Applications

CFD (Computational Fluid Dynamics)

AutoReaGas

Bence Gerber
Vice President, Marketing
Century Dynamics, Inc.
2333 San Ramon Valley Blvd.
San Ramon, CA 94583
USA
510-552-1600
510-552-1609 (fax)
big@centdyn.com
http://www.centdyn.com

AutoReaGas is an interactive, integrated Computational Fluid Dynamics program which analyzes combustion in flammable gas mixtures and subsequent blast effects. Gas cloud explosions, including flame propagation, turbulence, and the effects of obstacles in the flow field are modeled. Industrial applications include risk and safety assessment for petrochemical and process plants, nuclear plants, and offshore platforms. AutoReaGas is fully three-dimensional with an integrated analysis environment: model setup, analysis and post-processing of results are included in a single menu driven package. The graphics oriented AutoReaGas program incorporates state-of-the-art solutions to complex accident scenarios, thereby allowing realistic quantification of hazardous risk.

IRIX version compatibility:

CFD2000™

Ron Jewell
Marketing Manager
Adaptive Research Corporation
4960 Corporate Drive
Suite 100-A
Huntsville, AL 35805
USA
205-830-2620
205-830-2628 (fax)
rejewell@arc-cfd.com

CFD2000™ is an advanced computational fluid dynamics system for UNIX workstations, mainframe supercomputers, and high performance PCs. CFD2000 is a fully-integrated software system providing a common user interface based on advanced 3D windowing schemes, automatic mesh generators and the powerful Storm flow solver methodology. CDF2000/Storm uses an efficient and flexible structured-grid finite-volume solver for guaranteed conservation of mass, momentum and energy, and is the ultimate in user flexibility for special boundary conditions and fluid properties.

IRIX version compatibility:

CFX 4.1 (CFDS-FLOW 3D)

Michael Nieburg
President
AEA Technology
Engineering Software, Inc.
2000 Oxford Drive
Suite 610
Bethel Park, PA 15102
USA
412-833-4820
412-833-4580 (fax)
info@cfds.com

CFX 4.1 (CFDS-FLOW3D) is a flexible, mature, validated CFD package with fully integrated grid generator, solver and post processor. It is the only CFD code which is able to utilize either a multiblock or fully unstructured (quadrilateral/hexahedral elements) grid and handle the full range of physical models including:

- Multiphase (continuous Eulerian-Eulerian and dispersed Langrangian-Eulerian)
- Experimentally validated spray drier model
- Monte Carlo radiation model
- Experimentally validated coal combustion model
- Turbulence K-epsilon, Reynolds stress and RNG
- Compressible/incompressible fluids
- Newtonian/non-Newtonian fluids.

CFX is the only CFD code developed, validated and tested under an ISO9001 approved quality management system.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

COPS

Thorwald Herbert
President
Dynaflow, Inc.
3040 Riverside Drive
Columbus, OH 43221
USA
614-487-9939
614-487-1059 (fax)
tht@dynaflow.com

COPS is a code for the state-of-the-art analysis of transition in flows over aerodynamic bodies such as swept wings, nacelles, or turbine blades. The code is a full implementation of the compressible nonlinear PSE in general curvilinear coordinates, with interfaces to Navier-Stokes codes and two- and three- dimensional boundary-layer codes. Initial data representative of the disturbance environment in flight, wind tunnels, or gas turbines are selected with the aid of an automated local stability synopsis. COPS then tracks besides N factors - the transition process downstream, predicting the changes in the skin friction and heat transfer. A Graphical User Interface provides ease of use, and post-processing capabilities help interpret and visualize the results.

IRIX version compatibility:

DIGITAL PHYSICS AERO™: Solid Solutions to Fluid Problems

Wallace Smith
President & CEO
Exa Corporation
450 Bedford Street
Lexington, MA 02173
USA
617-499-7200
617-499-7299 (fax)
info@exa.com
http://www.exa.com/

Exa Corporation has developed a breakthrough, Digital Physics, for modeling fluid flow that is significantly faster, fundamentally more accurate, and easier-to-use than existing methods (CFD). It will revolutionize product design in many industries such as Automotive, Aerospace, Chemical, and Energy where understanding the fluids environment is critical.

Exa's first product, EXA RESOLUTE (Version 1.0), consists of Digital Physics software, and a single Fluid Accelerator chip embodied on a plug-in board for Silicon Graphics, Inc. workstations with seamless interface to Parametric Technology's Mechanical CAD program, Pro/ENGINEER® and SGI's visualization software, Explorer™.

IRIX version compatibility: 5.3, 6.x

Doublet/Source Aerodynamics (DSA)

Michael Shook
VP, Engineering
AeroHydro, Inc.
PO Box 684
54 Herrick Road
SW Harbor, ME 04679
USA
207-244-7347
207-244-7249 (fax)
mshook@aerohydro.com
http://www.aerohydro.com

Doublet/Source Aerodynamics (DSA) is a highly general and flexible potential flow code for aerodynamic analysis of arbitrary configurations. It uses combined source and doublet panels covering the surface of a body to solve for potential flow quantities such as lift, drag, moments, pressure distribution, and streamlines. It includes the unique feature of far field analysis for accurate calculation of induced drag.

