# *Design Solutions for HVAC&R*



# *Using Fluent CFD Software*

**FLUENT**
INCORPORATED

# HVAC Industry Challenges

*In the HVAC industry today, you face significant technical challenges that impact your company's market share and potential for growth in the increasingly competitive global market. Whether you design complete HVAC systems or manufacture HVAC components, computational fluid dynamics (CFD) software from Fluent can help you overcome your technical challenges.*

## Energy Efficiency

Increased efficiency of HVAC systems and components is important to your customers. Today, buildings account for over 30% of the energy consumed in developed nations. Tomorrow's HVAC systems need to employ new technologies and improved component performance. CFD modeling can help you optimize the efficiency of fans, compressors, heat exchangers, and ducts.

## Indoor Air Quality (IAQ)

Your challenge is to provide building HVAC systems that meet comfort and safety criteria like the ASHRAE standards 52, 55, and 62 and the newly proposed 154P/161P. You may be faced with stringent IAQ requirements for clean rooms or industrial ventilation applications.
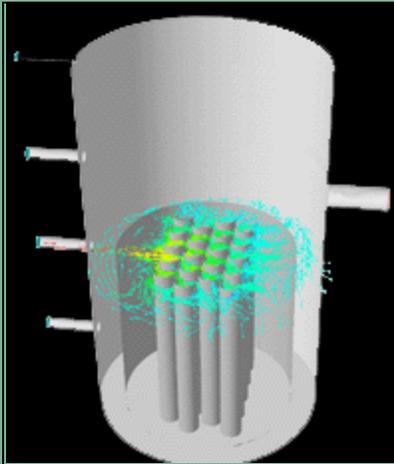
CFD modeling can help you determine ventilation air flows, temperature and humidity distribution, flow of contaminants, and trajectories of respirable particles.
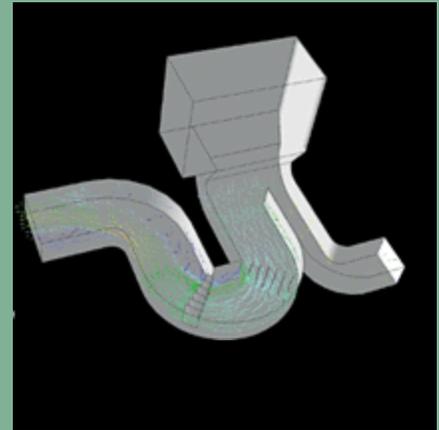
## Alternate Refrigerants

With the banning of ozone-depleting CFCs, you need to deliver new cost-effective designs for refrigeration and air conditioning systems. CFD modeling can help you assess the impact on system performance of alternate refrigerants with new thermodynamic and physical properties.
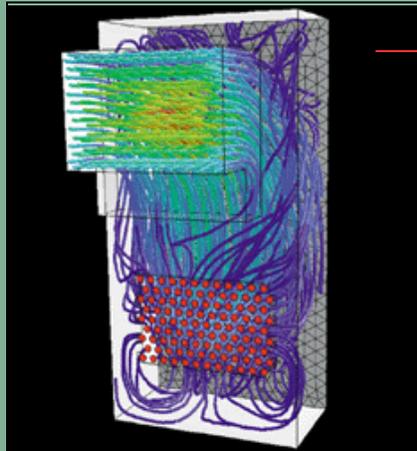
### *Minimize System Cost*

CFD modeling helps you optimize your HVAC system, ensuring that components are not oversized or overspecified due to excessive margins built into outdated design methods. CFD can predict non-idealized performance parameters that you can trust.
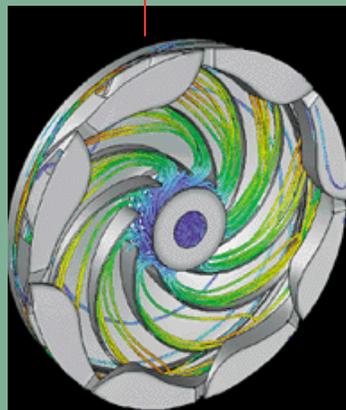


### *Reduce Design Time and Expense*

CFD cuts the time and expense of design by minimizing the number of prototypes you need to examine experimentally. Modeling is faster and less costly than the trial and error design process, and it accurately predicts system performance. Don't pay the high cost of reworking large systems during installation; use modeling and get the design right the first time.

# *CFD*
## Analysis
## Benefits
## You



### *Understand and Diagnose Problems*

By solving fundamental equations, CFD modeling gives you a detailed description of the fluid flow, heat transfer, and chemical species transport in your HVAC system or component. You gain the understanding you need for efficient troubleshooting, and insight that is difficult or impossible to get from experimental programs or field tests. CFD modeling minimizes the risk, time, and cost of retrofitting to fix field performance problems.



