



Version 14 : September 2020

Data Exchange Tutorials



Proprietary Notice

This document contains proprietary information of Cambrio Acquisition, LLC (“CAMBRIO”) and is to be used only pursuant to and in conjunction with the license granted to the licensee with respect to the accompanying licensed software from CAMBRIO. Except as expressly permitted in the license, no part of this document may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language or computer language, in any form or by any means, electronic, magnetic, optical, chemical, manual or otherwise, without the prior expressed written permission from CAMBRIO or a duly authorized representative thereof.

It is strongly advised that users carefully review the license in order to understand the rights and obligations related to this licensed software and the accompanying documentation.

Use of the computer software and the user documentation has been provided pursuant to a CAMBRIO licensing agreement.

Copyright © 2021 CAMBRIO. All rights reserved. The Gibbs and GibbsCAM logos, GibbsCAM, Gibbs, Virtual Gibbs, and “Powerfully Simple. Simply Powerful.” are either trademark(s) or registered trademark(s) of CAMBRIO in the United States and/or other countries. All other trademark(s) belong to their respective owners.

Portions of this software and related documentation are copyrighted by and are the property of Siemens Digital Industries Software.

Microsoft, Windows, and the Windows logo are trademarks, or registered trademarks of Microsoft Corporation in the United States and/or other countries.

Contains PTC Creo GRANITE® Interoperability Kernel by PTC Inc. All PTC logos are used under license from PTC Inc., Boston, MA, USA. CAMBRIO is an Independent Software Provider.

Portions of this software © 1994-2021 Dassault Systèmes / Spatial Corp.

Portions of this software © 2001-2021 Geometric Software Solutions Co. Ltd.

Contains Autodesk® RealDWG™ kernel by Autodesk, Inc., © 1998-2021 Autodesk, Inc. All rights reserved.

DMG MORI Models provided in conjunction with GibbsCAM © 2007-2021 DMG Mori Seiki Co., Ltd.

Contains VoluMill™ and VoluTurn™ software by Celeritive Technologies, Inc. © 2007-2021 Celeritive Technologies, Inc. All rights reserved.

This Product includes software developed by the OpenSSL Project for use in the OpenSSL Toolkit (<http://www.openssl.org/>). This Product includes cryptographic software written by Eric Young (eay@cryptsoft.com).

Portions of this software © MachineWorks Ltd.

Portions of this software and related documentation are copyrighted by and are the property of Electronic Data Systems Corporation.

Other portions of GibbsCAM are licensed from GibbsCAM licensors, which may not be listed here.

DATA EXCHANGE TUTORIALS

This chapter demonstrates how to handle bad surface import and how to repair invalid surfaces found in a model. A basic knowledge in Geometry Creation, Advanced CS and solid modeling is helpful before attempting this tutorial. The exercise involves importing an IGES surface file and “healing” or stitching the surfaces to create a solid body.

EXERCISE 1: HEALING

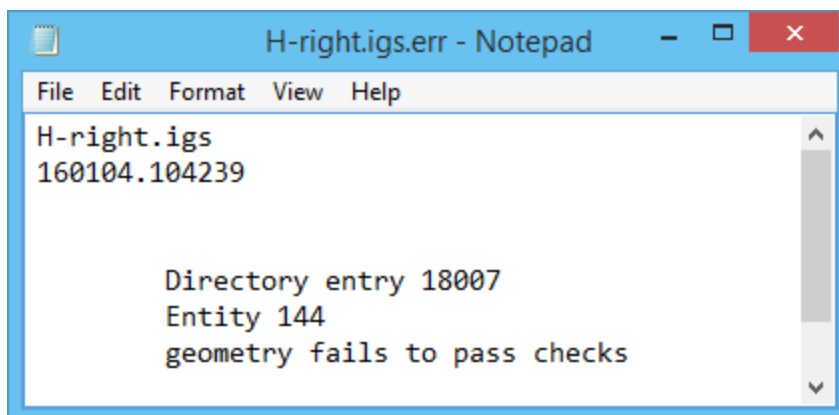
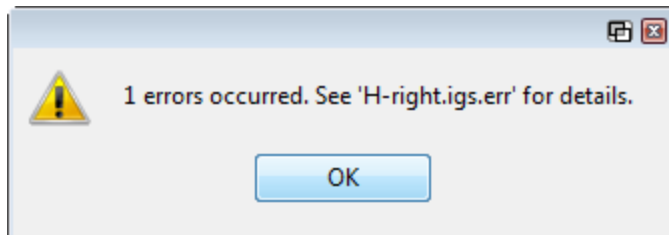
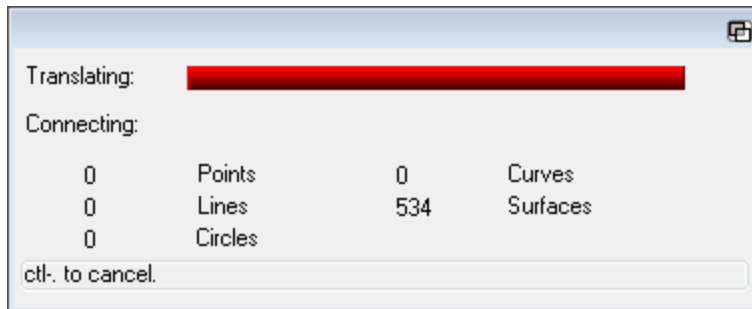
Opening an IGES part

