Fibersim 15 Pro - NX

Student Guide and Activities

CT2245_FS15NX_2

200 Fifth Avenue, Waltham, MA, USA +1.781.250.6800

Warranties and Liabilities

All warranties given by Siemens Product Lifecycle Management Software Inc. about equipment or software are set forth in the users' purchase contract, and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such warranties.

The information and software discussed in this document are subject to change without notice and should not be considered a commitment by Siemens PLM Software. Siemens PLM Software assumes no responsibility for any error that may appear in this document.

The software discussed in this document is furnished under a license and may only be used and copied in accordance with the terms of this license.

Trademarks and Copyright

Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth in subparagraph © (1) (ii) of The Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 or subparagraphs © (1) and (2) of Commercial Computer Software – Restricted Rights at 48 CFAR 52.227-19, as applicable.

Product Support

Siemens Product Lifecycle Management Software Inc. provides extensive support for all of its products. For technical assistance with any product you may contact the Siemens PLM Global Technical Access Center (GTAC), Monday through Friday, 8:00 am – 5:00 pm (local time), using any of the methods listed below.

TELEPHONE:

+1.800.955.0000 (US)

+1.714.952.5444 (International)

The following link is a list of phone numbers for various countries around the world:

http://support.industrysoftware.automation.siemens.com/services/global_number.pdf

WEBSITE:

http://support.industrysoftware.automation.siemens.com/services/gtac.html

Sign up for a user account, using your Siemens SoldTo ID and WebKey Access Code (WAC).

FTP SITE:

ftp.ugs.com

Log in using your **WebKey** account.

UNITED STATES MAIL:

Siemens Product Lifecycle Management Software, Inc.

Global Technical Access Center (GTAC)

200 Fifth Avenue

Waltham, MA 02451

FAX:

+1.781.250.6804

When reporting an issue please provide all relevant information to Product Support. This information includes:

1. Your company name and SoldTo ID.
2. Product name and version number.
3. CAD System and version number
4. A detailed and factual technical description of the issue. This includes a step-by-step account of the actions you performed prior to encountering the issue you are reporting.

To investigate a product issue, our technical staff will need to develop a clear and concise example that illustrates the issue being addressed. To expedite the generation of a solution to an issue you are reporting, it is useful for you to provide a straightforward example that clearly illustrates the problem.

Table of Contents

Chapter 1 Introduction to Fibersim 1-1

Introduction to Fibersim... 1-2

Chapter 2 Model Setup 2-1

Exercise 2a [NX8.5] Create a Model from
an Extended Surface and Planes... 2-2
Exercise 2b [NX9/10] Create a Model from
an Extended Surface and Planes... 2-12
Exercise 2c [NX8.5] Create a Model from a Solid 2-22
Exercise 2d [NX9/10] Create a Model from a Solid 2-30
Exercise 2e NX Model Setup from Planes and Surface
(Optional).. 2-39

Chapter 3 Introduction to Ply-Based Design...................................3-1

Exercise 3a Basic Part with 8 Full-body Plies........................ 3-2
Exercise 3b Reinforcement Plies, Sequencing,
and Cutouts 3-12
Exercise 3c Flat Pattern.. 3-27
Exercise 3d Design to Extended.. 3-32
Exercise 3e Wrapped Ply 3-38
Exercise 3f Create Basic Part.. 3-44

Chapter 4 Producibility..4-1

Exercise 4a How Geometry Affects Producibility................... 4-2
Exercise 4b How Simulation Options Affect Producibility
(Simulation Skin).. 4-29
Exercise 4c How Simulation Options Affect Producibility
(Two-Stage)... 4-38
Exercise 4d Resolving Producibility Issues 4-50

Project 1 Producibility P1-1

Exercise P1 Producibility P1-2

Chapter 5 Rosettes 5-1

Exercise 5a Translational Rosette.. 5-2
Exercise 5b Radial Rosette 5-16
Exercise 5c Spine-Based Rosette 5-24

Chapter 6 Utilities..6-1

Exercise 6a Curve Creation and Ply Drop-off Utilities 6-2
Exercise 6b Fiber Path Curve.. 6-16
Exercise 6c Symmetric Laminate 6-23

Project 2 Monolithic Panel P2-1

Exercise P2 Monolithic Panel P2-2

Chapter 7 Core 7-1

Exercise 7a Core Types 7-2
Exercise 7b Modeled Core with Overcore and IML Laminates 7-15
Exercise 7c Core Panel Design from a Solid (Optional) 7-28

Chapter 8 Documentation 8-1

Exercise 8a 3D Documentation 8-2
Exercise 8b Generating a Ply Book and Ply Table 8-13

Project 3 Core Panel P3-1

Exercise P3 Core Panel P3-2

Chapter 9 Introduction to Multi-Ply Design 9-1

Exercise 9a Basic Multi-Ply Design 9-2

Chapter 10 Transition Manipulation (Multi-Ply Based Design) 10-1

Exercise 10a Basic Offset Specification Options 10-2
Exercise 10b Splitting and Extending Transitions 10-12

Chapter 11 Layer Definition (Multi-Ply Based Design) 11-1

Exercise 11a Layer Sequencing (Multi-Ply Based Design) 11-2

Chapter 12 Surface Offset (Multi-Ply Based Design) 12-1

Exercise 12a Creating Parametric Surface Offset with Ramps 12-2

Chapter 13 Manufacturing Features 13-1

Exercise 13a Layer-based Manufacturing Laminate, with Splice and Dart Groups 13-2

Project 4 C Pillar P4-1

Exercise P4 C Pillar P4-2

Chapter 1

Introduction to Fibersim

Introduction to Fibersim

Fibersim is fully integrated with NX and launched from within the NX user interface. Engineers define composite parts by creating Fibersim objects, such as laminates, rosettes, plies, and cores. During definition, engineers associate these Fibersim objects to their corresponding CAD geometry. In this chapter, the Fibersim user interface will be discussed.

How to start Fibersim.

There are three ways to start Fibersim in NX:

- Press <F9> to start Fibersim.
- **[NX8.5]** Select *Start* > *All Applications* > *Fibersim* as shown in Figure 1–1.
- **[NX8.5]** Click (Fibersim). If the Fibersim icon is not on the screen, make sure **Fibersim** toolbar is turned on.
- **[NX9/10]** Select *File* > *All Applications* > *Fibersim* as shown in Figure 1–2.
- **[NX9/10]** Click Fibersim on the Fibersim ribbon menu.

It will start Fibersim window as shown in Figure 1–3.

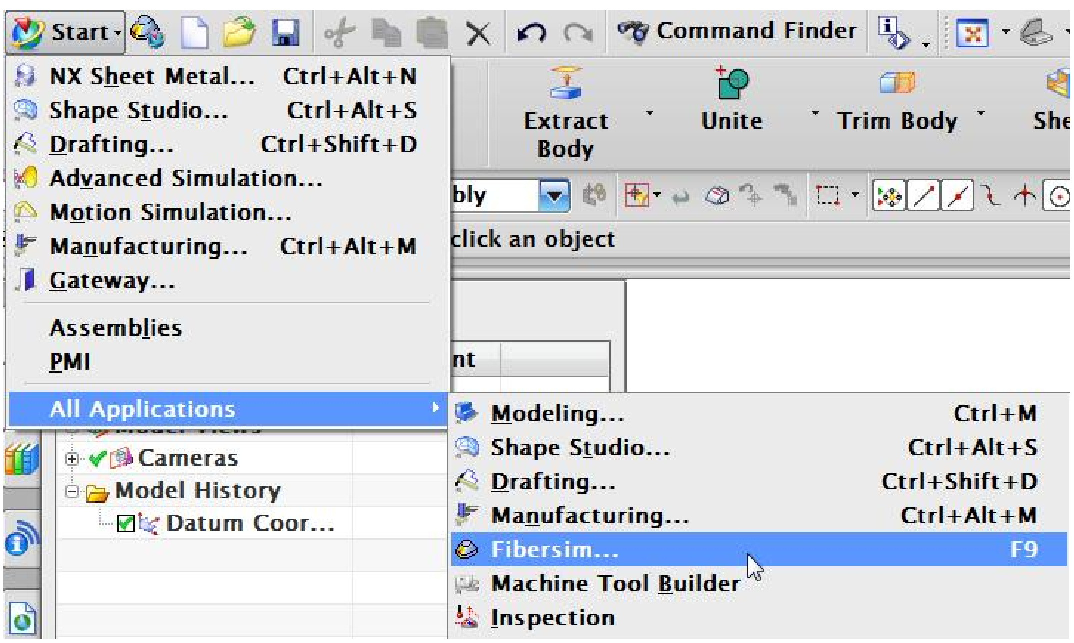


Figure 1–1

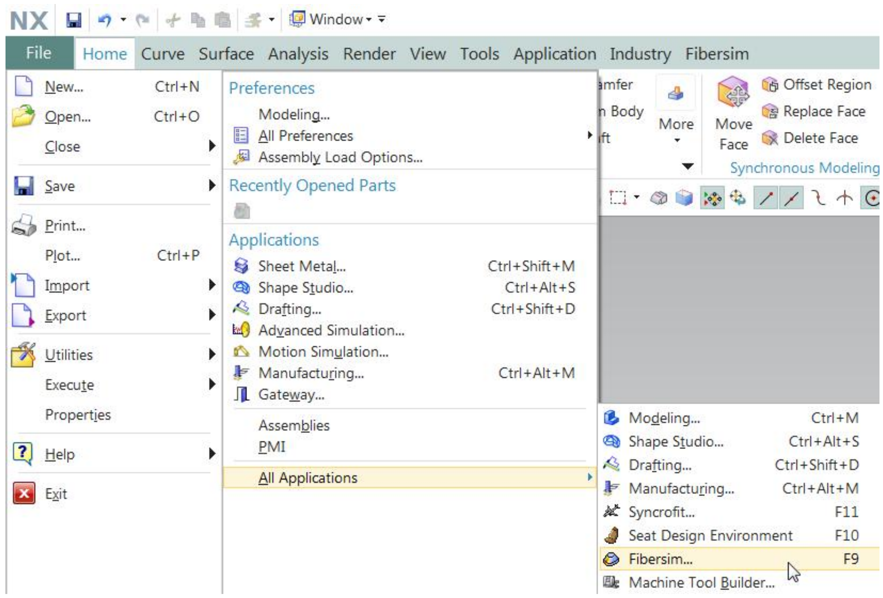


Figure 1–2

User Interface.

Figure 1–3 shows the Fibersim user interface when nothing is selected.

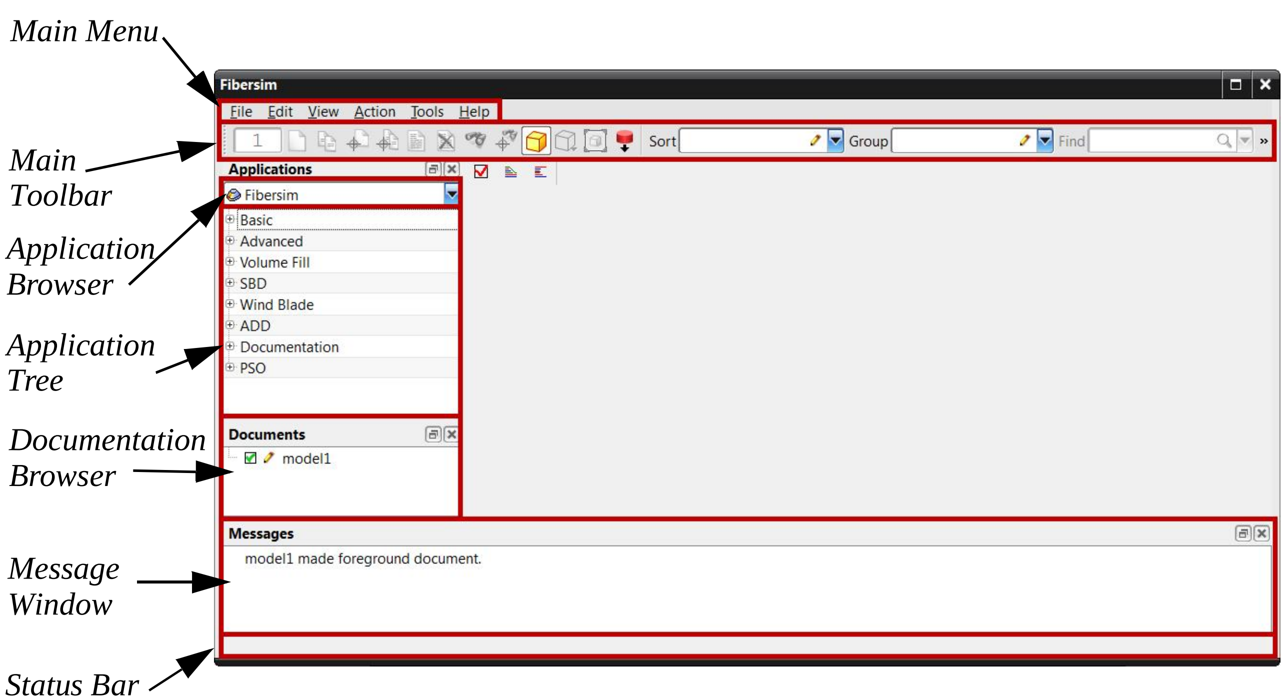


Figure 1–3

- Main Menu: These pull-down menus consist of import, export, basic editing tools, and utilities.
- Main Toolbar: This toolbar consists of icons to evoke common commands, such as Create New, Modify, Delete. and etc.
- Application Browser: This browser enables you to choose what you want to see in the Application Tree. You may choose *Fibersim,* which will display all available applications within Fibersim as shown in Figure 1–3, or choose a specific application.
- Application Tree: This tree displays applications available in Fibersim and components within each application. For more detailed explanation, see Fibersim User Guide *1.2 Fibersim User Interface*.
- Documentation Browser: This browser lists parts that exist in the current file. Commands related to documentation can be performed from right-click menu in this browser.
- Messages Window: This windows enables you to see what has been done in the current session of the file.
- Status Bar: This bar displays short messages and status of the commands you perform. It also displays progress while Fibersim performs tasks, such as highlighting objects, calculating producibility, creating flat patterns, and etc.

When an object is selected from the Application Tree, the Fibersim user interface updates. Based on what is selected, it displays commands that can be performed with the selected objects as shown in Figure 1–4.

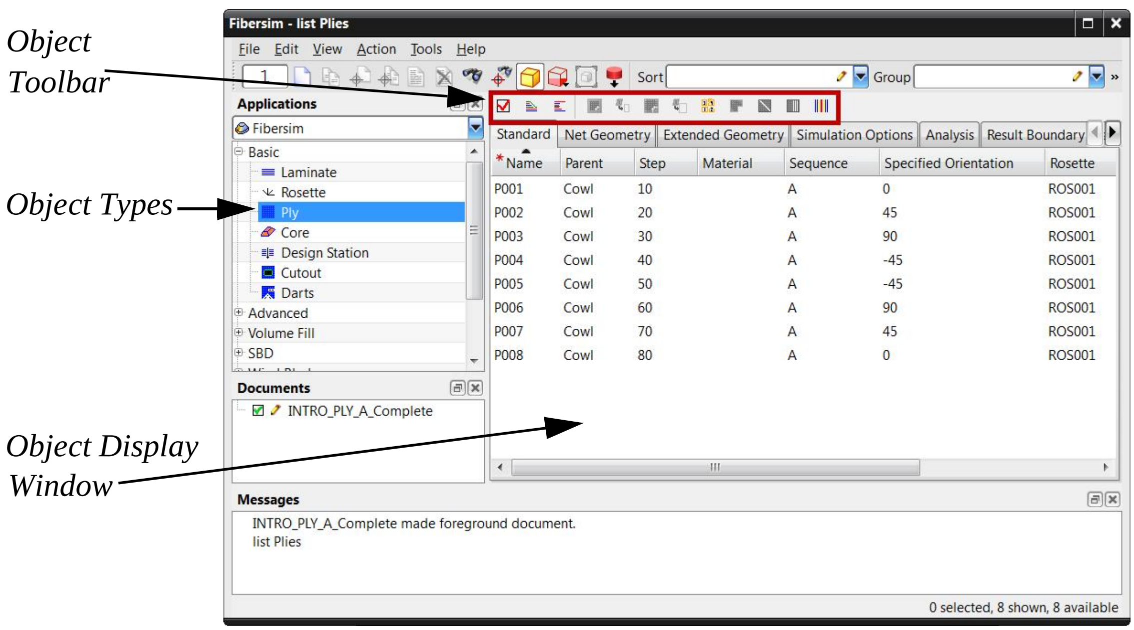


Figure 1–4

Add, Remove, Sort, and Move Columns

When a Fibersim component is selected from the Application Tree, list of objects for the selected component appears in the Object Display Window. The columns displayed at the top of the list can be used to sort the list. You can also add or move the columns.

- If you want to sort the list based on a column, click on the header of the column. A small arrow appears at the top of the column header to represent ascending/descending order as shown in Figure 1–5. Click on the header again to toggle between ascending and descending order.

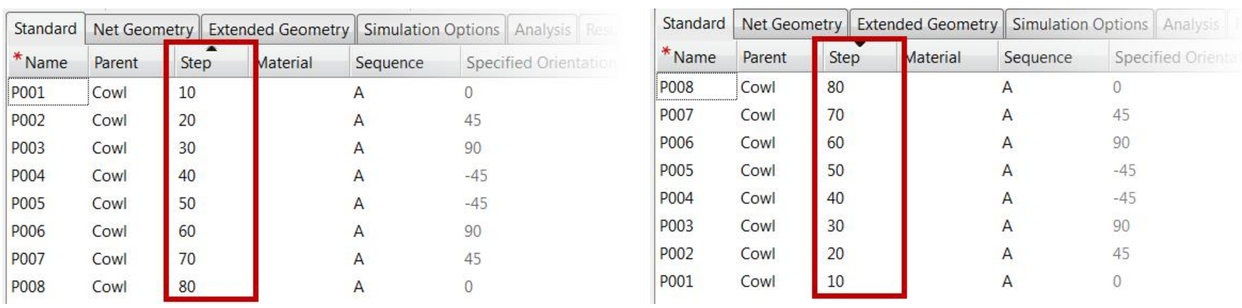


Figure 1–5

- If you want to add or remove a column, right-click on one of the column headers. The right-click menu should appear as shown in Figure 1–6. The check marks represent the columns currently displayed. Click on the items to add or remove them from the Object Display Window.

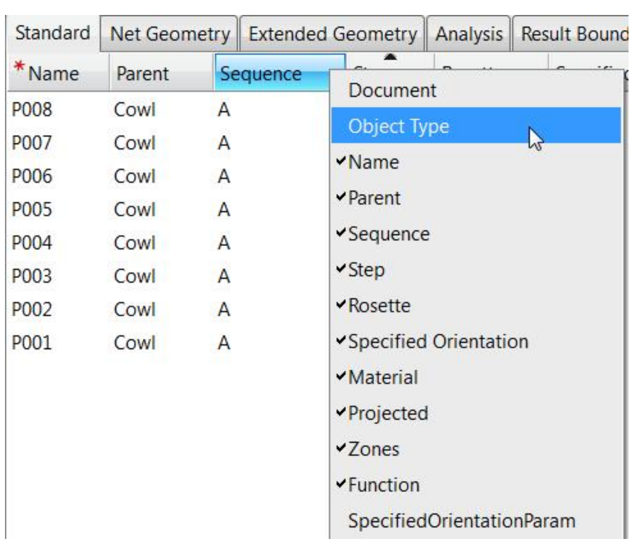


Figure 1–6

- If you want to move a column, click and hold on a column header. Drag it over to a desired location and release as shown in Figure 1–7.

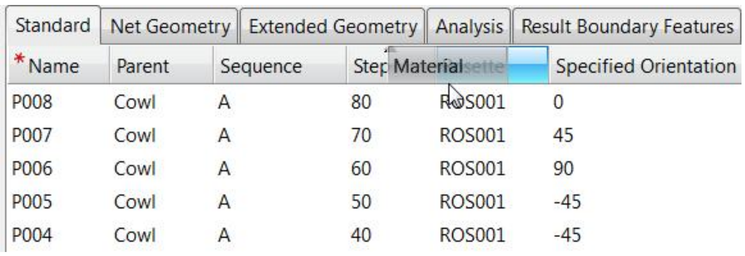


Figure 1–7

Sort, Group, and Find

Sort, Group, and Find filters located in the Main Toolbar are powerful tools that can help you to work efficiently in Fibersim.

- If you want to sort the list based on one member, it would be more efficient to sort with columns. However, if you want to sort the list of items based on more than one member, use this compound Sort tool:
  - By default, *Sequence, Step and Name* sort option exists. This option will sort the list based on Sequence, Step and Name in order.
  - You can create custom Sorts. To create a new Sort:

1. Click on  in the Sort field.
2. Enter a name for the Sort. Typically, it is a good idea to use a descriptive and unique name, such as *Orientation and Name* to indicate that it sorts by Orientation first then by Name.
3. Click on  (Create Sort).
4. Select parameters for *1st, 2nd, and 3rd Member* and the order as shown in Figure 1–8.
5. Click OK.

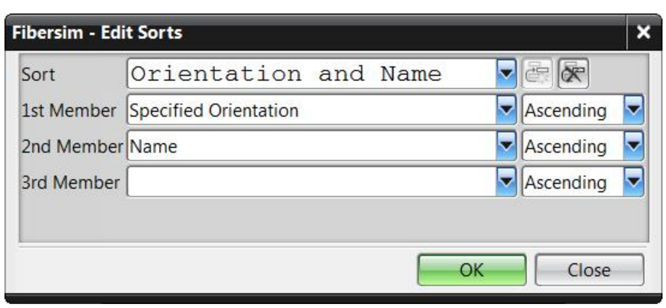


Figure 1–8

- If you want to edit objects that share common attributes, such as Specified Orientation, Zones, or Material, you can use the Group tool. There are some default groups already created, but you can create custom Groups. To create a new Group:

    1. Click on  in the Group field.
    2. Enter a name for the Group.
    3. Click on  (Create Group).
    4. Select values for *1st, 2nd, and 3rd Member*.
    5. Click OK.

- The Find tool lets you filter the list of objects based on your input. Fibersim will find all objects that contain your input and filter out the rest. The Find tool is most helpful if you are looking for a specific object.
    - To find a ply named P006:

    1. Type [P006] in the Find toolbar and hit Enter.
    2. The object list should be updated as shown in Figure 1–9.

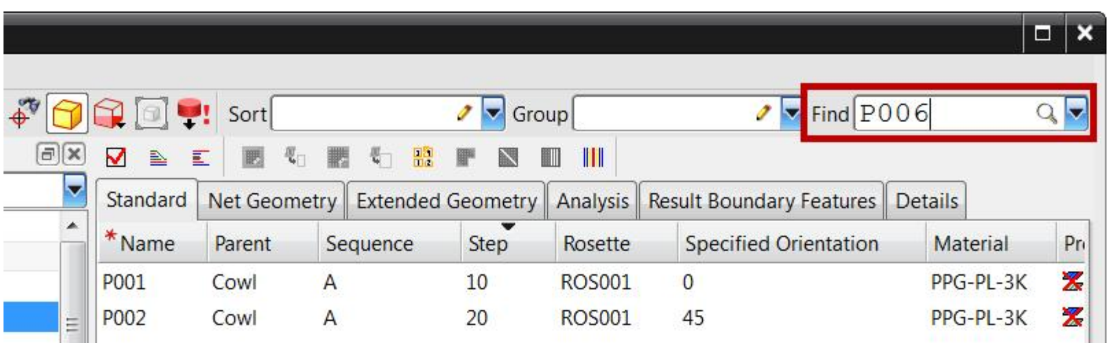


Figure 1–9

3. Note that the Status Bar at the bottom indicates the number of objects that are selected, shown, and available as shown in Figure 1–10.

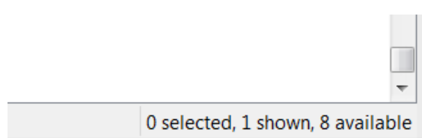


Figure 1–10

4. To show all objects again, right-click in the Object Display Window, and choose *Show Available*.

Active Laminate

Active Laminate option is new to Fibersim 15. It is located next to the Sort, Group, and Find options in the main toolbar as shown in Figure 1–11.

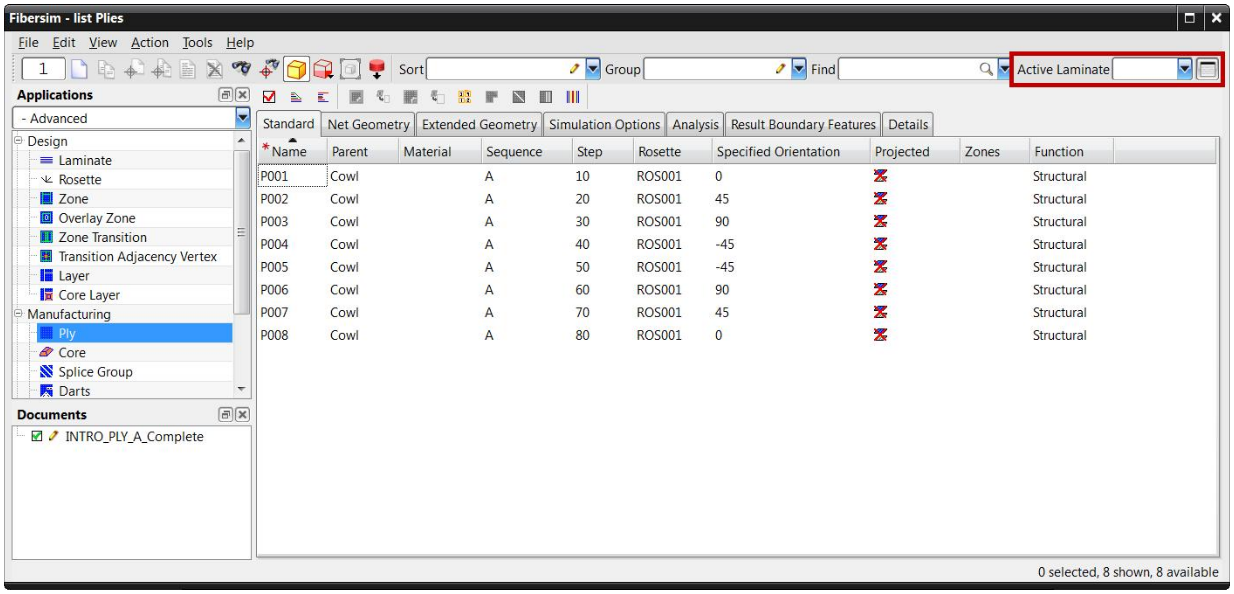


Figure 1–11

When a laminate is selected, only the objects that have the laminate as a parent are shown. For example, if you have multiple laminates and multiple plies that belong to those laminates, you can filter the list of plies by selecting an active laminate.

Selecting an active laminate will change the available value in the selected/shown/available status bar to match the number of objects that belong to the active laminate.

For certain cases (e.g. plies) objects can have another object as a parent. In that case, Fibersim will look at the parent's parent to find a laminate.

Highlighting Contexts in Fibersim

There are many different highlighting contexts in Fibersim, and they vary depending on the object displayed. For the ply object there are 7 separate contexts, as described below. To view the highlighting contexts, click on  in the Main Toolbar. You should see the options shown as Figure 1–12.

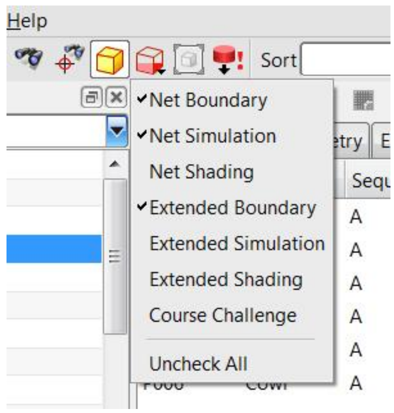


Figure 1–12

Figure 1–13 describes some of the different highlighting contexts for the ply object.

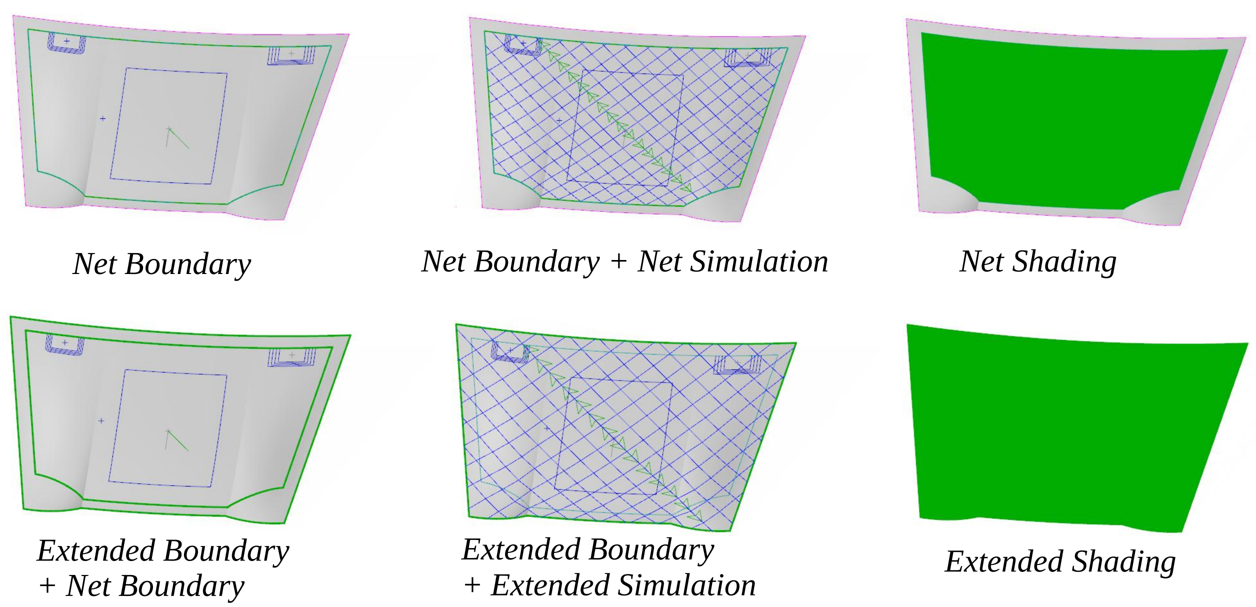


Figure 1–13

Options

There are three types of options in Fibersim; EnCapta options, Fibersim options - global, and Fibersim options - specific to part. They control different aspects of the application.

EnCapta Options control the User Interface of Fibersim. To access EnCapta options, select Tools > Options > EnCapta Options from the Fibersim window. EnCapta options dialog box should appear as shown in Figure 1–14.

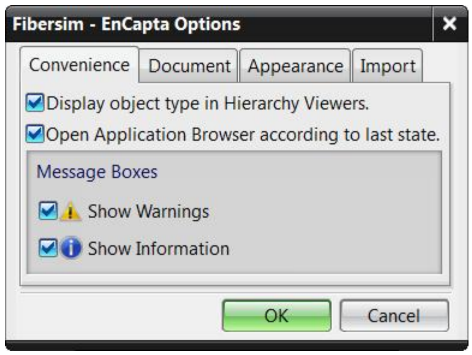


Figure 1–14

Fibersim Options have some controls for global settings and others for part specific. To access Fibersim Options, select **Tools > Options > Fibersim Options** from the Fibersim Window. Fibersim options dialog box should appear as shown in Figure 1–15.

As noted in Figure 1–15, the top section of the dialog box has global setting controls i.e. display colors for Fibersim components and zone text display options. Part-specific settings are located at the bottom of the dialog box.

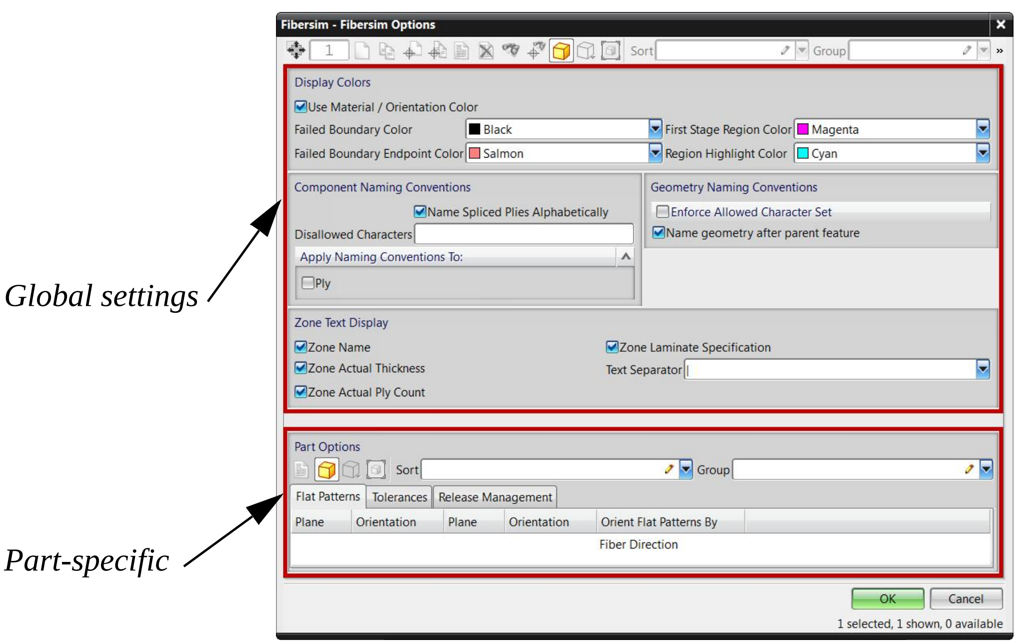


Figure 1–15

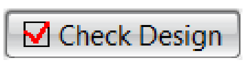
Design Checker

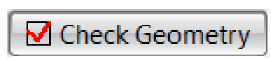
The Design Checker is a utility that can be used to check the basic geometry and design of the composite part for an error.

The Geometry check portion of the Design Checker will validate boundaries, holes, and origins on all appropriate objects; Laminate, Ply, Core, Zone, Layer, Core Layer, and etc. The Design check portion of the utility will run producibility on all the plies and check that plies do not exceed material width. In addition, Fibersim will report if the ply does not have an up-to-date flat pattern.

To run the Design Checker:

1. Click  (Design Checker) located on the object toolbar.
2. Choose the appropriate tab.
3. Click on  (Link with Link Dialog).
4. Link all the Laminates to run the Design Checker on.
5. Click 

.
6. Click 

or 

depending on the selected tab.
7. View Report for any errors and messages.

Actions: Ply Table - HTML and Ply Table - CSV

Actions are customizable operations that can be run on any Fibersim data in a given model. By default Fibersim comes with several pre-prepared actions, including two different Ply Table reports, as described below.

Ply Table - HTML and Ply Table - CSV commands are quick and simple way to export the ply table to either HTML or CSV format. They are available from *Action* Main Menu. The result of Ply Table - HTML is shown in Figure 1–16.

Composite Part Ply Table

Ply Name	Laminate	Sequence	Step	Material	Rosette	Orientation	Cured Thickness
P001	Hood	A	10	PPG-8H-3K	ROS001	45	0.0135
P002	Hood	A	20	PPG-8H-3K	ROS001	0	0.0135
P003	Overcore 1	B	40	PPG-8H-3K	ROS001	90	0.0135
P004	Overcore 1	B	50	PPG-8H-3K	ROS001	-45	0.0135
P005	Overcore 1	B	65	PPG-8H-3K	ROS001	0	0.0135
P006	Overcore 1	B	65	PPG-8H-3K	ROS001	0	0.0135
P007	Overcore 1	B	70	PPG-8H-3K	ROS001	45	0.0135
P008	Overcore 1	B	80	PPG-8H-3K	ROS001	0	0.0135
P009	Overcore 1	B	90	PPG-8H-3K	ROS001	90	0.0135
P010	Overcore 1	B	100	PPG-8H-3K	ROS001	-45	0.0135

Figure 1–16

Window Snapping

When working in Fibersim, the program may open multiple windows. The Window Snapping feature will help you manage these windows better.

Click (Move Dialog with Parent Dialogs), located at the top left corner of a window as shown in Figure 1–17.

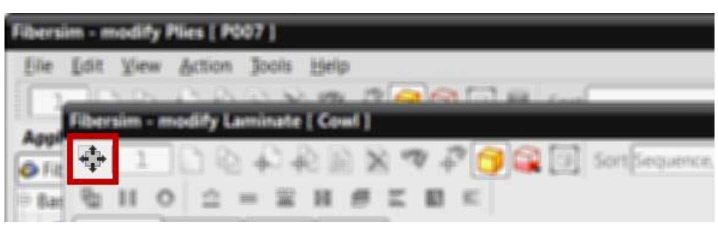


Figure 1–17

When the top dialog window is moved around, the parent dialog window should move with it.

To Un-snap the windows, click (Move Dialog with Parent Dialogs) again.

Chapter 2

Model Setup

This chapter includes:

- ✓**Exercise 2a: [NX8.5] Create a Model from an Extended Surface and Planes**
- ✓**Exercise 2b: [NX9/10] Create a Model from an Extended Surface and Planes**
- ✓**Exercise 2c: [NX8.5] Create a Model from a Solid**
- ✓**Exercise 2d: [NX9/10] Create a Model from a Solid**
- ✓**Exercise 2e: NX Model Setup from Planes and Surface (Optional)**

Exercise 2a

[NX8.5] Create a Model from an Extended Surface and Planes

User Guide Reference:
2.1 Introduction
2.2 CEE Organization
2.3 Understanding CEE Data Hierarchy
2.4 Defining Boundaries for CEE Objects
2.6 Laminates
2.7 Rosettes

In this exercise, you will prepare a NX model for composite design with Fibersim from an extended surface and planes using NX8 or NX8.5. The completed model displays as shown in Figure 2–1.

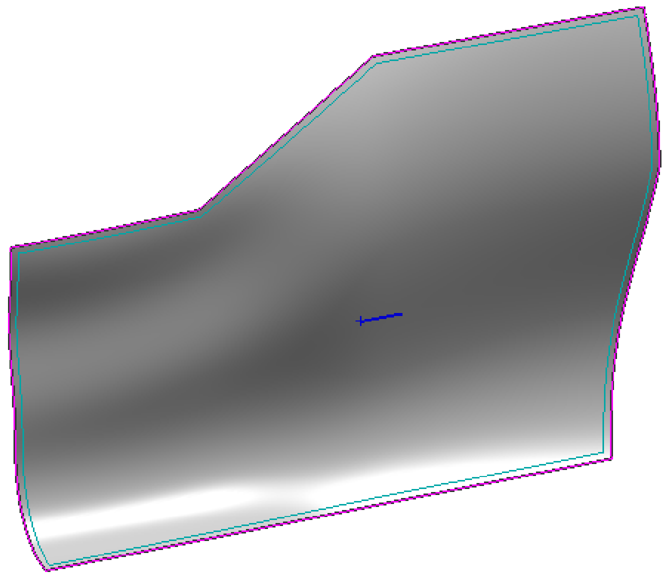


Figure 2–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Create Net Boundary geometry**
- ✓ **Create Extended Boundary geometry**
- ✓ **Create a Tool Surface**
- ✓ **Create Rosette geometry**

Estimated Time

10 min

Task 1 - Open a part.

1. Open **MODEL_SETUP_A.prt**. The model displays as shown in Figure 2–2.

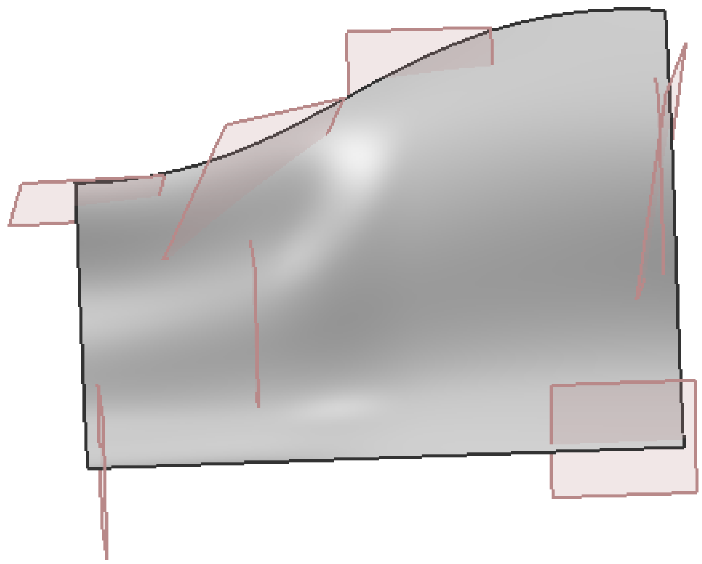


Figure 2–2

Task 2 - Create Net Boundary geometry.

1. Select **Insert > ~~Curve from Bodies~~ > Intersect** as shown in Figure 2–3.

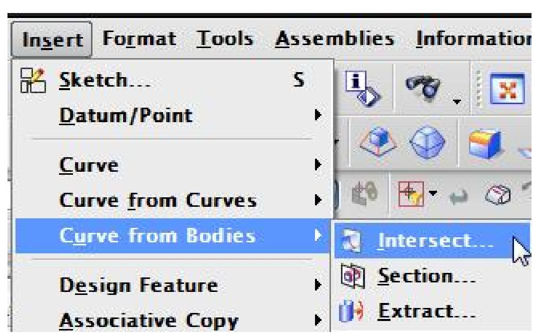


Figure 2–3

2. For the *Set 1* face, select **Loft_Surface**.

3. Select all six planes displayed in the model by making a selection box around the part as shown in Figure 2–4.

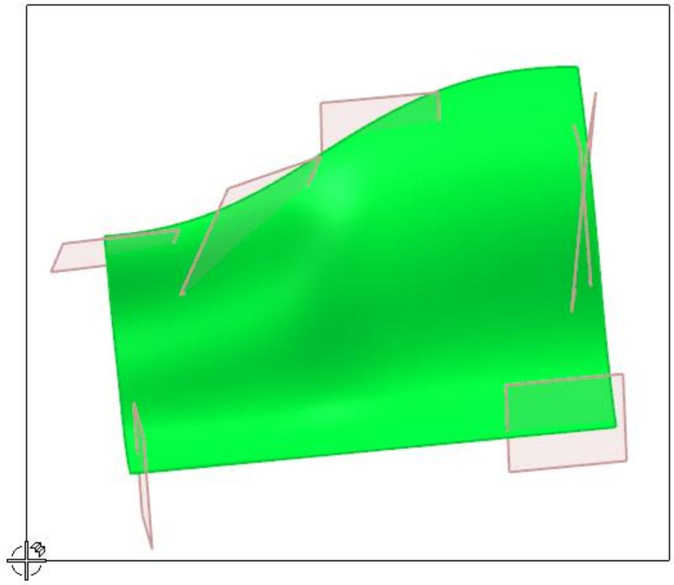


Figure 2–4

4. Click < OK >.

5. Select **Insert > Associative Copy > Extract Geometry** as shown in Figure 2–5.

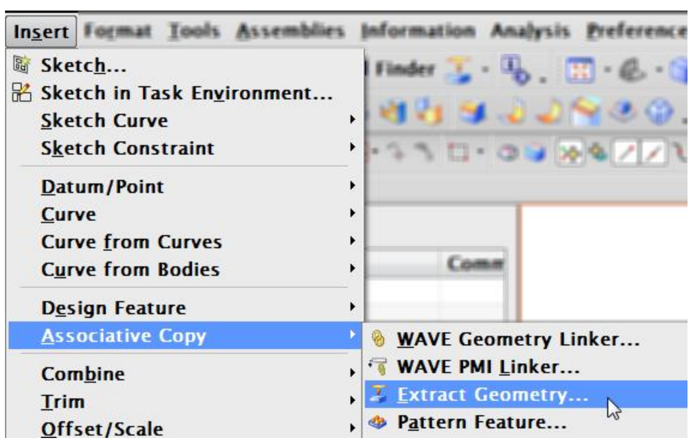


Figure 2–5

6. Select **Composite Curve** for *Type*.

7. Ensure that the **Stop at Intersection** ( ) option is enabled as shown in Figure 2–6.

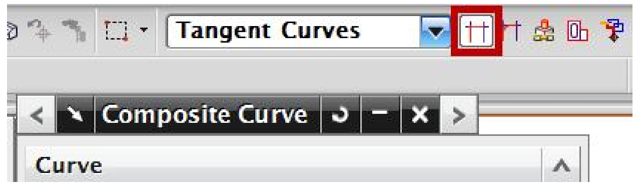


Figure 2–6

8. Select the curves in the location and order shown in Figure 2–7.

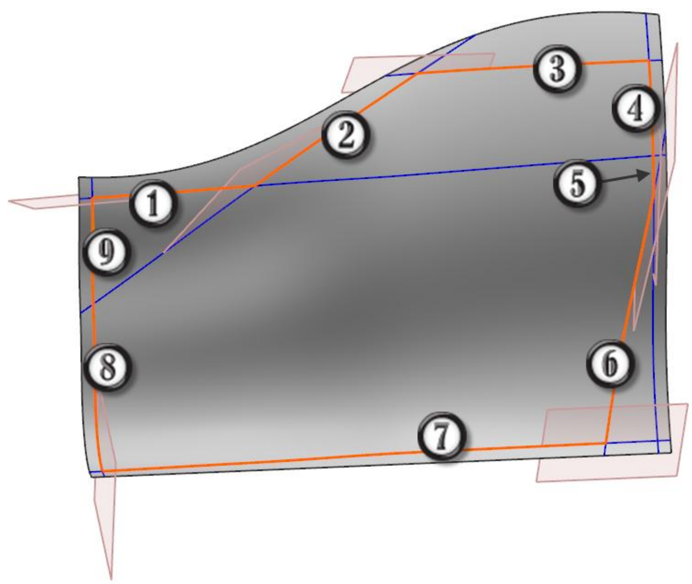


Figure 2–7

9. Under *Settings*, select **Hide Original** option and **General** for *Joint Curves*. as shown in Figure 2–8.

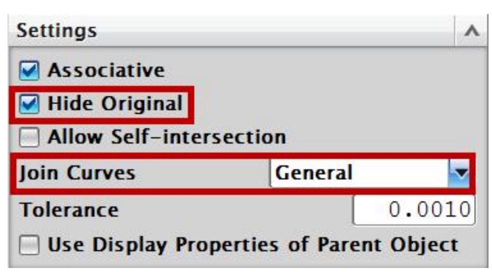


Figure 2–8

10. Click < OK >.

11. Select **Curve** from the *Selection Filter* list as shown in Figure 2–9.

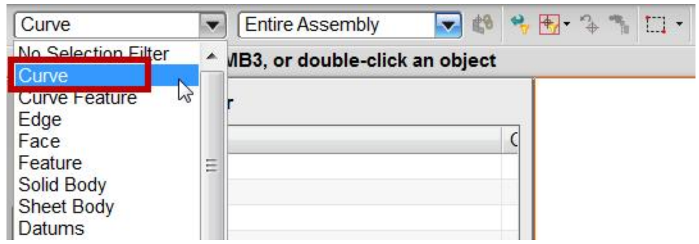


Figure 2–9

12. Select the composite curve in the model.
13. Right-click and select **Properties**.
14. Click on the *General* tab.
15. Enter [Ext_Boundary] for *Name* and *Feature Name*.
16. Click  .
17. Select the composite curve again.
18. Right-click and select **Edit Display**.
19. Change the color to **Magenta** (Color ID: 181) and width to **0.13mm** as shown in Figure 2–10.

The thinnest line width is used to minimize conflicts with Fibersim highlighting.

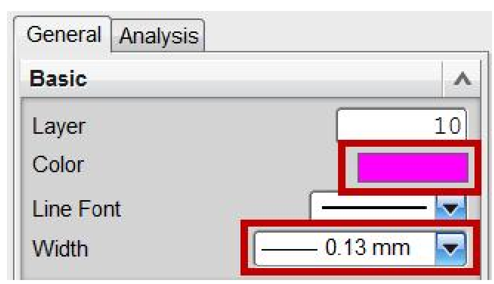
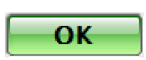


Figure 2–10

20. Click  .

Task 3 - Create Extended Boundary geometry.

1. Select **Insert > ~~Curve from Curves~~ > Offset in Face**.
2. Select **Ext_Boundary** for *Curve*.
3. Select **Loft_Surface** for *Plane*.

4. Enter [0.5] (12.7mm) for *Section1:Offset1* as shown in Figure 2–11. Click on  (Reverse Direction), if necessary, to match the Figure 2–11.

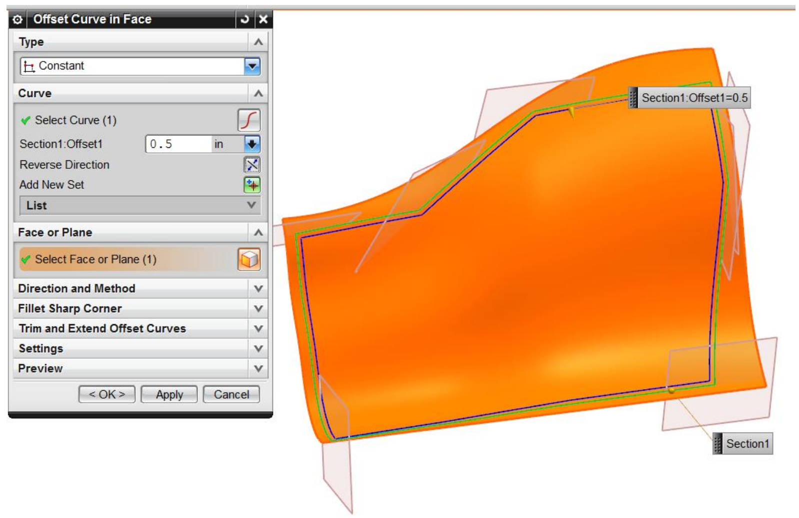


Figure 2–11

*If the **Offset in Face** command fails, flip the curve direction of the composite curve or change the Tolerance to **0.01**.*

5. Under Settings, select **General** for *Join Curves* field as show in Figure 2–12.

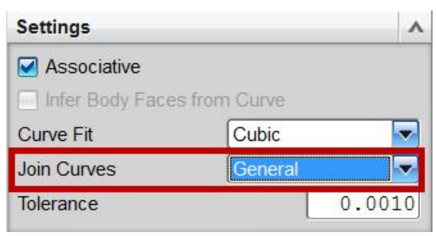
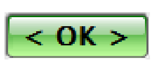
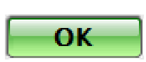


Figure 2–12

6. Click .

7. Select the Spline of the Offset in Face.
8. Right-click and select **Properties**.
9. Click on the *General* tab.
10. Enter [Net_Boundary] for *Name* and *Feature Name*.
11. Click .

12. Select **Net_Boundary**.
13. Right-click and select **Edit Display**.

14. Change the color to **Cyan** and width to **0.13 mm** as shown in Figure 2–13.

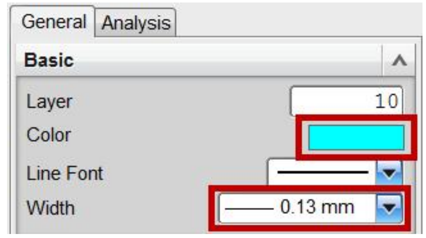


Figure 2–13

*Alternatively, you can select **Format > Layer Settings** to open Layer Settings.*

15. Press <*Ctrl*> + <*L*> to open Layer Settings and hide MASTER_LINES layer category. (Make sure that Category Display is enabled.)

16. Click Close.

Task 4 - Create a Tool Surface.

1. Select **Insert > Trim > Trimmed Sheet** as shown in Figure 2–14.

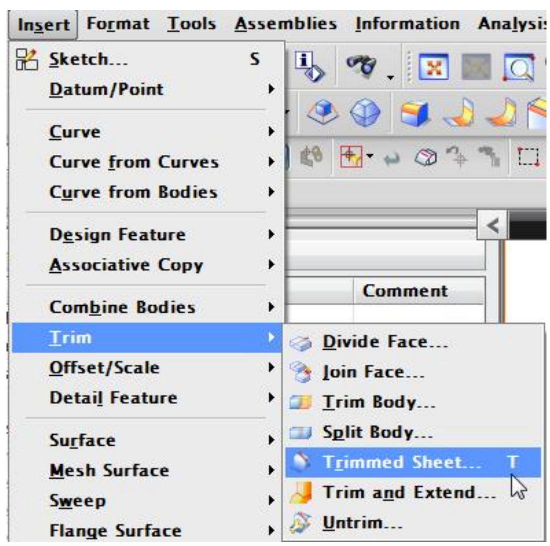


Figure 2–14

2. Select **Loft_Surface** for *Target* and **Ext_Boundary** for *Boundary Objects*.

3. Select the appropriate option for *Region* to keep the geometry shown as in Figure 2–15.

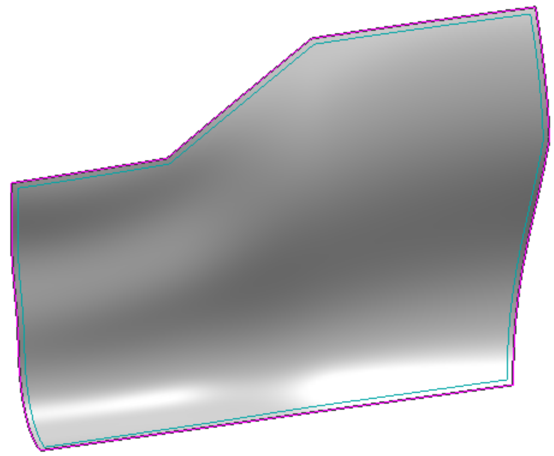


Figure 2–15

4. Click **OK**.

5. Select **Edit > Surface > Reverse Normal**. Fibersim uses the tool surface normal to determine the direction of the layup (male or female tool). To ensure the correct normal direction on the tool surface, the recommended practice is to use the **Reverse Normal** command.

6. Select **LOFT_SURFACE** and ensure that the normal direction points -Z direction as shown in Figure 2–16. (Press <W> to display WCS)

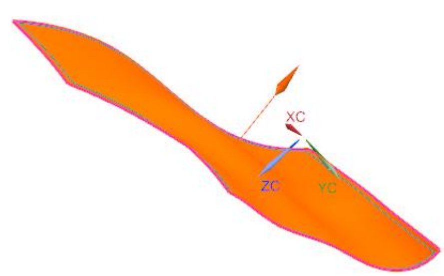
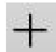
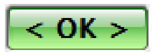


Figure 2–16

7. Since the surface normal is pointing to the desired direction, click **Cancel**.

Task 5 - Create Rosette geometry.

1. Click  (Point).
2. In the Point type drop-down list, select **Point on Face**.
3. Indicate a placement point somewhere in the middle of the Tool Surface.
4. Click 

.
5. Select **Point** from the *Selection Filter* list.
6. Select the created point.
7. Right-click and select **Properties**.
8. Enter [Rosette_Origin] for *Name* and *Feature Name*.
9. Click  (Line).
10. Select the following:
    - Start Option: **Point > select Rosette_Origin**
    - End Option: **Along XC**
    - Start Limit: **At Point**
    - Distance: **0 in (0 mm)**
    - End Limit: **Value**
    - Distance: **3 in (76.2 mm)**

The Line Definition dialog box updates as shown in Figure 2–17.

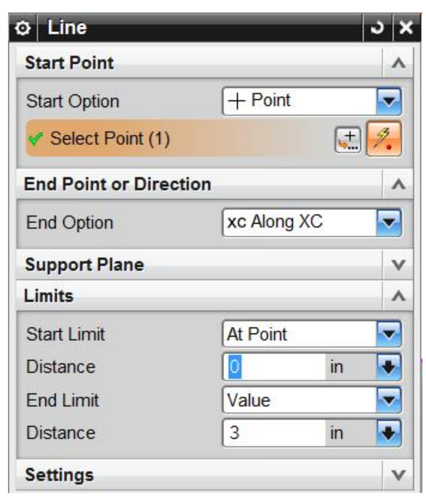


Figure 2–17

11. Click < OK >.
12. Select **Curve** from the *Selection Filter* list.
13. Select the created line.
14. Right-click and select **Properties**.
15. Enter [Zero_Direction] for *Name* and *Feature Name*.
16. Click OK.

Exercise 2b [NX9/10] Create a Model from an Extended Surface and Planes

User Guide Reference:
2.1 Introduction
2.2 CEE Organization
2.3 Understanding CEE Data Hierarchy
2.4 Defining Boundaries for CEE Objects
2.6 Laminates
2.7 Rosettes

In this exercise, you will prepare a NX model for composite design with Fibersim from an extended surface and planes using NX9. The completed model displays as shown in Figure 2–18.

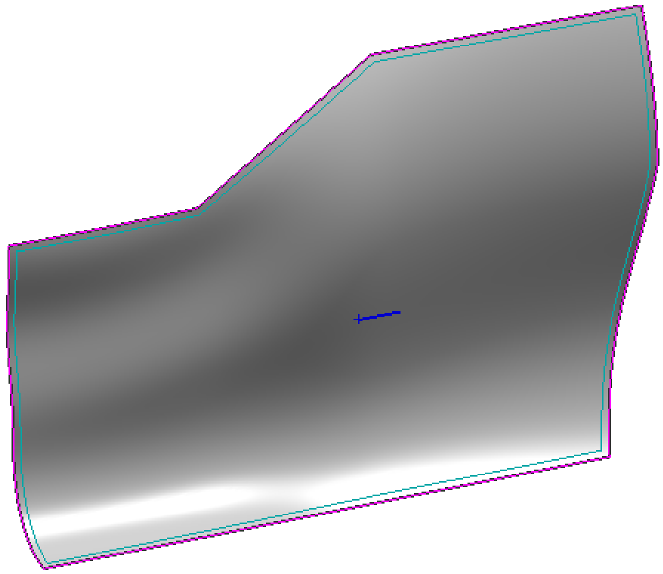


Figure 2–18

Goal

After you complete this exercise, you will be able to:

- ✓ **Create Net Boundary geometry**
- ✓ **Create Extended Boundary geometry**
- ✓ **Create a Tool Surface**
- ✓ **Create Rosette geometry**

Estimated Time

10 min

Task 1 - Open a part.

1. Open **MODEL_SETUP_A.prt**. The model displays as shown in Figure 2–19.

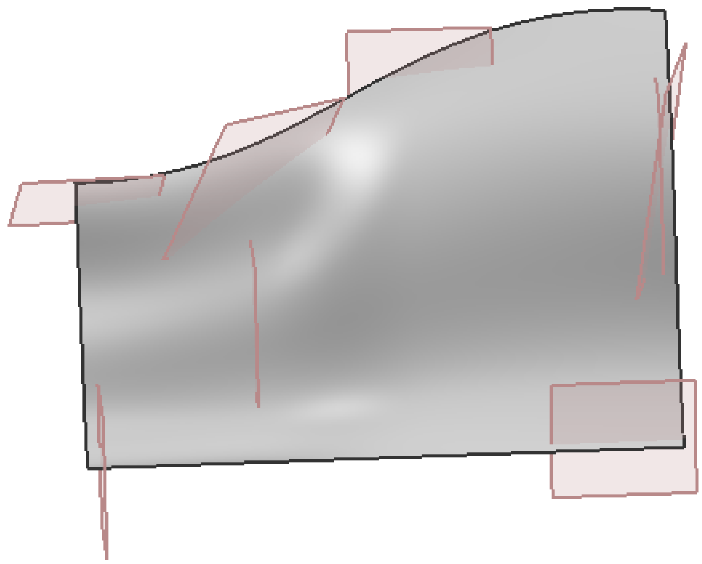


Figure 2–19

Task 2 - Create Net Boundary geometry.

1. Select **Intersection Curves** from the *Curves* tab as shown in Figure 2–20.

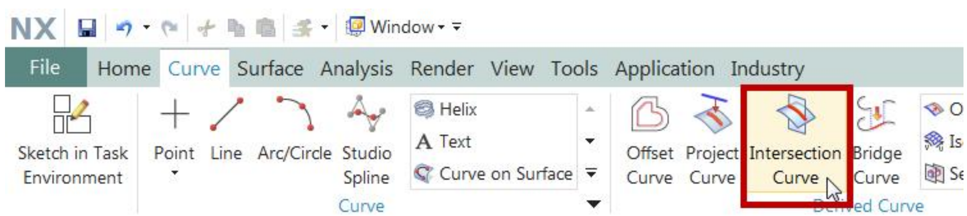


Figure 2–20

2. For the *Set 1* face, select **Loft_Surface**.

3. For the *Set 2* face, select all six planes displayed in the model by making a selection box around the part as shown in Figure 2–21.

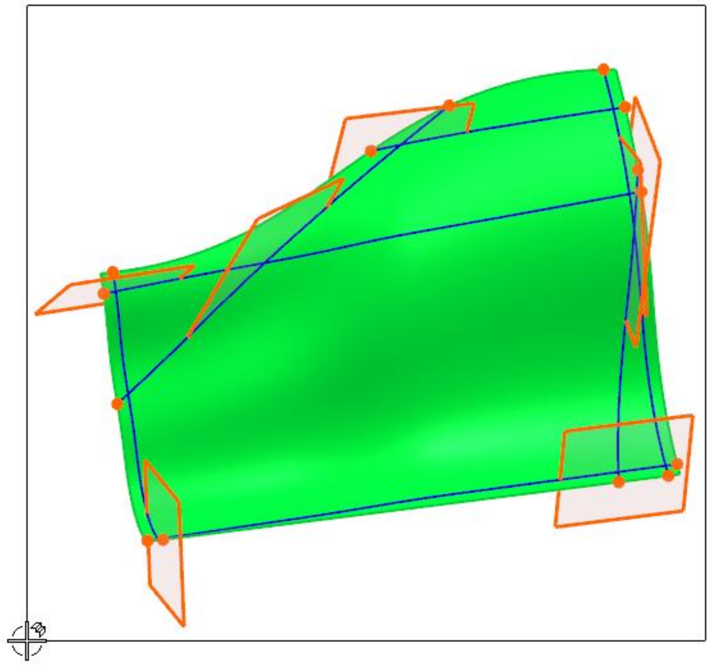


Figure 2–21

4. Click < OK >.

If you do not see ***Extract Geometry*** *under* ***More****, make sure that you are using* ***Advanced Role*** *on the Roles tab.*

5. Select **More > Extract Geometry** from the *Home* tab as shown in Figure 2–22.

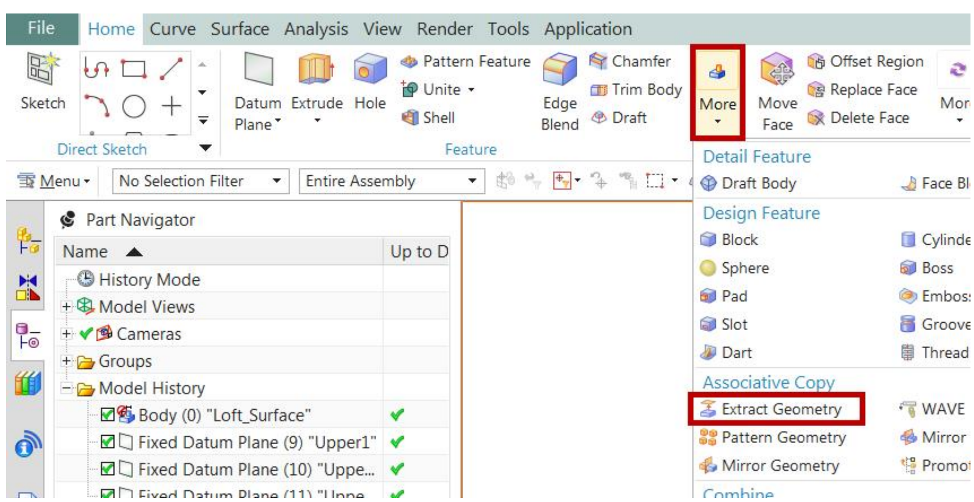


Figure 2–22

6. Select **Composite Curve** for *Type*.

7. Ensure that the **Stop at Intersection** ( ) option is enabled as shown in Figure 2–23.

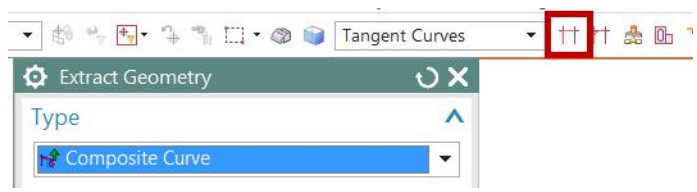


Figure 2–23

8. Select the curves in the location and order shown in Figure 2–7.

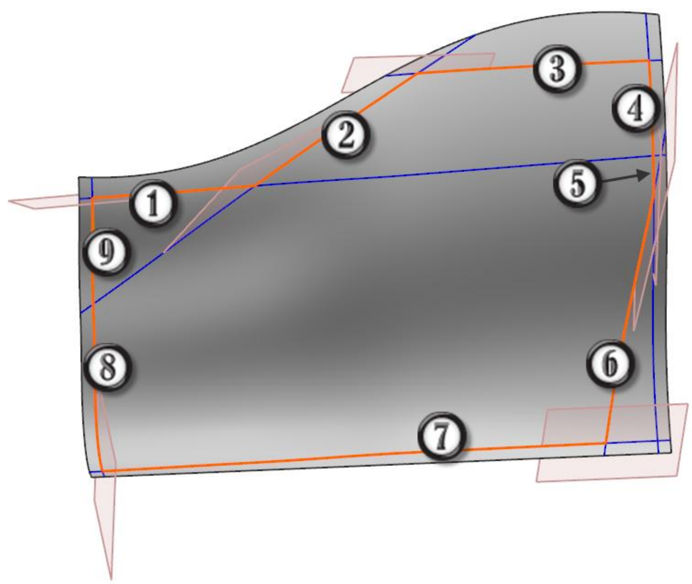


Figure 2–24

9. Under *Settings*, select **Hide Original** option and **General** for *Joint Curves*. as shown in Figure 2–25.

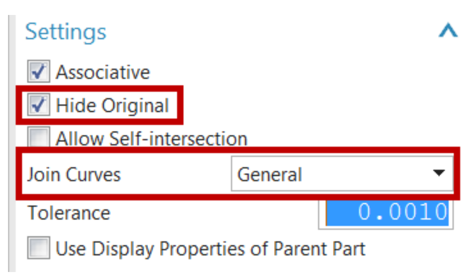


Figure 2–25

10. Click < OK >.

11. Select **Curve** from the *Selection Filter* list as shown in Figure 2–26.

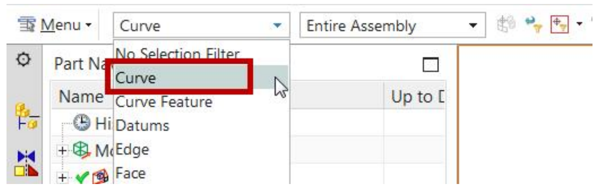


Figure 2–26

12. Select the composite curve.

13. Right-click on the curve and select **Properties**.

14. Select the *General* tab.

15. Enter [Ext_Boundary] for *Name* and *Feature Name*.

16. Click OK.

17. Select the composite curve again.

18. Right-click and select **Edit Display**.

19. Change the color to **Magenta** (Color ID: 181) and width to **0.13 mm** as shown in Figure 2–27.

The thinnest line width is used to minimize conflicts with Fibersim highlighting.

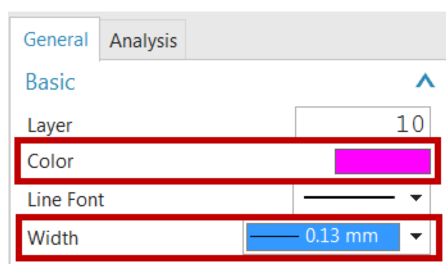


Figure 2–27

20. Click OK.

Task 3 - Create Extended Boundary geometry.

1. Select **Offset Curve in Face** from the *Curve* tab as shown in Figure 2–28.

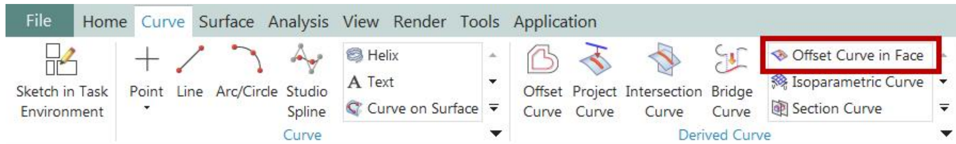
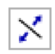


Figure 2–28

2. Select **Ext_Boundary** for Curve.
3. Select **Loft_Surface** for Face or Plane.
4. Enter [0.5] (12.7mm) for *Section1:Offset1* as shown in Figure 2–29. Click on  (Reverse Direction), if necessary, to match the Figure 2–29.

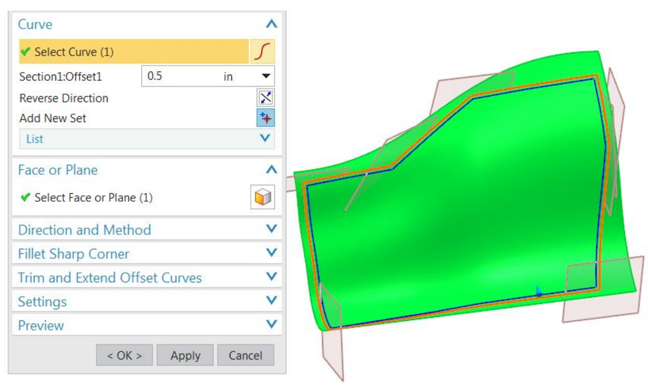


Figure 2–29

*If the **Offset in Face** command fails, flip the curve direction of the composite curve or change the Tolerance to **0.01**.*

5. Under Settings, select **General** for *Join Curves* field as show in Figure 2–30.

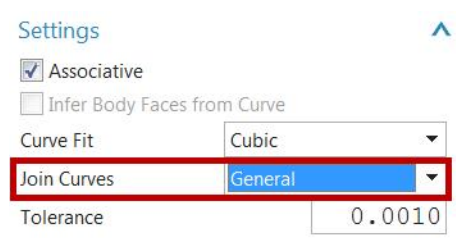
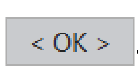


Figure 2–30

6. Click .

7. Select the curve of the Offset in Face.
8. Right-click and select **Properties**.
9. Select the *General* tab.
10. Enter [Net_Boundary] for *Name* and *Feature Name*.
11. Click OK.
12. Select the curve of the Offset in Face again.
13. Right-click and select **Edit Display**.
14. Change the color to **Cyan** (Color ID: 31) and width to 0.13 mm as shown in Figure 2–31.

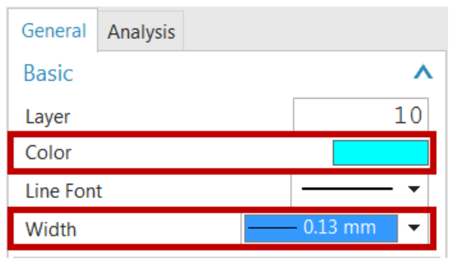


Figure 2–31

*Alternatively, you can select **Menu > Format > Layer Settings** to open Layer Settings.*

15. Press <Ctrl> + <L> to open the Layer Settings window.
16. Hide **MASTER_LINES** layer category. (Make sure that Category Display is enabled.)
17. Click Close.

Task 4 - Create a Tool Surface.

1. **[NX9]** Select **Trim~~med~~ Sheet** on the *Surface* tab as shown in Figure 2–32.

Figure 2–32

Hide all the Datum Planes from the Part Navigator.

1. **[NX10]** Select **Trim Sheet** on the *Surface* tab as shown in Figure 2–33.

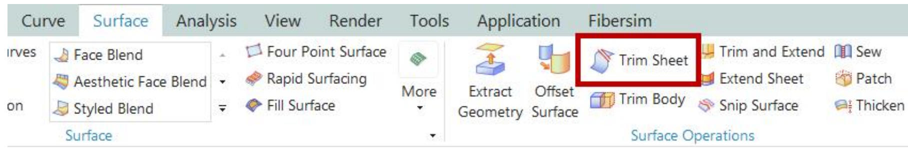


Figure 2–33

2. Select **Loft_Surface** for *Target* and **Ext_Boundary** for *Boundary Objects*.

3. Select the appropriate option for *Region* to keep the geometry shown as in Figure 2–34.

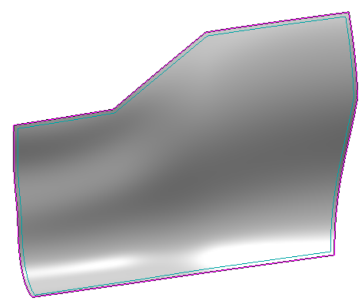


Figure 2–34

4. Click OK.

5. Select **Reverse Normal** from the *Surface* tab as shown in Figure 2–35. Fibersim uses the tool surface normal to determine the direction of the layup (male or female tool). To ensure the correct normal direction on the tool surface, the recommended practice is to use the **Reverse Normal** command.

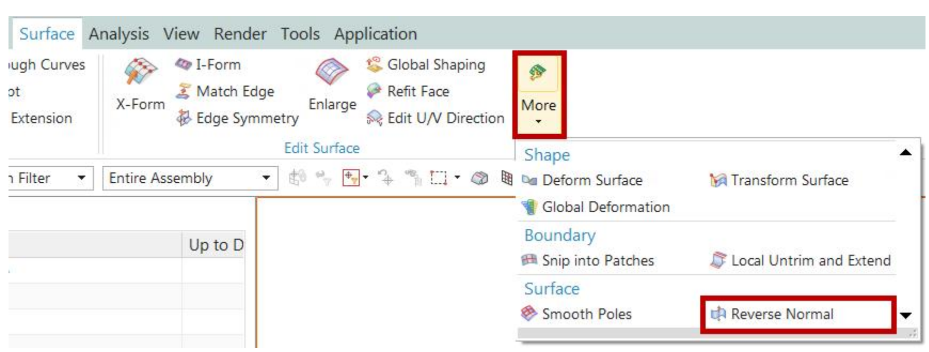


Figure 2–35

6. Select **Loft_Surface** and ensure that the normal direction points -Z direction as shown in Figure 2–36. (Press <W> to display WCS)

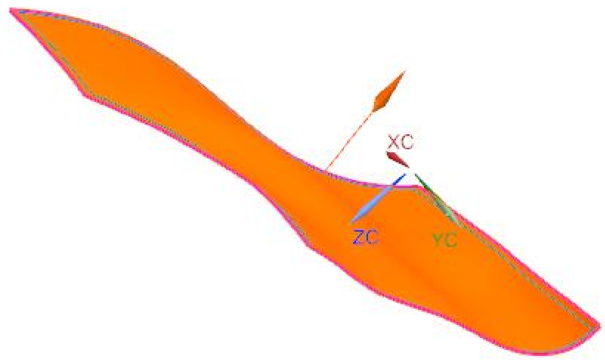


Figure 2–36

7. Since the surface normal is pointing to the desired direction, click Cancel.

Task 5 - Create Rosette geometry.

1. Click Point (Point) from the *Curve* tab.
2. In the Point type drop-down list, select **Point on Face**.
3. Indicate a placement point somewhere in the middle of the Tool Surface.
4. Click < OK >.
5. Select **Point** from the *Selection Filter* list.
6. Select the created point.
7. Right-click and select **Properties**.
8. Enter [Rosette_Origin] for *Name* and *Feature Name*.
9. Click Line (Line) from the *Curve* tab.

10. Select the following:

- Start Option: **Point >** select **Rosette_Origin**
- End Option: **Along XC**
- Start Limit: **At Point**
- Distance: **0 in (0 mm)**
- End Limit: **Value**
- Distance: **3 in (76.2 mm)**

The Line Definition dialog box updates as shown in Figure 2–37.

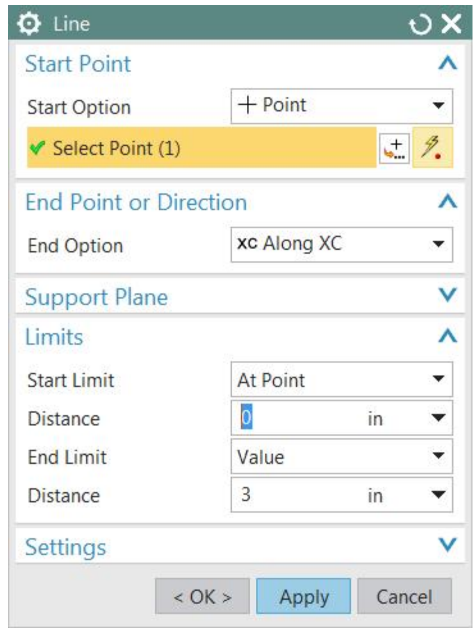


Figure 2–37

11. Click < OK >.

12. Select **Curve** from the *Selection Filter* list.

13. Select the created line.

14. Right-click and select **Properties**.

15. Enter [Zero_Direction] for *Name* and *Feature Name*.

16. Click OK.

Exercise 2c [NX8.5] Create a Model from a Solid

In this exercise, you will prepare a solid NX model for composite design with Fibersim using NX8 or NX 8.5. The completed model displays as shown in Figure 2–38.

