V European Conference on Computational Fluid Dynamics
ECCOMAS CFD 2010
J. C. F. Pereira and A. Sequeira (Eds)
Lisbon, Portugal, 14–17 June 2010

SIMULATING FIRE & SAFETY APPLICATIONS WITH ANSYS

Ilona Zimmermann*, **Elmar Schneeloch**†

*ANSYS Germany GmbH
Staudenfeldweg 12, 83624 Otterfing, Germany
e-mail: ilona.zimmermann@ansys.com

†ANSYS Germany GmbH
Birkenweg 14a, 64295 Darmstadt, Germany
e-mail: elmar.schneeloch@ansys.com

Key words: Fire & Safety, Combustion, Radiation, Multiphase, Fluent, CFX, ANSYS

Abstract. *Each year, several thousands of people are dying or get injured due to fire accidents. These accidents cause damages in billions of Dollars [1]. In order to reduce the injury to persons and property damages, the use of simulation tools is of steadily growing importance. Thus simulations provide a cost-efficient and secure option to improve fire safety engineering. Different simulation disciplines and possibilities will be shown in the framework of fire and safety applications. As CFD (Computational Fluid Dynamics) plays a major role, an overview of important models will be given, and their application area will be discussed.*

1 INTRODUCTION

Fire is an important energy source for power generation, for the process industry for the production of engineering materials or for heating of various types of buildings. But when it gets out of control, it may cause large damages to human lives, property or environment. Thus it is important to get an understanding for the sources of fire, its spreading, and its impact in order to prevent fire or optimize emergency systems.

Sources and locations of fire are various. Many fires occur in personal residences caused accidentally by radiators, curling irons, candles, cigarettes and so on, or intended by malicious arson. But fire can also occur in public buildings, tunnels, ships, airplanes, trains, cars, trucks, woods etc. by the already mentioned sources or also by explosions. Figure 1 shows global fires in August and December 2009 [2] based on Moderate Resolution Imaging Spectroradiometer (MODIS) on NASA's Terra satellite.

To avoid life hazards and injuries like burns, smoke inhalation, or injuries like crushing caused by collapsing structures it is important to optimize e.g. escape and emergency routes. Experiments are very challenging, expensive and risky to perform in reality, and scaling the data of simplified, smaller models is difficult. Furthermore only a "cold" mist is used in smoke experiments, which neglects important thermal buoyancy forces. Thus the simulation driven building design is a cost efficient and secure tool for fire safety engineering in order to ensure early detection of fires and alarm. Positions of smoke detectors, sprinkling or ventilations systems may be effectively determined, or even the structural integrity may be improved.

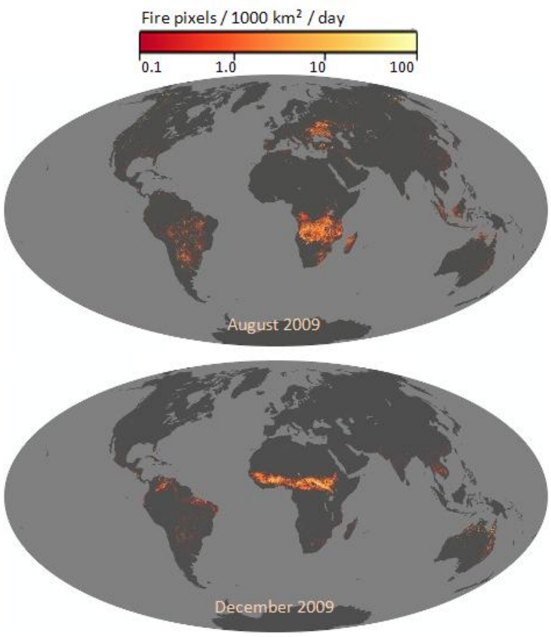


Figure 1.1: Global fires in August and December 2009 [2]

For such complex cases and physics, simple analytical models, norms and standards (e.g. DIN 18232) are not sufficient. Also simple simulation tools based on “zonal models”, where the geometry is divided into different zones (between 2 and 20) for which energy and continuity equations are solved in each time step, are not accurate enough or do not provide enough information. Due to the coarse zonal resolution, only simple geometries can be examined. Although zonal models are easy to use, they are not usable when external influences like wind are important, or the plume is not able to develop due to the room height or built-in components like balconies in atriums. VDI 6019 gives an overview of different accepted methods like analytical models, zonal models and CFD methods, their different applications in the design of smoke development in buildings and procedures for their applications.

Only with CFD it is possible to solve applications ranging from smoke or heat transport from fires to more challenging problems involving complex physics like conjugate heat transfer, combustion or radiation. In CFD methods, the computational domain is resolved by finite volumes, where transport equations for mass, momentum and energy are solved. The generic form of transport equations are defined as [3]

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho U_j\phi)}{\partial x_j}-\frac{\partial}{\partial x_j}\left(\frac{\partial\phi_i}{\partial x_j}\left(\frac{\mu_l}{\sigma_{\phi l}}-\frac{\mu_t}{\sigma_{\phi t}}\right)\right)=S_\phi \qquad (1.1)$$

Where P stands for any variable involved in the partial differential equations (e.g. mass with $\phi = 1$, momentum with $\phi = U_i$, energy with $\phi = E$). U is the velocity vector, μ is a viscosity, σ_ϕ is the Prandtl number for the variable ϕ, and the subscript l and t denote laminar and turbulent respectively. The source term S encompasses all terms other than the transient, advective and diffusive ones.