IRIX version compatibility:

ESTET

J. Chantrel
Business Dev. Engineer
SIMULOG
1, Rue James Joule
Guyancourt, CEDE78286
78286
France
33-1-30-12-27-00
33-1-30-12-27-27 (fax)
chantrel@simulog.fr

ESTET is a 3-D finite difference code for subsonic, viscous, steady, or transient flows in complex geometries. This code is developed by EDF/Laboratoire National d'Hydraulique. Phenomena that can be modeled include the effects of turbulence, gravity terms, heat transfer, a nonisothermal fluid model, variation of fluid density (high temperature gas flows), friction at the walls. Congugate heat triangles between fluids and walls, distributed head losses/porous media, combustion and radiative heat transfer, Lagrangian tracking model, turbulent dilute two-phase flow. The implemented finite difference method solves the Navier-Stokes equations coupled with a k-epsilon turbulence model. Structured meshes are built using Cartesian, cylindrical, or orthogonal curvilinear coordinate systems. The time differencing is based on a fractional time step, which allows adapting solvers to different equation operators.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

EURANUS/TURBO

Marc Tombroff
Product & Sales Engineer
Numeca International
R. de la Concorde 22
1050 Brussels,
Belgium
32-2-629-2378
32-2-629-2880 (fax)
num@stro10.vub.ac.be

EURANUS/TURBO is a powerful numerical simulator system, tuned to the simulation of 3D Reynolds-averaged Navier-Stokes flows for turbomachiniry applications. EURANUS is a general Navier-Stokes software system for very low speeds to hypersonic flows, combining control volume and multigrid methods, turbulence model, external or internal flow, steady or non-steady flow, rotating flows and chemical reactions. EURANUS/TURBO is available as a stand alone solver or, incorporated with IGG & CFView, in one easy-to-use in any environment.

IRIX version compatibility:

FIRE®

Hans-Peter Blahowsky
AVL List GmbH
Kleiststrade 48
Graz, 8020
Austria
011-43-316-987-618
011-43-316-987-777 (fax)
hpb@avl.co.at

FIRE® is a fully interactive fluid dynamics software package, including powerful, menu-driven pre- and post-processing, developed specially for computing compressible and incompressible turbulent fluid flow as encountered in engineering environments. 3-D or 2-D unsteady or steady simulations of flow and heat transfer within arbitrary complex geometries with moving or fixed boundaries can be performed. FIRE offers several additional modules like:

- Lagrangian Two Phase Flows
- Wall Film Model
- Conjugate Heat Transfer
- Full Reynolds Stress Closure
- Turbulence Controlled Combustion
- PDF Combustion
- Self Ignition
- Linear Acoustics Analogies.

IRIX version compatibility:

FLOVENT®

Raghav S. Gorur
Mgr, Customer Services
Flomerics, Inc.
2 Mount Royal Avenue
Suite 350
Marlborough, MA 01752
USA
508-366-9522
508-898-2582 (fax)

FLOVENT® is an engineering analysis software that uses the techniques of Computational Fluid Dynamics (CFD) to predict air flow and heat transfer in buildings. FLOVENT is unique because it has been conceived, specified and developed in collaboration with ESRIA (the Building Services Research and Information Association) with the practical needs of those in the field of accurate and inexpensive means of predicting the indoor thermal of heating, ventilation and air-conditioning (HVAC) equipment, as well as the location of smoke detectors, doors, windows, internal partitions, furniture and other obstructions to be optimized for a particular building and its occupants.

IRIX version compatibility:

FLOVIS

Paulo Leoncini
Head of Visualization Gro
CIRA - Centro Italiano
Recerche Aerospaziali
Via Maiorise
Capua, CE 81043
Italy
011-39-823-623134
011-39-823-623126 (fax)
lepa069@cira.it

FLOVIS is a visualizer for data on 3-D multiblock structured grids. Data representations include contour maps, iso-curvers, iso-surfaces, vector plots, cutting planes and X-Y line charts. Streamlines and streaklines can be powerfully traced with a separate module. Mesh geometry can be rendered in wireframe, hidden lines or lighting. Visualization techniques include real-time animation for parameter-dependent data, thresholding, color map manipulation, grid index sweeping, translucency and stereo viewing.

The input data file is the NLR Visdat, containing blocks of structured grid in a discipline independent format, with a per part grouping. Direct imput from other user data file formats can be arranged upon request.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

FLOW-3D®

Ron Harper
Senior Programmer
Flow Science, Inc.
1257 40th Street
Los Alamos, NM 87544
USA
505-662-2636
505-662-6564 (fax)
cfd@flow3d.com
http://www.flow3d.com/

FLOW-3D® analyzes fluid dynamics and heat transfer phenomena. Flows in three space dimensions and time are modeled in complex geometric regions and in regions bounded by free fluid surfaces. FLOW-3D consists of three separate programs written in standard FORTRAN 77 for easy portability. A preprocessor program simplifies problem setup activities for scientists and engineers by translating simple input data into a full numerical problem description for the main processor. The main processor contains the algorithms necessary for generating solutions under a wide range of physical conditions, and it also possesses some artificial intelligence in the form of control logic for time-step size selection to improve accuracy and eliminate numerical instabilities. Finally a postprocessor allows the user to easily extract and display the maximum amount of information from a calculation.

IRIX version compatibility:

GASP

Alan Davis
Asst Oceanographic Computing
Florida State University
Room 8E
Love Building
Tallahassee, FL 32306-3041
904-644-3798
904-644-4841 (fax)

GASP is a package for the visualization of time- dependent 3-D datasets composed of scalar and / or vector fields or discrete data points. GASP is useful for visualizing data from numerical analysis of fluid fields or satellite data. It is used by many researchers in oceanography, meteorology, computational fluid dynamics, groundwater pollution analysis, etc. The package is composed of two parts-the metacode production routines, which run on any computer from a Cray to a Personal IRIS, and the visualization part, which runs on any IRIS 4D or 3000 Series machine. GASP is extremely useful for visualizing very large, time-dependent datasets.

IRIX version compatibility:

GRIDGEN

Karen Newcomb
Information Analyst
COSMIC
382 East Broad Street
Athens, GA 30602
USA
706-542-3265
706-542-4807 (fax)
service@cosmic.uga.edu
http://www.cosmic.uga.edu/pub/SGI.html

GRIDGEN, Multiple Block Grid Generation Software, is a software system for the generation of 3D, multiple block, structured grids. GRIDGEN is a visually-oriented, graphics-based interactive code used to decompose a 3D domain into blocks, distribute grid points on curves, initialize and refine grid points on surfaces and initialize volume grid points. Wirtten in C-language and FORTRAN 77 for SGI IRIS 4D workstations. License agreement required. PROGRAM NUMBER: ARC-13371.