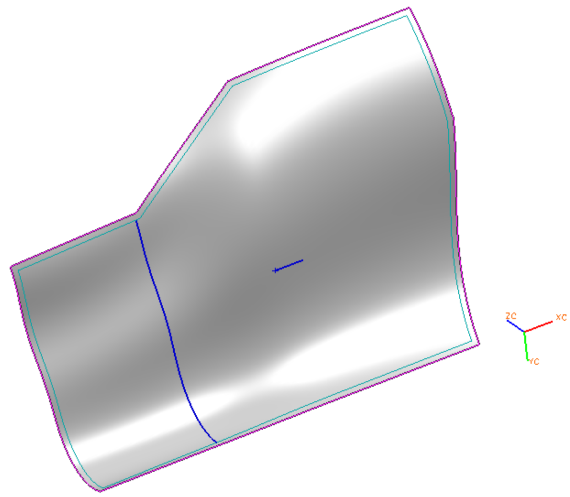


Figure 2–38

Goal

After you complete this exercise, you will be able to:

- ✓ **Extract a Tool Surface from a Solid**
- ✓ **Create Net Boundary geometry**
- ✓ **Extend a Tool Surface to account for manufacturing trim**
- ✓ **Create Extended Boundary geometry**
- ✓ **Extract a Ply Edge from a Solid**
- ✓ **Create Rosette geometry**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **MODEL_SETUP_B.prt**. The model displays as shown in Figure 2–39.

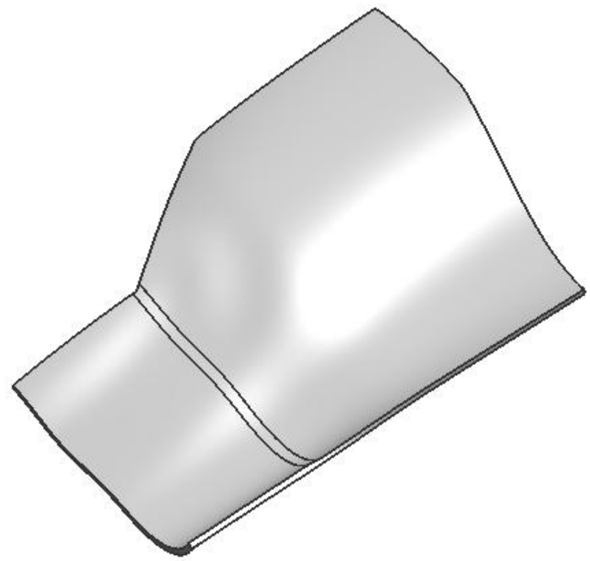


Figure 2–39

Task 2 - Extract Tool Surface from Solid.

1. Click  (Extract Geometry).
2. Select **Face** for *Extract Geometry Type* and **Single Face** under *Face Option* as shown in Figure 2–40.

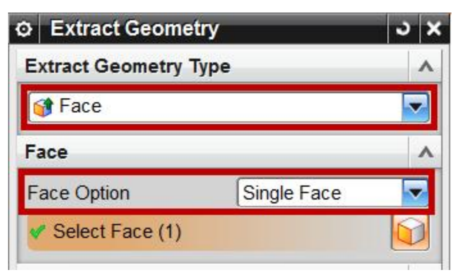


Figure 2–40

3. Select the rear surface of the solid as shown in Figure 2–41.

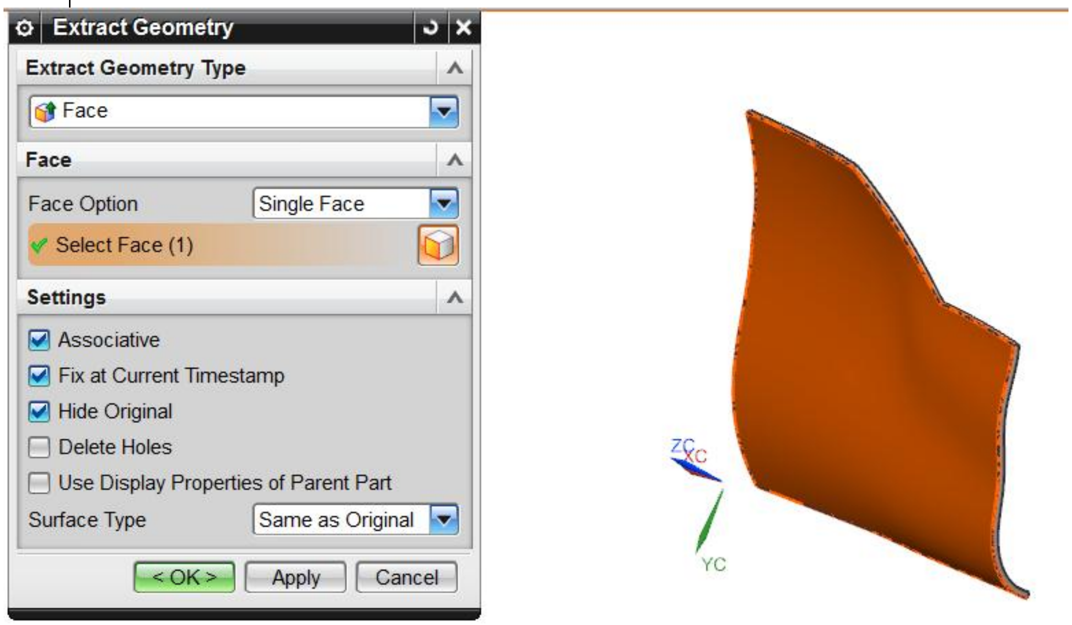


Figure 2–41

4. Click 

.

5. Select the extracted surface.

6. Right-click and select **Properties**.

7. Click on the *General* tab.

8. Enter [Tool_Surface] as shown in Figure 2–42.

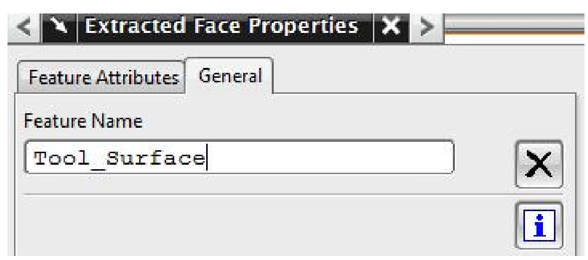
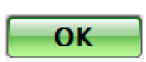


Figure 2–42

9. Click 

.

Task 3 - Create Net Boundary geometry.

1. Click (Extract Geometry).
2. Select **Composite Curve** for *Extract Geometry Type*.
3. ~~Select the 6 edges of the surface~~ and set *Joint Curves* to **General** as shown in Figure 2–43.

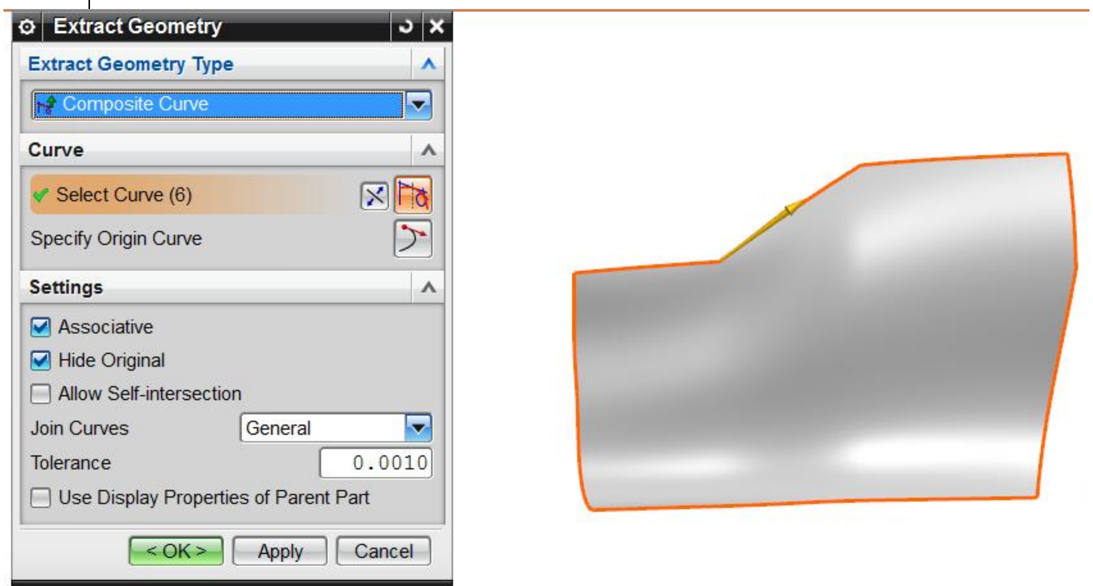


Figure 2–43

4. Click **< OK >**.
5. Select **Curves** for *Selection Filter*.
6. Select the composite curve.
7. Right-click and select **Properties**.
8. Enter [Net_Boundary] for *Name* and *Feature Name*.
9. Click **OK**.
10. Select **Net_Boundary**.
11. Right-click and select **Edit Display**.
12. Change the Color to **Cyan** (Color ID: 31) and Width to **0.13 mm**.
13. Click **OK**.

Task 4 - Create an Extended Tool Surface.

In this task, you will extend the net tool surface to provide a 0.6 in (15mm) material excess for the manufacturing trim.

1. Select **Insert > Trim > Trim and Extend**.

2. Select the 6 edges of **Tool_Surface** and enter ~~[0.6 in] (15 mm) for *Distance*~~ as shown in Figure 2–44.

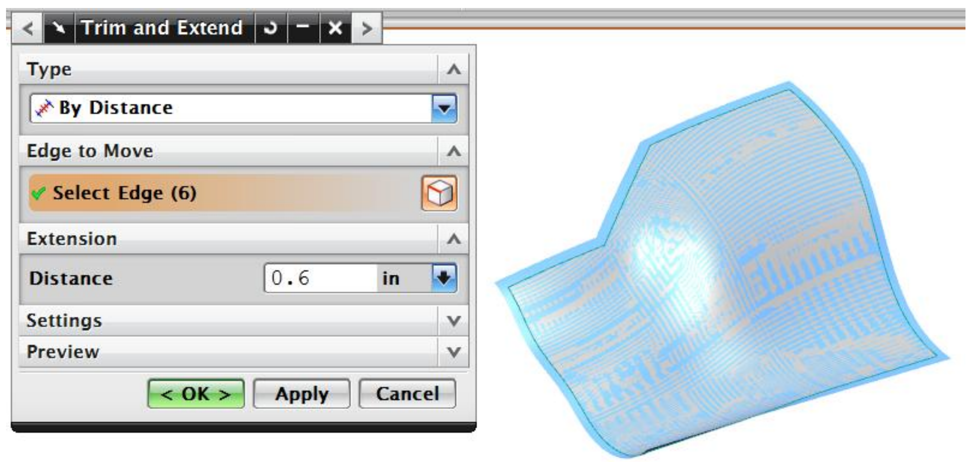


Figure 2–44

3. Click < OK >.

4. Select **Edit > Surface > Reverse Normal**.

Fibersim uses the tool surface normal to determine the direction of the layup (male or female tool). To ensure the correct normal direction on the tool surface, the recommended practice is to use the **Reverse Normal** command.

5. Select **Tool_Surface** and note the arrow in the middle of the surface indicating the normal direction of the surface as shown in Figure 2–45. (Press <W> to display WCS.)

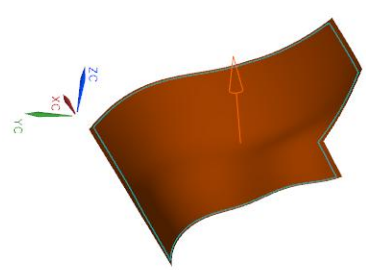


Figure 2–45

6. In this case the normal direction should point -Z direction. Click **Apply** to reverse the direction of normal.

7. Click **Cancel** to exit Reverse Normal command.

Task 5 - Create an Extended Tool Boundary.

1. Following the steps described in Task 3, extract the boundary edges of the **Tool_Surface**. Rename the new boundary as [Extended_Boundary], and change the color and line width to Magenta and Thin Width.

2. The model should display as shown in Figure 2–46.

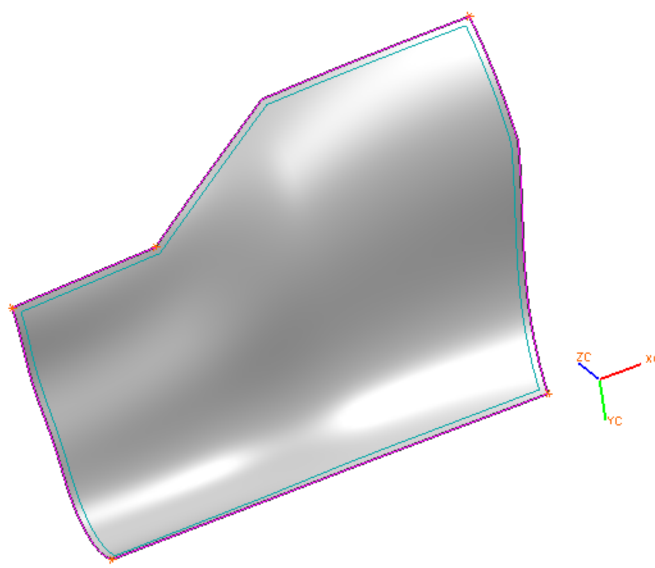


Figure 2–46

Task 6 - Extract a Ply Edge from a Solid.

The **Extended Boundary** curve created in Task 5 provides the dimensions for the full plies. In this task, you will create a curve that will provide dimensions for the shorter plies in the thicker area of the part.

1. Show **Body** (right-click in the Part Navigator and select **Show**).

2. Click  (Project Curve) and select the edges of the solid as shown in orange in Figure 2–47.

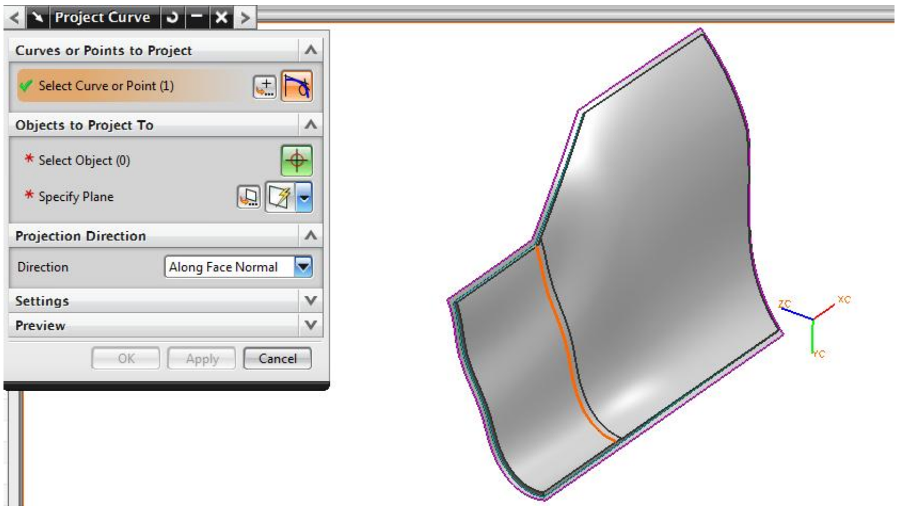


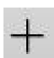
Figure 2–47

3. Select **Tool_Surface** for Objects to Project To.

4. Click **< OK >**.

5. Hide **Body** from the Part Navigator.

Task 7 - Create Rosette geometry.

1. Click  (Point).

2. In the Point type drop-down list, select **Point on Face**.

3. Indicate a placement point somewhere in the middle of the Tool Surface.

4. Click **< OK >**.

5. Select **Point** in the *Selection Filter* list.

6. Right-click and select **Properties**.

7. Enter [Rosette_Origin] for *Name* and *Feature Name*.

8. Click  (Line).

9. Select the following:

    - Start Option: **Point > select Rosette_Origin**
    - End Option: **Along XC**
    - Start Limit: **At Point**
    - Distance: **0 in (0 mm)**
    - End Limit: **Value**
    - Distance: **3 in (76.2 mm)**

The Line Definition dialog box updates as shown in Figure 2–48.

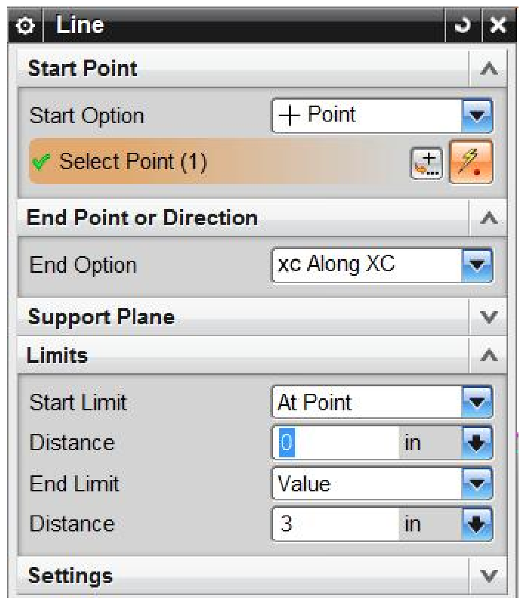


Figure 2–48

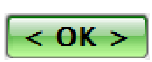
10. Click 

.

11. Select **Curve** in the *Selection Filter* list.

12. Right-click and select **Properties**.

13. Enter [Zero_Direction] for *Name* and *Feature Name*.

14. Click 

.

Exercise 2d [NX9/10] Create a Model from a Solid

In this exercise, you will prepare a solid NX model for composite design with Fibersim using NX9. The completed model displays as shown in Figure 2–49.

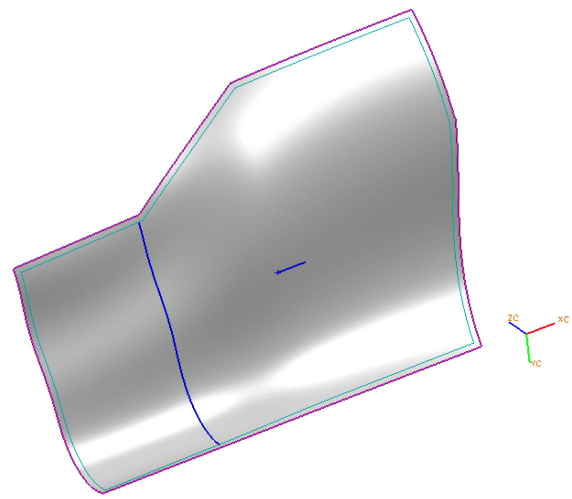


Figure 2–49

Goal

After you complete this exercise, you will be able to:

- ✓ **Extract a Tool Surface from a Solid**
- ✓ **Create Net Boundary geometry**
- ✓ **Extend a Tool Surface to account for manufacturing trim**
- ✓ **Create Extended Boundary geometry**
- ✓ **Extract a Ply Edge from a Solid**
- ✓ **Create Rosette geometry**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **MODEL_SETUP_B.prt**. The model displays as shown in Figure 2–50.

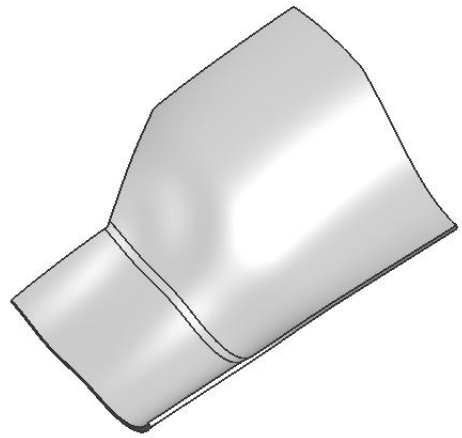


Figure 2–50

Task 2 - Extract Tool Surface from Solid.

1. Select **More** from **Feature > Extract Geometry** from the *Home* tab as shown in .

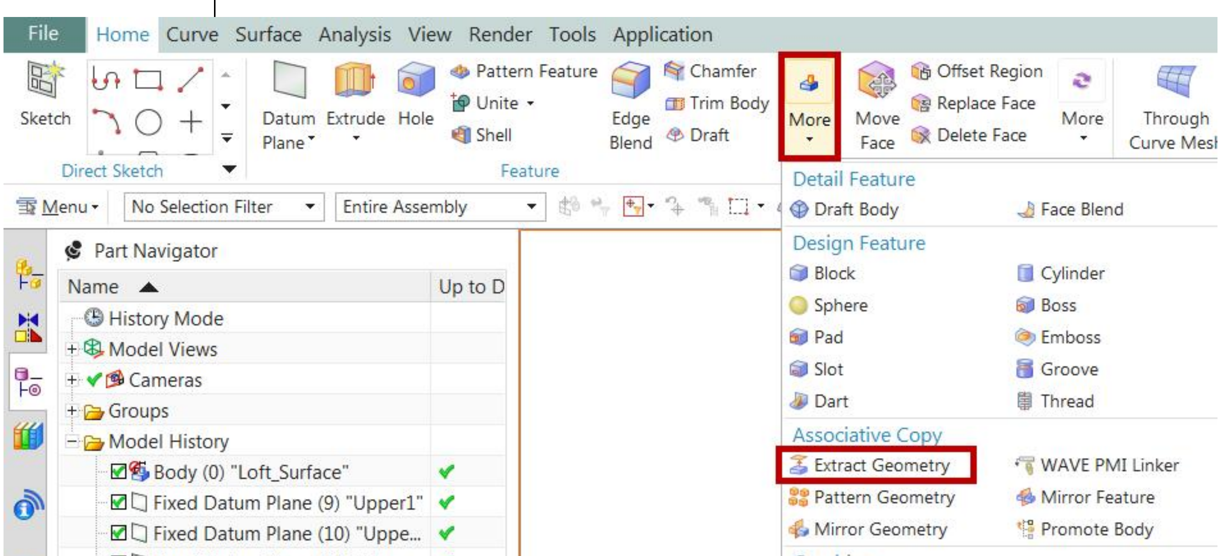


Figure 2–51

2. Select **Face** for *Type*.
3. Select the rear surface of the body as shown in Figure 2–52.

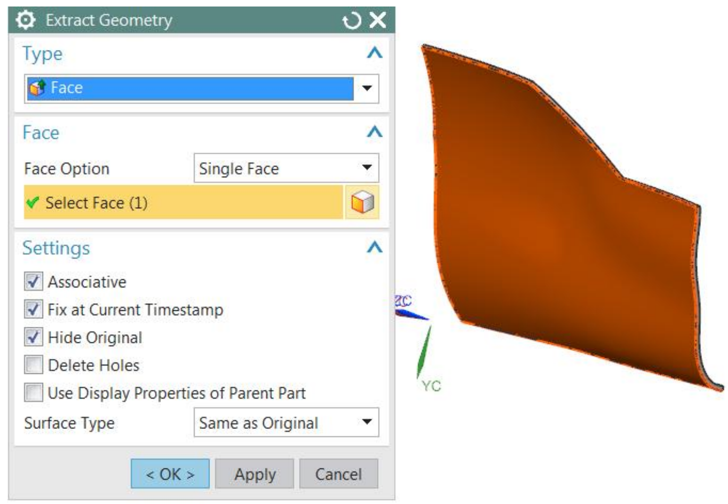


Figure 2–52

4. Click < OK >.
5. Select the extracted face.
6. Right-click and select **Properties**.
7. Select the *General* tab.
8. Enter [Tool_Surface] as shown in Figure 2–53.

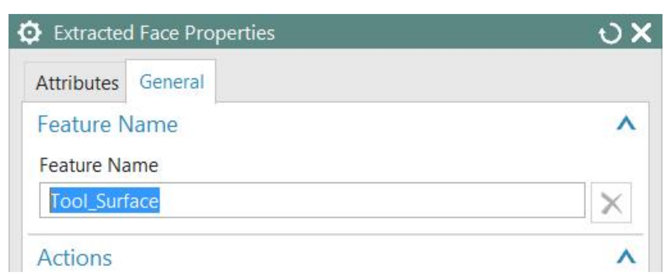


Figure 2–53

9. Click OK.

Task 3 - Create Net Boundary geometry.

1. Select **Extract Geometry** from the *Home* tab again and select **Composite Curve** for *Type*.
2. Select the 6 edges of the surface and set *Joint Curves* to **General** as shown in Figure 2–54.

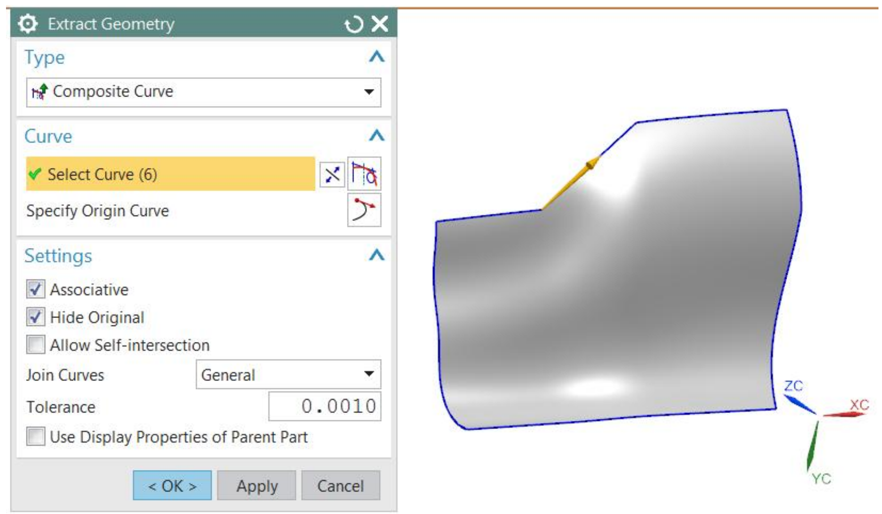


Figure 2–54

3. Click < OK >.
4. Select **Curve** from the *Selection Filter* list.
5. Select the composite curve.
6. Right-click and select **Properties**.
7. Enter [Net_Boundary] for *Name* and *Feature Name*.
8. Click OK.
9. Select Net_Boundary.
10. Right-click and select **Edit Display**.
11. Change the color to **Cyan** and Width to **0.13mm**.
12. Click OK.

Task 4 - Create an Extended Tool Surface.

In this task, you will extend the net tool surface to provide a 0.6 in (15mm) material excess for the manufacturing trim.

1. **[NX9]** Select **More** from **Feature > Trim and Extend** from the *Home* tab as shown in Figure 2–55.

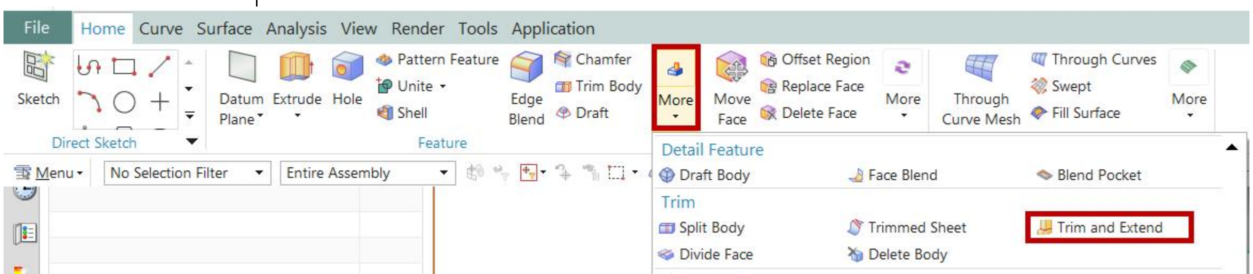


Figure 2–55

2. **[NX9]** Select the 6 edges of **Tool_Surface** and enter [0.6 in] (15 mm) for ~~Distance~~ as shown in Figure 2–56.

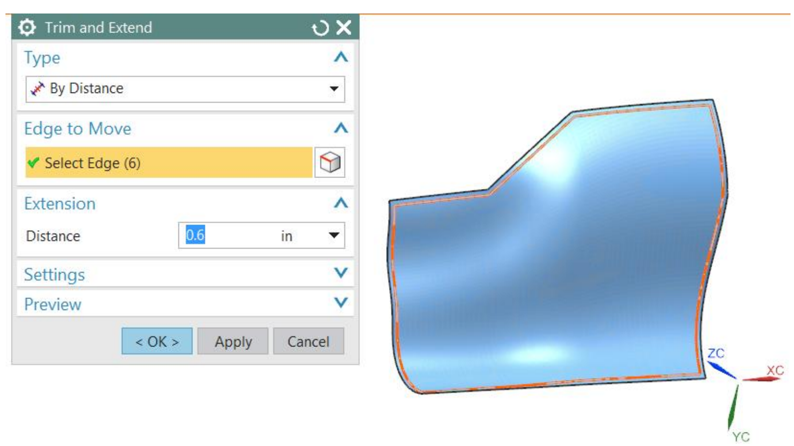


Figure 2–56

3. **[NX9]** Click < OK >.

1. **[NX10]** Select **Extend Sheet** from the *Surface* tab as shown in Figure 2–57.

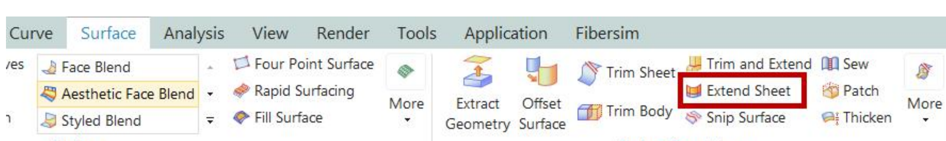


Figure 2–57

2. **[NX10]** Select the 6 edges of **Tool_Surface** and enter [0.6 in] (15 mm) for *Offset* as shown in Figure 2–56.

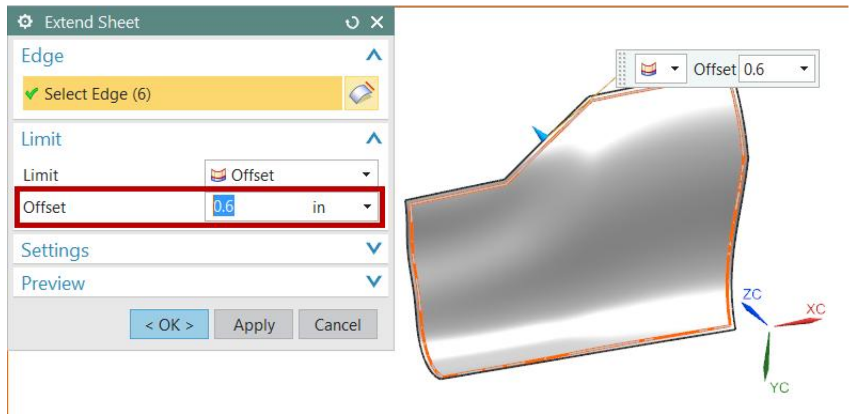


Figure 2–58

3. **[NX10]** Click < OK >.

4. Select **Edit Surface > More > Reverse Normal** from the *Surface* tab.

Fibersim uses the tool surface normal to determine the direction of the layup (male or female tool). To ensure the correct normal direction on the tool surface, the recommended practice is to use the **Reverse Normal** command.

5. Select **Tool_Surface** and note the arrow in the middle of the surface indicating the normal direction of the surface as shown in Figure 2–59. (Press <W> to display WCS.)

Figure 2–59

6. In this case the normal direction should point -Z direction. Click Apply to reverse the direction of normal.

7. Click Cancel to exit Reverse Normal command.

Task 5 - Create an Extended Tool Boundary.

1. Following the steps described in Task 3, extract the boundary edges of the **Tool_Surface**. Rename the new boundary as [Extended_Boundary], and change the color and line width to **Magenta** (Color ID: 181) and **0.13 mm**.

2. The model should display as shown in Figure 2–60.

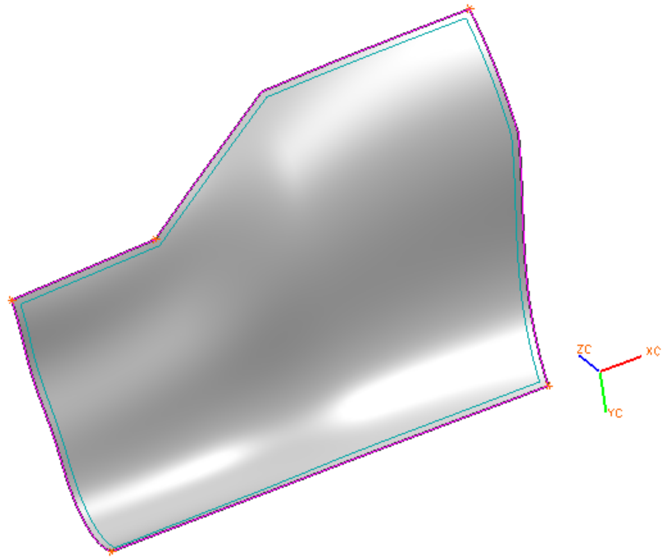


Figure 2–60

Task 6 - Extract a Ply Edge from a Solid.

The **Extended Boundary** curve created in Task 5 provides the dimensions for the full plies. In this task, you will create a curve that will provide dimensions for the shorter plies in the thicker area of the part.

1. Right-click on **Body** from the Part Navigator and select **Show**.

2. Click Project Curve (Project Curve) from the *Curve* tab and select the edges of the solid as shown in orange in Figure 2–61.

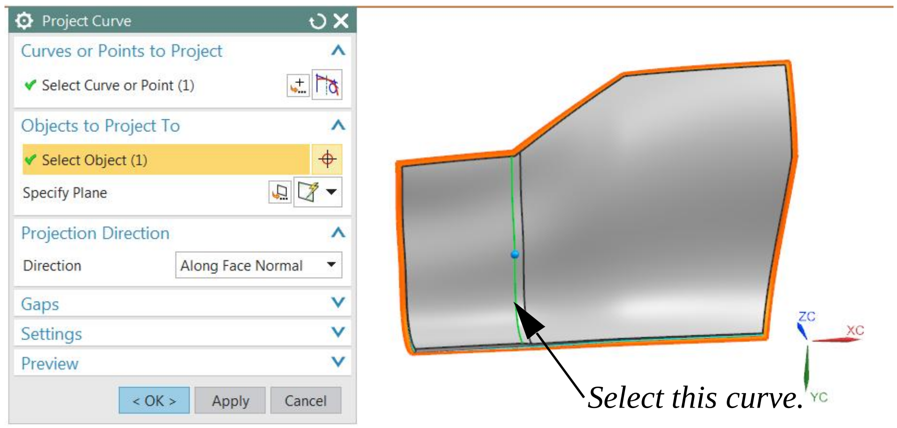


Figure 2–61

3. Select **Tool_Surface** for *Objects to Project To*.

4. Click < OK >.

5. Hide **Body** from the Part Navigator.

Task 7 - Create Rosette geometry.

1. Click Point (Point) from the *Curve* tab.

2. In the Point type drop-down list, select **Point on Face**.

3. Indicate a placement point somewhere in the middle of the Tool Surface.

4. Click < OK >.

5. Select **Point** from the *Selection Filter* list.

6. Select the created point.

7. Right-click and select **Properties**.

8. Enter [Rosette_Origin] for *Name* and *Feature Name*.

9. Click Line (Line) from the *Curve* tab.

10. Select the following:

- Start Option: **Point > select Rosette_Origin**
- End Option: **Along XC**
- Start Limit: **At Point**
- Distance: **0 in (0 mm)**
- End Limit: **Value**
- Distance: **3 in (76.2 mm)**

The Line Definition dialog box updates as shown in Figure 2–62.

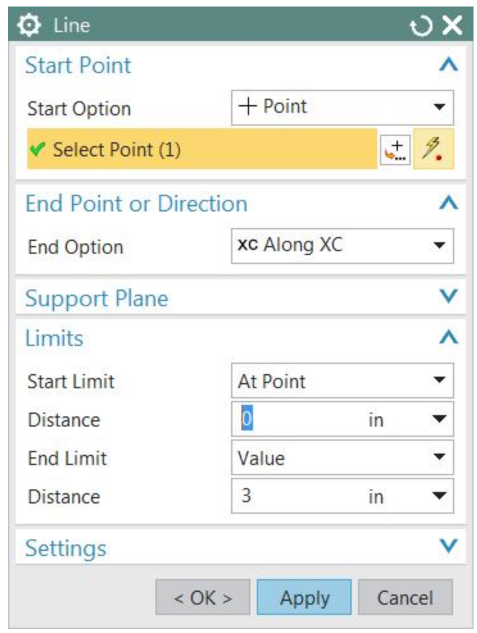


Figure 2–62

11. Click < OK >.

12. Select **Curve** from the *Selection Filter* list.

13. Select the created curve.

14. Right-click and select **Properties**.

15. Enter [Zero_Direction] for *Name* and *Feature Name*.

16. Click < OK >.

Exercise 2e NX Model Setup from Planes and Surface (Optional)

In this exercise, with minimal instruction, you will prepare a NX model for composite design with Fibersim. The completed model should display as shown in Figure 2–63.

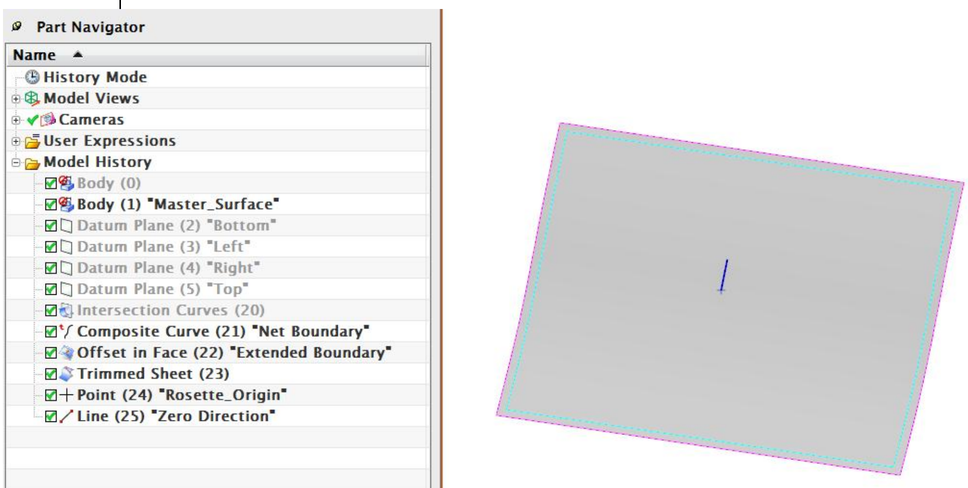


Figure 2–63

Goal

After you complete this exercise, you will be able to:

- ✓ **Create Net Boundary geometry**
- ✓ **Create Extended Boundary geometry**
- ✓ **Create a Tool Surface**
- ✓ **Create Rosette geometry**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **MODEL_SETUP_C.prt**. The model displays as shown in Figure 2–64.

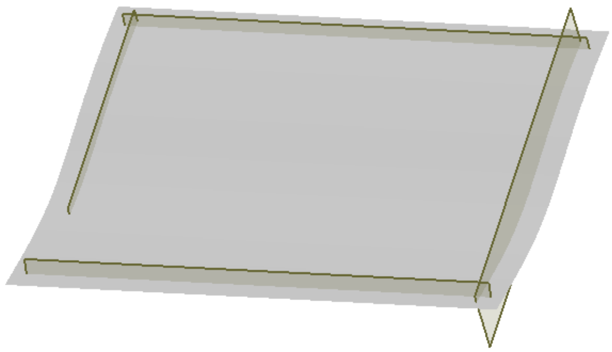


Figure 2–64

Task 2 - Create a Net Boundary.

1. Create a Net Boundary by intersecting the **Master_Surface** with the planes **LEFT**, **RIGHT**, **TOP**, and **BOTTOM** and then trimming the resulting curves to create a closed loop.

2. Modify the graphic properties of the Net Boundary. The resulting model should display as shown in Figure 2–65.

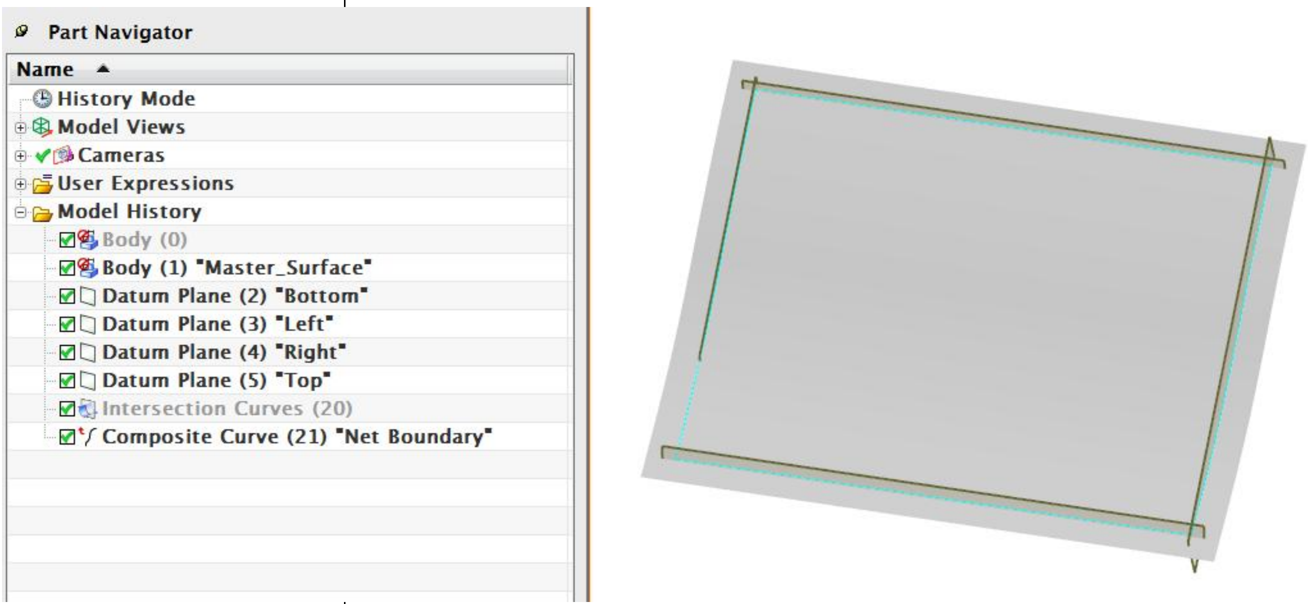


Figure 2–65

Task 3 - Create an Extended Boundary.

1. Create an Extended Boundary by offsetting the Net Boundary by 0.788 in (20 mm) on the Master Surface.

2. Modify the graphic properties of the Extended Boundary so that the resulting model displays as shown in Figure 2–66.

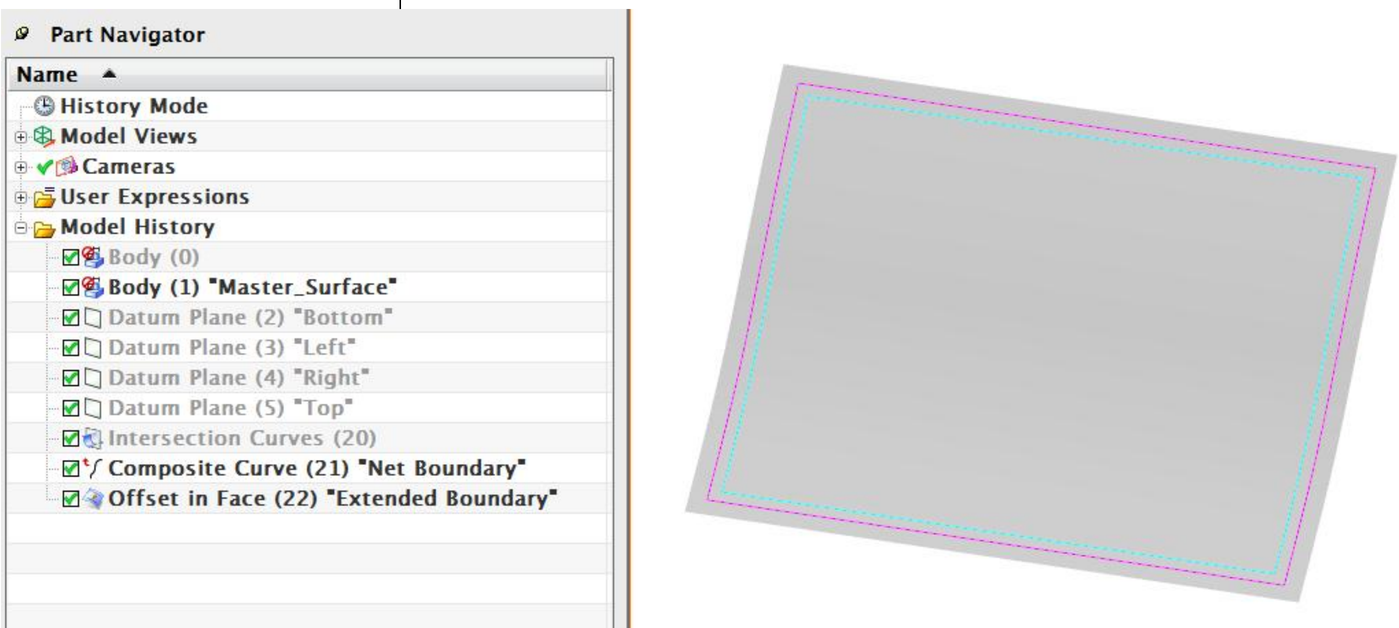


Figure 2–66

Task 4 - Create a Tool Surface.

1. Create a Tool Surface by trimming the Master Surface with the Extended Boundary.

Task 5 - Create the Rosette geometry.

2. Create a point in the middle of the **Master_Surface** and rename it [Rosette_Origin].

3. Create a 3.937in (100mm) line that starts at the Rosette Origin and is aligned with the part's Y-direction. Rename the line as [Zero Direction].

4. The model is now ready for composite design with Fibersim and should display as shown in Figure 2–67.

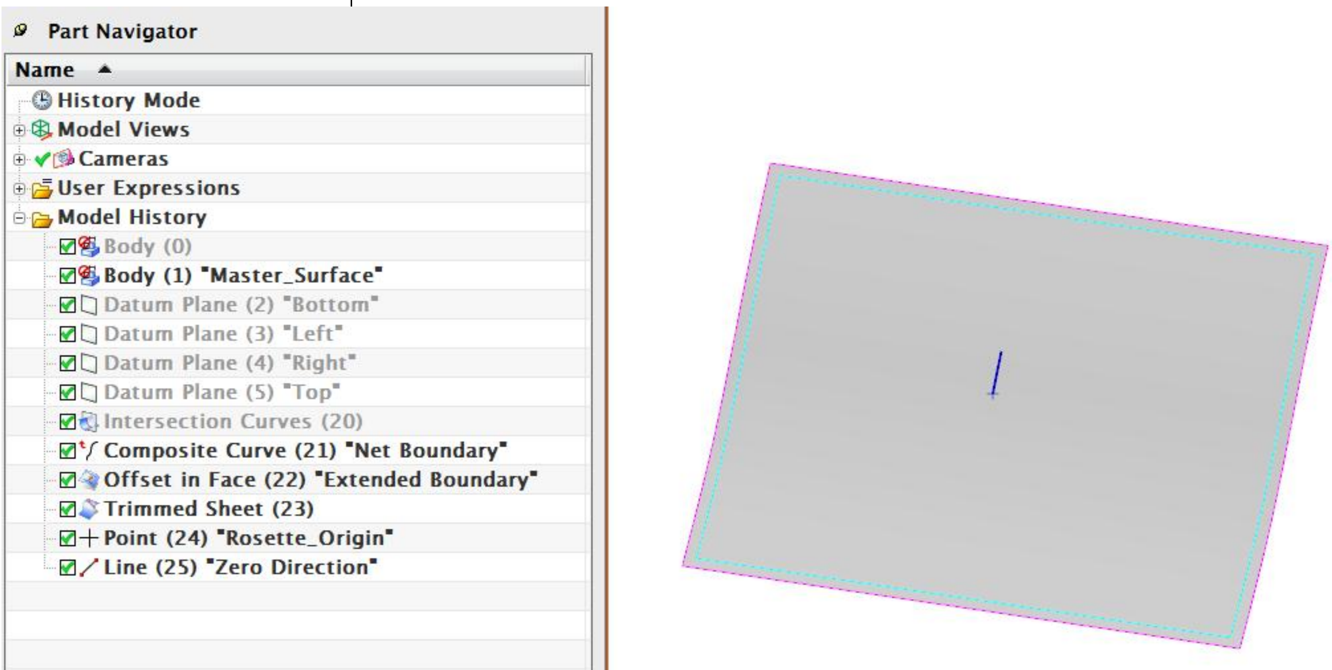


Figure 2–67

Chapter 3

Introduction to Ply-Based Design

This chapter includes:

- ✓**Exercise 3a: Basic Part with 8 Full-body Plies**
- ✓**Exercise 3b: Reinforcement Plies, Sequencing, and Cutouts**
- ✓**Exercise 3c: Flat Pattern**
- ✓**Exercise 3d: Design to Extended**
- ✓**Exercise 3e: Wrapped Ply**
- ✓**Exercise 3f: Create Basic Part**

Exercise 3a Basic Part with 8 Full-body Plies

User Guide Reference:
2.6 Laminates
2.7 Rosettes
2.8 Plies
B.3.2 Mirror Laminate

In this exercise, you will cover a part with eight full-body plies. The completed model displays as shown in Figure 3–1.

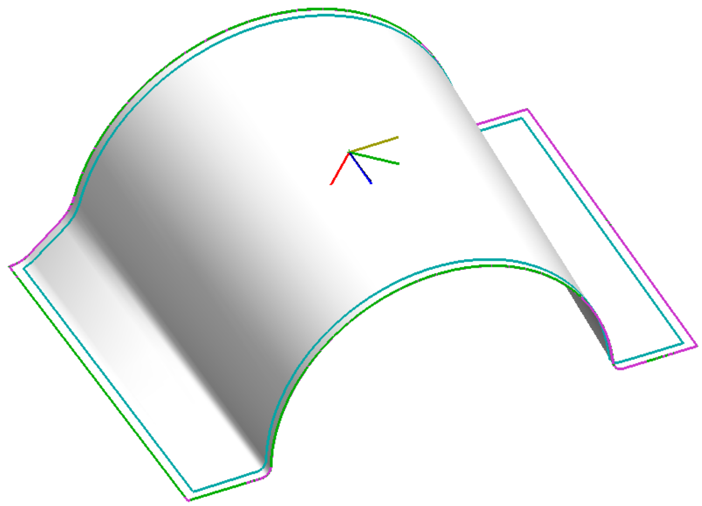


Figure 3–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a Laminate**
- ✓ **Create a Rosette**
- ✓ **Create a single full-body ply**
- ✓ **Create three full-body plies simultaneously**
- ✓ **Mirror four plies about the laminate neutral axis**

Estimated Time

10 min

Task 1 - Create a Laminate.

1. Open **INTRO_PLY_A.prt**. The model displays as shown in Figure 3–2.

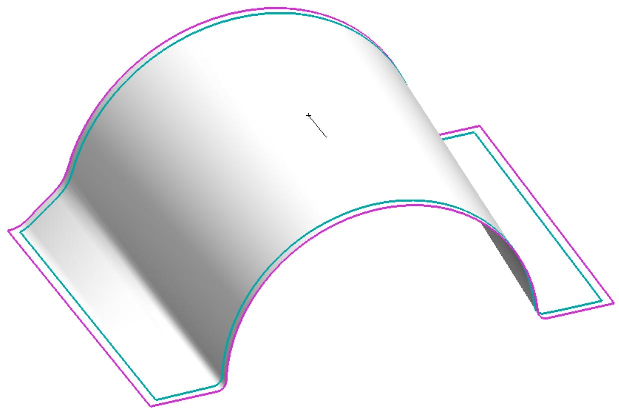


Figure 3–2

*Alternatively, you can go to **Start (or File) > All Applications > Fibersim** or click on the Fibersim icon to start Fibersim.*

2. Press <F9> to start Fibersim.

3. In the *Application Tree*, select **Laminate** under **Basics**. as shown in Figure 3–3.

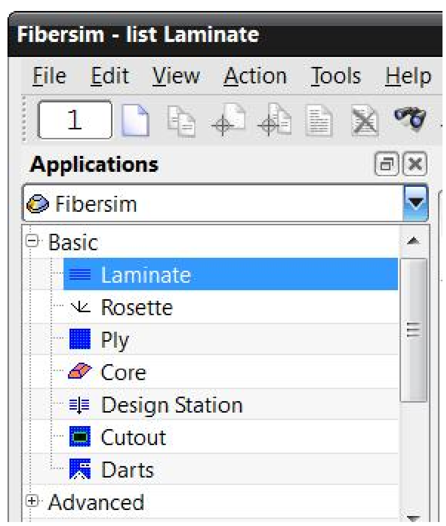


Figure 3–3

*Alternatively, you can highlight Laminate, right-click in the main Fibersim window, and select **Create New**.*

4. Click  (Create New).

5. In the Laminate's Standard form, enter the following parameters:
   - Name: **Cowl**
   - Step: **1**
6. Next to Default Material, click (Link with Database Link Dialog) to open the *link Material via Default Material* dialog box.
7. Click checkbox for **PPG-PL-3K** and click OK to link the material to the laminate.

Task 2 - Link Geometry.

1. Next to the *Layup Surface* field, click (Link Geometry) as shown in Figure 3–4.

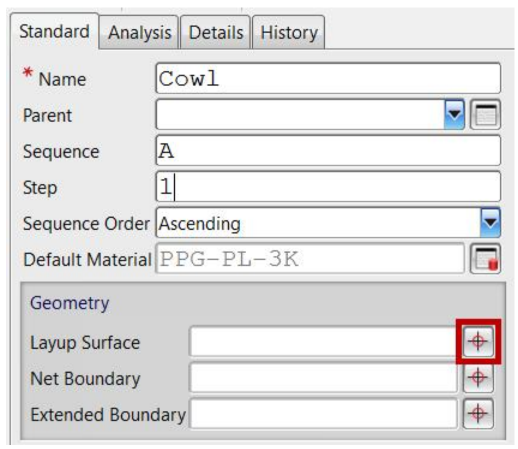


Figure 3–4

2. In NX, select the surface as shown in Figure 3–5.

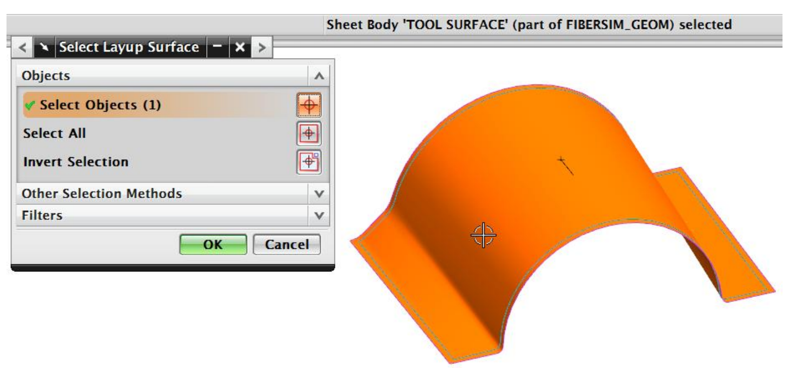


Figure 3–5

3. Click OK.

4. Use the same procedure to link the following geometry:
   - Net Boundary: **Net Boundary** (Cyan Boundary)
   - Extended Boundary: **Extended Boundary** (Magenta Boundary)
5. The Laminate Standard form updates as shown in Figure 3–6.

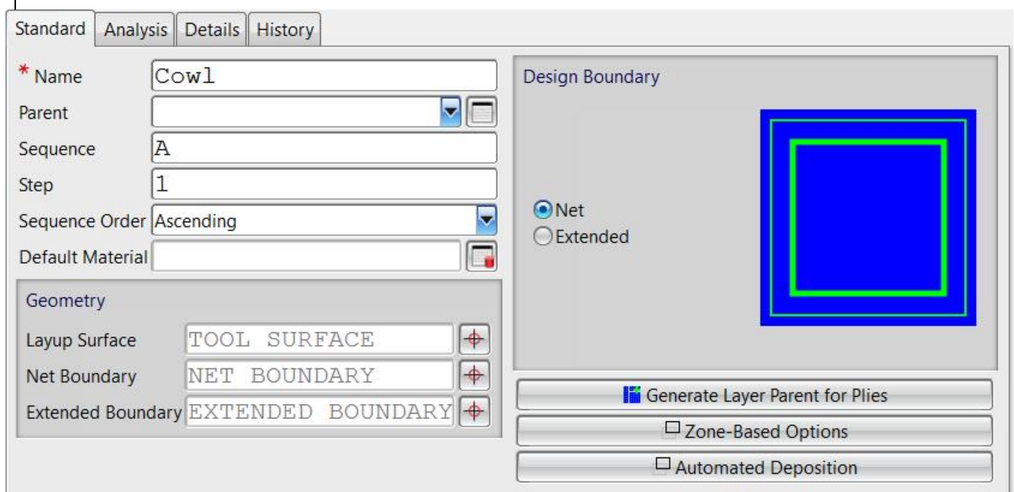


Figure 3–6

*If you need to delete a Laminate, select its name in the list, right-click, and select **Delete**.*

6. Click **OK** to complete the laminate creation and return to the list view. Select the **Cowl** laminate from the list view to display the laminate on the NX model as shown in Figure 3–7. The arrow represents the layup direction, which is inherited from the laminate surface normal.

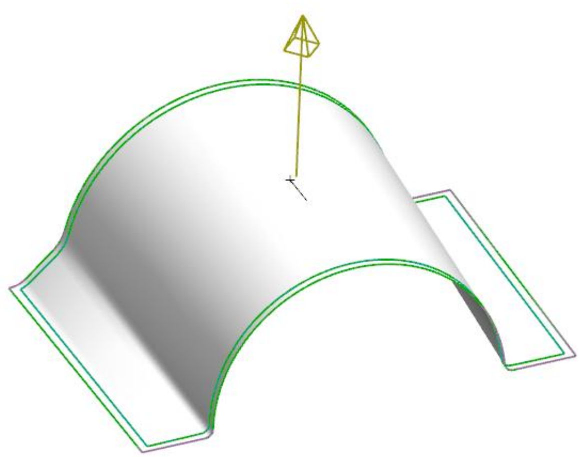


Figure 3–7

Task 3 - Create a Rosette.

1. In the *Application Tree*, select **Rosette** under **Basics** as shown in Figure 3–8.

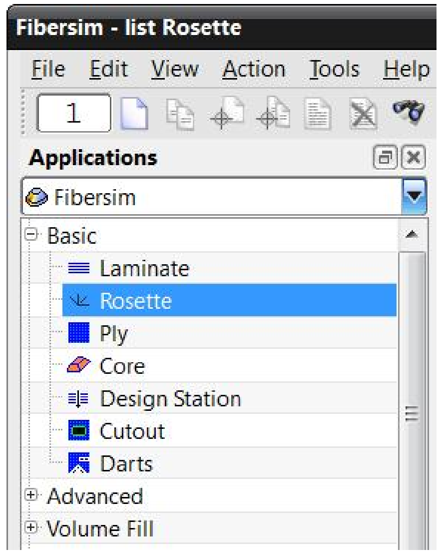


Figure 3–8

2. Click  (Create New).

3. Click  (Link Geometry) next to *Origin*.

4. Select only the **Existing Point** filter as shown in Figure 3–9.

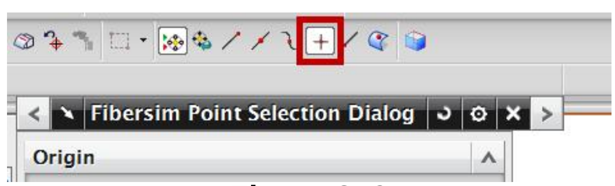
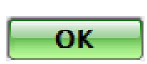


Figure 3–9

5. Select **Rosette Origin**.

6. Click .

7. Select **Zero Direction** for *Direction*.

8. The Rosette form updates as shown in Figure 3–10.

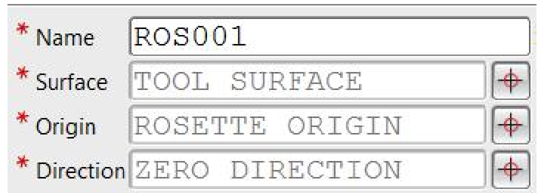


Figure 3–10

9. Click  to complete the rosette creation and return to the list view.

Task 4 - Create a single full-body ply.

1. In the *Application Tree*, select **Ply** under **Basics** as shown in Figure 3–11.

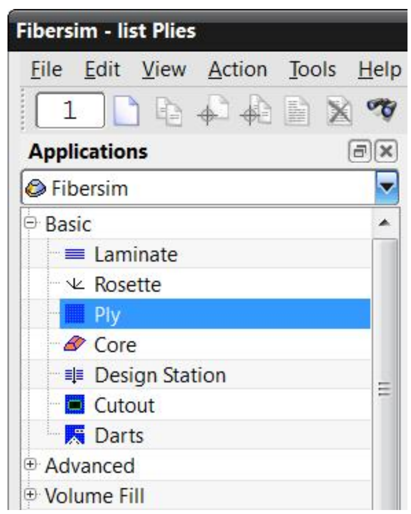


Figure 3–11

2. Click  (Create New).
3. For the P001 ply, enter the following parameter:
   - Step: **10**
4. Click  to complete the ply creation and return to the list view.

For [20,10], the 20 means that the first newly created ply has the number 20 and the other ply numbers are generated in increments of 10.

Task 5 - Create 3 full-body plies simultaneously.

1. In *Object Count*, enter [3] as shown in Figure 3–12.

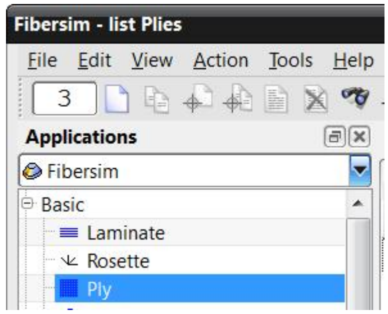


Figure 3–12

2. Click  (Create New).

3. In the *Step* field, enter [20,10] as shown in Figure 3–13.

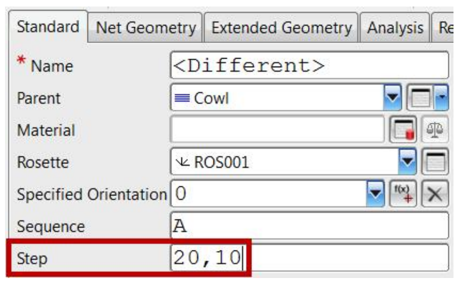


Figure 3–13

4. Press <Ctrl> + <T> to toggle to Table mode. The new plies, P002, P003, and P004, display in the table view as shown in Figure 3–14.

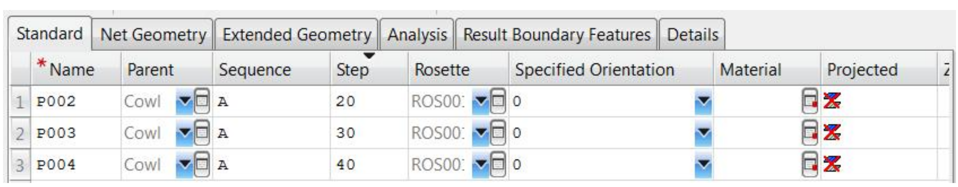


Figure 3–14

5. In the *Specified Orientation* field, enter [45], [90], and [-45] as shown in Figure 3–15.

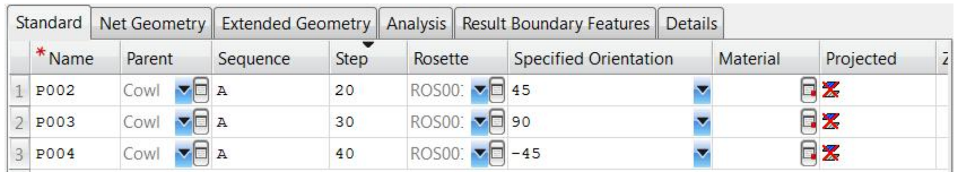
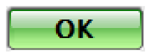


Figure 3–15

6. Press <Ctrl> + <T> to return to Form mode.

7. Click  to complete the creation of the three new plies and return to the list view. The four plies P001, P002, P003, and P004, are listed as shown in Figure 3–16.

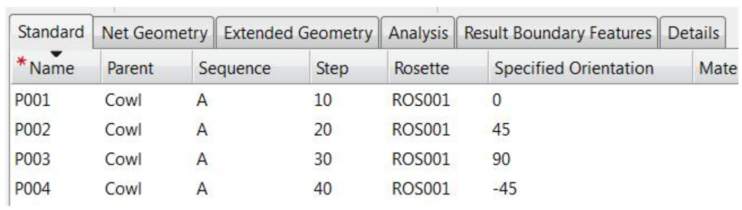


Figure 3–16

Task 6 - Mirror four plies about the laminate neutral axis.

1. Select **Tools > Operations > Mirror Laminate** as shown in Figure 3–17.

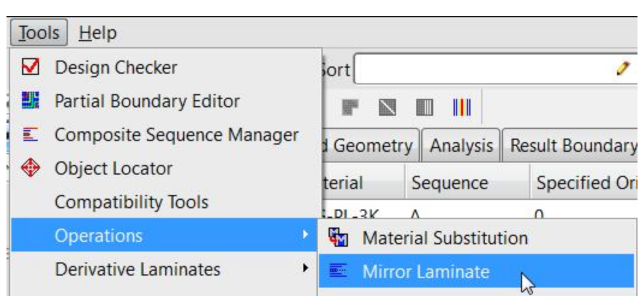


Figure 3–17

2. The Mirror Laminate dialog box opens as shown in Figure 3–18.

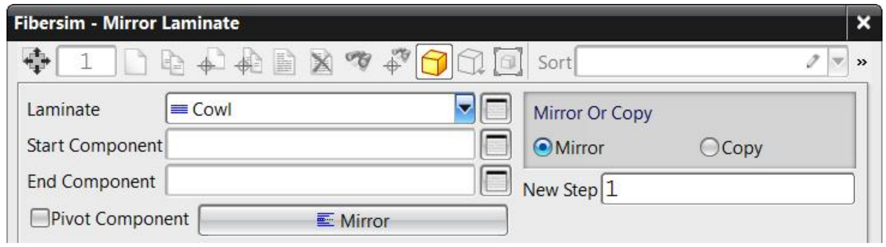


Figure 3–18

3. For the *Start Component,* click  (Link with Link Dialog) as shown in Figure 3–19.

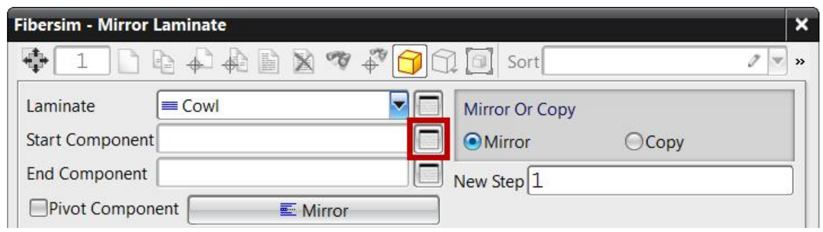


Figure 3–19

4. Click checkbox for **P001** as shown in Figure 3–20.

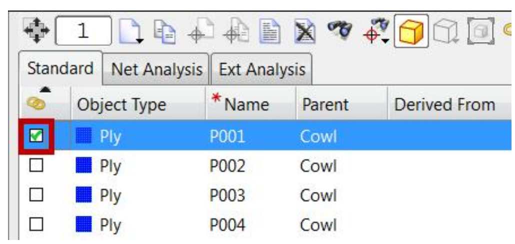
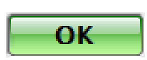


Figure 3–20

5. Click .

6. Repeat Steps 3 through 5 to link **P004** to *End Component*. The Mirror Laminate form updates as shown in Figure 3–21.

Figure 3–21

7. In the *New Step* field, enter [50] as shown in Figure 3–22.

Figure 3–22

8. Click Mirror.

9. The Fibersim message window opens prompting you that four components were generated successfully as shown in Figure 3–23.

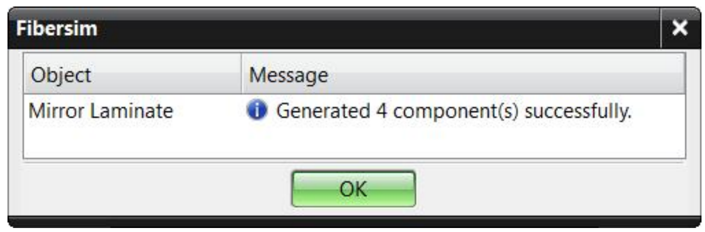
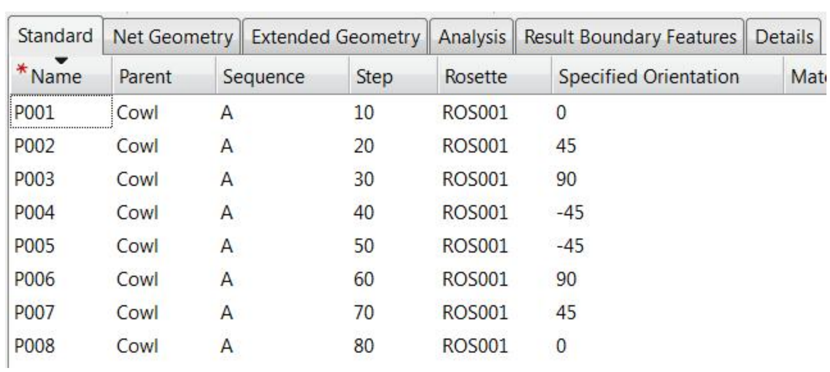


Figure 3–23

10. Click **OK**.

11. In the Mirror Laminate dialog box, click **OK** to complete the mirror operation and return to the list view. The updated list of plies P001 to P008 displays as shown in Figure 3–24.



Standard | Net Geometry | Extended Geometry | Analysis | Result Boundary Features | Details

*Name	Parent	Sequence	Step	Rosette	Specified Orientation	Mat
P001	Cowl	A	10	ROS001	0	
P002	Cowl	A	20	ROS001	45	
P003	Cowl	A	30	ROS001	90	
P004	Cowl	A	40	ROS001	-45	
P005	Cowl	A	50	ROS001	-45	
P006	Cowl	A	60	ROS001	90	
P007	Cowl	A	70	ROS001	45	
P008	Cowl	A	80	ROS001	0	

Figure 3–24

12. Close Fibersim.

13. Save the model.

Exercise 3b Reinforcement Plies, Sequencing, and Cutouts

User Guide Reference:
2.8 Plies
B.2.3 Composite Sequence Manager
2.10 Design Stations
2.11 Cutouts

In this exercise, you will create reinforcement plies and interleave them using the Composite Sequence Manager. You will also create a cutout and run an analysis comparing the results. The re-sequenced list of plies is shown in Figure 3–25.

Object Type	Object Name	Parent	Sequence	Step	Specified Orientation
Laminate	Cowl		A	1	
Ply	P001	Cowl	A	1	0
Ply	P002	Cowl	A	2	45
Ply	P003	Cowl	A	2	45
Ply	P004	Cowl	A	3	45
Ply	P005	Cowl	A	4	0
Ply	P006	Cowl	A	4	0
Ply	P007	Cowl	A	5	90
Ply	P008	Cowl	A	6	45
Ply	P009	Cowl	A	6	45
Ply	P010	Cowl	A	7	-45
Ply	P011	Cowl	A	8	-45
Ply	P012	Cowl	A	9	45
Ply	P013	Cowl	A	9	45
Ply	P014	Cowl	A	10	90
Ply	P015	Cowl	A	11	45
Ply	P016	Cowl	A	12	0
Ply	P017	Cowl	A	12	0
Ply	P018	Cowl	A	13	45
Ply	P019	Cowl	A	13	45
Ply	P020	Cowl	A	14	0

Figure 3–25

Goal

After you complete this exercise, you will be able to:

- ✓ **Create Reinforcement Plies**
- ✓ **Use the Composite Sequence Manager to interleave reinforcement plies**
- ✓ **Rename plies to match the lay-up order**
- ✓ **Create Design Stations and run a core sample analysis**
- ✓ **Create a Cutout**
- ✓ **Run an analysis and verify the results**

Estimated Time

15 min

Task 1 - Open a part.

1. Select **File > Open** and select **INTRO_PLY_B.prt**.

   If you completed Exercise 3a, you can continue working with **INTRO_PLY_A.prt** instead. The model displays as shown in Figure 3–26.

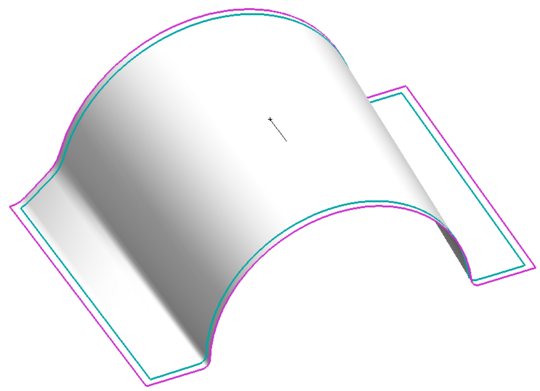


Figure 3–26

Task 2 - Create Reinforcement Plies.

1. Press <Ctrl> + <L> to open the Layer Settings window in NX.

*Enable **Category Display** option to see the category names in the **Name** column.*

2. In the Layers list, show **Reinforcement Plies** layer category as shown in Figure 3–27. The model displays the boundaries for reinforcement plies as shown in Figure 3–28.

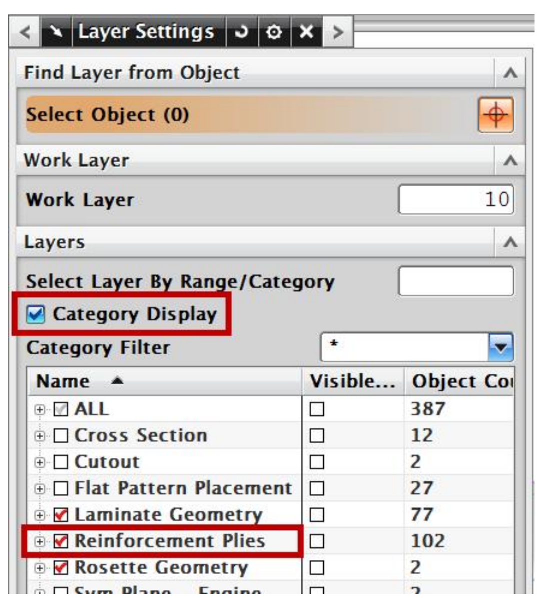


Figure 3–27

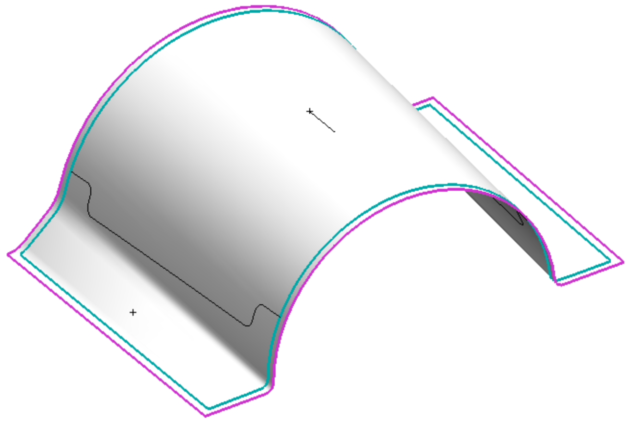
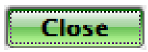


Figure 3–28

3. Click .
4. Press <F9> to start Fibersim.
5. In the *Application Tree*, select **Ply** under **Basics**.
6. For *Object Count*, enter [6] as shown in Figure 3–29.

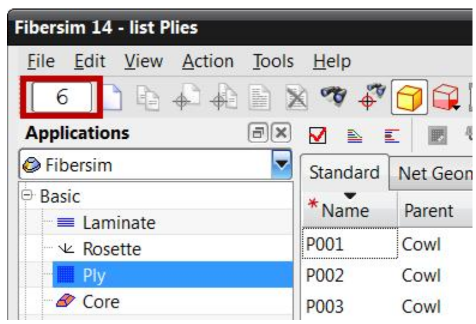


Figure 3–29

7. Click  (Create New).
8. In the *Geometry* area, click  (Link Geometry) for *Net* > *Origin*.
9. Make sure that the **Existing Point** filter is selected as shown in Figure 3–30.

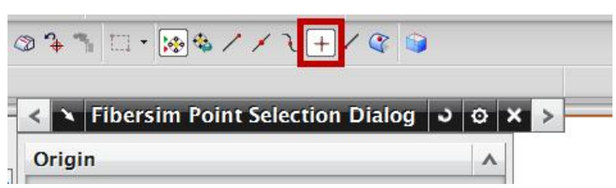


Figure 3–30

10. Select **LH Reinforcement Origin**.

11. Click OK.

12. For *Boundary*, click (Link Geometry).

13. Select **LH Reinforcement**.

14. Click OK.

15. Press <Ctrl>+ <T> to switch to Table mode. The new plies display as shown Figure 3–31.

Standard | Net Geometry | Extended Geometry | Analysis | Result Boundary Features | Details

	*Name	Parent	Sequence	Step	Rosette	Specified Orientation	Mate
1	P009	Cowl	A	90	ROS001	0	<Cov
2	P010	Cowl	A	100	ROS001	0	<Cov
3	P011	Cowl	A	110	ROS001	0	<Cov
4	P012	Cowl	A	120	ROS001	0	<Cov
5	P013	Cowl	A	130	ROS001	0	<Cov
6	P014	Cowl	A	140	ROS001	0	<Cov

Figure 3–31

16. In the *Specified Orientation* field, enter [45], [0], [45], [45], [0], and [45] as shown in Figure 3–32.

Standard | Net Geometry | Extended Geometry | Analysis | Result Boundary Features | Details

	*Name	Parent	Sequence	Step	Rosette	Specified Orientation	Mate
1	P009	Cowl	A	90	ROS001	45	<Cov
2	P010	Cowl	A	100	ROS001	0	<Cov
3	P011	Cowl	A	110	ROS001	45	<Cov
4	P012	Cowl	A	120	ROS001	45	<Cov
5	P013	Cowl	A	130	ROS001	0	<Cov
6	P014	Cowl	A	140	ROS001	45	<Cov

Figure 3–32

17. Click **OK** to display the new plies as shown in Figure 3–33.

Standard | Net Geometry | Extended Geometry | Analysis | Result Boundary Features | Details

*Name	Parent	Sequence	Step	Rosette	Specified Orientation	Material	Projected
P001	Cowl	A	10	ROS001	0	PPG-PL-3K	
P002	Cowl	A	20	ROS001	45	PPG-PL-3K	
P003	Cowl	A	30	ROS001	90	PPG-PL-3K	
P004	Cowl	A	40	ROS001	-45	PPG-PL-3K	
P005	Cowl	A	50	ROS001	-45	PPG-PL-3K	
P006	Cowl	A	60	ROS001	90	PPG-PL-3K	
P007	Cowl	A	70	ROS001	45	PPG-PL-3K	
P008	Cowl	A	80	ROS001	0	PPG-PL-3K	
P009	Cowl	A	90	ROS001	45	<Cowl : PP...	
P010	Cowl	A	100	ROS001	0	<Cowl : PP...	
P011	Cowl	A	110	ROS001	45	<Cowl : PP...	
P012	Cowl	A	120	ROS001	45	<Cowl : PP...	
P013	Cowl	A	130	ROS001	0	<Cowl : PP...	
P014	Cowl	A	140	ROS001	45	<Cowl : PP...	

