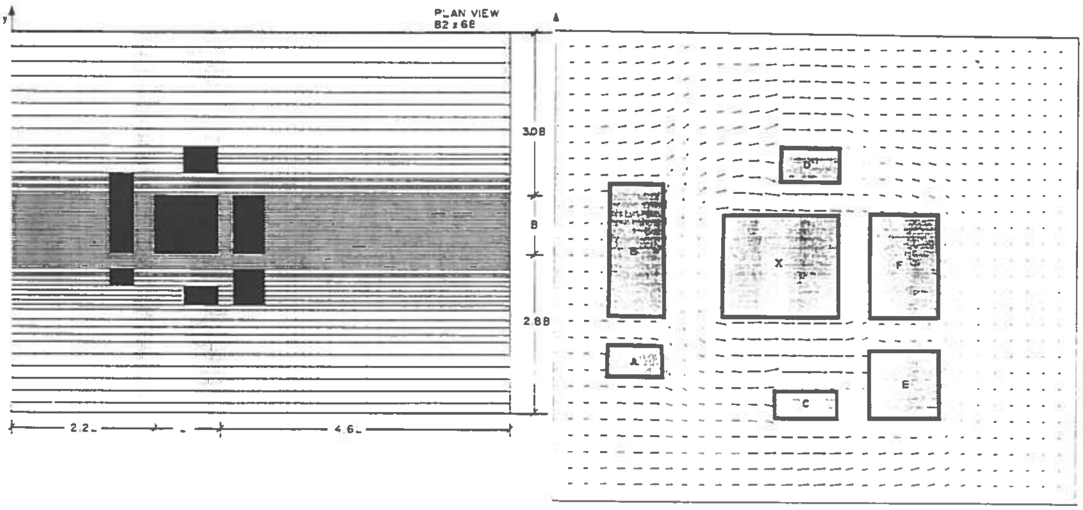
Dr. Achilles Karagiozis
Tel: (613) 993-0582

Dr. M. Nady Said
Tel: (613) 993-5938

Fax: (613) 954-3733



Moisture Distribution in Walls



Wind Flow around Buildings

Institute for Marine Dynamics Computational Hydrodynamics Laboratory NUMTANK: Numerical Wave Tank Program

Objectives:

Time domain simulation of the propagation of linear and nonlinear free surface waves in the presence of fixed or floating structures; in particular:

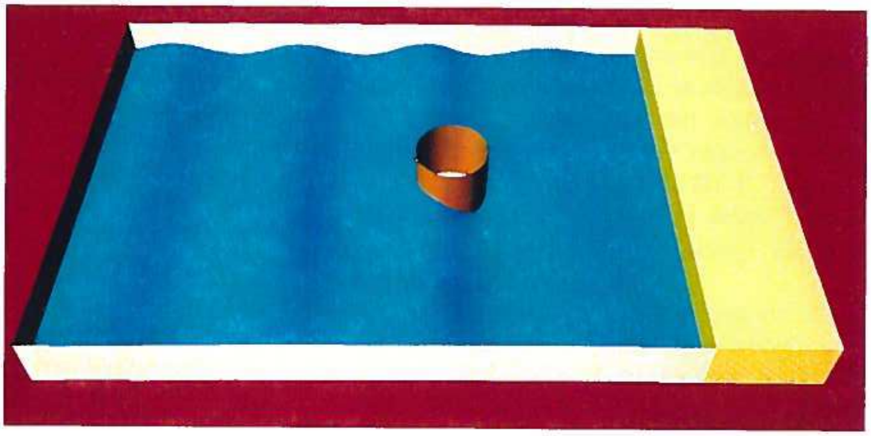
- 2D and 3D NUMerical wave TANK models with generatron facilities.
- propagation of linear and nonlinear waves.
- diffraction of waves due to the presence of fixed and floating surface piercing structures.
- wave patterns and wave-making resistance induced by submerged or floating three dimensional vessels.
- Extensive pre- and post-processing software, including 2D and 3D colour plots, 3D surface rendering and animation.

Hardware:

Stardent mini-super computer; IBM RS/6000 Powerstation 320; Colour plotter and laserjet printers; NCD 19C colour X terminals.