IRIX version compatibility:

GridPro®/az3000

Peter R. Eiseman
President
Program Development Corporation of Scarsdale, Inc.
300 Hamilton Avenue
Suite 409
White Plains, NY 10601
USA
914-761-1732
914-761-1735 (fax)

The multiblock grid generator with automatic zoning provides a fully automatic means to generate hexahedral grids (brick mesh elements) about a wide variety of complex geometric configurations. This represents a major technology breakthrough and provides a capability that is unavailable elsewhere. Unlike the prior grid generators, GridPro™/az3000 is both automatic and well structured. Moreover, the grid quality is maintained even with intense clustering. Because of the good structure, fluid dynamic solution procedures are more available and are more efficient. Because of the quality, they are more reliable, produce more accurate results, and generally operate faster. In addition, the automation is well set up for design variations since reuseable components are available or recurrent use and for parametric style actions (as in modern CAD).

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

GridPro®/sb3020

Peter R. Eiseman
President
Program Development Corporation of Scarsdale, Inc.
300 Hamilton Avenue
Suite 409
White Plains, NY 10601
USA
914-761-1732
914-761-1735 (fax)

Grid modeling is new and the product is GridPro™/sb3020. Grid modeling can be viewed like an extension of computer aided geometric design (CAGD). In distinction, pointwise distributions are important, the action is volumetric, and there is the concurrent integration of both control point arrays and specified boundaries. In addition, there is a rich supply of tools (eg. rubber banding, elliptic methods, etc.) to model by moving control points to new positions. Modeling can be performed with fixed boundary points, points moved along the boundary, a totally free-formable boundary, or some combination of these.

IRIX version compatibility:

I/PUNCH (Intergraph/Punch Option)

Kim Corbridge
Product Mktg Manager
Intergraph Corporation
MS GD 3000
Huntsville, AL 35894-0001
USA
205-730-3701
205-730-3453 (fax)
****
http://www.intergraph.com

I/PUNCH is an application enhancement package to the I/NC system. This optional package provides capabilities for two-axis turret punching centers, and includes the following features:

- area and perimeter nibbling and punching
- automatic tool selection (with manual override)
- machine "canned" cycles
- standard punches supported
- automatic indexed tool stations
- automatic clamp avoidance
- toolpath optimization
- sheet repositioning
- optimized gridding of multiple parts.

IRIX version compatibility:

ICEM CFD

Wayne Christopher
Chief Financial Officer
ICEM CFD Engineering
2600 Etna Street
Berkeley, CA 94704
USA
510-549-1890
510-841-8523 (fax)
info@icemcfd.com
http://icemcfd.com

ICEM CFD is the leading software for mesh generation for computational fluid dynamics and finite element applications, in the aerospace, automotive, nuclear, and semiconductor industries. High-quality multiblock structured and unstructured (tetrahedral, hexahedral, and mixed) meshes can be accurately and rapidly generated. ICEM CFD is based on the popular CAD package ICEM DDN, and provides full access to its advanced design capabilities. It also includes interfaces to IGES, DXF, and other interchange formats, can be interfaces to a wide variety of flow solvers. ICEM CFD is available on most UNIX workstations. The current version is 3.1.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

ICEM CFD Hexa

Wayne Christopher
Chief Financial Officer
ICEM CFD Engineering
2600 Etna Street
Berkeley, CA 94704
USA
510-549-1890
510-841-8523 (fax)
info@icemcfd.com
http://icemcfd.com

ICEM CFD Hexa is a new object based semi-automatic hexahedral meshing module of ICEM CFD, the leading mesh generation tool for fluid dynamics and finite element applications. It provides rapid generation of multiblock structured and unstructured volume frequent user. ICEM CFD Hexa utilizes advanced projection algorithms which automatically generate high quality grids, on CAD surfaces, regardless of the surface quality. Once the grid is set up the user can select a solver from over 2 dozen options. ICEM CFD Hexa will then generate an input file contianing all the necessary information for the solver.

IRIX version compatibility:

IGG

Marc Tombroff
Product & Sales Engineer
Numeca International
R. de la Concorde 22
1050 Brussels,
Belgium
32-2-629-2378
32-2-629-2880 (fax)
num@stro10.vub.ac.be

IGG 3.3 is a highly interactive geometry modeler and grid generator systems tuned to the generation of complex 2-D/3-D geometries as well as surfaces and volume grids for the solution of flow problems. Most generations constitute the most consuming and costly part of the numerical simulation process. IGG 3.3 provides original tools to create structured multiblock 2-D/3-D meshes as well as 2-D unstructured grids based on triangles. IGG 3.3 is a stand alone software based on the most advanced software tools and standards, such as object-oriented languages and PHIGS, integrated with X-Windows.

Prerequisite Hardware: 32MB main memory, 24 bit planes +Z buffer.

IRIX version compatibility:

INCA

Don Roberts
Vice President
Amtec Engineering, Inc.
3055 112th Avenue, NE
Suite 100
Bellevue, WA 98004
USA
206-827-3304
206-827-3989 (fax)
don@moclips.amtec.com
http://www.Amtec.com

INCA is a state-of-the-art flow code. It is valid for a wide range of compressible viscous flow fields, including transonic aircraft, hypersonic high altitude vehicles, automotive intakes/manifolds, and reacting combustors. The incompressible flow module extends the range of applicability to low speed flows.

INCA solves the complete compressible or incompressible Navier-Stokes equations using an advanced implicit finite-volume procedure on multi-block, body-fitted, computational grids. The flow domain can be divided into coupled grid blocks to simplify both the generation of the grid and the application of boundary conditions. A variety of flexible boundary-condition types, equation-of-state options, turbulence models and chemistry models makes INCA applicable to most flow fields of interest. A parallel-procesing version of INCA is also available.

