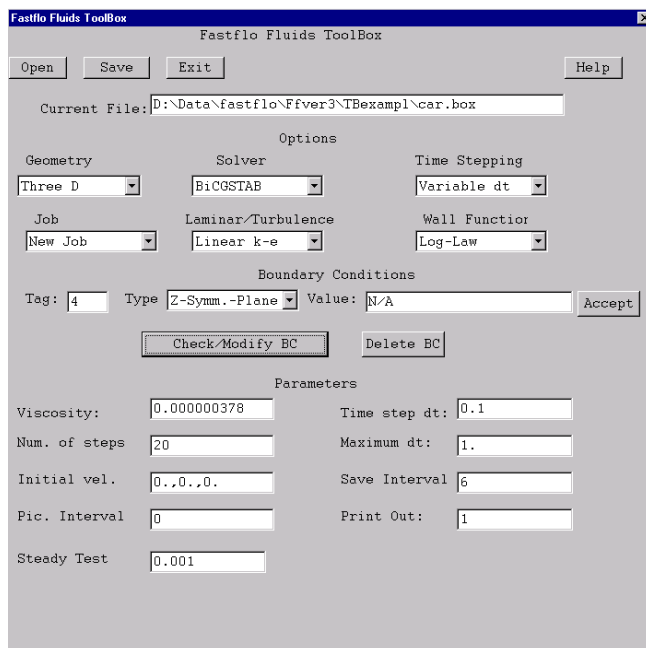


Fastflo

Fluids ToolBox

Technical information for prospective users

The Fluids ToolBox was released with Version 3 of *Fastflo*. This significant development makes available Computational Fluid Dynamics (CFD) expertise assembled by the *Fastflo* team since 1991.



The Toolbox consists of a User Interface, shown at left, accessed via the normal *Fastflo* workbench. The User Interface has four sections:

- file management (editing, saving etc.)
- options (this contains pull-down menus with options for fluid model, turbulent boundary layer treatment, solvers)
- boundary conditions (again with menus with various options)
- parameters (all with defaults, but generally requiring user specification)

Users need to have a general knowledge of the *Fastflo* package, including the way problems are specified (through mesh and problem files), the way that post-processing is carried out, broad details of *Fastflo*'s high level language *Fasttalk*, and the use of *Fastflo*'s graphical user interface.

The scope of the Fluids Toolbox is

- two- and three-dimensional flows in fixed geometries
- incompressible, viscous fluids
- laminar flow (for appropriate Reynolds numbers)
- three models for turbulent flow, namely linear k- ϵ model, nonlinear k- ϵ model and RNG model
- three alternative treatments of boundary conditions for turbulent flows: no-slip, mixing length effective viscosity and logarithmic wall functions
- time-dependent flows, with the option of pseudo-transient convergence to the steady state
- user-defined body forces, for example in natural convection

Users are expected to have background knowledge of

- the theory of fluid mechanics for incompressible, viscous fluids
- the theory of partial differential equations and numerical algorithms for their solution
- a working knowledge of the finite element method

Experienced users will be able to supplement the ToolBox with normal *Fasttalk* code. By this means, it is possible to incorporate capabilities additional to those explicitly included in the menu of the ToolBox. As an example, free and forced convection can be incorporated through an additional equation to describe the energy balance, thereby providing an additional forcing term to the Navier-Stokes equations.

The principal CFD algorithm used within the ToolBox is Glowinski's θ -scheme. This algorithm, also known as an operator-splitting or segregated solver, is well known in the literature, and we have found it accurate, fast and robust.

With the Fluids ToolBox is provided a user manual that gives summary details of the operator-splitting algorithm and the various turbulence models and their boundary conditions. The manual describes how to set up and run CFD calculations. Various examples are then discussed in detail. These range from common test examples to substantial industrial computations. Appropriate files are provided so that users can run these examples. The seven examples are

- 2D driven cavity problems for laminar flow at different Reynolds numbers
- 2D vortex shedding from a circular cylinder in a duct
- 2D turbulent flow over a backward facing step
- 2D turbulent flow around an airfoil
- 2D natural convection, for which a user supplied additional equation is required
- axisymmetric turbulent flow in a 180° narrowing bend
- 3D turbulent flow around a model car

For further information contact:

CSIRO Mathematical and Information Science
Private Bag 33, Clayton South, Victoria 3169, Australia
Tel: 61 3 9545 8005, Fax: 61 3 9545 8080