SolidWorks Flow Simulation
Local Initial Mesh

SolidWorks Flow Simulation is a computational fluid dynamics (CFD) package offered by SolidWorks. Instead of creating tetrahedrals, like the finite element analysis (FEA) tool SolidWorks Simulation, SolidWorks Flow Simulation creates cells with the finite volume method. This means that we have a computational volume that is divided into blocks. Each neighboring block can either be the same size, 1/8th the size, or 8 times larger. This stepping procedure is referred to binary refinement. Another difference from SolidWorks Simulation is that now we are analyzing environments with solids and fluids. The creation of these blocks, or “cells” as we refer to them in CFD, is independent of the contours of the geometry being analyzed. Therefore we will have cells that are categorized as either fluid, solid, or partial. Partial cells have some of their volume occupied by a solid and the rest filled with a fluid.

The solve time of a flow simulation is highly dependent on the number of cells in the computational domain. When an application arises where the cells need to be highly refined, the easiest procedure to take is to increase the level of mesh refinement in the model. While quick to implement, this process results in overly refined cells in regions that are not of much interest. To maintain the cell density in one region and leave lesser refinement elsewhere, the best approach is to create a local initial mesh. “Initial” in the name refers to the fact that study settings can be set such that a solution adaptive mesh is implemented. In that scenario, the mesh would be refined in between iterations is the solver senses high gradients in the results.

A local initial mesh can be added to either a solid region, fluid region, or a combination of the two. An effective way to define the region of interest is to add a solid body that occupies this space in the 3D model. From there, going back into the flow simulation, add a local initial mesh to that volume. The best way to select the body, as opposed to just the faces on the body, is to select the body in the FeatureManager. After applying refinement settings, exclude the body from the analysis by selecting the checkbox under Component Control. If Local Initial Mesh and/or Component Control are not in your Flow Simulation analysis tree, right-click your study and select Customize Tree, then add whichever categories are necessary.

Multiple local initial meshes can be applied in the study. By applying refinement to select areas the cell count in the computational domain can potentially be reduced by an order of magnitude while maintaining the same level of precision for the results. Local initial meshes are just one of the many methods to refine your model. Make sure to explore the effects of all these parameters. The most efficient way to refine the model is in steps starting from the lower end. Increase the refinement until the defined goals converge. This eliminates hours of unnecessary solve time due to a study that can result from starting with an overly complicated model and working in the reverse direction.