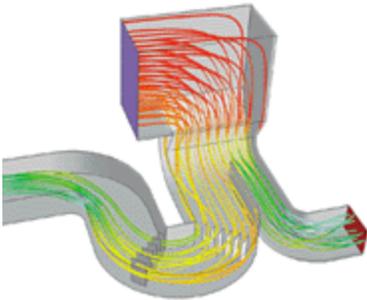**Courtesy of Türk Electrik Endüstrisi A.S.**

### *Improve Performance*

CFD modeling predicts important performance characteristics like pressure loss, flow and temperature uniformity, and heat transfer rate. Using modeling, you can study the impact of alternate design ideas on these performance parameters and improve the efficiency of your final design.

# Step Inside HVAC&R Design

## with Fluent CFD Software



*Surface of high smoke concentration in a multi-level indoor residential space.*

*Fluent's CFD software lets you examine the full range of HVAC design, from overall system design and troubleshooting to individual component performance.*
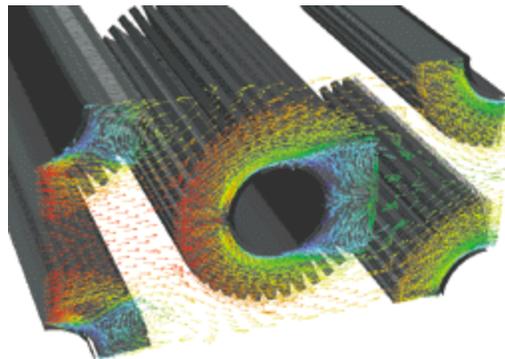
## Room Air Flows

Using your building geometry, vent locations, and fan performance curves, Fluent's software can predict the distribution of air velocity and temperature in room air flows. Buoyancy of heated gases and solar irradiation effects can be included. Plumes of hot gas and smoke from fires can be predicted in order to assist in safety analysis.

## HVAC Ductwork



*Flow pathlines colored by pressure quantify head loss and reveal that turning vanes in the 180° hard-bend are not able to eliminate a large separated region downstream of the fitting.*
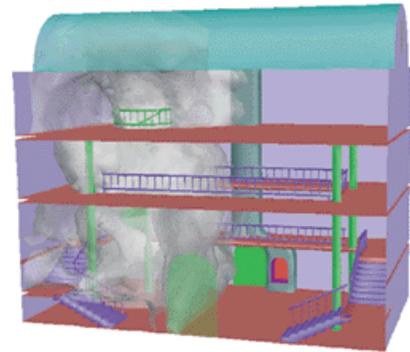
Pressure losses in ducts used for heating and ventilating have a significant impact on overall system energy requirements and component sizing. CFD modeling helps eliminate the guesswork when estimating duct losses. Traditional duct fitting loss coefficients are based on fully-developed flow and don't correctly account for close-coupled fittings. CFD allows you to quantify the real losses and avoid excessive margins of safety in your system design.

## Heat Exchangers



*Velocity vectors colored by temperature illustrate the complex flow and heat transfer in this enhanced surface fin/tube heat exchanger bundle. Courtesy of FINTUBE.*

Design of heat exchangers for enhanced heat transfer involves a critical tradeoff between pressure drop and heat transfer. Fluent's CFD software lets you understand the distribution of heat transfer and quantify the contribution to pressure drop from specific flow details. Fluent's periodic modeling options allow you to model a single, fully-developed heat exchanger section and sophisticated turbulence models ensure that you get accurate predictions.
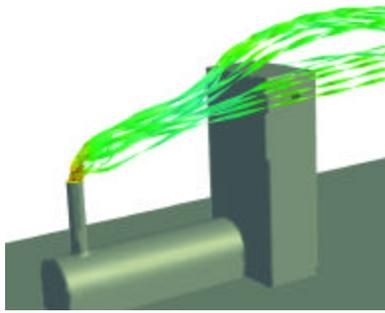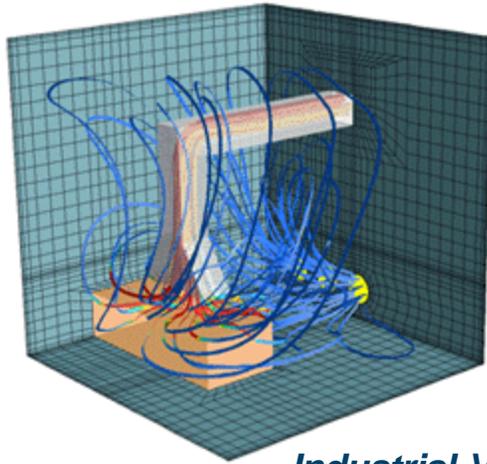
*Flow pathlines illustrate the vulnerability of a building ventilation intake to pollutants from stack exhaust.*



