Suggested Technique for Troubleshooting Incomplete or Incorrect Cross Section Views

When a cross section view cannot be created in Drawing mode, one of the following error messages may appear:

"Cross-section may be incomplete."
"Cross-section could not be created."

During the creation of a cross section view in Drawing mode, Pro/ENGINEER performs an operation on the model analogous to a cut feature. Therefore, if the cross section cutting plane intersects any "incorrect" geometry, the cross section view may not be created successfully. In addition, if the cutting plane passes through a tangency point, unattached edge, or vertex either directly or by function of the model accuracy, the cross section view will not be created.

When a cross section is created in Part or Assembly mode, it is simply a cosmetic which shows where the section lies in the model. Therefore, no error is given when a section is made through the previously mentioned entities. The following steps are recommended upon encountering an unsuccessful cross sectional view in Drawing mode.

As of Release 2000i^2, a new type of cross section can be created by Pro/ENGINEER. Instead of creating a hidden cut in the model to display the cross section, a more "graphic" type of section can be created. The view will look the same as other sections, but the chance of the cross section not being created is slim. Set the drawing setup file option "crossec_type" to "new_style" before creating a cross section view to enable this new functionality.

Procedure

1. Be sure that the design intent of the model is clear by first verifying that no geometry checks exist within the model in areas where the cross section intersects. Retrieve the model and select **Info** to determine whether **Geometry Check** is grayed out or not. If it is available for selection, the information provided in the subsequent menus allows for precise resolutions to geometry issues which could prevent a cross section view from being created.

2. If **Geometry Check** can not be selected, create a cut feature in the model using the exact same placement references and geometry that were used to create the cross section by selecting **Insert > Extrude** and click the **Remove Material** icon. When the cut feature fails, a **"Failure Diagnostics"** window will appear, along with an extensive amount of information concerning which feature and/or part the cut could not be made through. In that case, the cross section should be modified to avoid the highlighted features, most effectively through offsetting from the intersecting edges or points until the cross- section view can be created successfully in Drawing mode.

3. If the problem areas of the model for cross section creation are still in question, a series of feature and/or part suppressions should be performed in the top level model. If the drawing model is an assembly, select **Edit > Component Operations > Suppress** from the ASSEMBLY menu and suppress half of the assembly components. Change back to the drawing and attempt to create the cross sectional view. If the view creates successfully, the troublesome component is not active and will not interfere with the cross section cut. If unsuccessful at creating the view, try suppressing the other half of the assembly and resume the previously suppressed components. Once the problem component is determined, continue diagnostic testing at the part level by selecting **Edit > Feature Operations > Suppress**. After determining which feature is causing the cross sectional failure, modify the cross section so that it does not pass through any edges, vertices, or tangency points that could possibly cause the cross- section to incorrectly intersect this feature.

4. Continue to troubleshoot the failed cross section by indexing, or slightly increasing, the offset position of the cross section within the model. For planar corrections, select **Tools > Model Sectioning**, pick the name of the cross section, click **Edit > Redefine > Dim Values**. For offset corrections, select **Tools > Model Sectioning**, pick the name of the cross section, click **Edit > Redefine > Redefine** and either **Section** or **Scheme**. Minor offsets to the dimensions used to originally create and constrain the cross section should be added. Again, the modified cross section should continually be tested until the cross sectional view in the drawing is created successfully.