IRIX version compatibility:

INTERN™

Melvin Platt
Manager, CAE Group
Northern Research &
Engineering Corporation
39 Olympia Avenue
Woburn, MA 01801-2073
USA
617-937-4646
617-935-9052 (fax)

INTERN™ is a specialized finite-difference analysis system for gas-turbine combustors. The code calculates the 3-D, viscous, recirculating internal flow, both with and without combustion. It allows annular, can and can-annular geometries, and will accomodate swirlers, primary and dilution-zone jets of any orientation, and cooling films from wall slots. Gaseous, as well as liquid, fuels may be specified. A graphics processor previews the grid, and provides details of the calculated flow field through both velocity vector plots and contour plots that include temperature, pressure, turbulence intensity, and mass fraction of unburned fuel and carbon monoxide.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

LISA

Thorwald Herbert
President
Dynaflow, Inc.
3040 Riverside Drive
Columbus, OH 43221
USA
614-487-9939
614-487-1059 (fax)
tht@dynaflow.com

LISA (Linear Stability Analysis) performs an extensive set of tasks to analyze the stability of flows. A graphical user interface allows selection and interactive analysis of numerous built-in applications that reach from traditional stability problems to the stability of non-parallel or compressible boundary layers and secondary instability of parallel flows. Eigenvalue spectra, single eigenvalues and eigenfunctions, multi-dimensional tables, or neutral curves can be generated with numerical, graphical, or PostScript output. Besides the built-in applications, LISA provides a documented interface to the numerical (spectral) method and can be customized for other stability problems in fluid dynamics or elsewhere.

IRIX version compatibility:

Liquid Propellant Program (LPP)©

Stuart Dunn
Senior Research Engineer
Software & Engineering Associates, Inc.
333 S. Carson Meadows Dr.
Suite 44
Carson City, NV 89701
USA
702-882-1966
702-882-1827 (fax)
stu@seainc.com

The Liquid Propellant Program (LPP)© calculates the flow field and performance for liquid rocket engines using equilibrium, frozen, or finite-rate chemistry. It can accommodate axisymmetric or 2-D planar flow. It includes a SCRAM jet option, with arbitrary upper and lower wall definition. The solution includes an inviscid core flow module and two boundary layer modules. The boundary layer module can accommodate multiple supersonic slot injection for both upper and lower walls. The program can evaluate axi-symmetric plug nozzles with free-stream interaction.

The program includes an interactive front end (IFE) and interactive postprocessor graphic package. An export license is required outside of the U.S.

IRIX version compatibility:

M3F3CA/M3HAX

Daniel Lesieutre
Senior Research Engineer
Nielsen Engineering & Research, Inc.
526 Clyde Avenue
Mountain View, CA 94043-2212
USA
415-968-9457
415-968-1410 (fax)
00000

M3F3CA: Fast-running, engineering-level, missile aerodynamic predictions for analysis and preliminary design. Axisymmetric bodies with up to three cruciform sections with fully deflectable fins. Based on high resolution, high angle-of-attack experimental data bases and advanced equivalent angel-of-attack concept. Includes nonlinear effects of body and fin vortices. Subsonic, transonic, supersonic Mach numbers; roll angle arbitrary. Estimates axial force and pitch plane dynamic derivative.

M3HAX: M3F3CA extended to 90 degree angle-of-attack, options for including effects of rotational rates and nonuniform flow.

These code efficiently predict accurate longitudinal and lateral aerodynamic characteristics and are essential for generating aerodynamic databases for flight simulations.

IRIX version compatibility:

MACHMAKER

Laurence Feldman
President
Creative Visual Software, Inc.
P.O. Box 0329
Evanston, IL 60204
USA
408-252-0458
310-212-5046 (fax)

MACHMAKER is a 2-D finite difference compressible viscous flow solver providing transient field solutions over arbitrarily complex 2-D shapes. The user graphically inserts/edits the 2-D shape and the flowfield is computed and displayed in seconds. Arbitrary boundary and free-stream conditions from subsonic to hypersonic flow produce solutions containing non-linear effects such as shocks. Bodies may be oriented with variable pitch. Release 2.0 contains a full set of postprocessing visual analysis tools and employs a MOTIF user-interface. Solution accuracy is a function of grid resolution which is arbitrarily set.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

MGAERO

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

MGAERO is a 3-D Euler method for computing flow about arbitrary configurations in the subsonic through supersonic flight regimes. The technique uses a multigrid approach with an equally spaced Cartesian mesh structure. Swept, unswept, or rotated grids are used to develop local computational domains, which can be overlapped with either the surface or other multigrid levels. The ease of geometry and grid input allows for rapid turnaround of results on complete aircraft including nacelles. Coupling of an integral boundary layer method on surface streamlines allows prediction of viscous effects.

IRIX version compatibility:

MISES©

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

MISES© is a coupled viscuous/inviscid method for cascade airfoil analysis and design. It consists of menu driven routines which perform functions such as:

- Configuration geometric modification.
- Gridding of the flowfield.
- Subsonic, transonic and supersonic cascade airfoil.
- Computational design of cascade airfoils.
- Multi-point optimization of cascade airfoils with geometric and aerodynamic constraints.

MISES is based upon a streamline Euler discretization and a two-equation integral boundary layer formation coupled through the displacement thickness and solved simultaneously by a full Newton method.

IRIX version compatibility:

Material Ablation Conduction Erosion (MACE)©

Stuart Dunn
Senior Research Engineer
Software & Engineering
Associates, Inc.
333 S. Carson Meadows Dr.
Suite 44
Carson City, NV 89701
USA
702-882-1966
702-882-1827 (fax)
stu@seainc.com

The Material Ablation Conduction Erosion (MACE)© program solves the 1-D heat conduction equation, including mechanisms that control internal decomposition. The surface boundary can be determined by simple conduction, constant temperature sublimation, a Munson-Spindler-type relationship, a C-3 carbon-oxygen reaction, or a generalized chemistry boundary condition. In addition to the above boundary conditions, the surface material can be removed by mechanical erosion.