Research Activities:

- Improvement of the existing systems through the inclusion of Lagrangian and/or Eulerian formulations.
- Study of radiation boundary conditions.
- Verification of numerical models through comparison with experimental data.
- Active collaboration between CHL and: DTRC, Bassin d'Essais des Carenes (France), Canadian Coast Guard.



The development of state of the art software technology and expertise to help Canadian engineering firms to compete internationally. This work also supports government departments and agencies mandated to establish improved safety and effectiveness of ship operations in waves.

Numerical Tools:

- 2D-Higher Order Boundary Element models for the nonlinear time domain propagation of waves in the presence or absence of submerged or floating structures.
- 3D-Higher Order Boundary Element models for the time domain generation and propagation of waves and their interaction with structures.
- 3D double-body potential flow model.
- 3D Dawson wave resistance model.
- 3D Newmann-Kelvin wave resistance model.
- Body surface grid and panel generators.

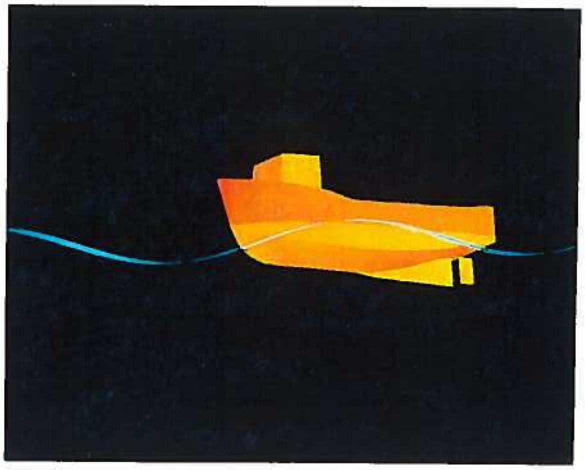
Potential users of CHL numerical tools:

Any institution in the private or public sector which needs numerical and computer simulations in support of research, design and development in the field of Marine Vessel Dynamics will be accommodated within the capabilities of the available systems. The programs developed by CHL may be utilized at various stages of research, design and operation evaluation. The prediction of forces and motions expands the capability of the regulator, designer and operator and can be used as an effective design optimization tool, especially to determine extreme responses of marine vessels systems.

Contact:

Dr. Jacek Pawlowski
Head of Computational Hydrodynamics Laboratory
Tel: (709) 772-2474
Fax: (709) 772-2462

Institute for Marine Dynamics Computational Hydrodynamics Laboratory SDSP: Ship Dynamics Simulation Program



Objectives:

Computation of time histories of fluid loads and motions of displacement type ships operating in heavy seas:

- The forward speed and arbitrary course angle are slowly varying relative to the wave excitation.
- Oncoming wave flow defined by a long-crested Stokes wave or by a long- or short-crested wave elevation spectrum.
- Modules for 3D-nonlinear, time domain response of vessels in extreme wave conditions.
- Vessel free floating moving forward or attached to any mooring system (viscous effects included).

Development of state-of-the-art software technology and expertise to help Canadian engineering firms to compete internationally. This work supports government departments and agencies mandated with establishing safety and effectiveness of ship operations in waves.

Numerical Tools:

- Isoparametric panel generators.
- Wave generator module: instantaneous elevation, velocity and pressure fields induced by regular and irregular waves.
- The M-WAMIT module: added mass and damping coefficients.
- Wet body generator: the instantaneous wetted surface of the ship hull at each time step.
- Normal equations solver: solves a set of normal equations for every wetted section of the hull.
- Scattering solver module: scattering modal generalized forces due to the modal excitations on every hull section.
- Viscous solver: generalized forces due to viscous flow effects.
- F-KS module: generalized Froude-Krilov wave forces on the hull.
- Motion solver module: integrates the six equations of rigid body motion of the ship due to the generalized total fluid loads.
- Slow DSP: slowly varying loads and motions (moving, wind and current forces).
- Extensive pre-and post-processing software, including 2D and 3D colour plots, 3D surface rendering and animation.

Hardware:

Stardent mini-super computer; IBM RS/6000 Powerstation 320; Colour plotter and laserjet printers; NCD 19C colour X terminals.

Research Activities:

- Validation of SDSP results with experimental data for a Series 60 vessel in steep waves and for a stern trawler in steep waves, and for statron keeping of a support vessel.
- Validation of SEMDSP results (semisubmersible version of SDSP) against experiment for the Ocean Ranger and ITTC semisubmersibles.
- Improvement of the existing systems through the inclusion of a Dynamic Positioning module.
- Active collaboration between CHL and: MIT, MARTEC Ltd. (Halifax), Memorial University, Marine Institute St. John's.