Errors in Surfaces

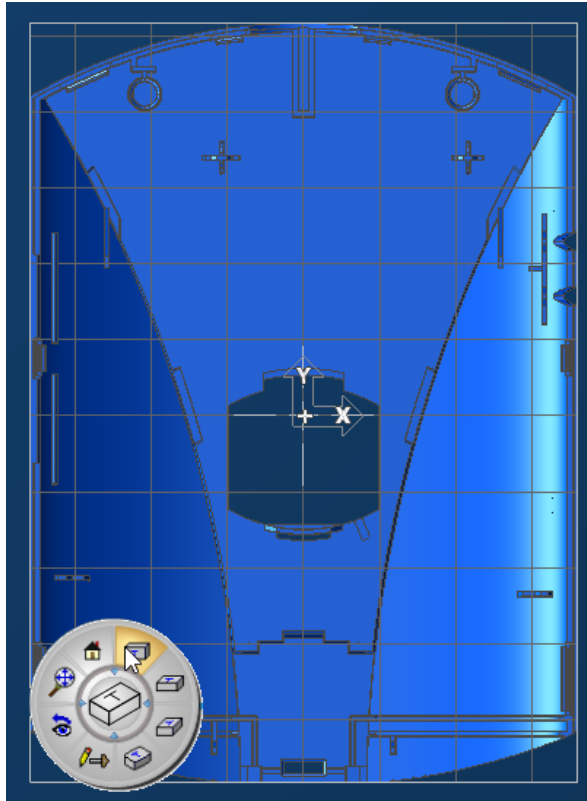
1. Open the IGES file `H-Right.igs` from the `Gibbs sample parts/Solids/Tutorial Parts - Required/SolidSurfacer` folder. If you cannot see this file, change the file type on the `Open part file` dialog to `IGES (*.igs,*.iges)`

Upon importing, you will see a message that an error was encountered in the translation of the IGES surface data. Press the `OK` button. When a valid sheet cannot be created from a surface, an error log is generated. This error log is a text file that contains brief information on the problematic surface. The error log can be found in the original folder of the imported file or in the user data folder (default

`C:\Users\\AppData\Roaming\CAMBRI0\GibbsCAM\).`



When a surface cannot be faceted, it will cause problems with stitching and solidifying the sheets. Therefore, these sheets must be identified, deleted and re-created if necessary.

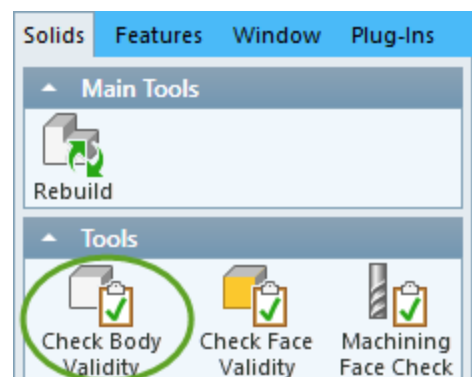


The first course of action is to perform a scan of all of the sheets in the file to check that they are valid entities.

