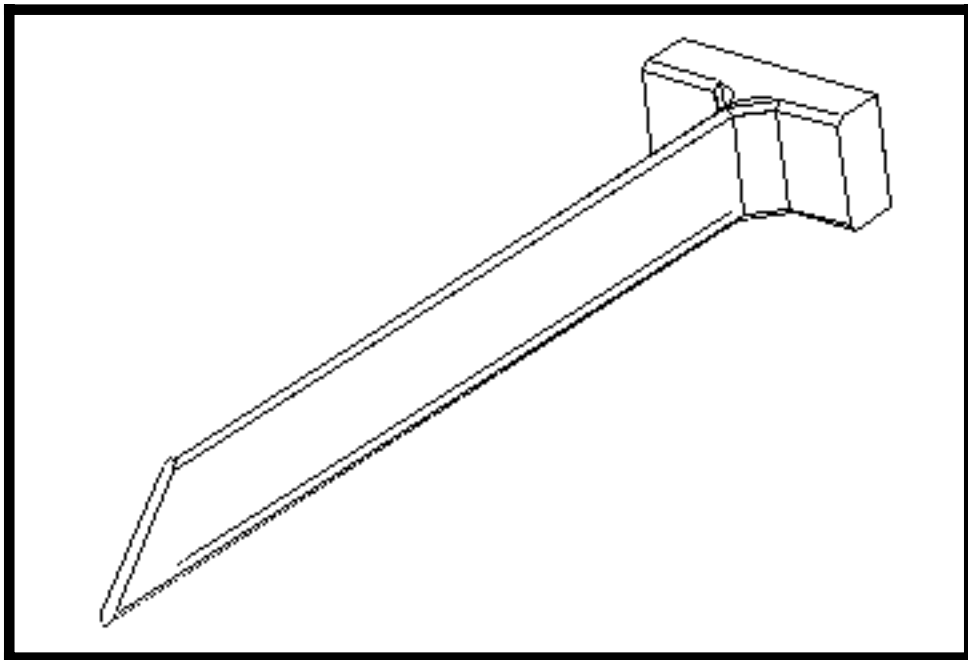

LESSON 6

Imported Geometry Cleaning



Objectives:

- Import CAD geometry.
- Identify problems with imported geometry.
- Fix geometry for the purpose of meshing.



Introduction:

Accessing Computer Aided Design (CAD) geometry in MSC/PATRAN is an essential step in the process of building a model for analysis. However, the requirement on the geometric representation of the analytical model is somewhat different from other applications, such as design and manufacturing.

The process of importing the geometry from a CAD file goes through a translation which is referred to as STEP/EXPRESS. This translation generates an express file that enables MSC/PATRAN to understand the native representation of the geometry.

Geometry information from CAD sources in general are too detailed for analysis purposes. The CAD geometry needs to be edited to help simplify the model.

There are many ways of building the same geometry in the CAD systems. The variety of representation leads to variation on the quality of the models. The quality of the geometric representation greatly impacts the quality of the translated data used in application like Computer Aided Engineering (CAE).

MSC/PATRAN provides tools to automate removal of unnecessary geometric details. The primary functions of these tools are verification, location and mending the problems.

In this exercise, a CAD file will be imported into MSC/PATRAN. Next, the geometry will be checked. After pinpointing these problems, a fix will be performed.

In particular, CAE requires a congruent model. Most CAD models are satisfactory if they just “look” like the part. They do not truly represent a continuum. Hence, it is necessary to edit the model to make it congruent.

Suggested Exercise Steps:

- Create a new database.
- Import the part file **blade.bxp** and update the model tolerance by applying the suggested value.
- Modify the geometry to be able to create a solid representation such as the boundary representation (B-

Rep).

- Verify the congruency in the geometry.
- Edit those surfaces that are noncongruent.
- Create a trimmed surface using the defined outerloop and the reference surface generated during the disassemble step.
- Verify the congruency of the edited model and create a B-Rep solid.

Files:

All the files used in this exercise are listed below. Each listing includes the file, where it originated, its format (text/binary) and summary information as to how it relates to this exercise.

File	Source	Description
blade.bxp	Supplied	This is a P3/PATRAN binary express part file. This file is the result of MSC/PATRAN traversing the CAD part and creating a STEP/EXPRESS file.
Clean_Geometry.db	Created	This is a binary MSC/PATRAN database which is created in this exercise and is used to import the bxp part file.