Convective heat transfers and boundary layer properties can be input directly or input through a files link with an aeroheating program. Heating rates can be modified by angle-of-attack, surface roughness, non-isothermal wall, or protuberance heating.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

N3S

J. Chantrel
Business Dev. Engineer
SIMULOG
1, Rue James Joule
Guyancourt, CEDE78286
78286
France
33-1-30-12-27-00
33-1-30-12-27-27 (fax)
chantrel@simulog.fr

N3S is a fluid mechanics software package based on the Finite Element method. It can handle 2-D, asymmetrical and 3-D arbitrary complex geometries with performance levels (CPU and memory load) equivalent to those of other industrial software programs based on structured grids. N3S is developed by the Research and Development branch (DER) of Electricite de France (EDF). The program is currently able to compute non-isothermal incompressible and compressible turbulent flows.

Space discretization in triangles and tetrahedral is well adapted to advanced capabilities available in N3S such as moving boundary flows, adaptive meshes, refinement of meshes, configurate heat transfer between fluids and walls, solid-fluid interactions, CAD interface.

Fields of application: confined flows, turbomachinery, thermal hydraulics, aerodynamics.

IRIX version compatibility:

NEARZEUS/ZEUSBL

Stanley Perkins
Senior Research Engineer
Nielsen Engineering &
Research, Inc.
526 Clyde Avenue
Mountain View, CA 94043-
2212
USA
415-968-9457
415-968-1410 (fax)
scp@nearinc.com

NEARZEUS CFD: Code for detailed missile aerodynamics, valid for supersonic/hypersonic flow. Predicts nonlinear details of flow surrounding the missile. Finite-volume Godunov scheme, second order accurate. Zonal Euler Solver, supersonic space marching, real gas option. External or internal flows, multiple zone semi-automatic gridding.

ZEUSBL: Boundary layer module for NEARZEUS, aerodynamic heating.

These codes run fast on workstations and provide accurate dynamic characteristics and skin friction/heat transfer estimates for high Mach number configurations. Reduces the requirements for high speed wind tunnel tests in designing new concepts.

IRIX version compatibility:

NEKTON™

Steve Rozov
Marketing Manager
Fluent, Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766
USA
603-643-2600
603-643-3967 (fax)
sr@fluent.com
http://www.fluent.com/

NEKTON™ is the code of choice for the simulation of coating, extrusion and viscous materials processing. NEKTON uses the spectral element method, a variable-order solution approach that outperforms all other CFD methods for the solution of laminar and non-Newtonian flows. With the geometric flexibility and convergence acceleration provided by large, unstructured spectral elements, NEKTON offers significant benefits over traditional finite element methods and provides accurate free surface modeling, a deforming mesh technique and high-order time stepping.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

NEWPAN

Chris Burkett
Senior Engineer
Flow Solutions Ltd.
7 Whinnetts Way
Pulloxhill, BEDFMK45
5EX MK45 5EX
UK
01525-718118
chris@flowsol.cyberphile.co.uk

NEWPAN is an advanced three-dimensional panel method for the computation of external aerodynamic and hydrodynamic problems in the aerospace, automotive, and marine industries. It is used extensively by championship winning Formula One teams and leading IndyCar motor racing teams. It is capable of modelling complex configurations in subsonic flow, if necessary including the effects of wind tunnel walls and mounting structures. It features a design capability, enabling, for instance, a wing to be designed to give a prescribed pressure distribution and returning the shape necessary to achieve it. NEWPAN has also been used for the hull, keel and rudder design of America's Cup yachts. Design consultancy is also available.

IRIX version compatibility:

NEWTUN

Laurence Feldman
President
Creative Visual Software, Inc.
P.O. Box 0329
Evanston, IL 60204
USA
408-252-0458
310-212-5046 (fax)

NEWTUN is a 3-D finite difference compressible viscous flow solver providing transient field solutions over arbitrarily 3-D shapes. REPLICORE, AutoCAD or other geometric modeler produces a 3-D body which is the inserted by NEWTUN into a uniform 3-D grid and solutions are provided shortly thereafter. A full complement of 3-D rendering and visual analysis tools are included. NEWTUN is unique in it ability to solve for flow over complex 3-D bodies at integrations rates orders of magnitude faster than comparable techniques. Release 2.0 contains a Motif™ preprocessor and produces PLOT-3D files for further postprocessing. A multitasked, vectorized version exists for supercomputers.

IRIX version compatibility:

NSBES

Robert Cavalleri
President
Applied Technology Associates, Inc.
P.O. Box 149434
Orlando, FL 32814
USA
407-894-6151
407-894-6577 (fax)

A unified flow and thermal analysis software package has been developed. The software package employs both a Computational Fluid Dynamics (CFD) code with a two phase flow capability and a Boundary Element Method (BEM) thermal analysis. The CFD code will determine the three dimensional flow field, typical properties of the flow that will be determined with the CFD code are velocity, pressure, temperature and particle distribution. This will allow predicting heat transfer rates which will be used to compute internal temperature distribution using a transient two dimensional BEM computer code. This design tool has application in a broad range of technology areas.

IRIX version compatibility:

OMNI3D

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

OMNI3D is a color graphics visualization program that displays aerodynamic simulation results in 3-D and arbitrary 2-D slices. OMNI3D displays surface pressure, velocity vectors, and Mach number contours on the body and in the flow field, streamline, and boundary layer parameters along on- and off-body streamline trajectories; it also displays wake geometry data. Time steps and iterations can be animated or stepped through. The interactive interface of OMNI3D with on-line help, pop-up menus, and the mouse or dial box, and its real-time manipulation of data in 3-D enable engineers to rapidly assess results on complex configurations.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

PAM-FLOW™

Lionel Bouet-Willaumez
Int'l Mktg &
Communications
ESI Group
20 rue Saarinen
Silic 303
Rungis CEDEX, 94588
France
33-1-49-78-28-00
33-1-46-87-72-02 (fax)
chaillou@cts.com

PAM-FLOW™ is an advanced FEA computational fluid dynamics (CFD) software using an automatic unstructured and adaptive meshing with its own automatic mesh generator and the possibility to handle compressible or incompressible 2D or 3D flows and coupled fluid/structure problems. Popular applications can be found in thermal transfers and climate control, blast waves simulations or lightweight structures testing, through coupling with PAM-CRASH/SHOCK software.

