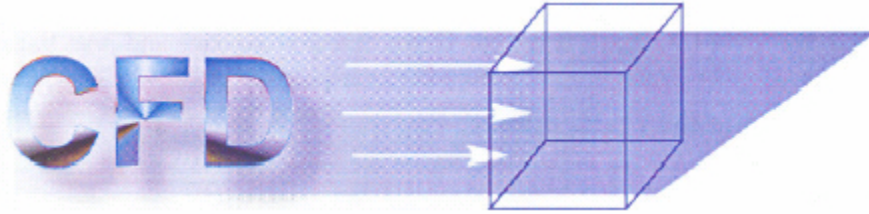


# Continental Research And Engineering, LLC

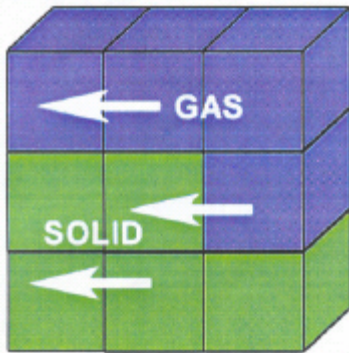


*Engineered solutions today for tomorrow's challenges.*

Computational Fluid Dynamics (CFD) has been an integral part of Continental Research and Engineering's computational "arsenal" for more than 2 years now. This state-of-the-art software package allows us to study complex geometrical problems involving fluid, heat and mass transfer with great accuracy. All of this can be accomplished without ever building a prototype or pilot plant. This is especially valuable in eliminating unwanted "surprises" in the operating plant, which has the potential to cause start-up or operational headaches. There is also a cost savings in studying existing equipment since it is now possible to rapidly study engineering design alternatives without costly modifications to existing equipment.

## *How does CFD work?*

This CFD package has a front-end drawing program that allows a three-dimensional (or two dimensional) object to be designed with ease. Once the object is constructed, the entire computational domain is "meshed." Meshing is an integral part of CFD. This means that an object is "carved" or sliced into tiny blocks. In this phase, objects are determined to be solids or fluids. For example, a simulated vehicle in a wind tunnel would have fluid blocks of air residing next to solid blocks of steel.



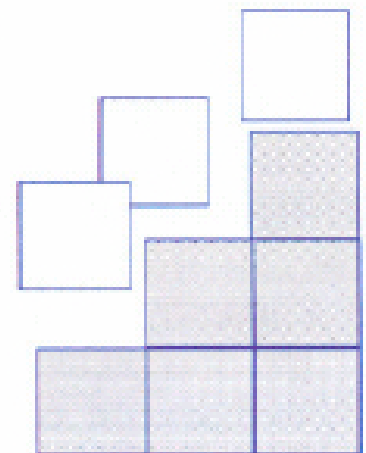
The meshed objects are then imported into the solver. The solver allows the user to input necessary physical properties of materials such as density and thermal conductivity. Initial boundary conditions are established such as the wind direction, speed and temperature, as is the case of the wind tunnel experiment. Models are added for important parameters such as turbulence, radiation, and diffusion. The problems can be transient or steady state, an important aspect for start-up phe-

nomena with plant equipment. The problem has initial values preset and is ready to solve from this point.

From this point the problem is "iterated." Each iteration solves the equations for fluid mechanics, heat transfer and mass transfer for each of the blocks within the problem. The initial guess the computer makes for variables is compared with the solved values. The difference between this initial value and the new calculated value is known as the "residual." These residuals become smaller and smaller as the final solution is approached. When the residuals meet a minimum criterion, the problem is "solved" or "converged." In the case of transient problems, the solver advances to the next time step and the process of iterating begins again.