Verifying the Bodies

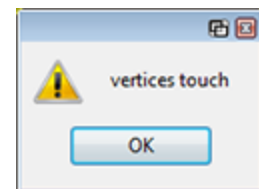
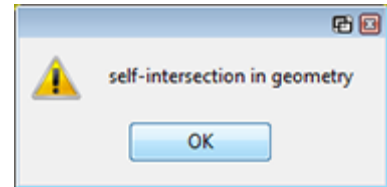
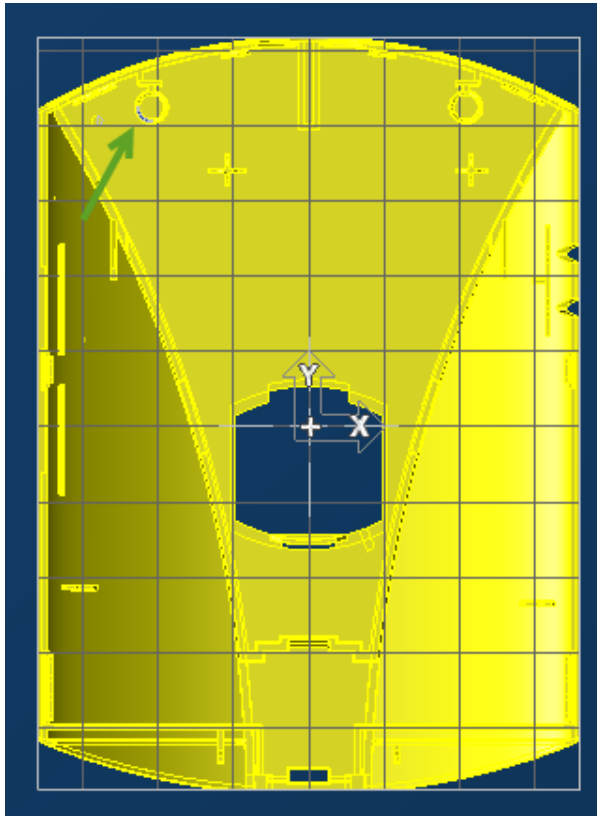
Identifying Bad Surfaces+

1. Select all (**Ctrl+A**) and then select Solids > Tools > Check Body Validity.



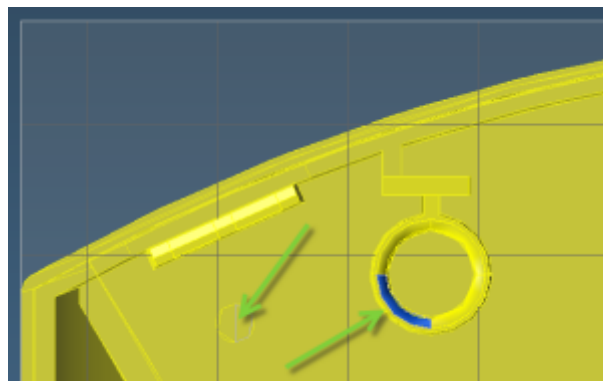
Check Body Validity will deselect any invalid sheets and an error message will appear identifying specific problems.

After performing the Check Body Validity, the following error messages will appear on the screen and a rounded edge in the top left corner of the part should be deselected. Click OK in each case.

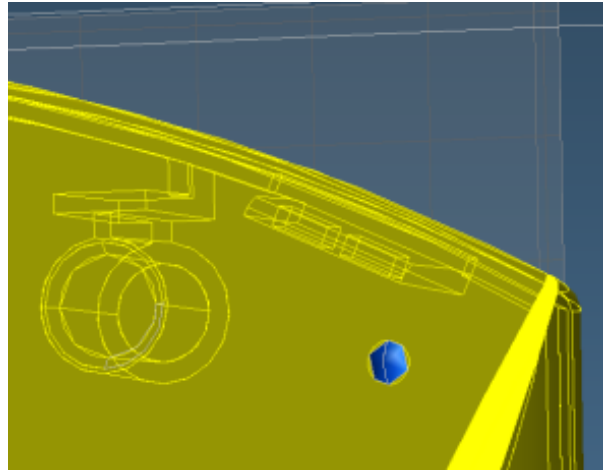


Now that the surfaces have been identified we must delete them. This is often a difficult process because the “bad” sheet cannot be selected by conventional selection methods, i.e. clicking on the sheet. Often times it is necessary to place all of the other sheets into the body bag and then select the workspace area, which will select the surface that cannot be faceted, and then delete it. However, in this case we can simply invert the current selection, and delete the bad sheet.

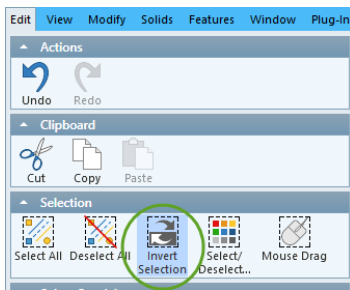
2. Zoom into the part to a blue selected face and what looks like a shadow on the surface.



Switch on Edge selection and look at the reverse side (right-click Top view) and we see that there is a face on the back side.