A virtual building dealing with complex geometries offers the advantage that it can be modeled with great accuracy, the origin of the event can be reproduced, and different scenario can be examined in what-if studies such that the best solution for safety can be identified.

Life safety is in all cases of fire simulations the most important aspect. In technical applications especially fires in (public) buildings are of interest. Already at the design stage of a building, emergency routes have to be taken into account that in case of fire all people are able escape. This includes the guarantee, that the emergency routes remain smoke free. CFD is able to cover this aspect. Different physical CFD models of Fluent and CFX as well as their application will be presented in Chapter 2, and

Chapter 3 gives an overview of additional tools which are helpful to successfully setup a simulation.

Although CFD plays a major role in fire & safety applications, also other disciplines of simulation are of growing importance. For example the structural behavior can be predicted so that buildings can be designed to resist extreme situations better such that structures sustain the induced thermo-mechanical deformations. Material behavior can be modeled by simplified or even complex material property laws. Electrical systems are often origins of fires due to defects or local overheating. Explosions causing significant damage to structures may also be simulated. An overview of these additional tools will be given in Chapter 4.

2 RELEVANT PHYSICAL MODELS

In fire simulations, many different physical phenomena have to be considered. Fire is an uncontrolled, self sustainable heat release where a burnable medium and oxygen are ignited by activation energy. In a building, this can be a chair, a curtain etc. which is further on heated by heat convection and radiation. During the combustion process, particles like soot or dust as well as gaseous products like flue gases are generated which results in a multiphase and multi-component situation. Multiphase flow simulation is also necessary when the fire is extinguished with liquid water.

In general fire and smoke extraction simulations have to be calculated transient. Thus the modeling effort has to be as small as possible without neglecting basic physical phenomenon and effects. As the spatial geometry extension might be very large, this also results in a large number of elements to resolve the geometry which in turn results in an increase in simulation time. In the following physical models [4] as well as their relevance to technical applications will be described.

2.1 Multi-component flows

To be able to track flue gases in space and time and to examine its distribution, it is necessary to model this gas. As the gas mixes with the air, it is not necessary to use a multiphase model. Using a multi-component mixture with multiple species is sufficient, which in the simplest case consists of two components, e.g. air and flue gas, where both components may have different material properties. In addition to the scalar transport equations solving for velocity, pressure, temperature and other quantities of the fluid for the bulk motion of the fluid, additional equations have to be solved for the components. According to the generic form of the partial differential equation (equation 1.1), the transport equation for each component can be written as

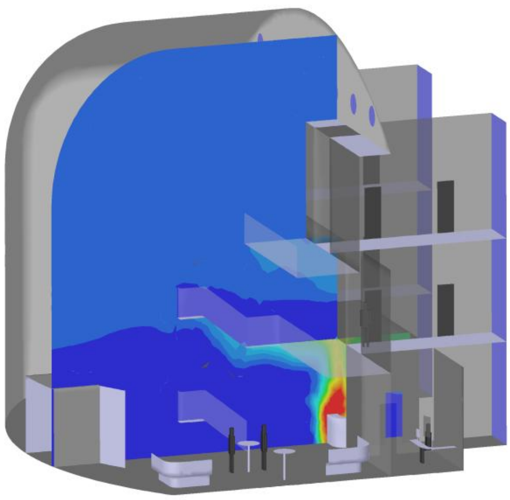


Figure 2.1: Smoke distribution in an atrium after 175 s

$$\frac{\partial(\bar{\rho}\widetilde{Y}_i)}{\partial t}+\frac{\partial(\bar{\rho}\,\widetilde{U}_j\widetilde{Y}_i)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\frac{\partial Y_i}{\partial x_j}\left(\Gamma_i+\frac{\mu_t}{Sc_t}\right)\right)+S_i \tag{2.1}$$

Where ρ is the density, Y_i the mass fraction of component i, U_j the velocity field, Γ_i the molecular diffusion coefficient, μ_t the turbulent viscosity and Sc_t the turbulent Schmidt number. As the mass fraction of all components sum up to unity, it is sufficient to solve N-1 mass fraction equations (where N is the number of components). S_i is the source term of component i, which also includes the effects of chemical reaction.

Figure 2.1 shows the smoke distribution in an atrium after 175 s simulated with a multi-component mixture. The entrance door and a window in the basement are open when the fire breaks out at the reception desk. Ventilation systems are positioned in the roof.

2.2 Heat release and combustion

For the injection of smoke gas as combustion products and the generation of energy as heat of reaction, as well as the dissipation of the consumed air, different approaches can be used. These approaches differ in their complexity and accuracy of the modeling.

Given mass flow for smoke gas and air as well as heat released

The amount of smoke and air (oxygen) that is injected or dissipated respectively and the amount of heat release is given in a pre-defined volume. These values can be constant in time in a worst case scenario, but in most cases vary in time. The magnitudes have to be supplied by the user and can be obtained by evaluation measurements or by normalized curves (e.g. ISO 834). The destruction of mass or the generation of heat is described by the source term (equation. 1.1). This simple approach can be approved by the possibility to change the source of fire in space and time to simulate a growth.