Potential users of CHL numerical tools:

Any institution in the private or public sector which needs numerical and computer simulations in support of research, design and development in the field of Marine Vessel Dynamics will be accommodated within the capabilities of the available systems. The programs developed by CHL may be utilized at various stages of research and/or design. The prediction of forces and motions expands the capability of the designer and can be used as an effective design optimization tool, especially to determine extreme responses of marine vessel systems.

Contact:

Dr. Jacek Pawlowski
Head of Computational Hydrodynamics Laboratory
Tel: (709) 772-2474
Fax: (709) 772-2462

Institute for Mechanical Engineering Combustion and Fluids Engineering Program

Objectives:

Development and evaluation of versatile CFD tools for combustion, fluid flow and heat and mass transfer problems.

Applications:

- Reduction of pollutant emissions in combustion systems (such as gas turbines, industrial furnaces and burners).
- Modelling of turbulence, chemistry, liquid spray with/without combustion, thermal radiation, soot radiation, and pollutant formation.
- Study of various fluid flow systems, such as:
  - Waterjet nozzles,
  - Turbine nozzles, and
  - Horseshoe vortices in front of obstructions.

Software:

- TURCOM (a rectilinear coordinate code) and TURCOM-BFC (a body-fitted coordinate code), for 2D/3D gaseous and liquid spray combustion and for other fluid flow processes. TURCOM-PDF for probability density functions f flow variables in turbulent flames.
- WATERJET code for cavitating water jets.
- ORTHO, a grid generation code for 2D/3D rectilinear coordinates and bipolar coordinates, with multiple openings and stairsteps.
- NRCPLOT, a post-processor using DISSPLA to display static pictures.
- NRCINTER, a pre-processor for NASA plotting software (FAST, PLOT3D, GAS) for flow visualization and animation.
- PRETURCOM, a frontend to integrate the grid generation code, the CFD codes, and the post-processors.
- FLOW3D, a 3D Harwell code using non-staggered and non-orthogonal grids, with three different turbulence models. Multi-block grids for complex geometries. Flows incompressible, weakly compressible or fully compressible. Combustion and many other important features.

Hardware:

- SGI 4D35A Personal IRIS workstation with 104 MB.
- SGI 4D220S Supercomputing Server with 64 MB.
- IBM RS6000 Powerstation 530 with 64 MB.
- Tektronix Phaser II PXi Colour Printer.

Benefits of CFD Approach:

- Better understanding of physical processes leading to optimization of performance of various combustion and fluid flow systems.
- Reduction of R&D effort (cost and time).

The laboratory welcomes inquiries from industry or other organizations regarding assistance or collaboration in numerical solution of combustion or fluid flow problems. This Program already collaborates with the Naval Research Laboratory, Washington, D.C., on sooting flames, and would be pleased to offer its expertise to other potential partners.

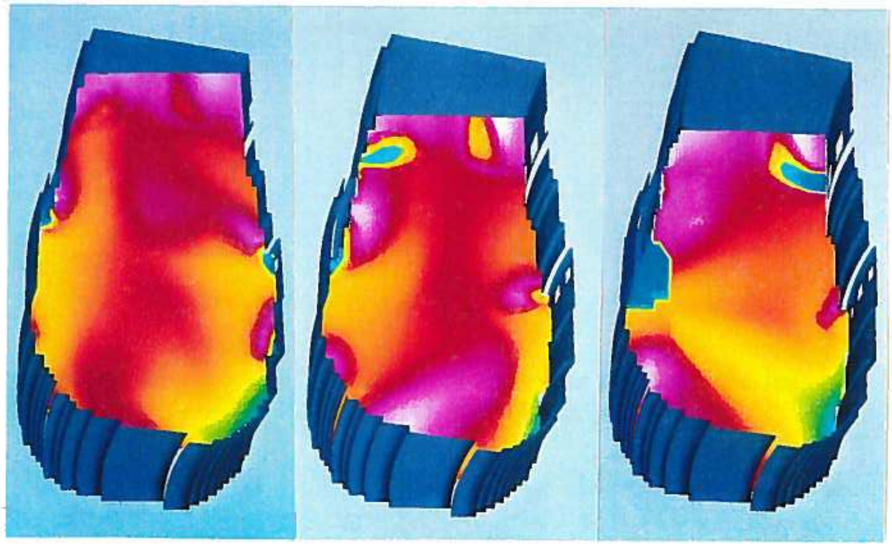
Contact:

Dr. A.H. Makomaski (Program Head)
Tel: (613) 993-2442

Dr. Mark Lai
Tel: (613) 993-9496

Dr. John Lau
Tel: (613) 993-9497

Fax: (613) 952-1395



Computed temperature distributions in a gas turbine combustor

Institute for Marine Dynamics Arctic Vessel Research Laboratory

Objective:

To model the "Constricted Hydrodynamic Flow due to Ice Blockage under Proximity Condition".

Background Information:

This work constitutes a part of the project proposed under a joint research program agreement (JRPA-6) between Canada and Finland for the development of theoretical models on the propeller/ice interaction to support the revision of the Arctic machinery regulations. The Canadian side, as funded by Canadian Coast Guard and Transportation Development Centre, both of the Ministry of Transport Canada, is responsible for the non-contact phenomena. The objective is to construct a physically-sound theoretical, experimentally-calibrated model for calculating the loading of propellers, that can be verified with existing full-scale data. The ultimate goal is to create a practical dimensioning tool for propulsion machineries in ice navigating vessels, and to provide a model capable of being developed for regulatory purpose as well.

As discovered during the full-scale trial (1984) of M.V. Robert LeMeur, the maximum non-contact blade force due to ice blockage is 5.3 times the open-water hydrodynamic force. A series of model tests has been conducted at the NRC/IMD ice tank to further verify this observation. It was identified that when the propeller blades pass the ice blocks with little clearance, high blade-load impulses and subsequently high peak loads occur.

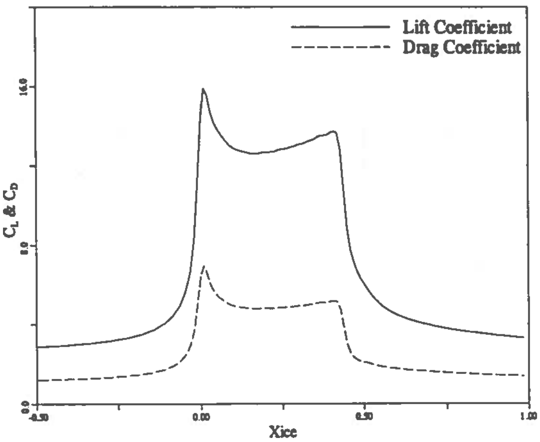
Task and Simulation Technique:

In the initial phase, we conduct a feasibility study by considering the flow over a blade profile in two dimensions with time dependence. To solve this problem, we use the direct boundary element method. The numerical results indicate a peak load about 5.7 times that computed for open water. A more realistic time-dependent flow simulation of three dimensions over the entire four-blade propeller will be conducted in the fiscal year 1992-93.

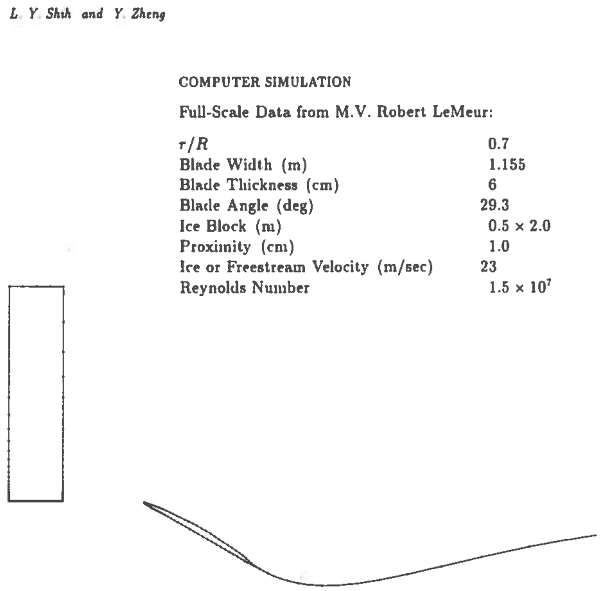
Contact:

Dr. L.Y. Shih
Tel: (709) 772-2475
Fax: (719) 772-2462

Mailing Address:
P.O. Box 12093, Station A
St. John's, Newfoundland A1B 3T5



Lift Coefficient & Drag Coefficient (Ice=0.5,X2.0, Vs=0.0m/s)



Constricted Flow due to proximate ice blockage over a blade profile in 2D

Institute for Aerospace Research
High Speed Aerodynamics Laboratory

Aerodynamic Applications and Developments

Objectives:

Besides supporting DND's CF-18 program, several other aerodynamic projects are under development at the High Speed Aerodynamics Laboratory. The objectives are to develop software tools in support of wind tunnel calibration and model testing (wind tunnel wall interference is a major concern) and also to design improved airfoils and analyze the flow about airfoils and wings.

Numerical Tools:

- Grid generation techniques, structured and unstructured.
- Small disturbance, full potential codes.
- Euler methods based on Jameson's finite volume method and on multiblock TVD schemes.
- EAGLE code for Euler solutions about full aircraft.
- Supersonic Euler methods SWINT and ZEUS.
- ARC2D Navier-Stokes method for airfoils.
- Small disturbance and full potential unsteady methods.
- ARC3D

Hardware:

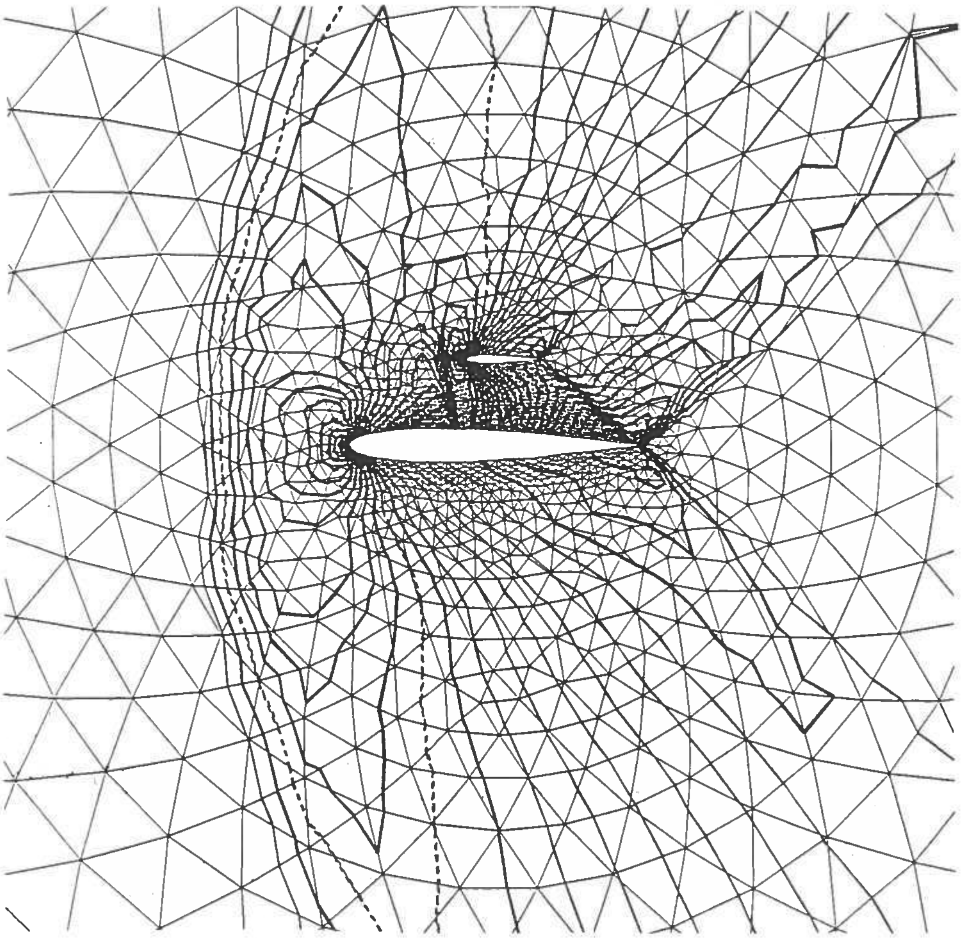
- IBM RISC 6000 workstations
- Silicon Graphics IRIS workstation
- IBM 3090 mainframe
- Colour plotter

