

Flow Through a Catalytic Converter

In this example, FLUENT is used to analyze the flow through a catalytic converter, a device used to clean automotive exhaust gases. To evaluate the performance of a catalytic converter, the automotive industry has established standards for several flow parameters. User-defined functions have been created for FLUENT to automate the process of computing these parameters, making quick work of assessing design modifications.

Catalytic converters are used to purify emissions from gasoline and diesel engines. Under certain operating conditions, these engines can release substances that are environmental hazards, such as carbon monoxide (CO), nitrogen oxides (NO_x), and unburned hydrocarbons (also known as volatile organic compounds, or VOCs). To convert these substances to more acceptable ones, catalytic converters force the exhaust gas through a substrate, a ceramic structure that is coated with a metal catalyst, such as palladium or platinum. For gasoline engines, exhaust gases react with these metals, causing CO to be converted to CO₂ and NO_x to be converted to nitrogen and oxygen. VOCs are also burned in the converter, leading to the formation of CO₂ and water. For diesel engines, catalytic converters are primarily used to treat the NO_x compounds.

The nature of the flow in a catalytic converter is very important, and CFD can play a key role in the design of these devices by allowing the engineer to visualize and analyze the

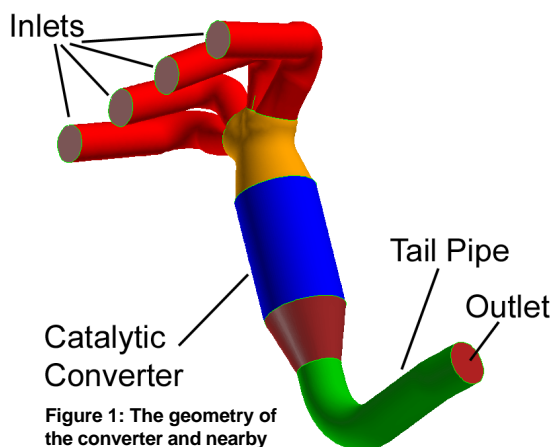


Figure 1: The geometry of the converter and nearby components

exhaust system flow. Key design criteria, such as uniform flow distribution across the substrate, can be easily quantified by CFD, and design performance can be judged via special parameters defined by the automotive manufacturers. To facilitate the extraction of these key design parameters, a special utility has been developed for FLUENT through user-defined functions (UDFs).

To test this utility, a representative geometry of an exhaust system (shown in Figure 1) is used. Heated exhaust gas enters through the four inlets of an exhaust manifold,

passes through the runners, and enters the substrate inside the catalytic converter. The substrate is modeled as a porous media in FLUENT, where viscous and inertial losses are specified in both the streamwise and transverse directions. By using the porous media model, the number of cells in the computational mesh can be reduced significantly, since the small geometric details of the substrate do not need to be resolved. After passing through the catalytic converter, the gas exits through the tail pipe. The standard k- ϵ model is used for turbulence, along with the standard wall function treatment. The fluid is assumed to be incompressible air.

Due to the varying complexity of the geometry, a hybrid mesh

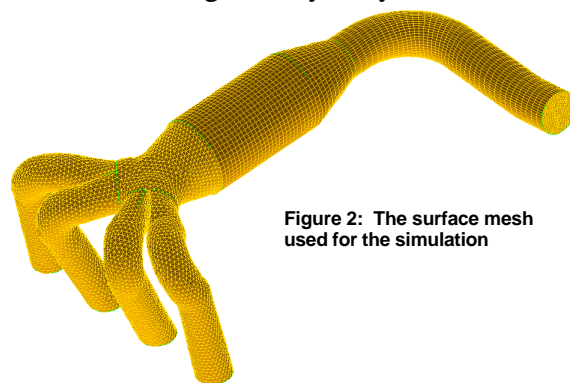


Figure 2: The surface mesh used for the simulation

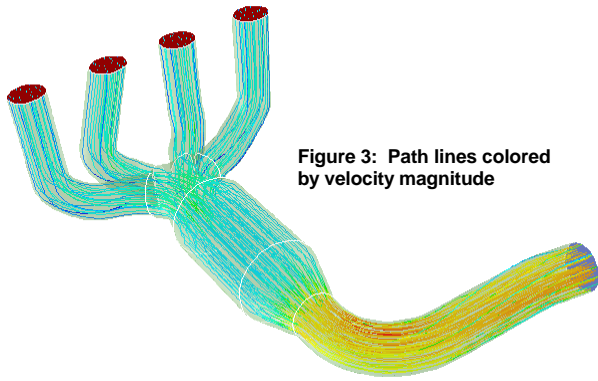


Figure 3: Path lines colored by velocity magnitude

(Figure 2) is chosen to minimize the preprocessing time. Tetrahedral cells are utilized in the exhaust runners, and wedge elements are used in the substrate and tail pipe. The final hybrid mesh consists of a total of 120,000 elements. A mass-flow boundary condition is specified at each of the exhaust runners (inlets), and a constant pressure boundary condition is prescribed at the tail-pipe exit.