Exercise Procedure:

1. Create a new database named **Clean_Geometry**.

File/New ...*New Database Name:***Clean_Geometry****OK****Open a New Database**

In the *New Model Preference* form set the model preference as default.

Tolerance:◆ **Default***Analysis Code:***MSC/NASTRAN***Analysis Type:***Structural****OK**

2. Import the part file found in the neutral express file **blade.bxp**. Update the model tolerance by applying the suggested value. When the suggested tolerance is within a reasonable range from the set value complete the importing by pressing the **Cancel** button.

Import the Geometry

The neutral express file is an intermediate file between the CAD part file and MSC/PATRAN. When the source of imported geometry is the CAD part file, this file will be first translated into an express file then read into PATRAN. Therefore, reading in a neutral express file is an alternative to importing the geometry directly from the part file.

File/Import...*Object:***Model***Source:***Express Neutral***Import File:***blade.bxp****Apply**

If you scroll up in the *command window*, you will read a note about an unresolved topological and/or geometrical gaps in the B-Rep solid representing the file we are trying to read. Furthermore, it is indicated that the B-Rep faces are written out as 24 surfaces.

The cause of this problem is improperly created geometry in the CAD system. This leads to an abnormal situation when PATRAN tries to resolve by readjusting the model tolerance. The first warning message suggests an alternative tolerance which in comparison with the original setting is quite significant. Hence, we will **Apply** the first suggested tolerance. The subsequent warning messages are indicative of a convergence problem that is best described by being caught in an infinite loop. Therefore, you can press the **Cancel** button at the second warning message.

First suggested tolerance:	<input type="button" value="Apply"/>
Second suggested tolerance:	<input type="button" value="Cancel"/>
Import summary:	<input type="button" value="OK"/>

Notice that there is a warning message in the *command window* stating there is *A gap between two of the faces in the Body. This may be due to edges not matching directly or a missing face.*

Clear the warning messages and import summary

- For a better view of the model, we need to do the following.

First, change the view angle to **Iso 3 View** by clicking on the following icon:



Iso 3 View icon

To easily visualize the problem in the model, we will modify the display properties. The display lines will be removed and the entities will be shrunk by a factor of 5%.

<input type="button" value="Display/Geometry..."/>	
<i>Number of Display Lines</i>	<input type="text" value="0"/>
<i>Geometric Shrink</i>	<input type="text" value="0.05"/>
<i>Point Size</i>	<input type="text" value="3"/>
<input type="button" value="Apply"/>	
<input type="button" value="Cancel"/>	

Change the Display Properties of the Model

Finally, we will adjust graphics preferences.

Preferences/Graphics...

Auto Extend

Auto Fit View

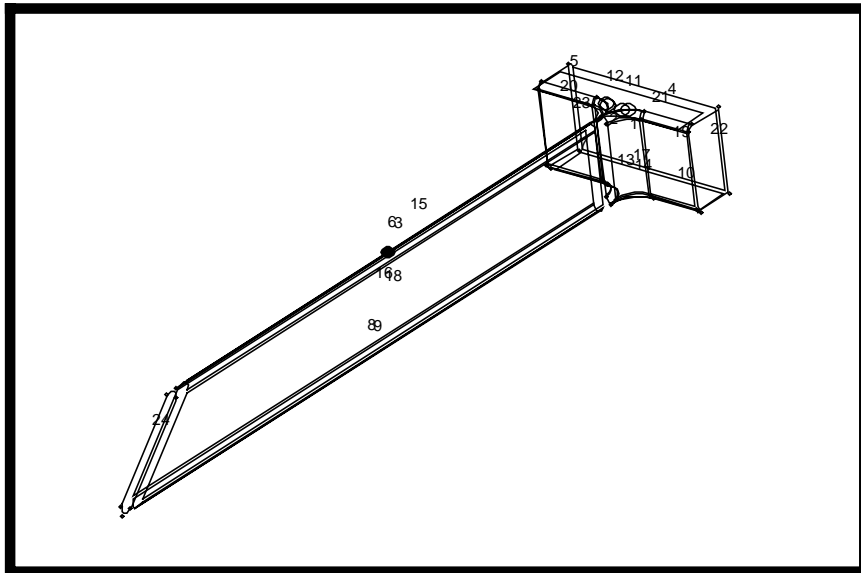
Auto Center

Highlight Colors: Primary Color: **(Change the color to white)**

Apply

Cancel

Your model should appear as shown below.