Figure 3–33

Task 3 - Use the Composite Sequence Manager to interleave the reinforcement plies.

1. In the ply toolbar, click (Composite Sequence Manager).
2. Drag and drop the plies to rearrange them in the following order: 1, 9, 2, 10, 3, 11, 4, 5, 12, 6, 13, 7, 14, and 8. The updated list is shown in Figure 3–34.

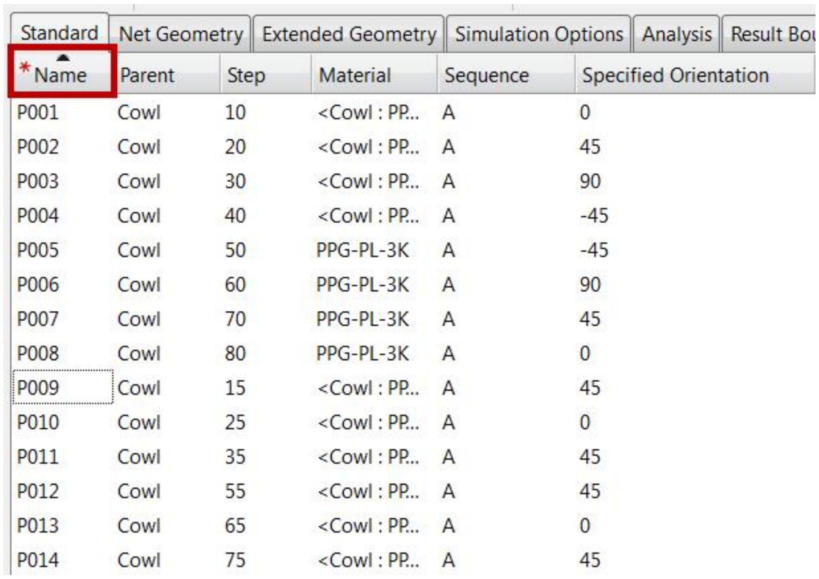
Object Type	Object Name	Parent	Sequence	Step	Specified Orientation	Material	Drc
Laminate	Cowl		A	1		PPG-PL-3K	
Ply	P001	Cowl	A	10	0	PPG-PL-3K	
Ply	P009	Cowl	A	15	45	<Cowl : PP...	
Ply	P002	Cowl	A	20	45	PPG-PL-3K	
Ply	P010	Cowl	A	25	0	<Cowl : PP...	
Ply	P003	Cowl	A	30	90	PPG-PL-3K	
Ply	P011	Cowl	A	35	45	<Cowl : PP...	
Ply	P004	Cowl	A	40	-45	PPG-PL-3K	
Ply	P005	Cowl	A	50	-45	PPG-PL-3K	
Ply	P012	Cowl	A	55	45	<Cowl : PP...	
Ply	P006	Cowl	A	60	90	PPG-PL-3K	
Ply	P013	Cowl	A	65	0	<Cowl : PP...	
Ply	P007	Cowl	A	70	45	PPG-PL-3K	
Ply	P014	Cowl	A	75	45	<Cowl : PP...	
Ply	P008	Cowl	A	80	0	PPG-PL-3K	

Figure 3–34

3. Click **OK**.

Task 4 - Create RH plies using Create Based On.

1. Select the *Name* column to sort the plies by name and ensure that P001 is at the top as shown in Figure 3–35.



Name	Parent	Step	Material	Sequence	Specified Orientation
P001	Cowl	10	<Cowl : PP...	A	0
P002	Cowl	20	<Cowl : PP...	A	45
P003	Cowl	30	<Cowl : PP...	A	90
P004	Cowl	40	<Cowl : PP...	A	-45
P005	Cowl	50	PPG-PL-3K	A	-45
P006	Cowl	60	PPG-PL-3K	A	90
P007	Cowl	70	PPG-PL-3K	A	45
P008	Cowl	80	PPG-PL-3K	A	0
P009	Cowl	15	<Cowl : PP...	A	45
P010	Cowl	25	<Cowl : PP...	A	0
P011	Cowl	35	<Cowl : PP...	A	45
P012	Cowl	55	<Cowl : PP...	A	45
P013	Cowl	65	<Cowl : PP...	A	0
P014	Cowl	75	<Cowl : PP...	A	45

Figure 3–35

Alternatively, you can select P009, press and hold <Shift> and then select P014 to highlight the range of plies.

2. Click **P009** in the list and drag down to **P014** to highlight the plies from **P009 to P014**.

3. With the six plies highlighted, right-click and select **Create Based On** as shown in Figure 3–36.

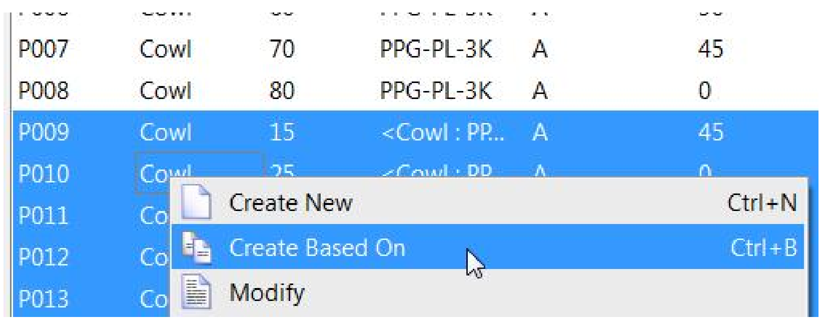


Figure 3–36

4. New plies P015 to P020 are created and displayed as shown in Figure 3–37.

   - Make sure the new plies are shown in Form mode as in Figure 3–37. You can press <Ctrl> + <T> to switch from Table mode to Form mode.

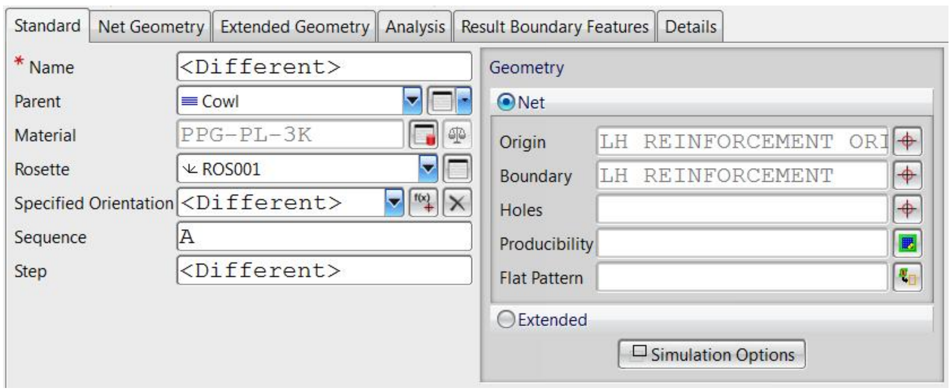


Figure 3–37

5. On the *Standard* tab, in the *Geometry* area, click (Link Geometry) for *Net* > *Origin*.

6. Turn on the Point on Surface option as shown in Figure 3–38.

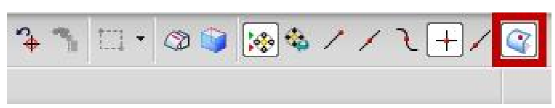


Figure 3–38

7. Rotate the model and indicate a location on the surface in approximately the location shown in Figure 3–39.

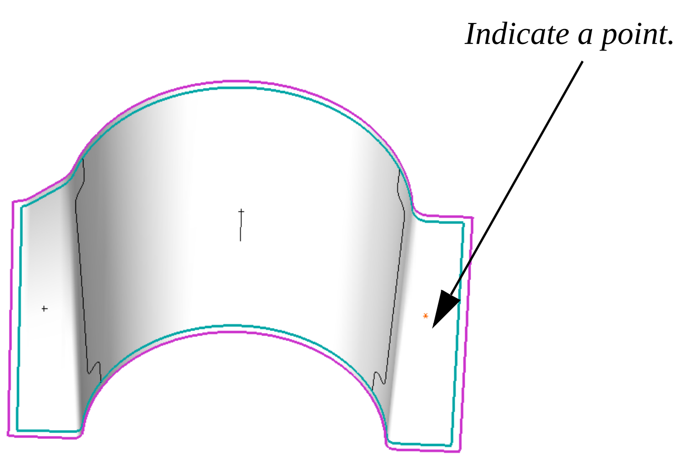


Figure 3–39

8. Click OK.

9. Click (Link Geometry) for *Boundary*.

10. Deselect **LH Reinforcement** by <Shift> + selecting.

11. Select **RH Reinforcement**.

12. Click OK.

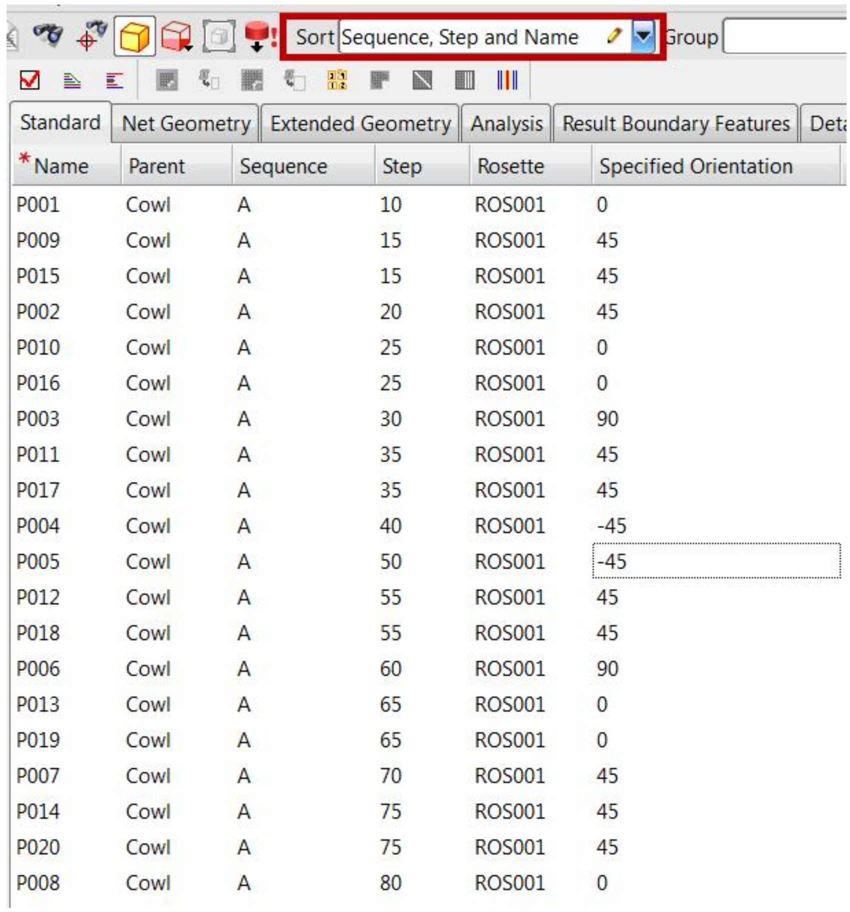
13. Click OK to display the modified plies in the list view as shown in Figure 3–40.

Standard | Net Geometry | Extended Geometry | Simulation Options | Analysis | Result

*Name	Parent	Step	Material	Sequence	Specified Orientation
P001	Cowl	10	<Cowl : PP...	A	0
P002	Cowl	20	<Cowl : PP...	A	45
P003	Cowl	30	<Cowl : PP...	A	90
P004	Cowl	40	<Cowl : PP...	A	-45
P005	Cowl	50	PPG-PL-3K	A	-45
P006	Cowl	60	PPG-PL-3K	A	90
P007	Cowl	70	PPG-PL-3K	A	45
P008	Cowl	80	PPG-PL-3K	A	0
P009	Cowl	15	<Cowl : PP...	A	45
P010	Cowl	25	<Cowl : PP...	A	0
P011	Cowl	35	<Cowl : PP...	A	45
P012	Cowl	55	<Cowl : PP...	A	45
P013	Cowl	65	<Cowl : PP...	A	0
P014	Cowl	75	<Cowl : PP...	A	45
P015	Cowl	15	PPG-PL-3K	A	45
P016	Cowl	25	PPG-PL-3K	A	0
P017	Cowl	35	PPG-PL-3K	A	45
P018	Cowl	55	PPG-PL-3K	A	45
P019	Cowl	65	PPG-PL-3K	A	0
P020	Cowl	75	PPG-PL-3K	A	45

Figure 3–40

14. In the Sort drop-down list, select **Sequence, Step and Name**. The list of plies updates as shown in Figure 3–41.



*Name	Parent	Sequence	Step	Rosette	Specified Orientation
P001	Cowl	A	10	ROS001	0
P009	Cowl	A	15	ROS001	45
P015	Cowl	A	15	ROS001	45
P002	Cowl	A	20	ROS001	45
P010	Cowl	A	25	ROS001	0
P016	Cowl	A	25	ROS001	0
P003	Cowl	A	30	ROS001	90
P011	Cowl	A	35	ROS001	45
P017	Cowl	A	35	ROS001	45
P004	Cowl	A	40	ROS001	-45
P005	Cowl	A	50	ROS001	-45
P012	Cowl	A	55	ROS001	45
P018	Cowl	A	55	ROS001	45
P006	Cowl	A	60	ROS001	90
P013	Cowl	A	65	ROS001	0
P019	Cowl	A	65	ROS001	0
P007	Cowl	A	70	ROS001	45
P014	Cowl	A	75	ROS001	45
P020	Cowl	A	75	ROS001	45
P008	Cowl	A	80	ROS001	0

Figure 3–41

Task 5 - Rename the plies to match the layup order.

1. Click in the list of plies, press <Ctrl> + <A> to select all of the plies, right-click, and select **Modify**.

2. In the *Name* field, enter [P001,1] and press <Enter>.

3. Click OK.

Task 6 - Re-sequence the plies to increments of 1 using the Composite Sequence Manager.

1. Click (Composite Sequence Manager).

2. Click Sequence Options (Sequence Options) as shown in Figure 3–42.

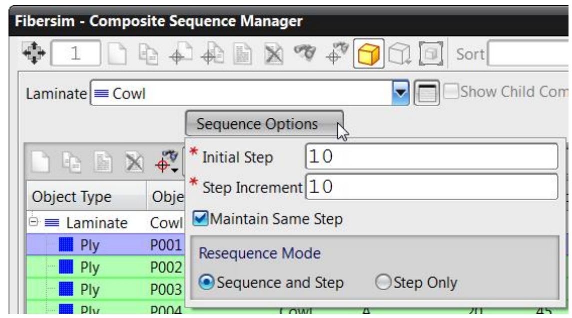


Figure 3–42

3. Modify the following parameters:

   - Initial Step: **1**
   - Step Increment: **1**

4. Select the **Cowl** laminate from the list and click (Resequence). The *Step* column updates with increments of 1 as shown in Figure 3–43.

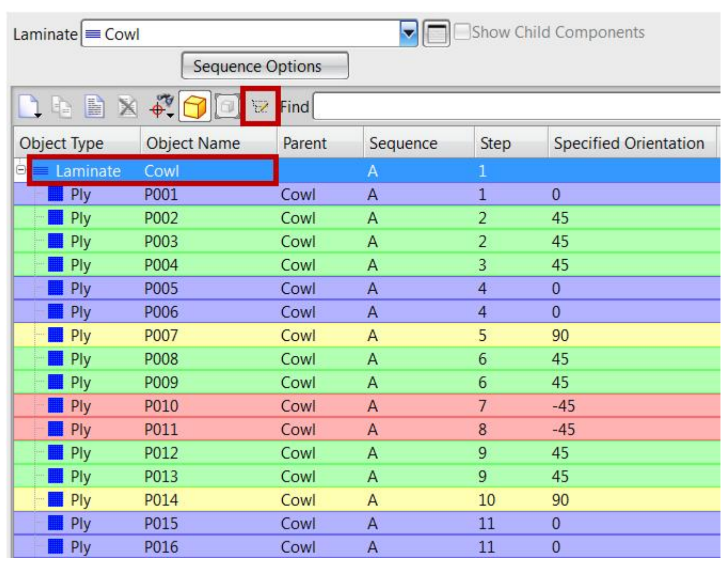


Figure 3–43

5. Click OK.

Task 7 - Create a Cutout.

1. In NX, press <Ctrl> + <L> to open the Layer Settings window.
2. Turn on the **Cutout** Layer Category to show Cutout geometry. The model displays the cutout geometry as shown in Figure 3–44.

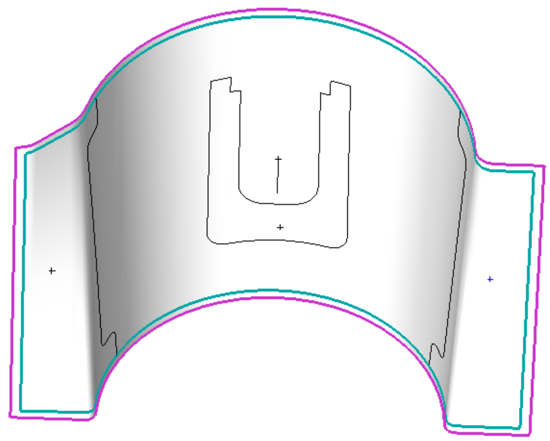


Figure 3–44

3. Click **Close**.
4. Press <F9> to start Fibersim.
5. In the *Application Tree*, select **Cutout** under **Basic** as shown in Figure 3–45.

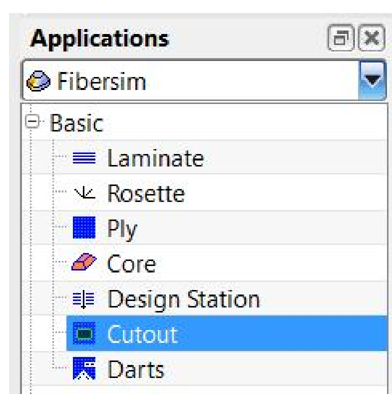


Figure 3–45

6. Click (Create New).
7. For the *Origin*, click (Link Geometry).
8. Make sure that the **Existing Point** selected and select **Cutout Origin**.

9. Click **OK**.

10. For the *Boundary*, click (Link Geometry).

11. Select **Cutout Boundary**.

12. Click **OK** twice to save **COUT001**.

Task 8 - Verify the results.

1. In the *Application Tree*, select **Ply** under **Basics**.

2. Highlight all of the plies in the list and click (Net Producibility) from the object toolbar area.

3. In the message window indicating that the material width has been exceeded, click **OK**.

4. In the *Application Tree*, select **Laminate** under **Basics**. Double-click on **Cowl** laminate to display its form.

5. Select the *Analysis* tab.

6. Click Update. The results display. Compare the **Net** results and the **with Cutout** results as shown in Figure 3–46.

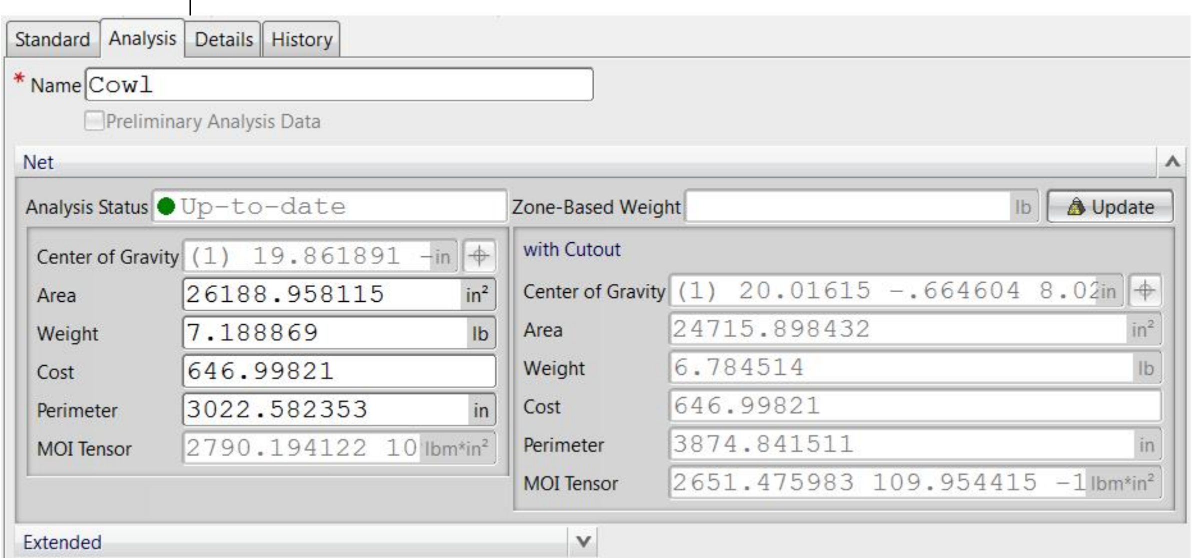


Figure 3–46

7. Click OK.

Task 9 - Create Design Stations and discuss the various types of core sample analysis.

User Guide Reference:
2.10 Design Stations

1. In the *Application Tree*, select **Design Station** under **Basic** as shown in Figure 3–47.

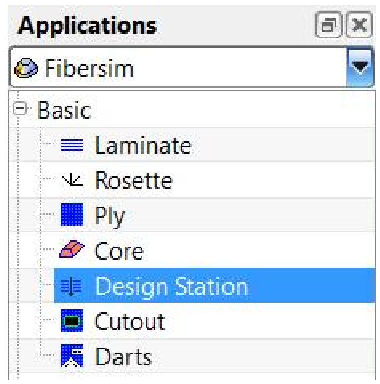


Figure 3–47

2. Click (Create New).
3. In the Design Station Standard form, in the Core Sample Type drop-down list, select **All**.
4. In the Design Station toolbar, click (Core Sample).
5. A Fibersim message window opens prompting you that the core sample has completed successfully as shown in Figure 3–48.

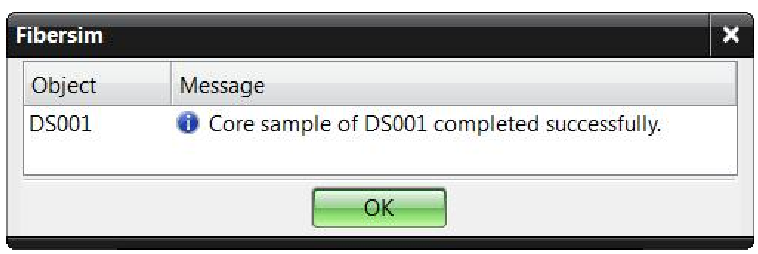


Figure 3–48

6. Click OK.

7. Core Sample Analysis report is displayed as shown in Figure 3–49.

Core Sample Analysis

Name: DS001
Parent: Cowl
Rosette: ROS001
Identifier: ROSETTE ORIGIN
Location: (19.185142, 0.095741, 18.677319)
Units: Inch (in)

Total Component Count: 8
Total Thickness: 0.060000

Name	Seq/Step	Orient	Material	Thickness
P001	A1	0	PPG-PL-3K	0.007500
P004	A3	45	PPG-PL-3K	0.007500
P007	A5	90	PPG-PL-3K	0.007500
P010	A7	-45	PPG-PL-3K	0.007500
P011	A8	-45	PPG-PL-3K	0.007500
P014	A10	90	PPG-PL-3K	0.007500
P017	A12	45	PPG-PL-3K	0.007500
P020	A14	0	PPG-PL-3K	0.007500

Figure 3–49

8. Scroll down to display the Detailed Design Station Information as shown in Figure 3–50.

Detailed Design Station Information

Name	Actual Fiber Direction		Relative Fiber Direction	
	Warp	Weft	Warp	Weft
P001	0.000000	90.000000	0.000000	0.000000
P004	45.000007	134.999993	45.000007	44.999993
P007	90.000000	179.999999	90.000000	89.999999
P010	-45.000007	45.000007	-45.000007	-44.999993
P011	-45.000007	45.000007	-45.000007	-44.999993
P014	90.000000	179.999999	90.000000	89.999999
P017	45.000007	134.999993	45.000007	44.999993
P020	0.000000	90.000000	0.000000	0.000000

Figure 3–50

9. Scroll down further to display the Laminate Rating Analysis as shown in Figure 3–51.

Laminate Rating Analysis

Rated Laminate Thickness:	0.060000
Rated Component Count:	(of 8)
% Symmetry:	100.000000
% Weighted Symmetry:	100.000000
% Mechanical Symmetry:	100.000000
% Laminate Balance:	100.000000
% Laminate Warpage:	0

List of Rated Components Only

Center	Name	Orient	Seq/Step	Material	Symmetry	Balance
	P001	0	A1	PPG-PL-3K	Yes	Yes
	P004	45	A3	PPG-PL-3K	Yes	Yes
	P007	90	A5	PPG-PL-3K	Yes	Yes
	P010	-45	A7	PPG-PL-3K	Yes	Yes
-C-						
	P011	-45	A8	PPG-PL-3K	Yes	Yes
	P014	90	A10	PPG-PL-3K	Yes	Yes
	P017	45	A12	PPG-PL-3K	Yes	Yes
	P020	0	A14	PPG-PL-3K	Yes	Yes

Figure 3–51

10. Click OK to close the report and return to the list view.
11. Close Fibersim.
12. Save the model.

Exercise 3c Flat Pattern

User Guide Reference:
2.5 Flat Pattern/Producibility Simulations

In this exercise, you will create a flat pattern. The completed model is shown in Figure 3–52.

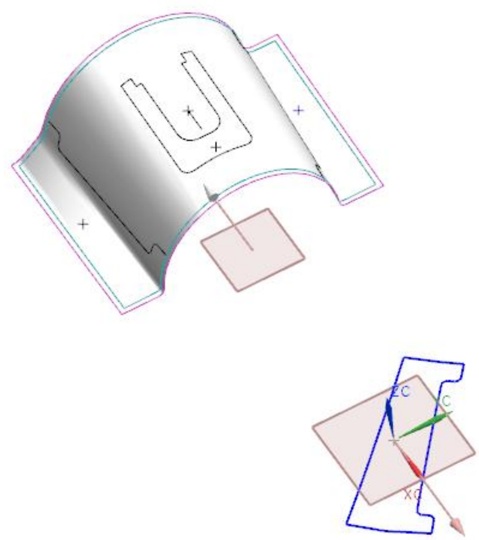


Figure 3–52

Goal

After you complete this exercise, you will be able to:

- ✓ **Run Producibility**
- ✓ **Generate a Flat Pattern**
- ✓ **Create the Flat Pattern Placement geometry in NX**
- ✓ **Set the Flat Pattern Placement Plane and Orientation**
- ✓ **Regenerate the Flat Pattern**

Estimated Time

15 min

Task 1 - Open a part.

1. Select **File > Open** and select **INTRO_PLY_C.prt**.

   If you completed Exercises 3a and 3b, you can continue working with **INTRO_PLY_A.prt** or **INTRO_PLY_B.prt** instead. The model displays as shown in Figure 3–53.

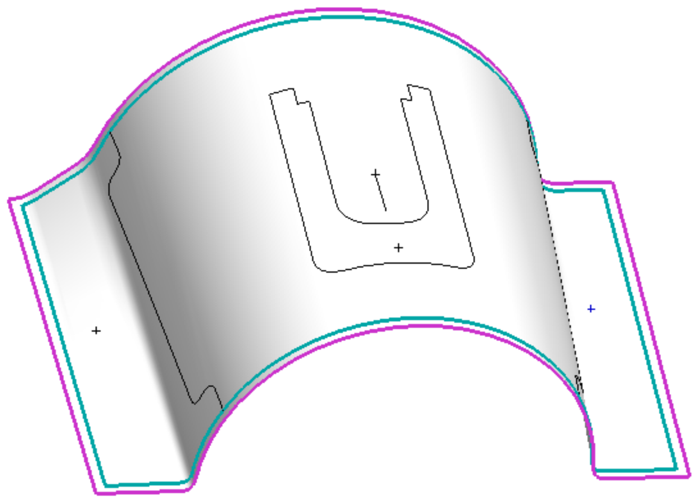


Figure 3–53

2. Launch Fibersim by pressing <F9>, if required.
3. In the *Application Tree*, select **Ply** under **Basics**.
4. Double-click on **P002** to display its details in the Standard form.

Task 2 - Generate Flat Pattern.

1. Click  (Net Producibility).
2. Click  (Generates the Net Flat Pattern).
3. Click  to save P002.

4. The Flat Pattern geometry is created on the default XY plane, as shown in Figure 3–54. Note that it is flipped in the opposite direction of the ply boundary direction.

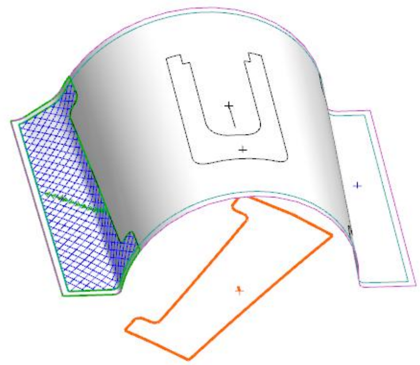


Figure 3–54

Task 3 - Create Flat Pattern Placement geometry in NX.

The default location for flat patterns can easily be changed. Fibersim will subsequently position all flat patterns directly onto a user-specified plane with a user-specified orientation. In this task, you will create the geometry in NX for defining a user-specified plane and orientation to be in the same direction as the ply boundary.

1. Close Fibersim.
2. In NX, click  (Display WCS) to turn on WCS.

*Alternatively, you can select **WCS > Dynamics** from the **Format** pull-down menu.*

3. Double click on WCS.
4. Move the WCS to an approximate location shown in Figure 3–55 and rotate X axis by 180° around the Z axis.

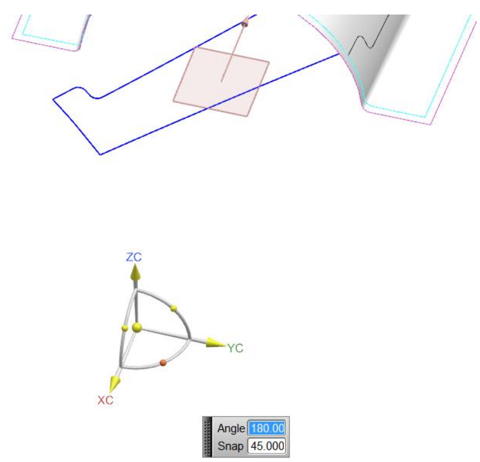
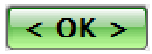


Figure 3–55

5. Click  (Datum Plane).
6. In the Type drop-down list, select **XC-YC plane**.
7. Click .
8. In the part navigator, rename the plane as [Net FP Placement Plane].
9. Click  (Datum Axis).
10. In the Type drop-down list, select **XC-Axis.**
11. Click .
12. In the part navigator, rename the datum axis as [Net FP Orientation].

Task 4 - Set the Flat Pattern Placement Plane and Orientation.

1. Press <F9> to start Fibersim.
2. In Fibersim, select **Tools > Options > Fibersim Options**. The Fibersim dialog box opens. Select the *Flat Patterns* tab if it is not already selected, as shown in Figure 3–56.

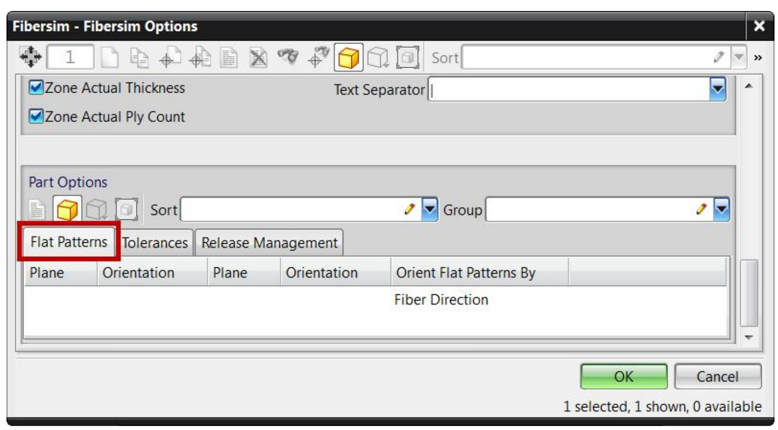


Figure 3–56

3. Double click on **Fiber Direction** to modify the Flat Patterns options.

4. In the Net Placement area, next to the *Plane* field, click (Link Geometry).
5. Select **Net FP Placement Plane** and Click **OK**.
6. In the Net Placement area, next to the *Orientation* field, click (Link Geometry).
7. Select **Net FP Orientation** and click **OK**.
8. In the Fibersim Options dialog box, click OK twice to save the change.

Task 5 - Regenerate Flat Pattern.

1. Highlight **P002** from the Ply List.
2. Click (Generates the Net Flat Pattern) on the toolbar.
3. The Flat Pattern updates. as shown in Figure 3–57. Note that it is now flipped in the correct direction, aligned with the ply boundary direction.

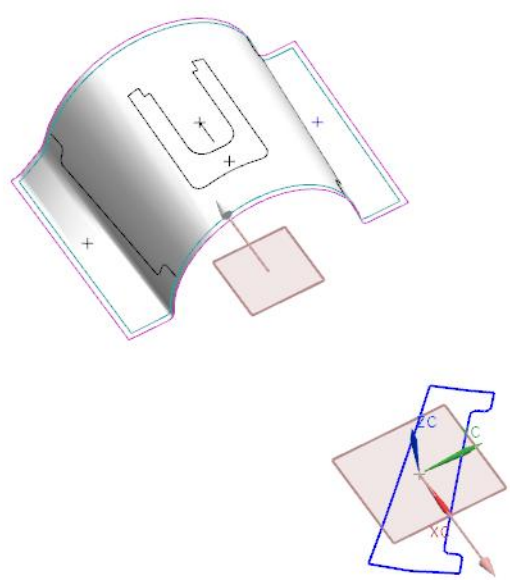


Figure 3–57

4. Close Fibersim and save the model.

Exercise 3d Design to Extended

User Guide References:
2.6.2 Laminates Form

In this exercise, you will explore the Design to Extended (Design to Manufacturing) method. You will create a ply object for the pad-up areas indicated in Figure 3–58 using the Design to Net (Design to Engineering) method first. After reviewing the result, you will do the same using the Design to Extended method and compare the results.

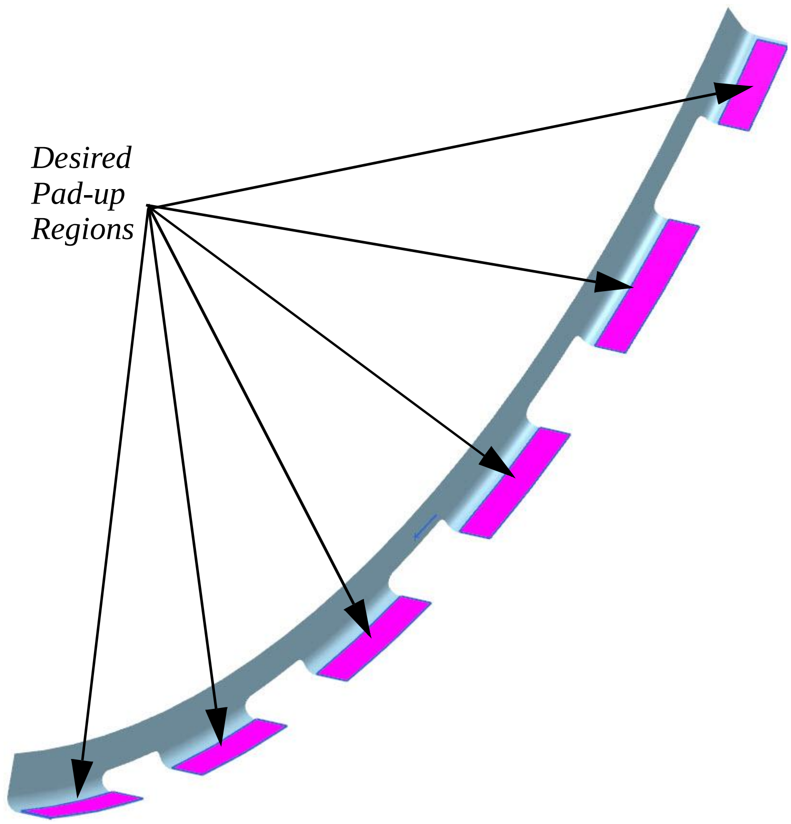


Figure 3–58

After you complete this exercise, you will be able to:

- ✓ **Understand when to use the Design to Manufacturing method vs. the Design to Engineering method**
- ✓ **Create a Laminate for the Design to Manufacturing method**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **INTRO_PLY_D.prt**. The model displays as shown in Figure 3–59.

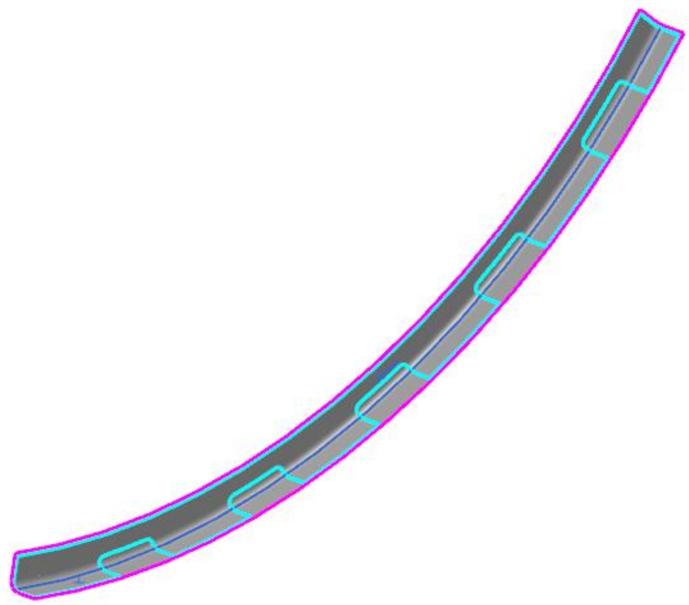


Figure 3–59

2. Press <F9> to start Fibersim.

Task 2 - Create a Design to Net Laminate and a New Rosette.

1. In the *Application Tree*, select **Laminate** under **Basics**.
2. Create a new laminate with the following parameters as shown in Figure 3–60:
    - Name: **Engineering Lam**
    - Step: **1**
    - Default Material: **PPG-PL-3K**
    - Layup Surface: **TOOL SURFACE**
    - Net Boundary: **NET BOUNDARY** (cyan curve)
    - Extended Boundary: **EXTENDED BOUNDARY** (magenta curve)

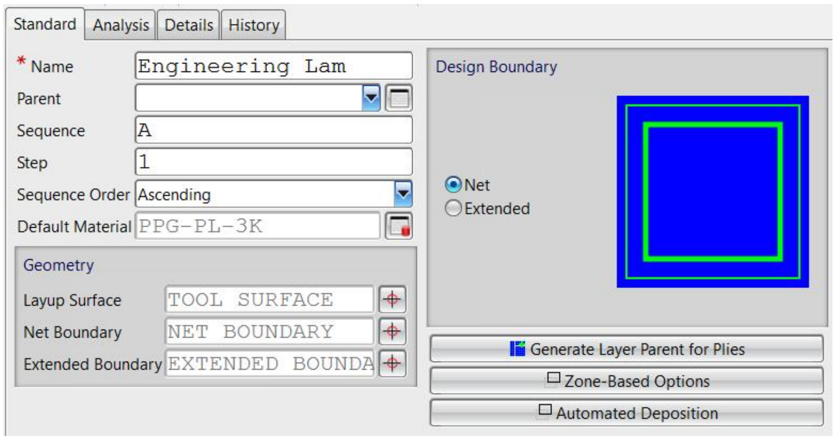


Figure 3–60

3. Click .

4. In the *Application Tree*, select **Rosette** under **Basics**.

5. Create a new rosette with the following parameters as shown in Figure 3–61:

   - Origin: **ROSETTE ORIGIN**
   - Direction: **ZERO DIRECTION**

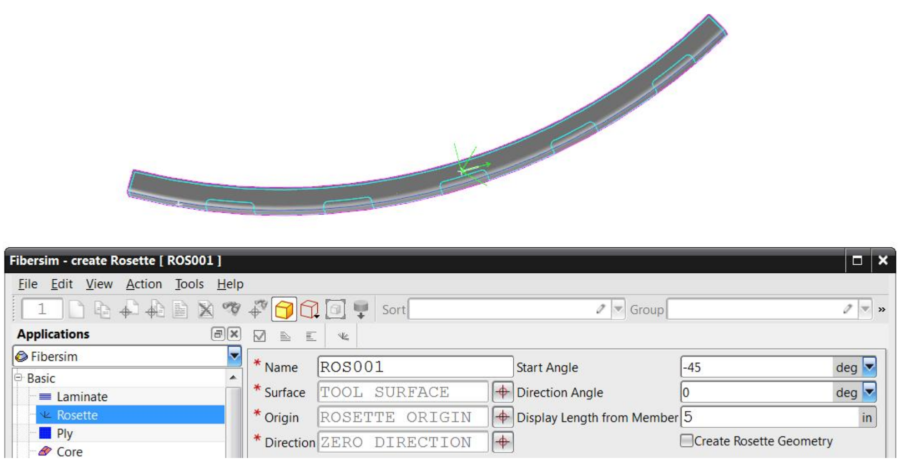
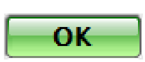


Figure 3–61

6. Click .

Task 3 - Create a New Ply for the Pad-up Area.

1. In the *Application Tree*, select **Ply** under **Basics**.

2. Click  (Create New).

3. Enter [**10**] for *Step*.

4. Select the indicated point for *Origin* as shown in Figure 3–62.

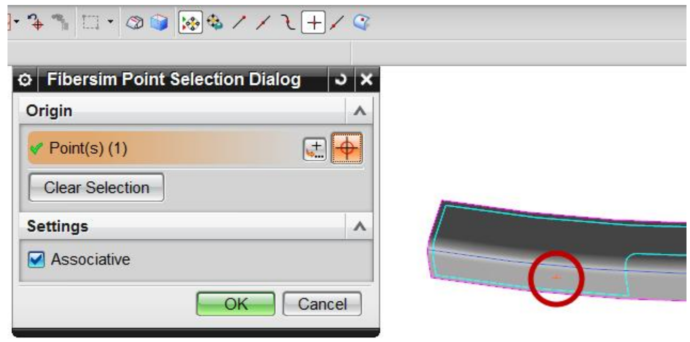
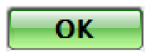


Figure 3–62

5. Click .

6. For *Boundary*, select the indicated curve as shown in Figure 3–63.

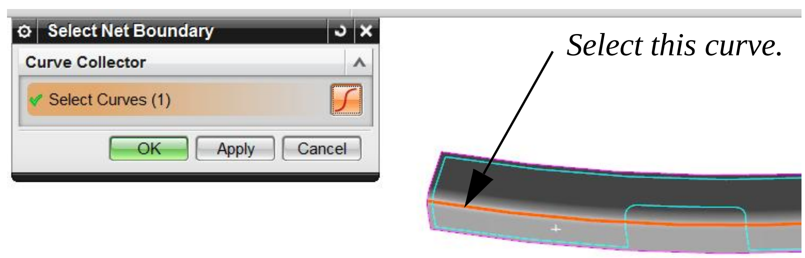


Figure 3–63

7. Click .

8. Click  (Extended Producibility) in the ply toolbar area.

9. Review the model and note that the Extended Boundary is not ideal as shown in Figure 3–64.

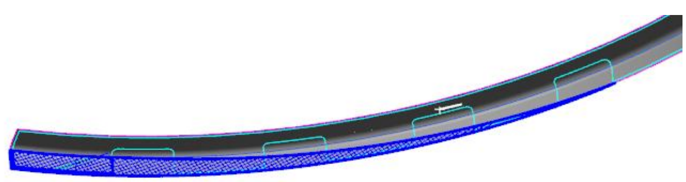
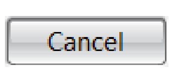


Figure 3–64

10. Click .

Task 4 - Create a New Laminate.

1. In the *Application Tree*, select **Laminate** under **Basics**.
2. Right-click on **Engineering Lam** and select **Create Based On**.
3. Modify the following parameters as shown in Figure 3–65:
   - Name: **Manufacturing Lam**
   - Design Boundary: **Extended**

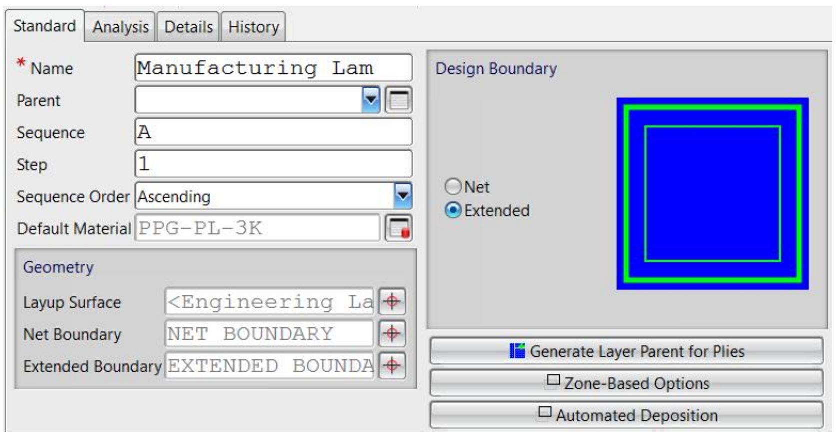
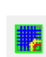


Figure 3–65

4. Click 

.

Task 5 - Create a New Ply.

1. In the *Application Tree*, select **Ply** under **Basics**.
2. Click  (Create New).
3. Enter the following parameters:
   - Parent: **Manufacturing Lam**
   - Step: **10**
4. Link the same point and curve for *Origin* and *Boundary* under **Extended** as outlined in **Task 3**.
5. Click  (Extended Producibility) to generate Extended Producibility.

6. The model should display as shown in Figure 3–66. Note that the pad-up areas are covered with one continuous ply by using the Design to Extended method, which is how the part will be manufactured.

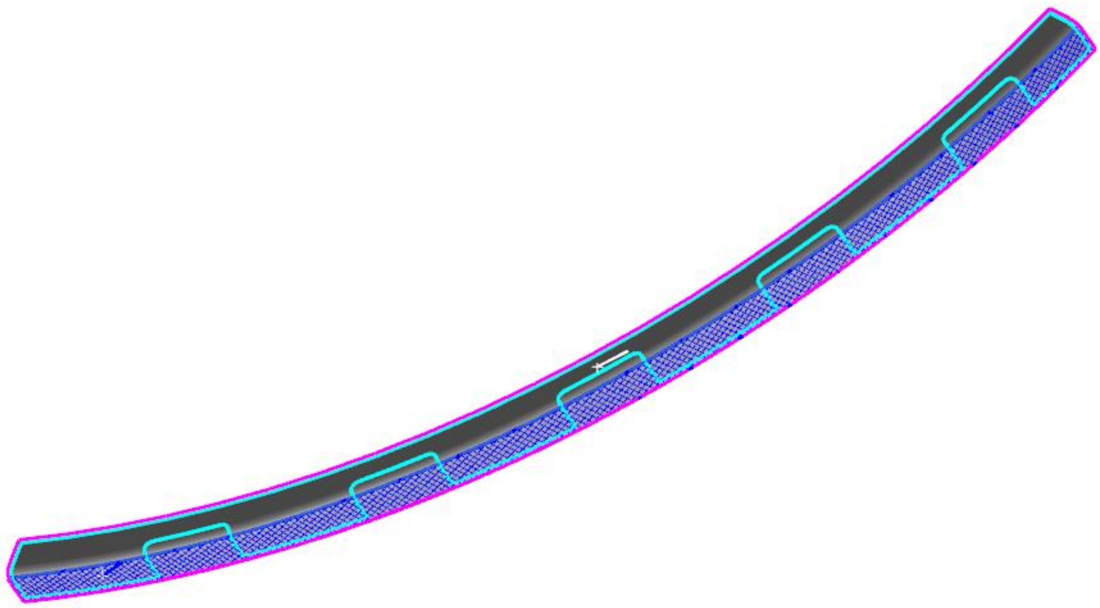


Figure 3–66

7. Click on the *Analysis* tab. Review and compare the analysis results between Net and Extended.

8. Click **OK** to save the ply object.

9. Close Fibersim and save the model.

Exercise 3e Wrapped Ply

In Fibersim, you can now define a ply that overlaps itself by selecting the "Wrapped" checkbox and defining a Start and End curve for the overlapping region. In this exercise, you will define a simple wrapped ply.The completed model is shown as in Figure 3–67.

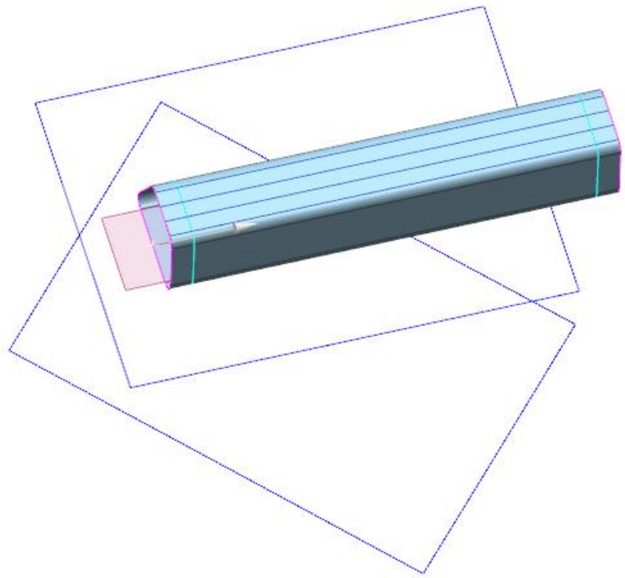


Figure 3–67

Goal

After you complete this exercise, you will be able to:

- ✓ **Define a ply that overlaps itself**
- ✓ **Run Producibility**
- ✓ **Generate a Flat Pattern**

Estimated Time

10 min

Task 1 - Open a part.

1. Select **File > Open** and select **WRAPPED_PLY.prt**. The model displays as shown in Figure 3–68.

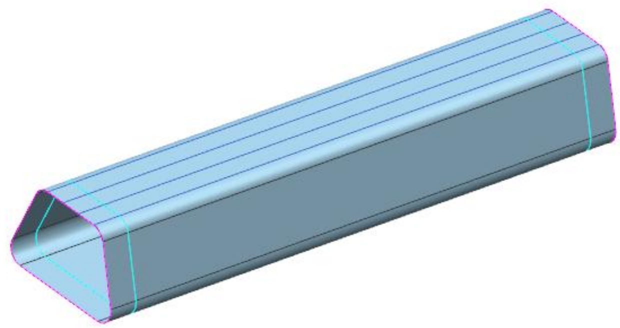


Figure 3–68

Task 2 - Create a Laminate and Link Geometry to CATIA.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate**.
3. Click (Create New).
4. Link or enter the following parameters:
   - Step: **1**
   - Material: **PPG-PL-3K**
   - Layup Surface: **Laminate** (blue surface)
   - Net Boundary: **Net 1, Net 2** (cyan curves)
   - Extended Boundary: **Extended 1, Extended 2** (magenta curves)
5. Click OK.

Task 3 - Create a Rosette and Link Geometry to CATIA.

1. In the *Application Tree*, select **Rosette**.
2. Click (Create New).

3. Link or enter the following parameters:
   - Origin: **Origin** (located at the bottom of the surface)
   - Direction: **Direction** (located at the bottom of the surface)

4. Click .

5. **ROS001** displays in model as shown in Figure 3–69.

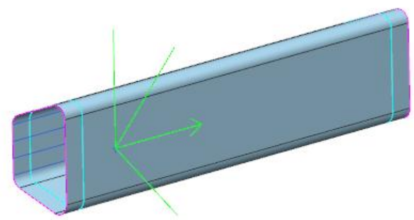


Figure 3–69

Task 4 - Create a new Wrapped Ply.

1. In the *Application Tree*, select **Ply**.

2. Click  (Create New).

3. Enter [10] for *Step*.

4. On the Standard tab, select the checkbox for Wrapped as shown in Figure 3–70.

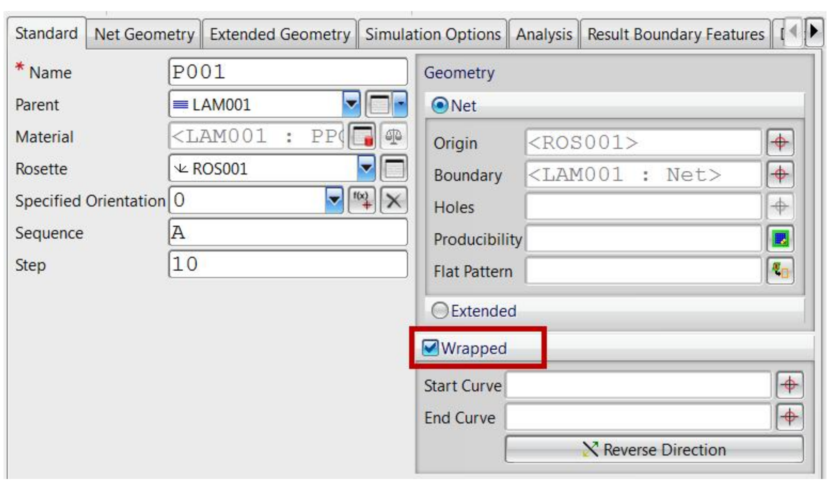
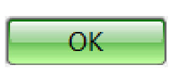


Figure 3–70

5. Click  to close the simulation message.

6. Click (Link Geometry) next to *Start Curve*.
7. Select **Start.1** as indicated in Figure 3–71.

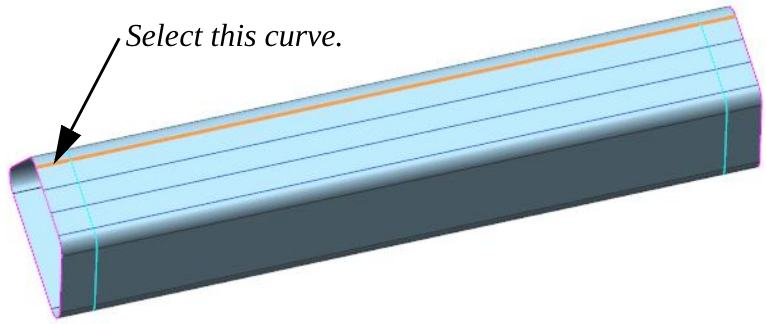


Figure 3–71

8. Click OK.
9. Link **End.1** for *End Curve* as indicated in Figure 3–72.

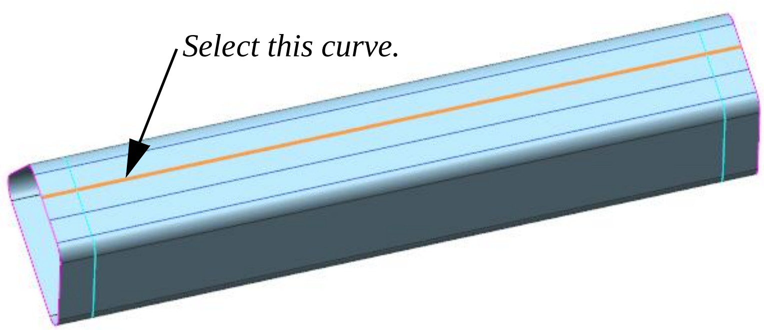


Figure 3–72

Task 5 - Run simulation and generate a flat pattern.

1. Click (Extended Producibility) and review the result in NX.
2. Click (Extended Flat Pattern).
3. Click OK to save **P001**.

Task 6 - Create another ply with orientation of 45.

1. Click (Create New).
2. Click (Link Geometry) next to *Boundary* under Geometry > Net.
3. Select **Start.2** (third curve from the top).
4. Click OK.
5. Fibersim gives you a warning message as shown in Figure 3–73.

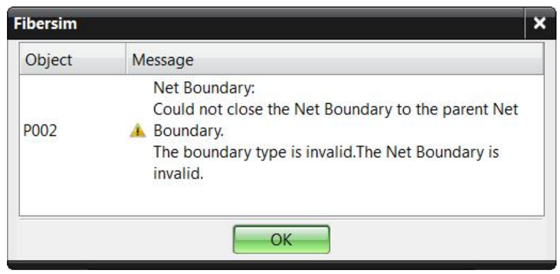


Figure 3–73

6. Click OK to close the message window.
7. Back in the Fibersim window, note that the Wrapped option is not available because the ply no long has a valid two-domain boundary.
8. Click (Link Geometry) next to *Boundary* under Geometry > Net.
9. Click (Deselect All).
10. Click OK.
11. Click the checkbox for **Wrapped**.
12. Click OK to close the message about simulation.
13. Link the following geometry to the parameters:
    - Start Curve: **Start.2** (third curve from the top)
    - End Curve: **End.2** (the last curve)

14. Enter the following parameters:

    - Specified Orientation: 45
    - Step: 20

15. Click (Extended Producibility) and review the result in CATIA.

16. Click (Extended Flat Pattern).

17. Click OK to save **P002**.

18. Close Fibersim.

19. Save and close the model.

Exercise 3f Create Basic Part

User Guide Reference:
2.6 Laminates
2.7 Rosettes
2.8 Plies
2.5 Flat Pattern/Producibility Simulations

In this exercise, with minimal instruction, you will cover a given model with full-body plies. The final flat pattern should display as shown in Figure 3–74.

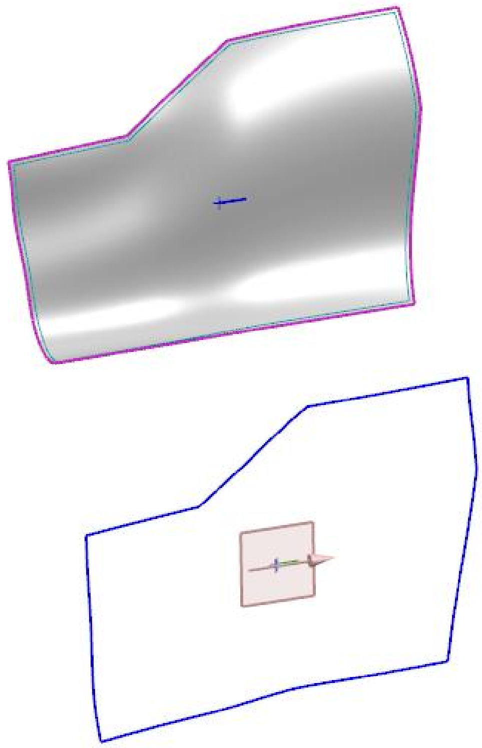


Figure 3–74

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a Laminate**
- ✓ **Create a Rosette**
- ✓ **Create four full-body plies**
- ✓ **Run Producibility**
- ✓ **Generate a Flat Pattern**
- ✓ **Set the Flat Pattern Placement Plane and Orientation**
- ✓ **Regenerate a Flat Pattern**

Estimated Time

10 min

Task 1 - Open a part.

1. Select **File > Open** and select **INTRO_PLY_E.prt**. The model displays as shown in Figure 3–75.

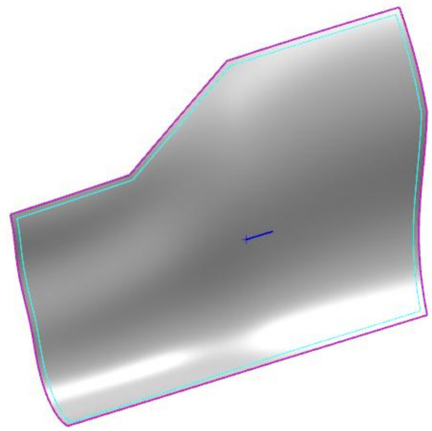


Figure 3–75

Task 2 - Create a Laminate and Link Geometry to NX.

1. In Fibersim, create a Laminate with the following parameters:
   - Name: **Skin**
   - Step: **1**
   - Material: **PPG-PL-3**
2. Link the following geometry as shown in Figure 3–76:

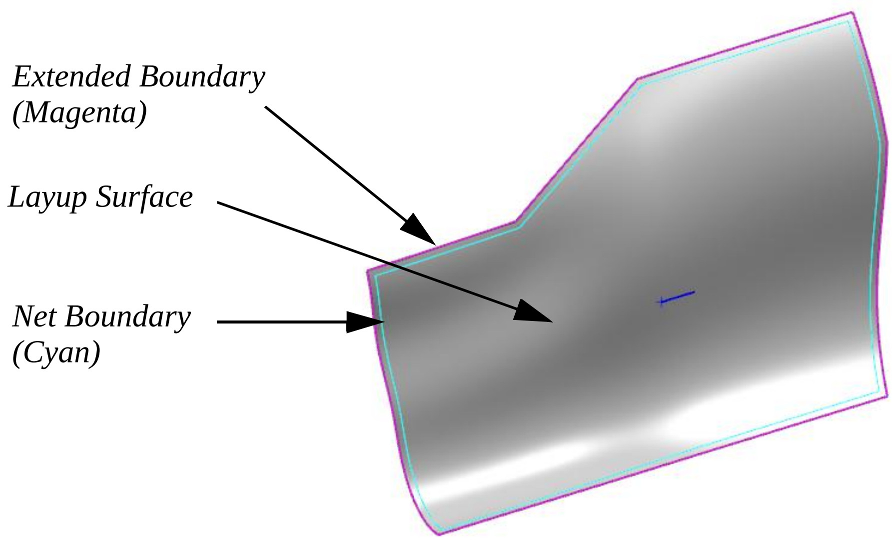


Figure 3–76

3. The Laminate's Standard form should update as shown in Figure 3–77.

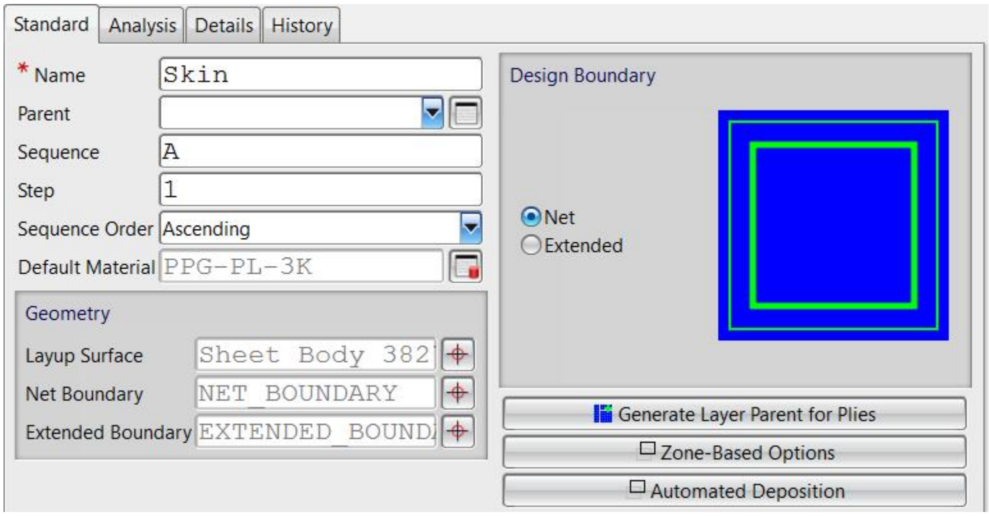


Figure 3–77

Task 3 - Create a Rosette.

1. Create a Rosette named [ROS001] and link the following geometry as noted in Figure 3–78:

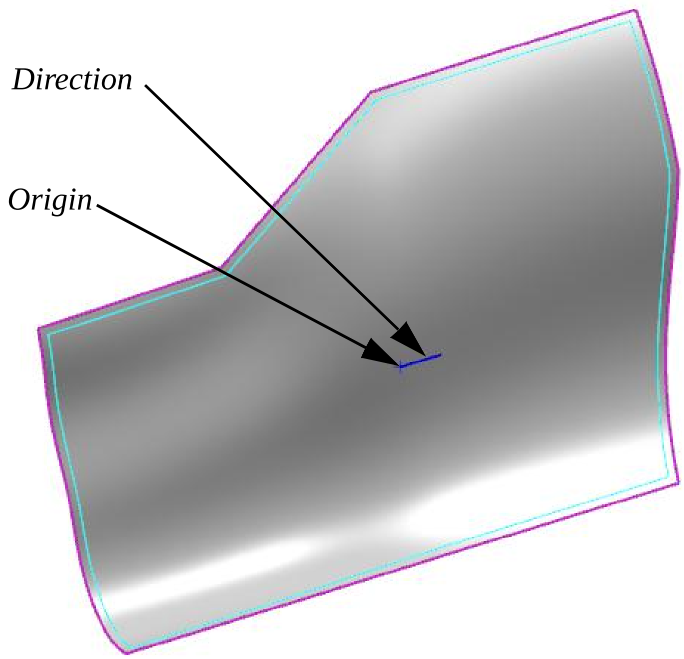


Figure 3–78

Task 4 - Create four full-body plies.

1. Create four plies with a *Step* of [10,10] and the following orientations:
   - P001: **0**
   - P002: **45**
   - P003: **-45**
   - P004: **90**

Task 5 - Run Producibility.

1. Run Net Producibility on P001.

Task 6 - Generate a Flat Pattern.

1. Using P001, generate the Net Flat Pattern. It should display as shown in Figure 3–79.

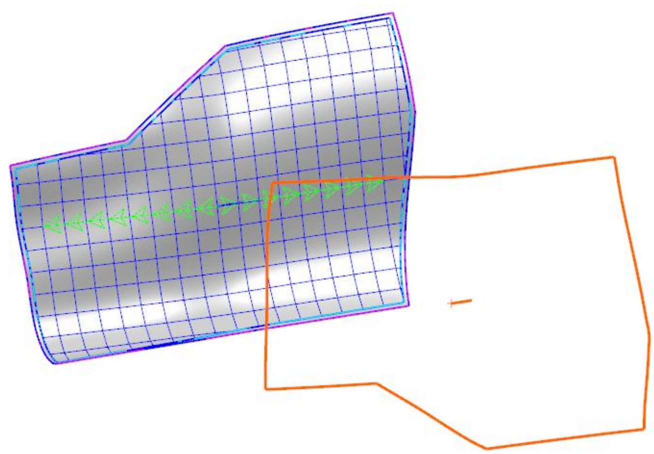


Figure 3–79

Task 7 - Set the Flat Pattern Placement Plane and Orientation.

1. Change the default Flat Pattern placement by linking the following geometry from the NX Part Navigator:
   - Plane: **Net FP Placement Plane**
   - Orientation: **Net FP Orientation**

Task 8 - Regenerate the Flat Pattern.

1. Generate the Net Flat Pattern again using P001. It should now display as shown in Figure 3–80.

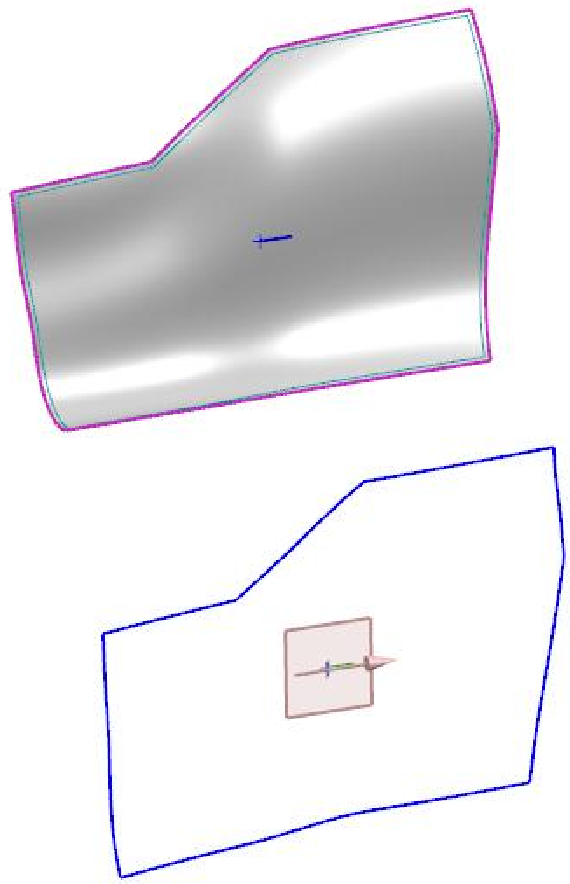


Figure 3–80

Chapter 4

Producibility

This chapter includes:

- ✓**Exercise 4a: How Geometry Affects Producibility**
- ✓**Exercise 4b: How Simulation Options Affect Producibility (Simulation Skin)**
- ✓**Exercise 4c: How Simulation Options Affect Producibility (Two-Stage)**
- ✓**Exercise 4d: Resolving Producibility Issues**

Exercise 4a How Geometry Affects Producibility

User Guide Reference:
2.5 Flat Pattern/ Producibility Simulations
2.12 Darting
B.3.7 Splice Ply

In this exercise, you will see how geometry affects producibility by changing ply origins and orientations. You will also create splices and darts to resolve producibility issues.

Goal

After you complete this exercise, you will be able to:

- ✓ **Change the ply origin point to observe the impact on producibility and flat pattern**
- ✓ **Change the ply orientation to observe the impact on producibility and flat pattern**
- ✓ **Create a Splice to resolve material roll width issues**
- ✓ **Create Slit darts and V-shape darts to resolve bridging and wrinkling**

Estimated Time

15 min without optional steps

30 min with optional steps

Task 1 - Open a part.

1. Open **PRODUCIBILITY_A.prt**. The model displays as shown in Figure 4–1.

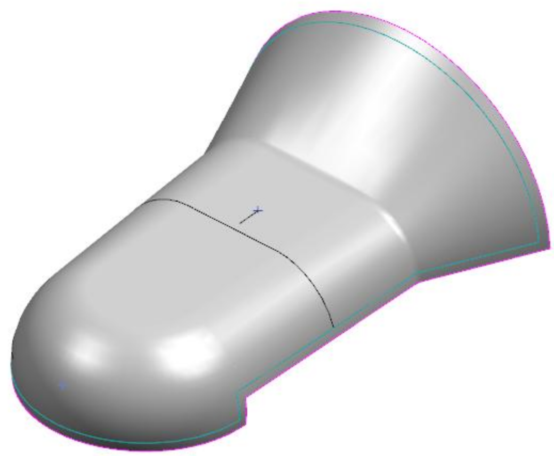


Figure 4–1

Task 2 - Create a Flat Pattern of P001.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Ply** under **Basic**.
3. Select **P001** from the list.
4. Click (Net Producibility) in the ply toolbar and click OK in the message box prompting you that the material width has been exceeded.
5. Click (Generates the Net Flat Pattern).
6. Click OK to save.

7. The **P001** producibility results and flat pattern geometry display as shown in Figure 4–2.

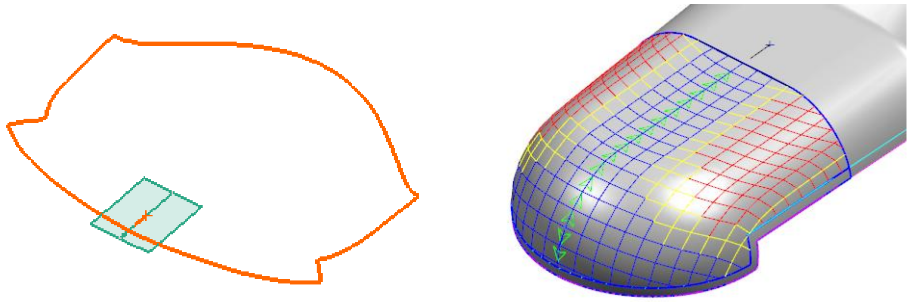


Figure 4–2

Task 3 - Create P002 from P001 and specify a different origin.

In this task you will create a new ply, **P002**, based on **P001** and change its origin. You will then note how the results compare with **P001**.

1. Right-click on **P001** and select **Create Based On**.
2. In the *Step* field, enter [20].
3. Next to the *Origin* field, click  (Link Geometry)
4. Press <Ctrl> + <L> to open the Layer Settings window.
5. Make sure that the option for Category Display is turned on, and turn on the **P002 GEOM** category as shown in Figure 4–3.

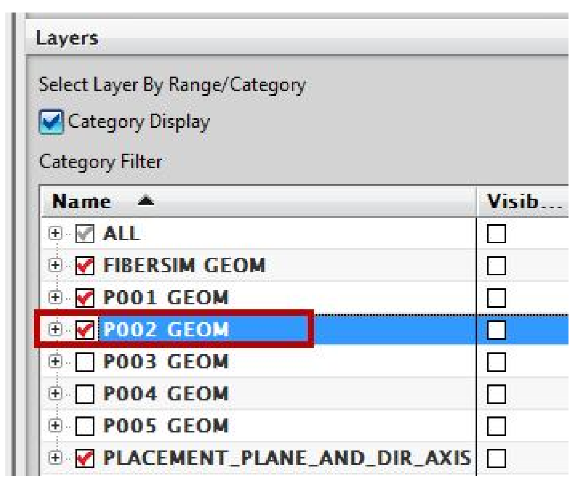


Figure 4–3

6. Click Close .

7. Make sure that the **Existing Point** filter is on as shown in Figure 4–4.

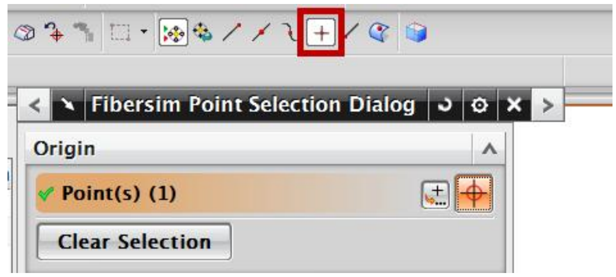


Figure 4–4

8. Select **P002 Origin** as noted in Figure 4–5.

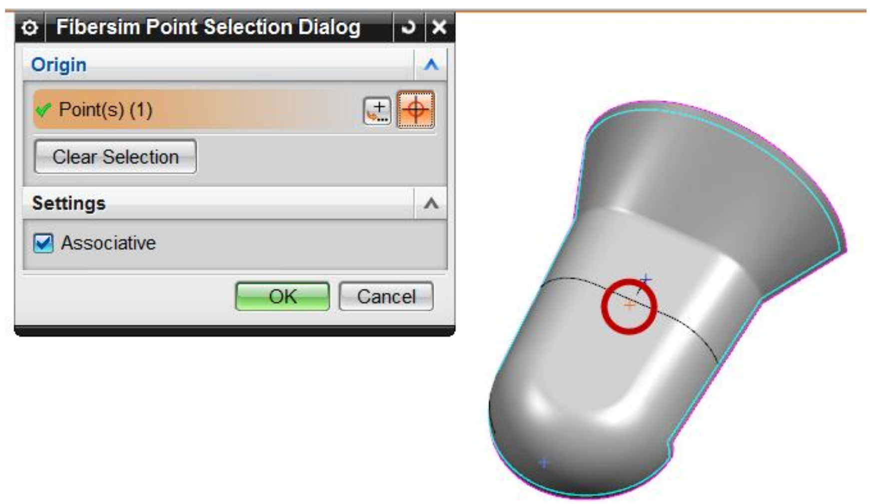


Figure 4–5

9. Click OK twice save **P002**.

10. Click (Net Producibility) in the ply toolbar and click OK in the message box prompting you that the material width has been exceeded.

11. Click  (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P002**. The results display as shown in Figure 4–6. **P001** Producibility is shown for comparison purposes.

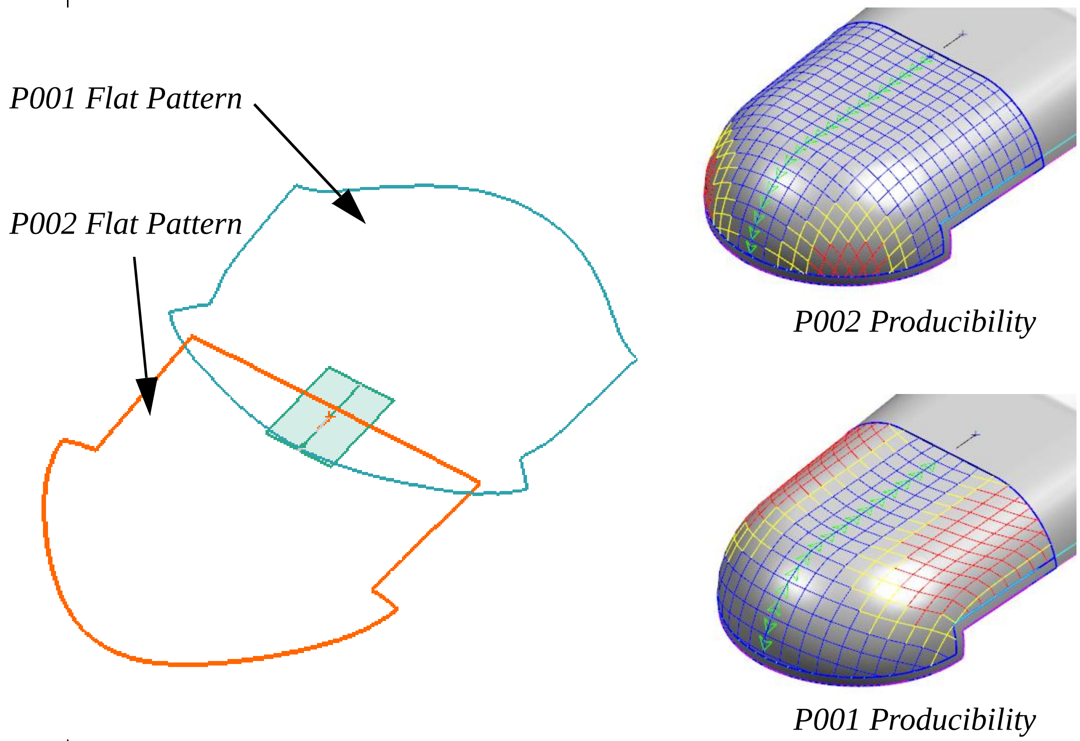
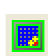
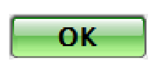


Figure 4–6

Task 4 - Create P003 from P002 and specify a different orientation.