IRIX version compatibility:

PAM-FLUID™

Lionel Bouet-Willaumez
Int'l Mktg &
Communications
ESI Group
20 rue Saarinen
Silic 303
Rungis CEDEX, 94588
France
33-1-49-78-28-00
33-1-46-87-72-02 (fax)
chaillou@cts.com

PAM-FLUID™ is a proven, effective solution for the Predictive Virtual Testing of turbomachines, engines or external aerodynamics for the transportation, automotive or aeronautical industries.

IRIX version compatibility:

PASSAGE™

Oguzhan Gurdogan
Director, Software R & D
Technalysis, Inc.
7120 Waldemar Dr.
Indianapolis, IN 46268
USA
317-297-6083
317-291-7281 (fax)

PASSAGE™ is a collection of finite element programs for flow and heat transfer analysis in complex 3-D geometries. PASSAGE supports the following standalone modules:

- PASSAGE/DUCTFlows through complex passages
- PASSAGE/WHEELFlows through rotating/stationary blade passages
- PASSAGE/PROCESSFluid flow and heat transfer in manufacturing processes
- PASSAGE/CASTINGCasting processes
- PASSAGE/SYSFLOWSimulation of flow networks

All modules are supported by preprocessors for geometry, mesh, and flow/process conditions definition, and postprocessors for color/monochrome results displayed as x-y, vector, and contour plots. Application areas are widespread in the automotive, fan/HVAC, appliance, aerospace, pharmaceutical/chemical, and equipment industries.

IRIX version compatibility:

PLASTEC®

Oguzhan Gurdogan
Director, Software R & D
Technalysis, Inc.
7120 Waldemar Dr.
Indianapolis, IN 46268
USA
317-297-6083
317-291-7281 (fax)

PLASTEC® is a collection of finite element programs for mold flow and heat transfer analysis in complex, 3-D, fiber-reinforced plastic parts. PLASTEC supports the following standalone modules:

- PLASTEC/INJECTIONInjection molding

- PLASTEC/COMPRESSIONCompression molding

Thermoplastic materials, Newtonian and non-Newtonian fluids, and isothermal and non-isothermal processes can be modeled. Fiber orientations can be predicted. Orthotropic mechanical properties are output for subsequent warpage/stress analysis. Both modules are supported by preprocessors for geometry, mesh, and material/process conditions definition, and by postprocessors for color/monochrome results display as x-y, vector, and contour plots.

IRIX version compatibility:

Polyflow

Christophe Waucquez
Research Assistant
Polyflow S.A.
16, Place de l'Universite
Louvain-la-Neuve, B-1348
Belgium
32-10-45-28-61
32-10-453-009 (fax)
support@polyflow.be

Polyflow is a unique, finite element-based, general purpose Computational Fluid Dynamics program which simulates the flow of Newtonian and non-Newtonian fluids, with a special emphasis on rheologically complex liquids in industrial processes.

Polyflow is a fully interactive package and its ease of use makes it the ideal tool for engineers and scientists to study 2-D and 3-D flow and heat transfer problems, steady or time-dependent, with fixed or moving boundaries.

Polyflow is used to understand, modify and improve a variety of industrial processes, among which:

* Extrusion and coextrusion

* Die and screw design in extrusion

* Coating

* Blow molding

* Fiber spinning

* Film casting and film blowing

* Laminar mixing

* Heat transfer

* Chemical reactions

* Chemical Vapor Deposition

* Glass furnace simulations

It is being used to model processes for fluids such as polymer solutions, polymer melts, rubber, oils, detergents, printing inks or paints, muds, liquid food stuff and molten glass, among others. Polyflow contains unique features for the calculation of the important and frequently encountered viscoelastic effects in industrial flows, for the prediction of free surfaces and interfaces and the prediction of the die shape to obtain an extrudate of prescribed shape.

More than 150 major industrial sites and research groups use Polyflow worldwide to benefit from the savings of numerical simulation.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

RAMPANT™ V2.0

Steve Rozov
Marketing Manager
Fluent, Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766
USA
603-643-2600
603-643-3967 (fax)
sr@fluent.com
http://www.fluent.com/

RAMPANT™ is the code of choice for the solution of high speed and compressible flows. Utilizing a fully-coupled algorithm that is the recognized method for these applications, RAMPANT contains physical models that address turbulent flows, heat transfer, compressible mixing, rotating flows and flows with strong shocks. Using unstructured meshes, RAMPANT gives you all the benefits of fast model creation and local mesh adaption for capturing shocks and other local flow details.

IRIX version compatibility:

RIP

Karen Newcomb
Information Analyst
COSMIC
382 East Broad Street
Athens, GA 30602
USA
706-542-3265
706-542-4807 (fax)
service@cosmic.uga.edu
http://www.cosmic.uga.edu/pub/SGI.html

RIP, Remote Interactive Particle-tracing, is a distributed graphics program that computes particle traces for computational fluid dynamics (CFD) solution datasets. A particle trace is a line that shows the path that a massless particle in a fluid takes; it is a visual image of where the fluid is going. The program can compute and display particle traces at a speed of about one trace per second because it runs on two machines concurrently. Implemented on the SGI IRIS 3000 and a CRAY 2 running Unico 1.0 or 2.0. Last updated in 1988.

IRIX version compatibility:

SAGE - Multidimensional Self-Adaptive Grid Code

Karen Newcomb
Information Analyst
COSMIC
382 East Broad Street
Athens, GA 30602
USA
706-542-3265
706-542-4807 (fax)
service@cosmic.uga.edu
http://www.cosmic.uga.edu/pub/SGI.html

SAGE - Multidimensional Self-Adaptive Grid Code, provides an efficient method to improve the accuracy of CFD flow solutions while simultaneously reducing computer processing time. SAGE enhances an inital computational grid by redistributing the mesh points into more appropriate locations. The movement of these points is driven by an equal-error-distribution algorithm that utilizes the relationship between high flow gradient and excessive solution errors. The method also provides a balance between clustering points in the high gradient regions and maintaining the smoothness and continuity of the adapted grid. Written in FORTRAN 77 to be machine independent. Requires a compiler which supports NAMELIST. Successfully implemented under IRIX 5.2.

