Fire and HVAC: Some recent applications by ANSYS
Yehuda L Sinai
ANSYS Europe Ltd., Milton Park, Oxfordshire OX14 4SA, UK

Abstract
This paper outlines several recent applications at ANSYS on simulations of fire and HVAC using ANSYS CFX, and focuses on three:

1. Web-enabled CFD for HVAC and fire.
2. IFC interface.

A limitation of space means that the material cannot be detailed. However, the paper provides a flavour of significant progress at ANSYS in the HVAC and safety spheres.

Introduction
ANSYS Europe Ltd has been active in the sphere of HVAC, fire, and safety generally. Funded developments and applications have, and continue to tackle a broad range of related issues, such as web-enabled HVAC and fire simulations, explosion risks in acoustic enclosures, fire suppression, flame spread, and under-ventilated fires. This paper outlines three of the activities, namely web-enabled CFD, IFC interfaces, and backdraft.

Consider the web-enabling technology first. CFD technology is evolving fast, and there is increasing awareness that its user friendliness and robustness means that non-specialists can become users. Some years ago, this fact prompted Fläkt Woods, a global manufacturer of ventilation equipment [1], to consider acquiring a web-enabled utility for running CFD for sales purposes by its sales force. In 2000, Fläkt Woods approached ANSYS Europe Ltd. with an outline of this long-term plan. It first commissioned two validation projects, in which ANSYS CFX [2] was subjected to blind validation against Fläkt Woods’ own experiments. Some of this work has been published [3]. The outcome was a good one, and Fläkt Woods then embarked on a series of projects in which the web-enabling functionality was developed, by combining ANSYS CFX with the EASA software [4]. This enables the salesperson to set up a case in minutes at the prospect’s office. The instruction set, which is small, is sent over the internet to the salesperson’s computer system, where scripting creates the geometry and mesh, sets up the CFD model, runs it, and post-processes the results. The results are then downloaded over the internet back to the salesperson’s laptop, in a pre-defined format, to be shown to the prospect. The user has a choice of a quick run, typically less than an hour, which can provide indicative results, or a longer run which is more accurate.

Next, consider IFC (Industry Foundation Class). In the example above, the building geometry is created by the ANSYS software. A common situation in the built environment, however, involves CAD data being used by several organisations. In 1995, the International Alliance for Interoperability (IAI) was formed [5]. The alliance is made up of building product manufacturers, designers, software companies, builders, owners, and others, with the goal of developing a universal standard for sharing object-based information for digital building models applicable throughout all phases of the building life-cycle (including planning, design, construction, operation, and decommissioning). Since then, IAI has been publishing IFCs (Industry Foundation Classes), which are non-proprietary building data elements, that represent parts of a building, or elements of a process, and which facilitate interoperability between the many disciplines and software packages which contribute to a building’s life cycle. The current standard is known as IFC 2x3, and a
listing of the attributes can be found at [6]. This development parallels another initiative known as STEP (Standard for the Exchange of Product model data), co-ordinated by ISO. Both efforts share the same modelling language, known as EXPRESS.

Recently, ANSYS Europe Ltd collaborated with Olof Granlund in creating an interface between IFC data and ANSYS CFX, aiming to automate the geometry and meshing process. The outcome is a module known as CFX-BSClient, which interacts with Olof Granlund’s BSPro software, which interprets the IFC data. It has been found that due to variability in the geometrical accuracy of typical IFC data, the mesh generation process sometimes requires manual intervention, and further work is called for. Nevertheless, this work is already useful, and offers a major potential speed-up of CFD set-up, as well as a smoother integration into the building processes. This paper provides a sample test of this utility.

Turning now to backdraft, this is one of the more hazardous events related to under-ventilated fires. Descriptions of this phenomenon may be found in any of the textbooks on fire safety, e.g. [7], or in a variety of papers and reports (e.g. [8], [9]). Briefly, backdraft is caused by fuel vapour being generated after a fire is extinguished, or reduced in intensity by oxygen starvation, and the subsequent introduction of fresh oxygen, for example by opening of a door. Following the mixing of the fresh air with the fuel rich mixture, concentrations can return to the combustible range, and since ignition sources are likely to exist, flaming combustion may be initiated and can develop into a deflagration. Backdraft was one of the topics analysed by ANSYS Europe during the EU FIRENET project, which is now being completed.

The work outlined here has been a team effort, involving colleagues in ANSYS as well as other companies; the author is only acting here as a compiler of information on that work. For details on who was involved, please refer to the Acknowledgements below.