In this task you will create a new ply, **P003**, based on **P002** and change its origin. You will then note how the results compare with **P002**.

1. In the ply list, select **P002**, right-click and select **Create Based On**.
2. In the *Step* field, enter [30].
3. In the Specified Orientation drop-down list, select **45**.
4. Click  (Net Producibility) and click  in the message box prompting you that the material width has been exceeded.

5. Create a Net Flat Pattern of **P003**. The results display as shown in Figure 4–7. **P002** Producibility is shown for comparison purposes.

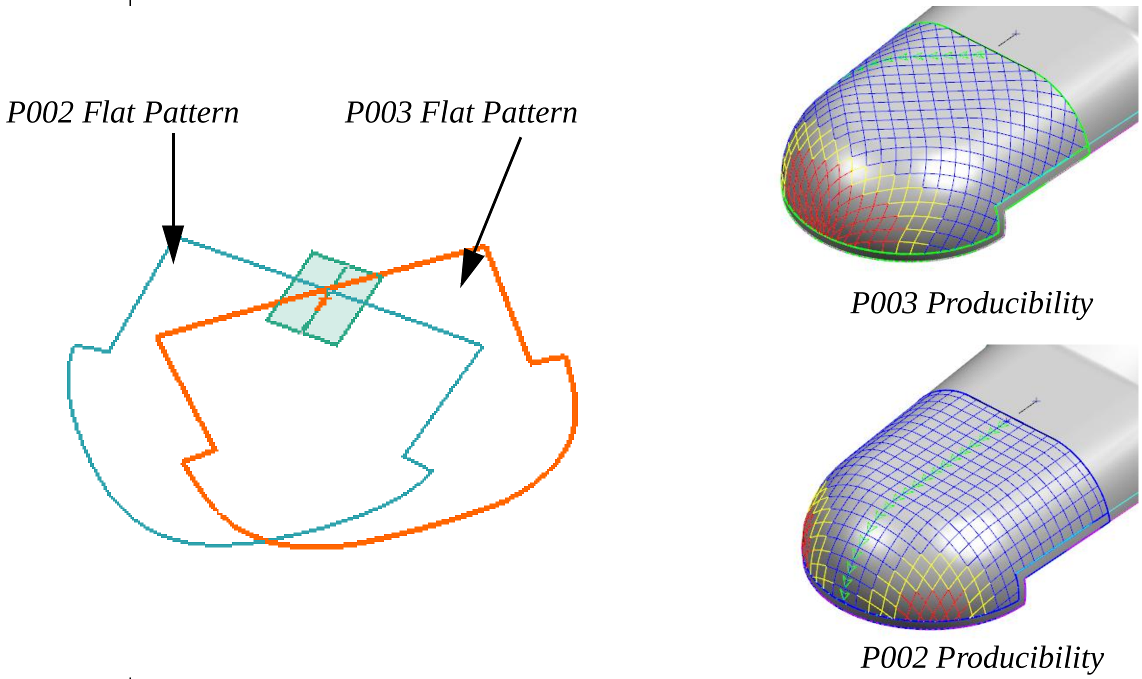


Figure 4–7

6. Click **OK** to save **P003**.

7. A comparison between the flat patterns and producibility of the 0 degree **P002** ply and the 45 degree **P003** ply is shown in Figure 4–8.

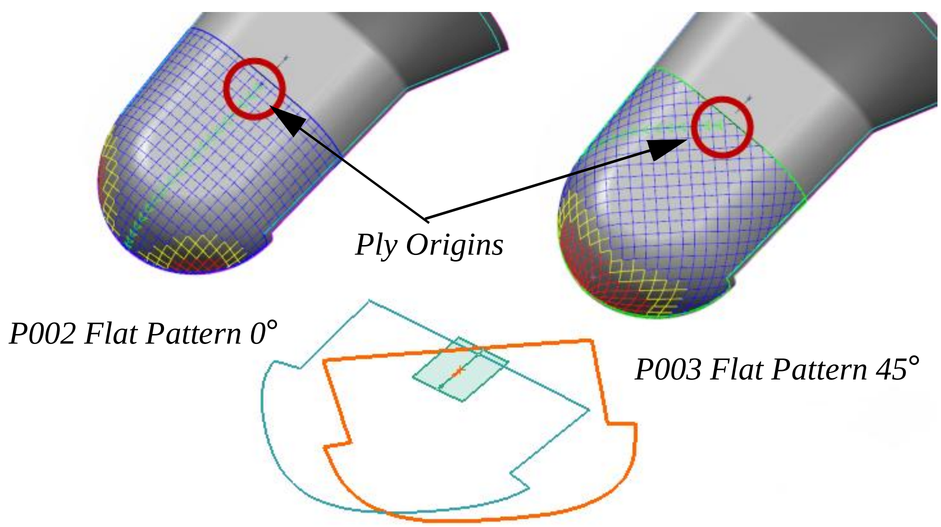


Figure 4–8

Task 5 - Create P004 from P002 and specify a width offset.

In this task, you will create **P004** based on **P002** and address the exceeded material width issue by specifying a width offset and splicing **P004** into two plies: **P004A** and **P004B**.

For manufacturing purposes, it might be more applicable to run ***Extended Producibility*** *to allow for the actual width of the bolt of material.*

1. In the ply list, select **P002**, right-click and select **Create Based On**.
2. In the *Step* field, enter [40].
3. Click (Net Producibility) and click OK in the message box prompting you that the material width has been exceeded.
4. Select the *Net Geometry* tab and expand Material Width Lines area as shown in Figure 4–9. Note that the *Material Width Offset* is 0.

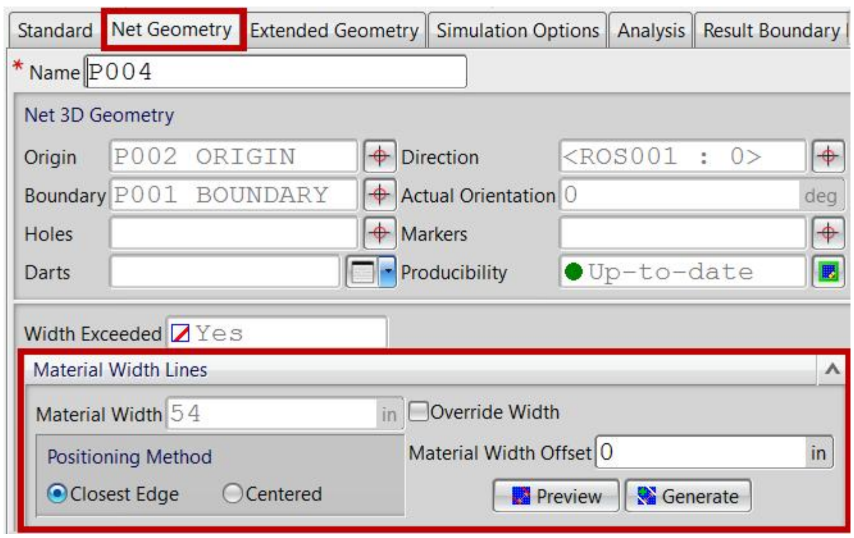


Figure 4–9

5. Click Preview. The material width offset with a value of 0 displays in NX as shown in Figure 4–10.

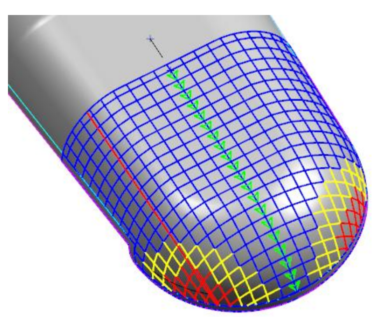


Figure 4–10

6. A recommended best practice is to set the material width line to be as centered as possible on the part for strength purposes. In the open Fibersim - Material Width Lines dialog box, enter [-18] inches (-457.2 mm). Click Preview to display the material width line to the center as shown in Figure 4–11.

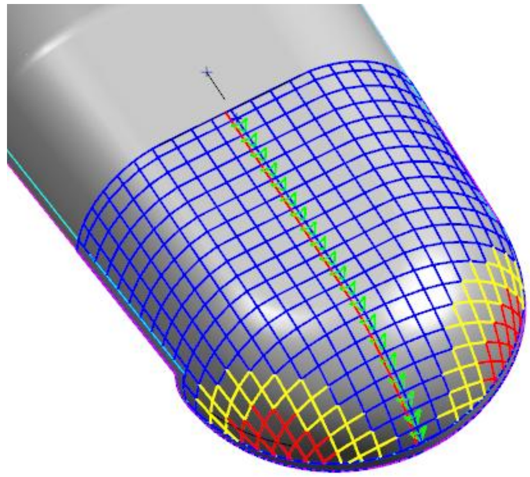


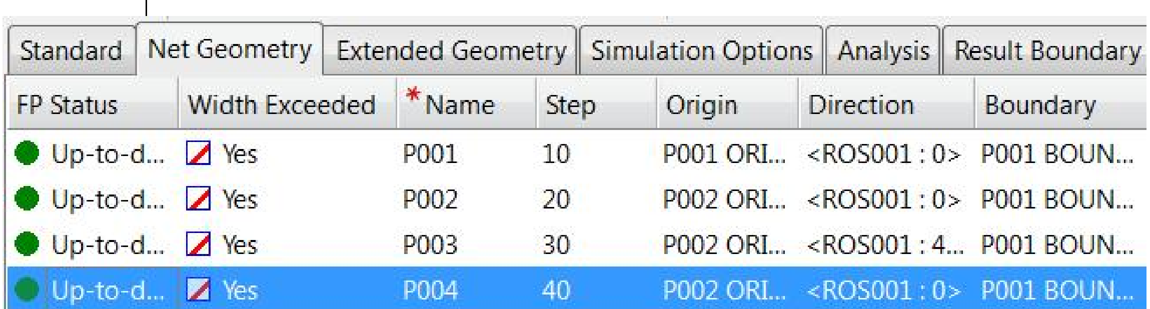
Figure 4–11

7. Click Generate.

8. Click OK to save **P004**.

Task 6 - Create a Splice Ply of P004A and P004B.

1. Back in Fibersim, list of plies are shown in Figure 4–12.



Standard | Net Geometry | Extended Geometry | Simulation Options | Analysis | Result Boundary

FP Status	Width Exceeded	*Name	Step	Origin	Direction	Boundary
Up-to-d...	Yes	P001	10	P001 ORI...	<ROS001 : 0>	P001 BOUN...
Up-to-d...	Yes	P002	20	P002 ORI...	<ROS001 : 0>	P001 BOUN...
Up-to-d...	Yes	P003	30	P002 ORI...	<ROS001 : 4...	P001 BOUN...
Up-to-d...	Yes	P004	40	P002 ORI...	<ROS001 : 0>	P001 BOUN...

Figure 4–12

2. Highlight **P004** and click (Splice Ply). The Fibersim - Splice Ply dialog box opens as shown in Figure 4–13.

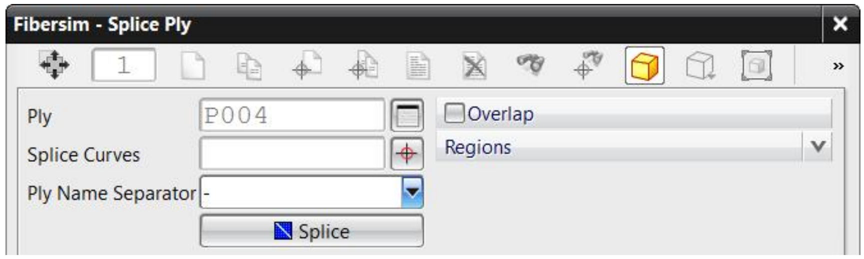


Figure 4–13

3. Next to the *Splice Curves* field, click (Link Geometry).
4. Select the material width curve as shown in Figure 4–14.

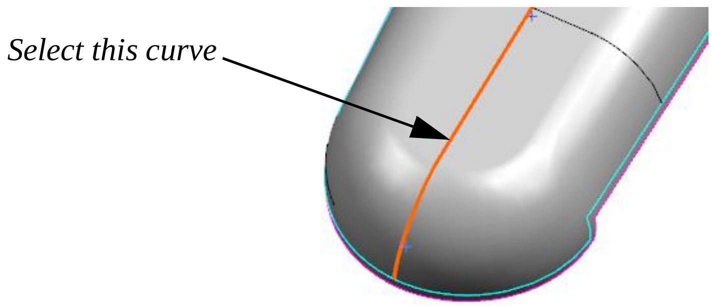


Figure 4–14

5. Click OK. The Fibersim - Splice Ply dialog box updates with the selected curve as shown in Figure 4–15.

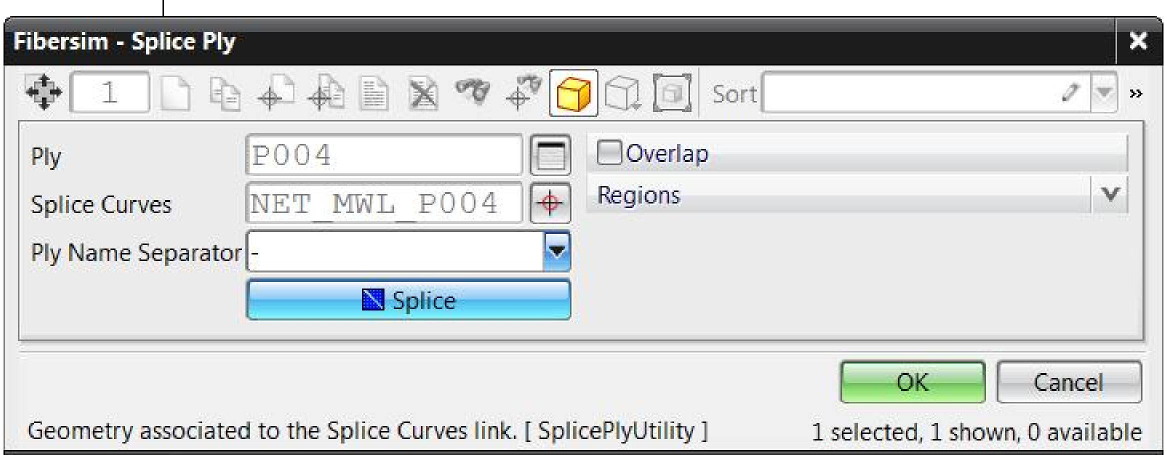


Figure 4–15

6. Click Splice to perform the splice.

7. The Fibersim message box opens, prompting you that two plies were spliced successfully as shown in Figure 4–16.

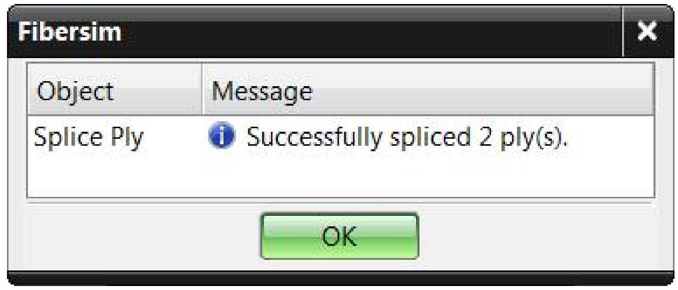
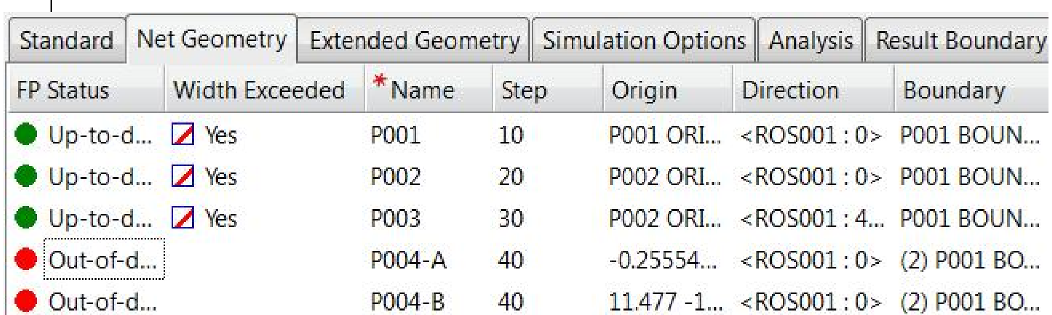


Figure 4–16

8. Click **OK** twice.

9. The **P004** spliced plies display in the ply list as **P004-A** and **P004-B** as shown in Figure 4–17.



Standard | Net Geometry | Extended Geometry | Simulation Options | Analysis | Result Boundary

FP Status	Width Exceeded	*Name	Step	Origin	Direction	Boundary
Up-to-d...	Yes	P001	10	P001 ORI...	<ROS001 : 0>	P001 BOUN...
Up-to-d...	Yes	P002	20	P002 ORI...	<ROS001 : 0>	P001 BOUN...
Up-to-d...	Yes	P003	30	P002 ORI...	<ROS001 : 4...	P001 BOUN...
Out-of-d...		P004-A	40	-0.25554...	<ROS001 : 0>	(2) P001 BO...
Out-of-d...		P004-B	40	11.477 -1...	<ROS001 : 0>	(2) P001 BO...

Figure 4–17

If the generated origin is suitable, you can use them. However, in many cases you will have to select a suitable origin that is based on the size of the part, which can limit the access to certain areas.

10. In the ply list, highlight **P004-A** and **P004-B**. The plies are highlighted in blue on the NX model. Note that the Splice utility generates an origin at the centroid of the ply that did not contain the origin as shown in Figure 4–18.

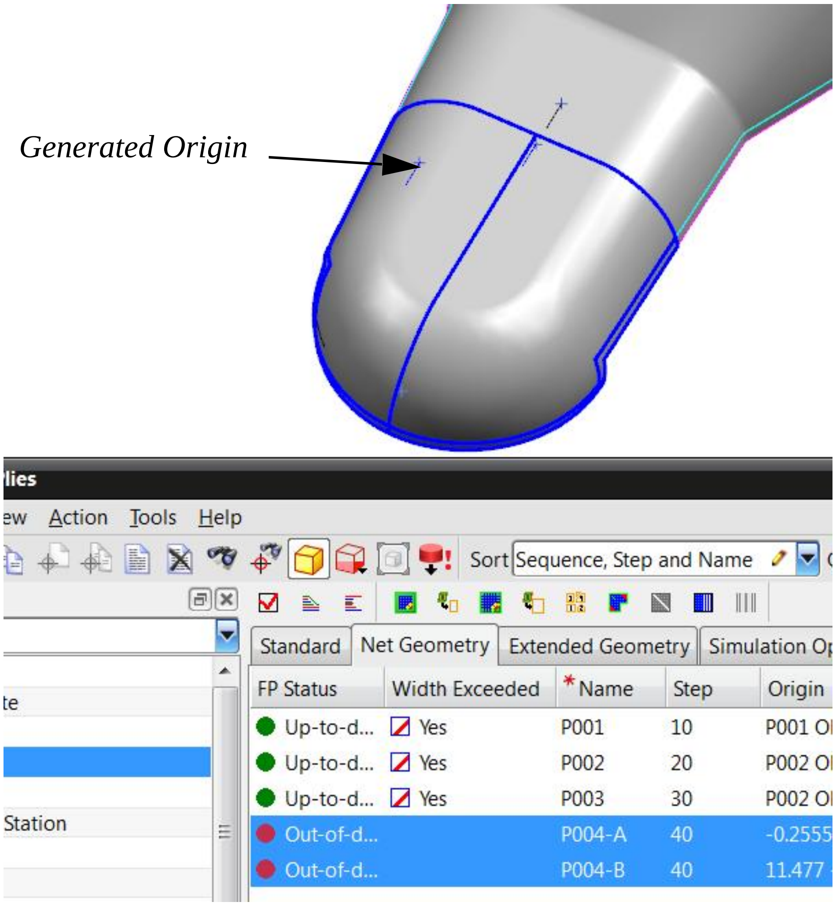


Figure 4–18

Task 7 - Move the origins of P004A and P004B to a new location.

In this task, you will move the generated origins created with **P004A** and **P004B** to more reliable origins, resulting in a better producibility.

1. Highlight **P004-A** and **P004-B**, right-click and select **Modify** as shown in Figure 4–19.

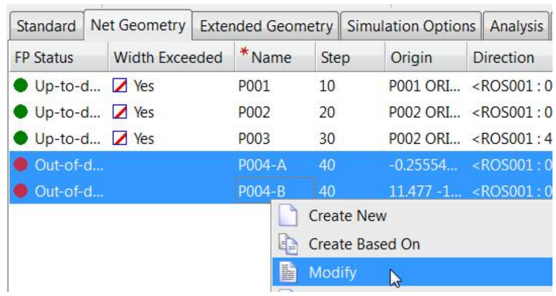


Figure 4–19

2. Press <Ctrl>+ <T> to switch to Table mode.
3. In the *Origin* column for **P004-A**, click  (Link Geometry).
4. Press <Ctrl> + <L> to open the Layer Settings window.
5. Turn on the **P004 GEOM** Layer Category.
6. Click Close.
7. Make sure that the **Existing Point** filter is selected.
8. Select the point indicated in Figure 4–20.

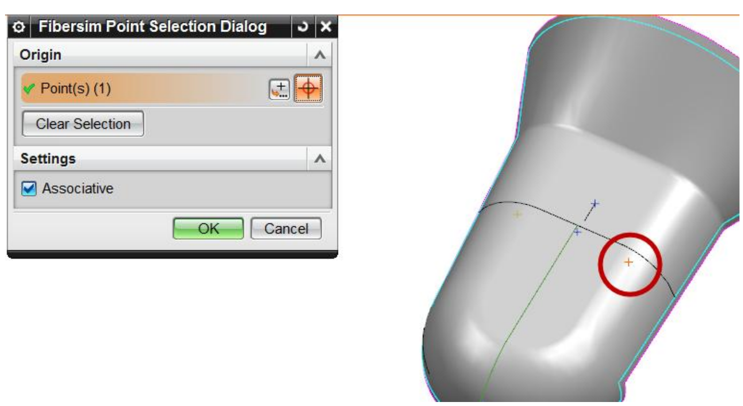


Figure 4–20

9. Click OK.

10. Using the same procedure as step 3 through 9 and link **P004-B** to the point indicated in Figure 4–21.

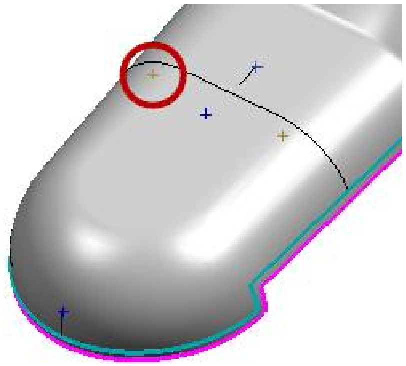
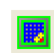
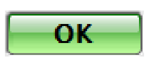


Figure 4–21

11. Click  twice to save the plies.

Task 8 - Compare the Flat Pattern of P002 to the Spliced Result of P004.

1. In the ply list, highlight **P004-A** and **P004-B** and click  (Net Producibility).
2. Click  (Generates the Net Flat Pattern).
3. Click . The Net Flat Patterns of **P004-A** and **P004-B** display as shown in Figure 4–22.

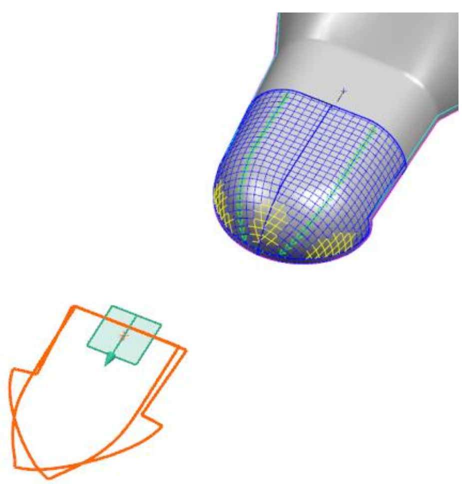


Figure 4–22

4. A comparison between the flat patterns of the **P002** ply and the spliced **P004** ply is shown in Figure 4–23 with the **P004A** and **P004B** flat patterns arranged side-by-side. Note the material difference as well.

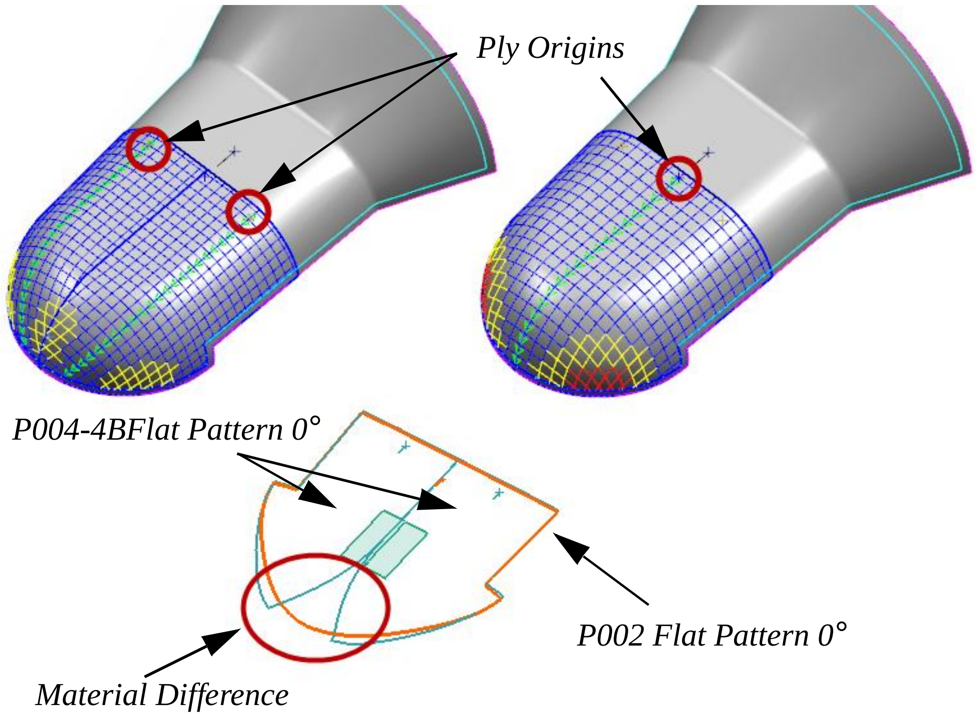


Figure 4–23

Task 9 - Create a new ply and check the producibility.

In this task, you will create **P005** based on **P002** and check the producibility.

1. Turn on **P005 GEOM** category in the Layer Settings window.
2. In the ply list, select **P002**, right-click and select **Create Based On**.
3. In the Ply's Standard form, enter the following parameters:
    - Name: **P005**
    - Step: **50**
    - Specified Orientation: **45**
4. Next to the *Net* > *Boundary* field, click  (Link Geometry)
5. Hold <Shift> and select **P001 Boundary** to deselect.

6. Select the boundary indicated in Figure 4–24.

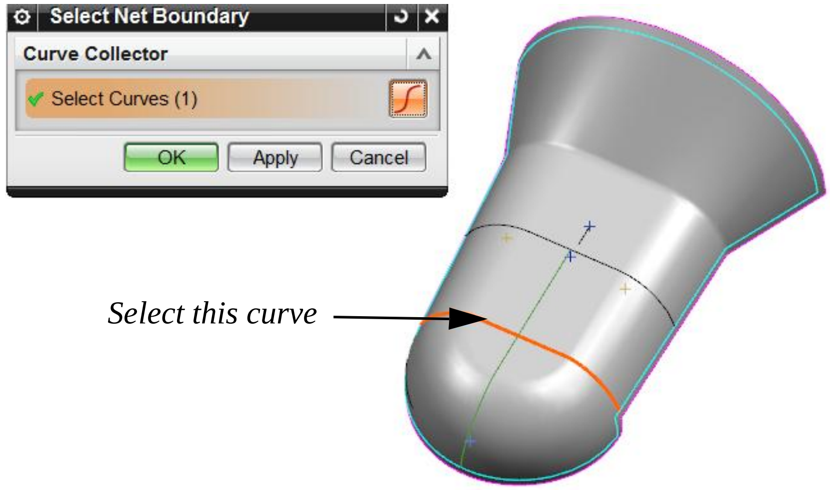


Figure 4–24

7. Click OK.

8. Click (Net Producibility) and click OK in the message box prompting you that the material width has been exceeded.

9. Click OK to save **P005**.

10. Switch to the NX window to display the producibility results as shown in Figure 4–25.

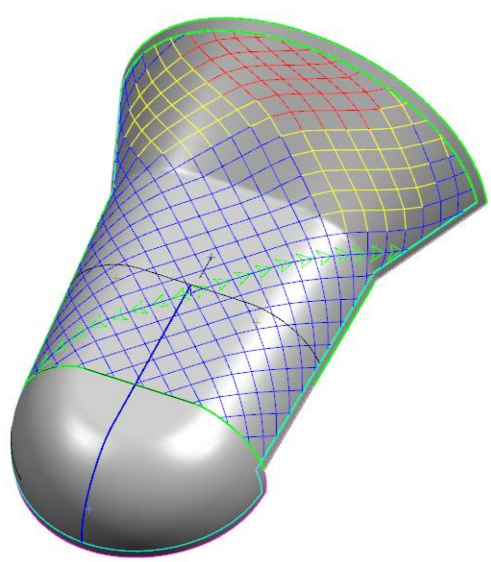


Figure 4–25

To identify material excess and shortage and to characterize deformation regions, observe the red fiber cell behavior in relation to the ply origin. The red fiber cell behavior for Bridging (material shortage) and Wrinkling (material excess) is shown in Figure 4–26.

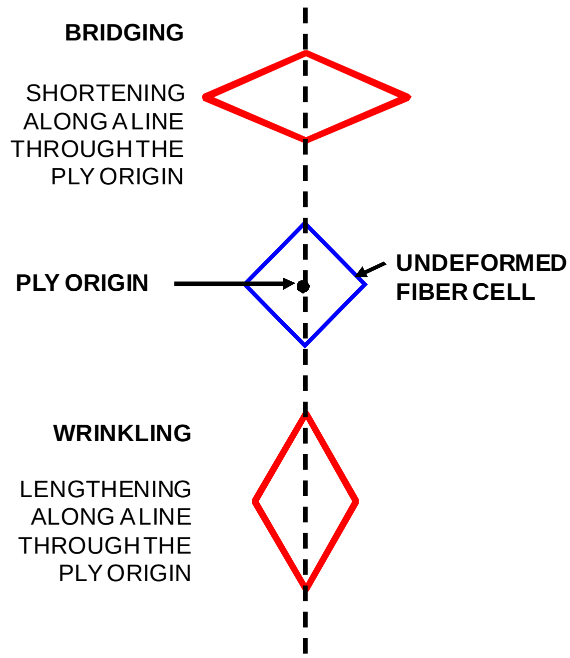


Figure 4–26

Task 10 - Create a Slit Dart.

In this task you will attempt to address the producibility issue by creating a Slit dart and applying it to **P005**.

1. Double-click **P005** to modify.
2. In the **P005** Standard form, select the *Net Geometry* tab.
3. Next to the *Darts* field, click on the arrow portion of the icon  (Link with Link Dialog) and select **Slit Dart** as shown in Figure 4–27.

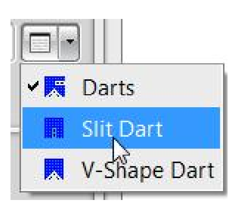


Figure 4–27

4. Fibersim - link Slit Dart via Darts window as shown in Figure 4–28.

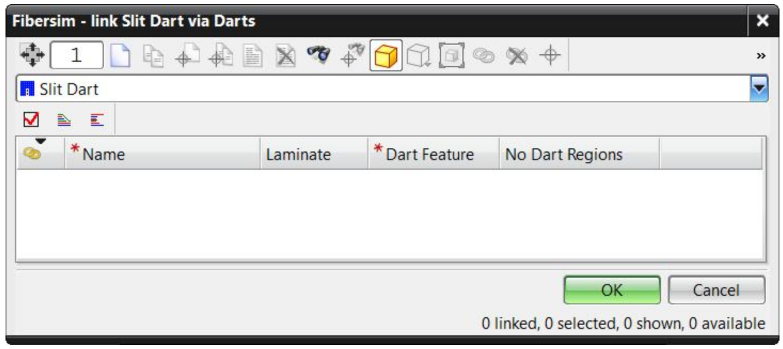


Figure 4–28

5. Click  (Create New).

6. Next to the *Base Curve Points* field, click  (Link Geometry).

7. Turn on the Point on Face option as shown in Figure 4–29.

Figure 4–29

8. Indicate twice on the surface at approximately the two points shown in Figure 4–30.

Always indicate dart points in order from the interior of the ply to the boundary.

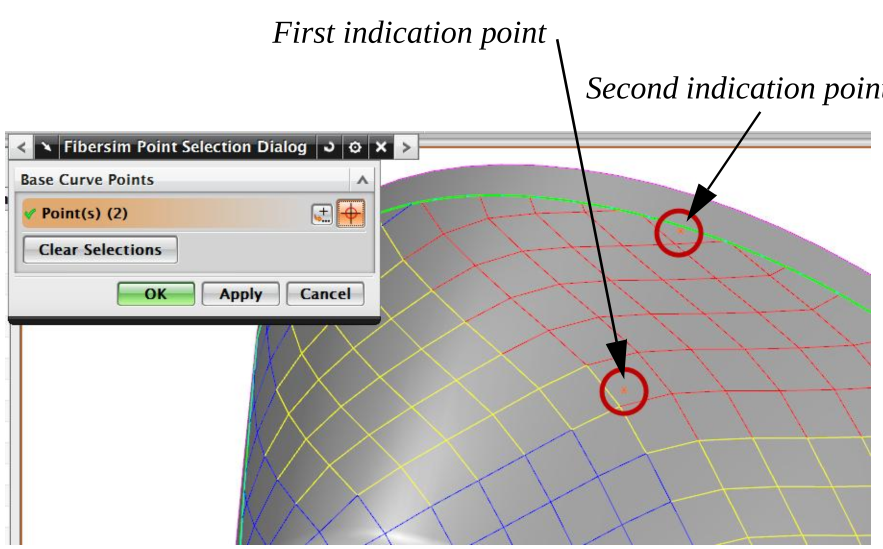
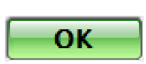


Figure 4–30

9. Click .

10. Click **Generate Dart Feature**. The Slit Dart form updates as shown in Figure 4–31.

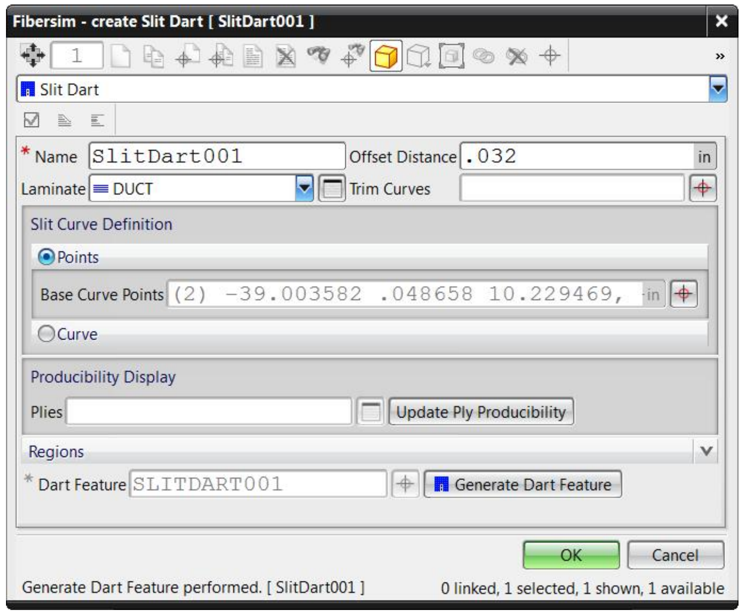


Figure 4–31

11. Click **OK** to save **SlitDart001**.

12. Click **OK** to return to the **P005** form.

13. The **P005** Net Geometry form updates with the dart information as shown in Figure 4–32.

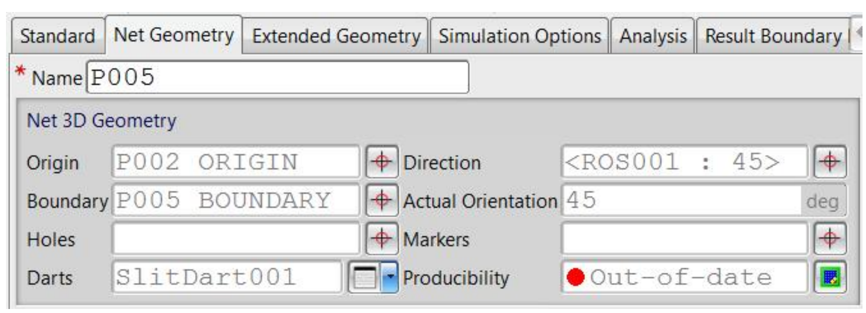


Figure 4–32

Task 11 - Run Net Producibility and Generate the Net Flat Pattern.

1. Click (Net Producibility) and click OK for the warning message.
2. Click (Generates the Net Flat Pattern) and click OK for the warning message.
3. Click OK to save **P005**.
4. The producibility and flat pattern results should be as shown in Figure 4–33. **P005** displays a bridging situation, indicating that there is a material shortage. Note the overlapping flat pattern.

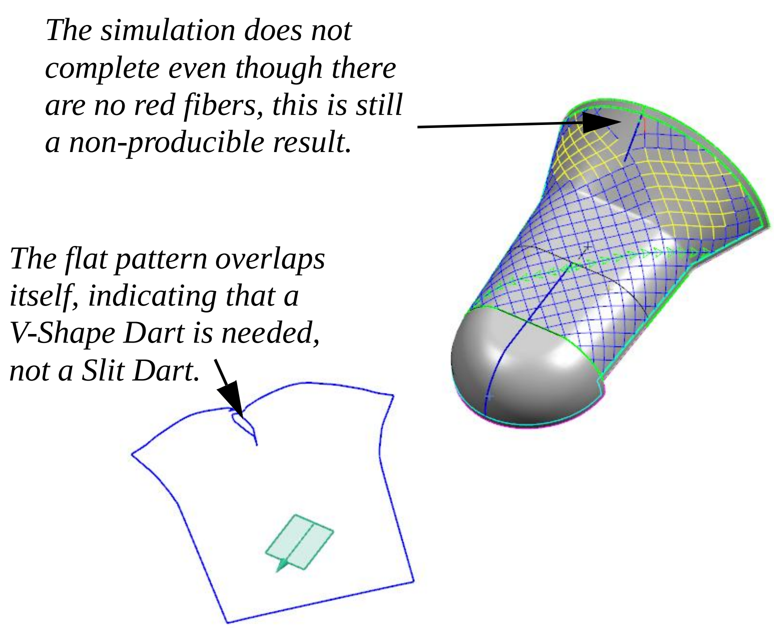


Figure 4–33

Task 12 - Create a V-Shape Dart to resolve bridging.

In this task you will address the bridging producibility issue by removing the **Slit Dart** from **P005** and creating a **V-Shape Dart** instead.

1. Double-click **P005** to modify.
2. In the ply **P005** *Net Geometry* tab, next to the *Darts* field, click (Link with Link Dialog). The Fibersim - link Darts via Darts dialog box opens.

3. Select the **SlitDart001** option to clear the current selection as shown in Figure 4–34.

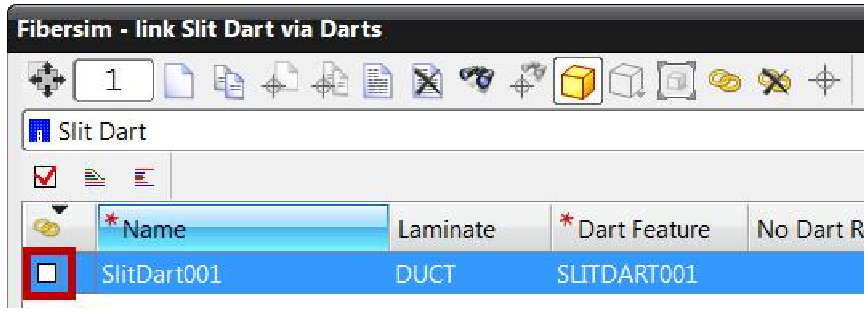


Figure 4–34

4. Click on **Slit Dart** and select **V-Shape Dart** as shown in Figure 4–35.

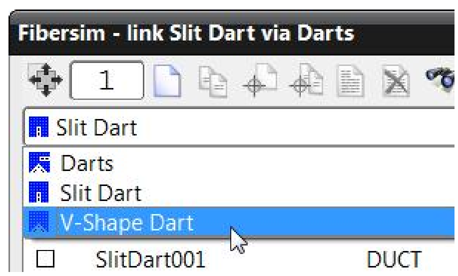


Figure 4–35

5. Click  (Create New).

6. The V-Shape Dart's form displays as shown in Figure 4–36.

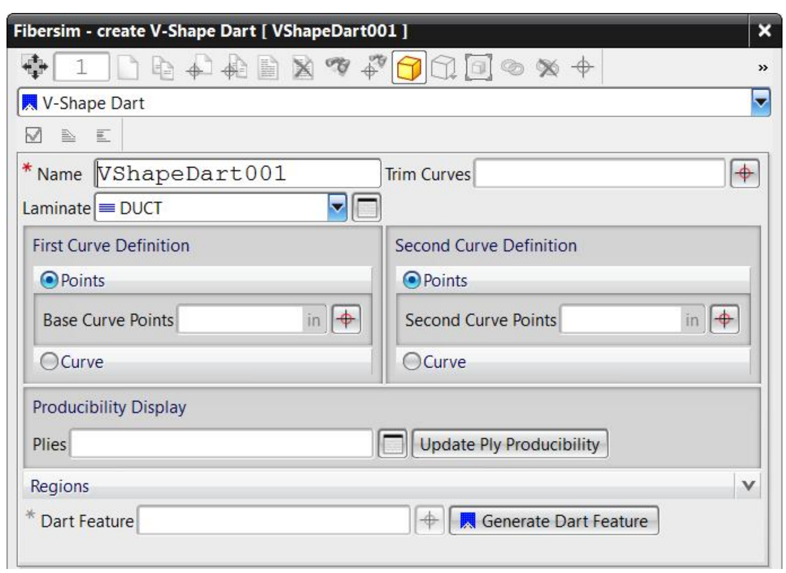


Figure 4–36

7. Next to the *Base Curve Points* field, click (Link Geometry).
8. Indicate twice on the surface at approximately the two points shown in Figure 4–37.

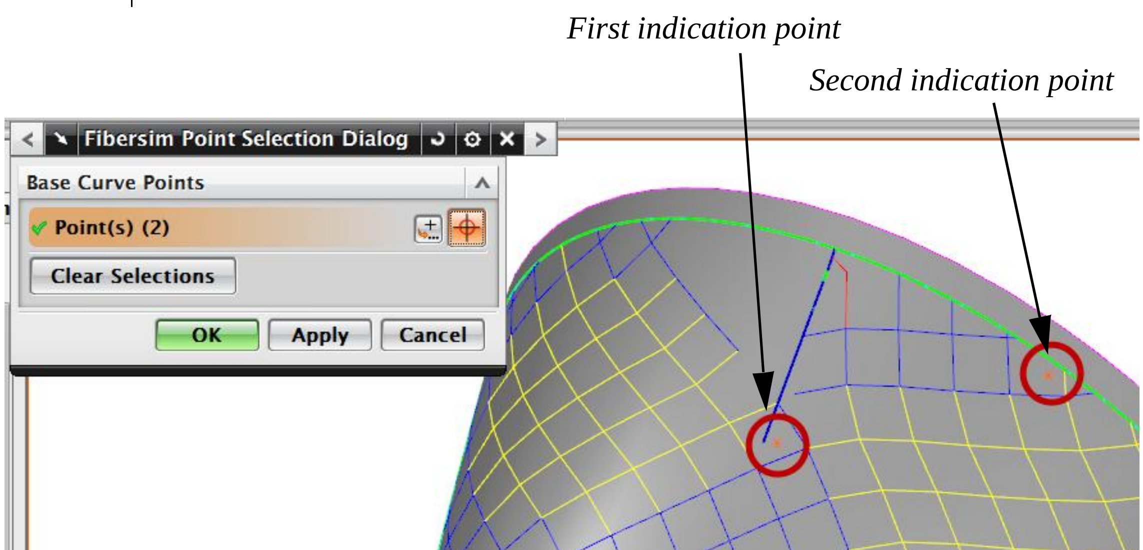


Figure 4–37

9. Click **OK**.
10. Next to the *Second Curve Points* field, click (Link Geometry).
11. Indicate once on the surface at approximately the point shown in Figure 4–38.

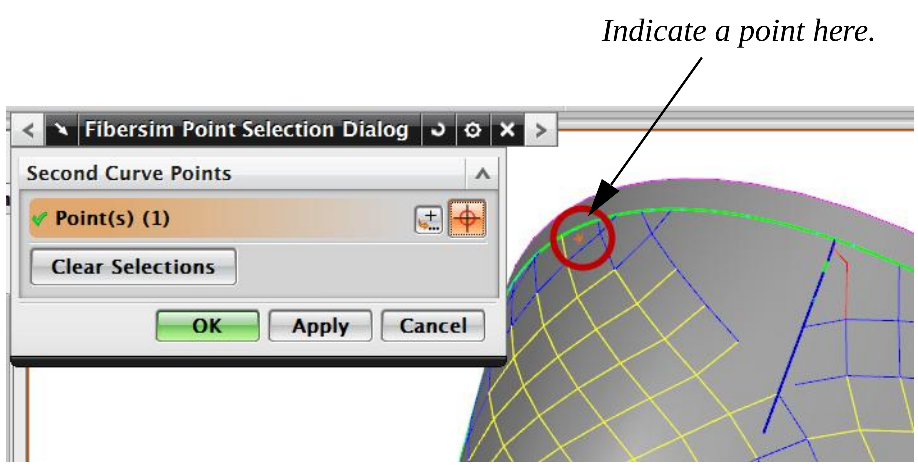


Figure 4–38

12. Click **OK**.

13. Click Generate Dart Feature . The V-Shape Dart form updates as shown in Figure 4–39.

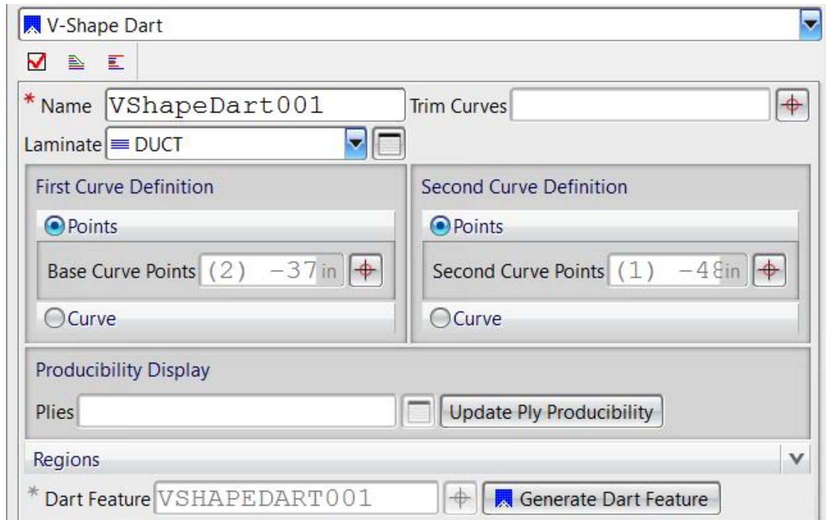


Figure 4–39

14. Click OK to save **VShapeDart001**.

15. Click OK to return to the **P005** form.

Task 13 - Run Net Producibility and Generates the Net Flat Pattern.

1. Click (Net Producibility) and click OK for the warning message.

2. Click (Generates the Net Flat Pattern).

3. Click OK to save **P005**.

4. Switch to the NX window to display the producibility and flat pattern results as shown in Figure 4–40.

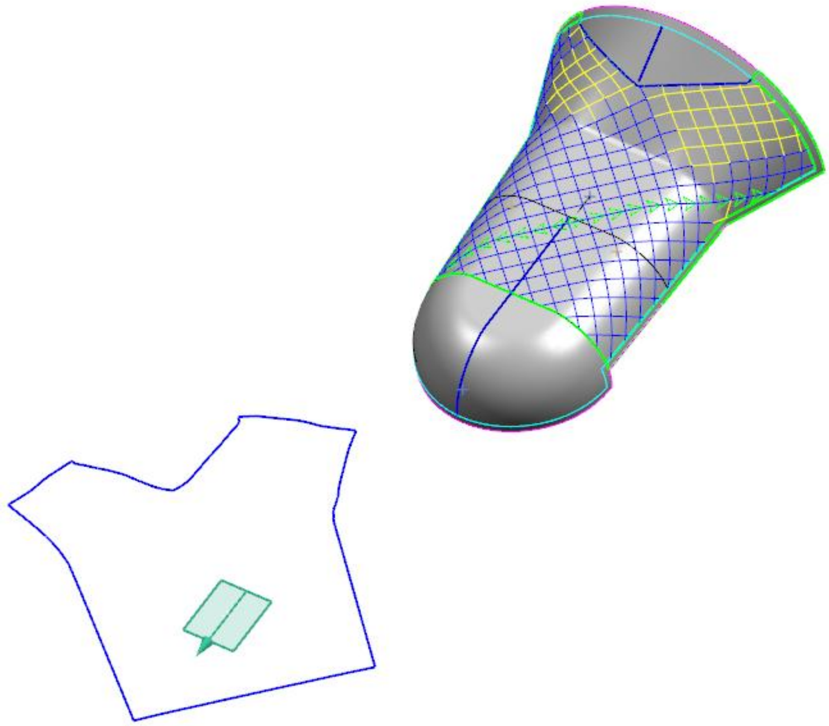


Figure 4–40

Task 14 - (Optional) Create a Patch Ply.

In this task you create a **Constant Curve Offset** to create a Patch ply to cover the gap.

1. In Fibersim, select **Tools > Curve Creation > Curve Offset**.

2. Click  (Create New) and link the following geometry:

   - Curves to Offset: the curves generated from the V-Shaped Dart
   - Surface: **Tool Surface (OML)**
   - Direction Point: Indicate a point outside the Dart curve as shown in Figure 4–41
   - Boundary Curves: **Net Boundary**

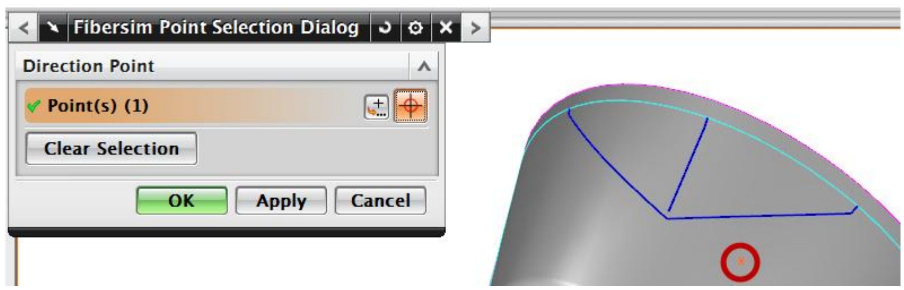
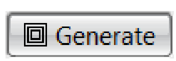


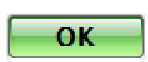
Figure 4–41

3. Enter the following parameters:

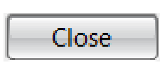
   - Number of Curves: **1**
   - Offset Value: **1 in (25.4 mm)**
   - Corner Type: **Straight**

4. Click 

.

5. Click 

twice.

6. Click 

.

7. Create a ply using the following parameters:

   - Name: **P006**
   - Step: **60**
   - Specified Orientation: **0**
   - Net > Boundary: Curves generated from previous steps
   - Origin: indicate a point within gap as shown in Figure 4–42

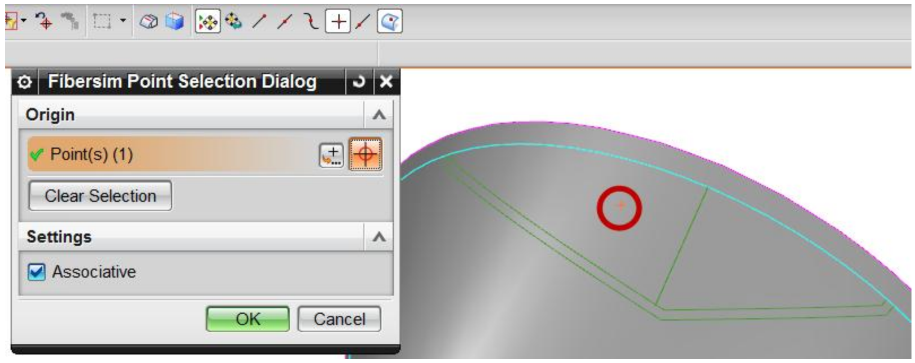


Figure 4–42

8. Run Net Producibility and create a Net Flat Pattern to display the results as shown in Figure 4–43. Now a full-coverage ply exists with a 1 inch (25.4 mm) overlap.

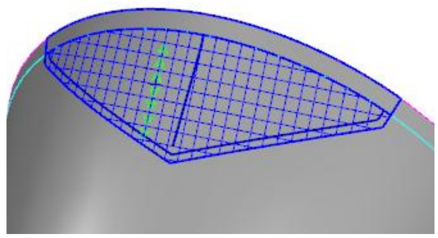


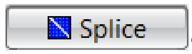
Figure 4–43

9. Click OK to save P006.

Task 15 - (Optional) Create a P007 Ply, Splice Ply and a Slit Dart.

In this task, you will create a new ply with a specified orientation of 45. Then, splice and create a slit dart to resolve producibility issues.

1. Show **P007 GEOM** layer category in the Layer Settings window.
2. In the ply list, select **P003**, right-click and select **Create Based On.**
3. In the Ply's Standard form, enter the following parameters:
    - Name: **P007**
    - Step: **70**
4. Click OK to save P007.
5. Highlight **P007** and click (Splice Ply).

6. Link the following geometry:
   - Splice Line Curve: **Splice Line 1 P007**, **Splice Line 2 P007**
7. Click 

.
8. Click 

twice.
9. Modify the origins of the spliced plies to use the following origins:
   - P007-A: **P007A Origin**
   - P007-B: **P007B Origin**
   - P007-C: **P007C Origin**
10. Run Net Producibility on the spliced plies, **P007-A**, **P007-B**, and **P007-C**. The results display as shown in Figure 4–44.

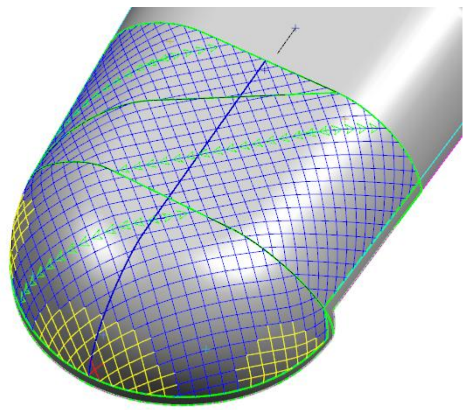


Figure 4–44

11. Modify **P007-C**.

Make sure that the Point on Face filter is turned on.

12. Create a slit dart with two points as shown in Figure 4–45.

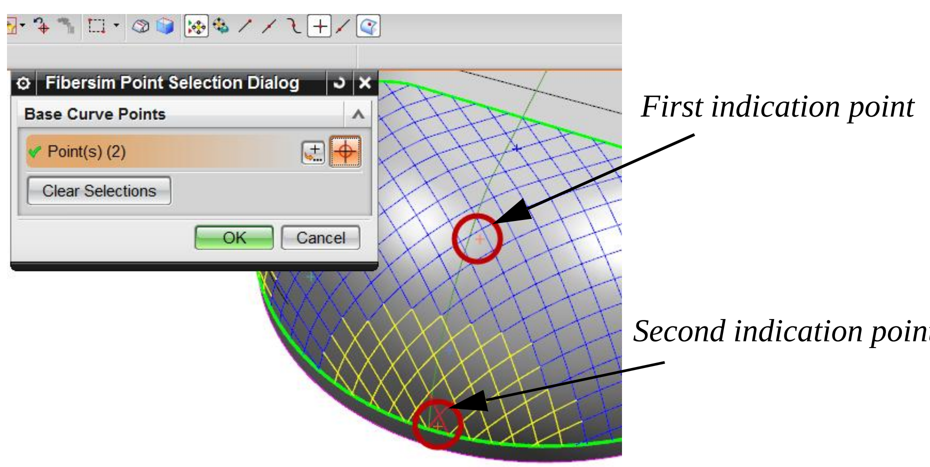


Figure 4–45

13. Run Net Producibility on **P007-C** to display the results as shown in Figure 4–46.

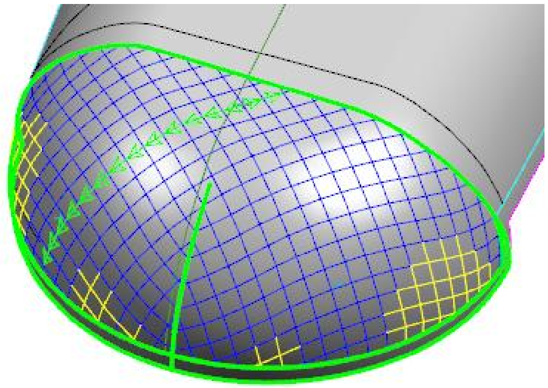


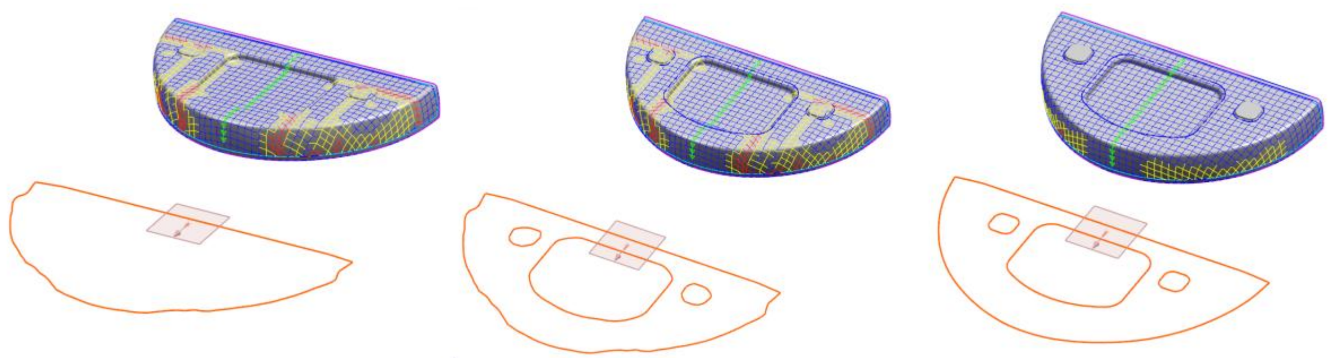
Figure 4–46

14. Close Fibersim and save the model.

Exercise 4b How Simulation Options Affect Producibility (Simulation Skin)

User Guide Reference:
2.8 Plies
2.5 Flat Pattern/ Producibility
2.5.2 Using Simulation Surfaces

In this exercise, you will see how simulation options, such as holes and simulation skin, affect producibility. The Net Flat Patterns you will create with each simulation option are displayed in Figure 4–47.



Before holes *With holes* *Simulation Skin*

Figure 4–47

Goal

After you complete this exercise, you will be able to:

✓ **Create a ply using a Fiber Spacing Factor**
✓ **Add holes to the ply and observe the impact on the Flat Pattern**
✓ **Set a Simulation Skin and observe the impact on the Flat Pattern**

Estimated Time

10 min

Task 1 - Open a part.

1. Open **PRODUCIBILITY_B.prt**. The model displays as shown in Figure 4–48.

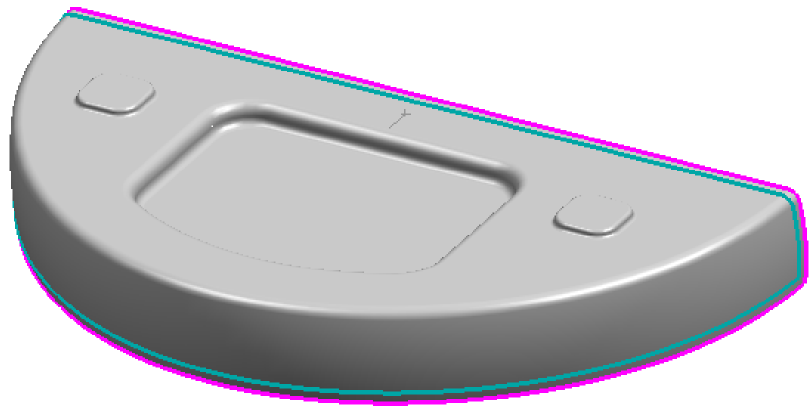


Figure 4–48

Task 2 - Create a Full-body Ply using the Fiber Spacing Factor.

In this task you will review the **Laminate** and **Rosette** that have already been created and then create a full-body ply using a *Fiber Spacing Factor* of [0.5].

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** under **Basics**.
3. Select **Sim Skin** from the list. In the NX window, Fibersim displays the associated tool surface and layup direction.
4. In the *Application Tree*, select **Rosette** under **Basics**.
5. Select **ROS001** from the list. In the NX window, Fibersim displays the associated geometry.
6. In the *Application Tree*, select **Ply** under **Basics**
7. Click  (Create New).
8. In the Ply's Standard form, enter the following parameters to create a full-body ply:
    - Name: **P001**
    - Step: **10**
    - Specified Orientation: **0**

For small features, it is necessary to reduce the Fiber Spacing Factor to conform the simulated fibers to the surface.

9. Select the *Simulation Options* tab.
10. For the *Fiber Spacing Factor*, enter [0.5].
11. Click (Net Producibility) and click OK in the message box prompting you that the material width has been exceeded.
12. Click (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P001**.
13. Click OK to save the ply **P001**.
14. Switch to the NX window and review the producibility and flat pattern results as shown in Figure 4–49.

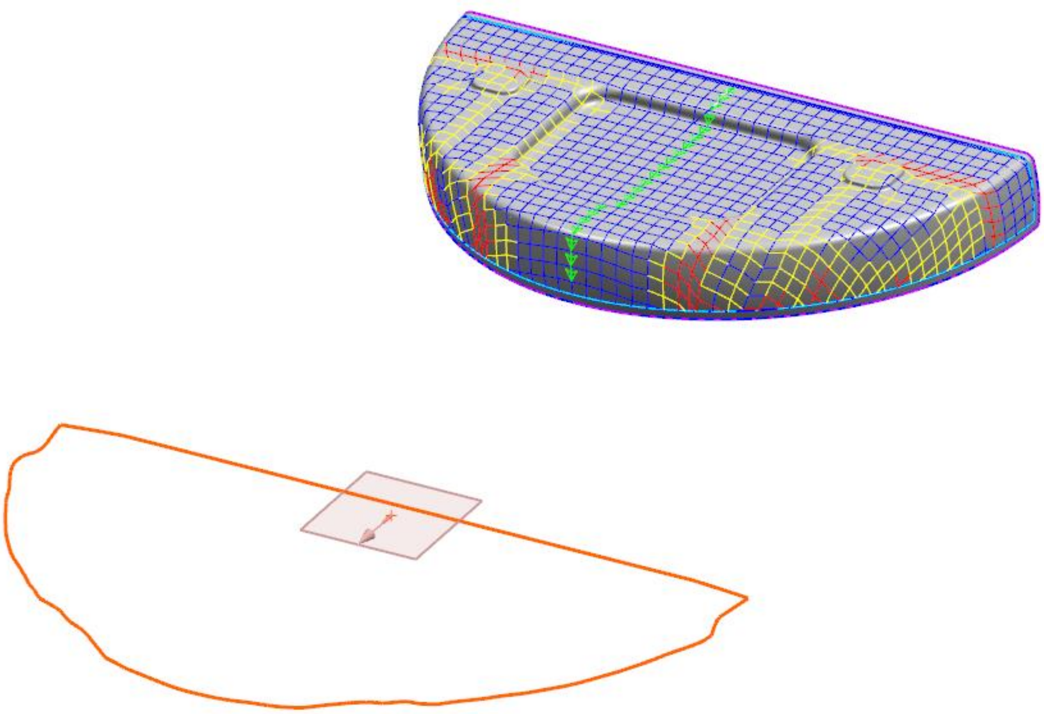


Figure 4–49

Task 3 - Add holes to ply P001.

In this task, since the ply material will not be required to cover the enclosed blue boundaries as shown in Figure 4–50, you will modify ply **P001** and add three holes.

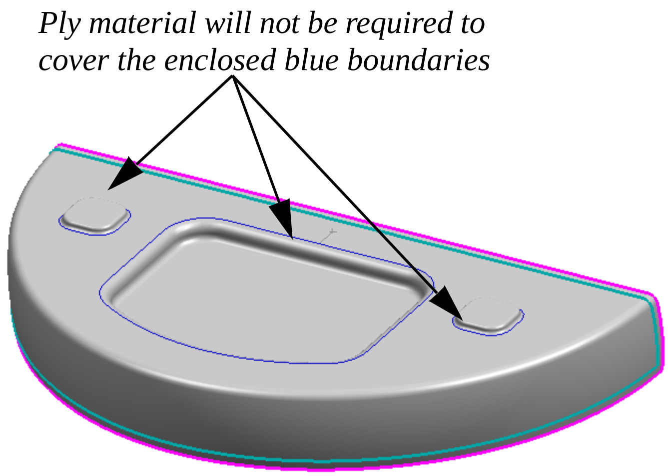
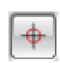


Figure 4–50

Press <Ctrl> + <L> to launch Layer Settings window.

1. Show **HOLES** layer category in the Layer Settings window.
2. Press <F9> to start Fibersim.
3. In the Ply list, double-click **P001** to modify it.
4. Select the *Net Geometry* tab.
5. Next to the *Holes* field, click  (Link Geometry).
6. Select the three curves: **Hole 1**, **Hole 2**, and **Hole 3** as shown in Figure 4–51.

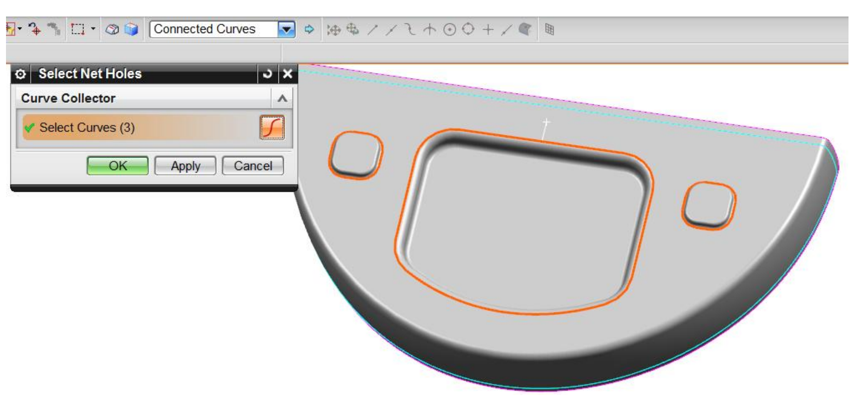


Figure 4–51

7. Click **OK**.

8. Click (Net Producibility) and click **OK** in the message box prompting you that the material width has been exceeded.

9. Click (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P001**. Note that the outside part of the flat pattern has not changed.

10. Click **OK** to save **P001**.

11. Switch to the NX window and review the producibility and flat pattern results as shown in Figure 4–52.

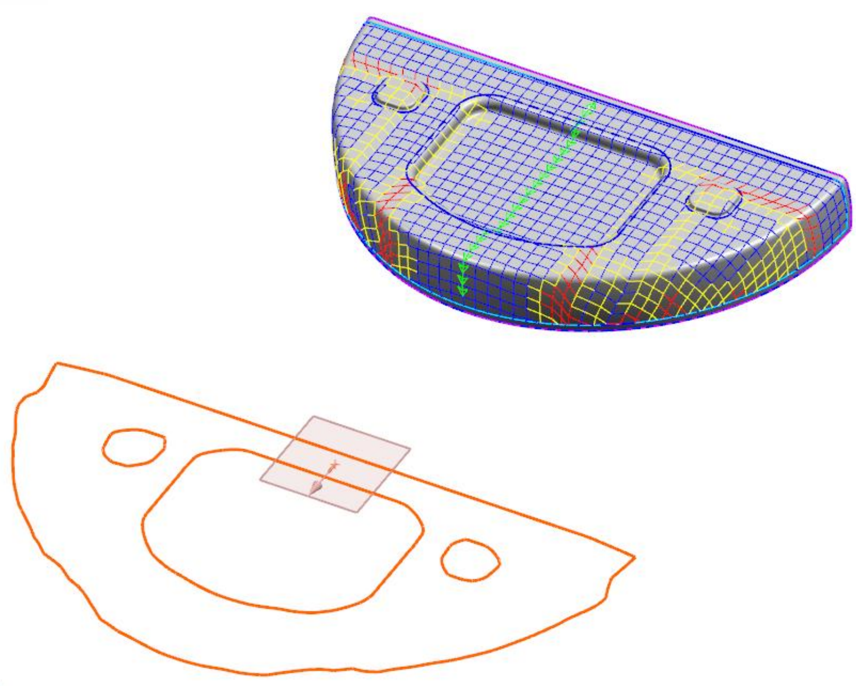


Figure 4–52

Task 4 - Set a Simulation Skin.

In this task, since a simulation surface must accurately reflect the topology of the tool surface to be covered by material, you will modify ply **P001** and set a **Simulation Skin** that does not include the Boss features as shown in Figure 4–53.

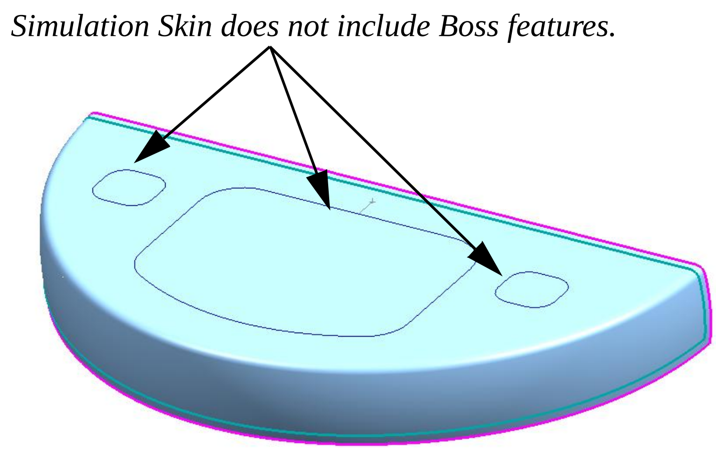


Figure 4–53

Press <Ctrl> + <L> to launch Layer Settings window.

1. Show **SIMULATION_SKIN** layer category and hide **LAMINATE_SURFACE** in Layer Setting. Note that the **Sim_Skin** surface does not include the bosses.
2. Press <F9> to start Fibersim.
3. In the Ply list, double-click **P001** to modify it.
4. Select the *Simulation Options* tab.
5. Next to the *Simulation Surface* field, click (Link Geometry).
6. Select **Sim_Skin** and click OK.
7. Click OK to save **P001**.
8. Show **LAMINATE_SURFACE** layer category and hide **SIMULATION_SKIN** in Layer Setting.
9. Press <F9> to start Fibersim.
10. Select **P001** from the list of plies.
11. Click (Net Producibility) and click OK in the message box prompting you that the material width has been exceeded.
12. Click (Generates the Net Flat Pattern).

13. Switch to the NX window and review the producibility and flat pattern results as shown in Figure 4–54.

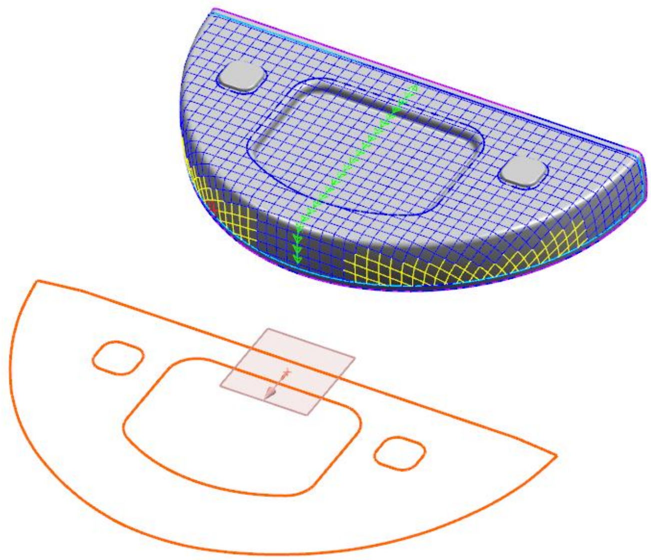


Figure 4–54

14. The producibility simulation now follows the **Simulation Surface** and ignores the bosses in the laminate surface as shown in Figure 4–55. The new flat pattern is much cleaner because it is not affected by the bosses in the laminate surface.

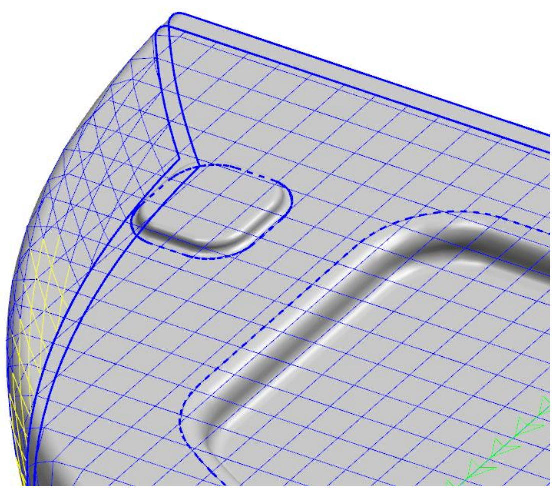
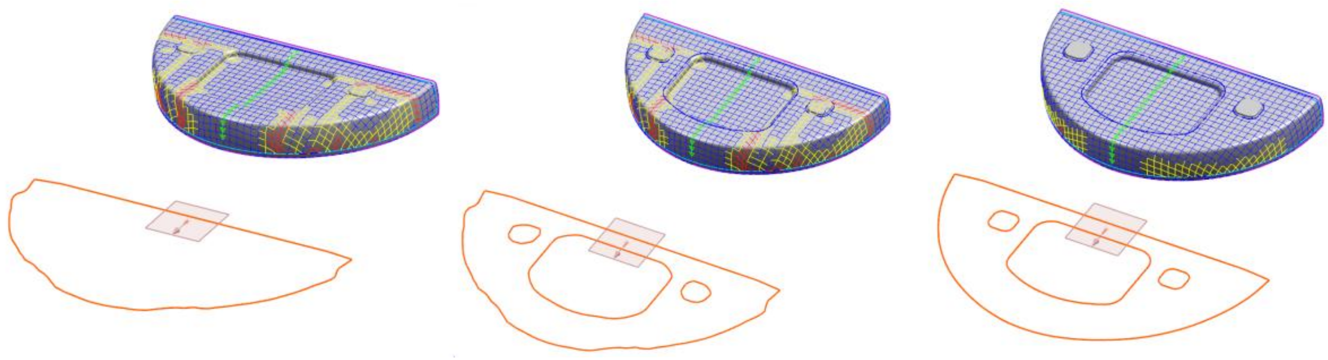


Figure 4–55

15. The Producibility and Flat Pattern Comparison of Tasks 2, 3, and 4 is shown in Figure 4–56.



Before holes *With holes* *Simulation Skin*

Figure 4–56

Task 5 - (Optional) Create a Splice through a Hole.

Fibersim enables splicing through holes.