The advantage of this approach is that it the models are very simple which reduces simulation time. A detailed description of combustion and thus a description of material properties of the burning commodity are not necessary. These data are often unknown or unsure. As the source of fire can be scaled, different scenarios can simply be compared.

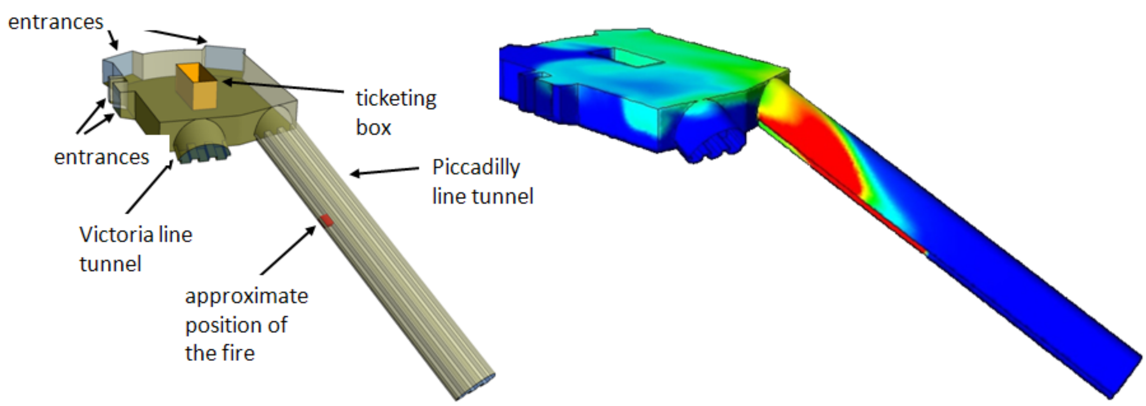


Figure 2.2: Kings' Cross Fire: Left – geometry, right – temperature distribution

Figure 2.2 shows the usage of this approach for the numerical simulation of a fire in an escalator tunnel of an underground station (Kings' Cross accident, UK, 1987). The left figure shows the geometric details, and the right figure shows the temperature distribution a few minutes after the beginning of the fire. Zavila [5] also simulated a motor-car fire in the road tunnel Valik with this approach.

Description of combustion by combustion models

A more detailed description of the combustion process requires more numerical effort and knowledge about material data. For example burning a sofa the description of volatile, combustible species in dependence of temperature needs to be described. But also the surface burn-off of the solid body and the change of geometry during the combustion process are difficult to describe. Thus, such simulations are often performed with alternative fuels like methane which is injected with a defined mass flow into the system. Horvat and Sinai [6] simulated a fire spread to a solid material. The solid material ignited under fire conditions with a subsequent pyrolysis and combustion. The consumption of oxygen and the reaction to combustion products (e.g. exhaust gas) as well as the heat release is then described by combustion models. CFD simulations provide a variety combustion models differing in complexity and application. The two most common combustion models used in fire simulations are the Eddy Dissipation Model (EDM), and the Laminar Flamelet model.

The Eddy Dissipation Model, derived by Magnussen and Hjertager [7] is based on a single step reaction and fast chemistry assumption. In addition to the common conservation and transport equations for cold flows, a transport equation for each species has to be solved (see equation 2.1). The process which determines the reaction rate is the mixing process on the smallest length scale. The local rate of combustion is then determined by the equation that gives the lowest reaction rate [8]

$$\overline{\rho S_F} = C_{mag} \bar{\rho} \frac{\varepsilon}{k} \min\left(\tilde{Y}_F, \frac{\tilde{Y}_O}{\nu}, \beta \frac{\tilde{Y}_P}{1+\nu}\right) \tag{2.2}$$

with the fuel mean burning rate S_F, the turbulent kinetic energy k, the dissipation rate ε, a model constant C_{mag} and the stoichiometric oxygen to fuel mass ratio ν.

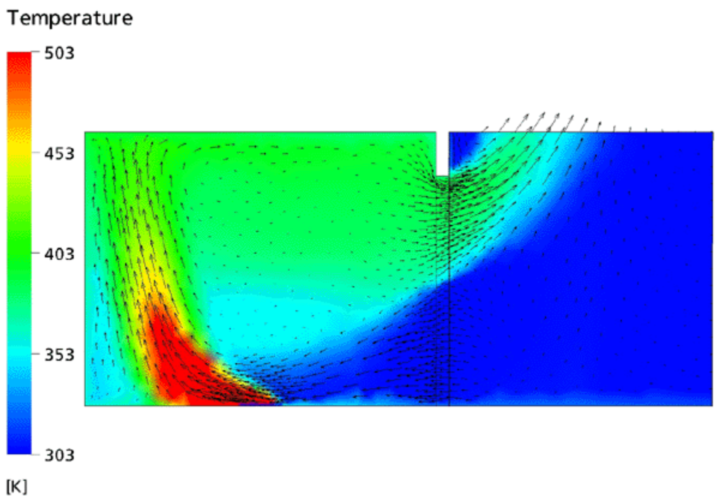


Figure 2.3: Compartment fire simulated with EDM

Figure 2.3 shows the predicted temperatures on a symmetry plane through the centre of the fire and the doorway. The fire simulated with the Eddy Dissipation Model consists of a round methane burner in the centre of the floor.