4. Reading the geometry from the part file had some flaws. This is evident from the unresolved gap in the topology and geometry that required the update of the geometric tolerance. The objective now is to modify the geometry to be able to create a solid representation for the model.

**Verify
Congruency**

It is required that all surfaces be congruent in order to create a B-Rep solid. This and the following steps outline the procedure to fix the geometry. This includes verifying the congruency of the surfaces representing the boundaries of the model and editing those surfaces that are noncongruent.

First, let us verify the congruency of the model.

◆ **Geometry**

Action:

Verify

Object:

Surface

Method:

Boundary

◆ **Topology**

Surface List

Surface 1:24

Apply

Notice that a warning message should appear in the *command window* stating that *Free edges and/or non-manifold edges exist*.

In the *Update Graphics* form press on the **Plot Incongruent Surfaces**. This will display on the screen only the surfaces that are not congruent.

Plot Incongruent Surfaces

OK

Note the blue maker on surface 15. Change the display as follow to get a clear idea of how surface 15 relates to the rest of the model.

Display/Highlight...

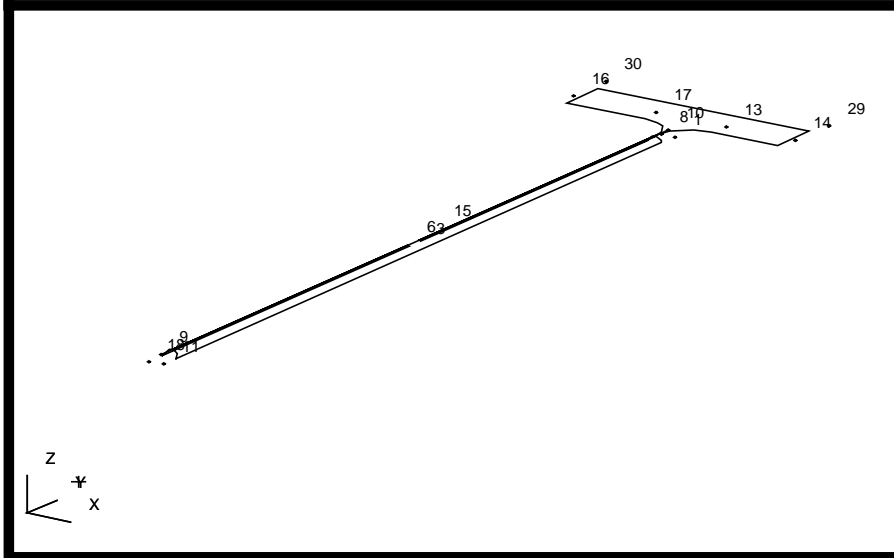
Selected Entities:

Surface 15

Highlight

OK

Your model should appear as shown below.



- Let us isolate the noncongruent surfaces so their manipulation is manageable.

We will create a new group and include in it only the geometry representing the noncongruent surfaces.

Group Non-congruent Surfaces

Group/Create...

New Group Name:

noncong_surf

Make Current

Unpost All Other Groups

Add Entity Selection

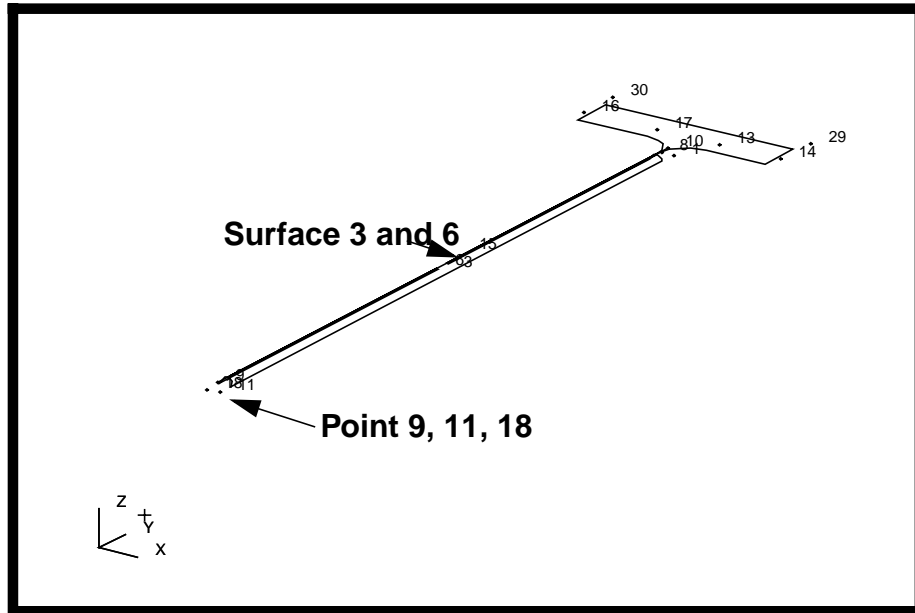
Entity Selection

Select all entities on the screen displayed after the Plot Incongruent Surfaces button was pressed.