1. In the ply list, double-click **P001** to modify it.
2. Generate a **Material Width Line** using a *Material Width Offset* of [24] inches (609.6 mm).
3. Splice **P001** into two plies, **P001-A** and **P001-B**, using the following parameters:
   - Splice Curves: curve created with Material Width Line
   - Overlap: **Yes**
   - Overlap Distance: **1 in (25.4 mm)**

4. Run Net Producibility and create a Net Flat Pattern to display the results as shown in Figure 4–57.

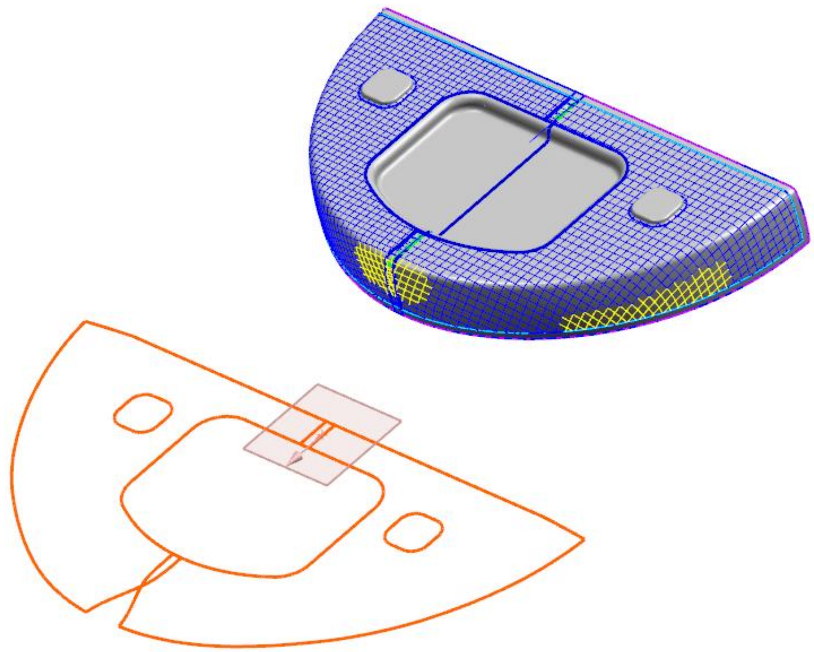


Figure 4–57

5. Close Fibersim and save the model.

Exercise 4c How Simulation Options Affect Producibility (Two-Stage)

User Guide Reference:
2.5 Flat Pattern/
Producibility Simulations
2.8 Plies
2.5.3 Performing
Multi-Stage Simulations
2.12. Darting

In this exercise, you will see the ply development using a two stage simulation. The completed standard simulation and two-stage simulation net flat patterns display as shown in Figure 4–58.

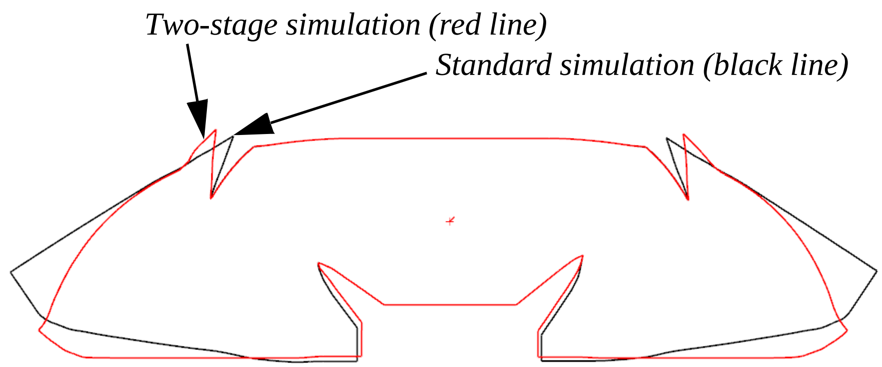


Figure 4–58

Goal

After you complete this exercise, you will be able to:

- ✓ **Compare the standard simulation to a Biased Geodesic simulation**
- ✓ **Add a First Stage Region**
- ✓ **Create V-Shape Darts to alleviate the inside corners**
- ✓ **Create Slit Darts to alleviate the outside corners**

Estimated Time

20 min

Task 1 - Open a part.

1. Open **PRODUCIBILITY_C.prt**. The model displays as shown in Figure 4–59.

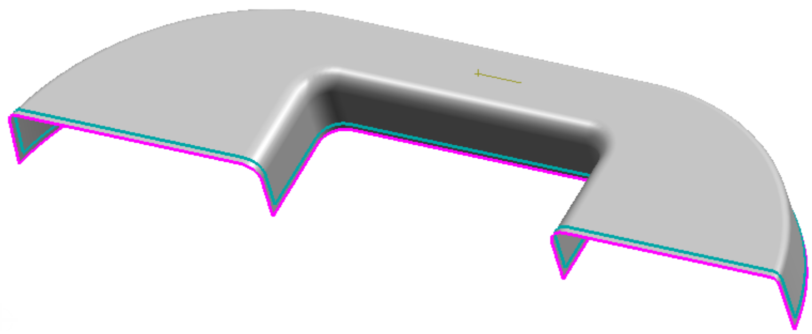


Figure 4–59

Task 2 - Create a ply and run the standard simulation.

In this task you will review the **Laminate** and **Rosette** that have already been created and create a full-body ply.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** under **Basics**.
3. Select **Center Support** from the list. In the NX window, Fibersim displays the associated tool surface and layup direction.
4. In the *Application Tree*, select **Rosette** under **Basics**.
5. Select **ROS001** from the list. In the NX window, Fibersim displays the associated geometry.
6. In the *Application Tree*, select **Ply** under **Basics.**
7. Click  (Create New).
8. In the Ply's Standard form, enter the following parameters to create a full-body ply:
    - Name: **P001**
    - Step: **10**
    - Specified Orientation: **45**
9. Select the *Simulation Options* tab.
10. For the *Fiber Spacing Factor*, enter [0.5].

11. Click  (Net Producibility). Click  to the message that appears.
12. Switch to the NX window to display the producibility results as shown in Figure 4–60.

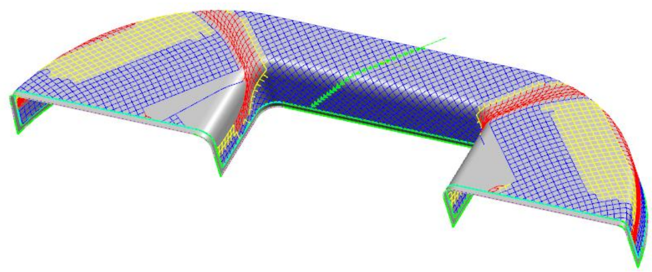
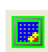


Figure 4–60

Task 3 - Run a Geodesic simulation and compare it to the standard simulation.

See the User Guide in section ***2.8 Plies*** *for more information on Propagation Method and Propagation Direction.*

1. On the *Simulation Options* tab, set the following parameters:
    - Propagation: **To Curve**
    - Constraint curve: **Geodesic**
    - Fiber Angle From Curve: **45**
2. Click  (Net Producibility) and close the warning message.
3. Click  to save **P001**.
4. Switch to the NX window to display the producibility results as shown in Figure 4–61.

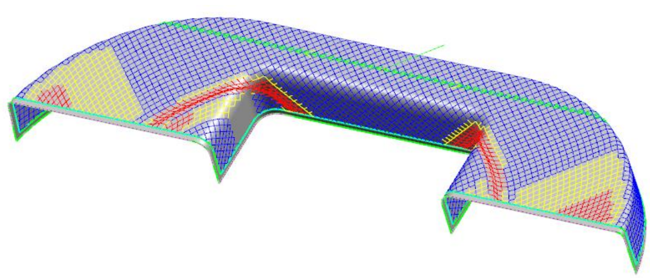


Figure 4–61

Task 4 - Add a First Stage Region.

In this task you will add a First Stage Region. The First Stage Region is used to define an area of simultaneous material contact (e.g., if material is placed on a flat surface). The simulation solves completely within this region before attempting to conform to the other areas and solving to the defined ply net boundary. The First Stage Boundary in this task is shown in Figure 4–62.

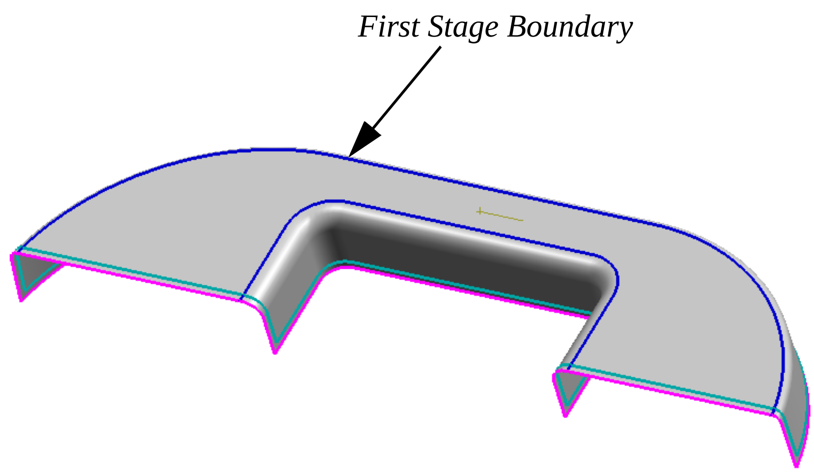
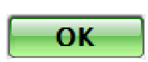
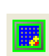


Figure 4–62

1. Show **FIRST_STAGE_REGION** layer category in the Layer Settings window.
2. Press <F9> to start Fibersim.
3. In the Ply list, double-click **P001** to modify it.
4. On the *Simulation Options* tab, next to the *First Stage Region* field, click  (Link Geometry).
5. Select the blue boundary noted as the First Stage Boundary shown in Figure 4–62.
6. Click 
.
7. Click  (Net Producibility) and close the warning message.

8. Switch to the NX window to review the producibility results as shown in Figure 4–63.

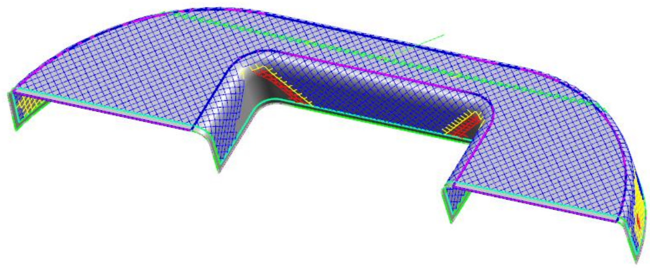


Figure 4–63

Task 5 - Create a V-Shape Dart to alleviate the left inside corner.

1. Back in the *modify Plies* window, select the *Net Geometry* tab.
2. Next to the *Darts* field, click on the arrow portion of the icon (Link with Link Dialog) and select **V-Shape Dart**.
3. Click (Create New).
4. Next to the *Base Curve Points* field, click (Link Geometry).
5. Make sure that the **Point On Face** filter is enabled.
6. Zoom in on the left inside corner and indicate on the surface at approximately the two points shown in Figure 4–64. These will form the first leg of the V that will cut out the red fibers.

The first indicated point is the apex of the V-shape.

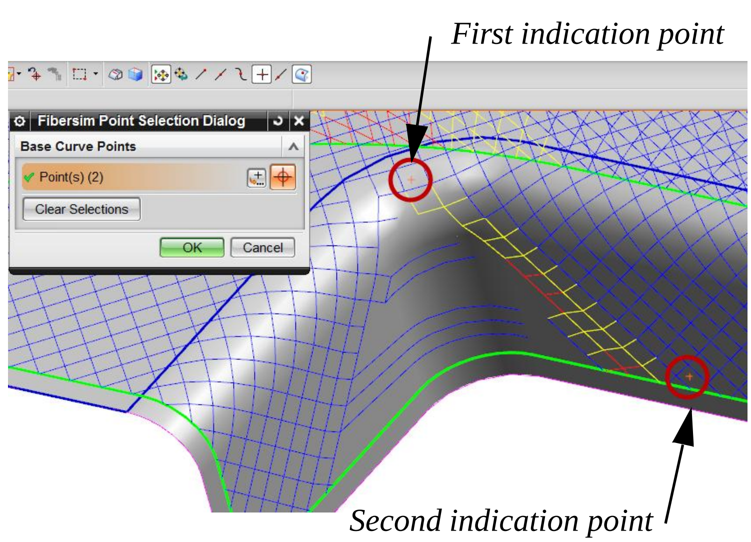


Figure 4–64

7. Click OK.

8. Next to the *Second Curve Points* field, click (Link Geometry).

9. Indicate on the surface at approximately the point shown in Figure 4–65. This will form the second leg of the V.

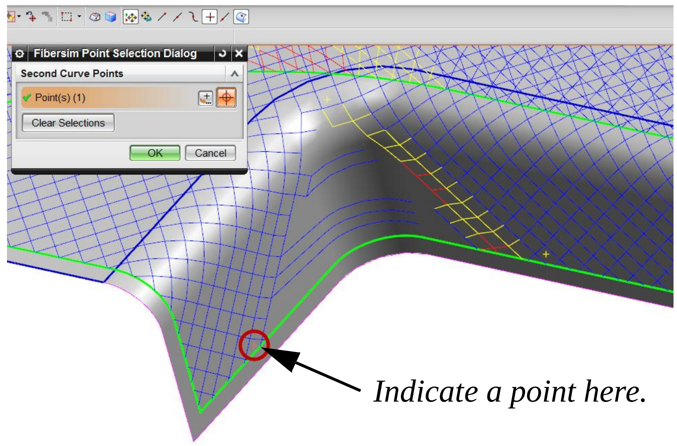


Figure 4–65

10. Click OK.

11. Click Generate Dart Feature.

12. Click OK to save **VShapeDart001**.

13. Switch to the NX window to display the completed V-Shape Dart as shown in Figure 4–66.

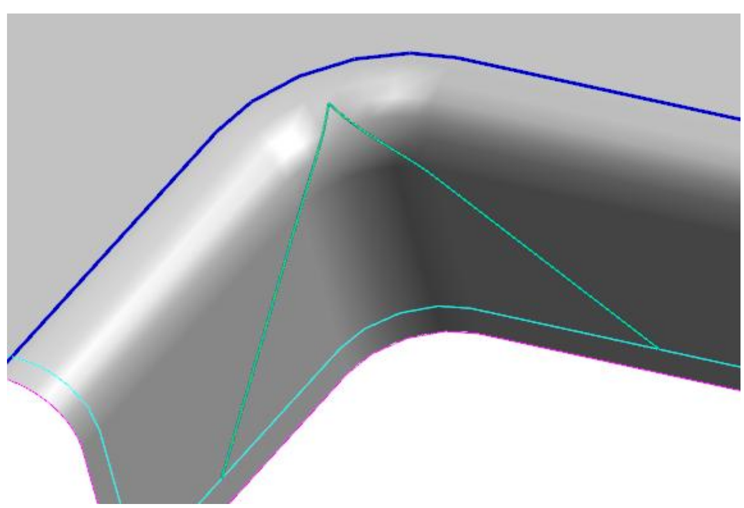


Figure 4–66

Task 6 - Create a V-Shape Dart to alleviate the right inside corner.

1. Use the procedure from Task 5 to create a V-Shape Dart on the right inside corner. The two Base Curve Points are shown in Figure 4–67 and the Second Curve Point is shown in Figure 4–68.

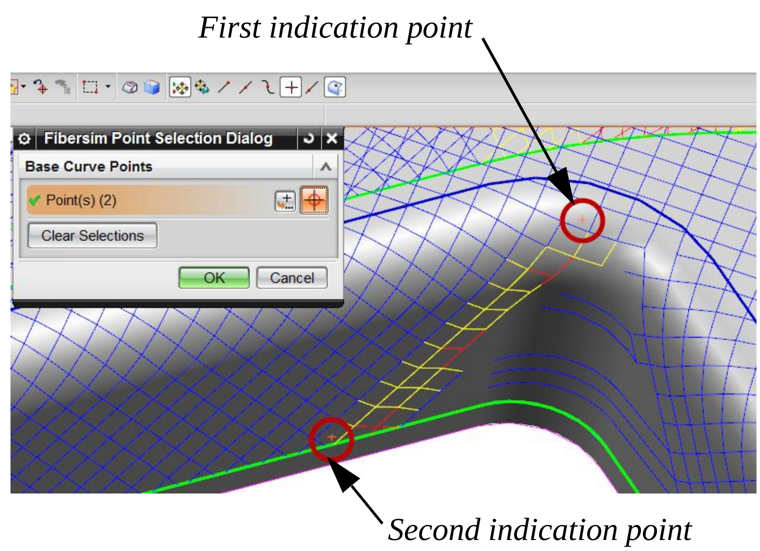


Figure 4–67

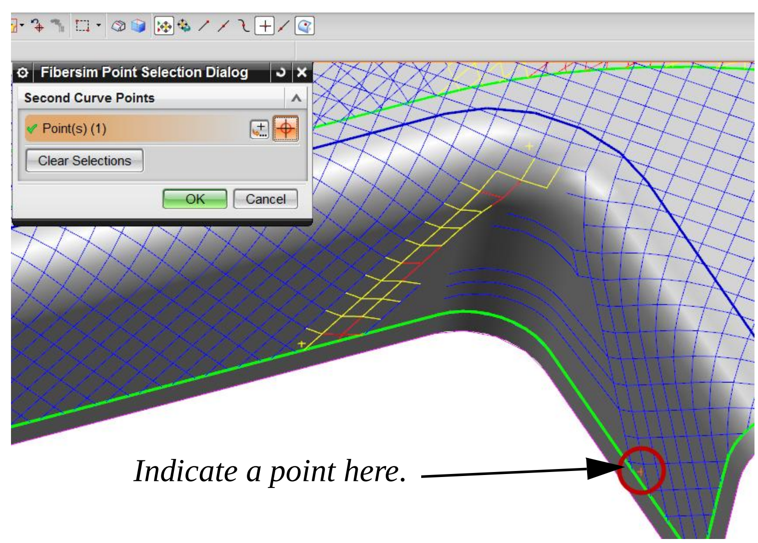


Figure 4–68

2. Click Generate Dart Feature.

3. Click OK to save **VShapeDart002**.

4. Ensure that both V-Shape darts are linked to the ply as shown in Figure 4–69.

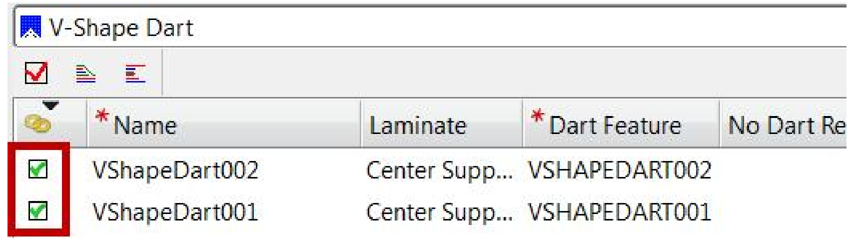
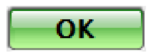


Figure 4–69

5. Click  .

6. Click  (Net Producibility) and close the warning message.

7. Switch to the NX window to display the producibility results with the V-Shape darts as shown in Figure 4–70.

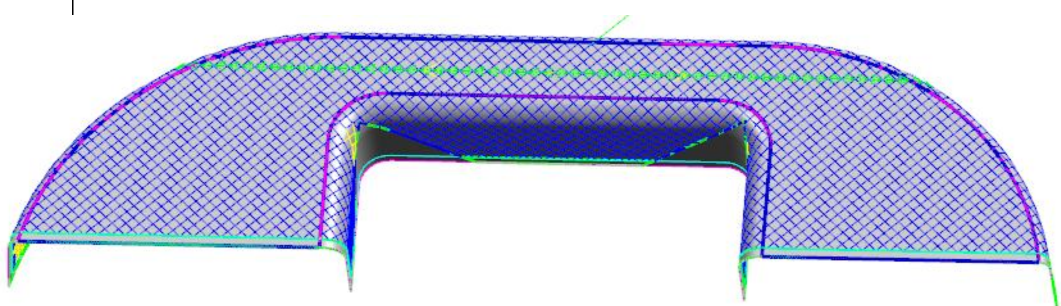


Figure 4–70

Task 7 - Create Slit Darts to alleviate the outside corners.

1. Back in Fibersim, next to the *Darts* field, click  (Link with Link Dialog).

2. In the **Darts** menu, select **Slit Dart** as shown in Figure 4–71.

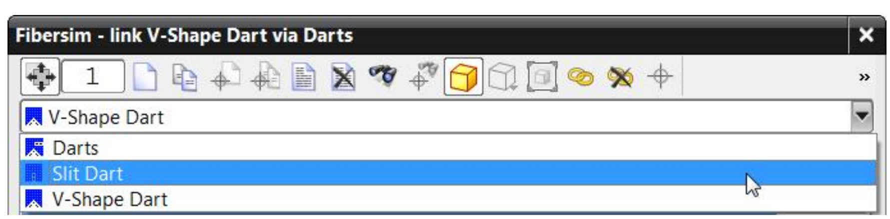


Figure 4–71

3. Click (Create New).
4. Select the **Curve** option as shown in Figure 4–72.

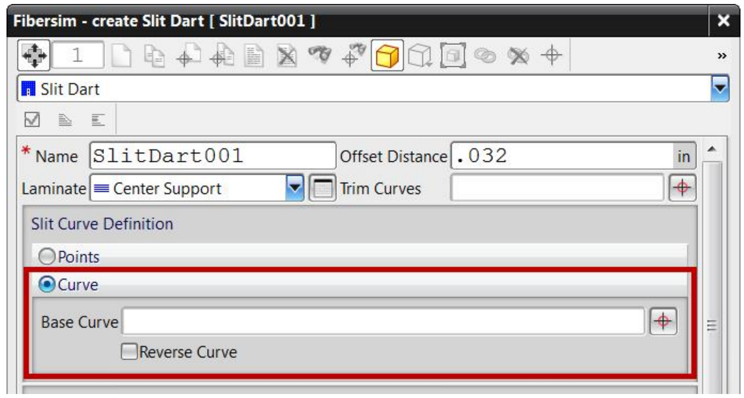


Figure 4–72

5. Next to the *Base Curve* field, click (Link Geometry)
6. In NX, show **SLIT_DARTS** layer category in the Layer Settings window.
7. Select the spline on the right when looking at the model from the back (**SLIT DART RIGHT**) as shown in Figure 4–73.

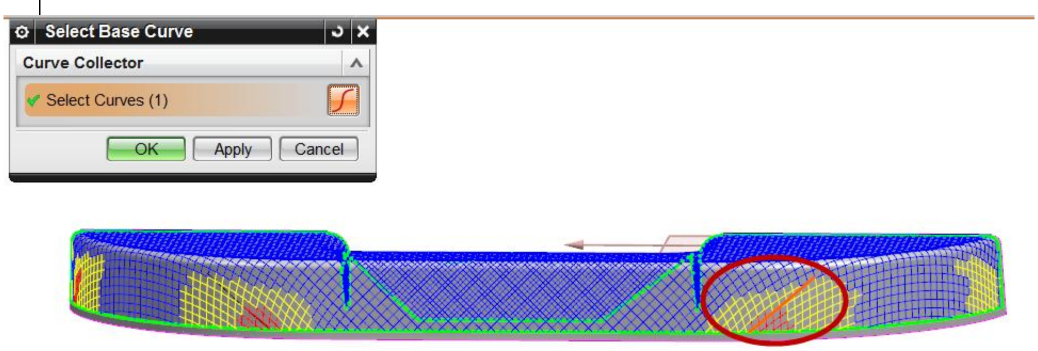


Figure 4–73

8. Click OK.
9. Click Generate Dart Feature.
10. Click OK to save **SlitDart001**.

The slit dart is a U-shaped curve that will be added to the ply boundary.

11. Switch to the NX window to display the Slit Dart as shown in Figure 4–74.

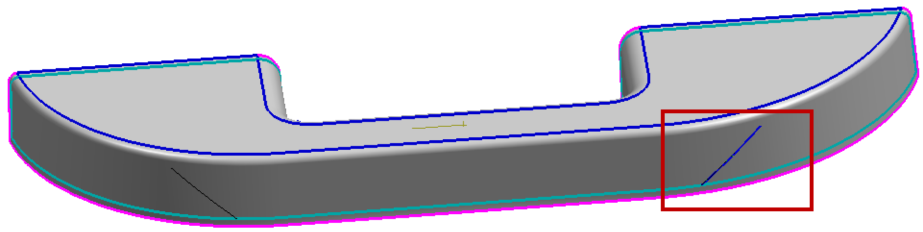


Figure 4–74

12. Repeat Steps 1 to 11 to create a Slit Dart on the left (when looking at the model from the back). The spline is named as **SLIT DART LEFT**.

13. Ensure that both Slit Darts are linked to the ply as shown in Figure 4–75.

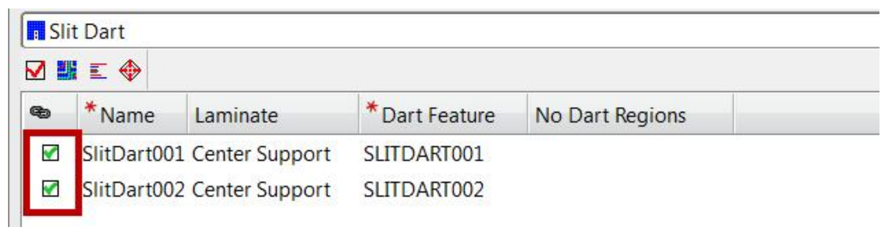
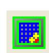


Figure 4–75

14. Click  to close the Fibersim - link Slit Dart via Darts dialog box.

15. Click  (Net Producibility).

16. Click  (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P001**.

17. Click  to save **P001**.

18. Switch to the NX window to display the producibility and net flat pattern results with the darts as shown in Figure 4–76.

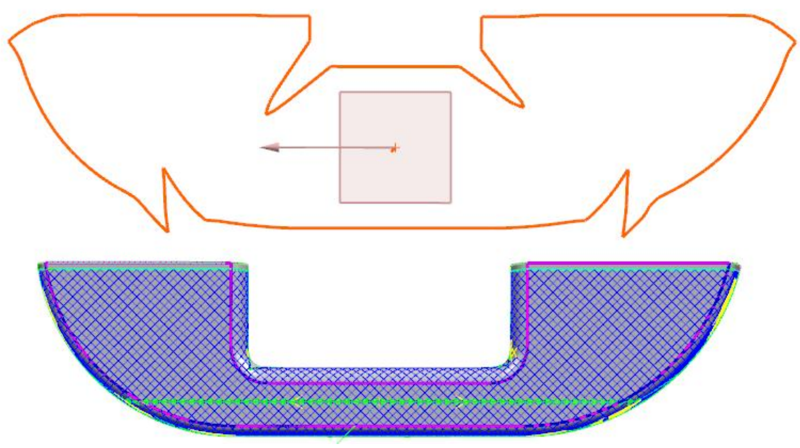
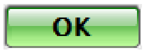


Figure 4–76

Task 8 - Compare the flat pattern to one without a two-stage simulation.

1. In the ply list, select **P001**, right-click and select **Create Based On** to create the ply **P002**.
2. In the *Step* field, enter [20].
3. Select the *Simulation Options* tab.
4. Next to the *First Stage Region* field, click  (Link Geometry).
5. Press <Shift> and select the First Stage Region spline to clear the selection and click 

.
6. Click  (Net Producibility).
7. Click  (Generates the Net Flat Pattern) to create the Net Flat Pattern of **P002**.

8. Switch to the NX window to display the producibility and net flat pattern results as shown in Figure 4–77.

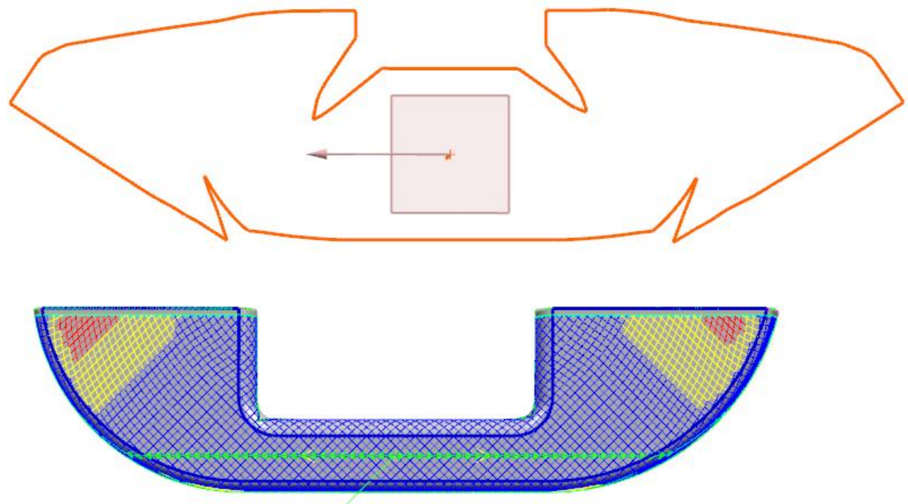


Figure 4–77

9. Click  to save **P002**.

10. The two-stage simulation produces a cleaner flat pattern (green) than the standard simulation (orange) as shown in Figure 4–78.

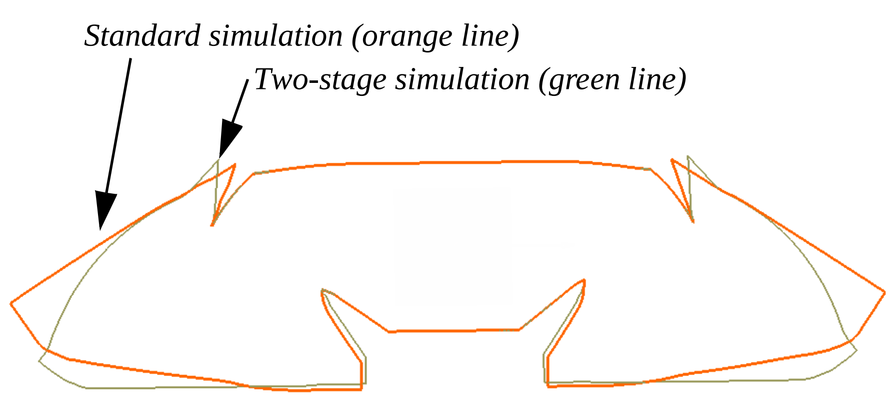


Figure 4–78

Exercise 4d Resolving Producibility Issues

User Guide Reference:
2.5 Flat Pattern/ Producibility Simulations
2.5.3 Performing Multi-Stage Simulations
2.12. Darting

In this exercise, with minimal instruction, you will resolve producibility issues. The producibility and flat pattern results for the acceptable solution are shown in Figure 4–79.

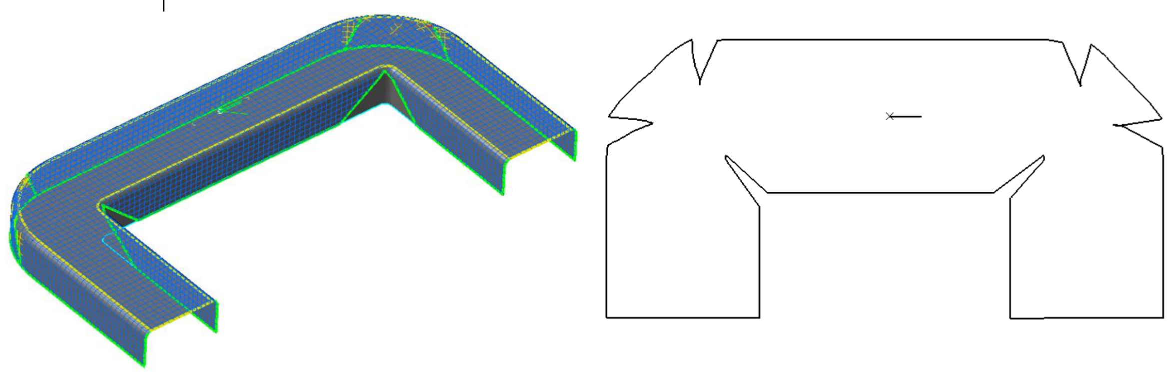


Figure 4–79

Goal

After you complete this exercise, you will be able to:

- ✓ **Set the Fiber Spacing Factor simulation option**
- ✓ **Compare the standard simulation to a Biased Geodesic simulation**
- ✓ **Add a First Stage Region**
- ✓ **Create V-Shape Darts to alleviate the inside corners**
- ✓ **Create Slit Darts to alleviate the outside corners**

Estimated Time

10 min

Task 1 - Open a part.

1. Open **PRODUCIBILITY_D.prt**. The model displays as shown in Figure 4–80.

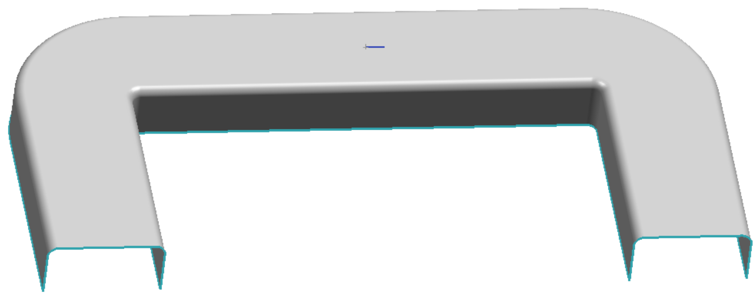


Figure 4–80

Task 2 - Create a full-body ply.

1. In Fibersim, review the existing **Laminate** and **Rosette**.
2. Create a full-body ply with the following parameters:
    - Name: **P001**
    - Step: **10**
    - Specified Orientation: **0**

Task 3 - Run the standard simulation.

1. Set the following simulation option for **P001**:
    - Fiber Spacing Factor: **0.2**
2. Run Net Producibility on **P001**. The producibility results should display as shown in Figure 4–81 and are not acceptable.

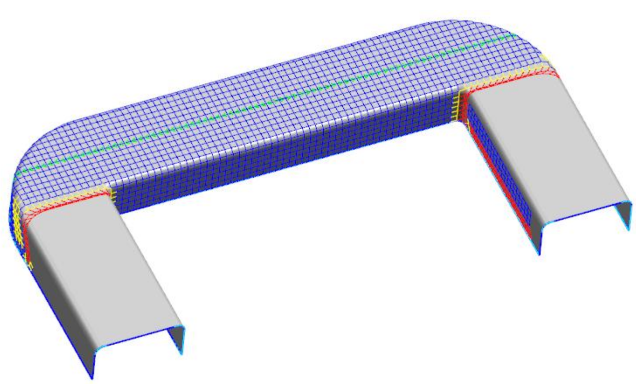


Figure 4–81

Task 4 - Run a Geodesic simulation and compare it to the standard simulation.

1. Set the following simulation options:
   - Propagation: **To Curve**
   - Constraint Curve: **Geodesic**
   - Fiber Angle From Curve: **0**
2. Run Net Producibility on **P001**. The producibility results should display as shown in Figure 4–82. Note that the simulation result has changed slightly but is still not acceptable.

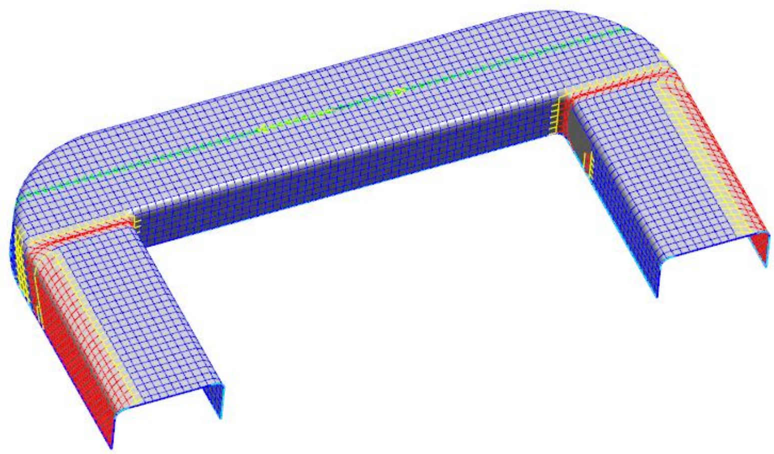


Figure 4–82

Task 5 - Add a First Stage Region.

In this task you will add a First Stage Region. The First Stage Boundary in this task is shown in Figure 4–83.

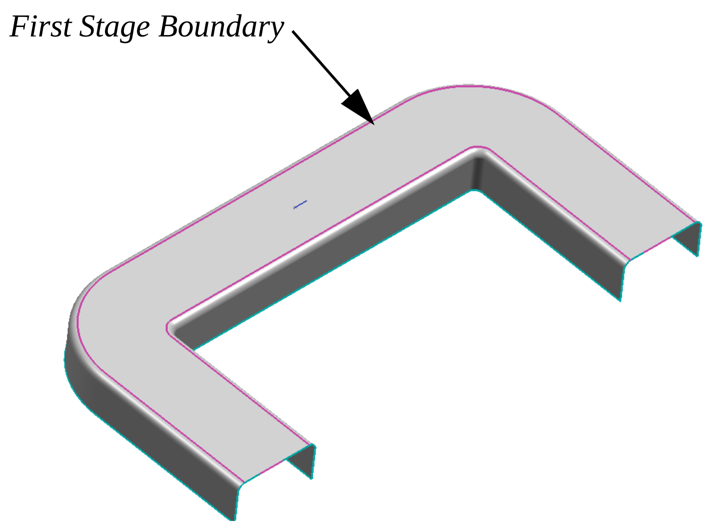


Figure 4–83

1. Show **FIRST_STAGE_REGION** layer category in the Layer Settings window.
2. Modify **P001** and set the following simulation options:
   - First Stage Region: **1st Stage Region**
   - Propagation: **Standard**
3. Run Net Producibility on **P001**. The producibility results should display as shown in Figure 4–84. The producibility display has changed but is still not acceptable.

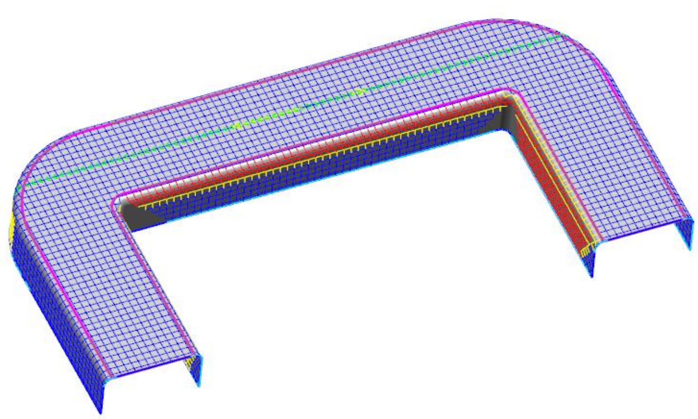


Figure 4–84

Task 6 - Develop a Darting Solution.

1. Modify **P001** and apply the six Darts as shown in Figure 4–85.

Note that an infinite number of darting solutions are available to solve this manufacturing challenge.

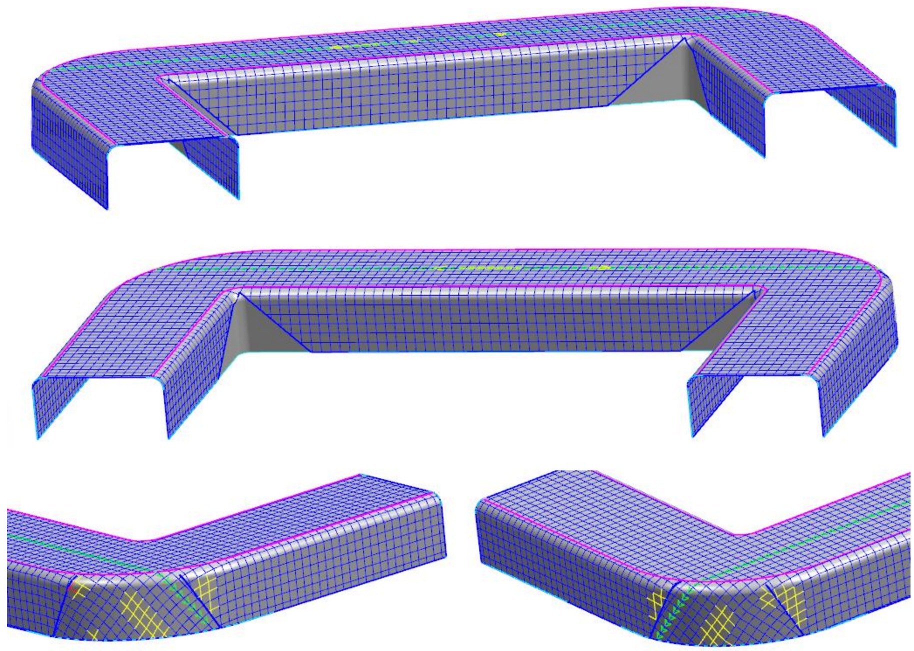


Figure 4–85

2. The producibility and flat pattern of an acceptable simulation is shown in Figure 4–86.

Note that the interior corners of the channel experienced a bridging situation and patches might need to be created to provide complete material coverage.

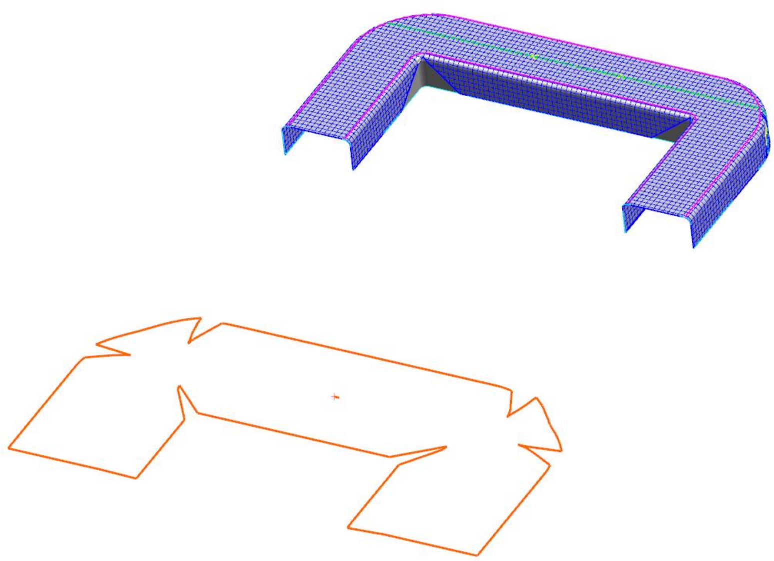


Figure 4–86

3. Compare the Net Flat Patterns of the standard simulation and the two-stage simulation as shown in Figure 4–87.

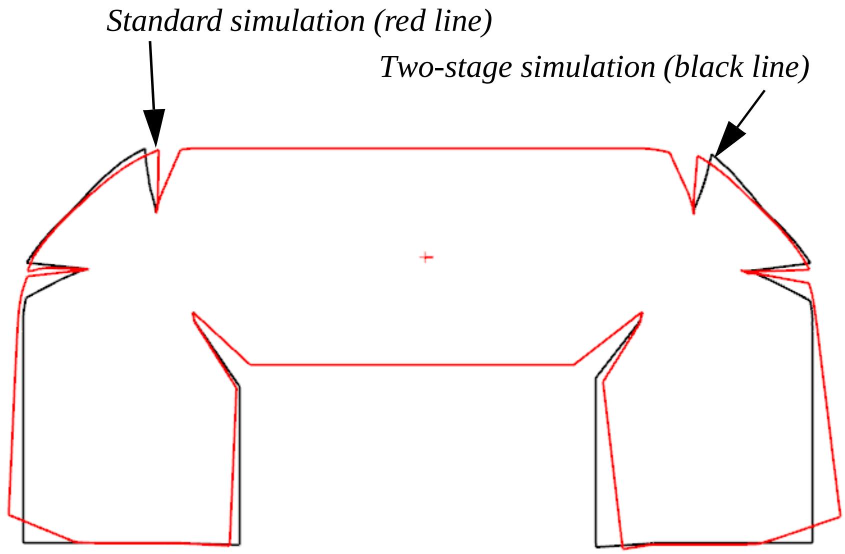


Figure 4–87

Project 1

Producibility

This chapter includes:

✓**Exercise P1: Producibility**

Exercise P1 Producibility

In this project, you will use ply based design to complete a layup scenario and resolve any producibility issues. The completed model displays as shown in Figure 1–1.

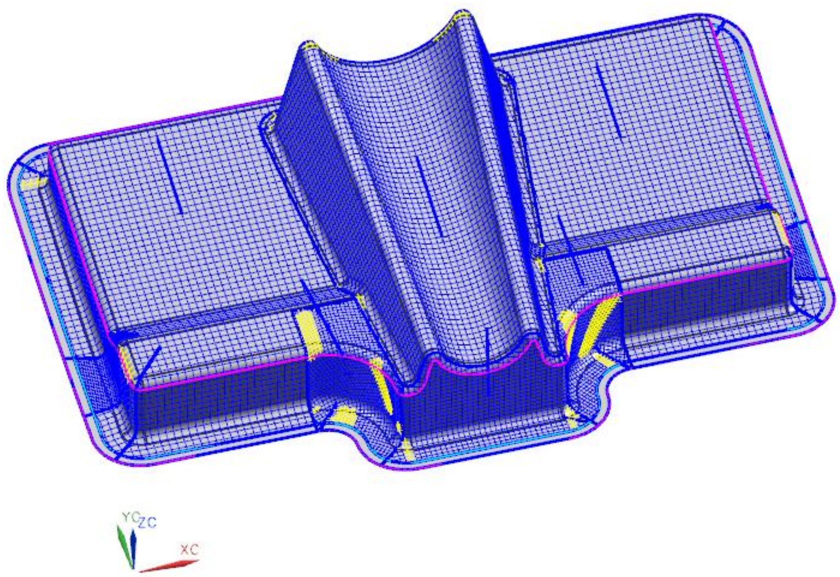


Figure 1–1

Goal

This project will test your knowledge on how to:

- ✓ **Create the required NX geometry**
- ✓ **Create Plies**
- ✓ **Resolve producibility issues with splicing and darting**

Given

The previously created geometry and design details are:

- ✓ **Net and Extended Boundaries have been created**
- ✓ **Laminate and Rosette have been created**

Estimated Time

2 hr

Task 1 - Open a part.

1. Open **PROJECT1_PRODUCIBILITY.prt**. The model displays as shown in Figure 1–2.

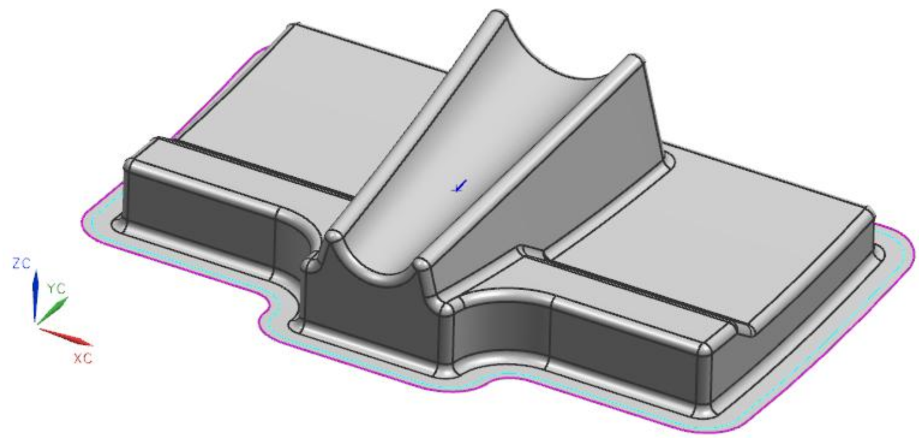


Figure 1–2

2. Review the Fibersim objects already created.

Task 2 - Create a full-body ply.

1. Create a new ply with the following parameters:
   - Step: **10**
   - Orientation: **0**
2. Generate net producibility and review the results.

The shape of the tool surface is too complicated for a full-body ply. You will need to either create separate plies with darts or splice the full-body ply.

Task 3 - Create ply objects.

1. Cover the part with a layer of 0 degree plies.
2. Resolve any producibility issues.

Task 4 - Cover the part with a layer of 45 degree plies. (Optional)

Repeat the same steps to cover the part with a layer of 45 degree plies.

Chapter 5

Rosettes

This chapter includes:

✓**Exercise 5a: Translational Rosette**

✓**Exercise 5b: Radial Rosette**

✓**Exercise 5c: Spine-Based Rosette**

Exercise 5a Translational Rosette

User Guide Reference:
2.7.2 Rosette Mapping Types
2.5.1 Simulation Display Types

In this exercise, you will learn how and why you use Translational Rosette.

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a Translational Rosette**
- ✓ **Analyze Producibility for Unidirectional Material**
- ✓ **Analyze Fiber Deviation**

Estimated Time

10 min

Task 1 - Create a Laminate.

1. Open **ROSETTES_A.prt**. The model displays as shown in Figure 5–1.

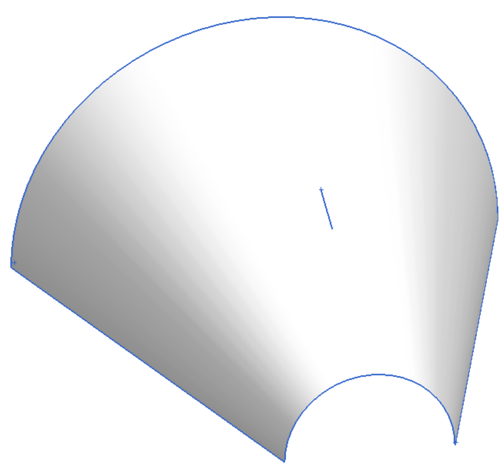


Figure 5–1

2. Press <F9> to start Fibersim.

T-24-in is a unidirectional tape with fibers running in one direction.

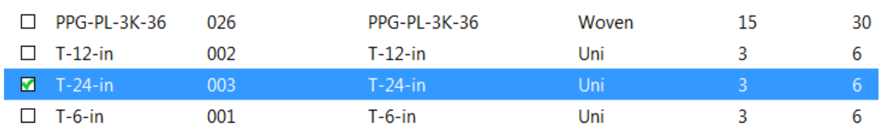
3. In the *Application Tree*, select **Laminate** under **Basic** and click (Create New).

4. In the Laminate's Standard form, enter the following parameters:

   - Name: **Cone**
   - Step: **1**

5. Next to *Default Material*, click (Link with Database Link Dialog) to open the Fibersim - link Material via Default Material dialog box.

6. Select **T-24-in** as shown in Figure 5–2 and click OK.



☐	PPG-PL-3K-36	026	PPG-PL-3K-36	Woven	15	30
☐	T-12-in	002	T-12-in	Uni	3	6
☑	T-24-in	003	T-24-in	Uni	3	6
☐	T-6-in	001	T-6-in	Uni	3	6

Figure 5–2

Task 2 - Link the geometry.

1. In the *Geometry* area of the form, link the Laminate to the NX geometry as shown in Figure 5–3:

   - Layup Surface: **Tool Surface**
   - Net Boundary: **Net Boundary**

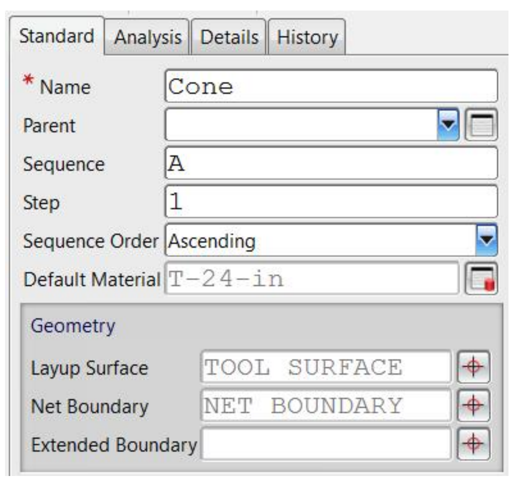


Figure 5–3

2. Click OK to complete the Laminate and return to the main window.

Task 3 - Create a Standard Rosette.

1. In the *Application Tree*, select **Rosette** under **Basic** and click (Create New).
2. Enter [Standard Rosette] for *Name*.
3. Link the following geometry:
   - Origin: **SR Origin**
   - Direction: **SR Zero Direction**
4. Ensure that the *Mapping Type* is set to **Standard** as shown in Figure 5–4.

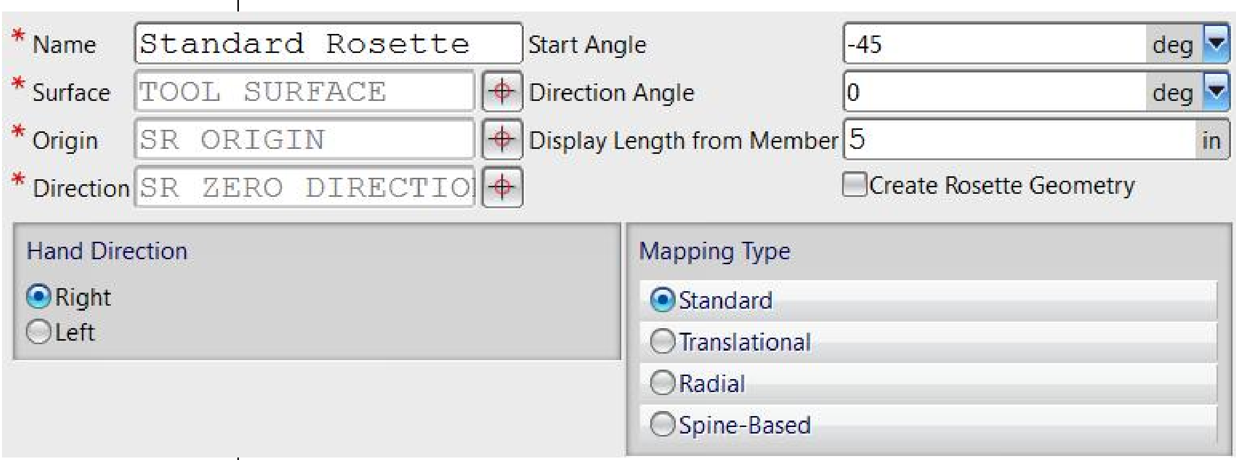


Figure 5–4

5. Click OK to complete the rosette and return to the main view.
6. Click (Highlight Type) in the main toolbar and select **Field** as shown in Figure 5–5.

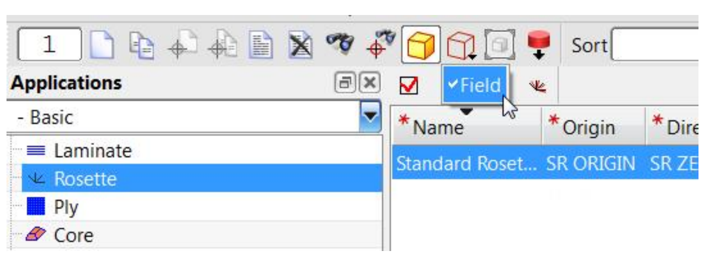


Figure 5–5

7. Activate the NX window. The model displays as shown in Figure 5–6.

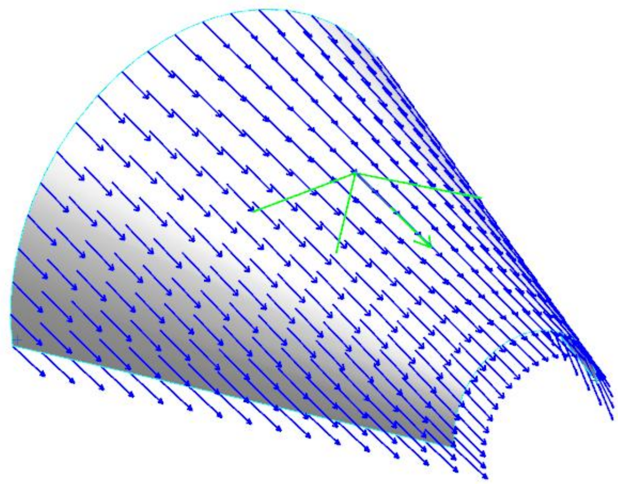


Figure 5–6

The axis of the cone in this exercise is assumed to be the primary strength direction for this particular part. Therefore, it is required that the material fibers be aligned in this direction.

8. Rotate the model and note that the fiber directions in the middle of the part are aligned along the rosette's 0-direction, which in turn, is parallel to the axis of the cone.

Task 4 - Create a full-body ply.

1. In the *Application Tree*, select **Ply** under **Basic** and click (Create New).
2. For P001, enter the following parameter:
   - Step: **10**
3. Click (Net Producibility). Click OK to close the message box.
4. Select the *Simulation Options* tab.

*Note that Fibersim automatically changes the Simulation Type to **Geodesic** for unidirectional materials.*

5. Note that Fibersim automatically changed *Propagation* to **To Curve** as shown in Figure 5–7.

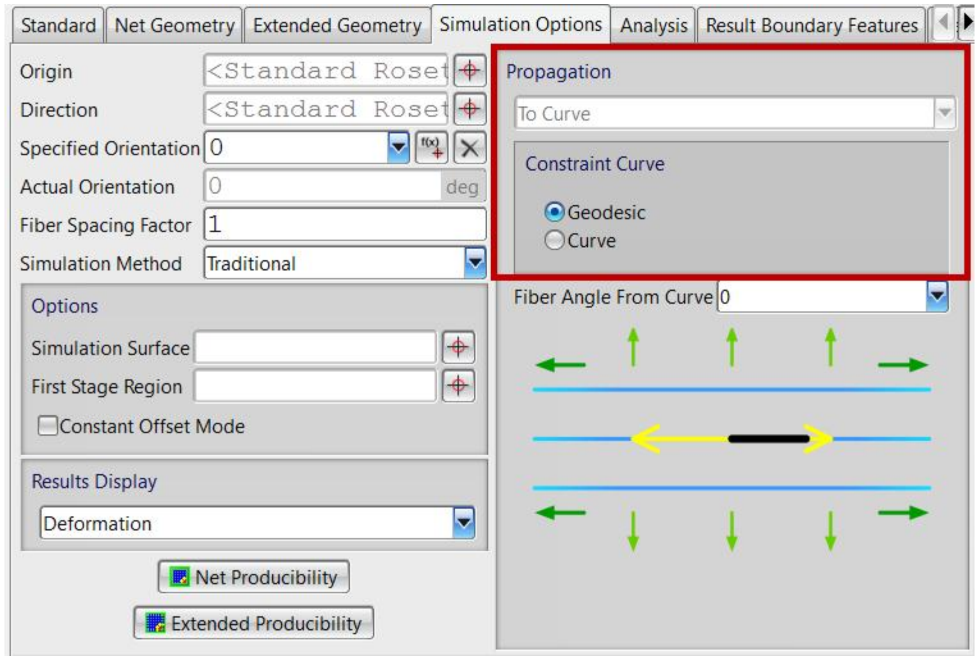


Figure 5–7

6. Activate the NX window. The model should display as shown in Figure 5–8.

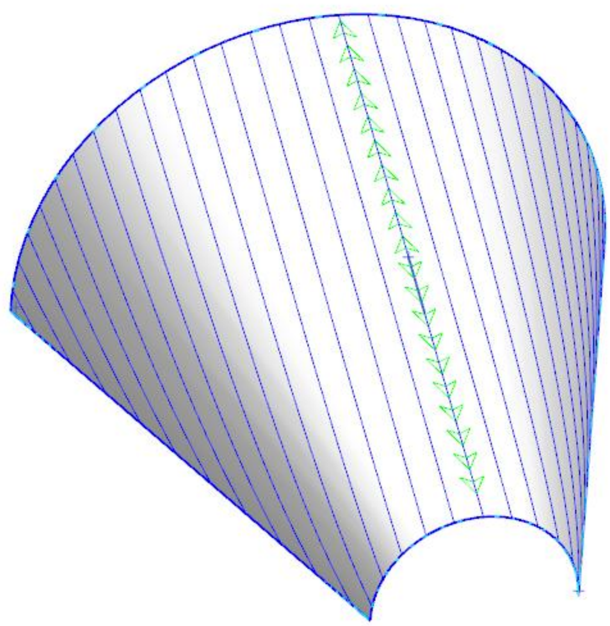


Figure 5–8

Task 5 - Change the ply origin.

1. Activate the Fibersim window.
2. Select the *Standard* tab.
3. Click (Link Geometry) next to the *Origin* field.
4. Make sure that the **Existing Point** filter is enabled.
5. In the model, select the **Ply Origin 2** point as shown in Figure 5–9.

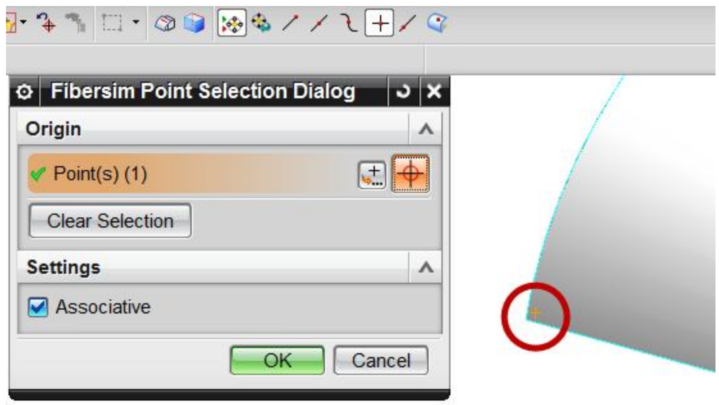


Figure 5–9

6. Click OK.
7. Click (Net Producibility) again. The model should now display as shown in Figure 5–10.

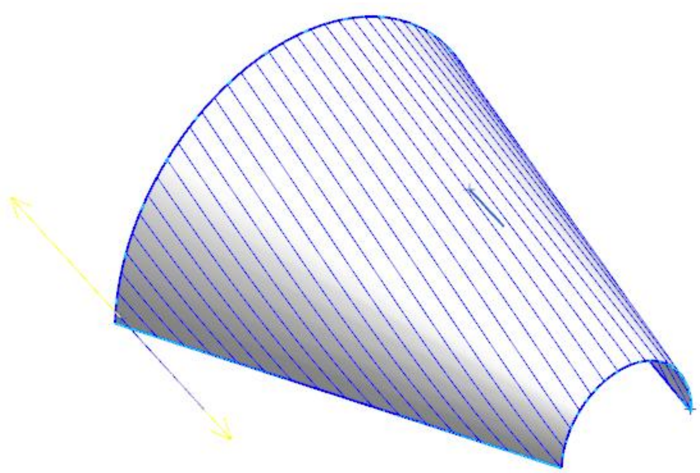


Figure 5–10

8. Note that the fiber paths (shown with blue lines) are no longer aligned with the part axis in the middle of the surface, where the rosette is located.

   The observed misalignment is due to the following:

   - The ply origin point is assumed to be where the fabric first touches the tool surface when laid up on the mold. At this location, for a 0-degree fabric orientation, the warp direction of the fabric roll is aligned with the 0-direction of the rosette. Similarly, for a 45-degree fabric orientation, the warp direction would be aligned with the 45-degree direction of the rosette, etc.
   - However, if the ply origin is selected away from the rosette, the rosette's directions need to be *transposed* to another location, (the ply origin). This is called *rosette mapping*.
   - With the Standard mapping method (the default option in Fibersim), orientation is mapped from the rosette so that it conforms to the part curvature. For example, a simple cube is shown in Figure 5–11. The arrows on the sides, top, and bottom of the cube indicate the 0-direction mapped to that location.

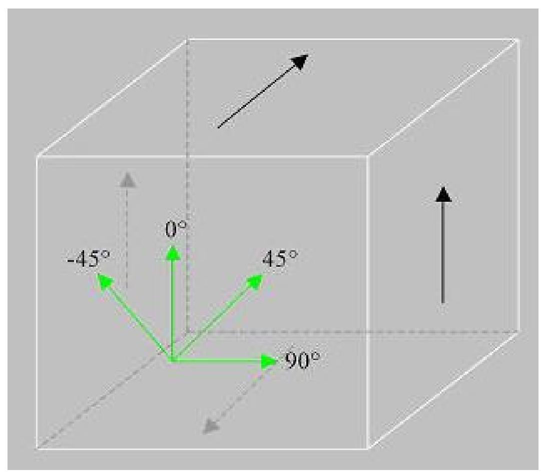


Figure 5–11

   - Since Standard mapping always takes surface normals into account, the result is always intuitively correct. Therefore, Standard mapping should be used on the majority of parts.
   - Standard mapping might cause a problem if the same *starting* 0-direction must be maintained no matter where on the surface you pick the ply origin. For this specific part, that should be the direction along the cone axis.
   - You will learn how to handle this requirement using Translational rosette mapping method in Task 6 and 7.

Task 6 - Analyze the Fiber Deviation.

1. Next to the *Origin* field, click (Link Geometry).
2. Select **SR ORIGIN**.
3. Click **OK**.
4. Select the *Simulation Options* tab.
5. In the Result Display drop-down list, select **Deviation** as shown in Figure 5–12.

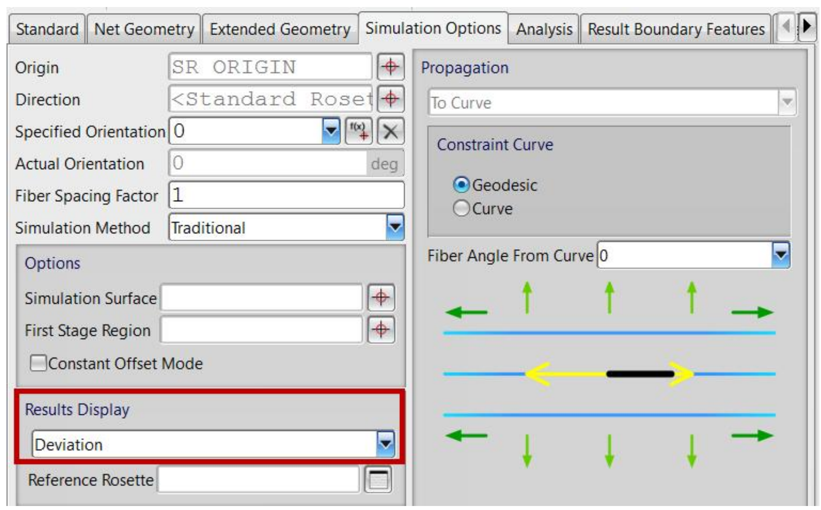


Figure 5–12

6. Click (Net Producibility). The model should now display as shown in Figure 5–13.

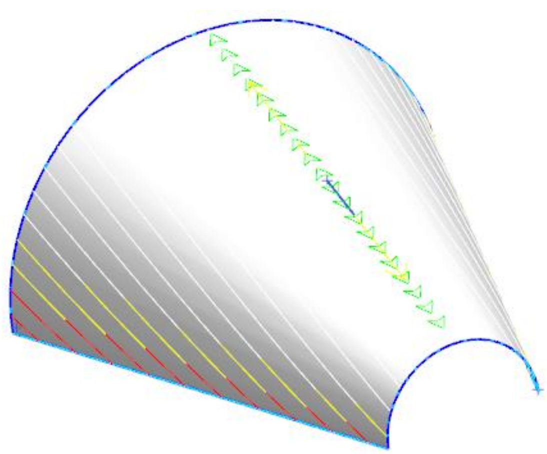
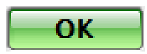


Figure 5–13

Warning (3 deg) and Limit (5 deg) angles can be adjusted in the Deviation area in the Rosette form.

7. The Deviation results display the amount of deviation between the simulated fiber orientations and the rosette’s orientation as follows:
   - White fiber paths indicate the difference under 3 degrees.
   - Yellow fiber paths indicate areas where the deviation is between 3 and 5 degrees.
   - Red fiber paths indicate areas where the deviation in direction exceeds 5 degrees.
8. Note that the deviation angles are computed between the fiber paths and the *mapped* rosette directions. Therefore, it is also important to select the proper type of rosette mapping to correctly assess the fiber deviations.
9. Click 

 to save **P001**.

Task 7 - Create a Translational Rosette.

Translational rosette mapping is a direct point-to-point translation technique, as shown in Figure 5–14. The 0-direction is directly translated, while the other fiber orientations are calculated by rotating the translated 0-direction around the tool surface normal to the point.

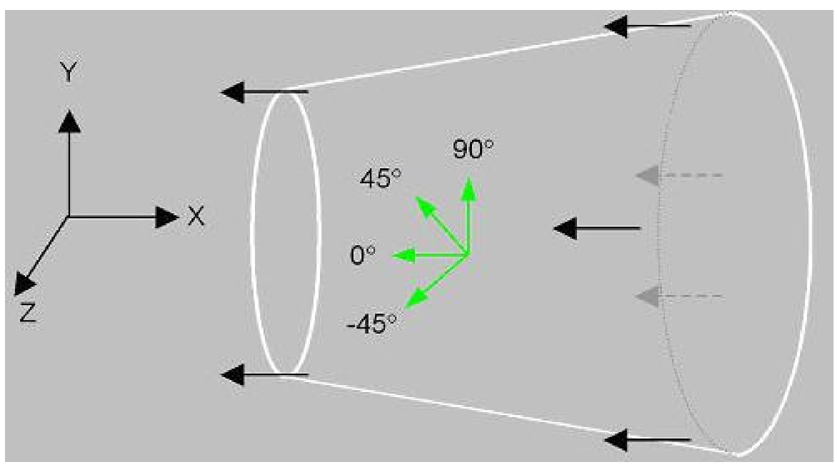


Figure 5–14

Commonly, point-to-point translation mapping used in surfaces of revolution where you want the 0-direction to always be parallel to an axis.

However, Translational mapping does not apply to the majority of parts. For example, the cube shown in Figure 5–15, uses Translational mapping. Note that point-to-point translation fails on two of the six faces.

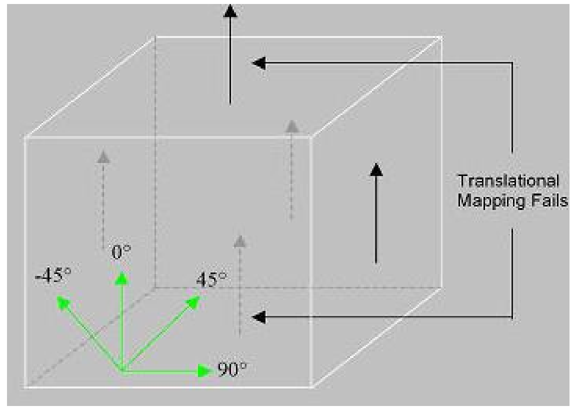


Figure 5–15

Translational Rosette requires the creation of a *reference plane* that is only used to determine the Rosette's directions rather than laying up actual plies.

1. In the *Application Tree*, select **Rosette** under **Basic**
2. Click  (Create New) and rename the rosette as [Trans Rosette].
3. Link the following geometry:
   - Surface: **TR Ref Surface**
   - Origin: **TR Origin**
   - Direction: **TR Zero Direction**
4. Set the *Mapping Type* to **Translational** as shown in Figure 5–16.

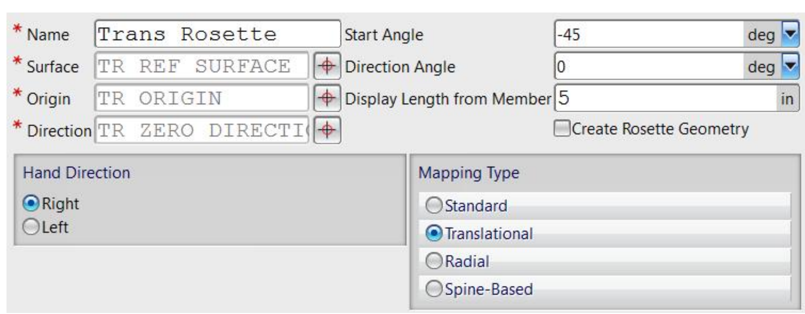


Figure 5–16

5. Click .

Task 8 - Compare Rosettes.

1. Highlight **Standard Rosette** from the list of objects.

2. Click (Compare Rosettes) in the main toolbar as shown in Figure 5–17.

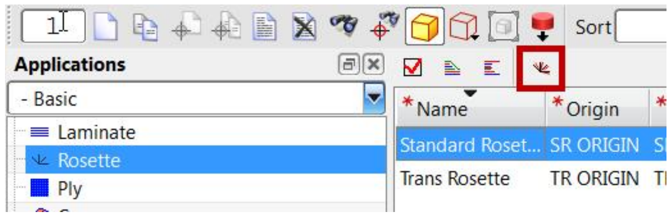


Figure 5–17

3. Click (Link with Link Dialog) next to the *Comparison Object* field.

4. Link **Trans Rosette** and click OK.

5. Activate the NX window. The model displays as shown in Figure 5–18.

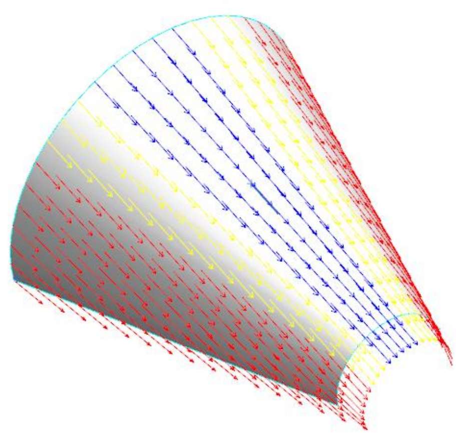


Figure 5–18

6. Click OK to close the Rosette Comparison window.

7. Click (Highlight Type) in the main toolbar.

8. Clear the option for **Field**.

Task 9 - Change the ply's rosette to Translational.

1. In the *Application Tree*, select **Ply** under **Basic.**
2. Double-click **P001** to modify.
3. Select the *Standard* tab.
4. In the Rosette drop-down list, select **Trans Rosette** as shown on Figure 5–19.

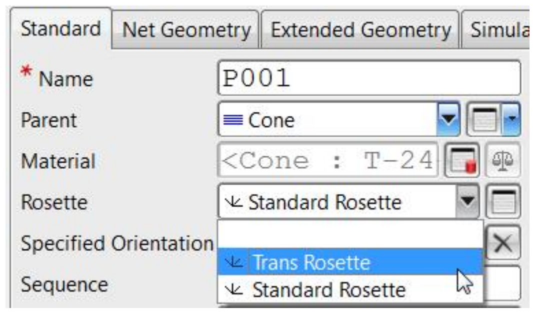


Figure 5–19

5. The ply form should update as shown in Figure 5–20.

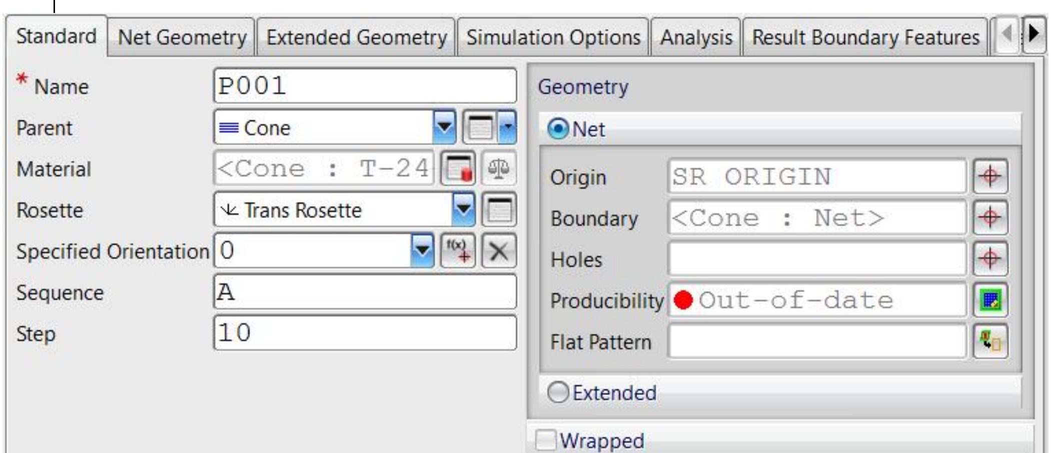


Figure 5–20

Task 10 - Analyze Fiber Paths

1. Select the *Simulation Options* tab.

2. In the Result Display drop-down list, select **Deformation** as shown in Figure 5–21.

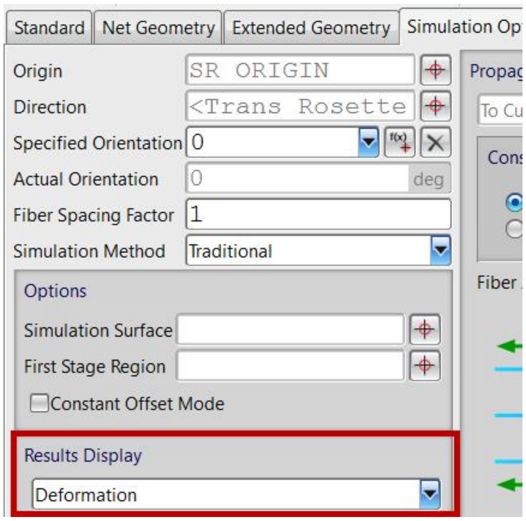


Figure 5–21

3. Click (Net Producibility). The model should display as shown in Figure 5–22.

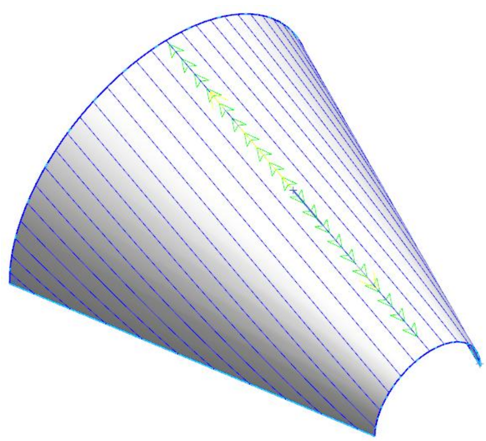


Figure 5–22

4. Note that all of the fiber paths are oriented along the part axis.

5. Select the *Standard* tab.

6. Click (Link Geometry) next to the ply *Origin* field.

7. Select **Ply Origin 2** in the model, as shown in Figure 5–23.

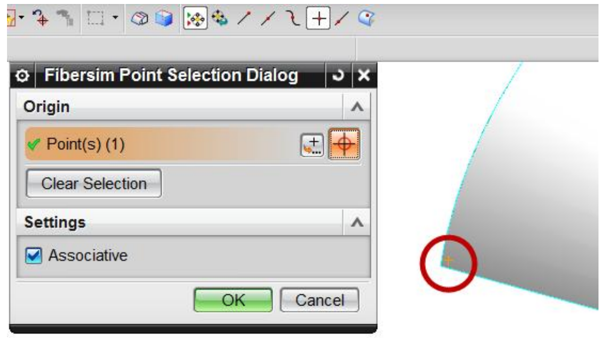
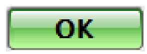
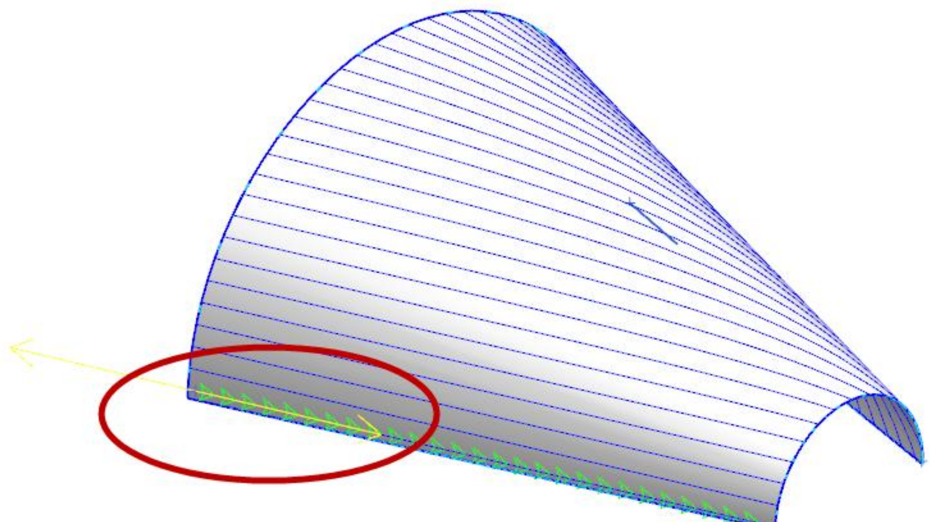


Figure 5–23

8. Click .

9. Click  (Net Producibility). The model should now display as shown in Figure 5–24.



Rosette mapped to the new ply origin

Figure 5–24

10. Note that the fiber paths are now aligned with the part axis at the new ply origin.

Exercise 5b Radial Rosette

User Guide Reference:
2.7.2 Rosette Mapping Types

Radial Rosette mapping is a specialized form of translation, used for spherical-shaped parts, such as satellite dishes and dome covers.