Figure 3 shows the path lines throughout the device, colored by velocity magnitude. The flow is axial and uniform in magnitude throughout the catalytic converter. The speed increases as the flow cross section reduces in the tail pipe. Contours of pressure throughout the device are shown in Figure 4. The results illustrate the significant pressure drop

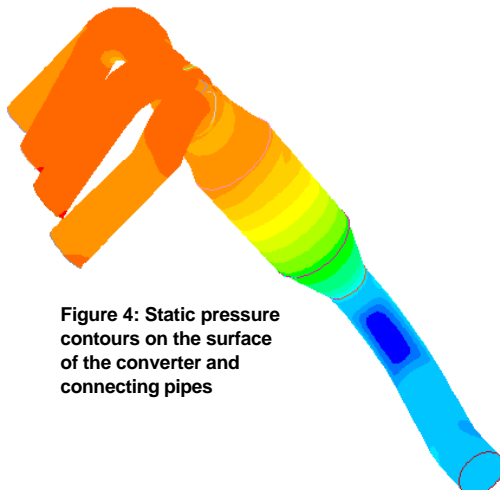


Figure 4: Static pressure contours on the surface of the converter and connecting pipes

occurring in the substrate region, owing to the specified loss factors in the flow and cross-stream directions within the porous media. To simulate the real-world conditions of the substrate, loss terms in the cross-stream direction are set three-

orders of magnitude higher than those specified for the flow direction, to ensure realistic straightening of the flow field. The flow in the substrate is treated as laminar while the remaining regions assume fully turbulent flow conditions.

Specific quantities have been established by automotive manufacturers for analyzing the uniformity characteristics of a catalytic converter. These include pressure loss, eccentricity (location of the maximum velocity within a cross-sectional cut of the substrate, as shown in Figure 5), and velocity ratio (of maximum to mean velocity). Other parameters of interest include the uniformity index, or percentage of cross-sectional area that contains velocity greater than a specified fraction of the maximum velocity, space velocity (product of mean velocity and substrate length), and gamma uniformity index, an integral measure of flow uniformity. These metrics can be used to define specific "pass/fail" criteria; a

catalytic converter design will fail if any of the indices falls outside a preset range. While these measures can be determined using the built-in capabilities in FLUENT, it is preferable to have a post-processing utility which automates the process. A user-defined function (UDF) has been written for this purpose. Using this utility, all of the relevant parameters can be reported to the user with a click of a button.

In summary, FLUENT's ability to simulate the flow in an automotive

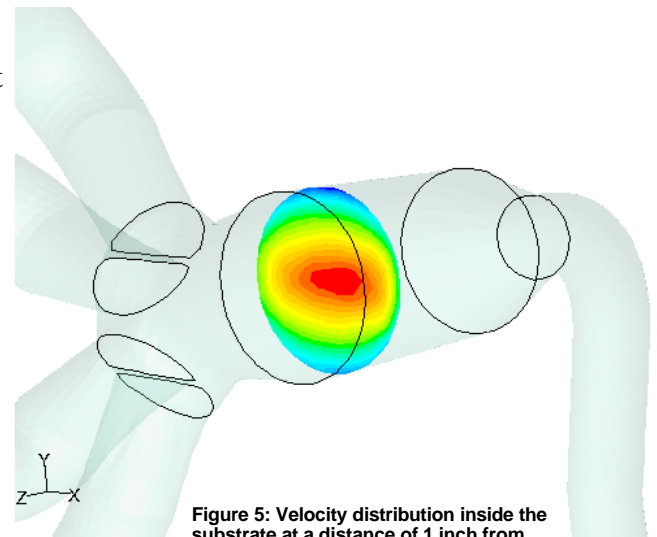


Figure 5: Velocity distribution inside the substrate at a distance of 1 inch from the entry of the gas to the substrate

catalytic converter has been demonstrated through this example. The current study is limited to steady state constant density flow, although a transient analysis using an ideal gas could also be done. A simplified calculation has been performed to test the post-processing features developed through user-defined functions. Using these functions, the flow uniformity characteristics of the catalytic converter can be quickly and easily determined.