Apply

Cancel

- Remove all the regular surfaces and the associated vertices from the viewport as follows:



Display/Plot/Erase...

Select Entities:

Surface 3 6 Point 9 11 18

Erase

OK

**Dis-
assemble
Surface**

- Second, edit *surface 15* such that it becomes congruent with the rest of the surfaces.

Now we are left with surface 15 in the display. This surface is a trimmed surface. It is made of curves representing the boundaries and a reference surface that defines its curvature. Perform the following procedure to disassemble the surface:

◆ **Geometry**

Action:

Edit

Object:

Surface

Method:

Disassemble

■ **Delete Original Surfaces**

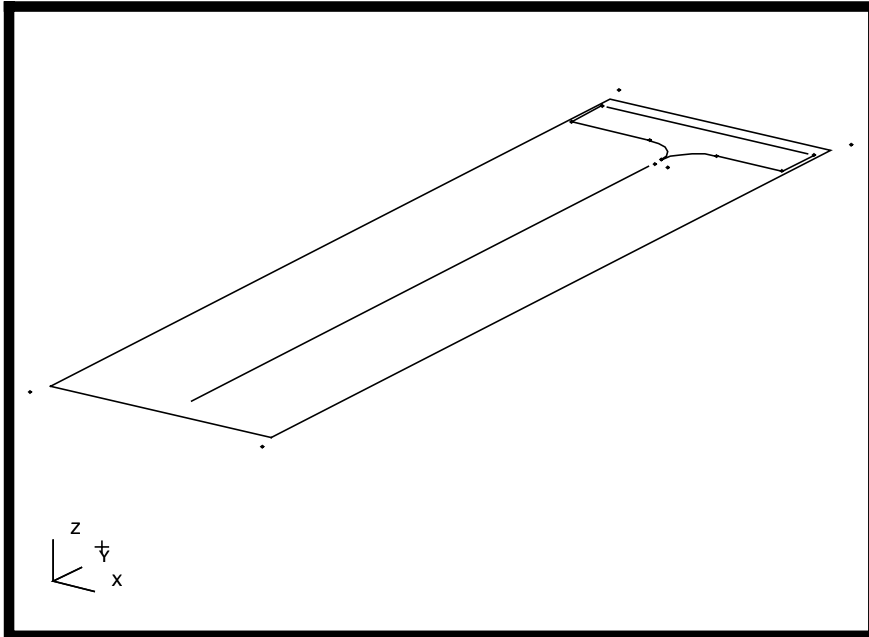
Trimmed Surface List

Surface 15

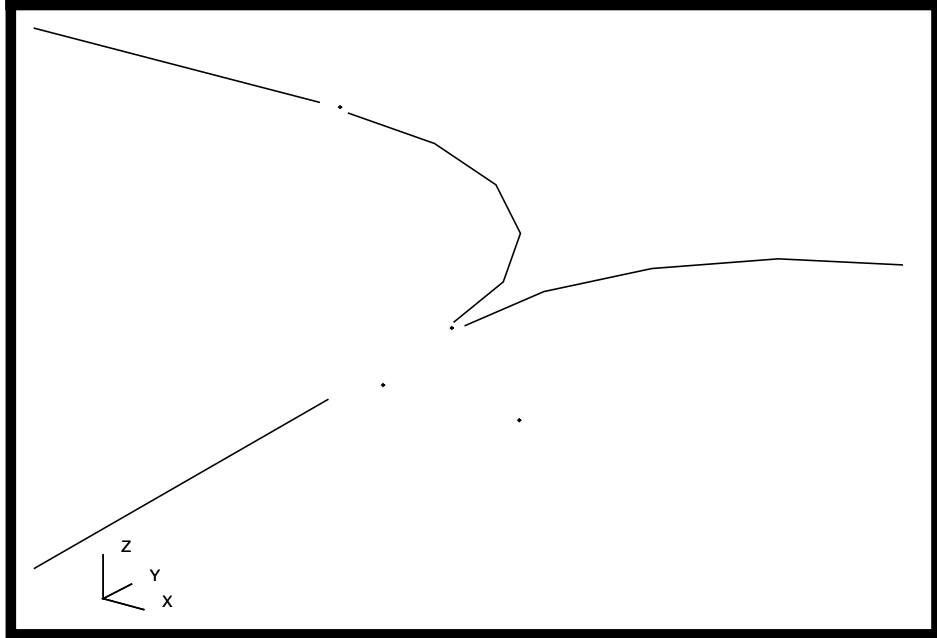
Apply

We don't want to create duplicates for entities. Respond with **No for all** to the warning message. Answer **Yes** to delete original geometry. You need to repaint the screen to see the change.

Your model should appear as shown below.



As you take a close up look at the boundary you will find that the outer boundary for surface 15 was not actually a closed one (see zoomed in image next page). In fact, you will find a gap that is approximately .001 units. This leads to the question; What do curves 1 and 2 represent?