Visualisation:

Sterling SSV package, PLOT3D, DISSPLA

Contact:

D.J. Jones
Tel: (613) 998-0504
Fax: (613) 998-1281



Mach Contours and Unstructured Grid for Supersonic Flow (M=1.2) about a bi-Airfoil Configuration

Institute for Aerospace Research
Unsteady Aerodynamics Program
Applied Aerodynamics Laboratory

Objectives:

The study of the unsteady aerodynamics of wings and bodies under various operational conditions. The two projects involved are:

- The simulation of delta wings at high angles of attach in ground effect. This will provide insight into the handling and stability characteristics of the high speed civil transport (HSCT) being developed in the United States.

- The investigation into the effects of surface roughness on boundary layers. The emphasis is on the performance degradation of helicopter rotor blades due to icing.

Numerical Tools:

- an unsteady Vortex Lattice Method with optional ground effect.
- a 2D steady Euler code.
- a 3D unsteady finite volume Euler code (under development).
- a 2D steady compressible Navier-Stokes code (under development).
- a 3D unsteady compressible Navier-Stokes code (proposed).
- a 3D multi-block elliptic grid generator.
- a 2D code for airfoils with rough surfaces.

Hardware:

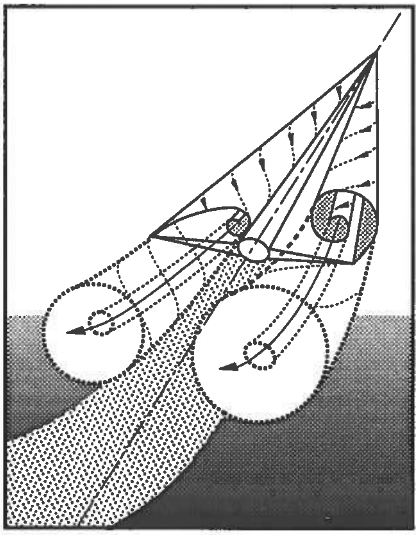
IBM RS6000 workstation

Research Activities:

The Applied Aerodynamics Laboratory is dedicated to the formation of an integrated research environment that includes both computational and experimental work. Alongside the code development and verification, complimentary research is performed in one of four wind tunnels which include the 2m x 3m Low Speed Wind Tunnel and the 5m Diameter Vertical Wind Tunnel.

Contact:

Dr. Jerry Syms
Tel: (613) 993-1145
Fax: (613) 957-4309



Institute for Mechanical Engineering
Cold Regions Engineering Program

Objectives:
Numerical analysis of engineering problems involving fluid flow, heat and mass transfer. All aspects of CFD from algorithm development to industrial application are considered.

Software:
Software based around the finite-volume method is capable of analysing 3D transient or steady-state, single- or two-phase viscous flow with turbulence, heat or mass transfer, chemical reactions, etc., using structured non-orthogonal body-fitting grids. The current software suite includes:

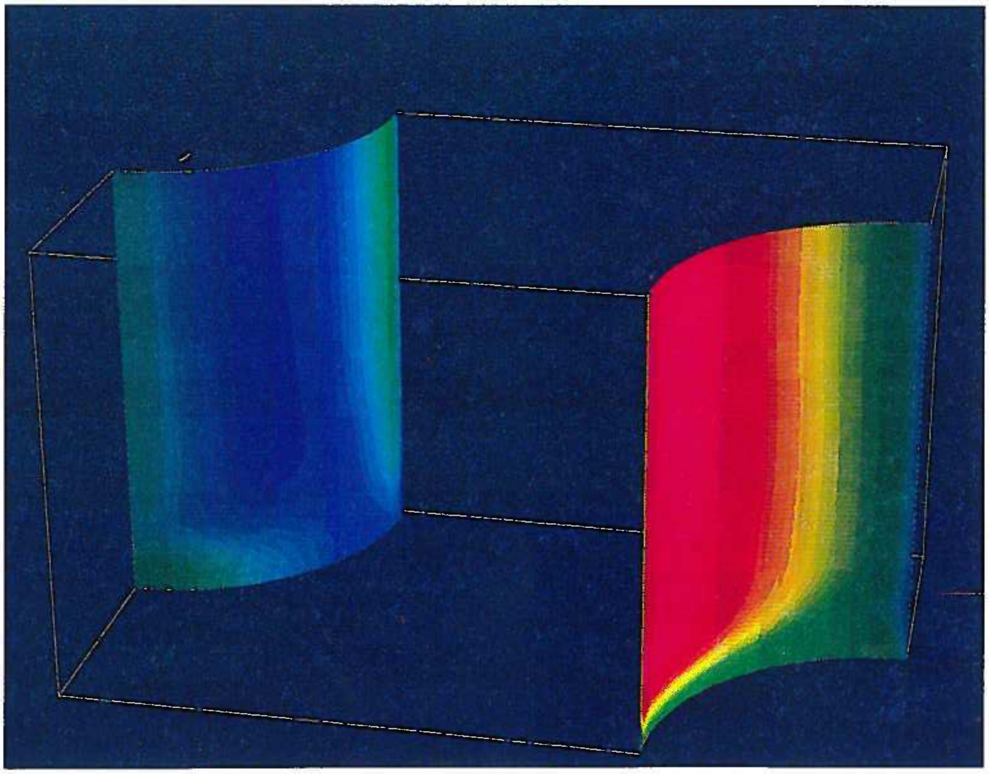
- PHOENICS - General purpose commercial CFD program (6 years experience)
- NASA PLOT3D, GAS, and FAST for flow visualisation and animation.
- In-house software. Used for grid generation, physical modelling.

