Computational Fluid Dynamics (CFD)

TRAX, a leader in engineering simulation, offers Computational Fluid Dynamic (CFD) analysis for a wide range of industrial applications.

We specialize in providing effective and realistic design solutions to provide clients worldwide with accurate, dependable results.

Our studies model the flow of liquids, gases and particulates through constrained surfaces such as pipes or ducts, or through unconstrained environments.

WHY CFD?

CFD analysis is used to numerically solve flow equations in complicated geometries. TRAX CFD analysis can be used to model:

- Single and multi-phase fluid flow
- Heat transfer
- Mixing
- Combustion and reacting flows

A wide range of variations in physical design and operational parameters can be tested and refined until optimum performance is identified—preventing inefficient operation or costly redesign.

Interface programs allow data to be easily transferred between a *ProTRAX* dynamic simulation and a CFD analysis. This provides an initial overall look at the plant, and then a more detailed analysis of critical flow areas.

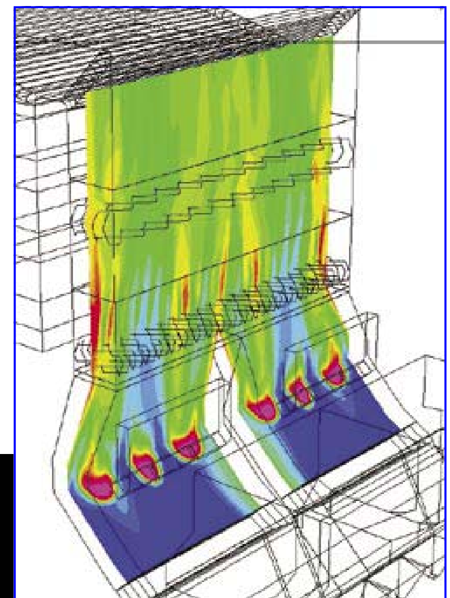
APPLICATIONS

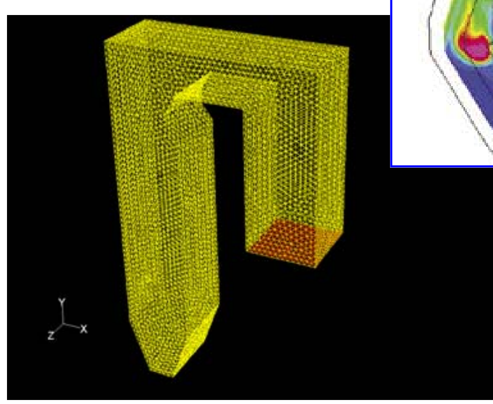
- **Determine temperature extremes**
- **Examine particulate separation**
- **Evaluate fluid mechanics**
- **Determine concentrations of mixed gases**
- **Evaluate coal combustion**
- **Examine heat exchange**

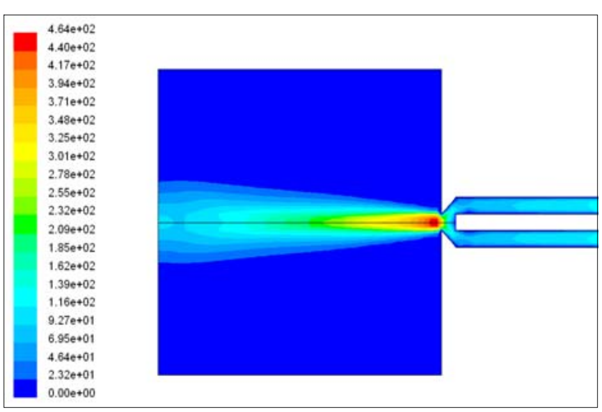
CFD represents an efficient and effective tool to optimize the design of pipes or ducts including internals such as mixers, flow straighteners or guide vanes.

Right, a CFD representation of temperature distribution

Below, meshed geometry of a typical furnace







Above, contours of velocity for atomizer geometry

TRAX INTERNATIONAL™
5061 Fort Avenue, Lynchburg, Va. 24502
Phone: 434-485-7174 Fax: 434-485-7101
E-mail: cgammon@traxintl.com
www.traxintl.com