Something in the Mix

Researchers use the Poincaré plane method to obtain quantitative time scale information from CFD simulations.

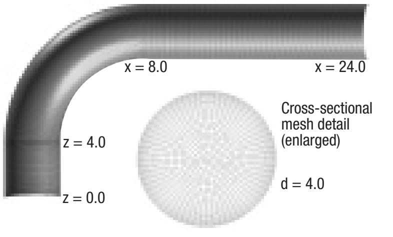
By M.N. Godo, Intelligent Light, New Jersey, U.S.A.

Mixing processes are involved at some level in nearly all chemical manufacturing processes. They are fundamental to the successful operation of combustion-driven systems. Today, many computational fluid dynamics (CFD) practitioners in the chemical process industry are able to use simulation to obtain detailed insight into overall performance of their process equipment. However, it is still difficult to relate CFD data to the effective management and control of a particular process. In addition, the cost of production delays due to sudden, unexpected changes in product quality provides strong motivation to understand the impact and relevance of CFD studies that are focused on these areas.

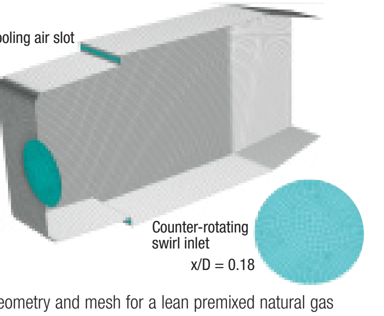
While CFD continues to be more accessible to analysts, managers and operators, problem complexity and sophistication also has increased. Relating flow data, such as mixing time scales to device performance, now is a major challenge. Flow visualization methods, which use iso-surfaces and cutting planes, can be used to help visualize flow topologies in an ad-hoc way. Streamlines and time-dependent streaklines also are effective at elucidating flow patterns. However, these approaches are limited in that they provide very little quantitative information on how flow patterns affect overall performance.

At Intelligent Light in the United States, engineers turn to the Poincaré plane method to obtain quantitative time scale information from CFD simulations. Poincaré planes, placed at various locations within a flow domain, display the time and locations at which streaklines cross these planes. Time scales obtained from these plots relate directly to how effective a mixing tank is or how efficiently a furnace or incinerator can be run. Being able to see time scales within mixers and combustion chambers offers much easier interpretation of the CFD data for everyone involved in the production process. For instance, Poincaré planes showing holes or concentric rings indicate flow regions that are strongly segregated, that is, poorly mixed. In general, this behavior is undesirable; knowing exactly where this occurs in a process vessel is a key step in resolving performance problems.

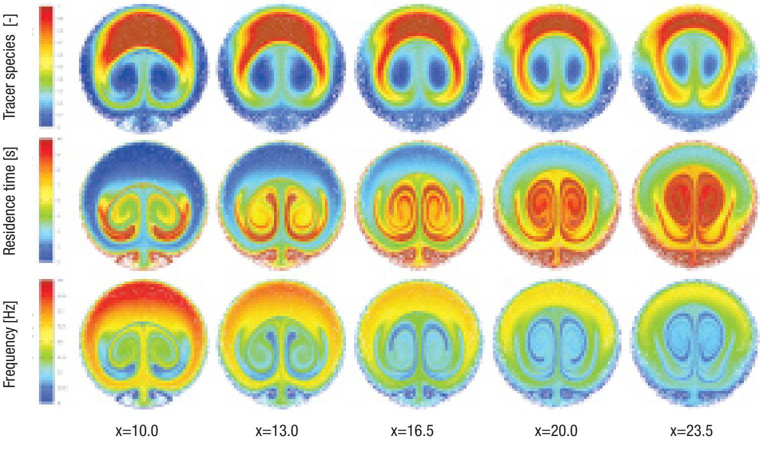
Since its introduction, the Poincaré plane method has been applied to mixing studies [1] and fundamental flow problems [2]. To create Poincaré planes, FIELDVIEW [3], a CFD post-processing tool from Intelligent Light, is used to interpret CFD simulation results that are generated by FLUENT



