Development of internal combustion engines is primarily driven by government regulations, emission requirements and demand for increased fuel efficiency. To address these constraints, engine manufacturers work to refine things like in-cylinder flow, mixture formation and the entire combustion process. However, the increased number of operating parameters associated with these processes coupled with the high cost and time-consuming nature of physical prototyping are prompting manufacturers to integrate engineering simulation technology into their engine design process.

Computational fluid dynamics (CFD) simulation of flow, mixing and combustion reduces or eliminates physical constraints, enabling simulated tests that otherwise might not be possible. Engine manufacturers can easily explore design alternatives to gain extremely detailed insight into these complex physical processes, ultimately helping to make better product development decisions.

Technical requirements for these massive sources of power present engineers with daunting design challenges in meeting the reliability, durability, safety and packaging standards of automotive applications.

In-cylinder engine CFD simulations can be quite challenging. A critical task in developing an accurate representation of an engine’s real-world performance is creating a CFD model that is ready for simulation. To take into account in-motion components, such as the piston and valves, the CFD model has to accommodate both moving and deforming meshes. For most commercial CFD tools, this effort takes anywhere between a couple of days to more than a week of time to complete. However, ANSYS has come up with a customized ANSYS® Workbench™ based analysis system (referred to as ICE system) that reduces the turnaround time for in-cylinder simulations to less than a few hours.

The ICE system is very easy to use. The user first specifies what type of CFD simulation he wants to perform. Then, he inputs engine specifications, such as piston stroke, connecting rod length, valve lifts, RPM, etc. The engine design inputs can also be defined as parameters, allowing the user to perform multiple design-point simulations. Once these inputs are provided, the tool is ready to be fed with the CAD model of the engine, which, as in this case, is usually the flow volume of combustion chamber, intake and exhaust ports, and the solid bodies of all valves. If a solid CAD model of the entire engine is read, the user can easily extract the flow volume using several options available in Workbench’s geometry component, ANSYS DesignModeler™.
DesignModeler can handle relatively dirty CAD models, which is usually the case with real-life engine CADs. The cleanup is made possible via DesignModeler’s diagnostic and repair tools. Once the required CAD starting point is defined, the user identifies geometric entities like cylinder liner, piston, valves and valve-seats. Since the relative motion of the valves and piston in an engine cylinder is complex, the geometry model must be decomposed into specific volumes and bodies so that a suitable mesh can be created on these bodies. Suitable in this context means that the mesh must allow for deformation when the body gets deformed. The tool then automatically decomposes the geometry into desired bodies and assigns names to each body and surface. Several different decomposition options allows the user to successfully handle different types of engine models — for example, racing engines, which operate on a very low squish height, need special treatment; engines with a pre-chamber also require special treatment during geometry decomposition. Once decomposition is completed, the model is ready to be meshed.

In the meshing component, the user specifies some global reference mesh sizes, which are then used to automatically mesh the entire engine model. He can verify the mesh and, if the mesh is not suitable, recreate it with a new group of mesh settings. The mesh is highly accurate, even though it’s generated automatically. It captures the region between the valve and valve seat with sufficient numbers of mesh layers, while the port region is effectively captured using boundary-layered mesh. Though the mesh is now ready for CFD simulation, there is still one big challenge. A number of factors need to be defined in the CFD solver: the motion of various moving bodies, events that control opening/closing valves, corresponding changes in solver parameters, etc. This is a cumbersome task that requires expertise, so it is traditionally performed manually. However, the ICE system automates this step and, therefore, reduces the chance of manual error — which is very common at this stage of model creation.

The basic model is now ready for CFD simulation. It takes only a few hours to reach this point from CAD preparation to final meshing. This includes one or two trials to tune meshing parameters. This is much more efficient than CFD model prep using other commercial tools.

The ICE system goes beyond offering ease of use and fast turnaround. For example, engine models are complex and, therefore, no CFD tool can boast that it handles every complex shape. The ICE system offers enough user-handled points to perform trouble-shooting and then introduce the model back into the system. This flexibility is of great help. Analysts can, for example, mesh a crevice volume separately and then attach it to the remaining engine model. If the engine model is not getting decomposed in the right way, a user can perform out-of-the-way operations and still use the ICE system for meshing and setup. Another significant advantage is its ability to handle parameters, in both design and operating conditions. For
example, if a user wants to study the effect of shape change on a piston bowl to mixing, traditionally he must generate CFD models with several piston shapes. If the user has a parametric CAD model, he can easily generate multiple CFD models for different parametric values without having to do it again and again for each design. The ICE system also provides hooks that users can employ to set up cases for combustion simulation.

The ICE system offers process compression and workflow streamlining. With the demand to cut design turnaround time, tools like this can help significantly reduce how long it takes to complete a simulation.