IRIX version compatibility:

SPIN

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

SPIN is a preprocessor for VSAERO which includes a point-and-click basic data generator and creation of wakes in 3D or arbitrary 2D cuts. It produces a syntactically correct VSAERO input file with descriptive comments. Geometry manipulation and grid generation are currently under development.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

SUBSTR/HASLSB/SUPSTR

Marnix Dillenius
President
Nielsen Engineering & Research, Inc.
526 Clyde Avenue
Mountain View, CA 94043-2212
USA
415-968-9457
415-968-1410 (fax)
mfed@nearinc.com

SUBSTR: 6-DOF store separation analysis for subsonic speeds. Aircraft components include fuselage, wings and up to three pylons, fixed stores. Stores with up to two finned sections, vortical wing-tail interference, specified ejection forces, specified thrust/time history, rail launch. Based on panel methods and slender body theory/reverse flow theorems.

- HASLSB SUBSTR extended to handle rail launch from pitching aircraft at high angle of attack.
- SUPSTR 6-DOF store separation analysis for supersonic speeds. Aircraft components include fuselage, wings and up to three pylons, fixed stores. Stores with up to two finned sections, vortical wing-tail interference, specified ejection forces, specified thrust/time history. Based on panel methods with simulation of nonlinear shock effects in flow field.

These codes run fast on workstations, reduce the need to conduct costly wind tunnel tests, and are indispensible in prelimanary safe store launch assessments and parent aircraft/store integration studies.

IRIX version compatibility:

SUPDL/SUBDL

Daniel Lesieutre
Senior Research Engineer
Nielsen Engineering & Research, Inc.
526 Clyde Avenue
Mountain View, CA 94043-2212
USA
415-968-9457
415-968-1410 (fax)
00000

SUPDL: Intermediate level code for predicting aerodynamic force distributions on supersonic missiles, valid in low supersonic Mach range. Post-processor generates force input data for NASTRAN structural analysis. Based on supersonic panel methods including nonlinear effects of body and fin vortices, fin loads, include stall model. Axisymmetric bodies with up to two finned sections in planar, triform, cruciform, low-profile layouts. Includes effects of rotational rates and nonuniform flow conditions.

SUBDL: Same as SUPDL but for subsonic Mach numbers.

Besides predicting longitudinal and lateral aerodynamic characteristics, the SUPDL codes are practical aerodynamic tools for use in design/optimization of lifting surfaces.

IRIX version compatibility:

Solid Performance Program (SPP)©

Stuart Dunn
Senior Research Engineer
Software & Engineering Associates, Inc.
333 S. Carson Meadows Dr.
Suite 44
Carson City, NV 89701
USA
702-882-1966
702-882-1827 (fax)
stu@seainc.com

The Solid Performance Program (SPP)© predicts rocket motor performance by calculating deviations from ideal performance by using a series of independent efficiency models. SPP analyzes nozzle and motor performance. The nozzle analysis includes models for describing the 2-D transonic and supersonic steady-state flow field within a given nozzle contour. 1-D models are used to analyze the nozzle wall boundary layer and finite-rate gas phase chemistry.

The motor performance analysis consists of grain design modules and an internal ballistics module. This program is restricted to U.S. companies only and requires DoD (AL/AFSC) approval.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

TASCflow™ for CAD

Glenn Smith
Advanced Scientific Computing, Ltd.
554 Parkside Drive
Waterloo, ON N2L 5Z4
Canada
519-886-8435
519-886-7580 (fax)
info@asc.on.ca

TASCflow™ for CAD (TfC) is general CFD software package functioning within your CAD/E environment. TfC was developed to deliver accurate and robust fluid flow and heat transfer analysis in complex geometries to a broad engineering segment by being easy to use and reducing set-up time. This is accomplished by providing five enabling software technologies:

- a functional and intuitive GUI with a consistent look and feel
- interfaces to your existing CAD/E software including I-DEAS, MSC/ARIES. Pro Engineer...in addition to FE file formats found in PATRAN, NASTRAN, ANSYS...
- support for any quality of unstructured meshes using any combination of hexahedral, tetrahedral, wedge and pyramidal elements.
- proven numerical methods including an unstructured coupled linear solver
- "on-line" documentation, accessed via NCSA's Mosaic, includes all help related functions and a complete "start to finish" CFD tutorial.

IRIX version compatibility:

TMRC

Robert Childs
Executive Vice President
Nielsen Engineering & Research, Inc.
526 Clyde Avenue
Mountain View, CA 94043-2212
USA
415-968-9457
415-968-1410 (fax)
childs@nearinc.com

Capabilities of the TMRC are:

- 3-D compressible Navier-Stokes flow solver
- High order differencing, finite volume algorithm
- Algerbraic or k-r turbulance modeling
- Cartesian or cylindrical grids
- Direct numerical simulations, large eddy simulations, or Reynolds-average calculations
- Aerocoustics and aero-optics applications
- Analysis tool for assessing optical distortion, accurate model of recirculating jets
- CFD educational tool

IRIX version compatibility:

USAERO

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

USAERO calculates the transient aerodynamic characteristics of complex single or multi-component configurations in arbitrary motion. The method is based on a time-stepping procedure that allows relative motions with close encounters. The nucleus of the code has been structured to take advantage of vectorizing compilers, thereby keeping computer time to a minimum. The program has reached a level of maturity for application to maneuvering aircraft, helicopter rotor/body interactions, and also gust response with three degrees of freedom. Other applications include rotor/stator/shroud, truck/automobile, train/train, train/station dynamic ground effect, oscillating surfaces (for example, spoilers), aeroelastic coupling, and yacht hull manuever.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

VECTIS

Anthony Smith
Software Sales Manager
Ricardo Consulting
Engineers Ltd.
Bridge Works
Shoreham-by-Sea
Shoreham-by-Sea, W BN43
5FG
UK
44-1273-455611
44-1273-464124 (fax)

VECTIS is an advanced CFD package, allowing the rapid analysis of fluid flows involving complex thermal and chemical interactions. The program incorporates fully automatic mesh generation irrespective of geometrical complexity, based upon geometry input from CAD systems, FE models and other unambiguous computer definitions. VECTIS also features a number of specific facilities for addressing flow problems within engines, such as, the optimization of combustion chambers, ports, manifolds and catalysts. The package has been well proven in its use by Ricardo and a number of licenses in Europe, Japan and the United States.