3. Select Edit >  Invert Selection.

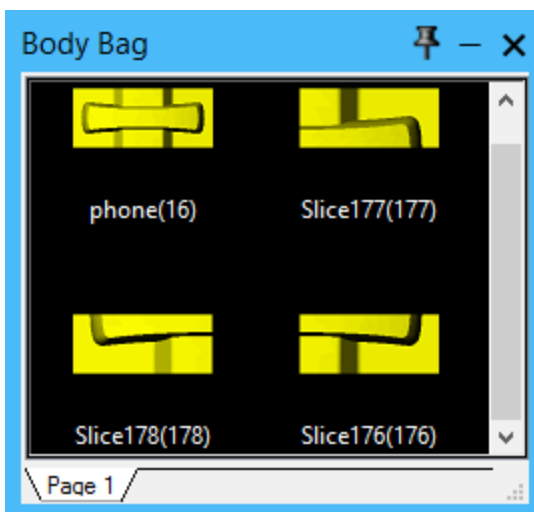


This will cause the invalid sheets that were deselected from the Check Body Validity to become selected and all the other sheets will be deselected.

4. **Delete** the invalid sheets (Hit the Delete button). We will need to recreate these invalid sheets.
5. Select all (**Ctrl+A**) and then select Solids > Tools > Check Body Validity again.

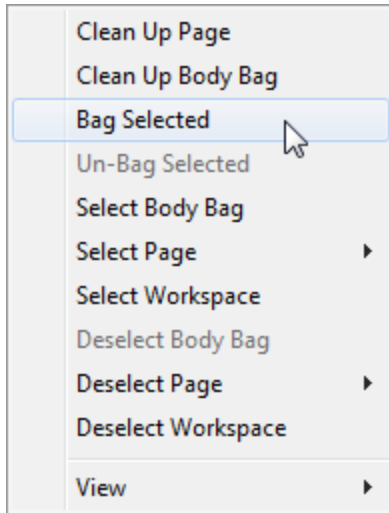
This time no error messages appear, indicating that all of the selected entities are valid.

6. Open the  Body Bag.



The Body Bag title bar context menu contains options for selecting and deselecting the entire workspace and for bagging or unbagging all selected items.

7. **Right-click** the title bar of the **Body Bag** and select **Bag Selected**. This may take some time to load.




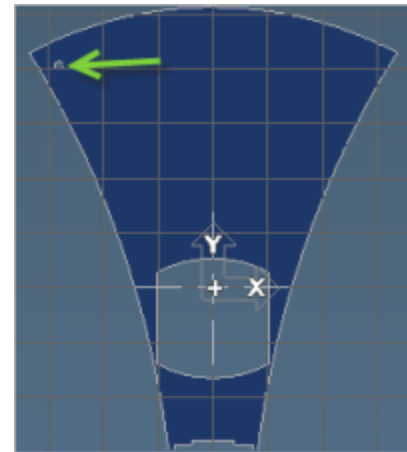
We now need to get the surface that needs work back in the workspace. With all the surfaces in the **Body Bag** it may be difficult to find the correct one. If we know the name of the body or surface we wish to use this is not a problem.

8. **Right-click** the title bar of the **Body Bag** and select **Deselect Body Bag**
9. Again in the right-click title bar menu of the **Body Bag**, choose **View > Show solid ID**.
10. Scroll to **Import_IJES10** and **double-click** it.

This will bring the selected sheet back into the workspace.

Notice the hole in the upper left corner of this sheet. We need to fill this hole from this surface. To accomplish this, we will

use the **Trim/Untrim** function in the  **Surface Modeling** palette.