**Web-enabled CFD for HVAC and fire**

The background to this activity has already been provided in the Introduction. Let us deal first with the EASA software, which is the package seen by the user. A schematic of a typical installation is shown in Figure 1.

The EASA user sees a web page using a standard browser. The contents of this page are designed by an ‘Author’, who ‘publishes’ an EASA Application (known as an EASAP). EASAPs can be run by any number of users, dictated by the licensing arrangements. The Author uses the EASA Application Builder to link engineering models and databases, and to provide automatic display of results, all within a secure remote environment. The typical sequence, once the application is installed, starts with the problem being defined by the Application User, then the data being sent over the internet to the EASA Server, which in turn triggers the pre-processing of CFX, the solver run, the post-processing, and then the transmission of a results set back to the User. For more details, the reader is referred to the EASA website [4].
The GUI in the browser allows static or dynamic display of the object which is being analysed. A static display is a schematic image, typically with dimensions and other characteristics displayed in a generic fashion. In a dynamic display, the true object shape is shown in response to the user input. The EASAPs developed for Fläkt Woods all have dynamic GUI displays. Typical GUIs are shown in Figure 2 & Figure 3. Standing people are represented as vertical cylinders, and patients lying on a hospital bed as horizontal cylinders which are on top of the horizontal slab which represents the bed. An ATD is an Air Terminal Device, or diffuser.

A significant amount of work has been carried out on what is termed ‘scripting’ here. This is the software which runs in batch upon receiving the primitive data from the GUI. A special geometry tool created by ANSYS Europe generates tetin files which are read by ANSYS ICEM, which in turn creates the mesh. Other data is sent to a session file needed by CFX-Pre, which in turn supplies the run definition which is read by the CFX solver. The results are examined by CFX-Post, and a pre-defined subset of the results is written in html format and made available to the User over the intranet.
Dispersion of a contaminant can be simulated. The source can be a point source, or a source distributed over the surface of an entity such as a cylinder representing a human being.
As far as fire is concerned, the present model uses an inert approach, in which the medium is represented only by air, with sources of heat and soot. Soot is modelled as a scalar, and produces the visibility as a field variable. The user can specify either steady-state or transient runs. For the latter, transient sources of heat are prescribed by the user. The soot source is derived from that on the basis of documented yield (usually empirical). Turbulence is represented by the industry-standard k-e model. Thus, the generic form of the eight conservation equations which are solved is (e.g. [10])

\[
\frac{\partial (\rho \phi)}{\partial t} + \nabla \cdot \left( \rho \vec{U} \phi \right) - \nabla \cdot \left( \mu_{\ell} \frac{\nabla \phi}{\sigma_{\phi,\ell}} \right) \tilde{\nabla} \phi = S_{\phi}
\]

(1)

Here \( \phi \) stands for any of the eight variables involved in the PDEs for mass continuity, linear momentum, enthalpy, \( k, \varepsilon \), and soot mass fraction. Also, \( \vec{U} \) is the velocity vector, \( \mu \) is a viscosity, \( \sigma_{\phi} \) is the Prandtl number for the variable \( \phi \), the subscripts \( l \) and \( t \) denote laminar and turbulent respectively, and \( S \) is a source which encompasses all terms other than the transient, advective and diffusive ones which already appear in Eqn (1).

Thermal radiation is also computed. The Radiative Transfer Equation ([11]), encompassing processes in both non-participating and participating media, is

\[
\tilde{\nabla} \cdot \bar{I}_{\lambda} + K \bar{I}_{\lambda} = K_a \bar{I}_{\lambda} + \frac{K_s}{4\pi} \int_{4\pi} I_{\lambda} (\bar{x}, \bar{\Omega}) p (\bar{\Omega}, \bar{\Omega}) d \bar{\Omega}
\]

(2)

Here \( I \) is the radiation intensity, which is generally a function of the wavelength \( \lambda \), but here the grey approximation is used. Scattering is ignored, so the integral vanishes, and \( K \) is therefore equal to the absorption coefficient \( K_a \). The RTE is coupled to the flow equation through the divergence of the radiation flux. The Shah-Lockwood algorithm for solving this equation is used.

All of these models are available as standard options in ANSYS CFX. The reader should note that many other models are available in CFX, for example an LES (Large Eddy Simulation) turbulence model and variants thereof, as well as Eddy Dissipation Model and Flamelet models for non-premixed and premixed combustion. These could be used in future extensions of this application.

