

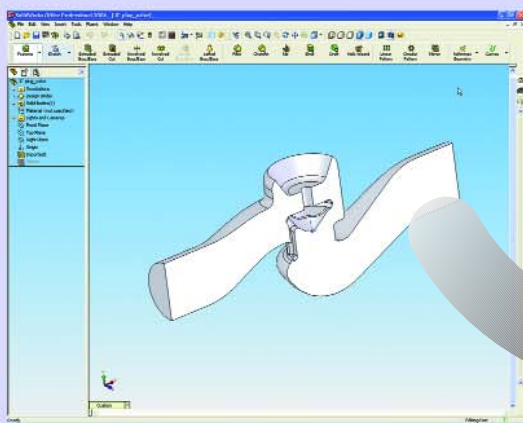
SolidWorks®

Fluent's SolidWorks Connection provides tools within the SolidWorks environment that enable you to prepare and verify that your model is CFD-ready. If there are problems with the geometry model, they can be identified early and resolved. With the new SolidWorks connection tools, you do not have to wait until you are ready to mesh or solve to determine that geometry repairs are needed. For example, the following checks can be done before you complete your SolidWorks session:

- ◆ The Check Interferences tool allows you to verify that no parts interfere with each other within an assembly
- ◆ The Clearance Check tool allows you to perform clearance checks between adjacent parts
- ◆ If your model contains various openings or holes, you can use the SolidWorks Connection to close them by adding capping faces, forming a "water-tight" flow volume
- ◆ The Check Flow Volume tool will verify if a flow volume can be extracted from the geometry model

The concept of the SolidWorks Connection is to identify and resolve technical nuances as the geometry model is being created, eliminating iterative hand-offs between staff or groups within an organization. This enhances productivity.

Once ready, with one click you can load the model directly into FloWizard or GAMBIT to set up your CFD analysis and solve for the flow in or around the geometry. The SolidWorks Connection provides seamless connectivity between your design and engineering processes.



Supported Operating System:

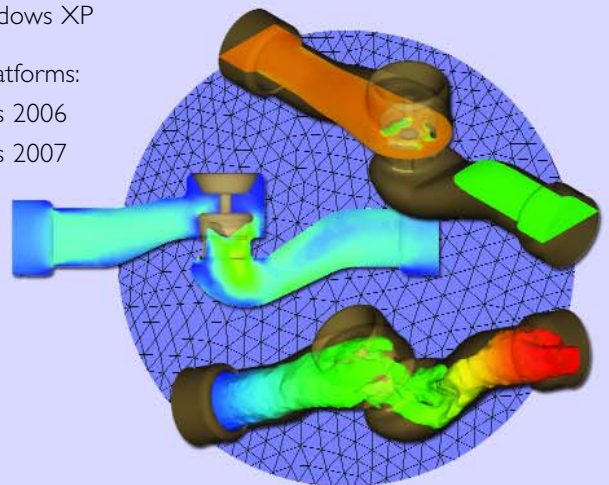
32-bit Windows XP

64-bit Windows XP

Supported Platforms:

SolidWorks 2006

SolidWorks 2007



Contact your local ANSYS/Fluent office to find out more

www.fluent.com • info@fluent.com

Fluent is a wholly owned subsidiary of ANSYS, Inc.



ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners. © 2007 ANSYS, Inc. All rights reserved.