Remove Un-necessary Geometry

8. Remove the extra curves, and redefine the outer boundary of what will be a replacement for surface 15. Use Create/ Curve/Chain to create the outer loop. You may need to adjust the global tolerance to complete the chaining step.

Now it is clear what is wrong with the geometry. This is one form degeneracy we may encounter. However, there are other forms as well. The procedure in this exercise can be applied to large number of problems.

We are not interested in the cause of the problem. But, we need to keep in mind that creating geometry with CAD systems is not unique, and there are implications of creating geometry one way over other ways.

Our objective is to create a surface that replaces surface 15. First, we need to delete the following curves because they have no relevance to the model.

◆ Geometry

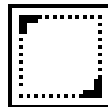
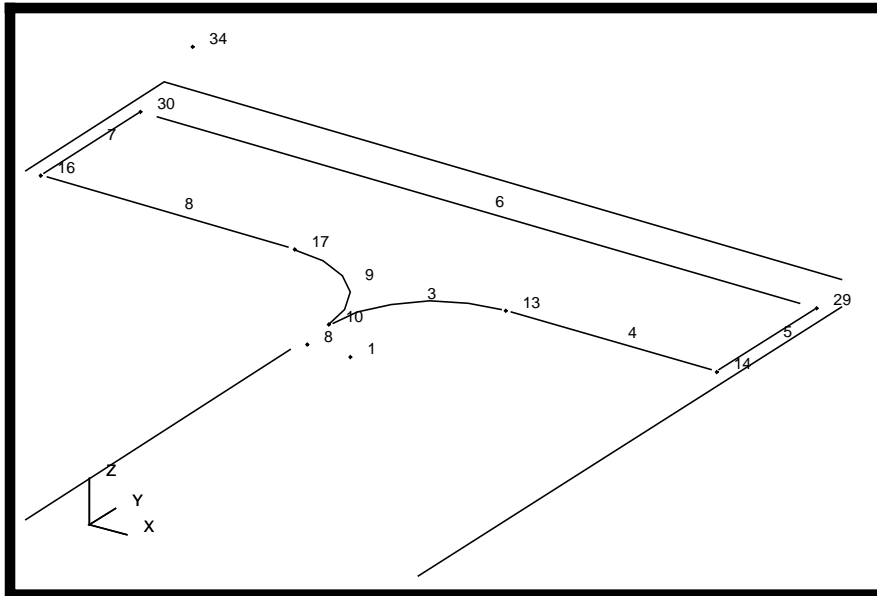
Action:

Object:

Curve List

Next, we need to define the outer boundary of the surface. This is done by chaining the appropriate curves and edges. We will use the manual Chain option.

Let us zoom in on the region to be modified. Use the **Select corners** icon to get the image shown below on the screen.



Select Corners Icon

Chain the Curves

◆ **Geometry**

Action:

Create

Object:

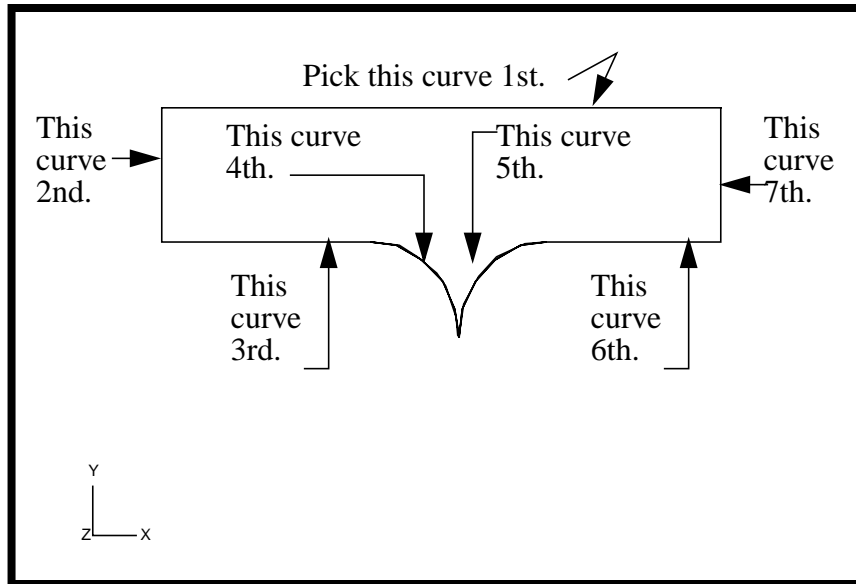
Curve

Method:

Chain

■ **Delete Constituent Curves**

Note the sequence of curve list picking is shown in the figure below.



Remember to use the appropriate icon in the select menu to pick the curves and surface edges for curve chaining.

Use the following icon to pick the curves:



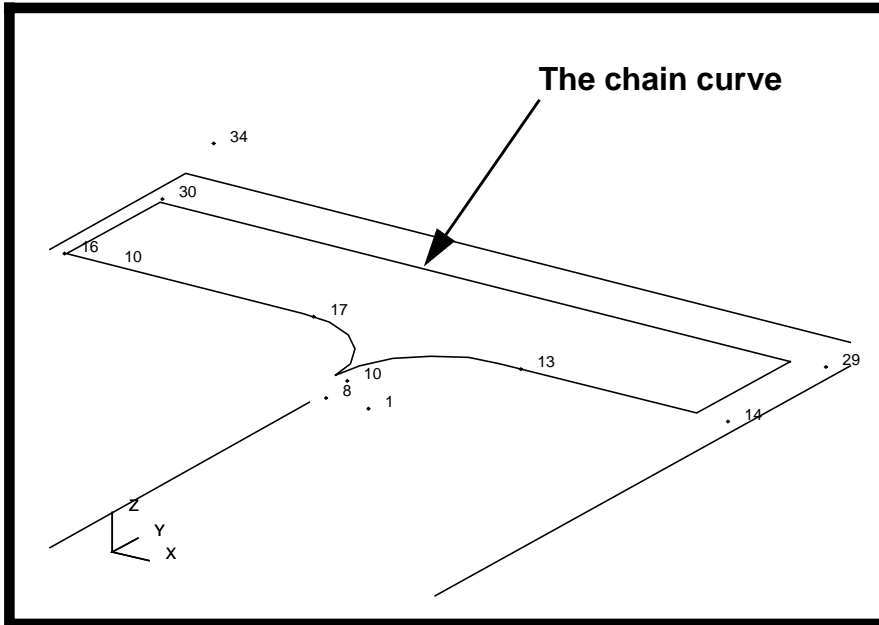
Curve List:

Select curves and edges as indicated in the illustration above.

Apply

Answer **Yes** to delete the original curves.

The resulting chained curve (as shown below) will be used as the outer loop of a trimmed surface.



9. Create a trimmed surface using the defined outer loop and the reference surface generated during the disassemble step.

**Create
Trimmed
Surface**

A trimmed surface is defined in PATRAN language as a general surface with multiple edges, more than 3 or 4 edges, that defines the outer boundary and possible inner boundaries such as holes. In light of this definition, we will use chained curve from the previous step as the outer loop and *surface 25* as the base. There are no inner loops involved with the trimmed surface.

◆ **Geometry**

Action:

Create

Object:

Surface

Method:

Trimmed

Option:

Surface

■ **Use All Edge Vertices**

■ **Delete Outer Loop**

Outer Loop List

Curve 10

■ **Delete Constituent Surface**

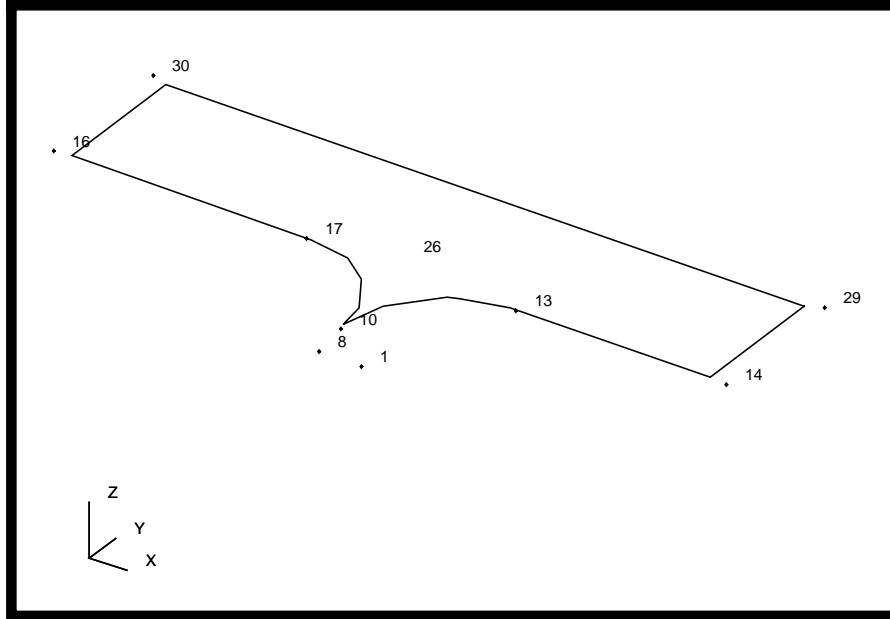
Surface List

Surface 25

Apply

Answer **Yes** to delete the original curves and surfaces.

Your model should appear as follows:



Modify the Default Group

10. Add the new surface to the **default_group**.

Remember that the newly created trimmed *surface 26* belongs to the current group. The current group is not the **default_group** that we imported the model. Therefore, this surface must be added to the **default_group**.

Group/Modify...

Change Target Group...

Existing Groups

default_group

Cancel

Member List to Add/Remove:

Surface 26

Add

OK

Change the Display

11. Post the default group.

Group/Post...

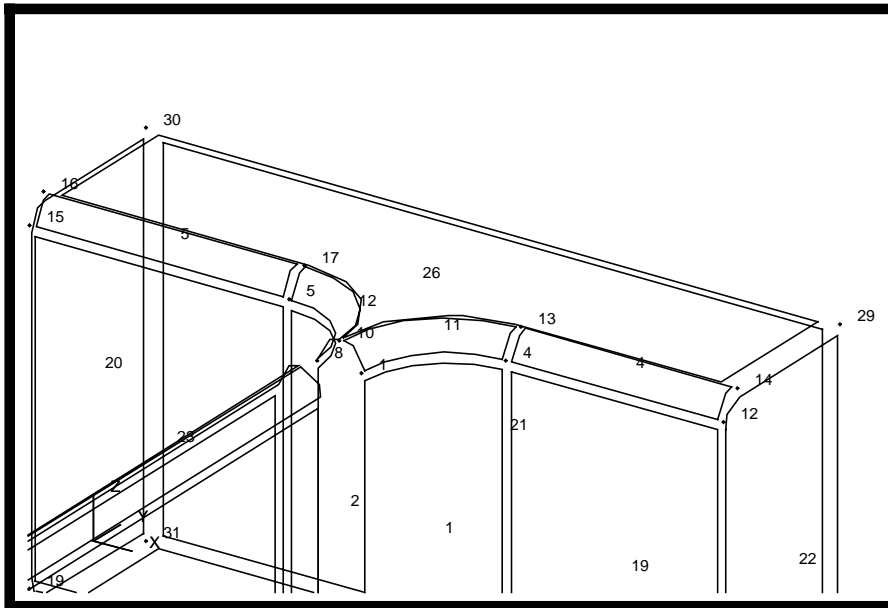
Select Groups to Post

default_group

Apply

Cancel

The following should appear in your viewport.



12. Again, verify the congruency of the model.

First, use the **Fit view** icon to resize the model to fit on the screen.



After that, apply the following:

◆ **Geometry**

Action:

Verify

Object:

Surface

Method:

Boundary

◆ **Topology**

Surface List

Surface 1:14 16:24 26

Apply

Check the *command window* to see that there are no free surface edges in the model.

