



Contribution ID: 195

Type: not specified

SIMULATING FLUID FLOW WITH COMPLEX PHYSICS IN ARBITRARY SHAPED DOMAINS BY CFD CODE FLOWVISION

Monday 3 July 2017 14:45 (15 minutes)

FlowVision software is designed for automation of engineering calculations in industrial computational fluid dynamics (CFD). It has a capability to solve complex non-tradition problems involving different physical processes. The paradigm of complete automation of labor-intensive and time-taking processes like grid generation makes FlowVision attractive for many engineers. FlowVision includes an advanced graphical interface, the system for specifying a computational project as well as the system for flow visualization by different methods –from planar to volume visualization.

The software is based on the finite-volume approach to approximation of the partial differential equations describing fluid motion and accompanying physical processes. It provides explicit and implicit methods for time integration of these equations. FlowVision has own split-based method for solving Navier-Stokes equations allowing to solve incompressible as well as supersonic gas flows existing simultaneously in one computational domain. The software includes automated generator of unstructured grid with capability of its local dynamic adaptation and resolving boundary layers. The solver involves two-level parallelism which allows calculations on computers with distributed and shared memory (coexisting in the same hardware). FlowVision incorporates a wide spectrum of physical models: different turbulence models, models for mass transfer accounting for chemical reactions and radioactive decay, several combustion models, a dispersed phase model, an electro-hydrodynamic model, an original VOF model for tracking moving interfaces. It should be noted that turbulence can be simulated within URANS, LES, and ILES approaches. FlowVision simulates fluid motion with velocities corresponding to all possible flow regimes: from incompressible to hypersonic. This is achieved by using an original all-speed velocity-pressure split algorithm for integration of the Navier-Stokes equations.

FlowVision enables solving multi-physic problems with use of different modeling tools. For instance, one can simulate multi-phase flows with use of the VOF method, flows past bodies moving across a stationary grid (within Euler approach), flows in rotary machines with use of the technology of sliding grid. Besides that, the software solves fluid-structure interaction problems using the technology of two-way coupling of FlowVision with finite-element codes. Some industrial examples of solving challenging problems in the FlowVision software are demonstrated in this paper. The first one is splashdown of a spacecraft after deceleration by means of jet engines. The second problem is simulation of the work of a human heart with artificial and natural valves designed on the basis of tomographic investigations with use of a finite-element model of the heart.

Author: Dr AKSENOV, A (TESIS Ltd, Moscow, Russia)

Presenter: Dr AKSENOV, A (TESIS Ltd, Moscow, Russia)

Session Classification: Mathematical methods and application software for modeling complex systems and engineering (I)