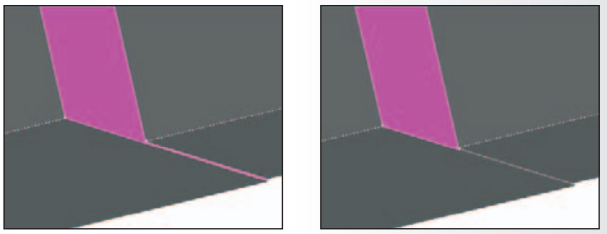
GAMBIT 2.4 Just Around the Corner

By Erling Eklund,
GAMBIT and TGrid Product Manager

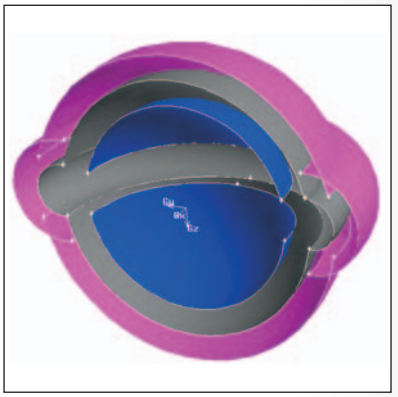
THE UPCOMING RELEASE of GAMBIT, planned for early 2007, continues to take advantage of the new CAD Connection capabilities (see the following article), making it possible to start GAMBIT directly from the CAD package while working on the CAD model.

GAMBIT 2.4 continues to expand on geometry operations with a new face-offset function which allows volumes with finite thickness to be created more easily. Real face- and volume-splits are enhanced to include a split-by-location option. A tolerant option has been introduced to Boolean operations. A new partial-sliver-face removal feature further strengthens the suite of GAMBIT cleanup tools.

In the area of meshing, prism layers are now smoother, linked meshes are easier to manipulate, and size function- and edge-meshing more exact. For example, copy-linked meshes can now be connected for easy conversion from periodic to full domain setup. GAMBIT also makes use of the new TGrid technology for improved tetrahedral meshing quality and stability. ■



A geometry before (left) and after (right) partial-sliver-face cleanup



The new face-offset tool is illustrated above; starting with the original face (gray), inward (blue) and outward (pink) offsets are shown

Using CAD

THERE HAS BEEN A SIGNIFICANT TREND in recent years for companies to bring engineering analysis forward in their design processes. In doing so, it has become increasingly important for these organizations to directly utilize their investment in production CAD geometries. Similarly, the need for CFD providers to offer direct connections to CAD has become critical. Unfortunately, the suitability of many, if not most production CAD models for CFD or CAE analysis remains a problematic issue. The problems are minimized if the geometries:

- are simple assemblies, consisting of prismatic geometry with no fasteners and very few rounds or fillets, for example
- are void of clearances or interference fits, typical of bearings, snap rings, inadvertent interferences (mistakes), or clearances for sliding fits for assembly or welding
- do not contain parts that are not relevant to the CFD solution

Production Models for CFD

By Leroy Stotler and Shane Moeykens, Fluent Inc.

Unless designed specifically for analysis, most production models do not meet the aforementioned criteria. Even so, the advantages of using the latest detailed CAD geometry far outweigh the alternative, which is to recreate an over-simplified model that does not evolve with the production model as the design process moves along. The balance is to understand how to take the production model and quickly make it compatible with CFD products whose solutions can be used to drive the design by providing performance feedback.

Supporting this need, Fluent released V1.1 Connections for Pro/ENGINEER® Wildfire®, UGS NX™, and SolidWorks® in September, 2006. Connections provides three significant benefits. First, tools are included that facilitate the identification of gaps and interferences between parts within an assembly. Gaps and interferences in the geometry model can lead to complications when creating the flow domain and can make the meshing process more difficult, so it is good to identify and fix them in the CAD environment before valuable time is lost. Second, Connections facilitates direct geometry transfer to the CFD software without the need to translate the geometry to any third-party generic CAD format. Finally, Connections helps the CAD user to identify and isolate the fluid region. Whether preparing for a complicated run using GAMBIT and FLUENT or a quick analysis in FloWizard, it is typically easier and more efficient to work with the CAD model directly in the native package instead of dealing with issues in the CFD package. To summarize, the drivers for success can be identified with three easy steps:

1. Have access to the native geometry CAD package or work more closely with the CAD designers
2. Use the integrated Fluent tools to identify interferences and clearances
3. Use the integrated Fluent tools to verify if a flow volume is recognized, or if existing geometry problems might curtail the creation of the flow volume