In Radial mapping, the point-to-point translation takes place with the 0-direction always set in a radial direction from the center of the part as shown in Figure 5–25. Therefore, the Radial Rosette origin must be always placed in the *exact center* of the part.

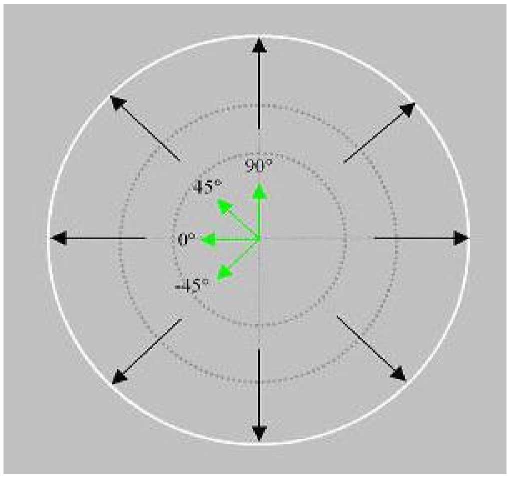


Figure 5–25

In this exercise, you will create a laminate consisting of four plies butt-spliced onto a model of a satellite dish. The completed model is shown in Figure 5–26.

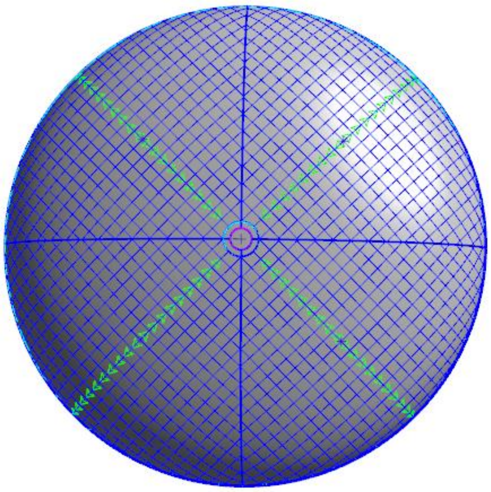


Figure 5–26

Goal

After you complete this exercise, you will be able to:

✓ **Create a Radial Rosette**
✓ **Create butt-spliced plies on multi-domain surfaces**
✓ **Analyze Producibility and Fiber Deviation**

Estimated Time

5 min

Task 1 - Open a part.

1. Select **File > Open** and select **ROSETTES_B.prt**. The model displays as shown in Figure 5–27.

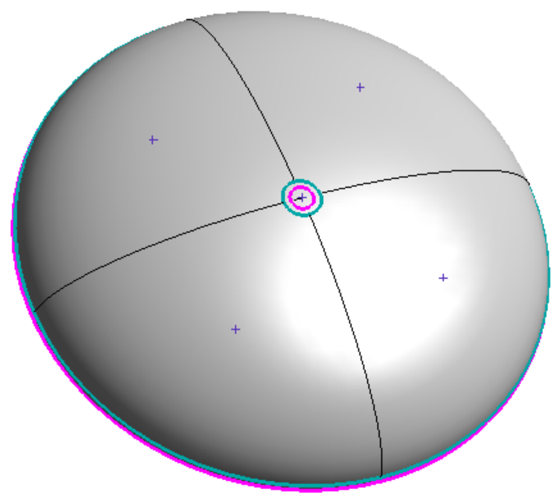


Figure 5–27

Task 2 - Create a Laminate.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** under **Basic** and click  (Create New).
3. In the Laminate's Standard form, enter the following parameters:
   - Name: **Dish**
   - Step: **1**

4. Next to *Default Material*, click (Link with Database Link Dialog).

5. Select **PPG-PL-3K** and click OK.

6. In the *Geometry* area of the form, link the Laminate to the NX geometry as shown in Figure 5–28:

   - Layup Surface: **Tool Surface**
   - Net Boundary: **Net_Boundary_1, Net_Boundary_2** (two cyan lines)
   - Extended Boundary: **Extnded_Boundary_1, Extended_Boundary_2** (two magenta lines)

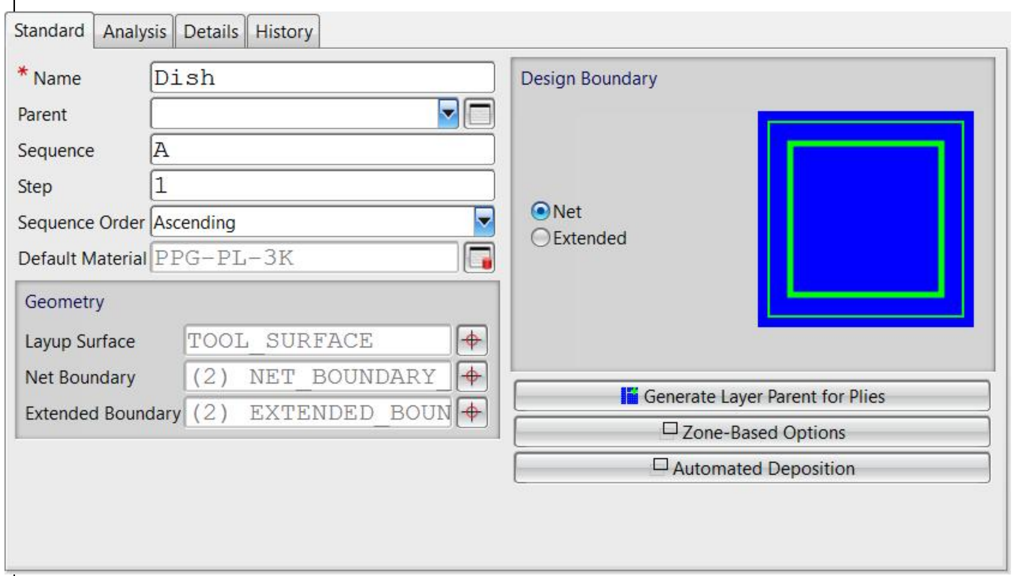


Figure 5–28

7. Click OK to save the Laminate and return to the main window.

Task 3 - Create a Radial Rosette.

1. In the *Application Tree*, select **Rosette** under **Basic** and click (Create New).

2. Link the following geometry:

   - Origin: **Rosette Origin**
   - Direction: **Zero Direction**

3. Set the *Mapping Type* to **Radial.** The form should display as shown in Figure 5–29.

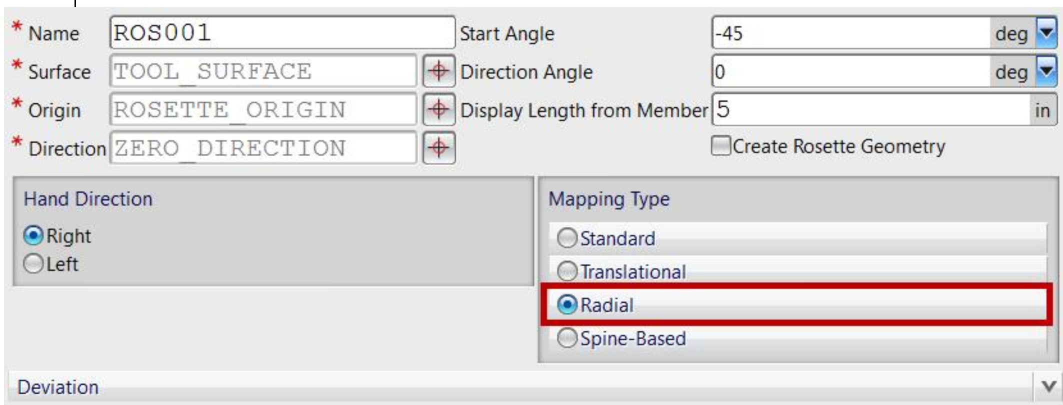


Figure 5–29

4. Click  .

Task 4 - Create the first ply.

1. In the *Application Tree*, select **Ply** under **Basic** and click  (Create New).
2. Enter [10] for *Step*.
3. Link **Gore_1_Origin** for *Net* > *Origin*. The ply form should update as shown in Figure 5–30.

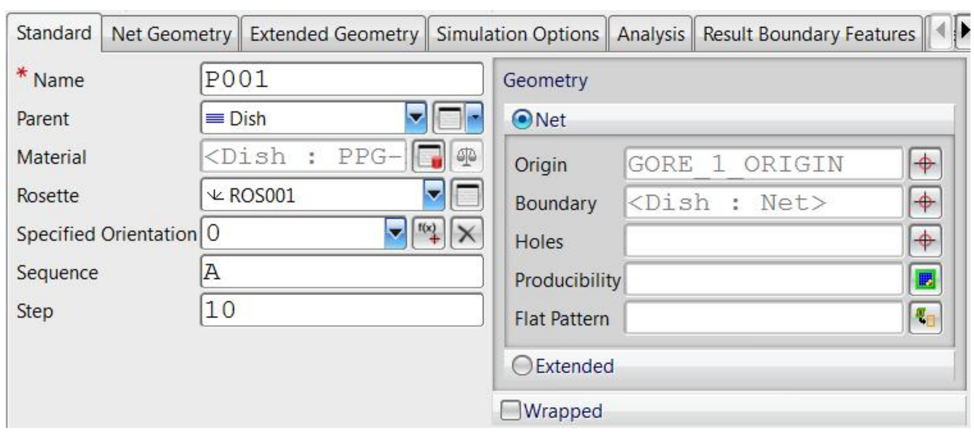


Figure 5–30

4. Click (Net Producibility). The warning message box opens as shown in Figure 5–31, because the Net Boundary of the ply consists of two disconnected contours. Click **OK** to close the message box.

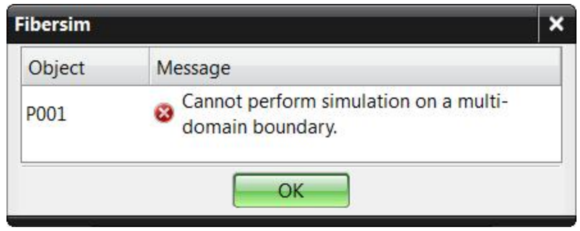


Figure 5–31

5. Click (Link Geometry) next to the *Boundary* field and select the curves **Radial 1** and **Radial 2**, as shown in Figure 5–32, to re-limit the ply to the first quadrant of the surface.

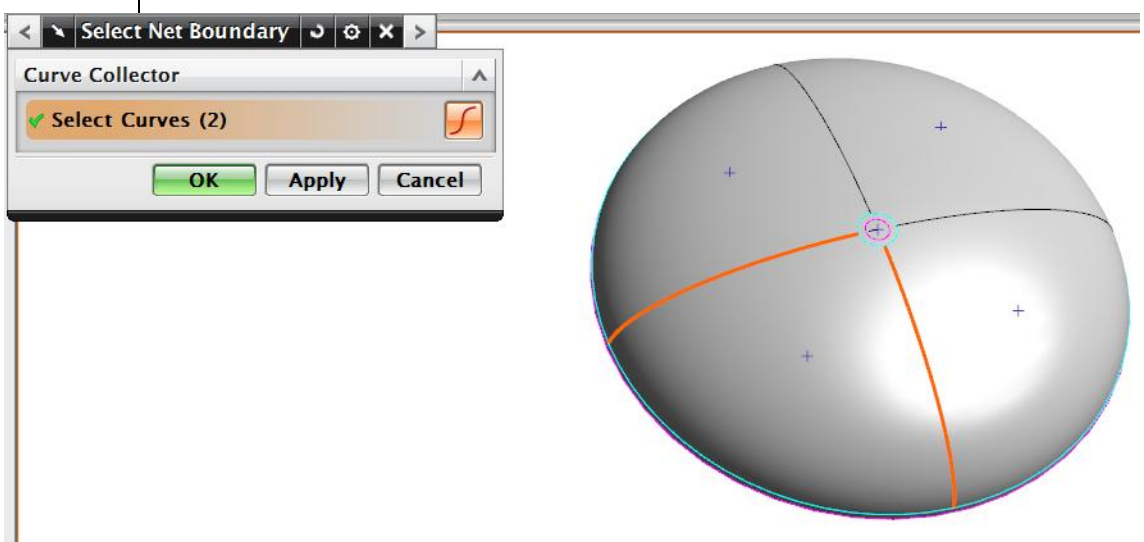


Figure 5–32

6. Click **OK**.

7. Click (Net Producibility). (Ignore the warning message that opens prompting you about the material width.)

8. The model should display as shown in Figure 5–33.

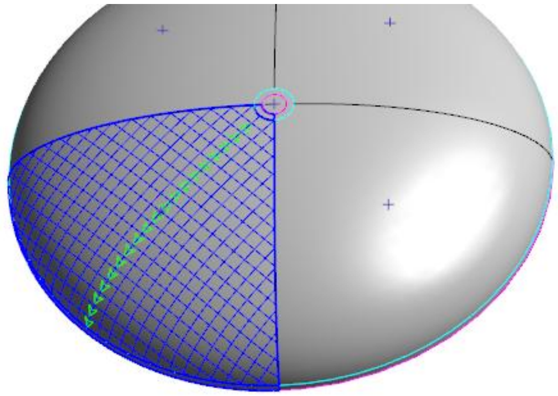


Figure 5–33

9. Click  to complete the P001 ply and return to the main view.

Task 5 - Create the remaining three plies.

1. Set the *Object Count* to [3] as shown in Figure 5–34.

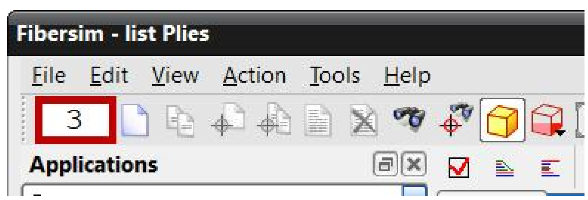


Figure 5–34

2. In the ply list, select **P001**, right-click and select **Create Based On**.

3. Press <Ctrl> + <T> to switch the ply form to the table view and select the *Net Geometry* tab. The form should display as shown in Figure 5–35.

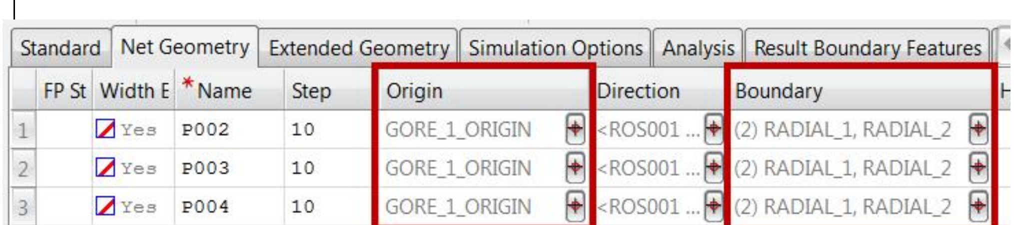


Figure 5–35

4. Note that the Origins and Boundaries (highlighted in Figure 5–35) of the new plies are incorrect (because they were copied from the first ply). Therefore, they need to be re-linked to the correct NX geometry.

5. Link the Origins of the new plies as follows:
   - P002: **Gore_2_Origin**
   - P003: **Gore_3_Origin**
   - P004: **Gore_4_Origin**

6. Link the Boundaries of the new plies as follows:
   - P002: **Radial_2, Radial_3**
   - P003: **Radial_3, Radial_4**
   - P004: **Radial_4, Radial_1**

7. The ply form should update as shown in Figure 5–36.

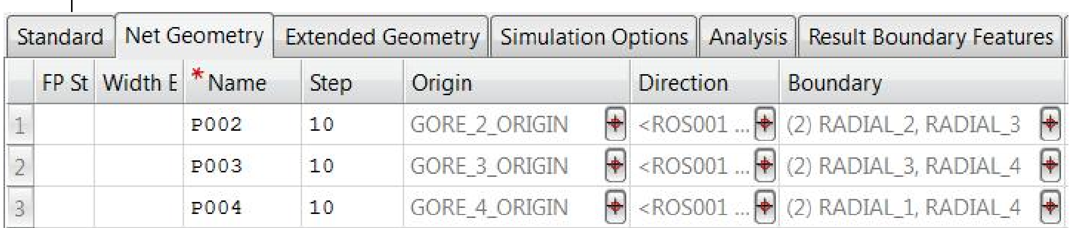


Figure 5–36

8. Click **OK** to complete the plies and return to the main view.

Task 6 - Analyze the producibility of all of the plies.

1. In the ply list, multi-select P001 through P004 as shown in Figure 5–37.

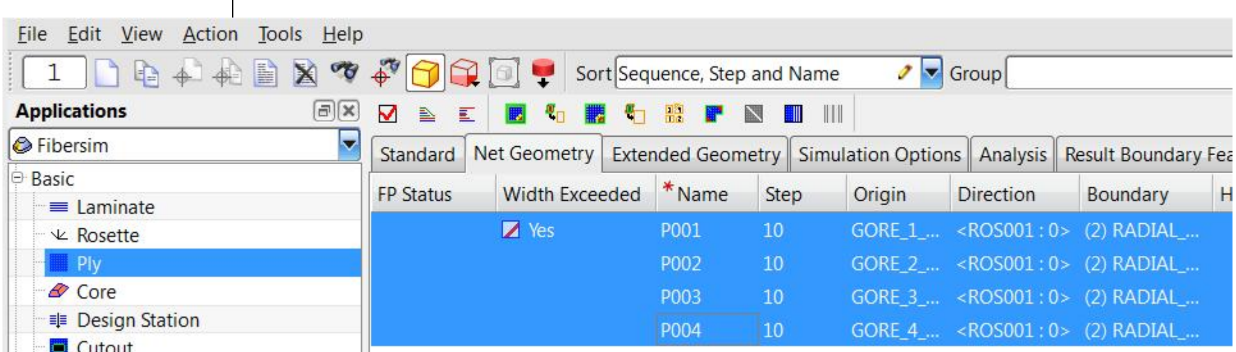


Figure 5–37

2. Click  (Net Producibility). (Ignore the warning message box that opens prompting you about the material width.)

3. Activate the NX window. The model should display as shown in Figure 5–38.

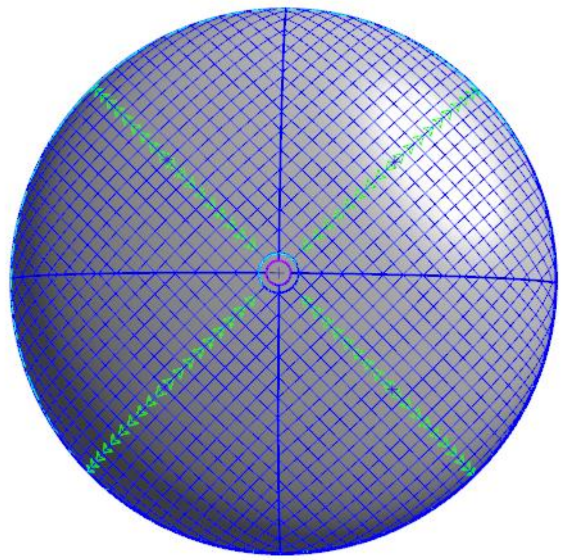


Figure 5–38

4. Rotate the model and note that the fiber directions in the middle of the quadrants are aligned along the radial direction of the surface, This is done using the Radial Rosette.

5. Close Fibersim and save the model.

Exercise 5c Spine-Based Rosette

User Guide Reference:
2.7.2 Rosette Mapping Types

Spine-Based Rosettes are used for spine-based part, such as stringers and floor beams. 0 direction of a Spine-Based Rosette follows a direction curve, whereas traditional rosettes yield an idealized 0 direction over an entire part as shown in Figure 5–39.

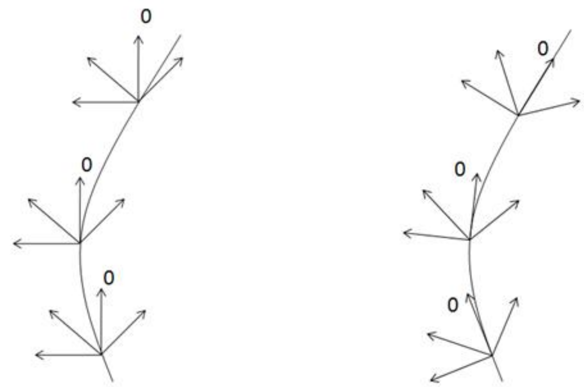


Figure 5–39

In this exercise, you will create Flat Patterns to compare the results from different types of Rosette mapping types and Simulation options.

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a Spine-Based Rosette**
- ✓ **Analyze Producibility utilizing Spine-Based Simulation**
- ✓ **Create Flat Pattern**

Estimated Time

15 min

Task 1 - Open a part.

1. Select **File > Open** and select **ROSETTES_C.prt**. The model displays as shown in Figure 5–40.

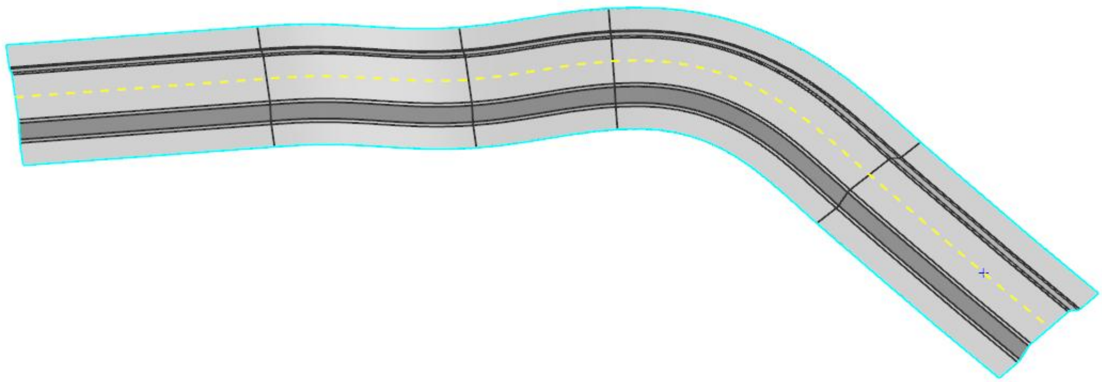


Figure 5–40

Task 2 - Create a Laminate.

Press <Ctrl> + <T> to toggle between the form and table view.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** under **Basic** and click (Create New).
3. In the Laminate's Standard form, enter the following parameters:
   - Name: **LAM001**
   - Step: **1**
4. Next to Default Material, click (Link with Database Link Dialog) and select **PPG-PL-3K** and click OK.

5. In the *Geometry* area of the form, link the Laminate to the NX geometry as shown in Figure 5–41:
   - Layup Surface: **Tool Surface**
   - Net Boundary: **Net Boundary**

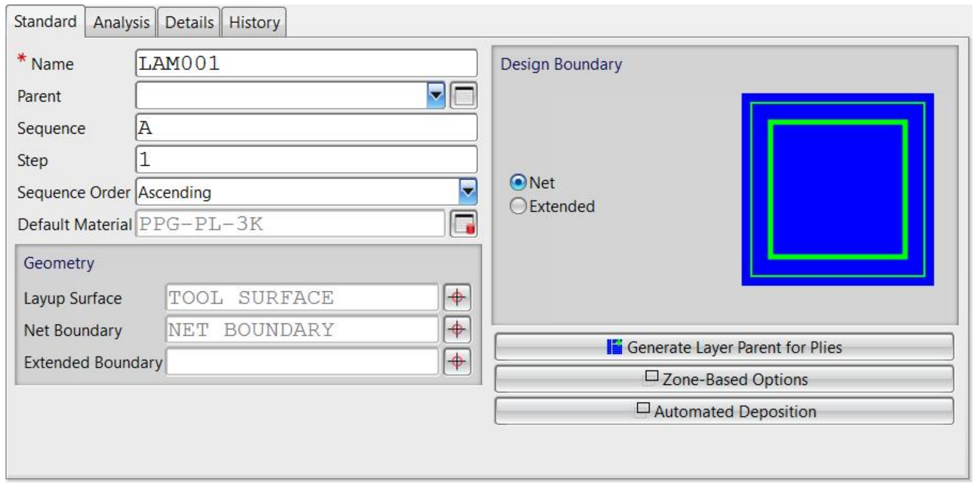


Figure 5–41

6. Click **OK** to complete the Laminate and return to the main window.

Task 3 - Create a Standard Rosette.

1. In the *Application Tree*, select **Rosette** under **Basic** and click (Create New).
2. Change the rosette name to [Standard Rosette] and link the following geometry as noted in Figure 5–42:
   - Origin: **Origin**
   - Direction: **Spine**

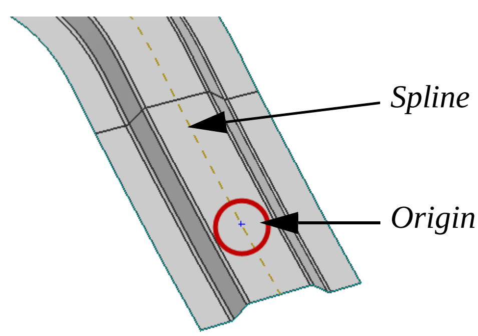


Figure 5–42

3. Ensure that the Mapping Type is set to Standard as shown in Figure 5–43.

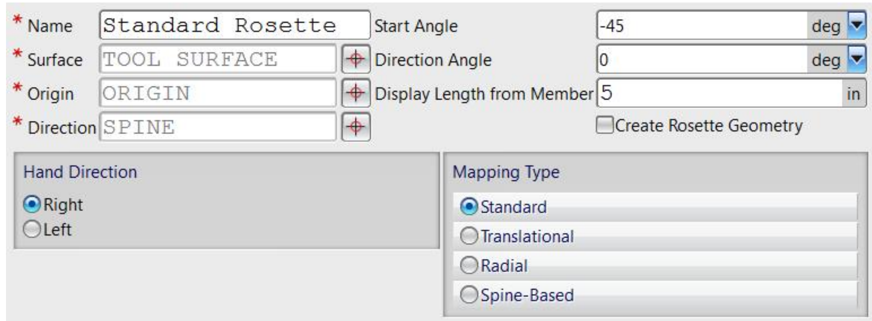


Figure 5–43

4. Click OK to save **ROS001**.

Task 4 - Create a full-body ply and a flat pattern with Standard Rosette.

1. In the *Application Tree*, select **Ply** under **Basic** and click (Create New).
2. Enter [10] for *Step*.
3. Select the *Simulation Options* tab.
4. Enter [0.3] for *Fiber Spacing Factor*.
5. Click (Net Producibility).
6. Click (Generate the Net Flat Pattern) and click OK.
7. Switch to NX and review the flat pattern created. Flat pattern for P001 is shown in Figure 5–44.

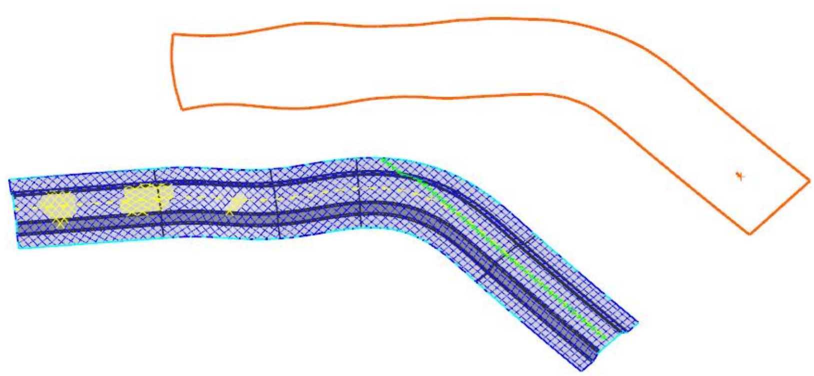


Figure 5–44

Task 5 - Create a new ply based on the first ply.

In this task, you will create a new full-body ply based on the first ply you created, but use a different simulation option to see how it affects the flat pattern.

1. Right-click on **P001** and select **Create Based On**.
2. Change *Step* to [20].
3. Select the *Simulation Options* tab.
4. Select **To Curve** from the list for *Propagation*.
5. Select the **Curve** option under *Constraint Curve*.
6. Link **Spine** for *Constraint Curve* as shown in Figure 5–45.

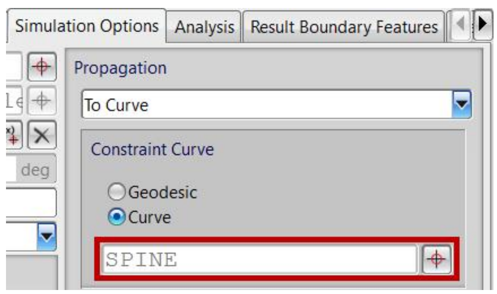


Figure 5–45

7. Click  (Net Producibility).
8. Click  (Generate the Net Flat Pattern) and click **OK**.
9. Switch to NX and review the flat pattern created. Flat pattern for P002 is shown in Figure 5–46.

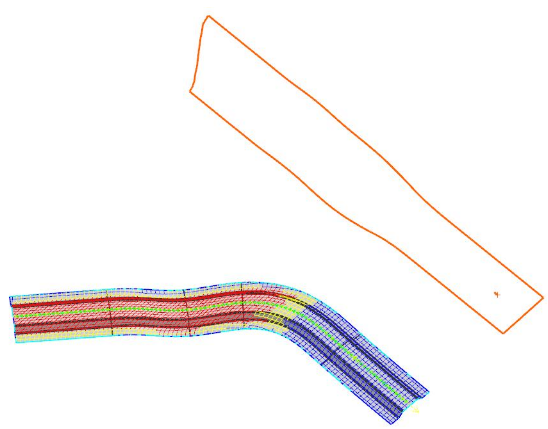


Figure 5–46

Task 6 - Create a Spine-Based Rosette.

In this task, you will create a Spine-Based Rosette to be used later with Spine-Based Simulation.

1. In the *Application Tree*, select **Rosette** under **Basic** and click  (Create New).
2. Change the rosette name to [SpineBased Rosette] and link the following geometry:
   - Origin: **Origin**
   - Direction: **Spine**
3. Change the Mapping Type to **Spine-Based** and turn on the option for *Reverse Spine Direction* as shown in Figure 5–47.

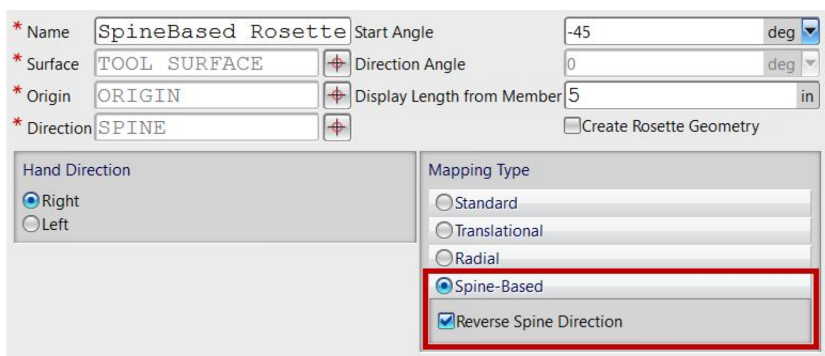


Figure 5–47

4. Click  to save **SpineBased Rosette**.

Task 7 - Create a full-body ply and a flat pattern with SpineBased Rosette.

1. In the *Application Tree*, select **Ply** under **Basic** and click  (Create New).
2. For the P003 ply, enter the following parameter:
   - Step: **30**
   - Rosette: **SpineBased Rosette**
3. Select the *Simulation Options* tab.
4. Enter [0.3] for *Fiber Spacing Factor*.

5. Select **Spine** for *Simulation Method*.
6. Click Net Producibility and review the results in NX window.
7. Change *Results Display* to **Deviation** and review the results in NX window.
8. Click (Generate the Net Flat Pattern) and click OK to save **P003**.
9. Switch to NX and review the flat pattern created. Flat pattern for P003 is shown in Figure 5–48.

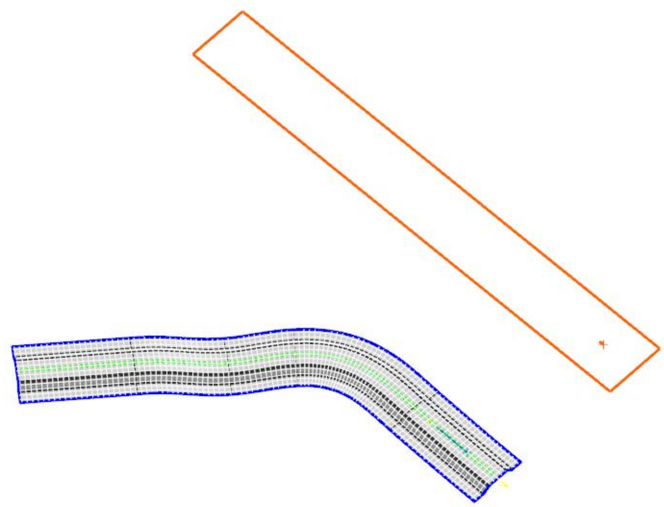


Figure 5–48

10. Close Fibersim and save the model.

Chapter 6

Utilities

This chapter includes:

- ✓**Exercise 6a: Curve Creation and Ply Drop-off Utilities**
- ✓**Exercise 6b: Fiber Path Curve**
- ✓**Exercise 6c: Symmetric Laminate**

Exercise 6a Curve Creation and Ply Drop-off Utilities

User Guide References:
B.5.2 Curve Creation
B.5.5 Curve Offset
B.3.6 Ply Drop-Off
B.5.4 Boundary Simplification

In this exercise, you will use various utilities to create curves and ply drop-offs. The completed model displays as shown in Figure 6–1.

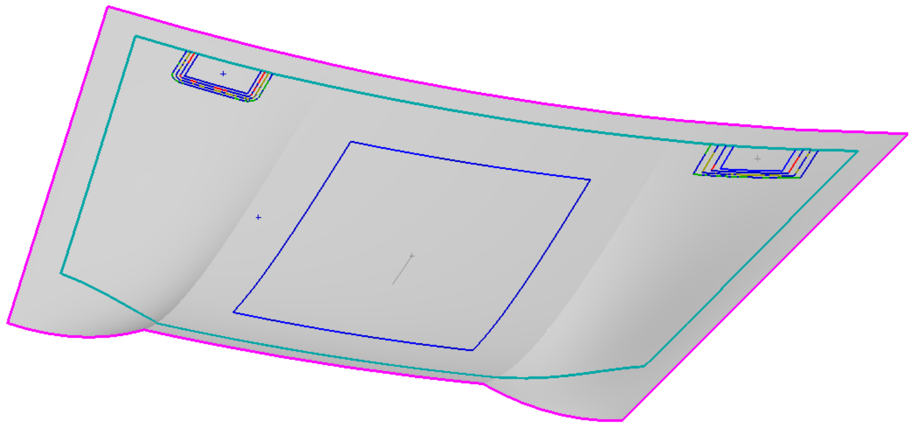


Figure 6–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a curve using Curve Creation**
- ✓ **Create a Curve Offset - Constant**
- ✓ **Create a Curve Offset - Directional**
- ✓ **Use Ply Drop-off**
- ✓ **Use Boundary Simplification**
- ✓ **Use Publish the Fibersim Composite Format**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **UTILITIES_A.prt**. The model displays as shown in Figure 6–2.

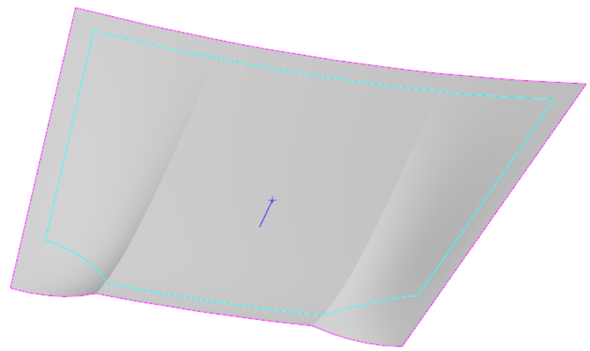


Figure 6–2

Task 2 - Create a curve using Curve Creation.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** and **Rosette** under **Basic**, to review the existing Fibersim objects.
3. In the *Application Tree*, select **Ply** under **Basic** to display the two plies: P001 and P002.

Press <Ctrl> + <L> to open the Layer Settings window.

4. Show the **HOLE_GEOMETRY** layer category in the Layer Settings window. The four points display as shown in Figure 6–3.

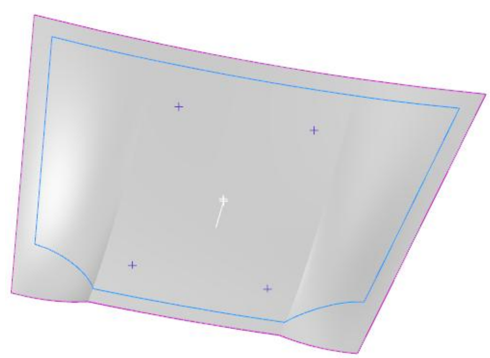


Figure 6–3

5. Press <F9> to start Fibersim.

6. In Fibersim, select **Tools > Curve Creation > Curve Creation** as shown in Figure 6–4.

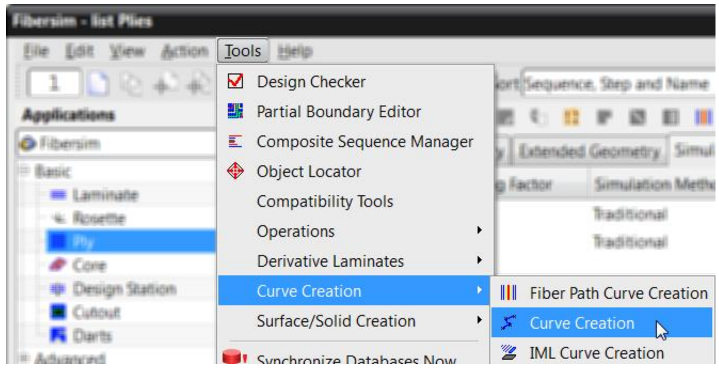
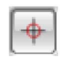


Figure 6–4

7. Next to the *Surface* field, click  (Link Geometry) and select the **Tool Surface**.

8. Click .

9. Next to the *Points to Connect* field, click  (Link Geometry).

10. Make sure that the **Existing Point** filter is enabled as shown in Figure 6–5.

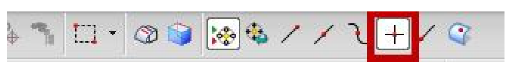


Figure 6–5

The curve will be created by connecting the selected points in the order of selection.

11. Select the four points: **Hole1**, **Hole2**, **Hole3**, and **Hole4** as shown in Figure 6–6.

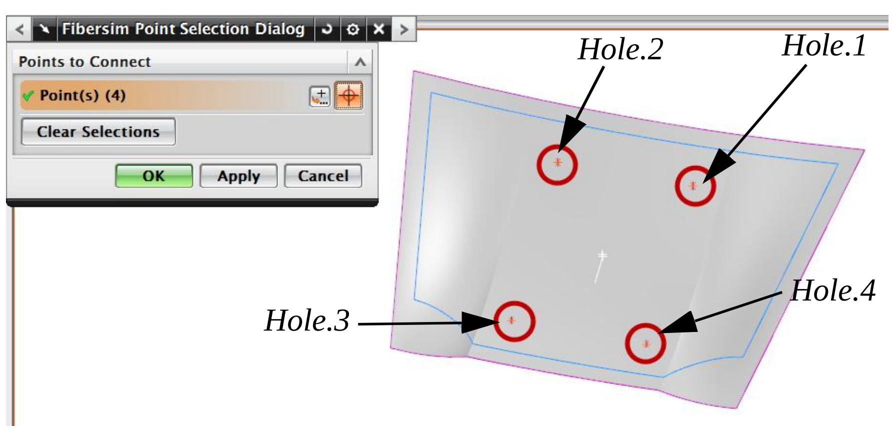


Figure 6–6

*If **Close Curve** is not selected, the points will be connected in order without connecting the last point to the first one.*

12. Click 

.

13. Select the checkbox for the **Close Curve** option. The Fibersim - Curve Creation window updates as shown in Figure 6–7.

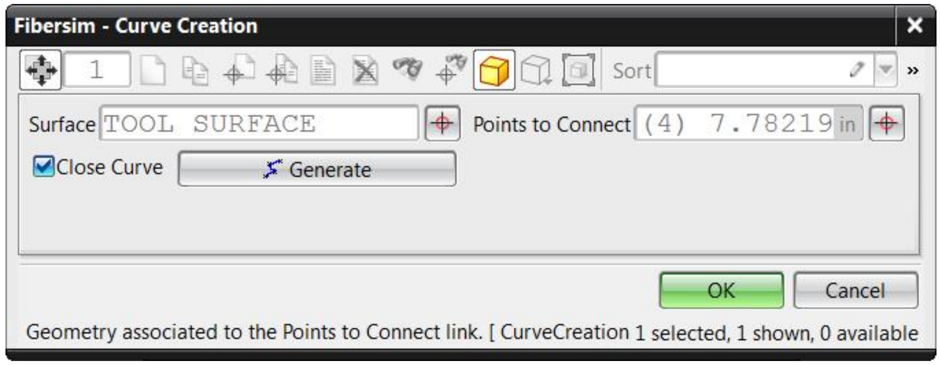
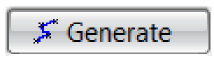
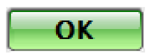


Figure 6–7

14. Click 

.

15. Click 

The model updates with the curve as shown in Figure 6–8.

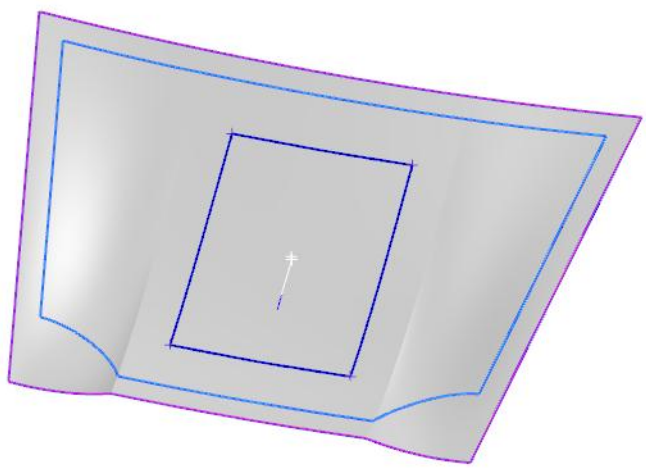


Figure 6–8

Task 3 - Use Curve to create a Hole.

1. In the ply list, double-click on **P002** to modify it.

2. In the *Geometry* area, next to the *Holes* field, click (Link Geometry) and select the curve you just created as shown in Figure 6–9.

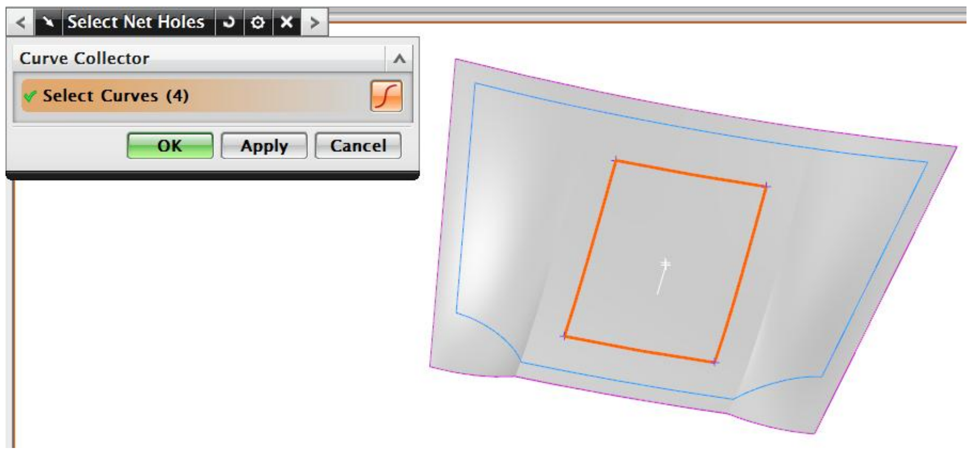


Figure 6–9

3. Click OK. A warning message about the origin should display as shown in Figure 6–10.

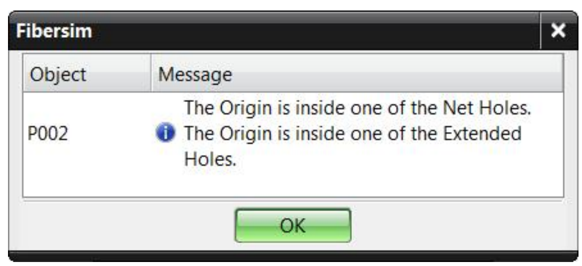


Figure 6–10

4. Click OK to close the message.

5. In the *Geometry* area, next to the *Origin* field, click  (Link Geometry) and indicate a new origin outside the hole boundary as shown in Figure 6–11. (Make sure to enable **Point on Face** selection filter.)

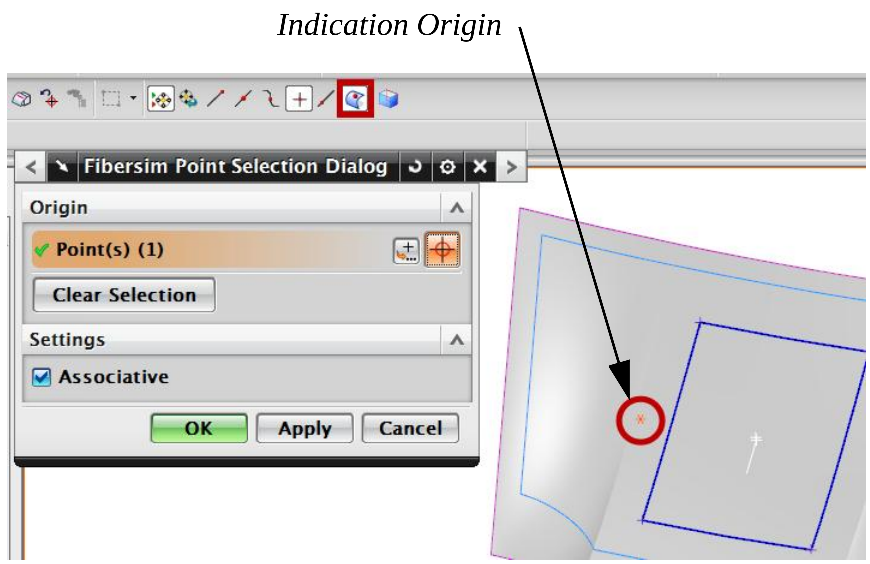


Figure 6–11

6. Click  twice to save **P002**.

Task 4 - Create a Constant Curve Offset.

Press <Ctrl> + <L> to open the Layer Settings window.

1. Show the **Reinforcement** layer category in the Layer Settings window. The two reinforcement boundaries and two origins display as shown in Figure 6–12.

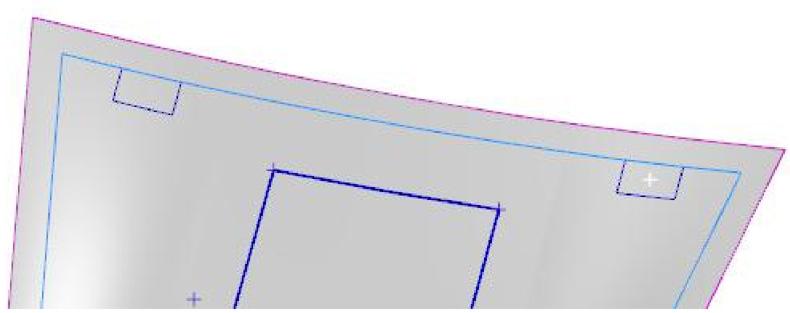


Figure 6–12

2. Press <F9> to start Fibersim again.
3. In Fibersim, select **Tools > Curve Creation > Curve Offset** as shown in Figure 6–13.

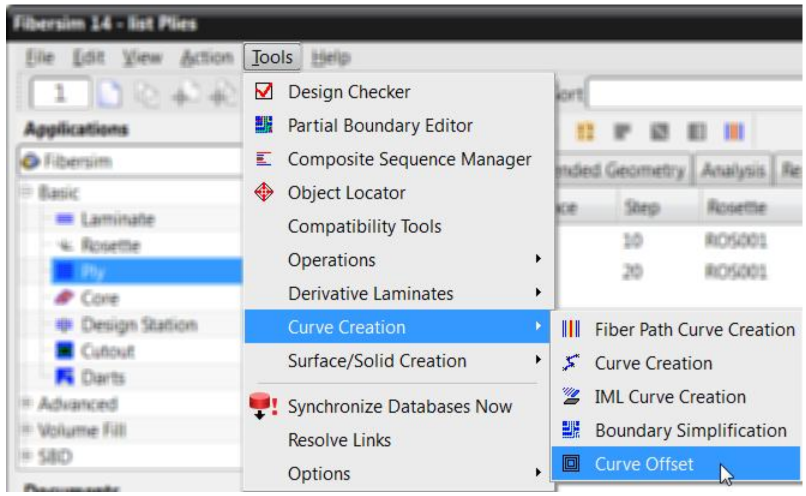


Figure 6–13

4. Select the *Constant Offset* tab.

*The **Direction Point** determines the side of the selected curves on which the offset curves are created.*

5. Click (Create New) and link the following geometry:
   - Curves to Offset: **RH Reinforcement Boundary**
   - Surface: **Tool Surface**
   - Direction Point: Indicate a point outside the RH Reinforcement Boundary as shown in Figure 6–14
   - Boundary Curves: **Net Boundary**

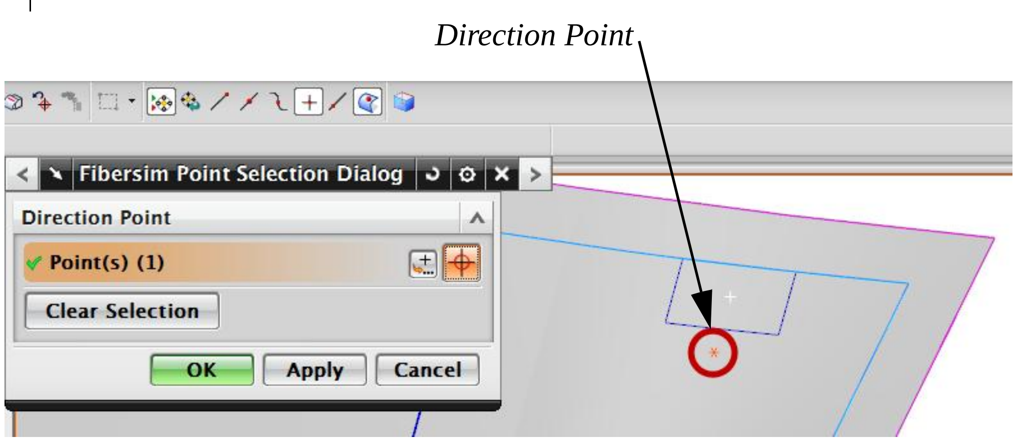


Figure 6–14

6. Enter the following parameters:
   - Number of Curves: **3**
   - Offset Value: **0.25 in (6.35 mm)**
   - Corner Type: **Fillet**

7. Click Generate in the message box as shown in Figure 6–15.

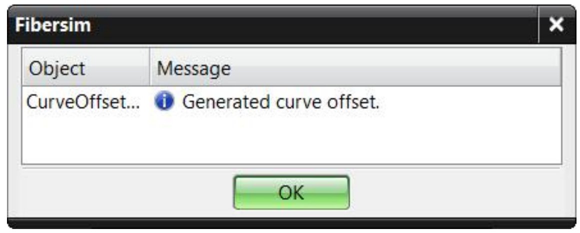


Figure 6–15

8. Click OK twice to save the curve offset.

9. Switch to the NX window to display the constant curve offset as shown in Figure 6–16.

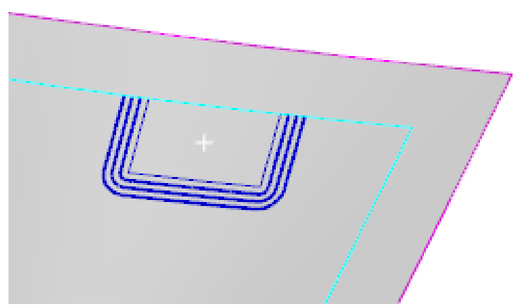


Figure 6–16

Task 5 - Change the Constant Offset Curves to be Directional.

1. Select the *Directional Offset* tab and double-click on **CurveOffset001** to modify.

The linked rosette determines the 0 and 90 degree offset directions.

2. Enter the following parameters:
   - 0 degree Offset Value: **0.25 in (6.35 mm)**
   - 90 degree Offset Value: **0.5 in (12.7 mm)**

3. Click Generate.

4. The Fibersim message window opens, prompting you that the curve offset was generated successfully as shown in Figure 6–17.

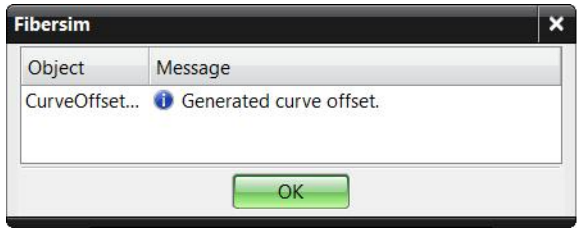
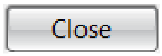


Figure 6–17

5. Click  twice.

6. Click .

7. Switch to the NX window to display the directional offset curves as shown in Figure 6–18.

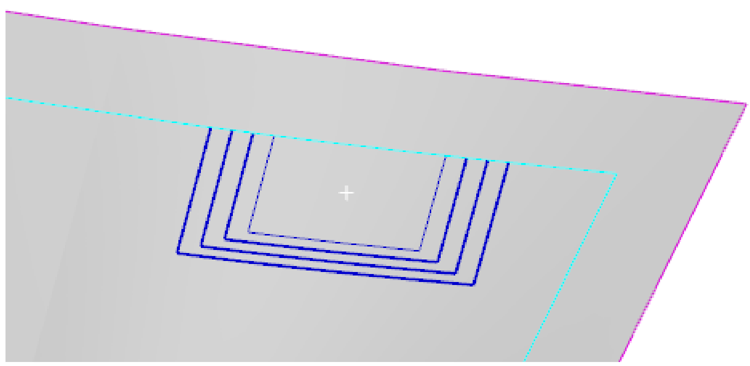


Figure 6–18

Task 6 - Create Plies from Directional Curve Offsets.

1. In the ply list, for the *Object Count*, enter [4] as shown in Figure 6–19.

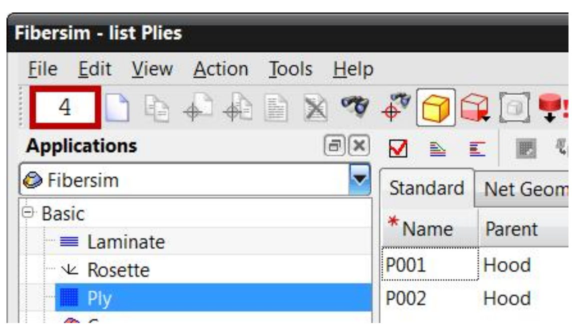


Figure 6–19

2. Click  (Create New).

3. Link the following geometry: (Make sure to disable **Point on Face** selection filter.)

   - Origin: **RH Reinforcement Origin**

4. Press <Ctrl> + <T> to switch to Table mode.

5. In the *Standard* tab, select the following for the *Specified Orientation* column:

   - P003: **0**
   - P004: **-45**
   - P005: **90**
   - P006: **45**

6. In the *Net Geometry* tab, for the *Boundary* column, link the geometry as shown in Figure 6–20.

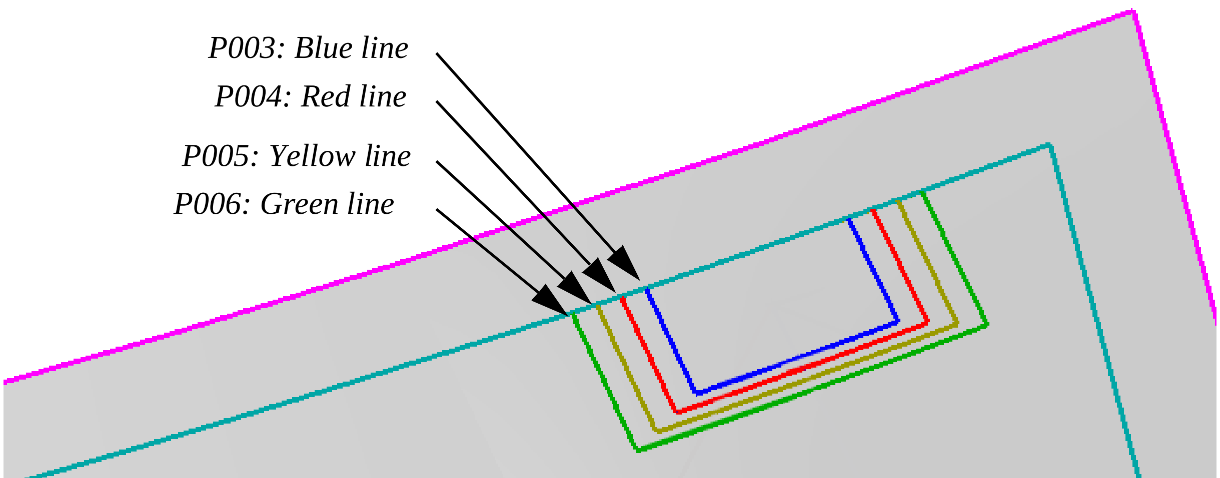
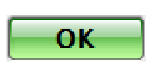


Figure 6–20

7. Click 

to save the plies.

Task 7 - Create 4 plies based on existing plies.

1. In the ply list, highlight **P003**, **P004**, **P005**, and **P006**, right-click and select **Create Based On**.

2. Press <Ctrl> + <T> to switch to Form mode.

3. Link the following geometry:
   - Origin: Indicate a point inside the **LH Reinforcement Boundary.**
   - Boundary: **LH Reinforcement Boundary**

*Because they were created based on the **RH Reinforcement** plies, the **LH Reinforcement** plies are already sequenced.*

4. Click **OK**

Task 8 - Create a Ply Drop-off.

1. In the ply list, highlight **P007**, **P008**, **P009**, and **P010**, if they are not already highlighted.

*You can also find **Ply Drop-Off** in the main menu by selecting **Tools > Operations**.*

2. In the Ply toolbar, click (Ply Drop-Off) as shown in Figure 6–21.

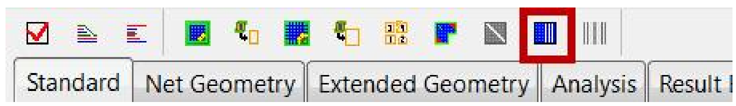


Figure 6–21

3. Specify the following parameters as shown in Figure 6–22:
   - Direction: **Outside**
   - Offset Distance: **0.25 (6.35 mm)**
   - Corner Type: **Fillet**

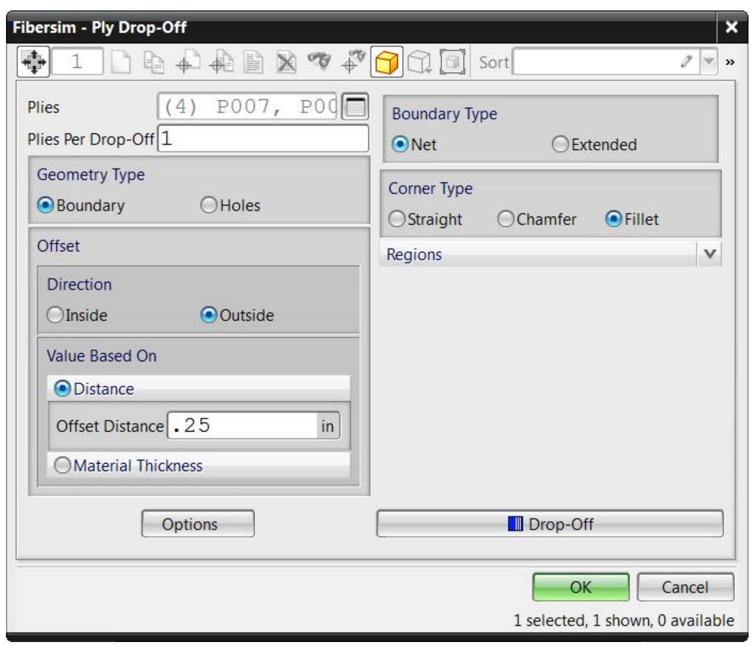


Figure 6–22

4. Click .

5. The Fibersim message window opens, prompting you that the generation of ply drop-offs is complete as shown in Figure 6–23.

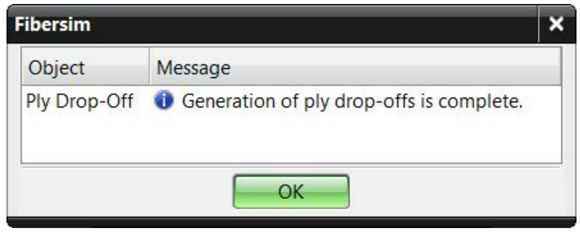


Figure 6–23

6. Click  twice.

*In Fibersim, select **P007** through to **P010** one at a time to display each ply in NX.*

Note that this Ply Drop-Off result is the same as the Constant Curve Offset.

7. Switch to the NX window to display the new plies as shown in Figure 6–24.

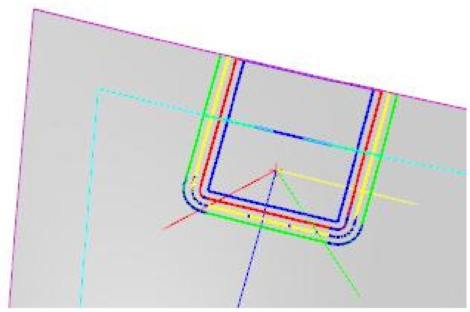


Figure 6–24

Task 9 - Show the ply drop-off using a 3D cross-section.

*You can also find **3D Cross Section** in the **Documentation** menu in the Application Tree.*

1. In the Ply toolbar, click  (3D Cross Section) as shown in Figure 6–25.

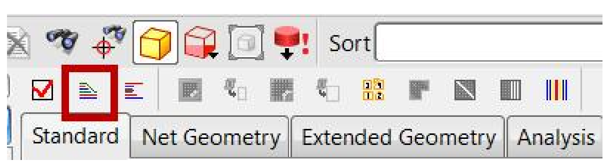


Figure 6–25

2. Select **CrossSection001** and click  (Generates the 3D cross section).

3. Zoom and orient the model to display the cross-section details as shown in Figure 6–26.

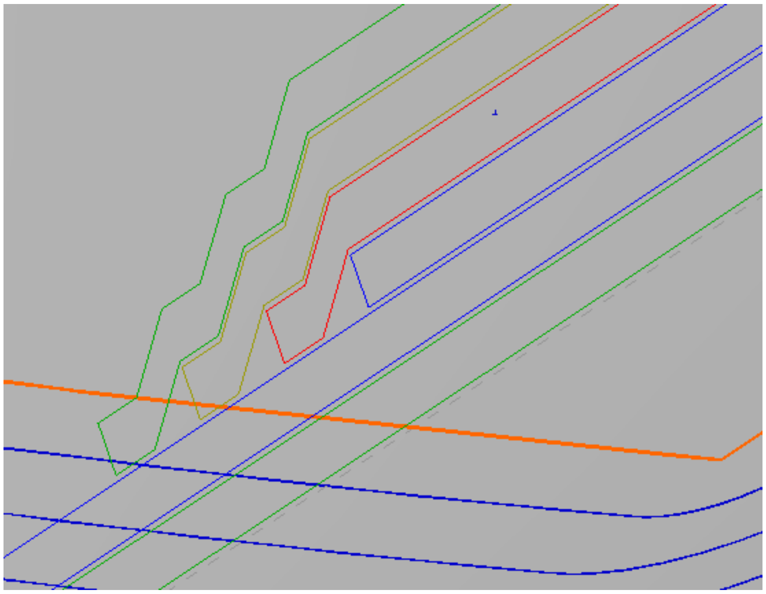
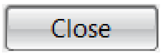


Figure 6–26

4. Click 

.

Task 10 - Boundary Simplification.

1. In Fibersim, select **Tools > Curve Creation > Boundary Simplification** as shown in Figure 6–27.

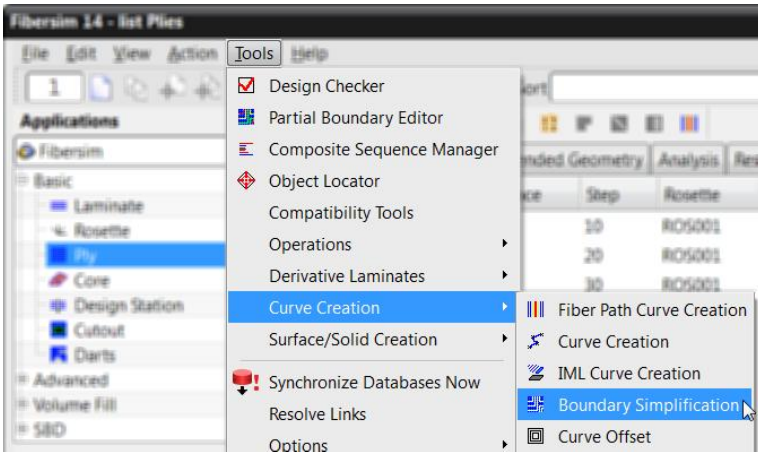


Figure 6–27

2. Click  (Create New).

3. Next to the *Components* field, click  (Link with Link Dialog).

You can also press <Ctrl> + <L> to link the selected plies.

***Internal Boundary Curves Only** will not create closed boundaries, but only the boundary curves that are not shared with the laminate boundary.*

4. Press <Ctrl> + <A> to select all of the plies, right-click, and select **Link** to link the selected plies.
5. Click OK.
6. Click Options to display the default options as shown in Figure 6–28.

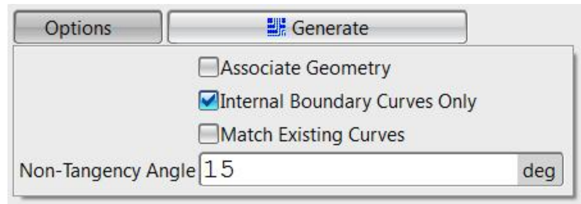


Figure 6–28

7. Click Generate.
8. Click OK in the message box prompting you that the boundary features were generated successfully.
9. Click OK to save **BNDSMP001**.
10. Click Close and close Fibersim.
11. Review the boundaries generated under **Group "FIBERSIM BOUNDARY SIMPLIFICATI**" as shown in Figure 6–29.

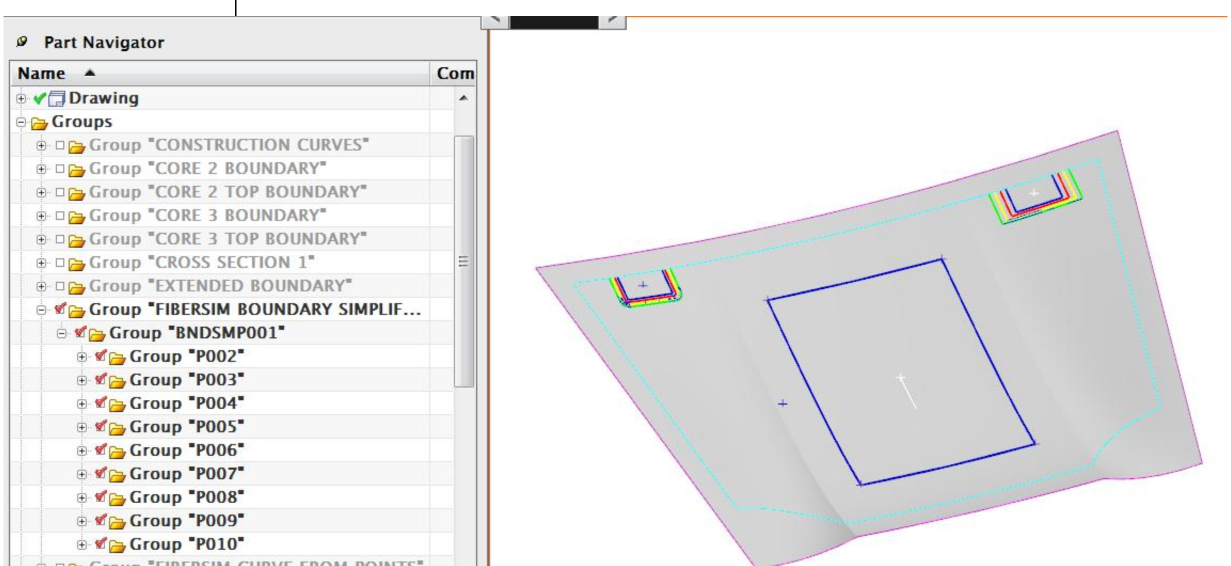


Figure 6–29

Exercise 6b Fiber Path Curve

User Guide Reference:
B.5.1 Fiber Path Curve Creation

In this exercise, you will create two Fiber Path Curves to use for Net Boundary to address deviation producibility issues. The completed model is shown in Figure 6–30.

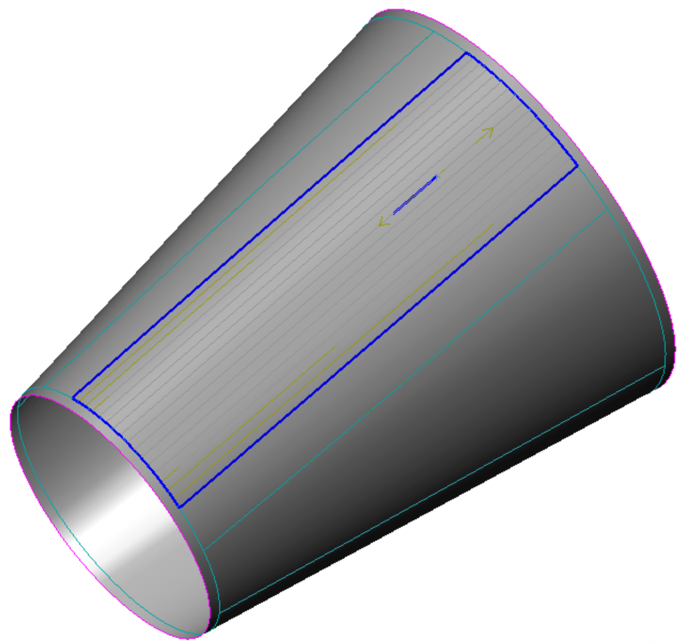


Figure 6–30

Goal

After you complete this exercise, you will be able to:

- ✓ **Display Fiber Deviation for a Ply**
- ✓ **Create a Fiber Path Curve**
- ✓ **Add Fiber Path Curves to a Net Boundary**
- ✓ **Create a Net Flat Pattern**

Estimated Time

10 min

Task 1 - Open a part.

1. Open **UTILITIES_B.prt**. The model displays as shown in Figure 6–31.

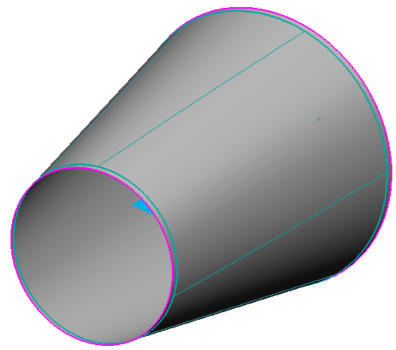
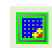


Figure 6–31

Task 2 - Set the Simulation options.

1. Press <F9> to start Fibersim.
2. In the ply list, highlight **P001** and click  (Net Producibility).
3. Double-click on **P001** to modify.
4. Select the *Simulation Options* tab and set the following simulation options as shown in Figure 6–32:
   - Results Display: **Deviation**

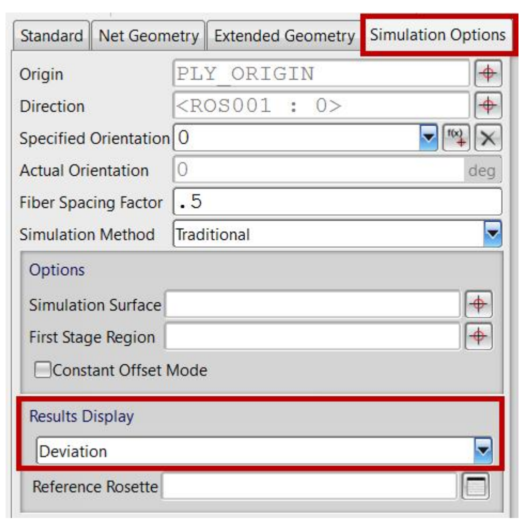
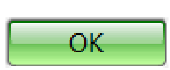


Figure 6–32

5. Click  to save **P001**.

Task 3 - Display Fiber Deviation for a Ply.

1. Switch to the NX window and display the producibility results as shown in Figure 6–33.

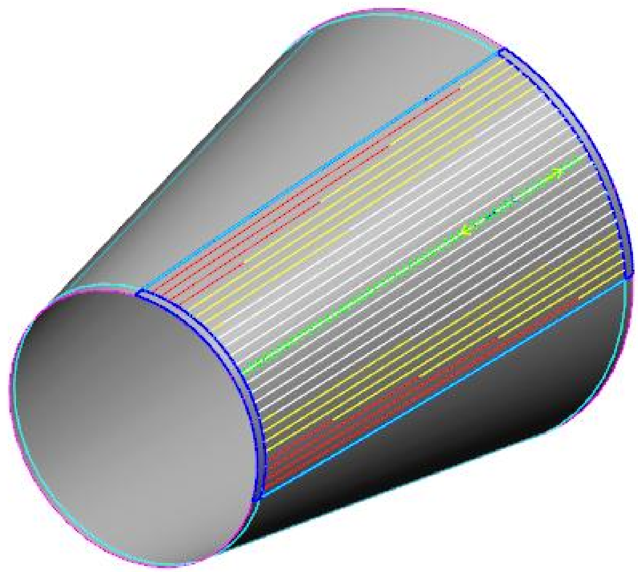


Figure 6–33

For more information on Deviation options, see the Rosettes chapter.

The Deviation results display the amount of deviation between the simulated fiber orientations and the rosette's orientation as follows:

- White fiber paths indicate deviation under 3 degrees.
- Yellow fiber paths indicate areas where the deviation is between 3 and 5 degrees.
- Red fiber paths indicate areas where the deviation in direction exceeds 5 degrees.

Task 4 - Create the First Fiber Path Curve.

*You can also find the **Fiber Path Curve Creation** utility in the main menu by selecting **Tools > Curve Creation**.*

1. In the ply list, select **P001**. In the Ply toolbar, click (Fiber Path Curve Creation) as shown in Figure 6–34.

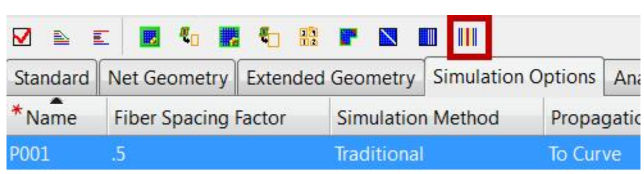


Figure 6–34

An indication must be made on the surface, not on the fiber path curve. Fibersim will not create an indication point unless it is not on the displayed curves.

2. Next to the *Fiber Path Position* field, click (Link Geometry) and indicate a point on the surface closest to the last full-yellow fiber path as shown in Figure 6–35. (Make sure to enable the **Point on Face** selection filter.