The EDM is still commonly used as it has the advantage that it can be implemented easily, needs low computational resources and is very robust, which is especially important in industrial applications with complicated geometries, complicated flow fields and large grids. Pathak and Aung [9] used the Eddy Dissipation Model to study the dynamics of a tunnel fire.

In order to uncouple chemical kinetics from turbulence, Peters [10] introduced the laminar Flamelet concept. It describes the interaction of chemistry with turbulence in the limit of fast reactions. The combustion is assumed to occur in thin sheets with inner structure called Flamelets. The turbulent flame itself is treated as an ensemble of laminar Flamelets which are embedded into the flow field. It is based on the assumption that the flame structure of the local mixture fraction is determining the main characteristics of the combustion process. The mixture fraction Z is defined as the mass percentage of fuel in the burned and unburned state in a gaseous mixture. It is equal to 1 in the fuel stream, and 0 in the oxidizer stream. The main advantage of the Flamelet model is that even though detailed information of molecular transport processes and elementary kinetic reactions are included, the numerical resolution of small length and time scales is not necessary. Information of laminar model flames are pre-calculated and stored in a library to reduce computational time. The coupling of laminar chemistry with the fluctuating turbulent flow field is done with a probability density function (pdf), where commonly a β-pdf shape is used.

Instead of solving a transport equation for each species only the transport equation for the Favre mean mixture fraction and its variance are solved. As the mixture fraction is a conserved scalar, its transport equation contains no source term.

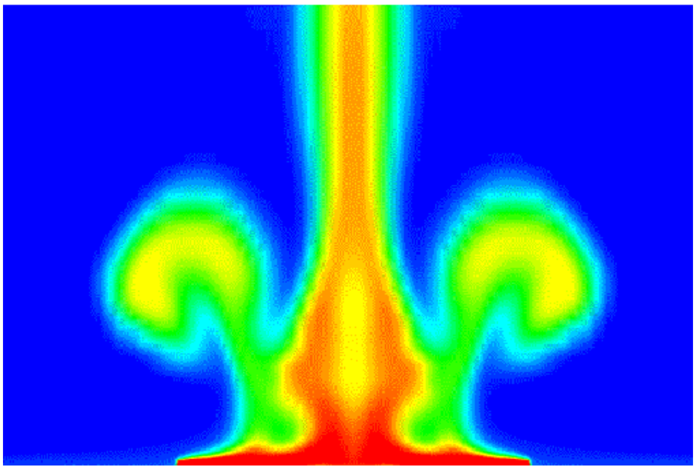


Figure 2.4: Pool fire simulated with laminar Flamelet

Figure 2.4 shows the temperature distribution of a pool fire simulated with the laminar Flamelet model. A liquid fuel distributed at the floor burns with the surrounding air.

2.3 Radiation

A further aspect which might not be neglected when dealing with high temperatures is the modeling of radiation as heat transport mechanism. Two aspects have to be considered, the radiation absorption and release from surfaces like walls and the emission and absorption of radiation heat in the gas. If multiphase flows with particles are considered, there might also be a radiation exchange with the particles.

The different approaches in radiation modeling are dependent on the physical phenomena that have to be modeled, and the modeling and simulation effort. Surface to surface models are only able to predict the radiation of walls. The gas phase and the particles are neglected. This model has the advantage, that the numerical effort is small. The Discrete Ordinate Model (DOM) is also able to account for the fluid and particle radiation up to the modeling of different radiation frequency bands, which also increases the numerical effort.

Further models especially used in HVAC simulations are the Discrete Transfer Model and the Monte Carlo Model. The Discrete Transfer Model is based on tracing the domain by multiple rays leaving from the bounding surfaces. The technique depends upon the discretization of the equation of transfer along rays. The path along a ray is discretized by using the sections formed from breaking the path at element boundaries. These rays have to be traced through the domain in the same way that the photons would be tracked in the Monte Carlo model. Therefore, the model description for both is identical.

The Monte Carlo method simulates the underlying processes that govern the system of interest (that is, the physical interactions between photons and their environment). A photon is selected from a photon source and tracked through the system until its weight falls below some minimum at which point it dies. Each time the photon experiences an event, a surface intersection, scattering or absorption for example, the physical quantities of interest are updated.

Figure 2.5 shows the temperature distribution of a solar radiation through a window (45 degrees).

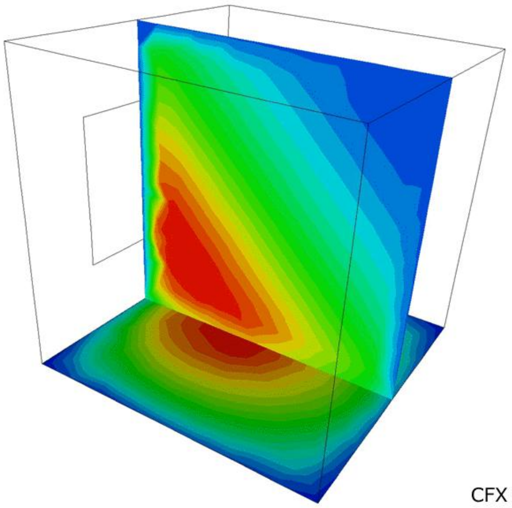


Figure 2.5: Thermal radiation through a window

2.4 Multiphase

Multiphase flow refers to the situation where more than one fluid is present: streams, bubbles, droplets, solid particles and free surface flows. Each fluid may possess its own flow field, or all fluids may share a common flow field. Unlike multi-component flow, the fluids are not mixed on a microscopic scale in multiphase flow. Although these models are rather seldom used, their usage is growing with the growing computational power. Schneeloch et al. [11] examined a sprinkling system above a burning vehicle. Extensive studies have also been performed by Edwards et al. [12] who used engineering simulations to develop effective methods to extinguish fires onboard ships.