13. The surface representation of the model is ready to create a B-rep solid.

◆ Geometry

Action:

Create

Object:

Solid

Method:

B-rep

■ Delete Original Surfaces

■ Auto Execute

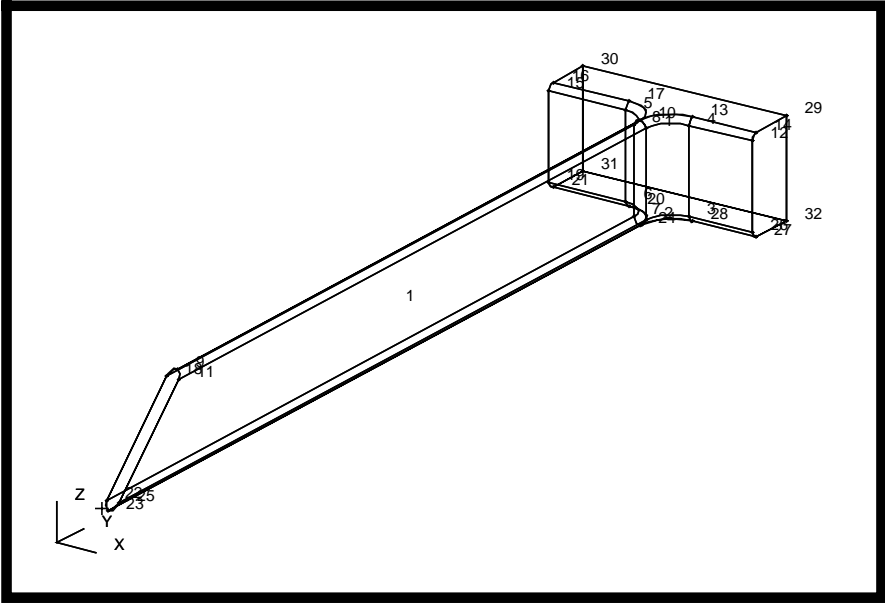
Surface List

Surface 1:14 16:24 26

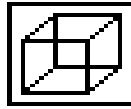
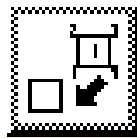
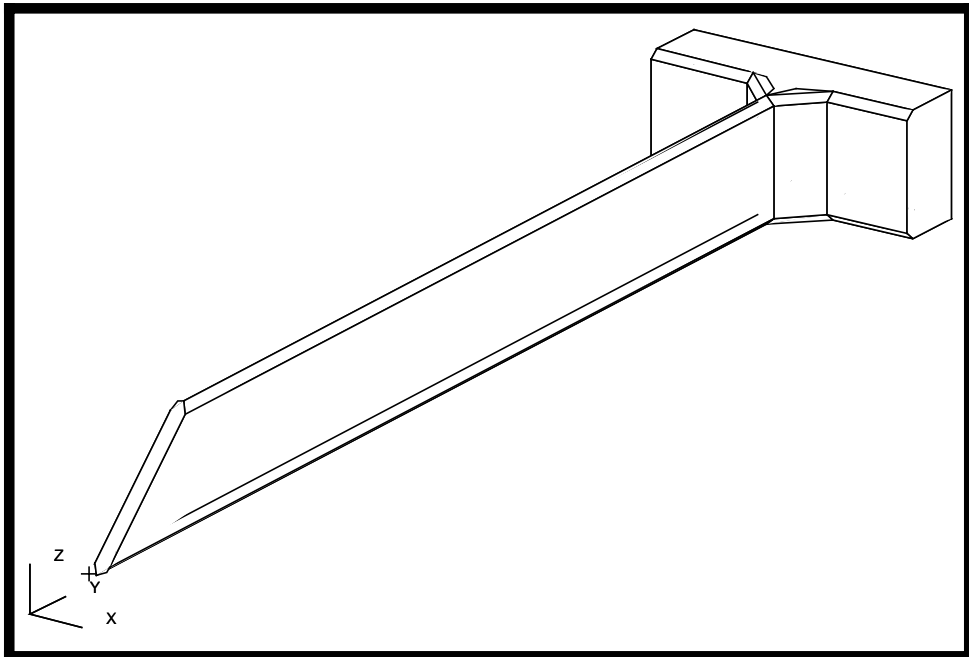
Apply

Answer **Yes** to delete the original surfaces.

A statement will show up in the *command window* to state that *solid 1* has been created (see the following figure). You need to reset the display properties for zero shrink factor (through *Display/Geometry*).



14. To get a better view of the model, change the display by hiding the labels and making the display **Hidden Line**:

**Hidden Line icon****Hide All Labels Icon**

Change the *render style* back to **Wireframe** and close the database.