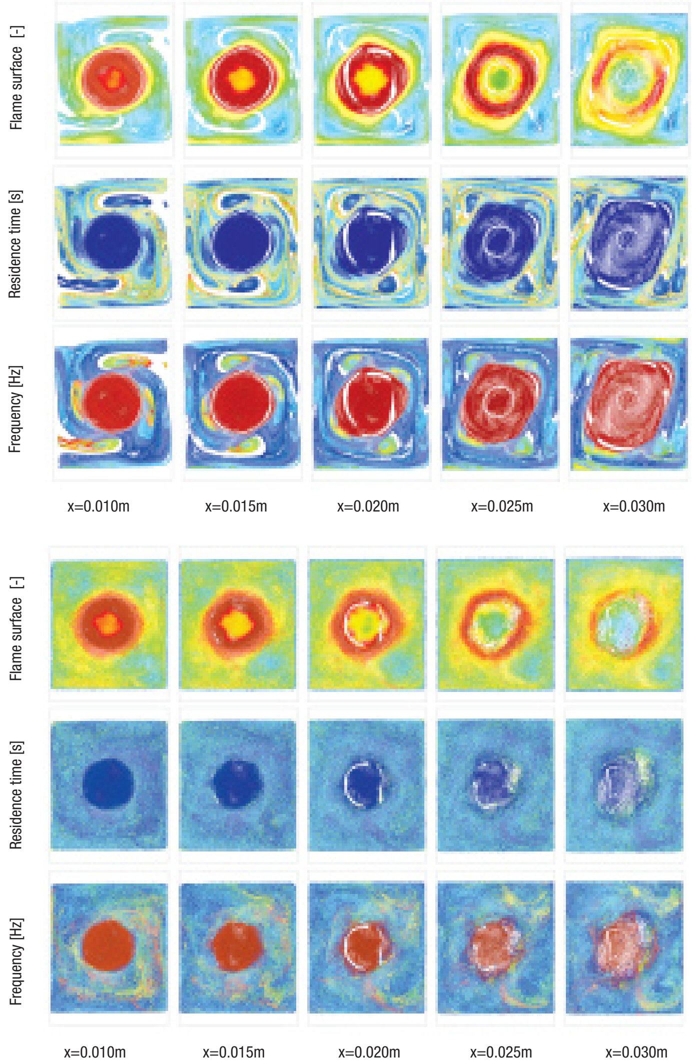
Geometry and mesh for a laminar flow case through a 90-degree bend



Geometry and mesh for a lean premixed natural gas power turbine CFD case, based on the General Electric Aircraft Engines LM6000 engine



A series of Poincaré planes in the downstream section of a 90-degree tube bend; rows are colored by tracer species, residence time and frequency (1/residence time), respectively, and x represents location in the tube after the bend, which ends at x = 8.0.



A series of Poincaré planes used in the analysis of a gas turbine case (specifically the GEAE LM6000) modeled using FLUENT software: (top) RANS turbulence model and (bottom) LES turbulence model

software. FIELDVIEW is able to read data exported directly from FLUENT tools as well as from the ANSYS CFX product. Using the velocity field information from the CFD simulations, FIELDVIEW calculates the large number of streaklines necessary to obtain accurate results for Poincaré planes. Because of the repetitive, quantitative tasks needed, the FIELDVIEW programming language, FVX™, was used to automate streakline trajectory calculations, identify streakline intersections with the Poincaré planes and visualize the final results.

Of two cases studied, the first simulated simple laminar flow through a 90-degree bend. The second case was a fully validated flow calculation for a lean premixed natural gas power turbine, based on the General Electric Aircraft Engines (GEAE) LM6000 engine. A counter-rotating swirl inlet boundary condition was provided directly by GEAE. Both Reynolds-averaged Navier–Stokes (RANS) and large eddy simulation (LES) turbulence models were calculated using FLUENT tools, and GAMBIT software was used to create the meshes for both cases.

For the 90-degree bend case, it was observed that flow details based on either the residence time or frequency are highly structured, and they exhibit significant local differences as the fluid is rolled up by the action of the vortices. Notably, the fluid in the center of the tube, which has a residence time roughly five times that of the flow near the upper section, has a significant impact on mixing effectiveness, as the flow has clearly become quite structured.

Within combustion chambers, a key goal in design assessment is to quantify mixing rates, particularly at time scales that are on the same order of magnitude as the chemical reaction and energy and mass transfer rates. For the RANS turbine case, areas of strong flow isolation are clearly seen near the inlet. In addition, the RANS solution exhibits significantly more structure than the LES solution. Time scales, observed in the Poincaré planes for the RANS case, cover a wide range. This strongly affects the extent of combustion predicted by this simulation. In contrast, Poincaré planes for the LES case show a very high level of chaotic mixing on a fine spatial scale. Apart from the region immediately downstream from the swirl inlet, there were no significant differences in either the residence time or frequency, and the central flame envelope is nearly gone at the farthest downstream plane for the LES case. Mixing time scales for the LES case are expected to provide more realistic predictions of the combustion physics in this particular case. ■

The author would like to express gratitude to Greg Stuckert and Graham Goldin of ANSYS, Inc. for providing the GEAE LM6000 combustion case and for sharing their considerable knowledge of best practices concerning the setup of the partially premixed combustion routines as well as the parameters for the large eddy scale calculations.

References

[1] Zalc, J.; Szalai, E.; Alvarez, M.; Muzzio, J., "Using CFD To Understand Chaotic Mixing in Laminar Stirred Tanks," *American Institute of Chemical Engineers Journal*, 48(10), 2002, pp. 2124–2134.

[2] Shariff, K.; Leonard, A.; Ferziger, J.H., "Dynamical Systems Analysis of Fluid Transport In Time-Periodic Vortex Ring Flows," *Physics of Fluids*, 18(4) , 2006, pp. 047104-1 – 047104-11.

[3] FIELDVIEW, CFD Postprocessor, Version 11, Intelligent Light, Rutherford, NJ, 2006.