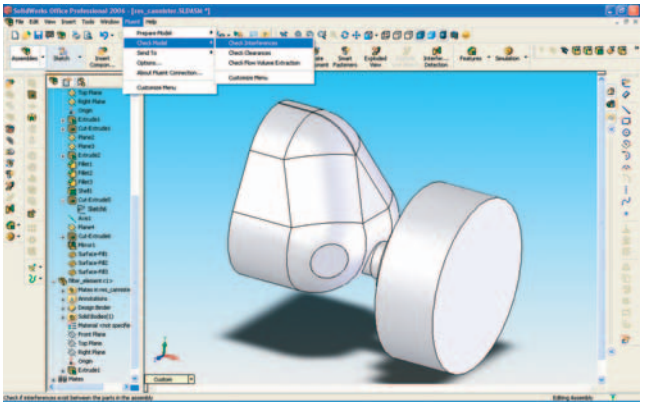


Figure 1: The integrated FLUENT menu in SolidWorks

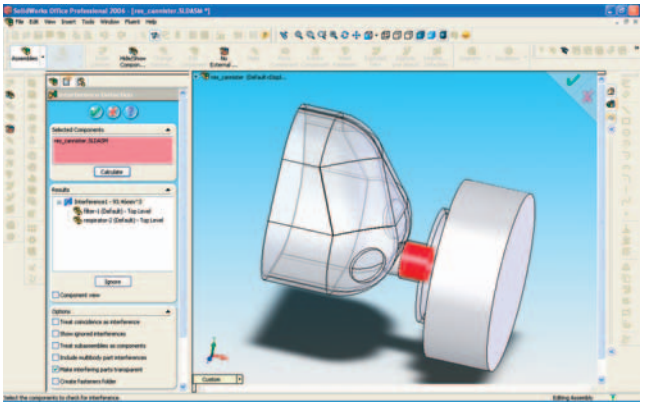


Figure 2: The Interference Detection alert in SolidWorks

As an example, a customer recently expressed an interest in trying the design tool, FloWizard. He began with the traditional process of requesting a neutral file (in a format such as SAT, Parasolid, STEP, or IGES) from the designer. After receiving that file and setting up a FloWizard run, it was determined that the geometry would not mesh. GAMBIT was used to identify the issues, and sliver faces were found as well as volumes that were not connected. The customer then requested the native CAD (in this case, SolidWorks) model from the designer and using the Fluent V1.1 Connections, the CAD model was reviewed and modified in about 15 minutes. It was then sent directly to FloWizard and solved in an additional 20 minutes.

Access to the native CAD geometry

In many cases simple communication between the designer and CFD user can be the key to a successful analysis with either FloWizard or FLUENT. Making sure that the designer is armed with the basic knowledge of what is appropriate for CFD is also beneficial. The integrated Fluent menu in the new V1.1 Connection products is designed to make it easy for analysts and designers alike to determine the CFD-readiness of their CAD geometry.

Identifying and fixing interferences

In Figures 1 and 2, the check interferences tool is applied to help identify potential issues with an assembly. In some cases, an interference is intended, but often it can be the result of a simple mistake. Once an unintended interference is detected, it can be easily fixed in the CAD system using native measuring tools and making a simple dimensional change, as shown in Figures 3 and 4.

Clearances

The clearance checking tool in Fluent Connections allows small clearances between parts within an assembly to be identified. A flow volume must be completely bound by fully connected surfaces to take advantage of Connection's automated flow volume extraction tool. Determining if unconnected parts exist allows any unintentional clearances in the native CAD environment to be dealt with quickly. Figure 5 shows the on-screen diagnostics when searching for clearances of less than 1 mm in the geometry. Two clearances have been detected. The panel displays the clearances and the parts associated with each. By selecting either clearance in the display panel, the clearance itself is displayed on the screen with automated zooming and dimensional brackets.

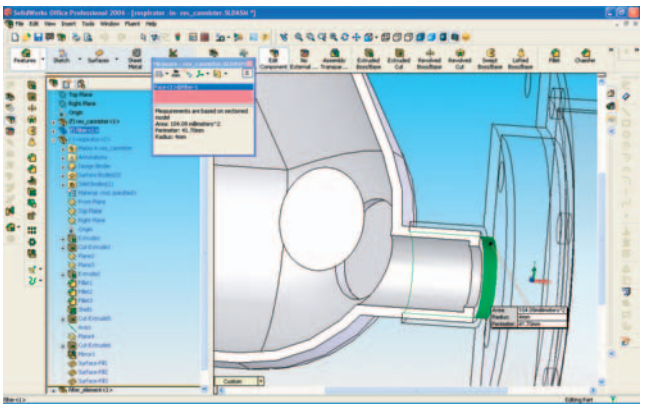


Figure 3: Using the Measuring Tool in SolidWorks to determine the required diameter