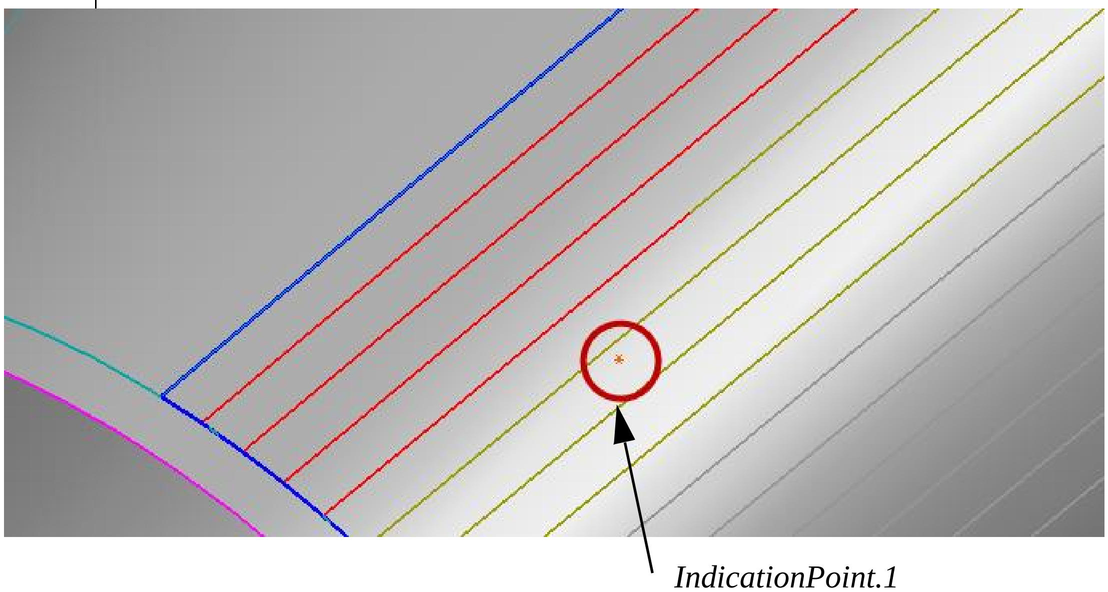


Figure 6–35

3. Click OK.

4. Click Generate.

5. Click OK in the message box prompting you that the curve was generated successfully.

Task 5 - Create a Second Fiber Path Curve.

1. Next to the *Fiber Path Position* field, click (Link Geometry).
2. Create a second fiber path curve by indicating a point on the other side of the surface nearest to the last full-yellow fiber path as shown in Figure 6–36.

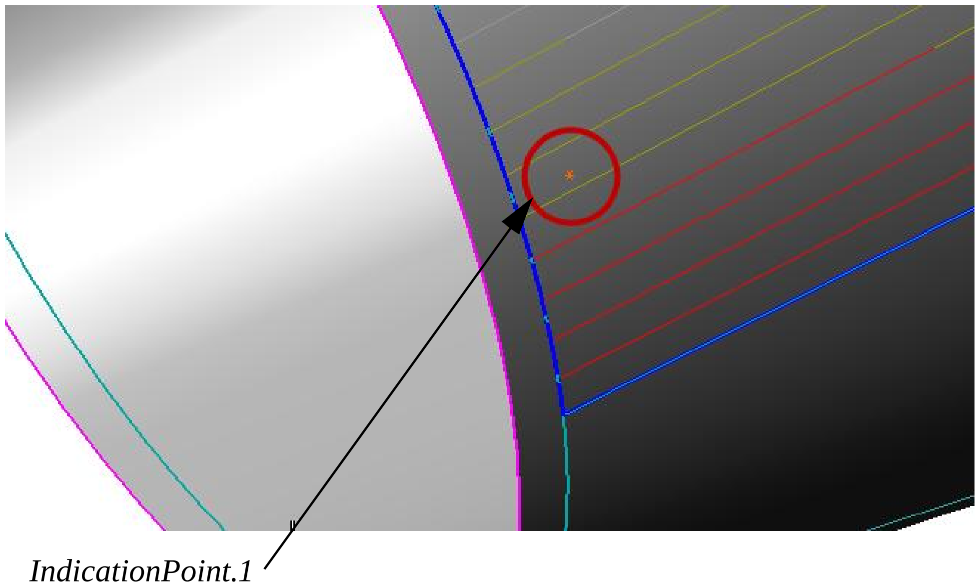


Figure 6–36

3. Click OK.
4. Click Generate.
5. Click OK in the message box prompting you that the curve was generated successfully.
6. Click OK.

Task 6 - Modify P001 to use the new Fiber Path Curves.

1. In the ply list, double-click on **P001** to modify it.
2. Select the *Standard* tab.
3. Next to the *Boundary* field, click (Link Geometry).

4. Clear the existing splines by <Shift> selecting and select the two new fiber path curves, as shown in Figure 6–37.

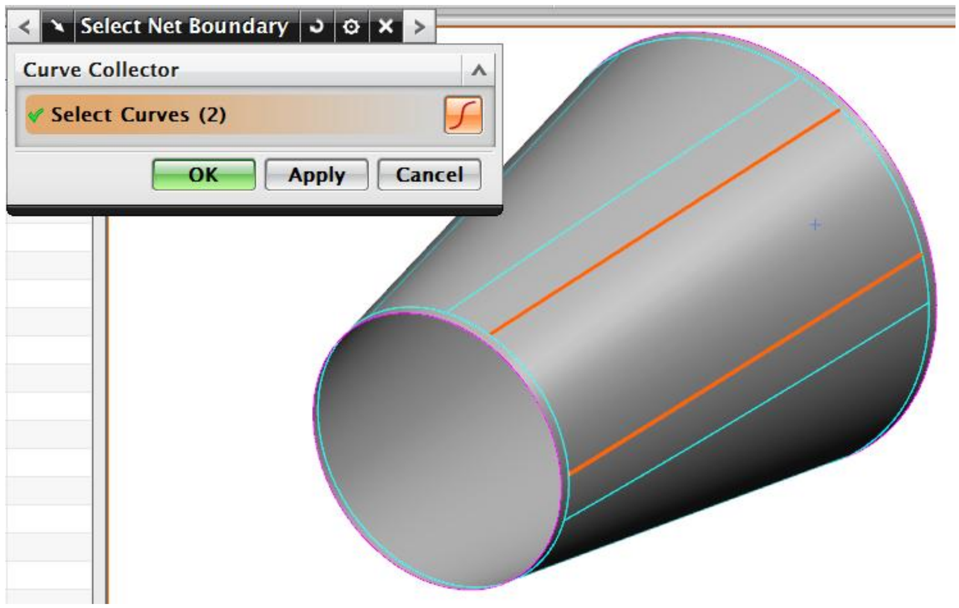


Figure 6–37

5. Click OK.

6. Select the *Net Geometry* tab.

7. Next to the *Markers* field, click (Link Geometry) and select the original material width curves as shown in Figure 6–38.

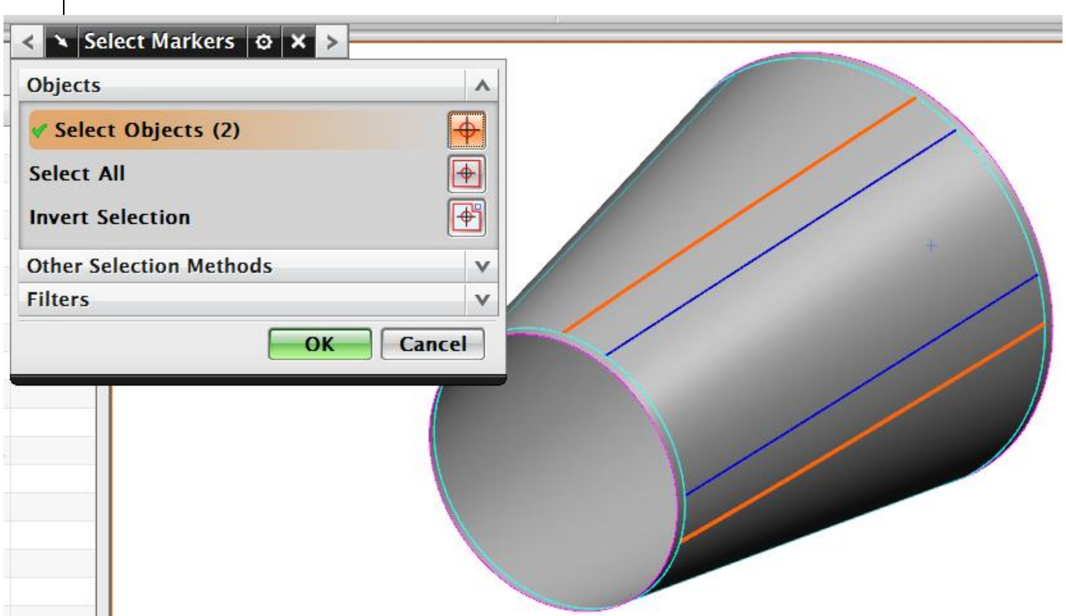
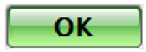


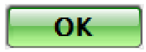
Figure 6–38

8. Click 

.

9. Click  (Net Producibility).

10. Click  (Generates the Net Flat Pattern).

11. Click 

to save **P001**.

12. Switch to the NX window to display the producibility and net pattern results. Note that the flat pattern satisfies the fiber deviation limits as shown in Figure 6–39.

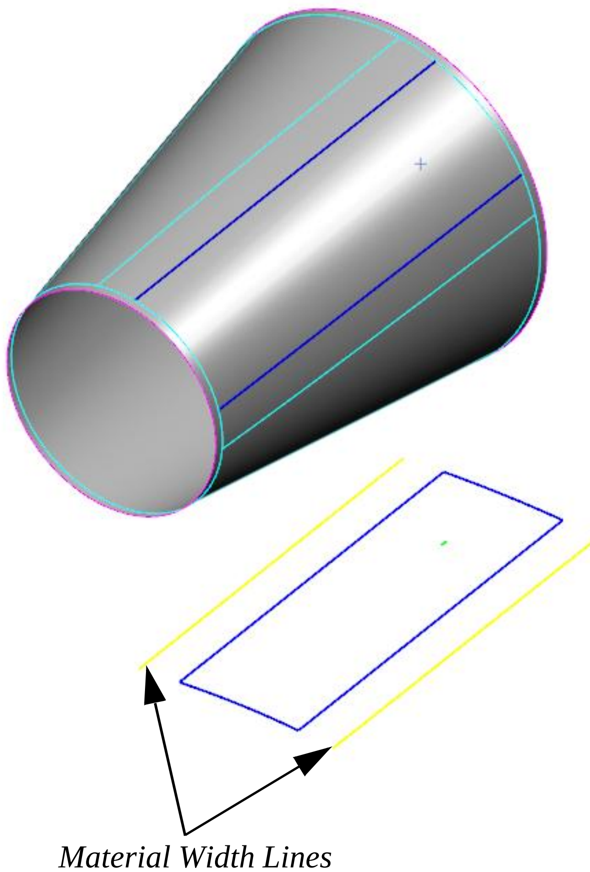


Figure 6–39

13. Close Fibersim and save the model.

Exercise 6c Symmetric Laminate

User Guide Reference:
B.4.1 Symmetric Laminate

*The **Symmetric Laminate** utility differs from the **Mirror Laminate** utility, which mirrors plies about a user-defined step value on the same laminate.*

In this exercise, you will use the **Symmetric Laminate** utility to create a second laminate dataset, by copying component data to a symmetric part. The completed model displays as shown in Figure 6–40.

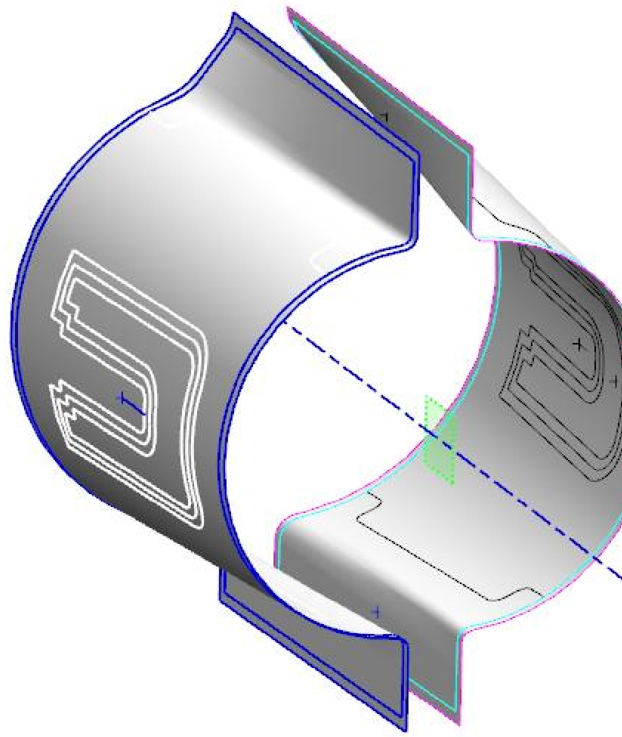


Figure 6–40

Goal

After you complete this exercise, you will be able to:

✓ **Create a Symmetric Laminate**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **UTILITIES_C.prt**. The model displays as shown in Figure 6–41.

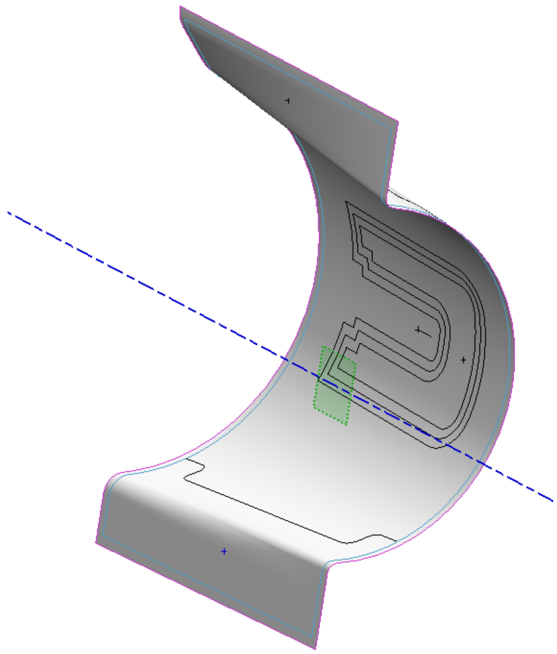


Figure 6–41

Task 2 - [NX8.5] Mirror CAD features about a symmetry plane.

Task 2 is different for each CAD system. Find the appropriate Task 2 and follow instructions.

1. In NX, click  (Extract Geometry) as shown in Figure 6–42.

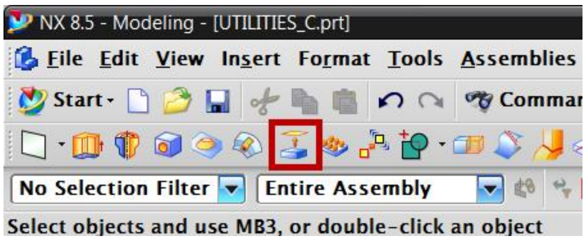


Figure 6–42

2. Select **Mirror Body** from the *Extract Geometry Type* pull-down list.

3. Select the body and the mirror plane as shown in Figure 6–43.

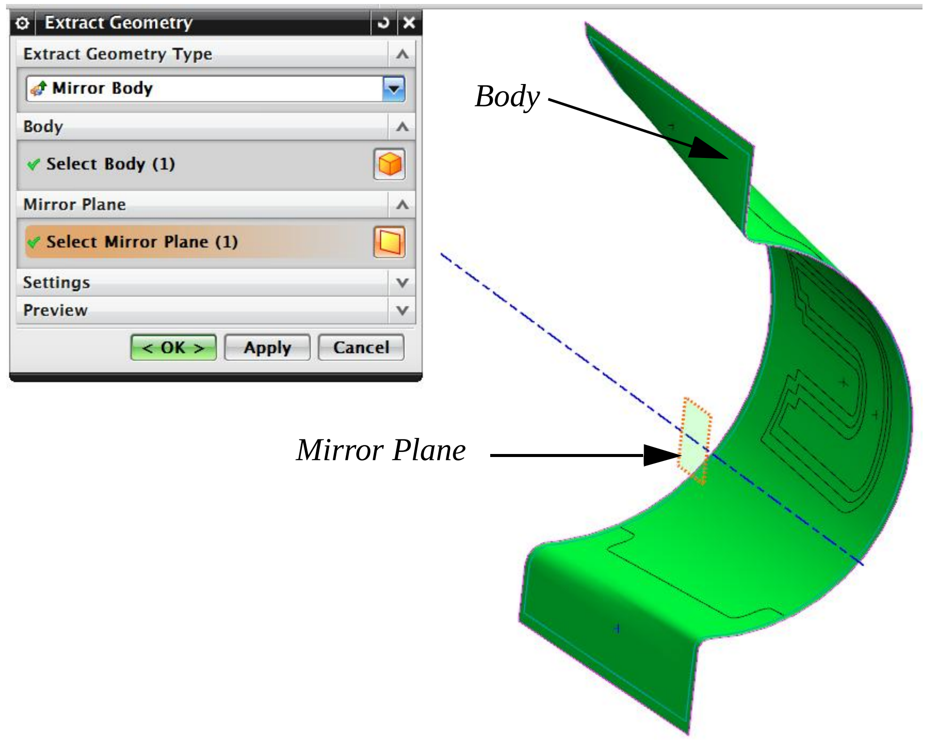


Figure 6–43

4. Click OK.

5. Select **Insert > Curve from Curves > Mirror** as shown in Figure 6–44.

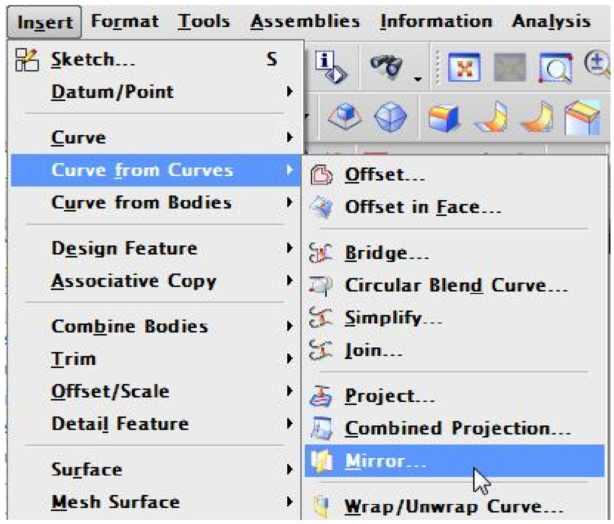


Figure 6–44

6. Select **Net Boundary**, **Extended Boundary**, and **Zero Direction** for the curves and the mirror plane as shown in Figure 6–45.

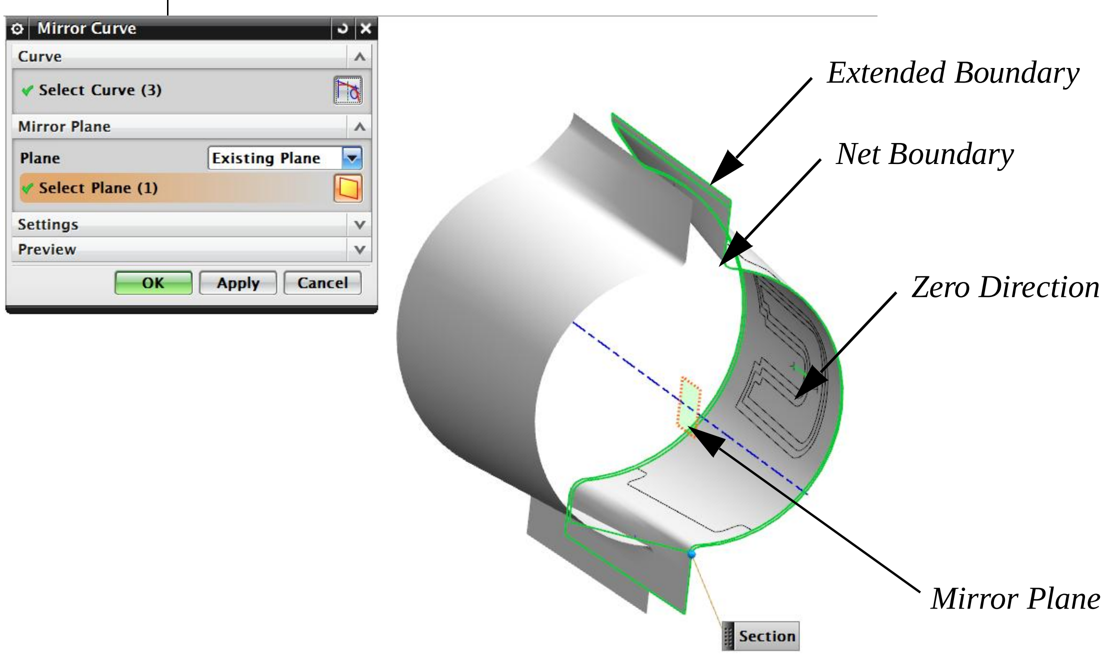


Figure 6–45

7. Click **OK** and the model updates as shown in Figure 6–46.

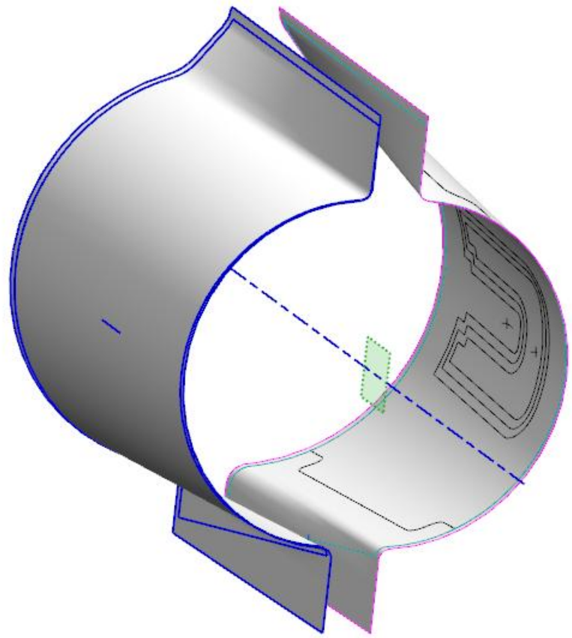


Figure 6–46

8. Select **Insert > Datum/Point > Point.**

9. Create a point at the end of the mirrored Zero Direction Curve as shown in Figure 6–47.

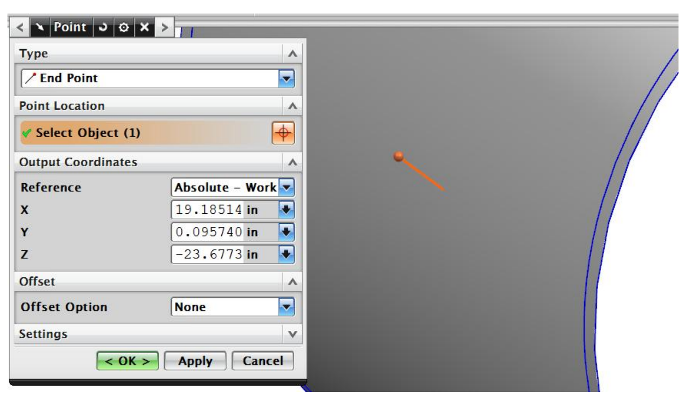


Figure 6–47

Task 2 - [NX9/10] Mirror CAD features about a symmetry plane.

1. Select **Mirror Geometry** from the *Home tab* > *Feature group* > *More* as shown in Figure 6–48.

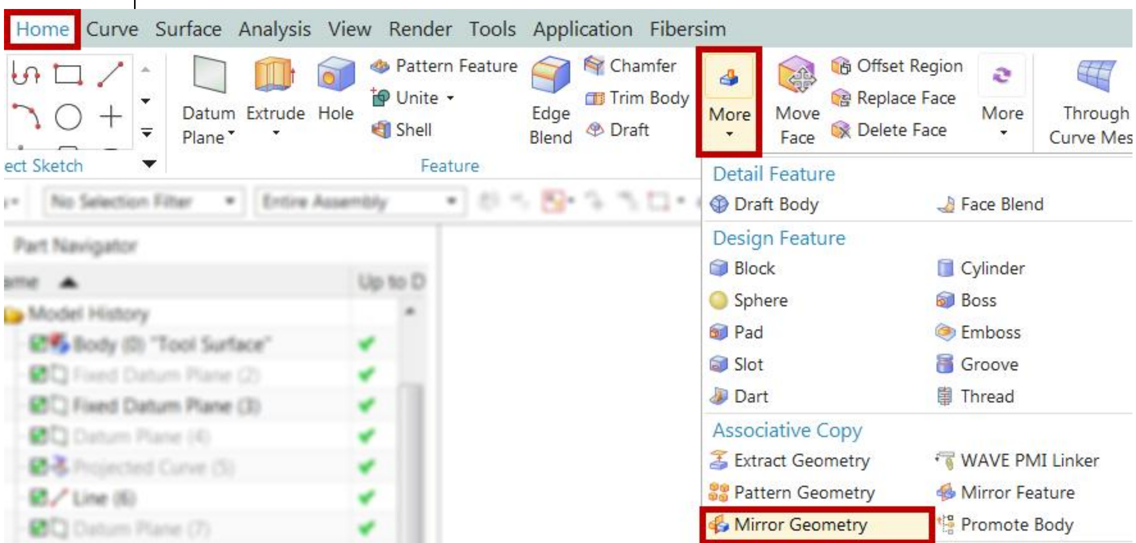


Figure 6–48

2. Select the following geometry as shown in Figure 6–49:

   - Geometry to Mirror: Tool Surface, Net Boundary, Extended Boundary, Rosette Origin, Zero Direction
   - Mirror Plane: Sym Plane

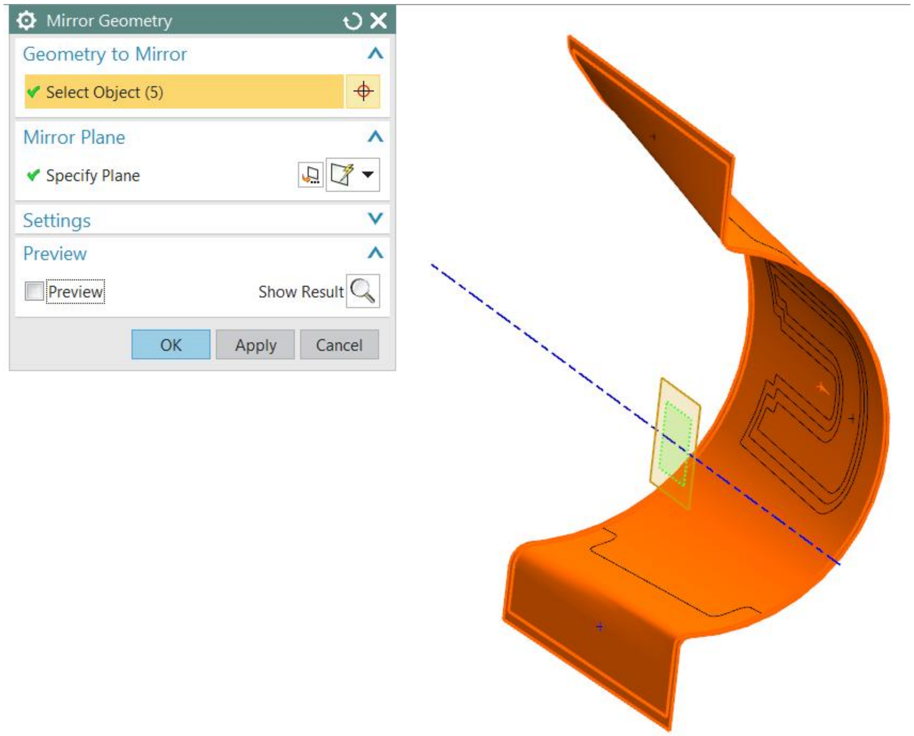


Figure 6–49

3. Click .

Task 3 - Create Symmetric Laminate and Rosette geometry.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Laminate** under **Basic**.
3. Click  (Create New).
4. In *Name* field, enter [Sym Lam].

5. Link the following geometry as shown in Figure 6–50:
   - Layup Surface: select the mirrored tool surface
   - Net Boundary: select the mirrored net boundary
   - Extended Boundary: select the mirrored extended boundary

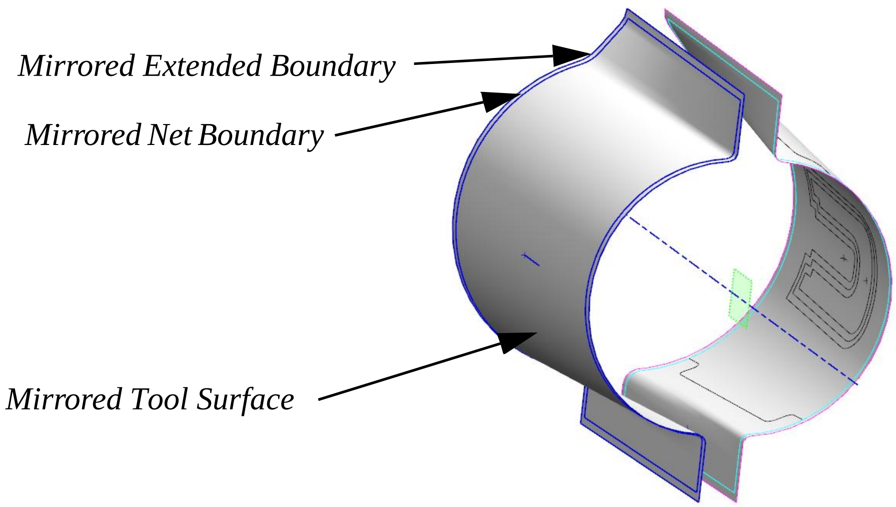


Figure 6–50

6. Click **OK** to save **Sym Lam**.
7. In the *Application Tree*, select **Rosette** under **Basic**.
8. Click (Create New) to create **ROS002**.

9. Rename **ROS002** as [Sym Ros].

10. Link the following geometry as shown in Figure 6–51:

- Surface: select the mirrored Surface
- Origin: select the mirrored Origin
- Direction: select the mirrored Zero Direction

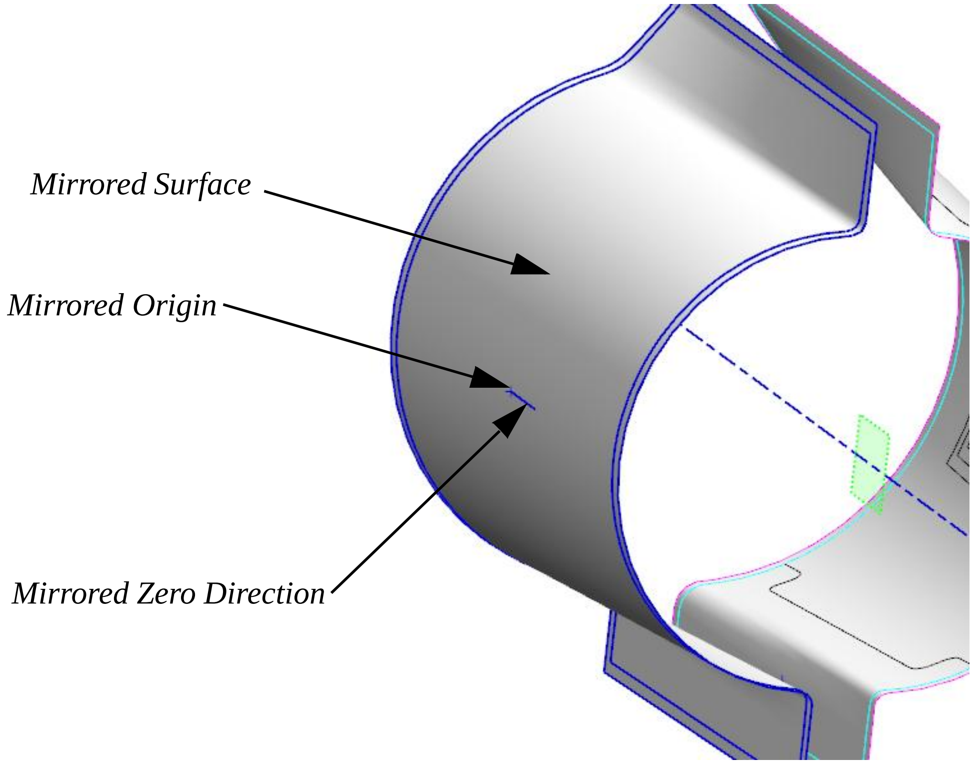
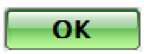


Figure 6–51

An exact symmetry of ply orientations requires an opposite handed (Left-Hand) rosette.

11. For Hand Direction, select **Left** as shown in Figure 6–52.

Figure 6–52

12. Click  to save **Sym Ros**.

Task 4 - Create a Symmetric Laminate.

1. In Fibersim, select **Tools > Derivative Laminates > Symmetric Laminate** as shown in Figure 6–53.

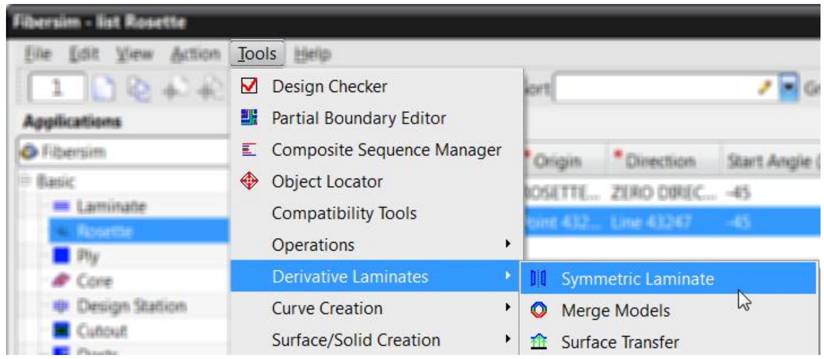


Figure 6–53

2. In the *Symmetric Laminate* dialog box, click  (Create New).

3. Specify the following parameters to create the symmetric laminate:

   - Source Laminate: **Cowl**
   - Source Rosette: **ROS001**
   - Symmetry Plane: **Sym Plane**
   - Symmetric Laminate: **Sym Lam**
   - Symmetric Rosette: **Sym Ros**

4. The updated Symmetric Laminate form displays as shown in Figure 6–54.

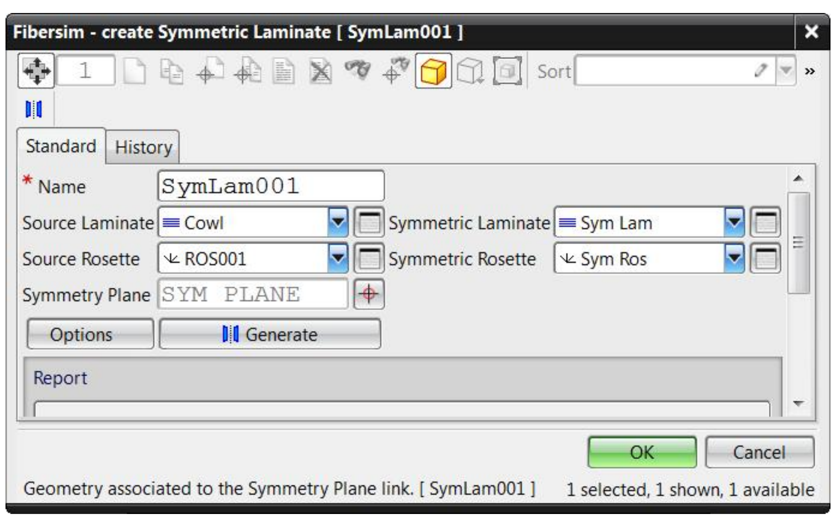


Figure 6–54

5. Click **Generate** to generate the symmetric laminate.

6. In the message box prompting you that 23 components have generated successfully as shown in Figure 6–55.

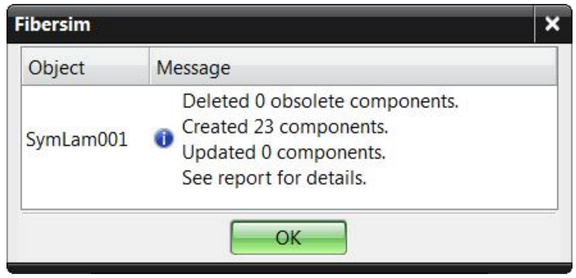


Figure 6–55

7. Click **OK**.

8. Note the Report created on the Symmetric Laminate's Standard form as shown in Figure 6–56.

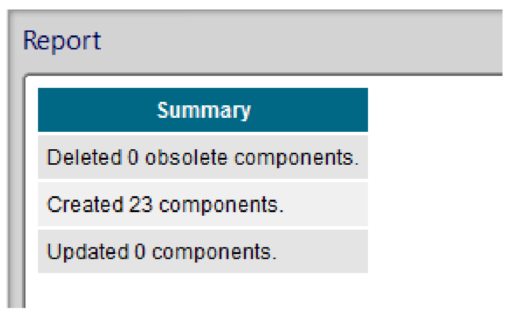


Figure 6–56

9. Click **OK** to complete the Symmetric Laminate creation.

10. Click **Close** to close the window.

Task 5 - Group plies by Parent.

1. In the *Application Tree*, select **Ply** under **Basic.**
2. In the Group drop-down list, select **Parent** to display the plies grouped by their parents as shown in Figure 6–57.

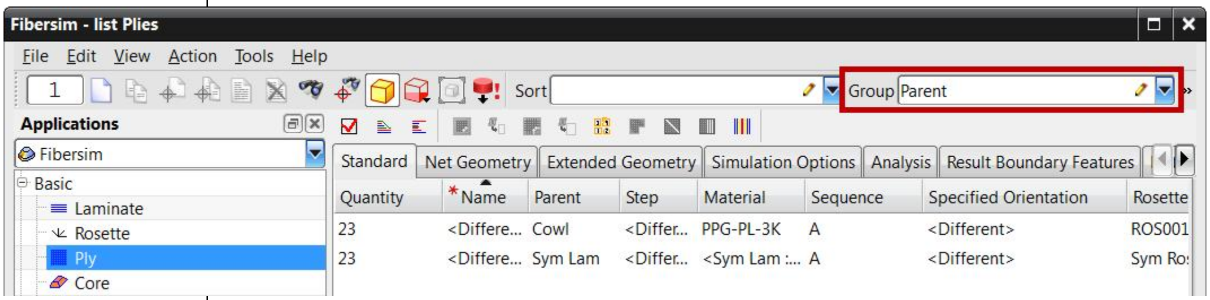


Figure 6–57

3. In the Parent ply list, highlight **Sym Lam** and **Cowl**, one at a time, to highlight them in the NX window as shown in Figure 6–58.

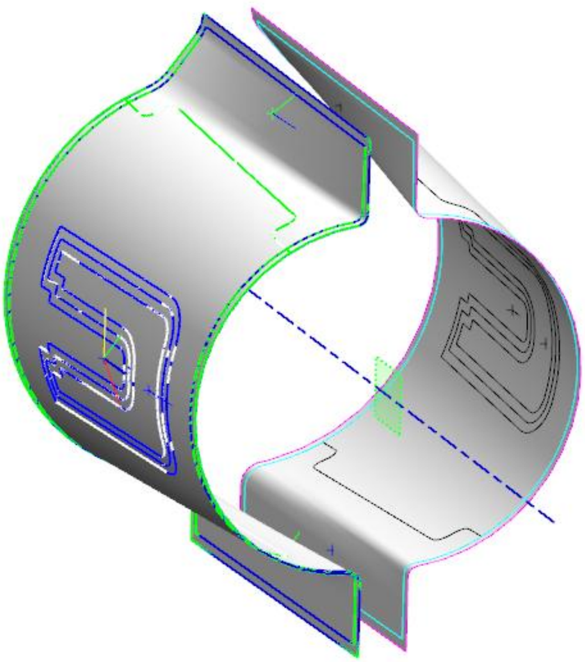


Figure 6–58

4. Close Fibersim and save the model.

Project 2

Monolithic Panel

This chapter includes:

✓**Exercise P2: Monolithic Panel**

Exercise P2 Monolithic Panel

User Guide Reference:
2.8 Plies
B.5.2 Curve Creation
B.5.5 Curve Offset

In this project, you will use ply based design to complete a layup scenario. The completed model displays as shown in Figure P2–1.

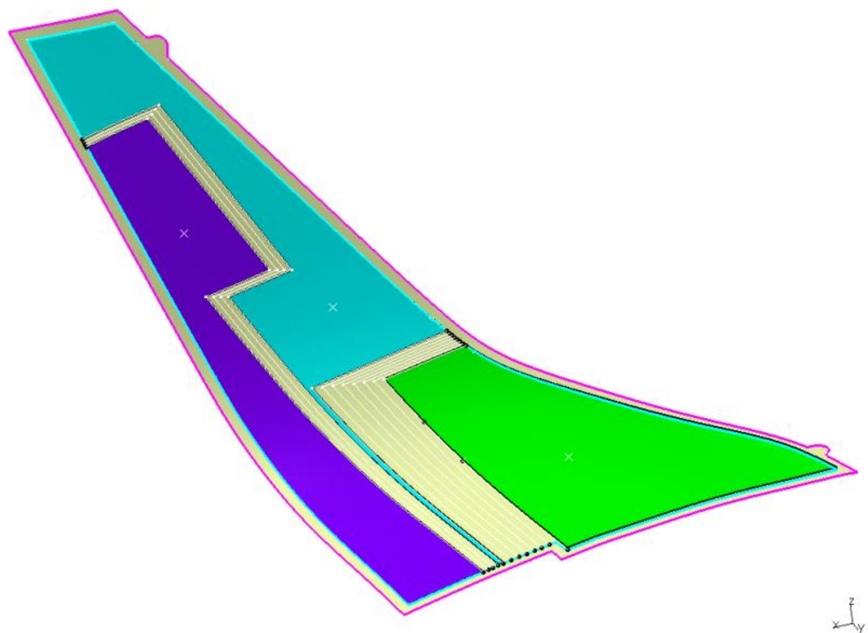


Figure P2–1

Goal

This project will test your knowledge on how to:

- ✓ **Create the required NX geometry**
- ✓ **Create ply objects**
- ✓ **Create a 3D cross-section and confirm layup**

Given

The previously created geometry and design details are:

- ✓ **Net and Extended Boundaries have been created**
- ✓ **Rosette geometry has been created**
- ✓ **Planes that define the locations of the substructure have been created**
- ✓ **The substructure is the basis for the ply boundaries**

Estimated Time

1 hr 30 min

Task 1 - Open a part.

1. Open **PROJECT2_MONOLITHIC.prt**. The model displays as shown in Figure P2–2.

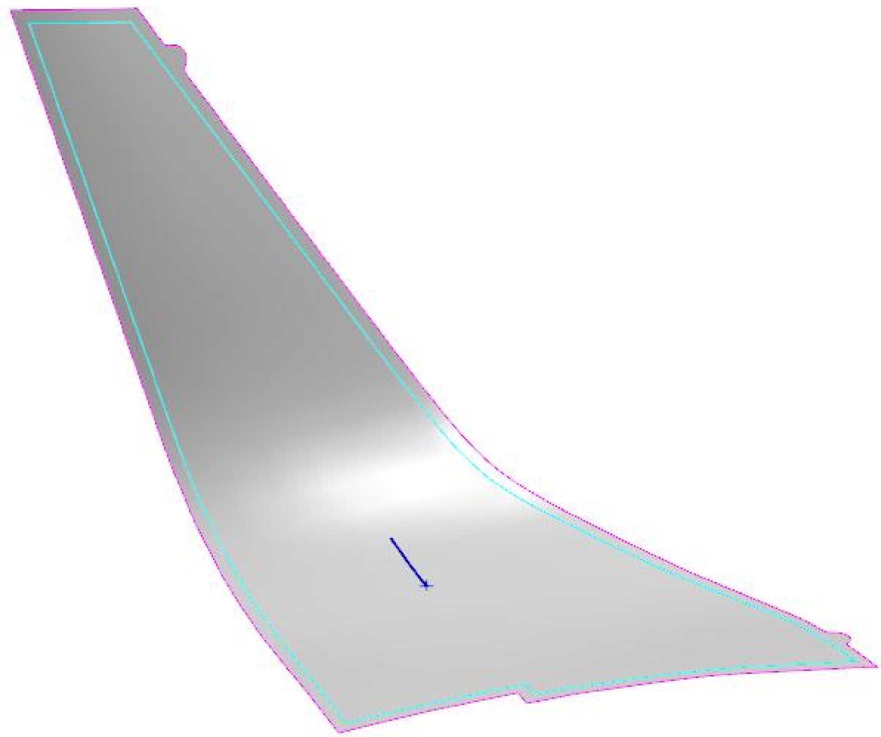


Figure P2–2

Task 2 - Review Fibersim geometry.

1. The Winglet Tool Surface, Manufacturing (Extended) Boundary, Design (Net) Boundary, Rosette Origin, and Zero Direction are displayed as shown in Figure P2–3.

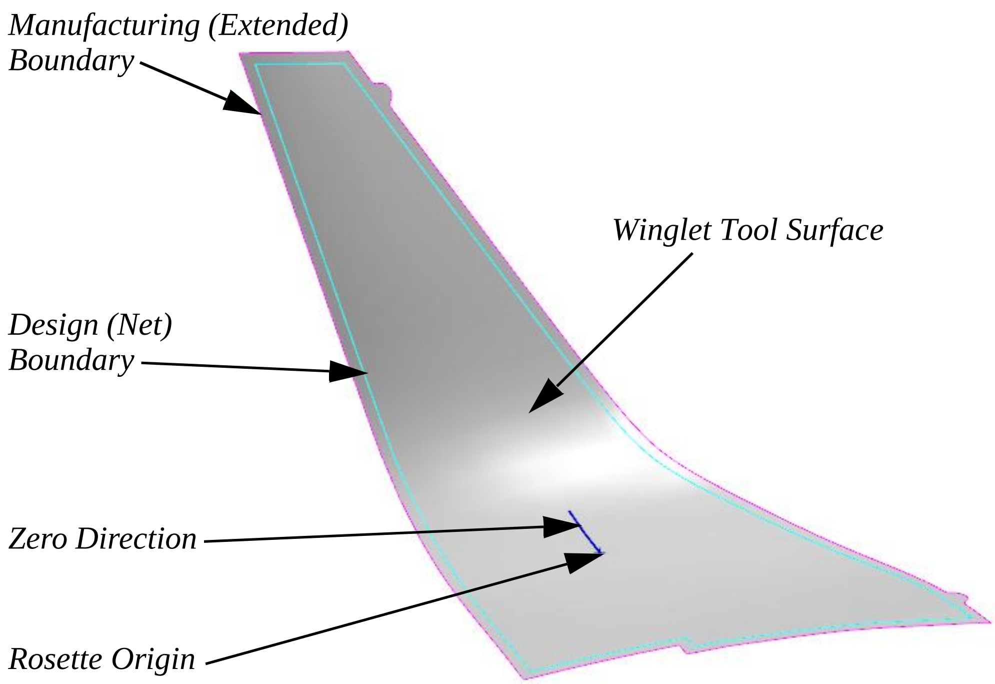


Figure P2–3

2. The Substructure geometry is shown in Figure P2–4.

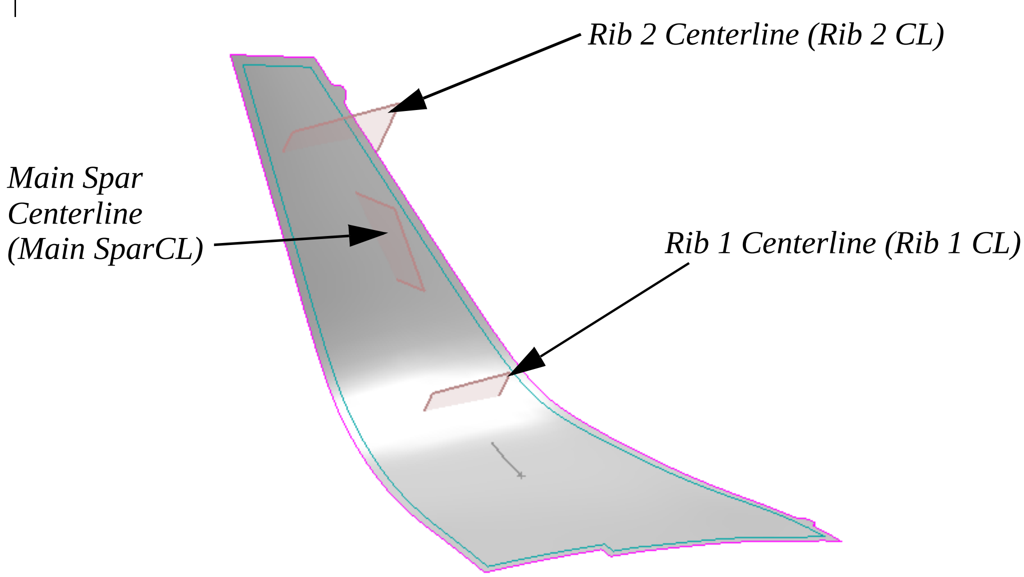


Figure P2–4

Task 3 - Create the required CAD geometry.

1. Create the required CAD geometry for the given plies as shown in Figure P2–5.

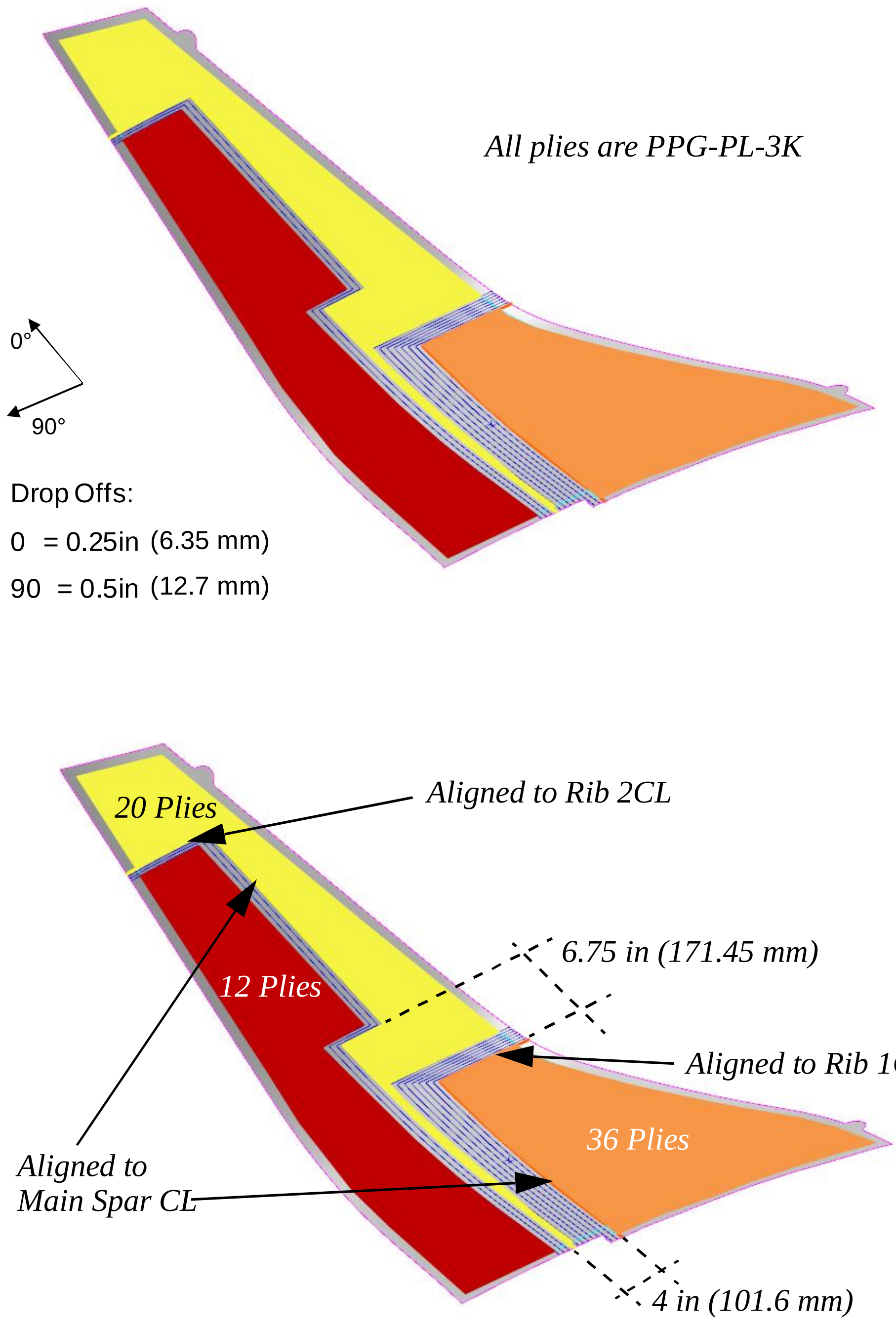
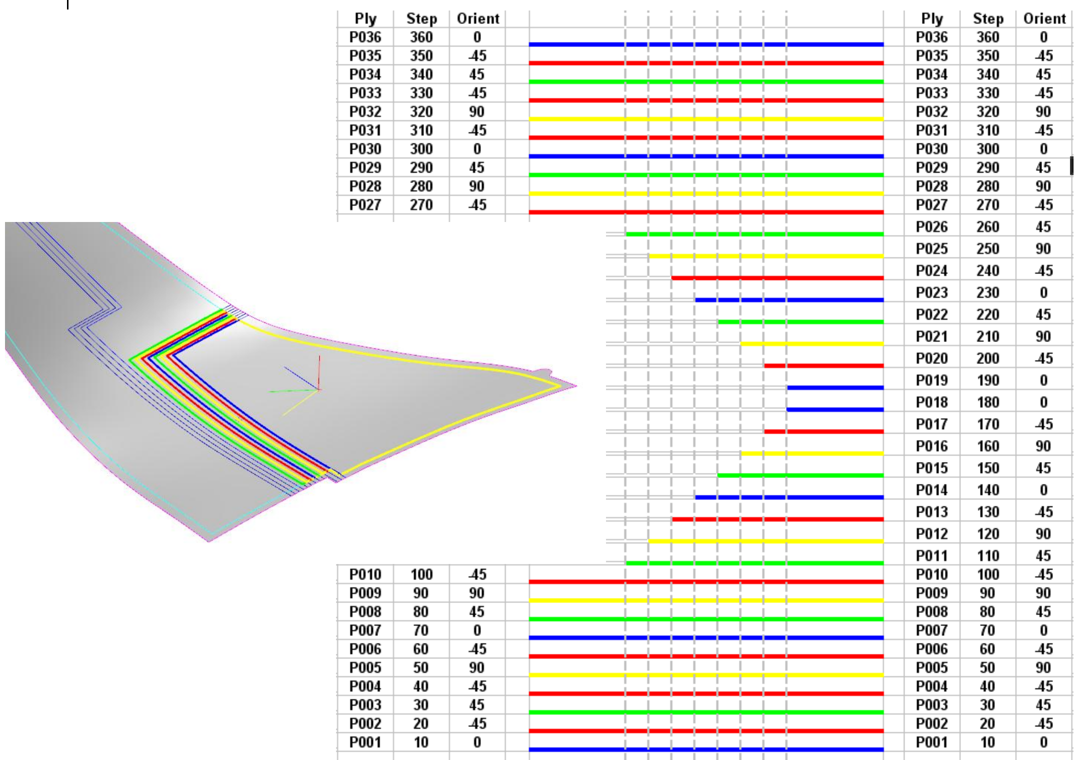


Figure P2–5

Task 4 - Create ply objects.

1. Create plies, using given quantity, ply drop-off values and orientations as shown in Figure P1–7 and Figure P2–7.

Ply Requirements (36 --> 20 Ply Drop)

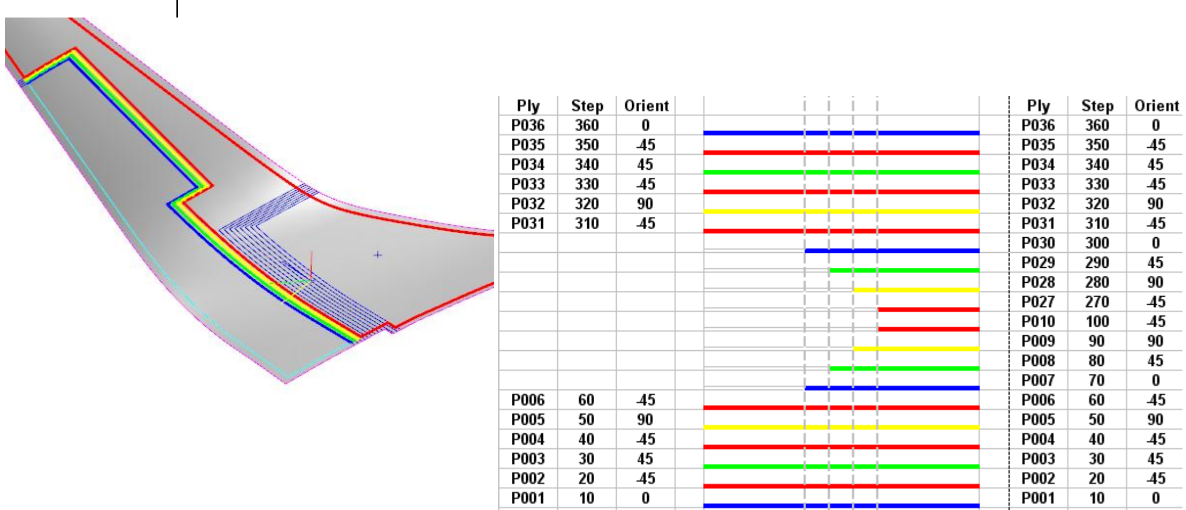


Ply	Step	Orient
P036	360	0
P035	350	-45
P034	340	45
P033	330	-45
P032	320	90
P031	310	-45
P030	300	0
P029	290	45
P028	280	90
P027	270	-45
P010	100	-45
P009	90	90
P008	80	45
P007	70	0
P006	60	-45
P005	50	90
P004	40	-45
P003	30	45
P002	20	-45
P001	10	0

Ply	Step	Orient
P036	360	0
P035	350	-45
P034	340	45
P033	330	-45
P032	320	90
P031	310	-45
P030	300	0
P029	290	45
P028	280	90
P027	270	-45
P026	260	45
P025	250	90
P024	240	-45
P023	230	0
P022	220	45
P021	210	90
P020	200	-45
P019	190	0
P018	180	0
P017	170	-45
P016	160	90
P015	150	45
P014	140	0
P013	130	-45
P012	120	90
P011	110	45
P010	100	-45
P009	90	90
P008	80	45
P007	70	0
P006	60	-45
P005	50	90
P004	40	-45
P003	30	45
P002	20	-45
P001	10	0

Figure P2–6

Ply Requirements (20 --> 12 Ply Drop)



Ply	Step	Orient
P036	360	0
P035	350	-45
P034	340	45
P033	330	-45
P032	320	90
P031	310	-45
P006	60	-45
P005	50	90
P004	40	-45
P003	30	45
P002	20	-45
P001	10	0

Ply	Step	Orient
P036	360	0
P035	350	-45
P034	340	45
P033	330	-45
P032	320	90
P031	310	-45
P030	300	0
P029	290	45
P028	280	90
P027	270	-45
P010	100	-45
P009	90	90
P008	80	45
P007	70	0
P006	60	-45
P005	50	90
P004	40	-45
P003	30	45
P002	20	-45
P001	10	0

Figure P2–7

Task 5 - Cross-section.

1. Use the Curve Creation Utility to create a curve for a cross-section. Review the 3D Cross Section and confirm that the layup matches the design requirements as shown in Figure P2–8.

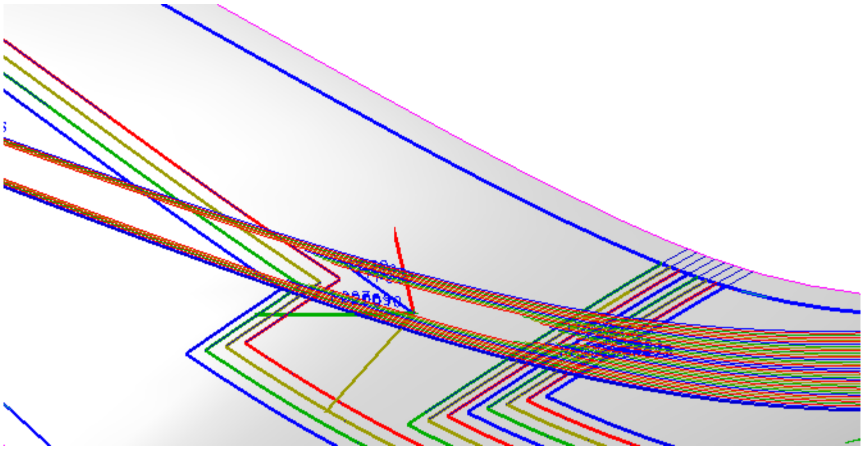


Figure P2–8

Chapter 7

Core

This chapter introduces:

- ✓**Exercise 7a: Core Types**
- ✓**Exercise 7b: Modeled Core with Overcore and IML Laminates**
- ✓**Exercise 7c: Core Panel Design from a Solid (Optional)**

Exercise 7a

Core Types

User Guide Reference:
2.9 Cores
5.2 3D Cross Section

In this exercise, you will create various types of cores within Fibersim Basic. (Note that designing the overcore plies/layers has not yet been discussed.) The completed model is shown in Figure 7–1.

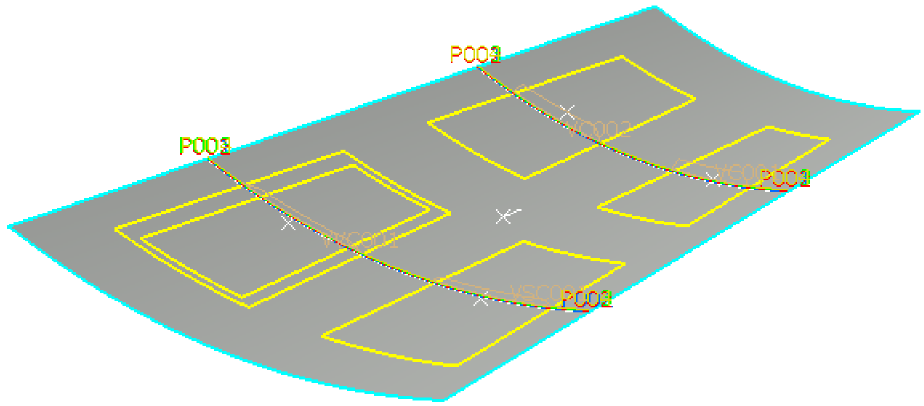


Figure 7–1

After you complete this exercise, you will be able to:

- ✓ **Create a Virtual Step Core object**
- ✓ **Create a Virtual Variable Core object**
- ✓ **Create a Virtual Core object**
- ✓ **Observe the differences between the different types of core using cross-sections**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **CORE_A.prt**. The model displays as shown in Figure 7–2.

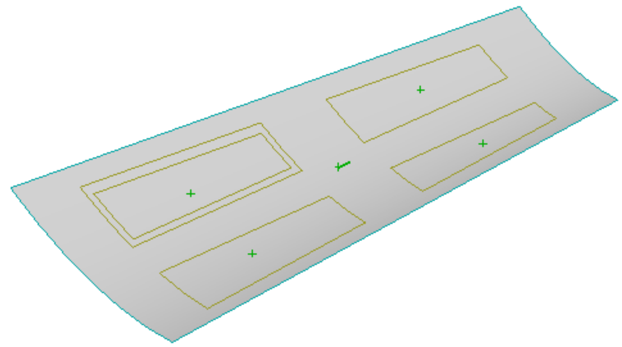


Figure 7–2

2. Press <F9> to start Fibersim.

3. Review the Fibersim objects in the model. Note that the model already contains a Laminate, a Rosette, and four full-body plies as shown in Figure 7–3. The plies are made of the PPG-PL-3K material and are oriented to the 0, -45, 45, and 90 deg directions.

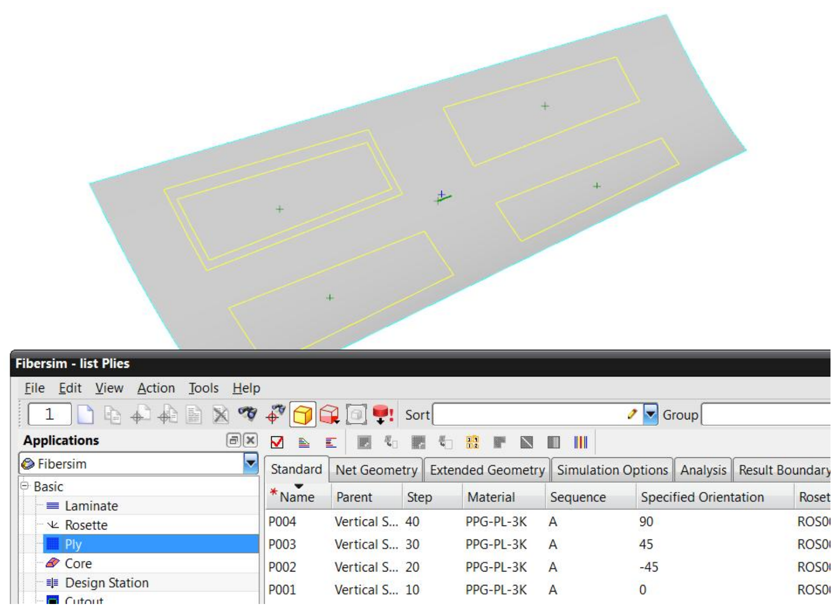


Figure 7–3

Task 2 - Create a Virtual Step Core.

Virtual Step Core is the most commonly used type of core, with the same bevel angle on all sides and a vertical step at the base.

1. In the *Application Tree*, select **Core** under **Basic**.

2. Click (Create New). Expand the drop-down list and click (Virtual Step Core). The Core definition form is displayed as shown in Figure 7–4.

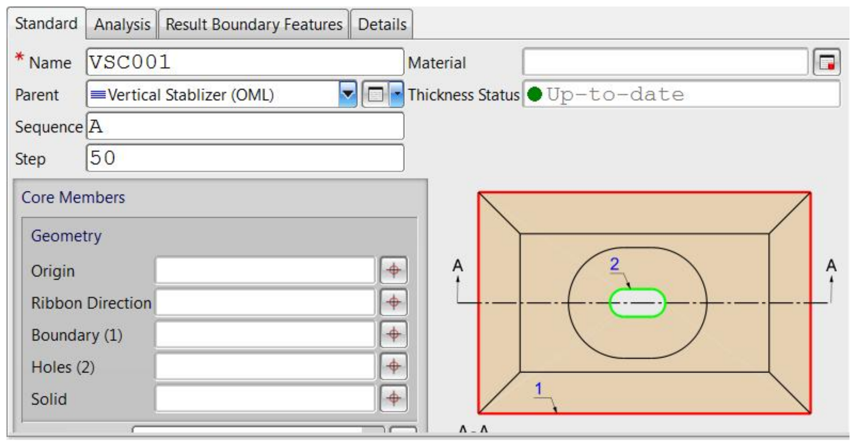
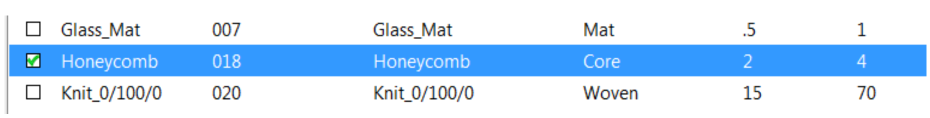


Figure 7–4

3. Next to the *Material* field, click (Link with Database Link Dialog).

4. Select the checkbox for **Honeycomb** material as shown in Figure 7–5.



☐	Glass_Mat	007	Glass_Mat	Mat	.5	1
☑	Honeycomb	018	Honeycomb	Core	2	4
☐	Knit_0/100/0	020	Knit_0/100/0	Woven	15	70

Figure 7–5

5. Click **OK**.

6. In the *Core Members > Geometry* area, next to the *Origin* field, click (Link Geometry).

7. Make sure that the **Existing Point** selection filter is enabled.

8. Indicate a point in the middle of the 1st Core as shown in Figure 7–6.

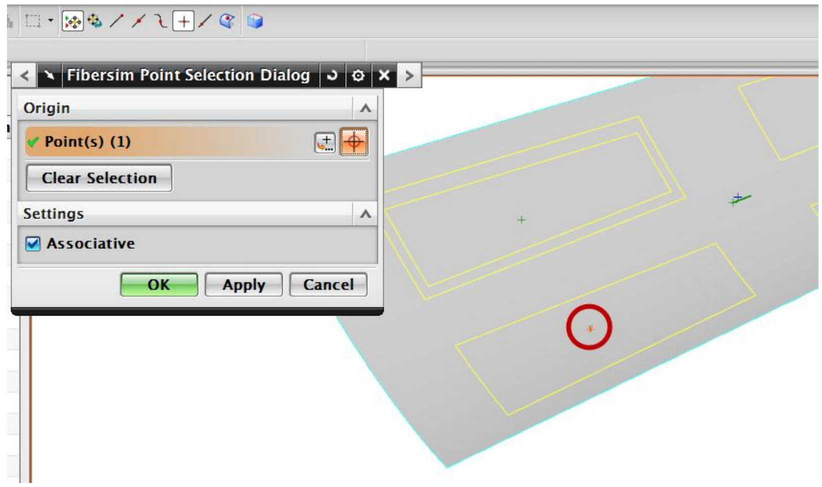


Figure 7–6

9. Click .

10. In the *Geometry* area, next to the *Boundary (1)* field, click  (Link Geometry).

If required for a given design, a curve inside the core boundary can be used to define the ribbon direction of the core.

11. Select the boundary contour of the 1st Core as shown in Figure 7–7.

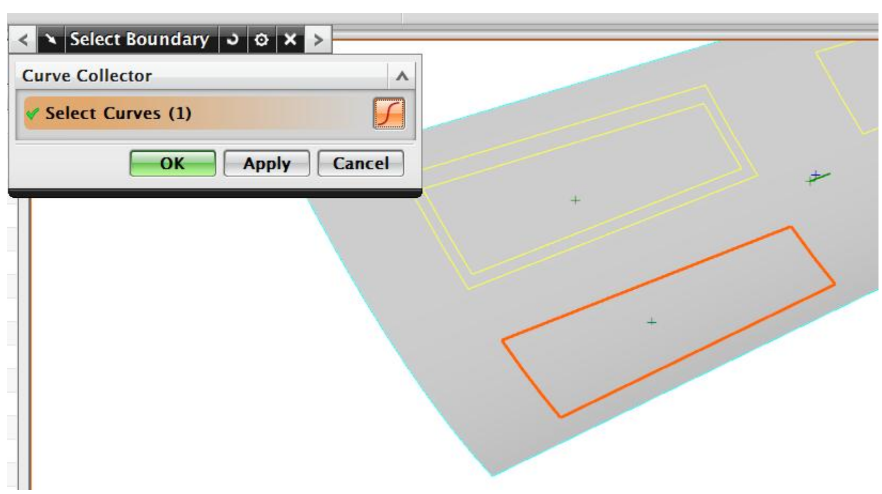


Figure 7–7

12. Click .

13. Enter the following parameters as shown in Figure 7–8.

- Thickness: **1.575 in (40 mm)**
- Bevel Angle: **30**
- Step Height: **0.095 in (2.413 mm)**

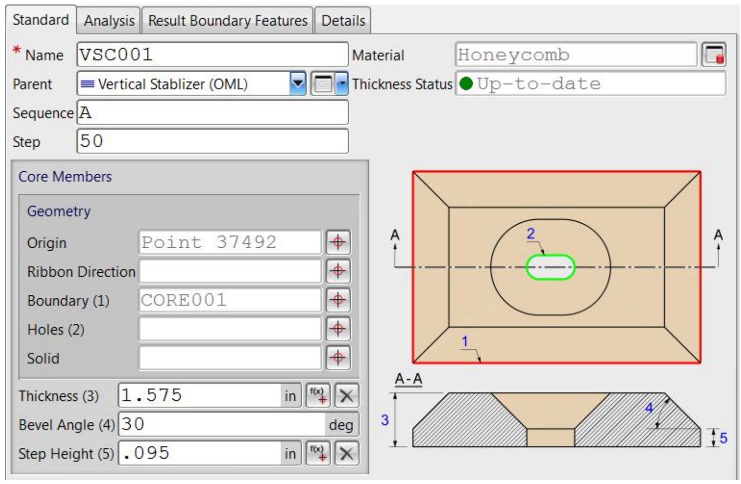


Figure 7–8

14. Click  to complete the Core.

Task 3 - Create a Virtual Variable Core.

Virtual Variable Core requires you to define both the Base Boundary and Top Boundary of the core. The Core bevel angles might be different on all sides and are defined by differences in the geometric shape of the Base Boundary and Top Boundary combined with the core height.

1. In the *Application Tree*, select **Core** under **Basic**.

2. Click  (Create New). Expand the drop-down list and click  (Virtual Variable Core). The Virtual Variable Core definition form is displayed as shown in Figure 7–9.

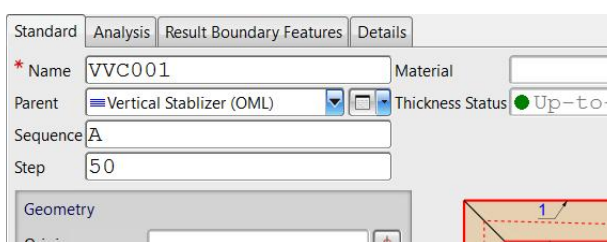


Figure 7–9

3. Next to the *Material* field, click (Link with Database Link Dialog).
4. Select the checkbox for the **Honeycomb** material.
5. Click OK.
6. In the *Geometry* area, next to the *Origin* field, click (Link Geometry).
7. Indicate a point in the middle of the 2nd Core as shown in Figure 7–10.

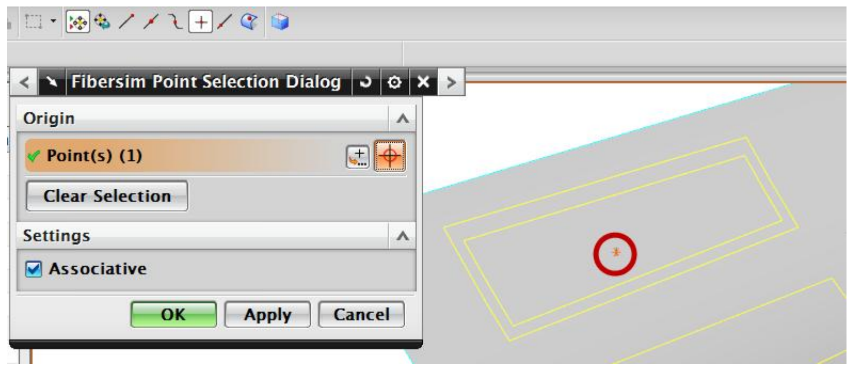


Figure 7–10

8. Click OK.
9. In the *Geometry* area, next to the *Boundary (1)* field, click (Link Geometry).
10. Select the outer contour of the 2nd Core as shown in Figure 7–11.

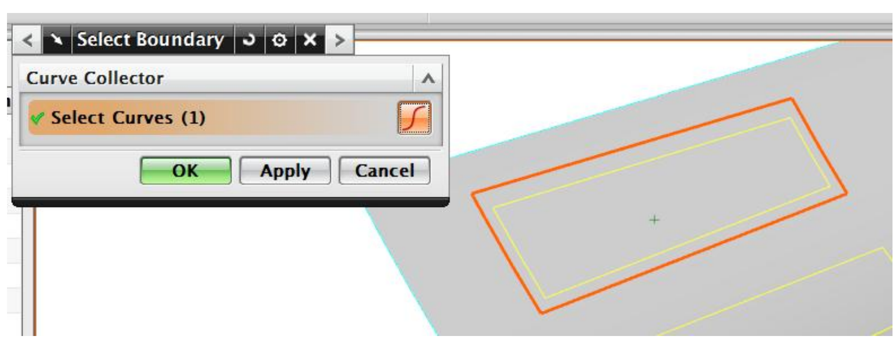


Figure 7–11

11. Click OK.

12. In the *Core Top* area, next to the *Boundary (4)* field, click (Link Geometry).
13. Select the inner contour of the 2nd Core as shown in Figure 7–12.

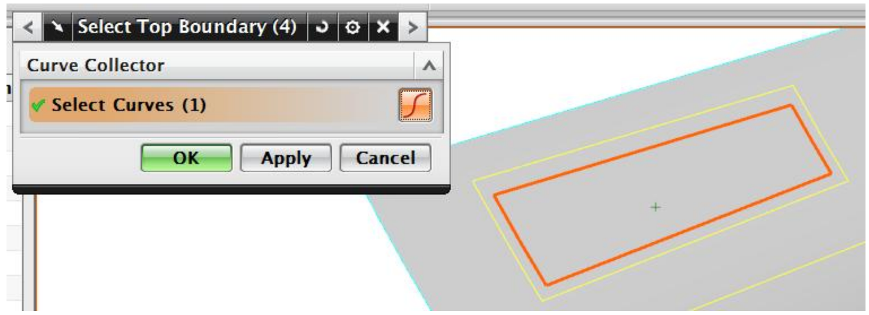


Figure 7–12

14. Click OK.
15. In the *Core Top* area, in the *Thickness (3)* field, enter [1.575 in] (40 mm). The form updates as shown in Figure 7–13.

