Combustion Modeling

A large percent of worldwide energy conversion processes are based on combustion. Software from ANSYS offers a wide range of user-friendly models to simulate the interactions between flow, turbulent mixing, heat release and chemical reactions.

Turbulent Combustion
Because chemical kinetics are highly nonlinear, turbulence–chemistry interactions should be taken into account to accurately predict turbulent reacting flows. Fluid dynamics software from ANSYS offers a wide range of turbulence–chemistry interaction models of various complexities suitable for almost every turbulent reacting flow application.

Fast Chemistry
The eddy dissipation model (EDM), in combination with the finite rate chemistry model, allows accurate simulation of the heat release and the distribution of the main chemical species for many industrial applications, such as gas turbines, internal combustion engines, furnaces and thrusters. The EDM can be used for all types of combustion as well as in combination with advanced scale-resolving turbulence and multiphase models.

Non-Premixed Combustion
With the non-premixed (or diffusion flame) model, the turbulence–chemistry interaction is described using an assumed shape of probability density function (PDF). The thermodynamic state of the fluid, related to the mixture fraction and the chemistry is calculated either from equilibrium or by using a tabulated steady flamelet approach. The non-premixed model offers a robust and fast solution for any turbulent reacting flow that is at or near chemical equilibrium. This model is often used for gas turbine, process furnace and IC engine applications.
Premixed and Partial Premixed Combustion
For a premixed flame, an additional transport equation is solved for the reaction progress variable, flame surface density or distance to the flame front. The solution of this variable predicts the position of the flame front. For closure of the source term, ANSYS offers different proven correlations related to the turbulent burning velocity. These correlations provide the best practical method to predict turbulent premixed flames. In combination with the equilibrium or flamelet model, this approach is applicable to partially premixed and fully premixed combustion.

Laminar Combustion and Stiff-Chemistry Systems
ANSYS® CFD software is capable of incorporating simple and complex reaction mechanisms to model reacting flows. Special solution algorithms compute numerically stiff chemical kinetic equations efficiently and robustly. It is also possible to import reaction mechanisms in CHEMKIN® format.

PDF transport and eddy dissipation concept (EDC) models incorporate detailed chemical kinetics in general turbulent reacting flows for which it cannot be assumed that the chemistry is close to equilibrium conditions such as flame ignition and extinction, and slowly forming pollutant species. In situ adaptive tabulation (ISAT), chemistry agglomeration and mechanism reduction are available to make detailed chemistry calculation more efficient and affordable.

Software from ANSYS also enables use of surface reactions as they occur, for example, in catalytic converters, chemical vapor deposition (CVD) or fuel cells. A stiff-surface chemistry solver is available to handle an arbitrary complex surface chemistry mechanism. It is possible to import surface reaction mechanisms in CHEMKIN format.

All combustion models can be used with most of the turbulence models including RANS, LES and hybrid turbulence models such as DES and SAS.

Pollutant Formation Model
Pollutant species, such as NOx, SOx or soot, typically can be calculated by post-processing a combusting flow solution. ANSYS offers several pathways to predict NOx formation in gas, oil and coal flames using thermal, fuel and prompt NOx mechanisms and to predict NOx reduction via re-burning or selective non-catalytic reduction (SNCR). One- and two-step soot models are available; soot radiation can be included in the simulation.

A general capability is available to solve any arbitrary pollutant chemistry mechanism as post-processing on a frozen flow and temperature field.
Multiphase (Fluid and Solid) Fuels

In many applications, fuel is fed into the combustion chamber as a fluid or a solid. Examples are gasoline or diesel for internal combustion engines or pulverized coal for power plants. In these cases ANSYS provides a variety of multiphase models that are fully compatible with the combustion models available in software from ANSYS.

Liquid Fuels

For liquid spray fuels, the common assumption is that the liquid can be described with a Euler–Lagrange or Euler–Euler model, and that vaporization is complete before the actual combustion process starts. Once the vapor is in the gas phase, any of the ANSYS CFD combustion models can be used to simulate the reactions.

Solid Fuels

Combusting solid fuels generally undergo moisture evaporation and devolatilization followed by combustion of volatiles in the gaseous phase and heterogeneous gas–solid reaction of the char. ANSYS offers several devolatilization and char combustion models that have been extensively tested on real applications. A general particle surface reaction model also is available to accommodate any type of heterogeneous reactions to model applications, such as sorbent injections and gasification.

The ANSYS Combustion Team

The ANSYS combustion team is actively engaged in high-level research and development projects and interacts directly with leaders in the combustion research community. A long-standing partnership with Robert J. Kee, George R. Brown Distinguished Professor of Engineering at the Colorado School of Mines and principal architect and developer of CHEMKIN software, provides knowledge that assists in simulating reacting and combusting flow. Interaction with Stephen B. Pope, professor at Sibley School of Mechanical and Aerospace Engineering at Cornell University, a leader in combustion flow modeling and one of the originators of the PDF transport method and ISAT, has delivered expertise in those areas. Long-term cooperation with Norbert Peters, emeritus at the Institute for Combustion Technology at RWTH Aachen University, provides insight into many aspects of combustion technology, including flamelet modeling. Expertise gained from these leaders and many others engaged in combusting and reacting flow research is incorporated into ANSYS combustion technology.

Industry-tested experience by a team focused on continued enhancements ensures that ANSYS combustion models remain at the leading edge of the technology and provides the highest quality of model formulation, consistency of implementation and careful validation across a wide range of industrial applications.
**Industry Solutions**

Simulation of reacting flow in a gas turbine combustor using scale adaptive simulation

*Courtesy German Aerospace Center (DLR), Institute of Combustion Technology*

**The ANSYS Advantage**

With the unequalled depth and unparalleled breadth of ANSYS engineering simulation solutions, companies are transforming their leading-edge design concepts into innovative products and processes that work. Today, almost all the top 100 industrial companies on the “FORTUNE Global 500” invest in engineering simulation as a key strategy to win in a globally competitive environment. They choose ANSYS as their simulation partner, deploying the world’s most comprehensive multiphysics solutions to solve their complex engineering challenges. The engineered scalability of solutions from ANSYS delivers the flexibility customers need, within an architecture that is adaptable to the processes and design systems of their choice. No wonder the world’s most successful companies turn to ANSYS — with a track record of 40 years as the industry leader — for the best in engineering simulation.

**Combustion chamber**

*Courtesy CADFEM Russia*

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

Toll Free U.S.A./Canada:
1.866.267.9724

Toll Free Mexico:
001.866.267.9724

Europe:
44.870.010.4456
eu.sales@ansys.com

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2010 ANSYS, Inc. All Rights Reserved. Printed in U.S.A. MKT0000534 11-10