Hardware:
- IBM RS6000 model 530 computer with 64 MB.
- SGI Indigo Elan Graphics workstation with 40 MB.
- DEC Local Area VAXcluster.

- TCP/IP and DECNET access.

Recent Research Activities:
- Flow and heat transfer in offset-fin and fin-and-tube heat exchanger modules.
- Transient analysis of shear layer instability/vortex shedding in in-line and staggered tube banks.
- Numerical generation of streamlines and surfaces in 3D.
- Stress analysis using finite-volume-based methods.

Contact:
Steven Beale
Tel: (613) 993-3487
Fax: (613) 954-1235
Email: BEALE@NRCM17.NRC.CA



Local Nusselt Number Distribution on the surface of a Cylinder in a Heat Exchanger

Industrial Materials Institute
Computer Integrated Materials Processing

Modelling of Casting Processes

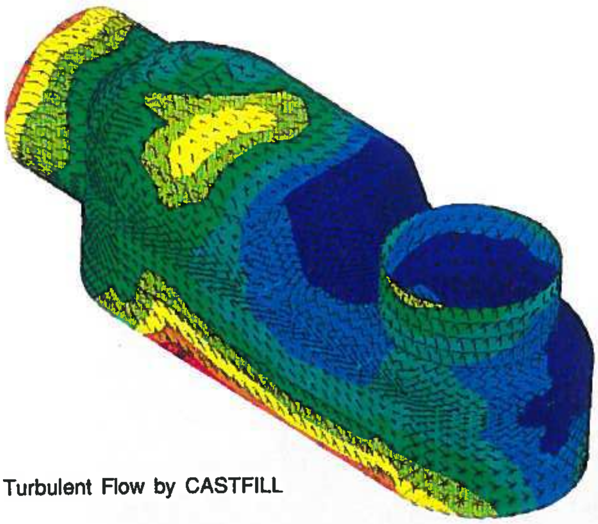
Objectives:
Computer modelling of activities related to pressure and gravity casting of metal alloys constitutes a major part of the research program of the Metals group within the Computer Integrated Materials Processing (CIMP) section of the Industrial Materials Institute.

Research Activities:
Expertise with the group includes modelling of metal flow under turbulent and laminar conditions and metal solidification in the die/mould cavity by the finite element method. Flow and solidification fronts and patterns are generated on 3D finite element mesh representations of the castings being investigated. The information developed can then be used by the manufacturer to evaluate and optimise cavity design, predict potential defects in the castings, and formulate manufacturing requirements.

Benefits:
As principal benefits, computer modelling provides more wholesome and rapid solutions to engineering and manufacturing problems, reduces reject rate through optimizing product and process design and shortens produced development cycle time.

Contact:
C.A. Loong
Program Manager - Metals
Tel: (514) 641-2280
Fax: (514) 641-4627

Address:
75 de Mortagne Boulevard
Boucherville, Quebec J4B 6Y4



Turbulent Flow by CASTFILL

Solidification by CASTSOL