

Pro/ENGINEER[®]
Wildfire[™] 2.0

Pro/COMPOSITE[™]
Help Topic Collection

Parametric Technology Corporation

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Using Additional Modules

Advanced Surface Extension

Pro/COMPOSITE

Basic Techniques

About Composite Design

The design of a composite part begins with a surface on a Pro/ENGINEER model. The composite part is a distinct Pro/ENGINEER entity based on the existing model surface. The composite part consists of plies and core material added to the surface, with beads and flanges to stiffen and connect the plies. Once all the material is added, the part can be solidified, analogous to being cured, to bind all the material together into one solid part.

To document the composite part, two types of drawings can be made: drawings that document the design part itself, showing dimensions, a bill of materials of the plies involved, and so on, or a drawing that automatically shows the material added with each sequence.

To Create a Composite Part

Once you have a design part with an available base surface, you can begin your composite part design. If you want to use a solid surface for applying composite material, copy it into a surface feature prior to creating the composite part.

1. From the Pro/ENGINEER menu bar, click **File > New**. The **New** dialog box appears.
2. Select **Part** as the Type, and **Composite** as the Sub-type, then enter the name of the new composite part.
3. Type or select the name of the existing design part that contains the base surface. If the part contains more than one surface feature, the part appears in a small window.
4. Select the surface to be used as the base surface. When the surface has been selected, the design model appears in the main window. A merge of the design part into the composite part is created automatically.

You can now add beads and flanges to the composite part. These must all be added before you start adding plies or cores.

Creating the Design Part

The design part is a regular Pro/ENGINEER part that can consist of surface features and solid features. However, only a single surface feature or merged surface of this part can be used as the basis for the composite part design. This surface forms the

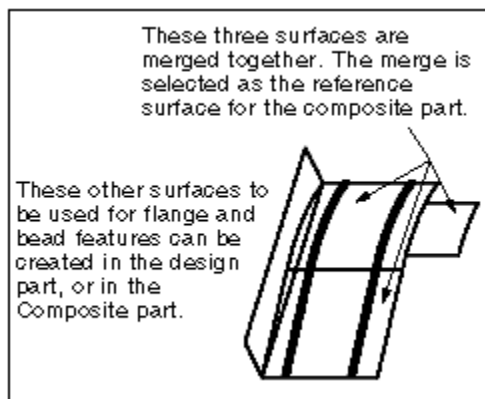
molded surface, referred to hereafter as the base surface, giving shape to all the composite material to be added later.

Use the following guidelines in creating a design part:

- The design part surface can be a flat panel, a ruled surface, or another surface with a curvature.
- The design part surface should be free of flanges and beads that are used to hold panels together or to stiffen the panels. These will be added in Composite mode.
- Other surfaces and datum curves can exist in the design part. These will be available in the Composite part and can be used for ply and core feature boundaries and for creating beads and flanges.
- Create at least one coordinate system before you begin composite feature creation. Coordinate systems are used as a reference for the angle of bias of the ply and core material.

The following figure shows an example of a design part.

Design Model for Composite Part



To Create Beads and Flanges

Beads and flanges are surface features that are added to the base surface feature. When all beads and flanges are added, the resulting surface forms the surface on which material is applied. This is called the bottom surface. Add all beads and flanges to the part before you add the first ply feature. Once the first ply is added, **Bead/Flange** is inaccessible.

1. Create a surface or surfaces to form the bead or flange geometry. These surfaces need to intersect or join to the base surface. Flanges must be added at the edges of the base surface; beads can be added to the interior of the surface.
2. Click **Applications > Legacy > Feature > Create > Composite > Bead/Flange**. The **SURFACE MERGE** dialog box opens.
3. Use the **SURFACE MERGE** dialog box to include the bead or flange surfaces in the base surface. The base or bottom surface is selected automatically for merging.

4. You can create round features along the two-sided edges of the beads and flanges for the composite features to have a smooth transition over them. Create the rounds using **Insert > Round**.