They used a transient Lagrangian particle transport model to assess the impact of water spray on fire and fuel, with two-way coupling of mass, momentum, convective heat and radiant heat. In such a simulation the water droplets are modeled as particles that vaporize into the gas phase due to the impact of heat. Thereby the heat of vaporization accounts for the decrease in temperature in the gas phase. In combination with the usage of combustion models, it directly influences the combustion process.

A further application of multiphase flow could be the particle modeling of the released soot, but a detailed modeling of the description of soot formation is difficult. Thus such kind of simulation is not yet industrial standard, and is more an area of research.

2.5 Turbulence

Especially in fire and smoke extraction systems where energy is released buoyancy effects are dominant. These effects are caused by differences in density due to the large temperature differences. The hot combustion gases rise, accumulate beyond the ceiling and spread further. Further on these large gravity forces induce a relatively high velocity field, so that a turbulent flow field can be expected.

The Direct Numerical Simulation (DNS) approach is most accurate, as the instantaneous transport equations are solved directly, but due to computer storage capacities and performance limitations, it is restricted to small Reynolds numbers.

A further approach is the Large Eddy Simulation (LES). Three dimensional, unsteady turbulent motions are directly simulated down to a certain dimension and the small eddies are modeled.

In Reynolds Averaged Navier Stokes equations (RANS) all turbulent structures are modeled, which allows an even coarser mesh than for the LES approach. In most applications the turbulent fluctuations are of less interest than the mean values of the flow field, which makes stationary simulations possible. Thus a variable is splitted into a mean and a fluctuating quantity. When the transport equations are averaged, an unclosed term occurs in the momentum equation, the so called Reynolds Stresses. They are closed by the applied turbulence model, which ranges from a zero-equation Prandtl mixing length model to Reynolds Stress Models where transport equations for all components of the Reynolds stress tensor and the dissipation rate are solved.

The most common turbulence models are the two-equation models as they are a good compromise between computational accuracy and numerical effort. For these models two additional transport equations are solved. The most common turbulence model in the ANSYS community is the SST (shear stress transport) model. To overcome the deficiencies of both, the k-ε model, which behaves well for free shear flows, and the k-ω model, which shows good performance in the near wall region, Menter introduced the SST model [13]. This model blends between the k-ω model near the wall and the k-ε model in the free stream flow. To combine these two models, the differential equations of the k-ε model are transformed to the k-ω model formulation and a blending function is introduced.

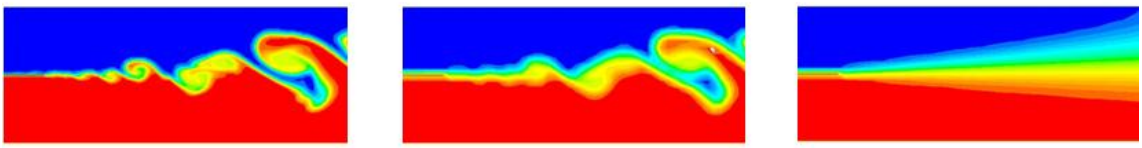


Figure 2.6: Shear layer; from left to right DNS, LES, RANS

The difference between DNS, LES and RANS (SST) simulations can be seen in Figure 2.6, where a shear layer has been calculated

2.6 Conjugate heat transfer between fluids and solids

Conjugate Heat Transfer (CHT) domains are solid domains that model heat transfer. Within solid domains, the conservation of energy equation can account for heat transport due to solid motion, conduction, and volumetric heat sources. Especially if the thermal stability of a building has to be ensured, it is necessary to also account for the solid bodies.

