MARTEC™

Chinook Solver 2006
Technical Specification

Chinook is blast analysis software that efficiently utilizes computational fluid dynamics (CFD) to provide physically-accurate solutions to multi-dimensional high-speed compressible flow problems.

Domain

- A number of Cartesian domain configurations are available, including:
  - One-dimensional grids representing planar, cylindrical and spherical coordinates
  - Two-dimensional grids representing planar and cylindrical coordinates
  - Three-dimensional grids in a planar coordinate system
- Structured quadrilateral (one- and two-dimensional), and hexahedral (three-dimensional) meshes may be generated automatically. Meshing tools are available which allow boundaries to be placed farther from the region of interest without having to maintain the same resolution throughout the grid.
- Unstructured meshes created using third-party mesh generators may be used in Chinook simulations. Supported cell types are:
  - One-dimensional - quadrilaterals
  - Two-dimensional - quadrilaterals or triangles
  - Three-dimensional - hexahedrons or tetrahedrons
- The Chinook Grid Converter tool, supplied with the Chinook solver, can be used to convert meshes created using third-party mesh generators into a form that can be used by Chinook. Supported mesh inputs are:
  - Meshes created in LS-DYNA keyword input format
  - Meshes created using the Easymesh generation tool
  - Meshes that have been exported to Martec's mesh format, which contains nodal coordinates and connectivities, and is described in the Chinook documentation.

Solver

- Explicit timestepping (single step, 2-step Euler, 4-stage Runge-Kutta), with up to 4th order temporal accuracy
- Fluxes are computed using an inviscid high-speed convection scheme that uses the HLLC approximate Riemann solver
- Second-order spatial accuracy, using Barth & Jesperson flux limiting scheme on unstructured meshes

Material Models

- Multiple-species gas mixtures can be simulated using an approach where mixed gas properties are calculated and used with the ideal gas equation of state.
- Four material types can be simulated:
  - Gas - Ideal gas equation of state
  - Water - Tait equation of state
  - Solid - Linear elastic equation of state
  - Explosive - JWL equation of state
- Multiple materials can reside at a mesh point at one time. Materials are combined using a mixed-cell approach where materials within a mesh point remain in pressure equilibrium and have the same velocity. This method allows states composed of multiple materials to be defined by specifying parameters for each material separately, as opposed to

defining a single equation of state and associated parameters for the entire mixture. This approach has been used to model underwater explosions (explosive in water with air at free surface), and landmine detonations (explosive in wet or dry soil).

Reactive Flow Models

- The Constant Reaction Time (CRT) burn model converts a fuel material component to products at a user-specified rate, while adding a specified heat of reaction to the system. Calculations performed using this model include simulations of the propagation of a detonation wave through a fuel cloud.

Boundary Conditions

- For meshes generated automatically by Chinook, boundary conditions can be applied to any of the domain edges. For imported meshes, boundary conditions are defined at the mesh generation stage and may be applied to select element faces and not necessarily along full domain edges.
- Available boundary conditions include:
  - Rigid reflecting surface
  - Far-field flow-through boundary
  - Entire state, whereby a specified state is applied at the boundary

Modelling and Analysis Tools

- Interim files can be written at user-defined intervals during the CFD simulation, from which Chinook can restart a terminated or cancelled calculation. This feature allows simulations to be continued when not run for sufficient time without having to re-execute the entire simulation.
- Regions of the mesh are assigned fluid states or designated as solids using on-grid shape detection. Using this approach, primitive shapes are assigned a state and positioned on the grid. Regions of the mesh within the shape bounds are set to the shape state. The on-grid shape detection method allows different fluid/solid regions to be defined in subsequent calculations using the same mesh but without returning to the mesh generation stage.
- Rectangular and elliptic shapes are available in one and two-dimensions, while blocks, ellipsoids and cylinders may be applied to three-dimensional meshes.
- A library of point-source blast results for 1 kg of TNT allow for simulations of blast effects on targets to be performed more effectively. Stored blast profiles are scaled to give equivalent waveforms for a range of explosive sizes and types. These scaled waveforms are used to initialize the mesh with the results of a blast at a later time. Using TNT profiles saves computing time as the initial expansion of the explosive is pre-calculated, and therefore allowing the simulation to be initialized to a point in time when the incident blast wave is close to the target.
- User-created one-dimensional blast results can be used to initialize two- and three-dimensional simulations.
- User-defined two-dimensional axisymmetric blast results can be applied to three-dimensional grids by rotating the two-dimensional results about an axis of symmetry.
- Hydrostatic pressure may be applied to underwater simulations to account for the variation in water pressure with depth.

Fluid-Structure Interaction Tools

- Fluid-structure interaction simulations can be performed in which the structure undergoes small deformations with respect to the size of the fluid mesh resolution.
- Simulations can be performed by coupling Chinook with either Martec's TRIDENT FEA software, or with LSCT's LS-DYNA.
- Simulations performed using the TRIDENT FEA package supports both one and two-way coupling. Two-way coupling allows for the effects of the motion of the structure to be applied back to the fluid. An example of this is in UNDEX calculations, where motion-induced cavitation has a significant effect on structural loading. FSI simulations using LS-DYNA only support one-way coupling.

Input Options

- Chinook 2005 is a command-line based program, which is supplied without a graphical user interface. Keyword-style input files can be created in any text editor.

Output Options

- In addition to outputting spatial results, other output options include:
  - Monitor point - outputs parameters of interest at a point as a function of time (for example, pressure vs. time)
  - Thrust surface - outputs x, y, and z components of force acting on a boundary as a function of time
- Specifying wall output in three-dimensional calculations exports a file containing only the pressures acting on solid boundaries, a feature which is useful for 3D visualization
- Support for DPlot and Amtec's Tecplot post-processors. DPlot output is limited to monitor point output, while all forms of Chinook output can be processed using Tecplot. Please contact Martec regarding exporting data formatted for other post-processors.

Minimum System Requirements

Minimum system requirements in order to run Chinook are as follows:

- Windows
  - Intel® Pentium® 4 processor, or equivalent
  - Microsoft® Windows 2000, Windows XP
  - 512 MB of RAM
  - 1GB of available hard-disk space
- Linux
  - 32 bit Intel Pentium processor
  - Redhat Enterprise Edition
  - 128MB of RAM
  - 1GB of available hard-disk space

Martec Limited
1888 Brunswick Street, Suite 400
Halifax, Nova Scotia
Canada B3J 3J8

tel. +1 (902) 425-5101
fax. +1 (902) 421-1923
info@martec.com
www.martec.com