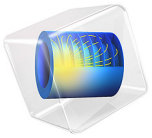

Creating a Fluid Domain Inside a Solid Structure

Introduction

The majority of 3D CAD files include only the geometry of the product to be manufactured. For finite element analysis, however, you find yourself often in a situation where additional geometry is needed, for example, to analyze the flow inside or outside a device. This tutorial, involving the geometry for an exhaust manifold, demonstrates how to create an extra domain for flow analysis after importing and defeaturing a CAD assembly. The following steps are covered:

- Importing a Parasolid® file,
- Deleting a larger detail from the geometry,
- Creating the union of all imported objects needed to generate a domain inside the objects,
- Moving the imported objects using the rigid transform operation to eliminate a gap,
- Using the cap faces operation to create the additional domain,
- Controlling where an operation is inserted in the geometry sequence,
- Finding and removing fillets and holes from the geometry.

For the step-by-step instructions of this tutorial, see the book *Introduction to LiveLink™ for PTC® Creo® Parametric™*.

Application Library path: `LiveLink_for_PTC_Creo_Parametric/Tutorials, _CAD_Import/exhaust_manifold_create_fluid_domain`