To Define Material During Feature Creation

When you are creating a ply, you are prompted to select or create the ply material after you select the direction of the ply.

1. Click **Create** from the **MATERIALS** menu.
2. Enter a name for the composite material.
3. Using the following options in the **PARAMETERS** menu, define the parameters for the material:
 - **Retrieve**—Uses the parameters from a saved material.
 - **Save**—Saves the parameters to a file with the extension `.mat`. Saving material parameters allows you to retrieve those parameters in other composite parts.
 - **Mod Params**—Edits the existing parameters using the table editor.
 - **Show**—Displays the current parameter values in an information window.

Note: During ply or core feature creation, you can only define a new material. You cannot rename or modify an existing material.

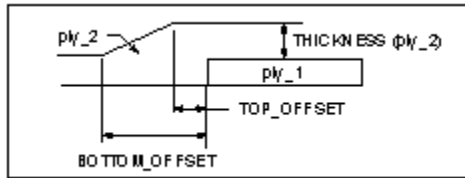
Composite Material Parameters

Composite material can be defined before or during composite feature creation. It consists of five parameters:

- **BOTTOM_OFFSET**—The bottom offset of material when overlapping other plies (see the figure below).
- **DENSITY**—Mass density of the composite material, default equals 1.
- **FIBER_ORIENT**—Whether the ply or core material has bias. Possible values are `Yes` and `No`.
- **THICKNESS**—Thickness of the material. The default changes with respect to the part size (see the figure below).
- **TOP_OFFSET**—The top offset of material when overlapping other plies (see the figure below).

The following figure illustrates composite material parameters.

Composite Material



Tip: Working with Composite Material Outside of Feature Creation

To work with composite material outside of feature creation, click **PART > Set Up > Material > Composite**. The **CMP MATERIAL** menu appears with the following options:

- **Add**—Adds a new composite material to the part. This option is automatically selected if no composite material exists in the part. Type the material name, and use the options on the **PARAMETERS** menu to retrieve, save, modify, or show the material parameters.
- **Remove**—Removes a previously defined material. A list containing all material names in the part will appear. Select the material to delete and click **Done Sel.**
- **Modify**—Modifies a material. Use the **PARAMETERS** menu options to retrieve, save, show, or modify the material parameters.

If you want to save the material data to a file for use in other parts, click **PARAMETERS > Save**.

Composite Features

About Composite Features

You can create composite plies and cores by specifying the boundaries and the material properties. As each feature is added, the top surface is modified to accommodate the contours of the new feature. The bottom surface is modified by having the ply boundaries projected onto it. These are characterized by curves appearing on the bottom surface in cyan. As you create additional features, you can use these curves.

Note: The first composite feature added to the part requires that you specify the direction in which the composite part will be built.

To Create a Ply

A ply is a type of composite feature. You can create a number of different plies using the **POSITION** menu.

1. Click **Feature > Create > Composite > Ply**.
2. Use the **POSITION** menu to specify the type of ply. The options are:

- **Full Body**—Creates a ply that automatically covers the entire bottom surface.
- **Boundaries**—Creates a ply that is bounded by the selected curves.
- **No Offset**—Creates the ply feature using the selected boundaries.
- **Offset**—Creates a ply that is offset from the selected boundary curves (similar to creating an offset sketch). This option is not available for full body plies.

To Create a Full Body Ply

1. Click **Feature > Create > Composite > Ply > Full Body > Done**. An arrow appears indicating the direction in which the ply will be created.
2. Click **Flip** or **Okay**.
3. Select or create a composite material.

Note: Only the names of composite material that have already been added to the part will be displayed for selection. If you have material defined that is not in the name list, enter the name to add it to the part and retrieve the appropriate parameters.

4. Select the reference coordinate system for aligning the material bias, and enter an angle for the bias. The angle is about the z-axis and with respect to the x-axis. The top quilt of the part will be adjusted for the new ply.

To Create a Boundaries Ply

1. Click **Feature > Create > Composite > Ply > Boundaries > Offset/No Offset > Done**. An arrow appears indicating the direction in which the ply will be created.
2. Click **Flip** or **Okay**.
3. The composite part is placed in Sketcher so you can select existing curves by using **Sel Curve/Edge**, **Sel Chain**, or the "bottom" surface edges. If you have an open curve (it must be aligned to the bottom surface edges), the system automatically closes the boundary by using the bottom surface edges.
4. Click **Done**. The system prompts you to select the side of the closed loop where the ply should be added.
5. Click **Flip** or **Okay**. If the ply boundary is Offset, the system prompts you to enter the boundary offset value in the direction shown. If any edges of the boundary have a different offset, select them and enter their offset.
6. Continue until all offsets have been established, then choose **Done Sel**.
7. Type the name for the material or create a new one.
8. Click **PARAMETERS > Done**. If the material has bias, enter a reference coordinate system for aligning the material bias.

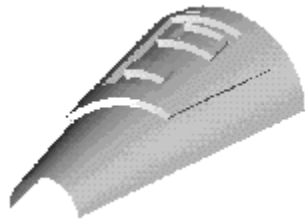
9. Select the coordinate system and enter an angle for the bias. The angle is about the z-axis and with respect to the x-axis. The top surface of the part is adjusted for the new ply.

Example: Creating a Ply from Boundaries

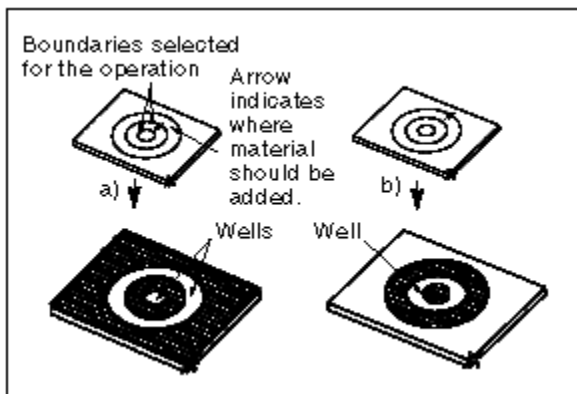
With the **Boundaries** option, you can create a ply with open areas (wells) by selecting multiple open and closed loops to define the boundaries of the ply.

Note: Datum curves delineating boundaries of a ply or well must be created before you start creating a ply.

An example of a ply with wells appears below. Note that for ease of visualization, the thickness of the ply has been exaggerated.

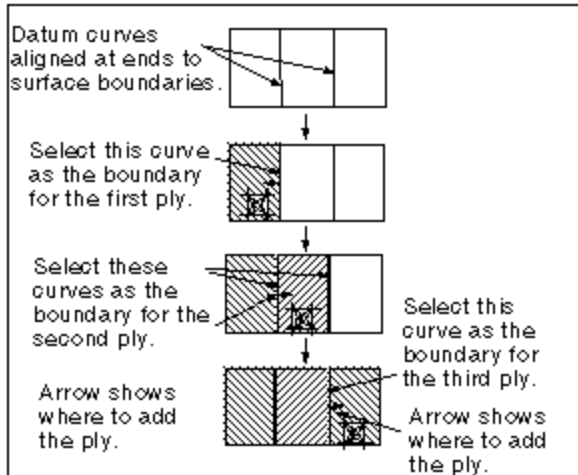


When creating a boundary ply with wells, remember that curves comprising boundaries of wells can be selected in any order. However, the boundary you define first is used by the system to determine the well location—within or outside the boundary. The system places an arrow at this boundary and prompts you to select the side where material should be added. The figure below illustrates how the system creates wells in a ply.



Example: Creating Plies with Common Boundaries

With the **Boundaries** option, you can create plies that share boundaries. An example of such a case appears in the following figure. Note that when you select an open curve aligned to surface edges, the system closes the boundary of a ply by using the surface boundaries; the direction of the arrow indicates where to add the ply.



To Add a Core

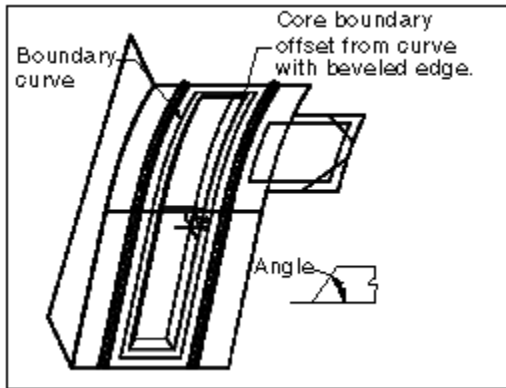
Cores are added to the part to provide stiffness and strength to the panel. A core is generally a much thicker ply with beveled edges so that additional plies to encapsulate the core have a smooth transition over the core. The angle of the core can be up to 90 degrees.

1. Click **Feature > Create > Composite > Core**.
2. Choose one of these options from the **POSITION** menu to specify the type of core:
 - **No Offset**—Creates the core feature using the selected boundaries.
 - **Offset**—Creates a core that is offset from the selected boundary curves (similar to creating an offset sketch).
3. Specify core boundaries, as you do for ply boundaries.
4. Type a bevel angle for the core edges.
5. Select the edges to which the angle applies (for every edge of the core boundary).
6. Click **Done Sel**.

Example: Offset Core

The following figure shows a composite part with an offset core.

Composite Part



To Create a Pre-Form Feature

Pre-forms represent a composite panel already made up of one or more plies/cores. Every ply/core represented by the feature will use the same boundaries.

1. Click **Feature > Create > Composite > Pre-Form**.
2. Click **POSITION > Full Body/Boundaries > Offset/No Offset > Done**.
3. Specify pre-form boundaries, as you do for ply boundaries.
4. When the Pro/TABLE editor opens, enter values for every ply/core in the preform. The data required is:
 - **Material**—The name of a material existing in the part or on disk.
 - **Coordinate system**—The name of the reference coordinate system for biased plies only (material with FIBER_ORIENT = YES).
 - **Angle**—The angle of bias for biased plies. The angle is about the z-axis and with respect the x-axis of the reference coordinate system.
 - **Quantity**—The number of plies with the same material and bias that are laid up consecutively in the pre-form.
5. The top surface of the part will be adjusted for the new ply.

To Attach Two Composite Parts

1. Assemble the two parts.
2. Click **Modify > Mod Part** and select the first (non-solidified) component.
3. Click **Feature > Create > Composite > Attach**.
4. Select the solidified composite part to attach.

5. Select appropriate surfaces for attaching, first from the solidified part, and then from the non-solidified part.
6. Repeat this process until all appropriate surfaces have been attached.

Using Attached Features

Composite parts can be attached to other composite parts using the **Attach** command. This command is available only in an assembly after the two composite parts have been assembled.

To make the parts attach, remember the following:

- The two must have similar surfaces where they attach. A flat surface can attach to a flat surface, a cylindrical surface to a cylindrical surface, and so on.
- The part that is assembled first must not be solidified; the part that is being attached must be solidified.

To Solidify the Composite Part

When you have added all the plies, pre-forms, and cores that you want, you can solidify the part to prepare for engineering analysis and mass properties or for adding other solid features. Once the part is solidified, you can no longer add composite features to the part.

1. Add all the composite features you want.
2. Click **Feature > Create > Composite > Solidify**.

Using Solid Features

Part features that have the ability to join with the ply surfaces (for example, rounds, area offsets, and so on) can be created on the composite part. If created before solidifying the part, they might be used to represent a threaded insert, metal backing plate, and so on. These can then be covered with additional composite features. Once the part is solidified, you can create solid features such as cuts, slots and holes, cosmetic features, and user-defined features to complete the part.

To Create a Cross Section

You can create a cross section of a composite part with the x-section button in the **Part** or **Composite** menu. The resulting cross section will show all plies that are intersected by the cutting plane.

This functionality is available for you if the configuration option `allow_ply_cross_section` is set to `yes` (default).

1. Click **X-section > Create**.
2. Enter the name for the cross section.
3. Select or make a datum plane to be the cutting plane.

To Create a Core Part

Core parts are extracted from the original composite part and flattened. These can then be used in drawings with dimensions added for flat patterns.

1. With the composite model active, click **PART > Flat Core**.
2. If there is more than one core, select which core to use.
3. Type a name for the part or accept the default name.

Modifying Composite Parts

To Insert a Composite Feature

1. Click **FEAT > Insert Mode > Activate**, and select the feature after which you want to insert features. Every feature after the one selected will be suppressed until you cancel Insert mode.
2. Create the appropriate features.
3. When finished, click **Insert Mode**.
4. Click **Cancel**. The features suppressed upon entering Insert mode will be resumed in the appropriate order.

Names for Selecting Composite Features

Plies, cores, and pre-forms are automatically given the following names:

- Ply—PLY_#
- Core—CORE_#
- Pre-form—PF_#

where # is an incremental integer. For example, PF_1, PLY_2, PLY_3, PF_4.

These names appear in the **NAMES** menu when you use the **Sel By Menu** option.

Note: These names can be changed using **Set Up, Name, and Feature**.

Inserting Composite Features

It is easy with Pro/ENGINEER to add features to the composite part. However, you must do this by inserting the features in the following situations:

- A ply feature has been added to the part. The **Bead/Flange** menu option becomes inaccessible. To add more beads or flanges, you must insert the features (surfaces, bead/flange, and round features) before the first ply.
- The part has been solidified, which makes all composite features inaccessible. To add a composite feature, you must insert it before the Solidify feature.

Deleting a Composite Feature

Composite features do not have the same parent and child relationships that other features have. You can delete most composite features without regeneration problems. For example, you can delete `ply_2` from the part without affecting `ply_8`. The part will regenerate from the deleted feature on, adjusting the top of the composite part appropriately.

You can also delete flanges and beads. Composite features will adjust themselves to the new boundaries as necessary.

To Modify Dimensions

1. Click **Modify > Dimension** in the **PART** menu.
2. Select the feature whose dimensions you want to modify.
3. Click **Ok**. The **Dimension Properties** dialog box opens.
4. Modify the dimension.
5. Click **OK**. The dimension is modified.

To Modify the Design Part

Note: The design part (the model referenced by the composite part for its design surface) cannot be modified when in Composite mode.

1. Open the design model in Part mode.
2. Make any modifications to the model.
3. Regenerate the composite part to see the changes in Composite mode.

To Change the Material Parameters

Composite material can be modified for the part and also saved to disk for use by other parts. A different material can also be assigned to a ply or core when modifying the feature.

Note: This procedure modifies the material wherever it is used.

1. Click **PART > Set Up > Material > Composite > Modify**.
2. If more than one material exists in the part, select the material to modify from the **MOD MATERIAL** menu. The **PARAMETERS** menu appears with the following options:
 - **Retrieve**—Retrieves a composite material file from disk to replace the values for the selected material.
 - **Save**—Saves the modified parameters to disk for use by other parts.
 - **Mod Params**—Modifies the current parameter values.
 - **Show**—Shows the current parameter values.

3. Select another option when done.

To Change the Material Assigned to a Ply or Core Feature

The composite material characteristics are assigned to a feature, but exist in the part for every feature to use.

1. Click **PART > Modify**, then select the feature to modify.
2. Click **MOD INTF OPT > Table**. The Pro/TABLE editor opens with the material name, feature name, reference coordinate system and angle—if the feature has bias—and the sequence to which the ply belongs. You can modify every value but the sequence.
3. Modify the appropriate values. Remember that if you change the bias of the feature, you must add or delete the coordinate system name and angle.

Note: Pro/ENGINEER checks the values against the specified material and will not allow invalid changes. For example, if the material has no bias, entering a coordinate system and angle is not allowed, and the changes won't be kept. To do this, modify the material for the feature first, then modify the feature.

4. Exit Pro/TABLE and regenerate the part.

To Modify Sequences

1. Click **PART > Set Up > Sequences**. The Pro/TABLE editor opens with the list of sequences and features.
2. Modify the PLY NAMES column, specifying all the plies to be added with a specific sequence. The plies for a single sequence cannot overlap. Delete any unused sequences.
3. Modify the sequence names if desired.
4. Save the table.
5. Exit Pro/TABLE to save the changes.

Sequences

Composite material is always added in sequence from the bottom up, that is, you cannot lay down a layer of composite material and then lay another underneath it. You can, however, for non-overlapping plies, cores, or pre-forms, build several composite features on the part at the same time. Pro/ENGINEER does not allow you to create these features all at the same time, but it does allow you to specify the building sequence in which the material is added. This information is used to create sequence drawings, in which the material added during a specific sequence is shown.

Pro/ENGINEER establishes a sequence for each composite feature added. It starts with SEQ_0 for the first ply, core, or pre-form feature added, and increments for every feature added after. To modify the plies associated with a sequence, in the second column containing the ply name, enter a list of ply names with a space between them.

You can also rename the sequences without. For example, you can rename SEQ_0 to BASE_PLY. This does not affect the order of the sequences.

Composite Drawings

About Composite Drawings

Drawings of composite parts can document the completed composite panel, or they can document the manufacturing sequence. Sequence drawings are a special application of drawings for Composite parts and views of Plies in a flattened state.

To Create a Sequence Drawing

Sequence drawings allow you to step through the layup sequences of composite material. For each sequence, Pro/ENGINEER will show the new material added, as you specified when you created the composite part or set up sequences in Composite mode.

1. Create a new drawing.
2. Add a view of the composite model using the **Views** command.
3. When Pro/ENGINEER finds that the part is a composite model, it prompts you whether you want to create sequence sheets on the drawing. Answer this prompt with **Yes**. If you answer **No** you will create a regular drawing and will have to start a new drawing to get composite sequences.
4. Add views of the model to the drawing. Every view that you add at any time will automatically appear in all sequences of the drawing.

To Document the Sequences

When you create a sequence drawing, the **Sheets** command in the **Drawing** menu is replaced with **Sequences**. The sequences all have the same number of views of the part, but each sequence can be annotated and dimensioned separately.

1. Click **DRAWING > Sequences**. The **SEQUENCES** menu appears.
2. Click one of the following options:
 - **Finished**—Go to the view of the finished part, where all composite material has been added.
 - **Empty**—Go to the view of the part before any composite material has been added.
 - **Next**—Go to the next sequence.
 - **Previous**—Go to the previous sequence.
 - **Select**—Go to the sequence that you select from the **SEQUENCES** menu.

3. When you have reached the desired sequence, repaint the drawing by clicking **View > Repaint**. This will regenerate the model up to the last feature in the selected sequence.
4. Annotate and dimension the sequence. You can show the dimensions for the plies, pre-forms, and cores. Bias angles are shown at the reference coordinate system.

To Create a Flat Ply View

1. Click **Add View > Of Flat Ply**.
2. Select the center for the view on the drawing.
3. From a name list, select the ply you want to show in the view.
4. Orient the view of the ply. You can only rotate it about the z-axis (normal to screen).
5. When finished, click **Done Orient**.

Flat Ply Views

Any ply that is developable can be shown in a flattened state so that it can be dimensioned and used for a pattern. A flat ply view is always of a single ply, and it can exist in a regular drawing or in sequence drawings. When it exists in sequence drawings, changing the sequence and repainting does not affect a flat ply view.

Note: The **Of Flat Ply** command is not available for full body plies.

Flat views of boundary plies with bias will be shown with cross-hatching indicating the angle of the bias.

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