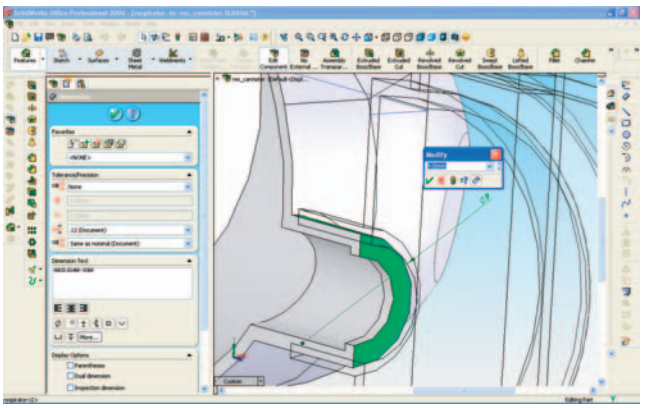


Figure 4: Correcting the diameter to eliminate an interference

It is important to recognize that a clearance will not be displayed between two surfaces of two parts that are connected within an assembly. This is because the clearance checking tool is intended, by definition, to identify gaps between surfaces along the boundary of the flow domain. Thus where parts are connected, there is no concern regarding the watertight integrity of the flow domain. An example of a clearance that will not be detected by the clearance checking tool is shown in Figure 6.

Flow volume determination

Prior to sending a geometry to FloWizard, an important step is to perform the flow volume extraction check, which is available in the integrated Fluent menu within the CAD interface. This step complements the interference check and the clearance check. The number of flow volumes identified in the assembly is displayed on the screen. If no water-tight flow volume is detected, an alert indicates this fact so that the problem can be addressed in the CAD system immediately. Later in FloWizard, the flow volume or volumes will automatically be created in a reliable fashion, because the mechanism for creating the flow volume is precisely the same as that employed by Connection's flow volume extraction check in the CAD system.

One additional note

Because of geometry associativity and the downstream activities that rely on the production CAD model, the need to save a separate CFD version of the assembly should be addressed. The reason, simply, is that changes should be made to the model to make it more efficient for CFD without impacting the downstream deliverables. For example, information in support of Bills of Material or detailed drawings that capture the desired detail not significant to the CFD analysis may need to be preserved. In other words, caution should be used when simplifying the geometry. Keep in mind, however, that it is possible to remove a geometric detail that was assumed to have an insubstantial effect on the flow field, only to find later on that the detail was actually significant. ■

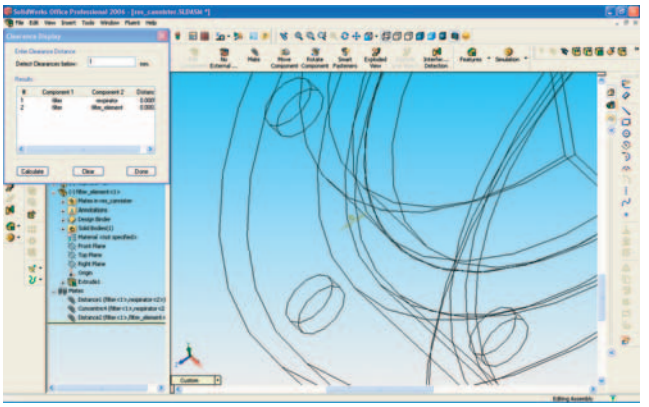


Figure 5: The Clearance Check diagnostic tool in SolidWorks

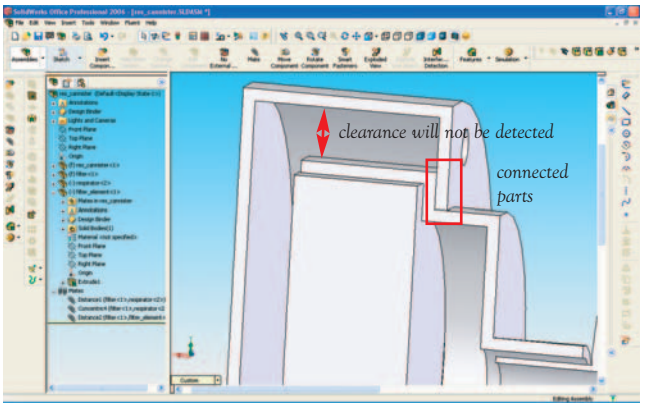


Figure 6: Noting acceptable clearances between connected parts

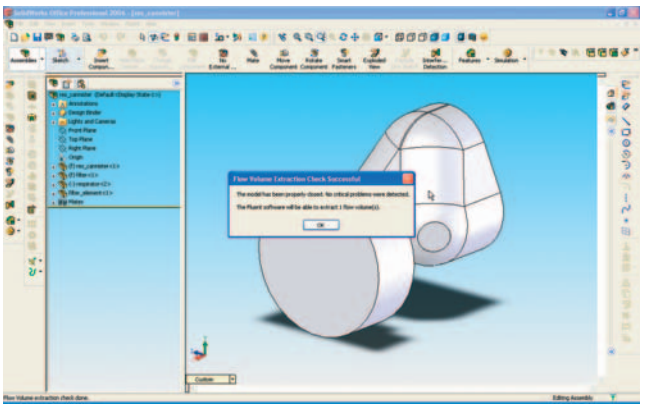


Figure 7: A successful flow volume extraction check indicates that the model is CFD-ready