INGENIERÍA y SISTEMAS COMPASS

Tdyn 11 Features

Introduction

Tdyn is an environment for computational fluid-dynamics (CFD), multi-physics simulation, and fluid-structure interaction calculations, based on the leading stabilized finite element method (FIC-FEM).

Tdyn includes different modules that allow to solve Heat Transfer in both solid and fluids, Turbulence, Advection of Species in fluids and solids and Free Surface problems using the same stabilised scheme mentioned above. Tdyn also offers advanced solvers for beams, shells and solid structures, and includes cutting-edge tools for solving fluid-structure interaction (FSI) problems.

By using Tdyn it is also possible to configure additional user defined partial differential equations (PDE) solvers and to couple them with any of the other variable of the problem.

The different analyses available in Tdyn are fully integrated in an advanced graphic user interface (GUI), for geometry and data definition, mesh generation and post-processing the analysis results. The Tdyn GUI uses a versatile tree-like interface for data managing, allowing an easy control of the whole process of entering the analysis data.

To help the data definition process, Tdyn provides tools to easily configure the type of the analysis to be carried out (structural analysis, CFD, multi-physics, fluid-structure interaction, ...). Based on the initial selection done, Tdyn pre/processing system is adapted to minimize the time required to insert the required analysis data.

Furthermore, all the modules of Tdyn include context-sensitive online help for panels / menus, and help / tutorial documentation.

Tdyn meshing technology includes a suite of tools to create high-quality unstructured, structured and semi-structured meshes, including boundary layer meshes, in an automatic way and verify the quality of the generated elements.

Finally, Tdyn provides a variety of tools allowing having a perfect control over the process and to verify its quality.

A more detailed explanation of the capabilities of every Tdyn module is given next.

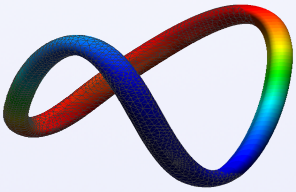
Ransol module

This module features the latest technology for solving real fluid flows in 3D (transient or steady), including turbulence effects. The fluid models available allow to solve from fully compressible to incompressible fluid flow problems and porous media flows (Stokes problem in solid materials).

Physical properties and other problem data used in this module can be defined in terms of any other variable of the problem (i.e. temperature, species concentration, ...). This way it is possible to analyse fluids with variable viscosity or define external floatability forces acting on the fluid. Ransol module also allows to define non-linear Darcy's law effects.

Tdyn offers a choice of 14 different predefined turbulence models, including RANS, ILES (Implicit Large Eddy Simulation) and DES (Detached Eddy Simulation) types. It also allows the user to define new ones based on analytical expressions or by coupled systems of PDEs (see Ursolver module).

Ransol module includes powerful tools for defining space varying, transient and advanced boundary



Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - C/ Tuset 8, 7º 2ª, Barcelona 08006 (Spain)
Sociedad inscrita en el R.M. de Barcelona, T.33954, fol.128, hoja B236843, inscrip. 1. CIF. A-62485180

Tdyn 11 Features

conditions. This way, profiles of velocity and pressure can be specified on openings and walls.

Heatrans module

Heatrans module is able to solve complex forced, natural, and mixed convection heat transfer in fluids and conduction in solids.

This module also includes conjugate heat transfer technology where calculation of thermal conduction through solid materials is coupled with the calculation of the temperature in the working fluids.

All the problem data can be defined in terms of any other variable of the problem. This way it is possible to analyse solids with variable heat capacity, define profiles of heat fluxes in walls or even simulate problems including phase change.

Advect Module

This module allows to solve problems of advection/diffusion of species in fluids, including reactive effects. It is also able to solve species diffusion problems in solids (Fick's law).

Advect allows to define and track the evolution of a number of new species, being the physical properties and behaviour of the fluid flow defined in terms of the species concentration.

Tdyn incorporates finite calculus (FIC) algorithms to increase stability and accuracy of advection / diffusion problems.