When the problem is converged, the next and final step is to view the results. The software's post-processing capabilities are extremely flexible and revealing of the ongoing phenomena. The results can be viewed from many variables. Some of the possibilities are listed below:

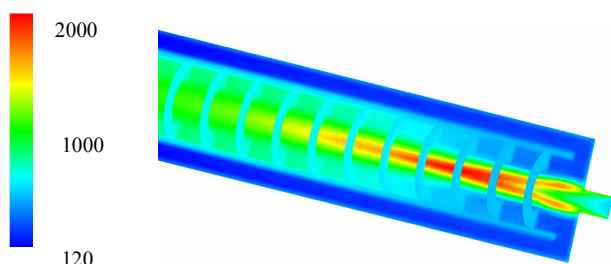
- Temperature
- Enthalpy
- Density
- Pressure
- Flow rate
- Flow direction
- Vorticity
- Momentum
- Shear stress
- Viscosity
- Radiation
- Turbulence
- Reaction rates
- Reaction species
- Particulate paths
- NOx kinetics



Interestingly enough, flow direction in an arrow format can be overlaid on colored arrows to incorporate any one of the above variables. Even “fly-about” movies can allow the user to view the variables at all aspects of the problem or process equipment.

### *What kind of problems does CR&E solve?*

Since the program has been incorporated into CR&E’s capability, it has been used to study a variety of thermal and flow problems.

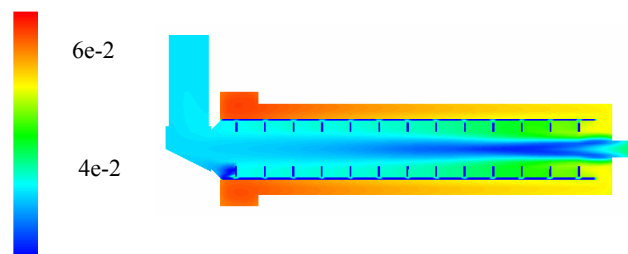


Contours of Temperature [F]

The program has been used to study furnace residence times of various plant components at Johnston Island. The facility at Johnston Island processes chemical weapons, which contaminates various components throughout the facility over the life of the plant. Concrete/ water sludge was simulated within a furnace to determine the required time to reach 1,000 °F, a temperature required for complete decontamination. The water vaporization model which can model the effect of different water content in the concrete has been developed. Using turbulence and radiation heat transfer models coupled with water vaporization model, the process time for concrete sludge can be accurately determined.

A flow problem involving a study of methods to alter airflows within baghouse ductwork was investigated. The objective was to push airflow into the center portion of the duct before the duct funneled the process gas around a 90 degree elbow.

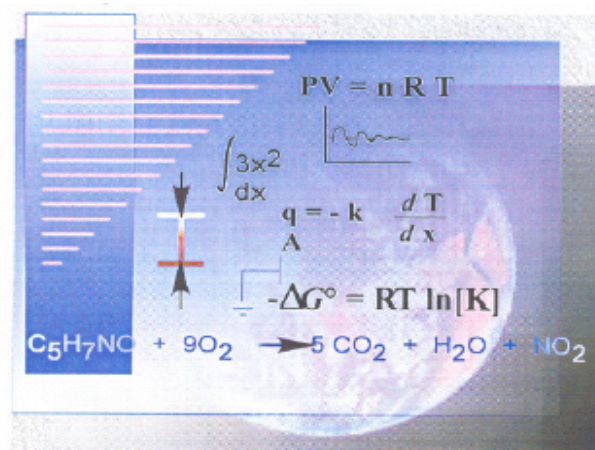
In the past, CR&E has used the software to study gas flow patterns within a rotating kiln and furnace. The kiln is designed to consume propellants and chemical agents within rockets and mines. Temperature profiles, gas flows, reaction kinetics and heat transfer are just



Contours of Density [lb/cu.ft]

some of the variables of interest. The software is being engaged to study charcoal particulates as they are injected into a burner tunnel. Coal or raw material erosion from yard piles is being examined for various power plant facilities. As you can see, the software is designed to study a diverse set of practical engineering problems, wherever mass, fluid flow or heat transfer is important.

The numerical modeling team at CR&E is lead by our PhD engineer, who has over ten years of numerical modeling and research experience. Talk to our on-staff engineers and scientists for your process modeling needs.



**Continental Research and Engineering, LLC**  
**6825 E. Tennessee Ave Suite 101 Denver Colorado 80224**  
**Tel: (303) 758-7373 Fax: (303) 758-1072**  
 Email: cre@cre-denver.com www.cre-denver.com