*Flow pathlines colored by contaminant concentration in a bench-top ventilation hood.*
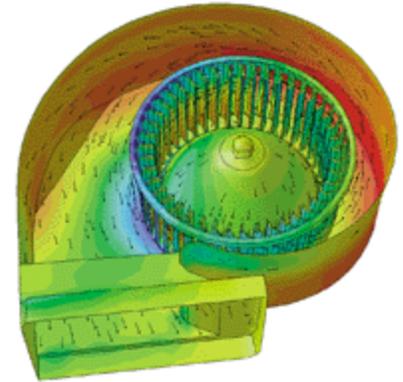
## Airflow Around Buildings

Airflow around buildings can be predicted using Fluent's CFD software. The transport of pollutants from exhaust stacks or vents, under postulated wind conditions, can be analyzed to help decide where to place ventilation intakes on the building exterior.

## Industrial Ventilation

Fluent's CFD software models the flow of air induced by fans and ventilation systems in industrial workplaces. Fluent predicts contaminant concentration levels throughout the work area and captures efficiencies generated by alternate ventilation designs.
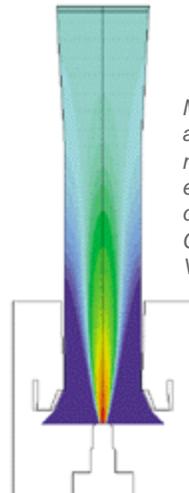
## Blowers, Fans & Compressors



*Fluent's CFD model revealed separated flow and high loss at the exit of this centrifugal blower*

## Heating System Design

Burners and furnace flows can be modeled using Fluent's outstanding combustion simulation capabilities.
You can examine the impact of design changes on flame temperature and pollutant formation. Hot spots and local heat transfer coefficients can be determined, allowing you to study thermal stresses and lifetime reliability issues.
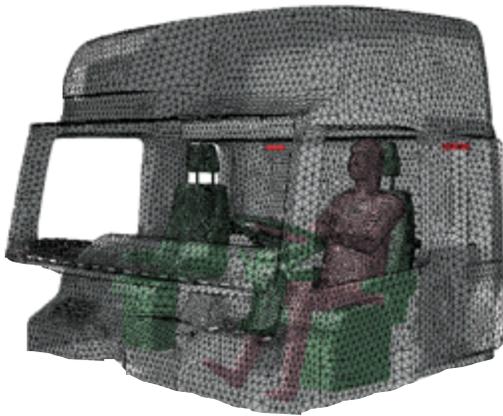


*Mixing of fuel and air in a premixed gas burner used for residential heating. Courtesy of Vaillant GmbH.*

Fan performance curves can be predicted using Fluent's CFD software, including the effect of poorly designed inlets or exits and off-design conditions. Flow separation, stall, and flow reversal in fans can negatively impact performance and can be avoided by using modeling during the design optimization. Fluent provides sophisticated physical and numerical models to handle rotating components.

# The Fluent
# Advantage

- **Compatible with your design process**

- **Fastest tools for model building**

- **Accurate prediction of HVAC physics**

- **Software designed for you to use**

*Fluent provides the best CFD software for design and analysis of HVAC systems and components. From model generation to solving and examining results, Fluent outperforms other CFD software and provides unique features that help ensure your success. Fluent has the leading CFD software for a reason: we deliver the solutions you need.*



