

Solutions for the Heating, Ventilation, Air Conditioning and Refrigeration Industries

HVAC&R

The latest engineering simulation technology from ANSYS, Inc. is used by the heating, ventilation, air conditioning and refrigeration industries for simulation and analysis of ventilation, smoke and fire dispersion as well as HVAC&R equipment. ANSYS delivers the ability to shorten design cycles, optimize existing processes to improve energy efficiency, assist in efficient design of new structures and products, and improve health, safety and the environment.

ANSYS Simulation Solutions

- ▶ ANSYS® CFX®
- ▶ ANSYS® Mechanical™
- ▶ ANSYS® ICEM CFD™
- ▶ ANSYS® DesignModeler™
- ▶ ANSYS® DesignXplorer™
- ▶ ANSYS® BladeModeler™
- ▶ ANSYS® TurboGrid™

CASE-IN-POINT

Analysis with ANSYS CFX allowed engineers at Hatch Mott MacDonald to evaluate the cost and benefits of a wide range of potential designs during renovation of the trainshed at Grand Central Terminal in New York City. Simulations, like the cross section below, made it possible to improve ventilation and obtain greater return on investment.

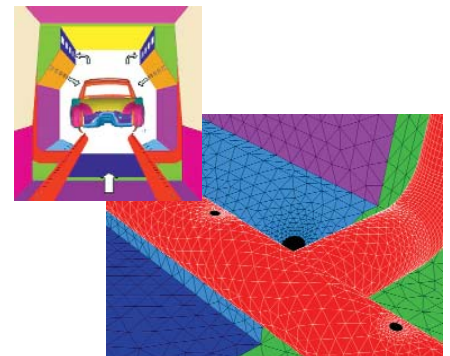


“We use ANSYS CFX because we have found that it can quickly and easily model the complex geometries found in most large building projects.”

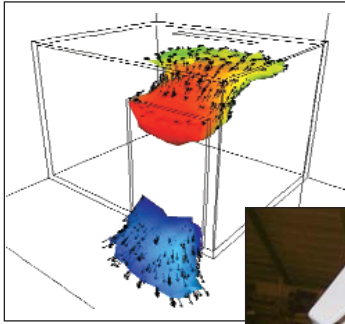
– Tuomas Laine & Sami Lestinen
Olof Granlund Oy

ANSYS provides solutions to:

- ▶ Optimize design to meet requirements through simulation of compressors, blowers, fans, diffusers, VAVs and other equipment
- ▶ Improve occupant comfort and reduce energy consumption through analysis of building ventilation for indoor air quality, age of air and temperature
- ▶ Improve safety in public buildings and industrial applications through simulation of smoke diffusion and fire spread
- ▶ Provide consistent indoor ventilation by modeling of external flows
- ▶ Accurately model refrigerant flows in which, due to the complex fluid nature, ideal fluid approximations can be invalid
- ▶ Increase cabin comfort in the transportation industries
- ▶ Decrease material and energy costs through simulation during the design phase
- ▶ Reduce noise in rotating equipment



At Dürr Systems GmbH, ANSYS ICEM CFD allows quick generation of various types of high-quality mesh over a large, complicated three-dimensional geometry, such as a paint spray booth.



CASE-IN-POINT

Ingress of warm moist air during the opening of cold-store doors is a major issue in refrigeration plants as it increases energy consumption and causes an undesirable rise in product temperature. ANSYS software was used by the University of Bristol, U.K., to help alleviate this problem for several large food suppliers. Later confirmed through testing, the ANSYS analysis determined that air curtains were the most promising method to control the temperature without restricting access.



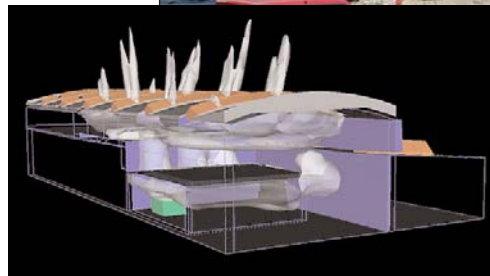
The ANSYS Advantage

ANSYS software provides customers with a competitive advantage:

- ▶ Extensive experience in engineering simulation in the HVAC&R industry, available through software that delivers the ability to model a wide variety of applications, including specialized equipment
- ▶ ANSYS® Workbench™, which provides a unified product development environment offering integration across a wide range of design processes — ranging from geometry modeling and editing, meshing and pre-processing, advanced analysis (structural, thermal, electro-magnetics, CFD, etc.) and robust design optimization
- ▶ Specialized turbomachinery tools for blade design, meshing, simulation and analysis of fans and compressors

CASE-IN-POINT

Air Flow Consulting has been using ANSYS CFX for many years with an increasing number of applications in the HVAC field. Many of the projects involve modeling the airflow in very complex building geometries for major HVAC planning companies in Switzerland. Due to project size and complexity, ANSYS ICFM CFD is used for meshing. The simulation shows smoke distribution inside a restaurant.



About ANSYS, Inc. Solutions

ANSYS designs, develops, markets and globally supports engineering simulation solutions used to predict how product designs will behave in manufacturing and real-world environments. Its integrated, modular and extensible set of solutions addresses the needs of organizations in a wide range of industries. ANSYS solutions qualify risk, enabling organizations to know if their designs are acceptable or unacceptable — not just that they will function as designed. ANSYS helps organizations achieve:

- Innovative and high-quality products and processes
- Fewer physical prototypes and test setups
- Faster return on investment due to reduced development time
- A more flexible and responsive information-based development process, enabling the modification of designs at later stages of development
- A front-end simulation strategy that offers a superior method for bringing products to market in less time and with fewer costs

About ANSYS, Inc.

ANSYS, Inc., founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries. The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-conscious product development, from design concept to final-stage testing and validation. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 25 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ approximately 600 people and distribute ANSYS products through a network of channel partners in over 40 countries.



www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
USA
724.746.3304
ansysinfo@ansys.com

Toll Free USA/Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com

Publication subject to change without prior notice. ANSYS, ANSYS Workbench, CFX, AUTODYN and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICFM CFD is a trademark licensed by ANSYS, Inc. All other trademarks or registered trademarks are the property of their respective owners.