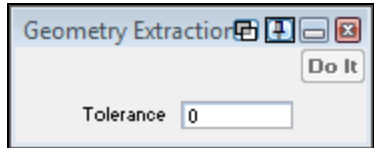


Repairing Bad Sheets

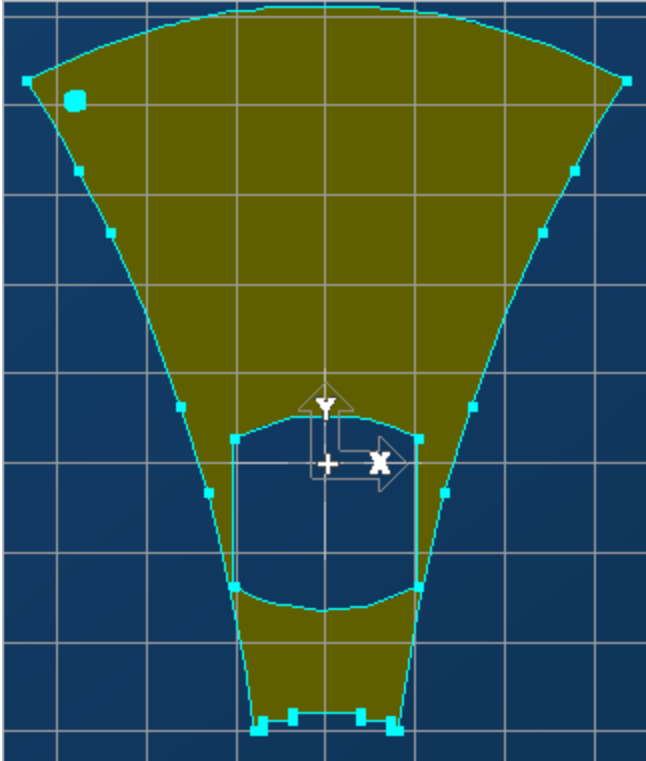
Extracting Geometry from a Sheet

1. Turn on Face selection.  Select the face you unbagged.
2. Right-click the face.



3. From the menu, select  Extract Edges.

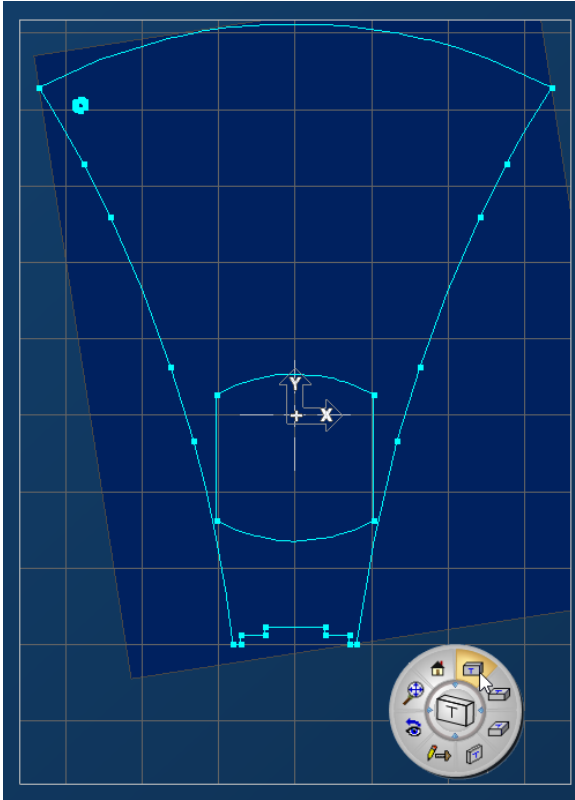


4. Click the Do It button  or (Alt+.).



Untrimming and Trimming Sheets

1. Open the Surface Modeling  palette.
2. Select the sheet and click the Trim/Untrim button. 



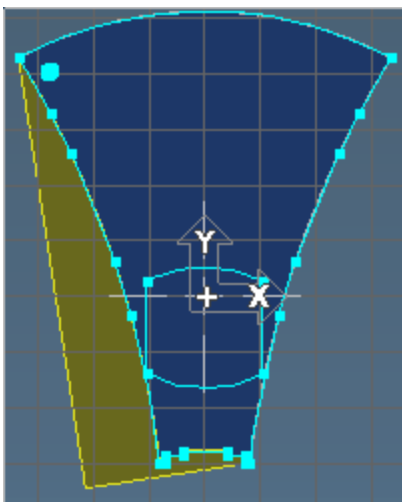
With only a sheet selected, the system will perform the Untrim operation. When a sheet is untrimmed, it removes the bounding edge of the sheet and replace it with the underlying surface that defines the sheet.

Now, we need to trim this surface using the geometry that we extracted. The trim function is performed with the same button that we used to untrim. The trim function will be performed when a sheet and geometry are selected. In order to trim a sheet the selected geometry must completely separate the selected sheet into two distinct areas.


3. Select the sheet and (**Ctrl+double-click**) anywhere on the outside bounding geometry.

4. Click the  Trim/Untrim button.

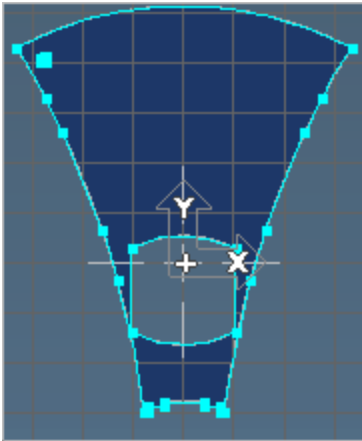
5. **Delete** the sheet that is created on the outside of the geometry. You will get a warning about Deletion losing history. Ignore this and click **OK**.



The exterior sheet is created differently depending on which part of geometry was selected. Exterior sheets and smaller sheets will always be the selected sheet after a trim, making it easy to delete.