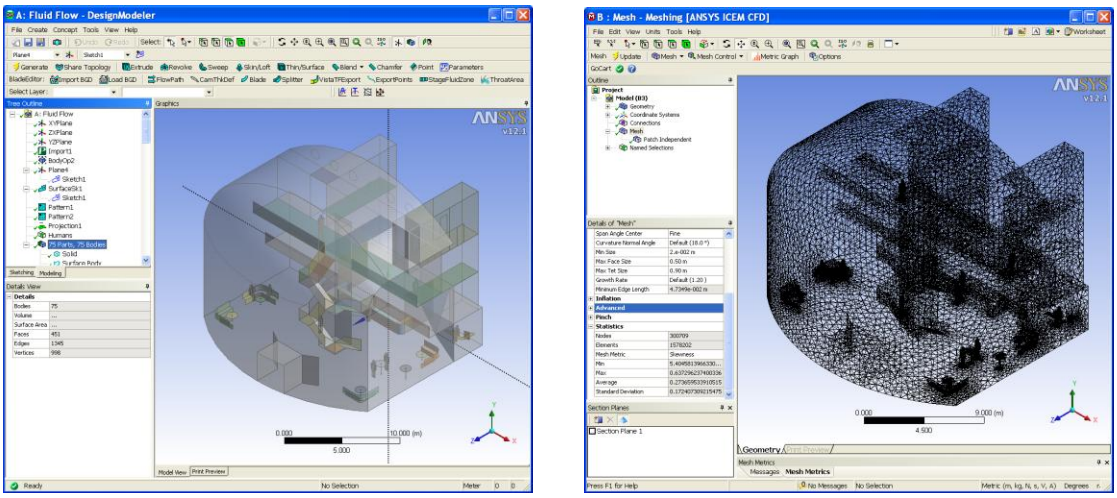
In order to correctly calculate the temperature distribution, it is important to consider the heat transfer between walls and fluid, which results from thermal conduction, radiation and convection as the dominant mechanism. For that a realistic boundary conditions has to be set to insure the temperature adjustment in the solid (e.g. the ceiling). Unrealistic boundary conditions may lead to wrong temperatures in the smoke what in turn results in a wrong prediction of smoke distribution.

3 ADDITIONAL TOOLS

To successfully setup and perform a simulation and to evaluate the results, ANSYS offers some helpful tools, which are all integrated in the ANSYS Workbench platform. The project schematic view ties together the entire simulation process, guiding the user every step of the way. Even complex multiphysics analyses can be performed with drag-and-drop simplicity. Rather than offer a simple list of files, the project schematic presents a comprehensive view of the entire analysis project in flow chart form in which explicit data relationships are readily apparent. Some major components of the Workbench platform are presented in the following.

3.1 Preprocessing – DesignModeler and Meshing

All engineering simulation starts with geometry to represent the design, be it a solid component for a structural analysis or the air volume for a fluid or electromagnetic study. The engineer either has geometry that has been produced in a CAD (computer-aided design) system or builds the geometry from scratch. The ANSYS Design Modeler is a gateway to geometry handling for an ANSYS analysis.



a) DesignModeler b) Meshing

Figure 3.1: Geometry preparation and meshing of an atrium

Especially in safety engineering simulations, the geometry includes details not needed for simulation. Simulating such a fully detailed model will increase solver run times. It can be more efficient to spend a short time removing these details to reduce the total run time by hours or days.

The quality of the CFD results is strongly dependent on the mesh quality. Mesh generation is one of the most ciritcal aspects of engineering simulation. Too many cells may result in long solver runs and too few may result in inaccurate results. ANSYS Meshing allows the user to find the balance and get the right mesh for their simulaiton in the most automated way possible.

As can be seen in Figure 3.1.a and b, where the DesignModeler and ANSYS Meshing graphical user interfaces (GUI) are shown, the layout of both is the same. This makes it easier for the user to get used to a new system, and to be quickly able to generate results.

3.2 Fire Protection module

Most CFD codes are designed to cover a broad range of applications. So they are not adjusted for special application. For fire engineering simulations, ANSYS Fluent offers a special Fire Protection module (see Figure 3.2). With this module, the user is able to geometrically define the fire source. It is not necessary to define the position already in the geometry and mesh generation. The fire source may also change its size depending on time. Smoke and heat release may be imported as text data, which makes it easy to use measurement data. This application fitted module makes it easy to use and the full capability of an in depth CFD tool is still available.