Ursolver Module

Ursolver module is able to solve user-defined PDE problems in fluids and solids. This module allows to define a number of new variables (Φ-phi problems) and specify and solve the differential equations that reign their behaviour. New user-defined problems can be coupled among them or with any other variable used in Tdyn (i.e. velocity, pressure, temperature, ...).

Ursolver module features a powerful Tcl interface enabling integration into design processes, faster completion of repetitive tasks and further extension of Tdyn capabilities. Tdyn Tcl interface provides an interpreter for the standard Tcl scripting language, enabling users to enhance simulations without recourse to external compiled subroutines. Furthermore, Tdyn Tcl interface allows access to advanced features, including operations on internal structures and execution / communication with external programs.

Tdyn Tcl interface also offers a library for vector and matrix operations, including mesh interpolation tools, that makes easy to connect Tdyn with other solvers.

Alemesh Module

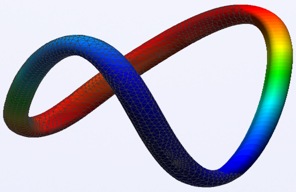
When the analysis require geometry updating, Alemesh offers several mesh updating strategies: prescribed / free bodies movement with automatic mesh updating, explicit 3D mesh movement via user functions and combinations of these strategies. These algorithms cover almost every conceivable mesh movement need.

Furthermore, Alemesh module features advanced Arbitrary Lagrangian-Eulerian (ALE) algorithms, offering faster and accurate results in problems with moving meshes.

For fluid structure interaction (FSI) analyses, Alemesh algorithms manages the necessary mesh updating in an automatic way.

Ramseries Module

Ramseries module features an advanced environment for structural analysis based, based on the latest Finite Element



Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - C/ Tuset 8, 7º 2ª, Barcelona 08006 (Spain)
Sociedad inscrita en el R.M. de Barcelona, T.33954, fol.128, hoja B236843, inscrip. 1. CIF. A-62485180

Tdyn 11 Features

Method technology.

Ramseries module includes 3D beams, cables, shells, membranes and solids models, as well as a full support for nonlinear elements, nonlinear and linear material laws, and inelastic material models. Ramseries easily simulates even the largest and most intricate structures. Furthermore, Ramseries features the latest technology for solving structural dynamics analyses, including contact-impact algorithms which permit the analysis of most practical problems in engineering.

As for composite materials, Ramseries allows to study laminated beams and shells, including the capability of setting predefined standard laminate sequences giving each ply direction, based on local or global frames of reference. A key feature here is the possibility of viewing the critical strains and stresses in each of the laminated material plies or tissues.

Ramseries module can perform coupled analysis, ranging from thermo-mechanical analyses to the most sophisticated fluid-structure or wave-structure interaction problems.

Furthermore, Ramseries module includes specific tools for naval applications, including stiffened shells, wave loads (sagging, hogging and generic), automatic equilibration of the ship (sinkage and trim), specific utilities for composite materials definition, etc.

It is also possible to define wave loads, using the wave significant parameters relative to the hull. For static analysis, an automatic equilibration of the ship in that conditions is performed, adjusting sinkage and trim. This allows a complete structural analysis of the hull, with the self-weight loads, specific weights loads and the loads due to the pressure of sea water.

Fsurf Module

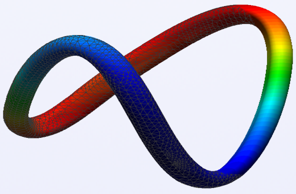
Fsurf module features the leading overlapping domain decomposition level set technique, developed by Compass IS for free surface simulation. This methodology is based on the application of domain decomposition techniques and allows increasing the accuracy of the free surface capturing (level set equation) as well as precisely solving governing equations in the interface between two fluids, taking into account surface tension effects. The greater accuracy in the solution of the interface between the fluids allows the use of non-structured meshes with larger elements in the free surface. Fsurf module includes robust mass conservation techniques and optimized surface tracking algorithms, increasing the accuracy in capturing sharp interfaces between two fluids and allowing efficiently solving problems with large free surface deformation. Furthermore, Fsurf module features a state-of-the-art algorithm for towing analysis, based on the transpiration technique, largely improving the resolution of the free surface deformation, even for coarse meshes.