6. Select the sheet and the inside geometry and click the  Trim/Untrim button.


7. **Delete** the inner sheet, again ignoring the warning about history being lost.



In the **Body Bag** select all (**Ctrl+A**) then **Right-click** the **Body Bag** title bar and click **Un-bag Selected** to bring all the sheets into the workspace.

8. Hide the geometry. 

Simplifying the Model


1. With all sheets selected (**Ctrl+A**), select **Solids > Tools >**  **Simplify**.

This attempts to convert NURBS surfaces into analytic surfaces. Often times when surface files are imported into the system, analytic surfaces are converted into NURBS. This function converts those NURBS back into analytics. This may take some time to complete.

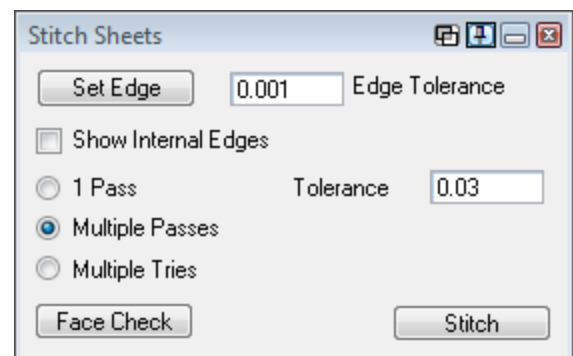
2. Select **Solids > Tools >**  **Remove Unused Topology**.

The Remove Unused Topology function attempts to merge any sheets that are defined by the same underlying surface definition into one sheet, thus reducing the number of individual sheets and simplifying the part file. Again this may take some time to complete.

Stitching


1. Open the **Stitch Sheets**  dialog.

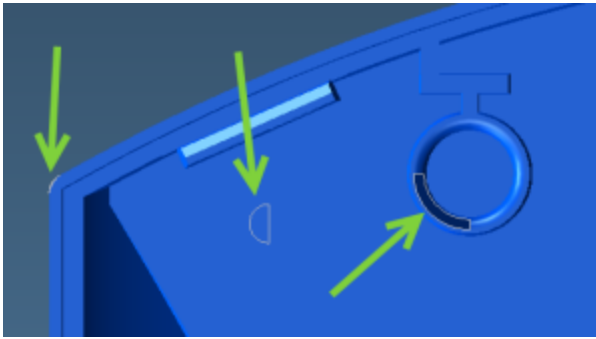
2. Enter the information shown in the **Stitch Sheets** dialog.



Make sure the **Show Internal Edges** option is unchecked.

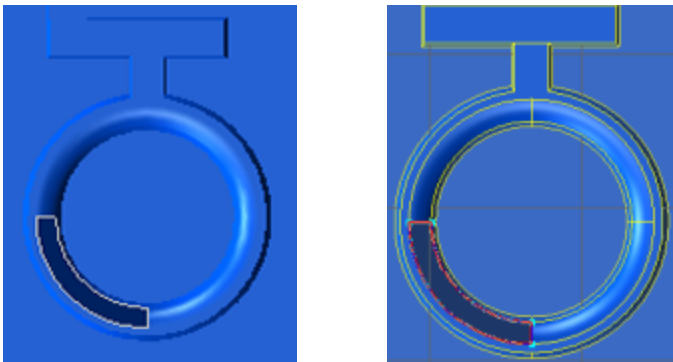
This will allow us to view any edges that do not successfully stitch in the subsequent stitching operation. Edges that are successfully stitched become internal edges and are not displayed on the screen when this item is unchecked. Only the external edges, edges that did not stitch will be shown. We will need to have the edges in view in order to view the results.



3. Turn on  Edge selection.
4. **Click** the **Stitch** button.

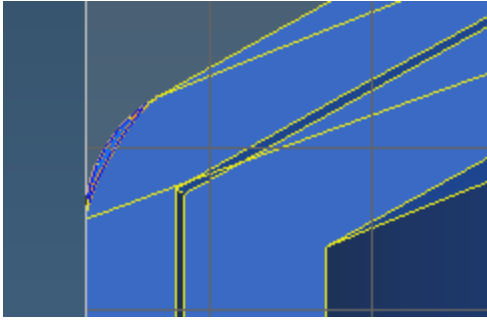


When the stitch is complete, you should be able to see three external edges (white) that were not able to be stitched. (Delete the circle geometry as that may obstruct the view) They can be found on the upper left-hand corner of the part. We will start on the right and then work our way to the left since they are all in the same area.

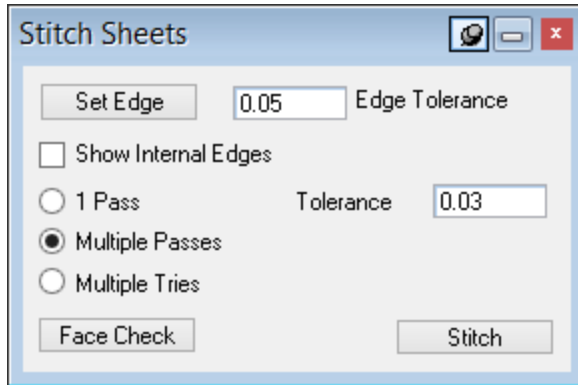
5. Zoom in to the far left boss to the external edge as shown below left.



6. **Double-click** the edge around the boss as shown above right, and extract the geometry from it. 
7. **Double-click** the geometry and **click** the Coons Patch button. 
8. Move to the far left corner of the model.
9. **Double-click** the external edge in the upper left-hand corner.



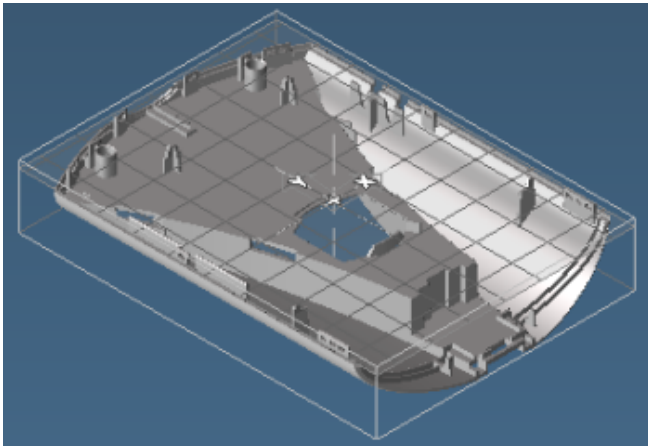
- In the Stitch Sheets dialog which should still be open, enter a value of **0.05** for the Edge Tolerance and **click** first the Set Edge and then Stitch buttons.



The edge will disappear because it was successfully stitched and has become an internal edge.

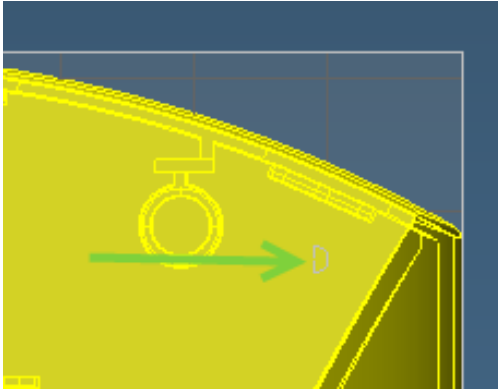
Solidifying the model

- Select all (**Ctrl+A**) of the sheets and **click** the **Stitch** button in the Stitch sheets dialog.



When the stitching operation is complete, the sheets should be solidified into a solid body. This will be apparent by the fact that the part model is drawn in gray, indicating a solid body, rather than in blue, indicating sheets.

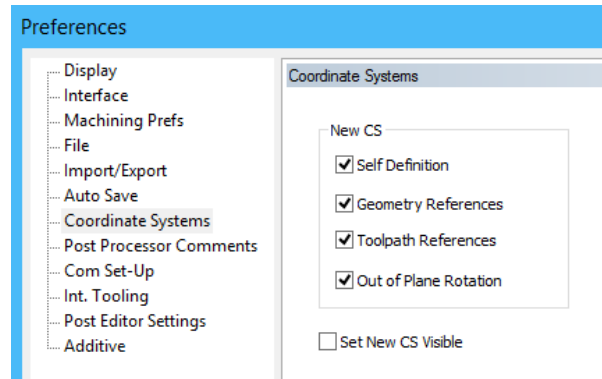
- Switch to the opposite home view (**Ctrl+Alt+H**).




Remember there was also a semicircular unstitched area. This was part of a dimple in the back of the surface. Since we missed creating a sheet for the other half of the dimple, we can easily recreate it with solids by creating a sphere and subtracting it from the model. This is a difficult task because the full diameter is not at the surface.


- Place the solid model in the Body Bag. The unstitched sheet will be left behind.