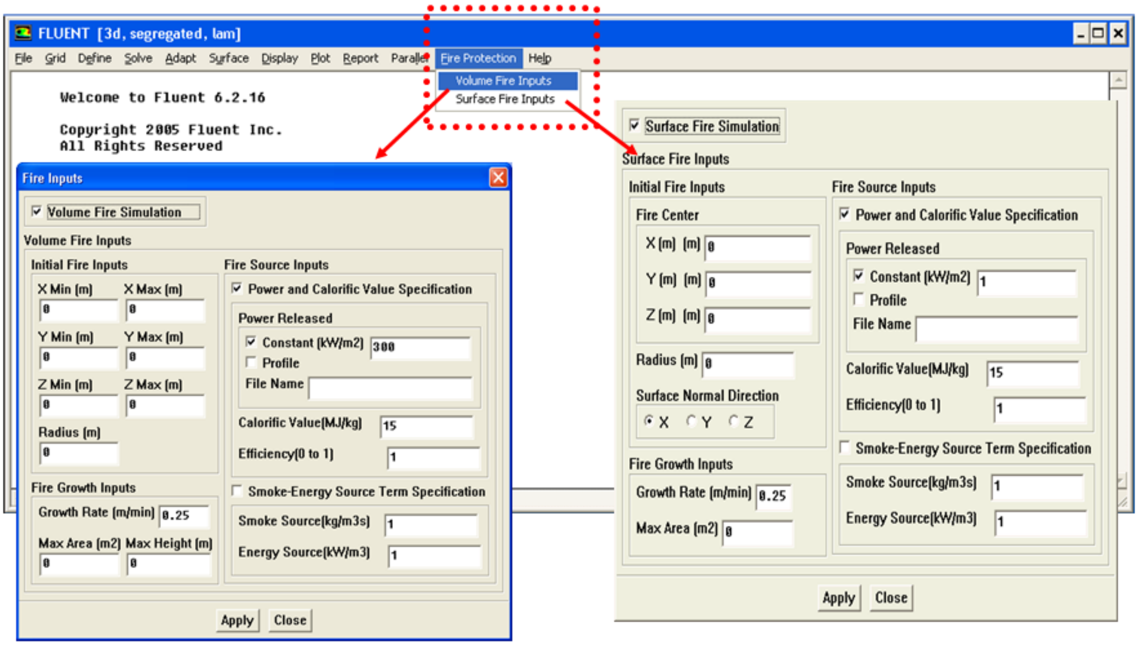


Figure 3.2: Fluent Fire Protection module

In ANSYS CFX offers the possibility to adjust the GUI layout to specific needs of individual customers or applications. Users may create their own extensions to the GUI and define additional panels that contain their application- or company specific terminology to clarify the precise input required to ensure established best practices are followed.

3.3 Postprocessing

CFD simulations don't end with the fluid flow prediction. To benefit from the prediction requires post-processing that gives users complete insight into their fluid dynamics simulation results. ANSYS CFD-Post (see Figure 3.3) is the common post-processor for all ANSYS fluid dynamics products, but may also read in mechanical results to be able to directly evaluate e.g. FSI (fluid structure interaction) simulations.

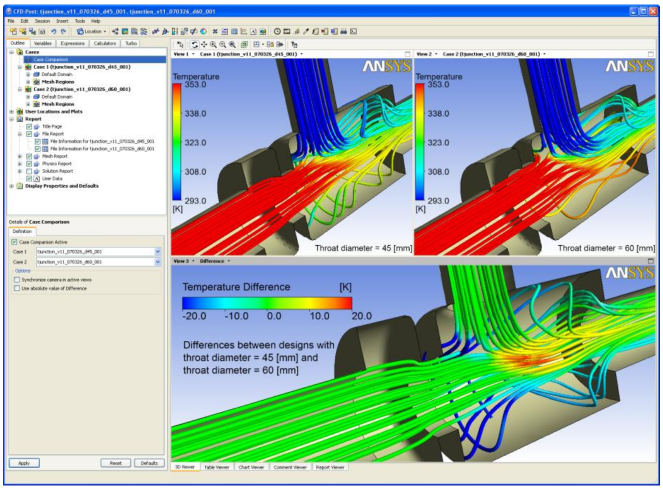


Figure 3.3: ANSYS CFD-Post

Beside the common features a Post processor should provide (like streamlines, vector plots, iso-surfaces etc.), it offers some additional helpful tools. For example multiple solution datasets may be loaded simultaneously, easing the comparison of different design alternatives or operating conditions. Differences between two results can be computed and analyzed. All images can be saved in standard 2-D formats, but it is also possible to write 3-D image files that anybody can view with a freely distributable 3-D viewer. Each session includes a standard template for report generation, which allows the user to customize the content with user-defined text, images, charts or tables.

4 FURTHER SIMULATION TOOLS

Sources, locations and impact of fires are various. ANSYS does not only offer CFD tools to simulate smoke spreading and so on, but also tools that range beyond CFD applications.

Fires may be caused by overheating or sparking in electrical systems. These aspects of system design maybe studied through a combination of circuit simulation technology and electromagnetic field solver technology. ANSYS Simplorer provides a circuit and system simulation environment which can be applied to this type of problems. Simplorer provides a basis for a complete system simulation as shown in Figure 4.1 combining electromagnetic field solvers and thermal solvers to study the electro-thermal problem.

Detailed electromagnetic analysis of system components like motors, transformers, wires etc. can be performed with ANSYS Maxwell, which applies Finite Element solvers or magnetic and/or electric field problems. In Maxwell, the electric field strength can be calculated, leading to an analysis of insulation breakdown which can lead to sparking. Maxwell can also calculate the power loss due to the interactions of electric and magnetic field in such components.

The power loss calculated in Maxwell can then be coupled to a thermal solver, such as ANSYS CFD, in order to calculate the resulting temperatures, and therefore the risk of product failure and/or ignition.