IRIX version compatibility:

VGRID

Shahvar Pirzadeh
Research Engineer
ViGYAN, Inc.
30 Research Drive
Hampton, VA 23666
USA
(757) 864-2245
(757) 864-8469 (fax)
pivgrid@vigyan.com
http://www.vigyan.com

VGRID™ is a robust, user-friendly computer program for the generation of three-dimensional unstructrued (triangular surface and tetrahedral volume) grids in geometrically complex domains. The base code (VGRID3D™) has been developed by VIGYAN, Inc. under a NASA Small Business Innovation Research (SBIR) contract. Since completion of that project (1990), considerable extensions to VGRID have been developed at NASA Langley Research Center. These extensions are an outcome of several years of research, development, extensive application, and user-feedback improvements. VGRID is a key componenet to the NASA unstructured-grid software system TetrUSS, a recipient of the 1996 NASA Software of the Year award. Along with a graphical user interface utility GridTool and a grid post-processing program POSTGRID, the VGRID system provides a reliable tool for the fast and convenient generation of unstructured grids around complex configurations. The code, in conjunction with the unstructured-grid solver USM3D and post-processing visualization program ViGPLOT has been widely used by many researchers and engineers from academia, industry, NASA, and other government agencies for the past several years.

IRIX version compatibility:

VISIUN™

Melvin Platt
Manager, CAE Group
Northern Research &
Engineering Corporation
39 Olympia Avenue
Woburn, MA 01801-2073
USA
617-937-4646
617-935-9052 (fax)

VISIUN™ is a specialized software system for the time-dependent analysis of internal flows in turbomachinery. It employs an implicit, approximately factored finite-difference scheme to solve the 3D Euler or Navier-Stokes equations, with body-fitted grid generation, and considers flow in ducts and either stationary or rotating blade rows. The compressible flow solution is able to handle supersonic flows with shocks, as well as subsonic flows. VISIUN™ has been applied successfully to both radial and axial turbomachinery. Its processor allows the designer to quickly create a finite-difference grid, a complete set of boundary conditions, and an initial flow field. Tracking and steering minimize convergence time for a solution. The postprocessor provides important plots including blade loading diagrams. It also supports powerful flow visualizers.

IRIX version compatibility:

CFD (Computational Fluid Dynamics)

VISIUN-2FR™

Melvin Platt
Manager, CAE Group
Northern Research &
Engineering Corporation
39 Olympia Avenue
Woburn, MA 01801-2073
USA
617-937-4646
617-935-9052 (fax)

VISIUN-2FR™ is a specialized software system for the two-frame-of-reference analysis of internal flows in turbomachinery. It employs an implicit, approximately factored finite-difference scheme to solve the 3-D Euler or Navier-Stokes equations, with body-fitted grid generation, and considers flow in ducts, and both stationary and rotating blade rows. The compressible flow solution is able to handle supersonic flows with shocks, as well as subsonic flows. VISIUN-2FR™ has been applied successfully to both radial and axial turbomachinery. Its processor allows the designer to quickly create a finite-difference grid, a complete set of boundary conditions, and an initial flowfield. Tracking and steering minimize convergence time for a solution. The postprocessor provides important plots including blade loading diagrams. It also supports powerful flow visualizers.

IRIX version compatibility:

VSAERO

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

VSAERO is a three-dimensional panel method for calculating the nonlinear characteristics of arbitrary configurations in subsonic flow. Wake separation and vortex/surface interaction are included in an interative wake-shape calculation procedure. The effects of viscosity are treated in an outer iteration loop which couples the potential flow and integral boundary layer methods. Additionally, VSAERO's simulation capabilities include internal flows, hydrodynamic wake effects, rotor/fuselage interaction, and propeller/fan perfomance. Consequently, applications include fluid flow problems in the aerospace, automotive and marine industries.

IRIX version compatibility:

ViGPLOT

Brent Bates
UNIX Sys. Admin.
ViGYAN, Inc.
30 Research Drive
Hampton, VA 23666
USA
(757) 865-6350
(757) 865-8177 (fax)
blbates@vigyan.com
http://www.vigyan.com

ViGPLOT is an interactive, menu-driven graphics post-processor program for manipulation and display of data. ViGPLOT can read 2-D and 3-D data from a variety of sources including structured and unstructured CFD as well as experimental data. Its capabilities include surface plots of grid, solution contours and velocity vectors, ability to interpolate data on arbitrary user defined planes and field options of particle traces and stream ribbon plots. A built-in calculator lets a user manipulate scaler data. ViGPLOT's animation capability helps produce display quality graphics while its volume rendering algorithm facilitates detailed feature extraction from a 3-D flow field.

IRIX version compatibility: Pre 5.x

XFOIL©

Frank Dvorak
President
Analytical Methods, Inc.
2133 - 152nd Avenue N.E.
Redmond, WA 98052
USA
206-643-9090
206-746-1299 (fax)
frank@amiwest.com
http://www.am-inc.com/

XFOIL© is an interactive program for the design and analysis of two-dimensional airfoils. It consists of a collection of menu-driven routines which perform functions such as:

- Viscous (or inviscid) analysis of an existing airfoil, allowing forced or free transistion.
- Airfoil design and redesign by interactive specification of a surface speed distribution.
- Airfoil redesign by interactive specification of new geometric parameters.

XFOIL is based upon a linear vorticity panel method coupled to a lag-dissipation boundary layer scheme. The program has been used in the design of aircraft wings, fan blades, propeller blades and hydro-foils.

IRIX version compatibility: