Removing Small Geometric Entities with Repair

Introduction

In this example, the Parasolid® file of a wheel rim contains a few small details, which are not removed during import, since they fall outside the default import tolerance. The step-by-step instructions demonstrate one way to locate and remove these features using the Repair operation. The general workflow is:

- Import the file.
- Create a mesh for quick examination of the geometry.
- Measure the size of the features you would like to remove.
- Repair the object.
- Create a new mesh for comparison.

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

Application Library path: LiveLink_for_SOLIDWORKS/Tutorials,_CAD_Import/
wheel_rim_repair