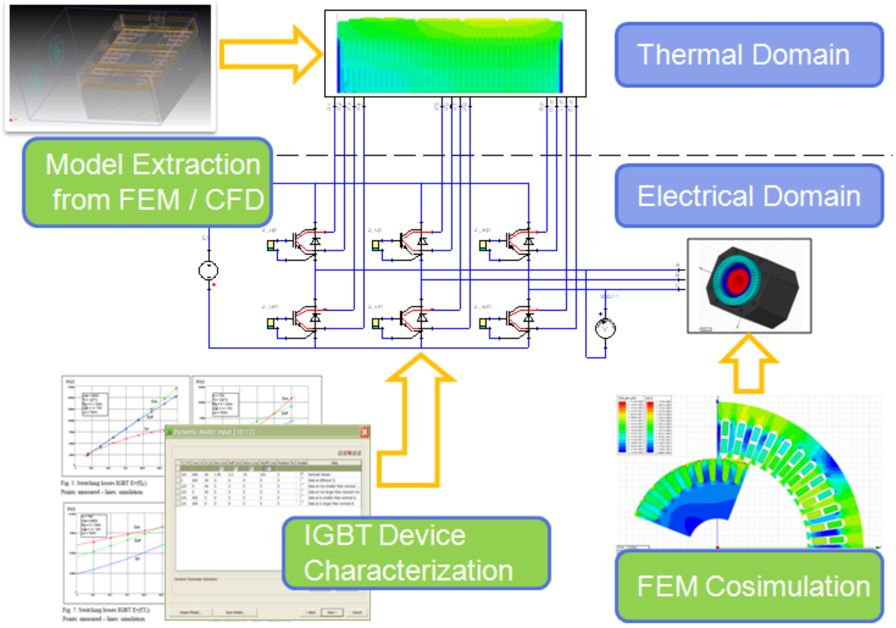


Figure 4.1: Electrothermal simulation of electronic and electromagnetic components – analysis of overheating and electrical insulation breakdown using ANSYS Simplorer, ANSYS Maxwell and ANSYS Icepack

Fires and shock waves may also be caused by explosions which can be simulated with ANSYS AUTODYN. ANSYS AUTODYN is a uniquely versatile explicit analysis tool for modeling the nonlinear dynamics of solids, fluids, gas and their interaction. It offers a finite element (FE) solver for computational structural dynamics and a finite volume solver for fast transient computational fluid dynamics. Mesh free particle solvers may calculate high velocities, large deformation and fragmentation.

The most common fire and safety application beside the CFD simulations is the structural fire response and collapse analyses with ANSYS Structural Mechanics Solutions. This method is based on a finite element solver. Benes et al. [14] numerically examined the structural integrity of multi-story buildings under fire using a non-linear solution with spread of plasticity, large strain, large deflections and temperature linear distribution on elements. Within ANSYS Workbench, a CFD simulation can easily be coupled with a mechanical (e.g. static structural or thermal) analysis, either one way, where the temperature and pressure loads are transferred to the mechanical analysis, or even fully coupled, where the mechanical analyses exchange the data with CFD. Thus the temperature distribution within a fire can be used as boundary condition in a structural analysis.

5 CONCLUSIONS

In fire and safety applications, several physical effects play a major role. The corresponding CFD models have been outlined briefly. The virtual building in a CFD simulation places no restrictions to the complexity of the geometry model, allows the effects of different origins of the fire to be assessed, and permits different scenarios to be examined in what-if studies, so that the best solution for safety can be identified.

CFD already plays a major role in fire & safety applications; however other simulation technologies are also growing in importance. For example, structural behavior can be simulated so that buildings can be designed to better cope with thermo-mechanical deformations induced in the harsh environment of a fire; material behavior can be modeled to include complex and non-linear properties; the effect of defects or overheating in electrical systems can be assessed, as they may serve as the source of a fire; and explosions causing significant damage to structures may also be simulated.

6 ACKNOWLEDGEMENTS

The authors would like to thank the colleagues of ANSYS, especially L. Voss, T. Marchal, H. Forkel who provided pictures and information.

REFERENCES

[1] World Fire Statistics, The Geneva Association (2008)
http://www.genevaassociation.org/PDF/WFSC/GA2008-FIRE24.pdf

[2] NASA images by Reto Stockli and Jesse Allen using data courtesy the MODIS Land Science Team at NASA Goddard Space Flight Center.
http://earthobservatory.nasa.gov/GlobalMaps/view.php?d1=MOD14A1_M_FIRE#

[3] B.E. Launder, D.B. Spalding, Mathematical Models of Turbulence, *Academic Press, London* (1972)

[4] ANSYS User Manual 2009

[5] O. Zavila, CFD Simulation of Motor-Car Fire in the Road Tunnel Valik, European Built Environment CAE Conference (2008)

[6] A. Horvat, Y. Sinai, Computational Prediction of Fire Spread to a Solid Material with ANSYS CFX, European Built Environment CAE Conference (2008)

[7] B.F. Magnussen, B.H. Hjertager, On Mathematical Modeling of Turbulent Combustion with Special Emphasis on Soot Formation and Combustion, *Symposium (International) on Combustion*, **16**, pp. 719-729 (1976)

[8] T. Poinsot, D. Veynante, Theoretical and Numerical Combusiton, Edwads, Inc., Philadelphia (2001)

[9] K. Pathak, K. Aung, Numerical Simulations of Dynamics of a Tunnel Fire, *Proc. of HT-FED04*, AMSE Heat Transfer/Fluids Engineering Summer Conference (2004)

[10] N. Peters, Laminar Diffusion Flamelet Models in Non-Premixed Turbulent Combustion, *Prog. Energy Combustion Science*, **10**, pp. 319-339

[11] E. Schneeloch, M. Adler, A. Schälin, Transient Burning of a Vehicle as a Leading Edge Example of CFD Simulations in Safety and Environment Engineering, *ECCOMAS* (2004)

[12] M. Edwards, M. Smerdon, Y. Sinai, C. Staples, Fighting Fire with Simulation, *ANSYS Advantage*, **Vol. III**, Issue 1, pp. 44-45 (2009)

[13] F.R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA-Journal*, **32(8)**, pp. 1598-1605

[14] M. Benes, F. Wald, Z. Sokol, H.E. Pascu, Numerical study to structural integrity of multi-story buildings under fire, *in Eurosteel 2002*, Coimbra, pp. 1401-1411 (2002)