*Fluent's CFD software provides all the physical models you need combined with ease of meshing. The triangular surface mesh shown here allowed this complex geometry to be modeled in a matter of hours.*
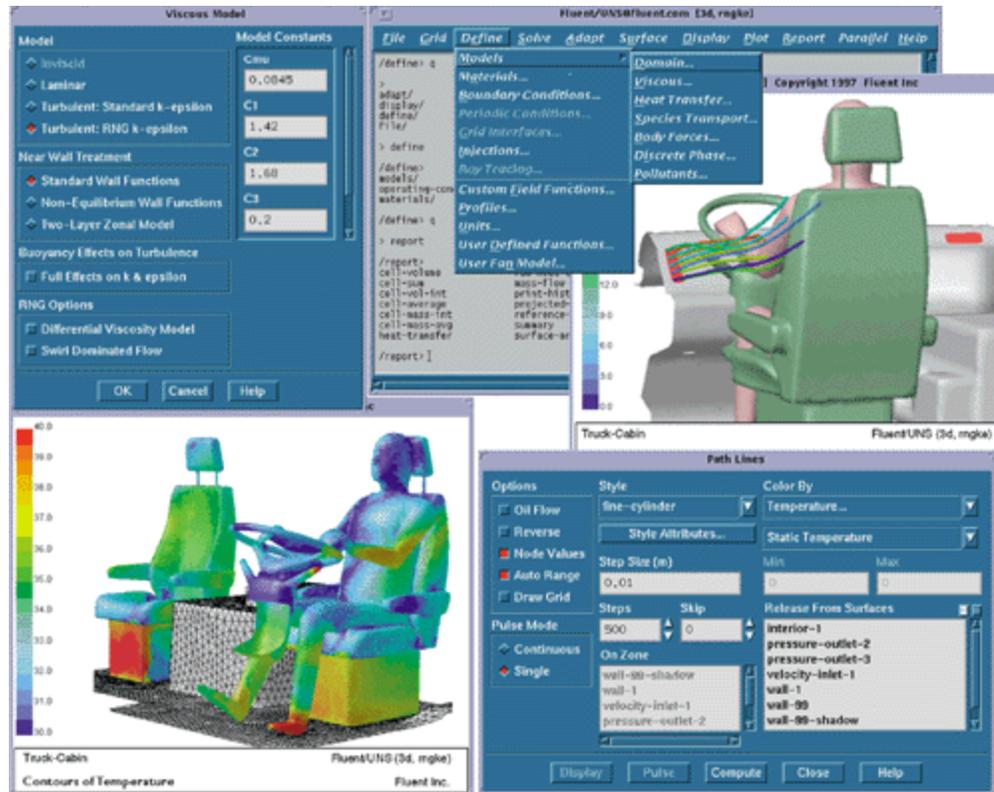
### Compatible with your design process

If your company already uses a CAE/CAD package (PATRAN, ANSYS, I-DEAS, AutoCad, HP M.E., ProEngineer, Unigraphics, Catia, etc. ), your existing geometry data can be used by Fluent for CFD model generation. Fluent also provides its own CAD-level geometry modeler, so you can create your model using our tools, too.

Connecting your CFD results back to other CAE analysis packages is easy, too: we export fluid flow solution data like temperature, heat transfer coefficients, and pressure back to the software you use for structural analysis.

### Fastest tools for model building

Creating a mesh for CFD analysis can take a lot of time. At Fluent, we recognize this and provide the technology you need to overcome this bottleneck in the analysis process. Fluent's software can use any kind of mesh: structured or unstructured, hexahedral or tetrahedral, or even mixed element types. This means you can use our automated tools for meshing – and save hours or days over the time required for

*Prediction of ventilation flow in the confined space of a truck cabin. Fluent's interactive interface is unique and helps contribute to your success with CFD.*

traditional block structured meshing. Only Fluent offers this meshing ease in a CFD code that is time-tested and contains a wealth of physical models.

◆ *Accurate prediction of HVAC physics*

Fluent provides the full range of physical models that you need for modeling HVAC systems and components. You can model laminar or turbulent flows, and our emphasis on accuracy means that you can trust your results – even for mid-range Reynolds numbers that many codes can't handle. You can model steady flow or transient phenomena such as flow instabilities. Heat transfer modes include natural convection, forced or mixed convection, conduction, and radiation. You can model particle or droplet laden flows as well as gas-liquid multiphase flows. Mass transfer and species mixing or reaction can be included. Fluent also provides the input controls you need at model boundaries, from handling periodic conditions in heat exchangers to including profiles determined from upstream models.

◆ *Software designed for you to use*

You don't need a Ph.D. in computational fluid dynamics to succeed with Fluent. Fluent's user interface works with you, interactively, from problem setup to solution to analysis of results. This interactive design, not available in many other CFD packages, contributes to your productivity and minimizes the possibility of errors.

*Join the Leaders in the HVAC&R Industry who use Fluent CFD Software*

**Carrier**

**Nordyne**

**Trane**

**ThermoKing**

**Parsons Brinkerhoff**

**Amana**

**Whirlpool**

**FINTUBE**

**Forney**

**NREL**

**Rheem**

# Choose Fluent . . .

*When you choose Fluent, you choose the world's leading CFD software, used today at nearly 2000 sites worldwide. You also choose a company to work with, a company dedicated to ensuring your success. Fluent's staff of technical experts takes pride in offering the services you need:*

## Training

*Our comprehensive, customized training courses teach you how to apply CFD to your design problems. You'll work with our technical support staff to learn how to plan the analysis, use the software, and interpret results.*

## Technical Support

*The relationship that started during training is strengthened as we stand behind you with unlimited technical support. Successful use of CFD depends upon quality support and Fluent is renowned for providing it.*

## Consulting

*When your organization chooses to use external services for CFD analysis, Fluent's consulting staff can provide the resources you need. We can help you with turn-key solutions or with model generation, high-performance computing, or custom software development for your application.*

# . . . Your Partner in HVAC&R Flow Modeling

**FLUENT**
INCORPORATED