Seakeeping Module

This module offers an advanced tool for seakeeping analysis. While most seakeeping softwares available are based on the frequency-domain and boundary element method (BEM), Seakeeping module has been conceived for more realistic simulations working in the time-domain and using the FEM, which allows the use of unstructured meshes recommended for the simulation of complex geometries. Furthermore, seakeeping module features an utility to perform waves-structure interaction analysis.

Technical Specifications

- Tdyn solvers are based on the latest Stabilised Finite Element Method technology, offering the highest stability and accuracy.



INGENIERÍA y SISTEMAS COMPASS

Tdyn 11 Features

- Fluid Flow solvers library includes; 2n order implicit Fractional Step and 2n order implicit Monolithical (predictor / corrector) scheme.

- Fluid models available in Tdyn are; fully incompressible, slightly compressible, barotropic, incompressible ideal gas and ideal gas.

- Tdyn incorporates a choice of 1^{st} and higher order finite increment calculus (FIC) algorithms to increase stability and accuracy of advection / diffusion problems.

- Coupled fluid / solid heat transfer is solved by Conjugated heat transfer technology. This technology is also used to solve coupled problems in Ursolver and Alemesh modules.

- Tdyn is fully integrated in a graphic pre/post-processor environment based on GiD system.

- Mesh generator is based in the advancing front technique, and includes tools for automatic generation of unstructured and structures meshes, and boundary layer meshes.

- Library of elements includes:

Generic elements: Linear 2 or 3 nodes, Triangle 3 or 6 nodes, Quadrilateral 4, 8 or 9 nodes, Tetrahedron 4 or 10 nodes, Hexaedron 8, 20 or 27 nodes and Triangular Prism 6 nodes.

Structural elements: Triangle DKT and 6 nodes, Quadrilateral 4-8 nodes, Tetrahedron 4-10 nodes, Hexahedron 8-20 nodes and beam.

Note: Tdyn algorithms are optimised for 4 nodes tetrahedron.

- Structural analysis solver features: Linear / Nonlinear, Static / Dynamic, Transient and Natural frequency, Harmonic Response, Response spectrum, Random vibration, Coupled added mass vibration, Buckling P-Delta method.

- Tdyn provides advanced utilities for the generation of parametric models, based on XML descriptions.

- Tdyn post-processing tools include; contour fill, contour lines, display vectors, iso-surfaces, graphs, streamlines, result surface and line diagram among others. Tdyn also includes utilitites for quasi-automatic generation of results, based on the leading *Lognoter* technology.

System Requirements

Tdyn is developed using C++ Tcl/Tk and OpenGL and is optimised for the best performance possible in workstations under Windows or Linux, with seamless transfer of data between Windows NT and Linux/Unix.

- *Windows*

Windows NT / XP / XP64 / Vista / Vista64 / 7 / 7 64

Minimum requirements: 1.0 GB RAM (+2GB recommended) and 1.0 GB of hard disk space

3 button mouse recommended (Space-Ball supported)

Support any graphics card with OpenGL acceleration

- *Linux / Unix*

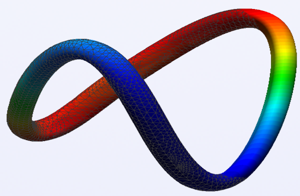
Linux kernel version 2.0.30 or higher

Minimum requirements: 1.0 GB RAM (2.0 GB recommended), 1.0 GB hard disk space

3 button mouse recommended (Space-Ball supported)

Support any graphics card with OpenGL acceleration

Note: 1.0 GB RAM allows the analysis of 3D problems up to 1.8 million linear tetrahedron (32 bits systems, approx. 1.4 million linear tetrahedra in 64 bits systems). Every additional GB increases this limit in about 2.5 million linear tetrahedra.



Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - C/ Tuset 8, 7º 2ª, Barcelona 08006 (Spain)
Sociedad inscrita en el R.M. de Barcelona, T.33954, fol.128, hoja B236843, inscrip. 1. CIF. A-62485180