Computer Models
For
Fire and Smoke

Model Name:	Star-CD
Version:	V3100A
Classification:	Field Model
Very Short Description:	General purpose unstructured CFD code, which contains industry standard models for modeling fire and smoke movement.
Modeler(s), Organization(s):	Computational Dynamics Ltd.
User's Guide:	Star-CD V3.100A User Guide
Technical References:	Star-CD V3.100A Methodology Manual
Validation References:	Can provide validation test cases to perspective clients.
Availability:	Available from Computational Dynamics and its agents Worldwide (http://www.cd.co.uk)
Price:	on application
Necessary Hardware:	Unix workstation or a PC running NT or Linux
Computer Language:	Fortran
Size:	-----
Contact Information:	Computational Dynamics Ltd. 200 Shepherds Bush Road Hammersmith London W6 7NY

Detailed Description:

STAR-CD is a powerful general purpose CFD code. It benefits from an easy to use Graphical User Interface, which allow complex scenarios to be developed, simulated and analysed without difficulty. Star-CD is widely used by building and transport service companies to investigate fire and smoke movement in different types of buildings.

The scenario of interest can be generated via the GUI or imported from many popular CAD packages. This allows for simple geometries and the extremely complex scenarios often encountered in many industrial situations to be simulated.

A wide range of physical models are available. Turbulence models range from industry standard k-ε through non-linear k-ε models, to LES models. Combustion can be accounted for using either a heat source, an Eddy Break Up model or a PPDF approach. Radiative heat transfer is simulated using the Discrete Transfer method. Smoke movement can be simulated using a transported scalar.

Later versions of the code will also include additional turbulence models such as Reynolds Stress, alternative radiative heat transfer models such as P1, and allow the user to obtain complex physical properties by interacting with the chemical kinetics package Chemkin.

A range of boundary conditions are available; fluid flow related conditions such as fixed velocity and pressure boundaries as well as conjugate heat transfer conditions to allow the interaction of fluid and solids.

Analysis of the simulation can also be carried out using the GUI. Powerful post processing allows the users to investigate the simulation results in great detail. The information gained can be used to easily manipulate or refine the mesh for further simulations and parametric studies. While, 2 and 3 dimensional plots can be exported to well known image formats, such as GIF and POSTSCRIPT.