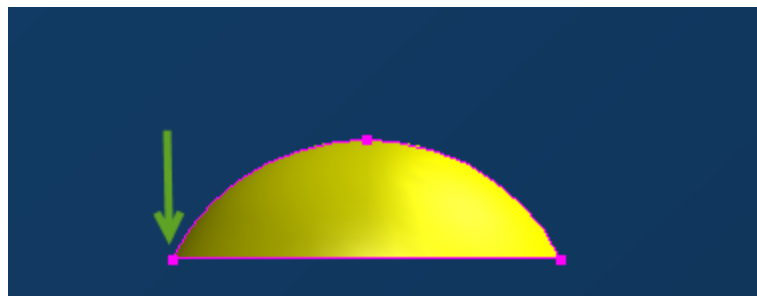
We will extract the geometry and align a CS to get the correct center point and radius for the sphere. Before we continue be sure that your CS preferences are set as shown here.



- Create a new CS and Workgroup. Ensure Face selection is enabled and extract  geometry from the sheet.
- Switch to the right side view (**Ctrl+G**), select the three points and **click** the Align CS button.

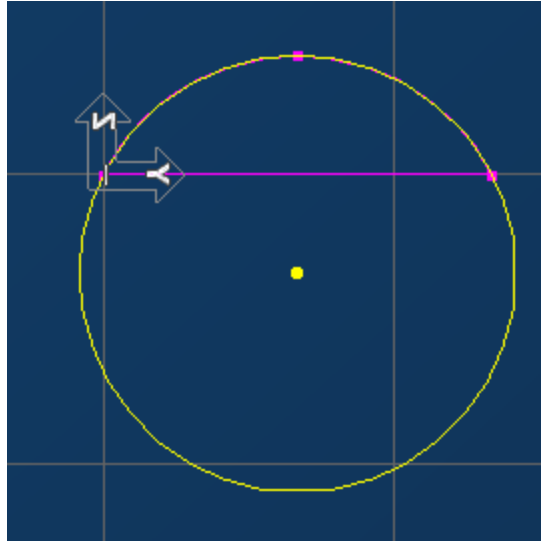



- Change the CS origin  to the left point as shown.

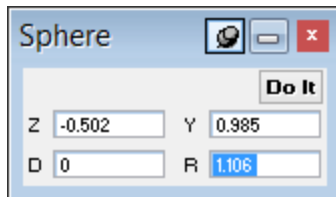


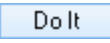

- Delete** the sheet.

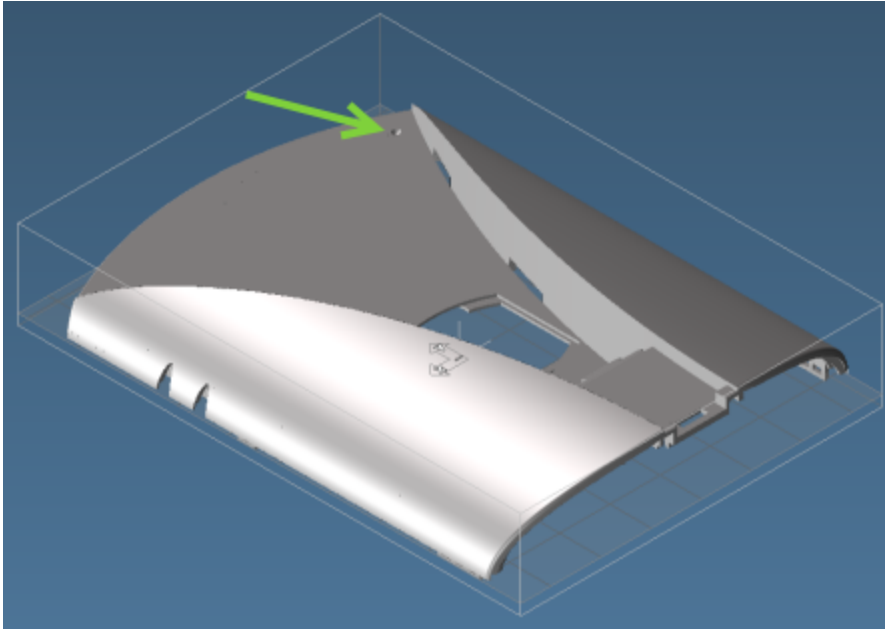
8. Create a circle from the three points.



9. Open the Sphere  dialog from the Solid Modeling palette.
10. With the D field highlighted, (**Shift+Alt+Click**) on the center point of the circle.
11. Place the cursor in the R field and (**Alt+Click**) the circle to interrogate the radius.



12. **Click** the **Do It** button  or (**Ctrl+.**).
13. Subtract the sphere from the model by placing it into the body bag, then selecting first the model
then the sphere and clicking the subtract button .



Viewed from the opposite isometric view the model shows the corrected dimple.

Save this part.