



3D turbulent flow solver TUDflow3d

TUDflow3d is a fully 3 dimensional CFD (computational fluid dynamics) solver which can give detailed turbulent results on engineering scale. The Large Eddy Simulation approach is used to capture the influence of turbulence; hence the most important larger scale eddies are resolved instead of modeled like in RANS. Density currents can be solved, for example warm/cold water outlets, salinity flows, sediment transport.

TUDflow3d is based on a numerical code from the TU Delft and further developed to be applicable for engineering scale at Svašek Hydraulics. It uses a fast solver on structured grids and runs parallel through MPI. On a 16 core workstation engineering scale flow problems with high Reynolds number can be simulated within hours-days on grids involving 1-50 million cells.

TUDflow3d has been applied successfully for¹:

- Release of sediment plumes from a moving dredging vessel
- Release of tailing plumes near the seabed
- Salinity plume in a crossflow
- Turbulent channel flow with or without high concentration sediment
- Density currents with and without deposition

¹See for further description and extensive validation: De Wit (2015), 3D CFD modelling of overflow dredging plumes, PhD dissertation, Delft University of Technology

Developer

Svašek Hydraulics / TU Delft

Main features

3D turbulent flow
Sediment transport
Density currents (warm/cold, salinity, sediment)
Large Eddy Simulation

Date

2008-2014

Svašek Hydraulics

Schiehaven 13G, P.O. Box 91, 3000 AB Rotterdam, The Netherlands.
Phone +31 10 467 13 61, Fax +31 10 467 45 59, Internet: www.svasek.com, E-mail: info@svasek.com

TUDflow3d