**IFC interface**

Following on from the Introduction, an example is given here of a single floor containing room and corridors. An IFC file was processed by CFX-BSCClient, producing a tetin file. Figure 4 shows the surfaces after importing that file into ANSYS ICEM. Figure 5 displays the surface mesh which emerges for this case.
Figure 4. A tetin file after loading into ANSYS ICEM.

Figure 5. The surface mesh.
The volume mesh is then successfully created, and the physics and other aspects of the run are defined in ANSYS CFX-Pre. It should be pointed out that whilst some work has been done in defining one type of diffuser, using the so-called ‘box method’, this is limited in scope, and more work is needed before one can easily incorporate the many categories of diffusers which exist in the marketplace.

**Backdraft**

The backdraft activities at ANSYS have focused on one of the experiments conducted at Lund University by D. Gojkovic [12], in a shipping container. A methane burner was operated with the container nominally closed. The term ‘nominally’ is deliberately used here to explain that the container was not perfectly sealed, and it has been shown previously [13] that leaks and wall heat transfer can have a major effect on stratification in under-ventilated fires. Since conditions inside the container were not measured, there is uncertainty about conditions after the burner had been operated for some time. The flame eventually died, and the door was opened rapidly. In the Lund experiment, a heated wire was used as the ignition source. After the door opened, a gravity current developed, and in most cases, ignition occurred at the wire, leading to a propagating flame which passed through the door and produced an external fireball.

The simulation [14] used the DES turbulence model (a hybrid of the SST and LES models, [15]), a variant of the Eddy Dissipation combustion model [16], and an ignition model which has been developed specifically for this topic. In the computations, the ignition model initiated the combustion process when conditions anywhere along the wire were in the combustible range for methane. The ignition model was initiated automatically as part of the computation.

The reaction rate, i.e. fuel consumption rate per unit volume, was expressed as ([14])

\[
R_f = -F_T \max \left( t_{low}, C_A \frac{\varepsilon}{k} \right) \min \left( \zeta_f, \frac{\varepsilon}{\varepsilon_F} \right)
\]

(1)

Here \( \zeta \) is mass fraction, \( s \) is the stoichiometric ratio, \( \varepsilon \) and \( k \) are the turbulence dissipation and kinetic energy, \( C_A \) is a constant, \( t_{low} \) is a low-turbulence timescale, \( F_T \) is a function of temperature, and the subscripts \( f \) and \( O_x \) denote fuel and oxygen respectively.

Typical results ([14]) are shown below.

![Figure 6. Methane concentration 7.2 s after door opening.](image)

![Figure 7. Temperature field 12s after door opening.](image)

Figure 6 shows the gravity current generated by the heavier fresh air, at a stage when its head has struck the rear wall on the left and has begun the reflection process towards the right. In Figure 7 one can see the external fireball to the right of the compartment, which is so dangerous to firefighters.
Conclusions

Three examples of recent activities, in the HVAC and safety spheres, have been outlined briefly. Two of these, namely web-enabled CFD and IFC compatibility, relate to significant advances in accelerating CFD set-up time, in linking CFD to other tools and processes, and in placing CFD tools in the hands of non-specialists. The third example, dealing with backdraft, is a relatively novel application of CFX in an area of concern to firefighters.

Acknowledgements

Fläkt Woods sponsored the web-enabled work described in this paper. It is the author’s pleasure to acknowledge this fact, and the constructive and helpful role of Fläkt Woods’ staff, who contributed to modelling and performed experiments, especially but not only R Kohonen, J Hokkanen, R Critchlow, M Lahtimo, Y Tallgren, and N Danielsson. The batch scripting and coding of geometry, meshing, and CFX pre-processing, solution, and post-processing were done by M Owens of ANSYS Europe Ltd. It has been a pleasure to collaborate with A Kendall of EASA Software. The EASA GUI was done mostly by M Gerring and P Sobczak of EASA Software.

The IFC activities reported here were carried out in collaboration with Olof Granlund. It is the author’s pleasure to acknowledge the roles of Reijo Hänninen, Tuomas Laine, Hannu Lahtela and Sami Lestinen. The ANSYS work was done by M Owens of ANSYS Europe Ltd..

The Firenet work outlined here was performed as a part of the project "Under-Ventilated Compartments Fires (FIRENET)" (Co. No. HPRN-CT-2002-00197). The project was supported by the EU Research Training Network FP5, which is gratefully acknowledged. The ANSYS researcher was A Horvat of ANSYS Europe Ltd..

It has been my pleasure to work on these projects with my colleagues M Owens and A Horvat.

References