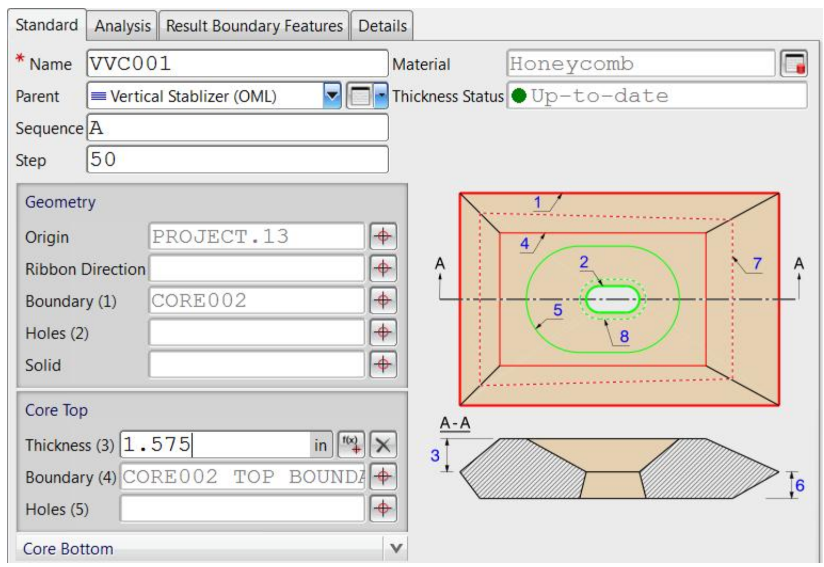


Figure 7–13

16. Click OK to complete the Core.

Task 4 - Create Two Virtual Cores.

Virtual Core only requires explicitly defining geometry for the Base Boundary. The cross-sectional shape of the Core is then implicitly defined by specifying numeric values for thickness, bevel angle, etc.

1. In the *Object Count* field, enter [2] to create two cores simultaneously as shown in Figure 7–14.

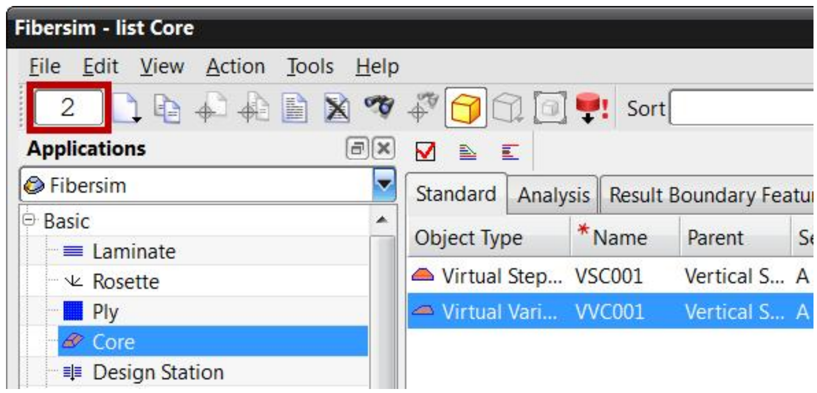


Figure 7–14

2. Click  (Create New). Expand the drop-down list and click  (Virtual Core). The Virtual Core definition form is displayed as shown in Figure 7–15.

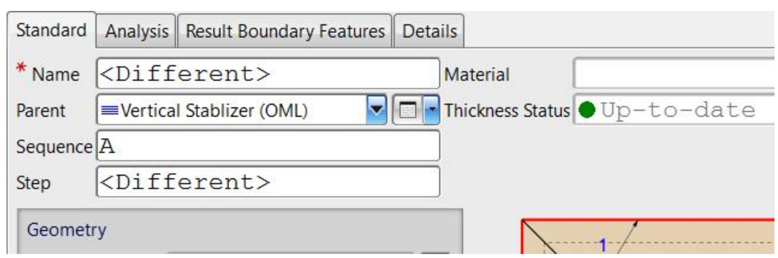


Figure 7–15

3. Next to the *Material* field, click  (Link with Database Link Dialog).

4. Select the checkbox for the **Honeycomb** material.

5. Click **OK**.

6. Enter the following parameters in the *Core Top* area as shown in Figure 7–16.
   - Thickness: **1 in (25.4 mm)**
   - Bevel Angle: **30°**

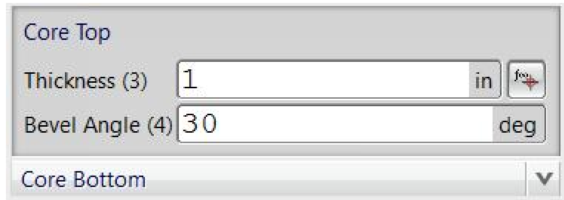


Figure 7–16

7. Press <Ctrl>+ <T> to switch to the Table view. The updated form displays as shown in Figure 7–17.

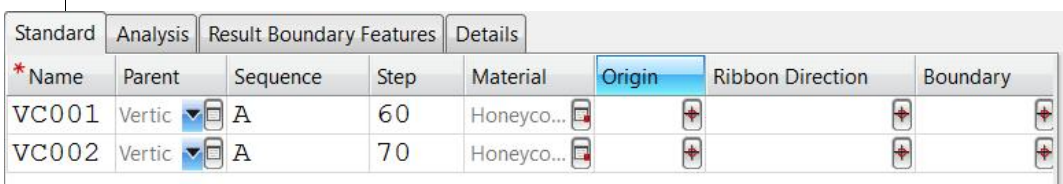


Figure 7–17

8. For the **VC001** core, next to the *Origin* field, click (Link Geometry).
9. Select the indicated point as shown in Figure 7–18.

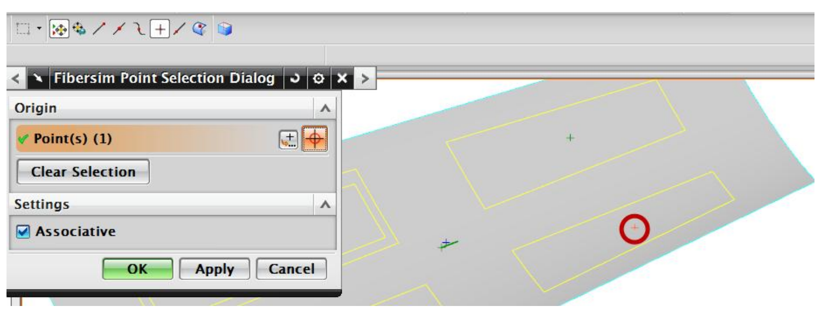


Figure 7–18

10. Repeat the steps 8 and 9 for the **VC002** core, selecting the point in the middle of **Core004** curve as shown in Figure 7–19.

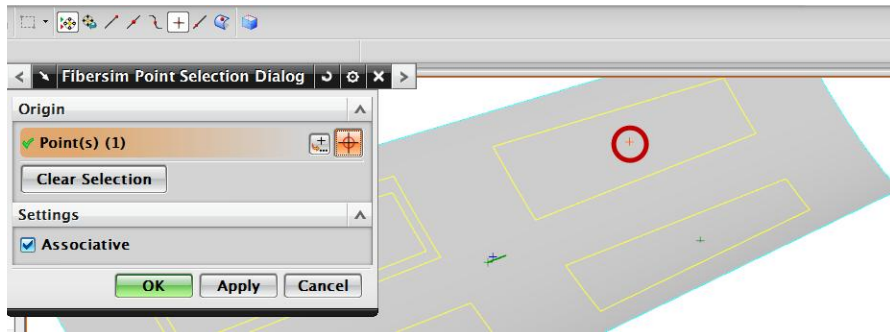


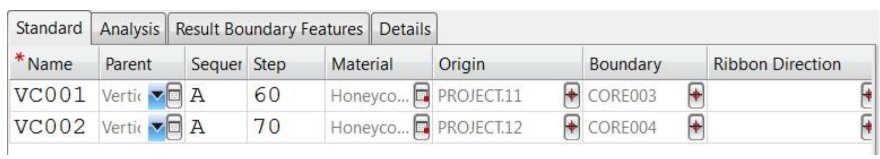
Figure 7–19

11. Click **OK**. If a warning box appears, click OK.

12. Link the Boundary geometry (in the *Boundary* column) for both Cores as follows:

    - VC001: select the **Core003** curve
    - VC002: select the **Core004** curve

13. The updated form displays as shown Figure 7–20.


Standard | Analysis | Result Boundary Features | Details

*Name	Parent	Sequer	Step	Material	Origin	Boundary	Ribbon Direction
VC001	Vertic	A	60	Honeyco...	PROJECT.11	CORE003	
VC002	Vertic	A	70	Honeyco...	PROJECT.12	CORE004	

Figure 7–20

It is a recommended best practice to return to Form Mode to prevent future confusion.

14. Press <Ctrl> + <T> to return to Form mode.

15. Click **OK** to complete both Cores.

Task 5 - Create 3D cross-sections.

1. Switch to NX and press <Ctrl> + <L> to launch Layer Settings dialog box.

2. Show the **Cross_Section_Curve** layer category.

3. Click **Close**.

***3D Cross Section** can also be found in the Fibersim **Documentation** menu.*

4. Press <F9> to start Fibersim.

5. In the Core toolbar, click (3D Cross Section) as shown in Figure 7–21.

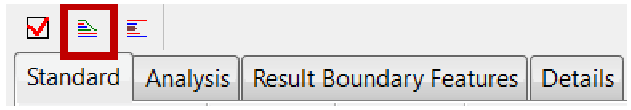


Figure 7–21

6. Click (Create New).

7. The cross-section form displays as shown in Figure 7–22.

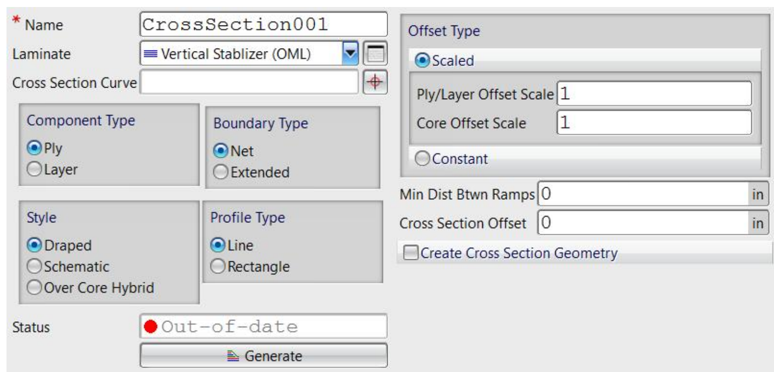


Figure 7–22

8. Next to the *Cross Section Curve* field, click (Link Geometry).

9. Select the **Cross Section Curve 1** as shown in Figure 7–23.

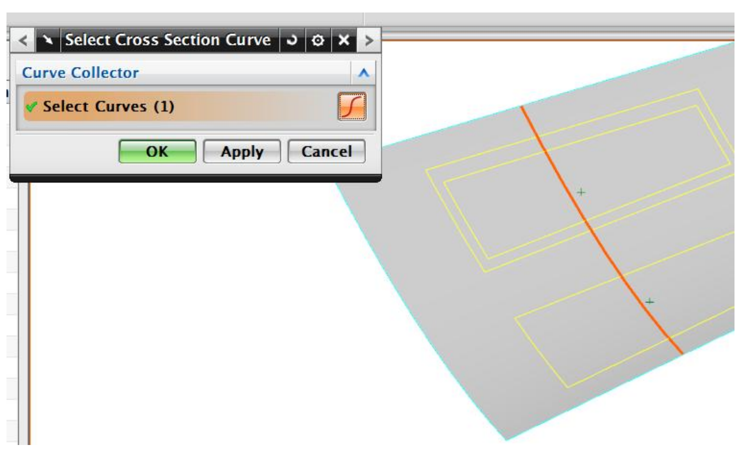
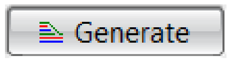


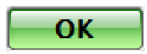
Figure 7–23

10. Click OK.

11. Enter the following parameters:
    - Component Type: **Ply**
    - Style: **Draped**
    - Ply/Layer Offset Scale: **20** (Note that this parameter is used to exaggerate the offset distances between plies in the cross-section for better visibility.)
    - Core Offset Scale: **1** (Note that this parameter is used to scale the core dimensions. A value of [1] means that the core(s) in the cross-section will be displayed at real size.)

12. Click 

.

13. Click 

to complete the cross-section.

14. Switch to the NX window. The model displays as shown in Figure 7–24.

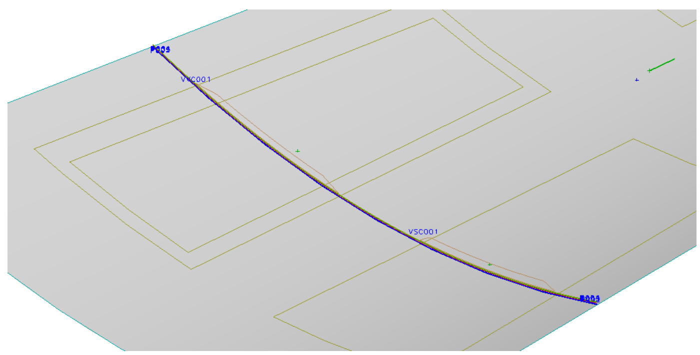


Figure 7–24

15. Zoom in to examine the cross-sections of the cores. Note the different bevel angles in the **VVC001** core, and the step geometry in **VSC001** core as shown in Figure 7–25.

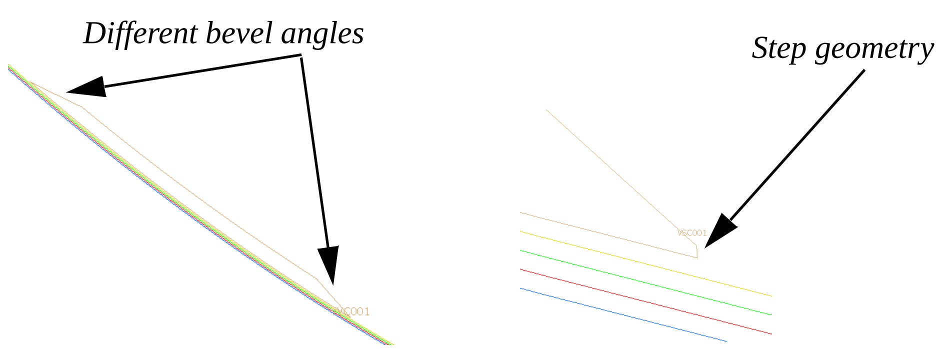


Figure 7–25

16. In the 3D Cross Section list, highlight **CrossSection001**, right-click and select **Create Based On** to create a **3D Cross Section** for **Cross Section Curve 2**.

17. Display both cross-sections (multi-select them in the Fibersim window), and hide the NX geometry. The model should display as shown in Figure 7–26.

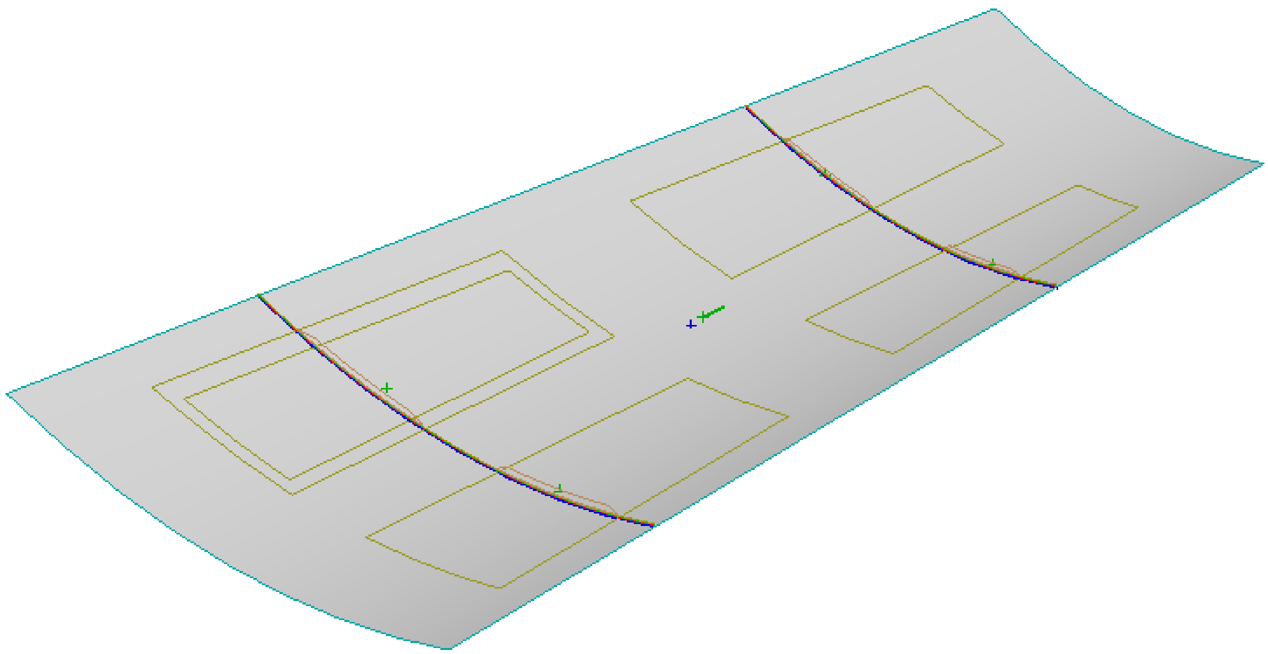


Figure 7–26

Exercise 7b Modeled Core with Overcore and IML Laminates

User Guide Reference:
2.9 Cores
9.7 Laminate Tab
2.10 Design Stations

In this exercise, you will create a Modeled Core object and create overcore and IML laminates. This will demonstrate the process of defining a core panel, display the results on producibility, and indicate why multiple laminates must be used.

After you complete this exercise, you will be able to:

- ✓ **Create a Modeled Core object**
- ✓ **Create an Overcore Laminate**
- ✓ **Re-sequence the overcore plies to reflect the overcore topology**
- ✓ **Create an IML laminate with a descending sequence order**
- ✓ **Interrogate the model using three Design Stations**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **CORE_B.prt**. The model displays as shown in Figure 7–27.

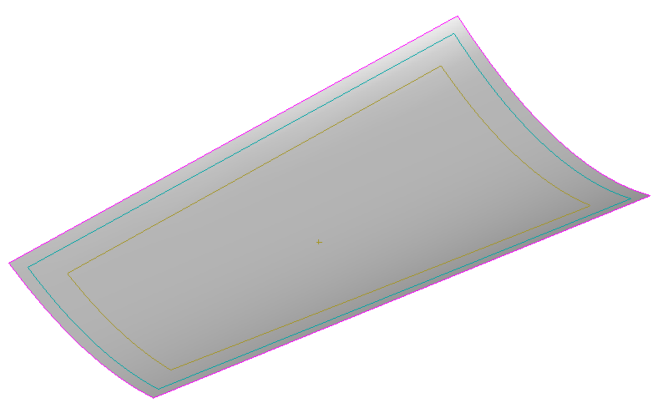


Figure 7–27

Task 2 - Create a Modeled Core object.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Core** under **Basic**.
3. Click (Create New).
4. In the drop-down list, click (Modeled Core) as shown in Figure 7–28.

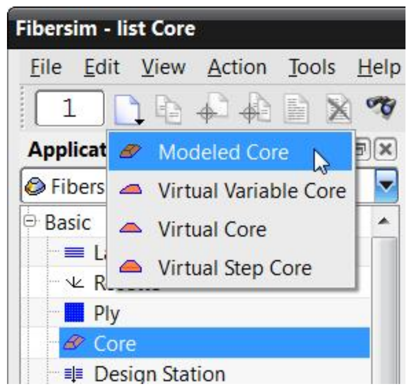


Figure 7–28

5. Enter the following parameters as shown in Figure 7–29:

- Parent: **OML**
- Step: **50**
- Material: **Honeycomb**
- Thickness: **0.5 in (12.7 mm)**
- Origin: **OML_PLY_ORIGIN**
- Boundary: **CORE_BOUNDARY**

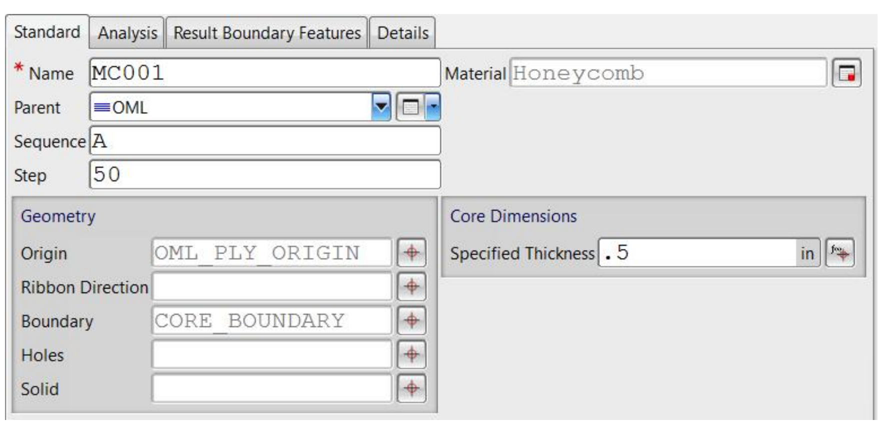
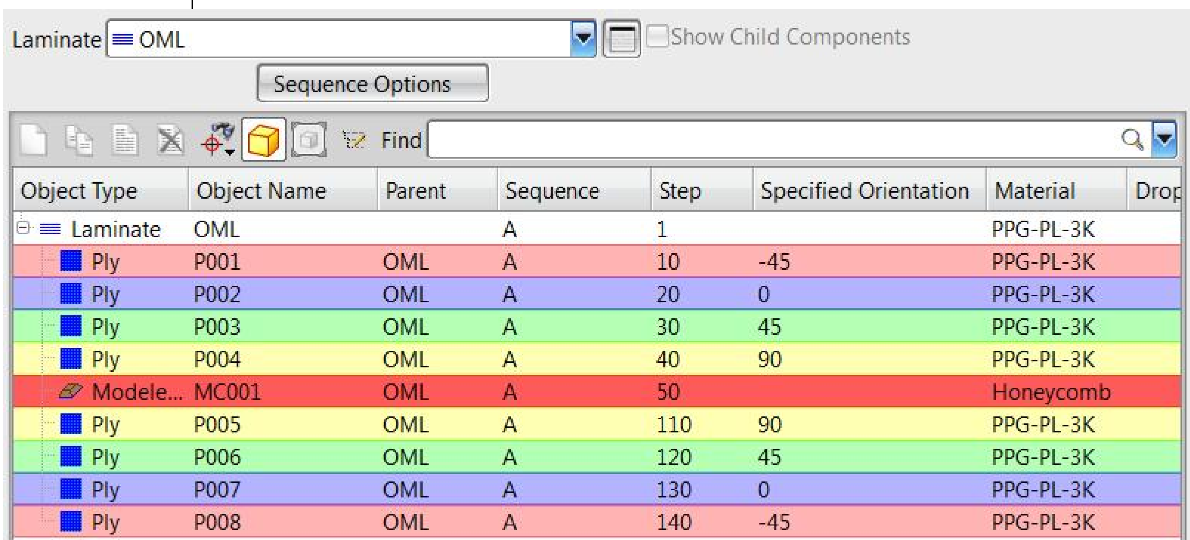


Figure 7–29

6. Click **OK** to save **MC001**.

7. Click (Composite Sequence Manager) to review the component stack as shown in Figure 7–30.



Object Type	Object Name	Parent	Sequence	Step	Specified Orientation	Material	Drop
Laminate	OML		A	1		PPG-PL-3K	
Ply	P001	OML	A	10	-45	PPG-PL-3K	
Ply	P002	OML	A	20	0	PPG-PL-3K	
Ply	P003	OML	A	30	45	PPG-PL-3K	
Ply	P004	OML	A	40	90	PPG-PL-3K	
Modele...	MC001	OML	A	50		Honeycomb	
Ply	P005	OML	A	110	90	PPG-PL-3K	
Ply	P006	OML	A	120	45	PPG-PL-3K	
Ply	P007	OML	A	130	0	PPG-PL-3K	
Ply	P008	OML	A	140	-45	PPG-PL-3K	

Figure 7–30

8. Click **OK**.

Task 3 - Create an Overcore Laminate.

1. In NX, hide the **OML_Geometry** layer category and show the **OverCore_Geometry** layer category in Layer Settings window.
2. Press <F9> to start Fibersim.
3. In the *Application Tree*, select **Laminate** under **Basic**.
4. Click (Create New).

A different alphabetically ordered Sequence letter is assigned to each laminate (Sequence A = OML; Sequence B = Overcore).

5. Enter the following parameters as shown in Figure 7–31:
   - Name: **Overcore**
   - Parent: **OML**
   - Sequence: **B**
   - Step: **100**
   - Layup Surface: **Overcore**
   - Net Boundary: **Overcore_Net_Boundary**
   - Extended Boundary: **Overcore_Extended_Boundary**

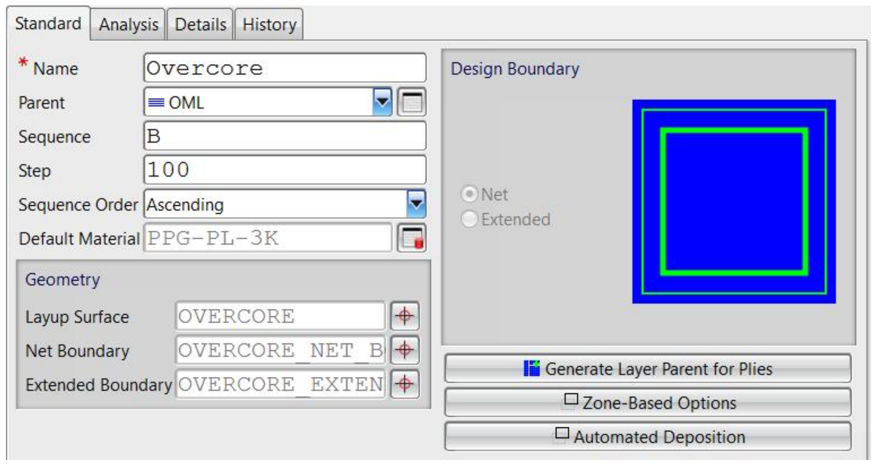
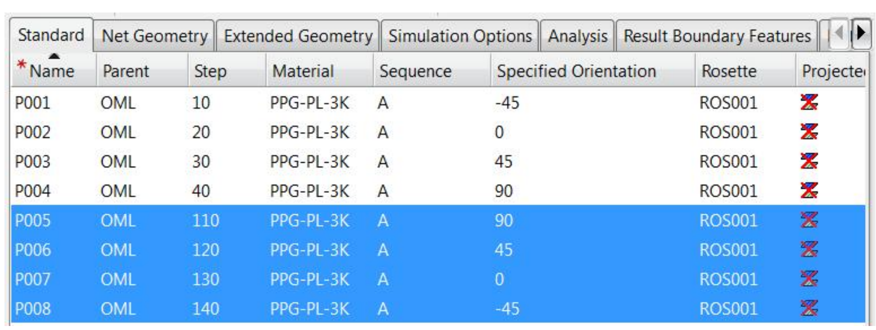


Figure 7–31

6. Click **OK** to save the **Overcore**.

Task 4 - Link the plies to Overcore Laminate

1. Select **Ply** from **Basic** menu.
2. Select **P005**, press <Shift> and select **P008** to multi-select plies **P005**, **P006**, **P007**, and **P008** as shown in Figure 7–32.



* Name	Parent	Step	Material	Sequence	Specified Orientation	Rosette	Projecte
P001	OML	10	PPG-PL-3K	A	-45	ROS001	
P002	OML	20	PPG-PL-3K	A	0	ROS001	
P003	OML	30	PPG-PL-3K	A	45	ROS001	
P004	OML	40	PPG-PL-3K	A	90	ROS001	
P005	OML	110	PPG-PL-3K	A	90	ROS001	
P006	OML	120	PPG-PL-3K	A	45	ROS001	
P007	OML	130	PPG-PL-3K	A	0	ROS001	
P008	OML	140	PPG-PL-3K	A	-45	ROS001	

Figure 7–32

3. With the four plies selected, right-click and select **Modify**.
4. For the *Parent* field, select **Overcore** from the pull-down list as shown in Figure 7–33.

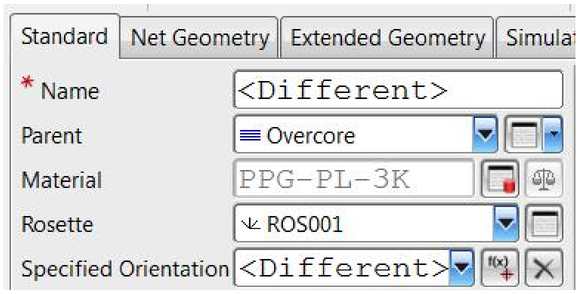
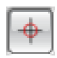


Figure 7–33

5. Click  (Link Geometry) next to Origin field in the Net Geometry area.
6. Select **OVERCORE_PLY_ORIGIN**.
7. Click  .

8. For reference, the component sequence and steps are shown in Figure 7–34.

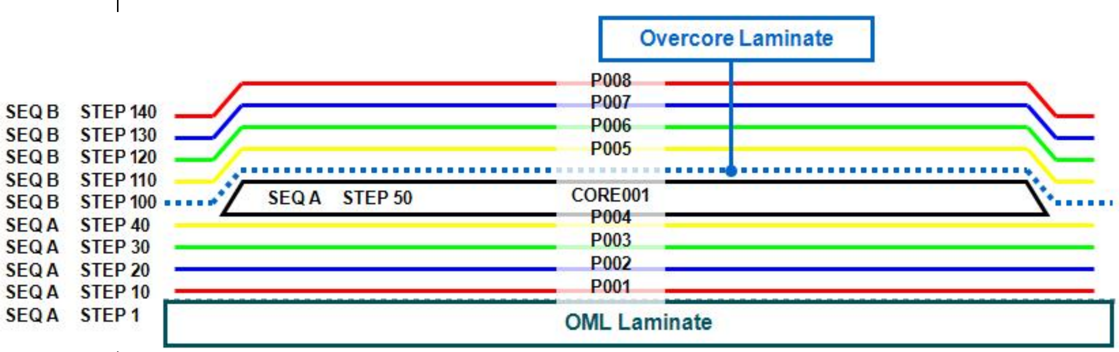


Figure 7–34

Task 5 - Run Net Producibility and generate Net Flat Patterns.

In this task, the **Fiber Spacing Factor** is reduced for plies (**P005** to **P008**) to ensure that the simulation follows the core ramps accurately. The simulation results should closely conform to the topological changes in the simulation surface (default laminate surface). Changing the **Fiber Spacing Factor** in this situation will result in a more accurate flat pattern for the plies covering the core object as shown in Figure 7–35.

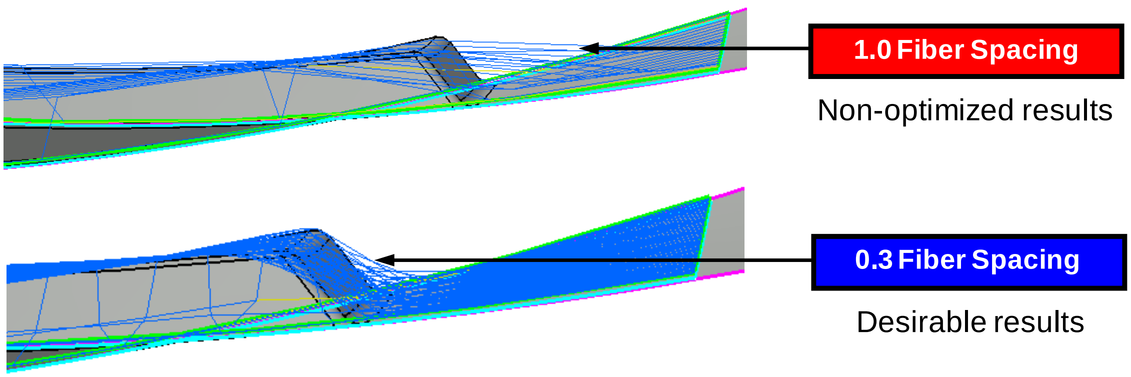


Figure 7–35

1. Select the *Simulation Options* tab.
2. For the *Fiber Spacing Factor*, enter [.3].
3. Click (Net Producibility) to run Net Producibility.
4. Click (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P005**, **P006**, **P007**, and **P008**.
5. Click OK to save.
6. Close the Fibersim window.

7. Switch to the **NX** window. In the Part Navigator, highlight **NET_FP_P001** to **NET_FP_P008** to display the net flat pattern results. Note the bulges in the updated flat patterns due to the different topology of the **Overcore** surface as shown in Figure 7–36.

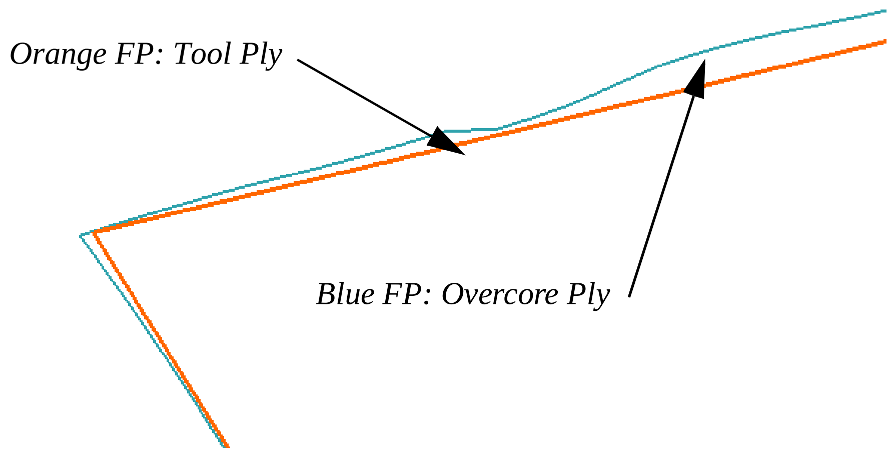


Figure 7–36

Task 6 - Create an IML laminate with a descending sequence order.

For a layup that progresses in opposite directions (closed mold or caul plate with bag side loading operations), as shown in Figure 7–37, the Sequence-Step order of the second laminate is typically defined as **Descending**. In this task you will change the Overcore laminate to be an IML laminate with a descending sequence order.

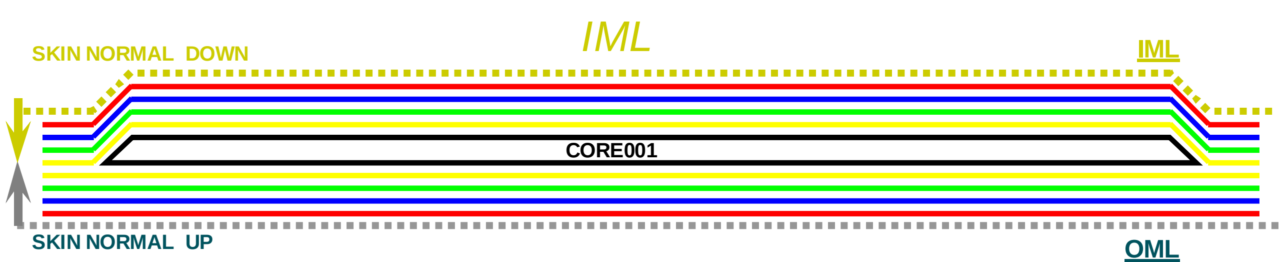


Figure 7–37

1. Hide **OVERCORE_GEOMETRY** and show **IML_GEOMETRY** layer category in the Layer Settings window.

2. Press <F9> to start Fibersim.

3. Create a new Laminate with the following parameters as shown in Figure 7–38:

   - Name: **IML**
   - Parent: **OML**
   - Step: **100**
   - Sequence Order: **Descending**
   - Layup Surface: **IML**
   - Net Boundary: **IML_Net_Boundary**
   - Extended Boundary: **IML_Extended_Boundary**

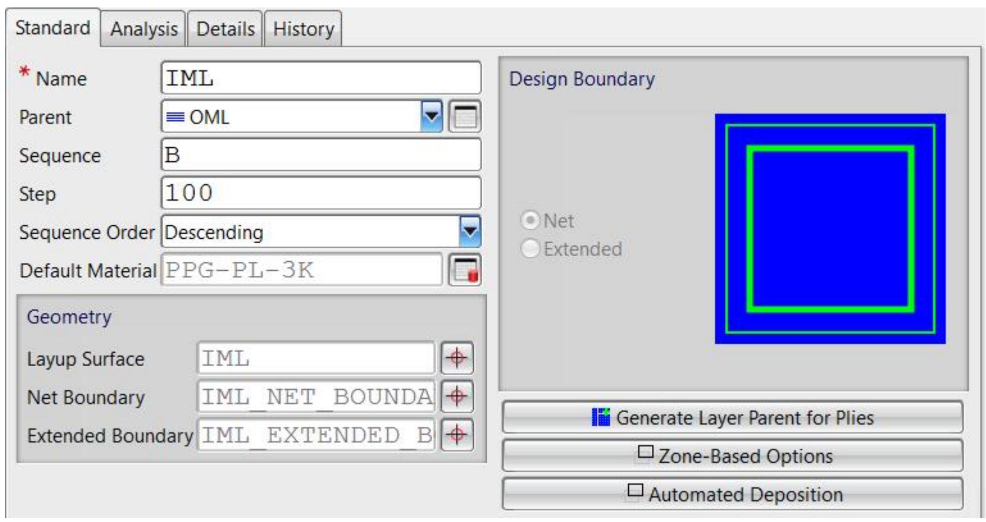


Figure 7–38

4. Click **OK** to save the IML laminate.
5. Modify **P005, P006, P007**, and **P008**.
6. Change Parent to **IML**.
7. Click **OK** to save the changes to **P005, P006, P007**, and **P008.**

Task 7 - Update the P005 to P008 Flat Patterns.

1. In the Ply listing, make sure **P005** to **P008** are highlighted.
2. Click (Net Producibility).
3. Click (Generates the Net Flat Pattern) to create a Net Flat Pattern of **P005**, **P006**, **P007**, and **P008**.

4. Switch to the NX window to display the net flat pattern results as shown in Figure 7–39.

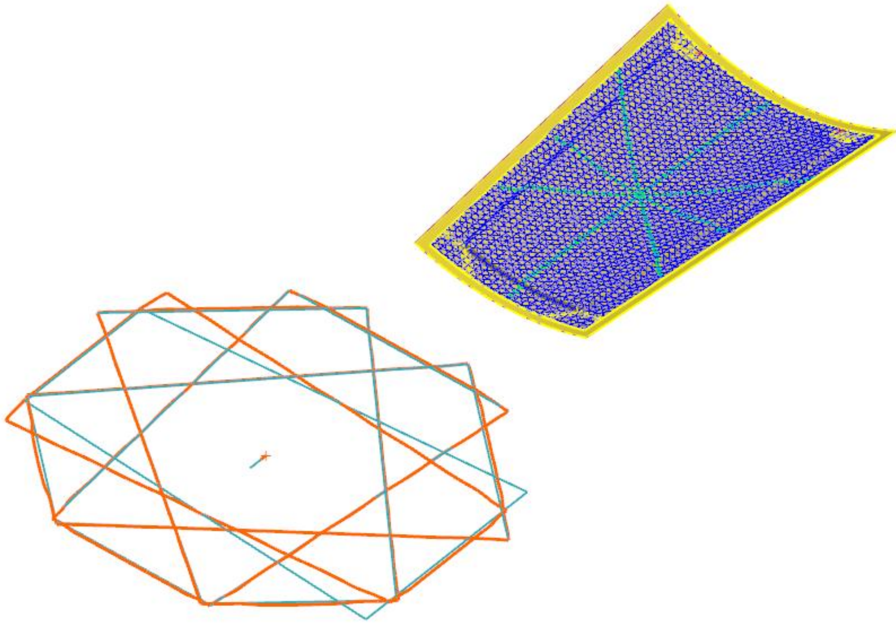


Figure 7–39

Task 8 - Interrogate the model using three Design Stations.

1. In the *Application Tree*, select **Design Station** under **Basic** as shown in Figure 7–40.

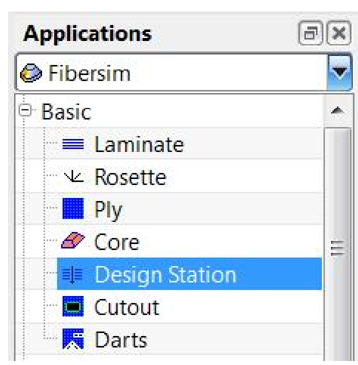


Figure 7–40

2. Click  (Create New).
3. For the *Laminate* field, select **OML** from the pull-down list.
4. Click  (Link Geometry) next to *Origin*.

5. Hide **IML_GEOMETRY** and show the **Design Stations** layer category in the Layer Settings window.

6. Select **DS001_Origin** as shown in Figure 7–41.

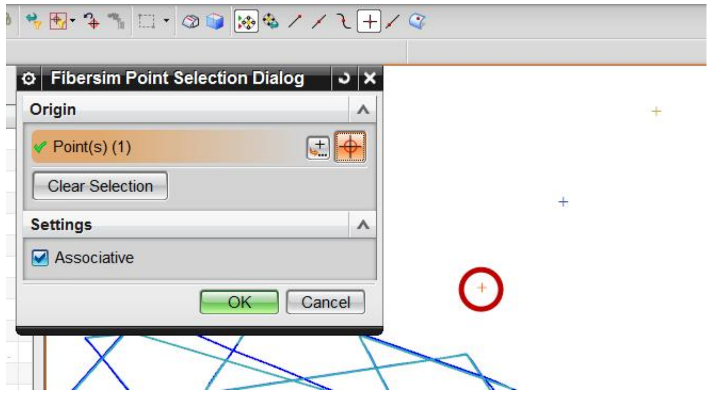


Figure 7–41

7. Click 

.

8. In the Design Station Standard form, expand the Core Sample Type drop-down list and select **Summary** if it is not already selected.

9. Click  (Core Sample). The Fibersim message box opens prompting you that the core samples have completed successfully as shown in Figure 7–42.

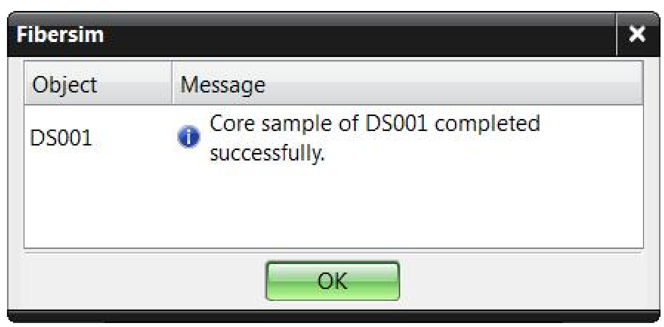
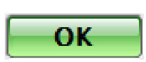


Figure 7–42

10. Click 

twice to go back to the Design Station list.

11. Enter [2] in the object counter as shown in Figure 7–43.

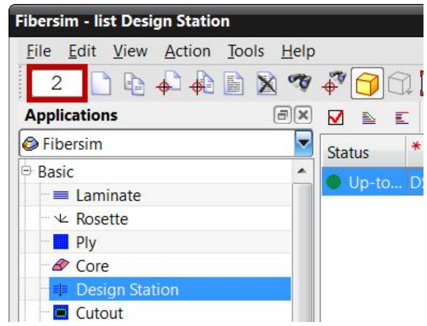


Figure 7–43

12. Right-click on **DS001** and select **Create BasedOn**.

13. Press <Ctrl> + <T> to switch to the List View.

14. Link the following geometry for *Origin*.

- DS002: DS002_Origin
- DS003: DS003_Origin

15. Press <Ctrl> + <T> to switch to the Form View.

16. Click (Core Sample).

17. Click **OK** twice.

18. In the Design Station list, double-click on **DS001**.

19. Scroll down to view the results of the core sample for **DS001**. The measured thickness of the core is less than the specified thickness, as shown in Figure 7–44, because the origin point lies on a core ramp.

Core Sample Analysis

Name: DS001
Parent: OML
Rosette: (None)
Identifier: DS001_ORIGIN
Location: (-16.745115, -0.000000, 1.622058)
Units: Inch (in)
Total Component Count: 9
Total Thickness: 0.294997

Name	Seq/Step	Orient	Material	Thickness
P001	A10	-45	PPG-PL-3K	0.007500
P002	A20	0	PPG-PL-3K	0.007500
P003	A30	45	PPG-PL-3K	0.007500
P004	A40	90	PPG-PL-3K	0.007500
MC001	A50	N/A	Honeycomb	0.204987 (1.000000)
P005	B110	90	PPG-PL-3K	0.015003
P006	B120	45	PPG-PL-3K	0.015003
P007	B130	0	PPG-PL-3K	0.015003
P008	B140	-45	PPG-PL-3K	0.015003

Figure 7–44

20. Click **OK**.

21. In the Design Station listing, double-click on **DS002** to display the **DS002** core sample results as shown in Figure 7–45. The measured thickness of the core is equal to the specified thickness because the origin point lies within the constant thickness region.

Name	Seq/Step	Orient	Material	Thickness
P001	A10	-45	PPG-PL-3K	0.007500
P002	A20	0	PPG-PL-3K	0.007500
P003	A30	45	PPG-PL-3K	0.007500
P004	A40	90	PPG-PL-3K	0.007500
MC001	A50	N/A	Honeycomb	0.500000 (1.000000)
P005	B110	90	PPG-PL-3K	0.007500
P006	B120	45	PPG-PL-3K	0.007500
P007	B130	0	PPG-PL-3K	0.007500
P008	B140	-45	PPG-PL-3K	0.007500

Figure 7–45

22. Click **OK**

23. In the Design Station listing, double-click on **DS003** to display the **DS003** core sample results as shown in Figure 7–46.

A core is not present at this Design Station

Name	Seq/Step	Orient	Material	Thickness
P001	A10	-45	PPG-PL-3K	0.007500
P002	A20	0	PPG-PL-3K	0.007500
P003	A30	45	PPG-PL-3K	0.007500
P004	A40	90	PPG-PL-3K	0.007500
P005	B110	90	PPG-PL-3K	0.007500
P006	B120	45	PPG-PL-3K	0.007500
P007	B130	0	PPG-PL-3K	0.007500
P008	B140	-45	PPG-PL-3K	0.007500

Figure 7–46

24. Close Fibersim and save the model.

Exercise 7c

Core Panel Design from a Solid (Optional)

User Guide Reference:
2.9 Cores
5.2 3D Cross Section
2.10 Design Stations

In this exercise you will design a multi-laminate composite part with a core, starting from a solid NX model with minimal guidance.

The solid model of the part is shown in Figure 7–47. The part consists of 16 full-body plies made of PPG-PL-3K pre-preg fabric [cured thickness 0.0075 in (0.1905 mm)], and a 0.394 in (10 mm) thick core made of foam. Eight plies are laid underneath the core, and another eight plies are placed over the core. Therefore, the total thickness of the part is 0.454 in (11.5316 mm) in the core area and 0.06 in (1.524 mm) in all other areas.

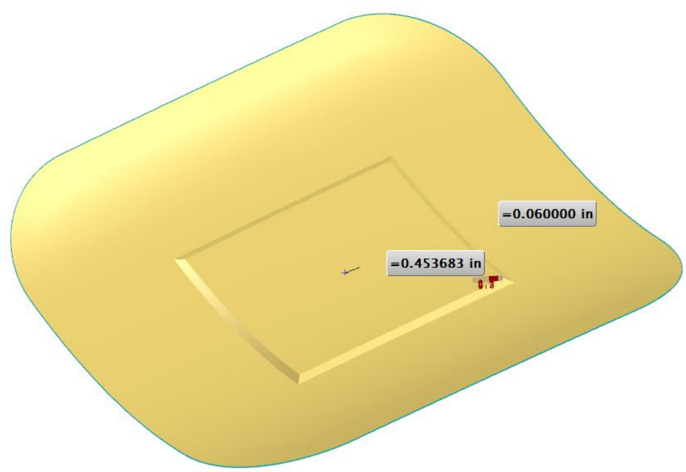


Figure 7–47

After you complete this exercise, you will be able to:

- ✓ **Create an OML Laminate**
- ✓ **Create a Modeled Core**
- ✓ **Create an Overcore Laminate**
- ✓ **Review the design-in-progress with Design Station and 3D Cross Sections**
- ✓ **Analyze producibility and obtain Flat Patterns for OML and Overcore plies**

Estimated Time

20 min

Task 1 - Open the part.

1. Open **CORE_C.prt**. The model displays as shown in Figure 7–48.

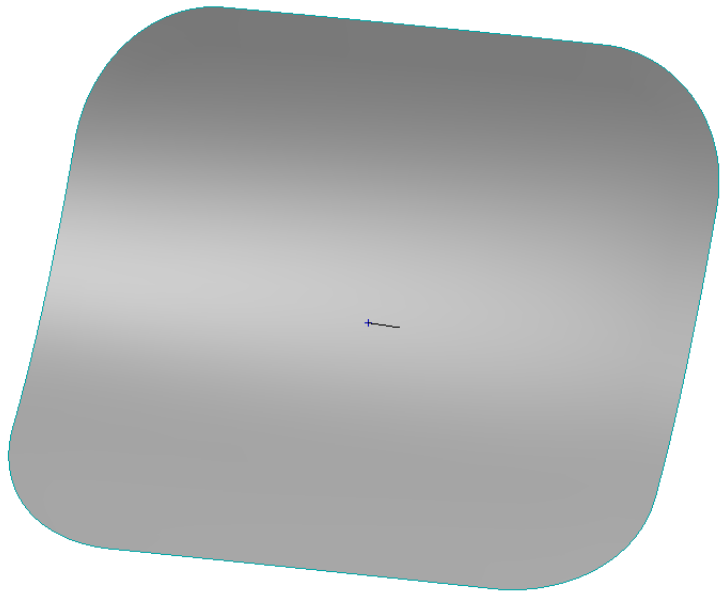


Figure 7–48

Task 2 - Review the part.

The model is already prepared for composite design with Fibersim, with all of the necessary surfaces and curves extracted, etc. In this task, you will review the prepared geometry.

1. In NX, press <Ctrl> + <L> to open Layer Settings window as shown in Figure 7–49.

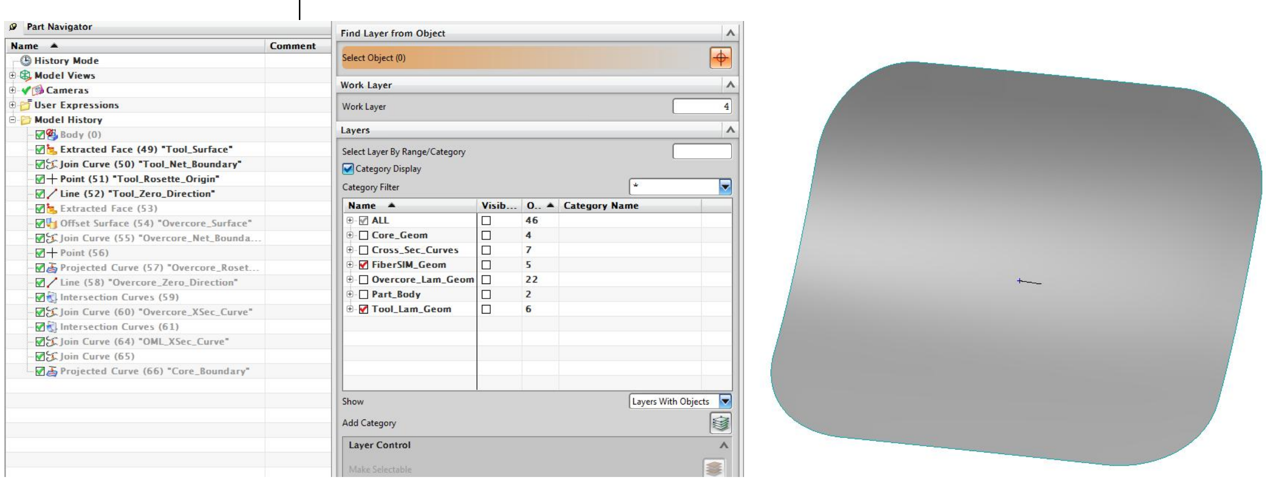


Figure 7–49

The **Tool_Lam_Geom** layer category contains the geometry required to create the undercore plies:

- **Tool Surface**: Surface extracted from bottom of solid model.
- **Tool Net Boundary**: Boundary edges of Tool Surface.
- **Tool Rosette**: Geometric set containing origin and zero-direction of rosette.

2. Hide **Tool_Lam_Geom** and show the **Overcore_Lam_Geom** layer category set as shown in Figure 7–50.

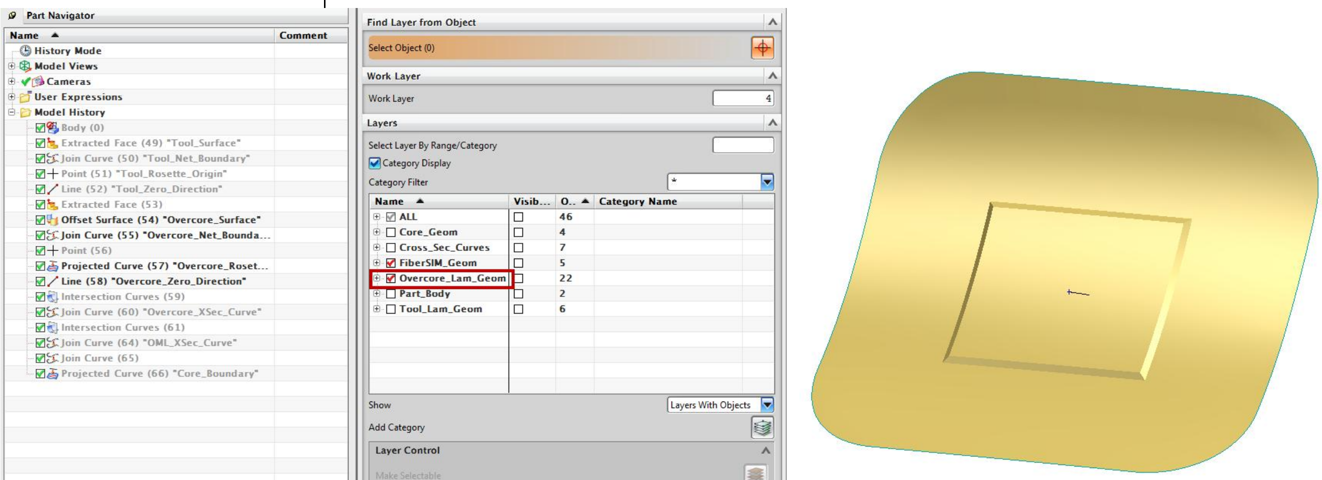


Figure 7–50

The **Overcore_Lam_Geom** layer category contains the geometry required to create the overcore plies:

- **Overcore Surface**: Surface extracted from top of solid and offset by 0.12 in (3.048 mm), the total thickness of the overcore plies.
- **Overcore Net Boundary**: Boundary edges of Overcore Surface.
- **Overcore Rosette**: Geometric set containing origin and zero-direction of rosette.

3. Hide **Overcore_Lam_Geom** and show both the **Tool_Lam_Geom** and **Core_Geom** layer categories as shown in Figure 7–51. Note that the **Core_Geom** category contains the Core Boundary curve (extracted from the solid edges and projected onto the **Tool Surface**).

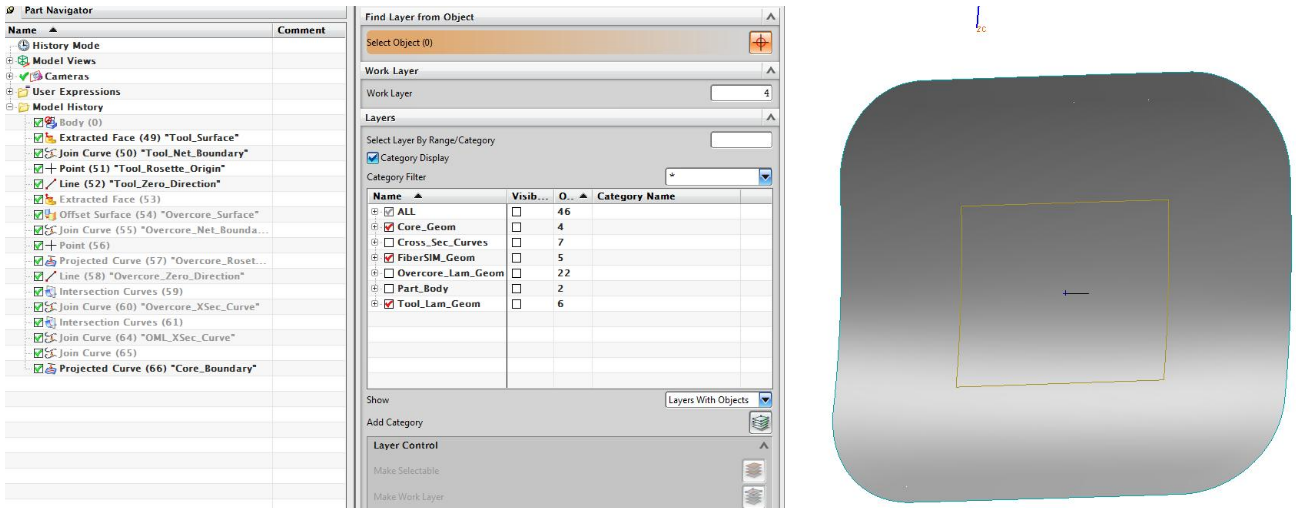


Figure 7–51

4. Click **Close**.

Task 3 - Create the OML Laminate.

1. In Fibersim, create a laminate with the following parameters:

   - Name: **OML Laminate**
   - Sequence: **A**
   - Step: **1**
   - Default Material: **PPG-PL-3K**
   - Layup Surface: **Tool_Surface**
   - Net Boundary: **Tool_Net_Boundary**

2. The laminate form is shown in Figure 7–52.

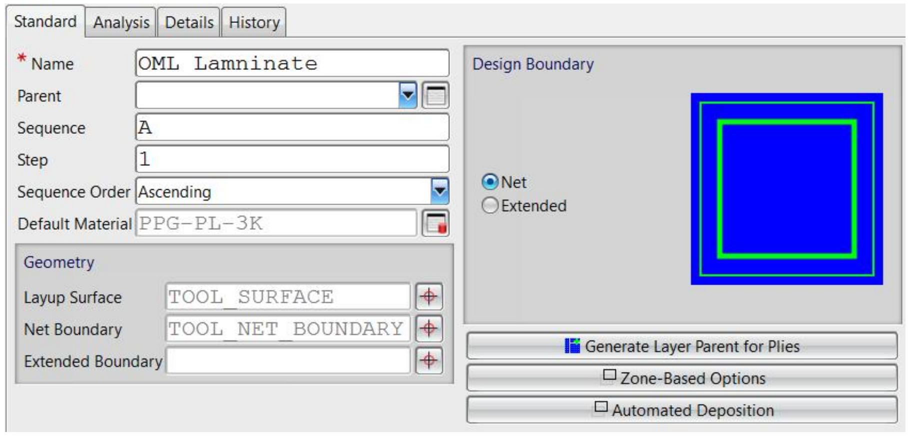


Figure 7–52

3. Create a new Rosette with the following parameters:

   - Name: **OML Rosette**
   - Origin: **TOOL_ROSETTE_ ORIGIN**
   - Direction: **TOOL_ZERO_DIRECTION**

4. Create eight full-body plies with the following parameters:

   - Name: **Accept the default**
   - Sequence: **A**
   - Step: **10,10**
   - Orientations (**P001** to **P008**): **0, 45, 90, -45, 45, 90, -45, 0**

5. The plies are shown in Figure 7–53.

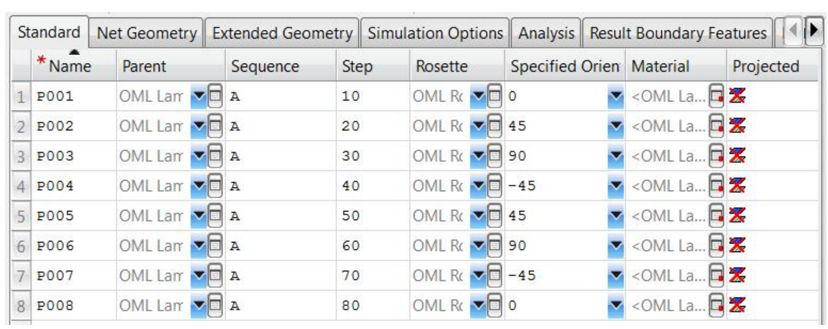


Figure 7–53

6. Create a Design Station (DS001) at **OML Rosette** and run a core sample to verify that the total thickness of the **OML Laminate** is currently 0.06 in (1.524 mm) (8 plies with 0.0075 in (0.1905 mm) each cured thickness).

*Note: Show the **Core** geometric set to locate the Boundary.*

Task 4 - Create a Modeled Core.

1. Create a new core with the following parameters:
   - Name: **MC001**
   - Type: **Modeled Core**
   - Parent: **OML Laminate**
   - Sequence: **A**
   - Step: **90**
   - Material: **Foam-3.1**
   - Origin: **TOOL_ROSETTE_ORIGIN**
   - Boundary: **Select all 4 curves**
   - Thickness: **0.394" (10 mm)**

Task 5 - Create an Overcore Laminate.

1. Create a laminate with the following parameters:
   - Name: **Overcore Laminate**
   - Sequence: **B**
   - Step: **1**
   - Sequence order: **Ascending**
   - Default Material: **PPG-PL-3K**
   - Layup Surface: **Overcore_Surface**
   - Net Boundary: **Overcore_Net_Boundary**
2. The laminate form is shown in Figure 7–54.

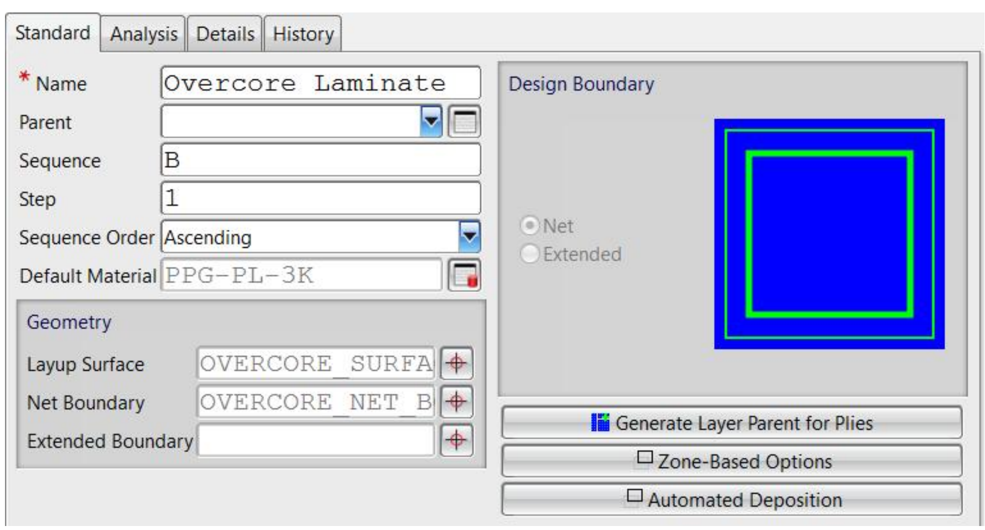
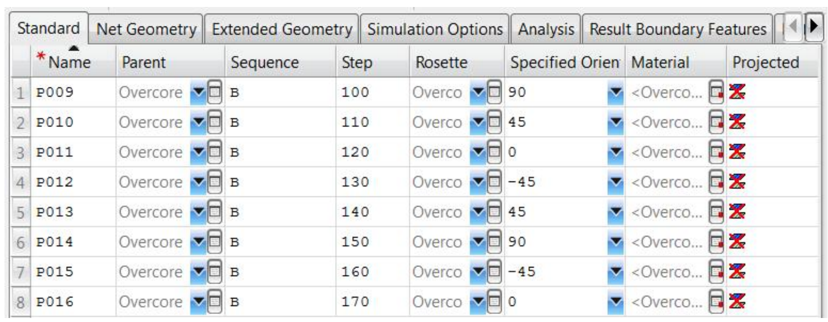


Figure 7–54

3. Create a new Rosette with the following parameters:
   - Name: **Overcore Rosette**
   - Surface: **Overcore_Surface**
   - Origin: **Overcore_Rosette_Origin**
   - Direction: **Overcore_Zero_Direction**

4. Create eight new full-body plies, with the following parameters (ignore the warning messages regarding the invalid Net Boundary type):
   - Name: **Accept the default names**
   - Parent: **Overcore Laminate**
   - Rosette: **Overcore Rosette**
   - Sequence: **B**
   - Step: **100,10**
   - Orientations (**P009** to **P016**): **90, 45, 0, -45, 45, 90, -45, 0**

5. The plies are shown in Figure 7–55.



	*Name	Parent	Sequence	Step	Rosette	Specified Orien	Material	Projected
1	P009	Overcore	B	100	Overco	90	<Overco...	
2	P010	Overcore	B	110	Overco	45	<Overco...	
3	P011	Overcore	B	120	Overco	0	<Overco...	
4	P012	Overcore	B	130	Overco	-45	<Overco...	
5	P013	Overcore	B	140	Overco	45	<Overco...	
6	P014	Overcore	B	150	Overco	90	<Overco...	
7	P015	Overcore	B	160	Overco	-45	<Overco...	
8	P016	Overcore	B	170	Overco	0	<Overco...	

Figure 7–55

6. Create a new Design Station (**DS002**) and verify that the thickness of the **Overcore Laminate** is 0.06 in (1.524 mm) (eight plies, each with 0.0075 in (0.1905 mm) cured thickness).

Task 6 - Create 3D Cross Sections.

1. Create two 3D Cross Sections with the following parameters:
   - Name: **CrossSection OML**
   - Laminate: **OML Laminate**
   - Cross Section Curve: **OML_Xsec_Curve** (in the Cross_Sec_Curves layer category)
   - Ply/Layer Offset Scale: **1**
   - Core Offset Scale: **1**

- Name: **CrossSection Overcore**
- Laminate: **Overcore Laminate**
- Cross Section Curve: **Overcore_Xsec_Curve** (in the Cross_Sec_Curves layer category)
- Ply/Layer Offset Scale: **1**
- Core Offset Scale: **1**

2. In NX, hide all other layer categories except for **Fibersim_Geom** and **Part_Body** in Layer Settings window.

3. In Fibersim, multi-select both 3D Cross Sections. The model should display as shown in 4..

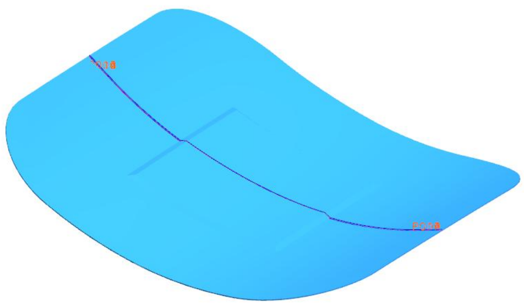


Figure 7–56

4. In NX, change the render style to **Static Wireframe**.

5. Zoom in on the cross-section and verify whether the laminate thickness, etc., matches the original solid model as shown in Figure 7–57.

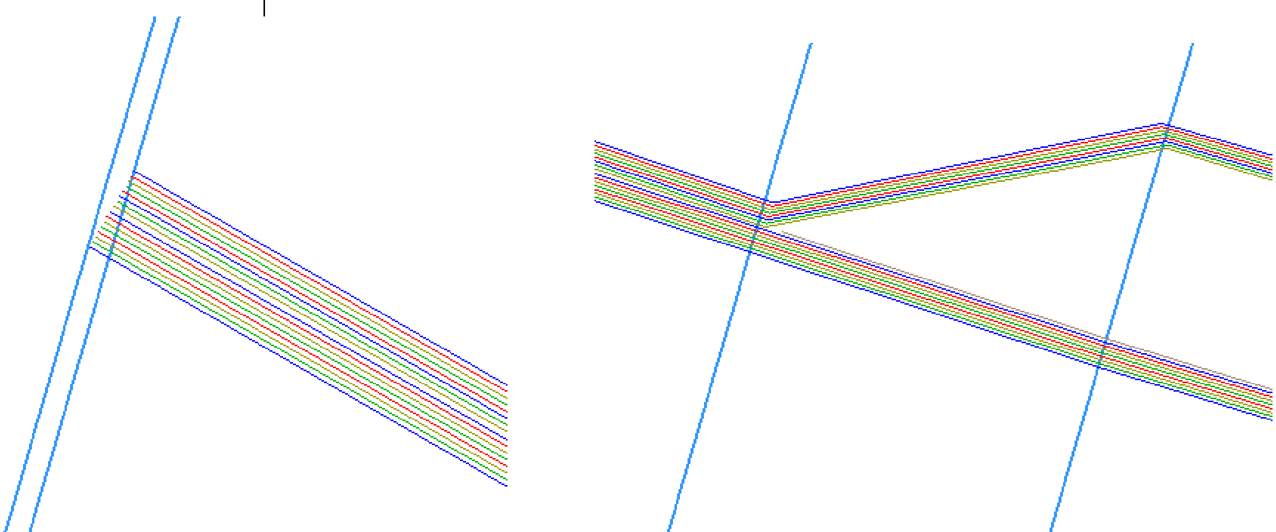


Figure 7–57

Task 7 - Create flat patterns for the undercore and overcore plies.

1. Run **Net Producibility** (set the *Fiber Spacing Factor* as [0.2]) and create a **Net Flat Pattern** for the **P001** ply.
2. Run **Net Producibility** (set the *Fiber Spacing Factor* as [0.2]) and create a **Net Flat Pattern** for the **P009** ply.
3. In the Fibersim window, highlight plies **P001** and **P009**. Examine the differences in the flat patterns for the undercore (**P001**) and overcore (**P009**) plies as shown in Figure 7–58.

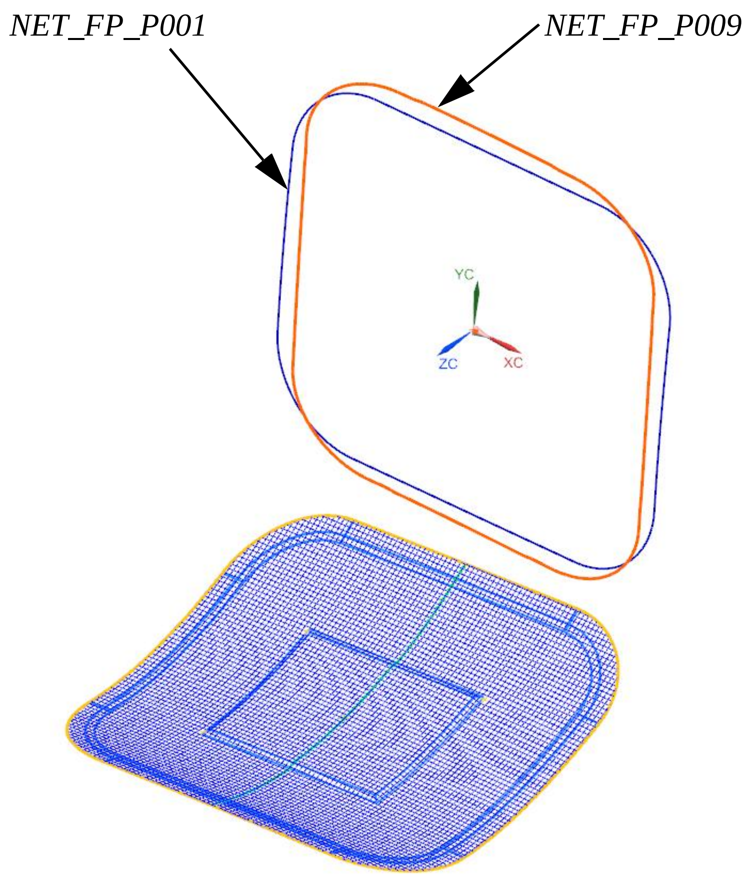


Figure 7–58

4. Close Fibersim and save the NX model.

Chapter 8

Documentation

This chapter introduces:

✓**Exercise 8a: 3D Documentation**

✓**Exercise 8b: Generating a Ply Book and Ply Table**

Exercise 8a 3D Documentation

User Guide Reference:
1.5 Documentation
5.1 Introduction
5.2 3D Cross Section
5.3 Explode Laminate
5.5 3D Text Annotations

In this exercise, you will use the Documentation tools available in Fibersim to create 3D cross-sections and annotations, and to generate an exploded view of the ply stack-up. The completed model displays as shown in Figure 8–1.

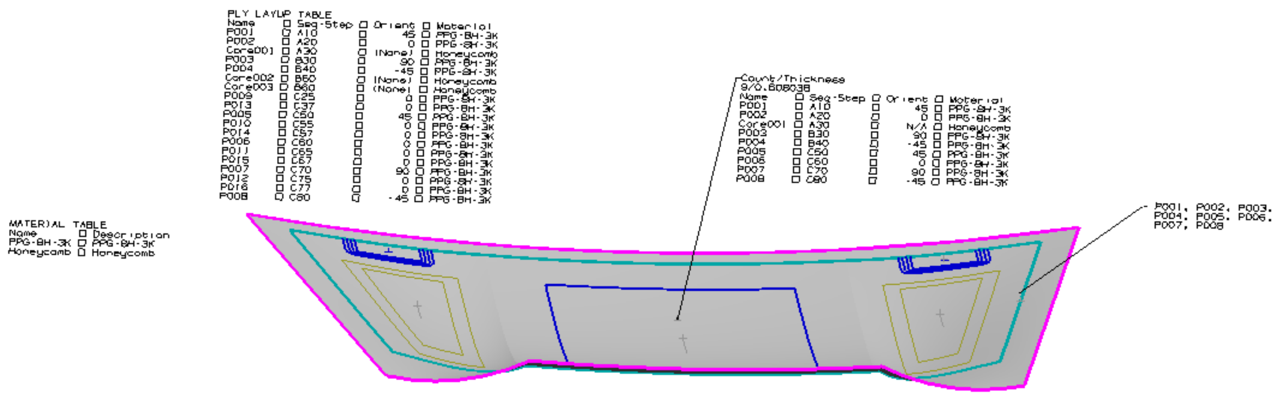


Figure 8–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Generate and update 3D cross-sections**
- ✓ **Create a core sample annotation**
- ✓ **Generate a ply table**
- ✓ **Create a ply callout**
- ✓ **Generate a material table**

Estimated Time

20 min

Task 1 - Review the model.

The model and much of the Fibersim data have already been completed. This exercise will focus on the documentation tools of Fibersim.

1. Open **DESIGN_AND_DOC_A.prt**. The model displays as shown in Figure 8–2.

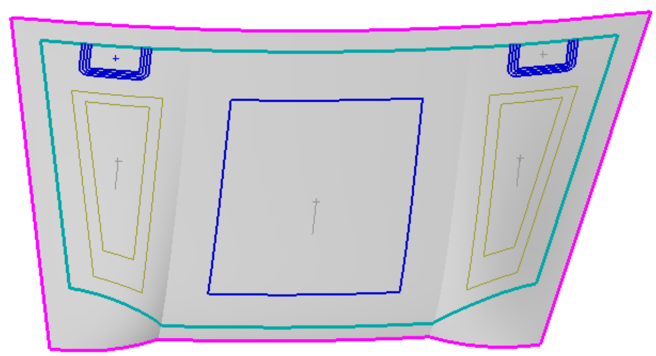


Figure 8–2

2. Show the **CROSS_SECTION_CURVE_1** and **CROSS_SECTION_CURVE_2** layer categories as shown in Figure 8–3. These curves determine where on the Tool Surface the cross-section will be created. The curves have been created on the Tool Surface. The Tool Surface is the skin for the Hood laminate on which the cross-section will be generated.

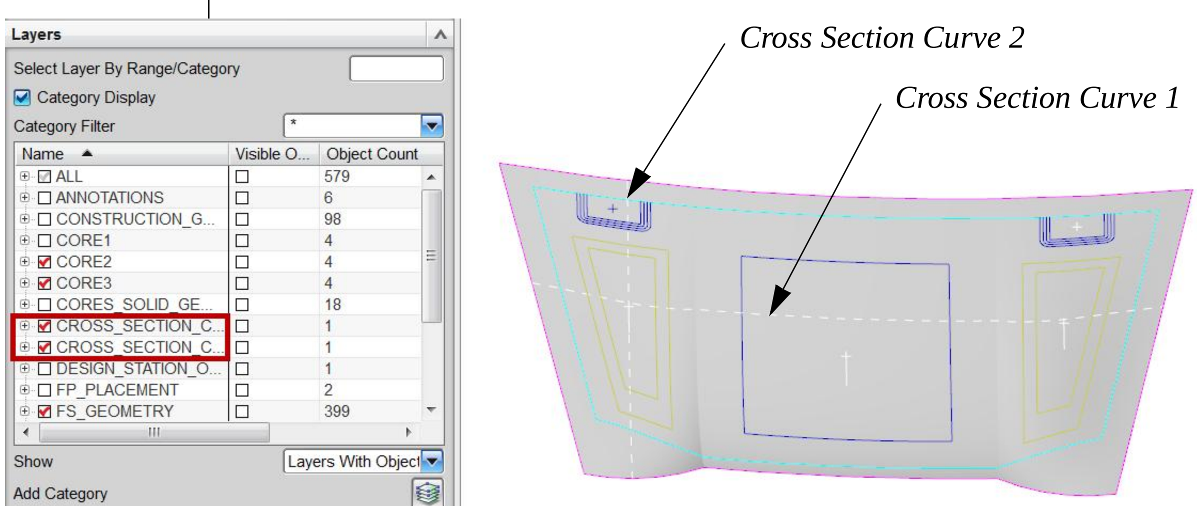
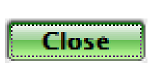


Figure 8–3

3. Click 
.

Task 2 - Generate 3D Cross Section.

In this task, you will create two 3D cross-sections. 3D cross-sections enable you to visually inspect the laminate stack and help in the manufacturing of the composite part.

1. Press <F9> to start Fibersim.
2. From the *Application Browser* select **Documentation** as shown in Figure 8–4.

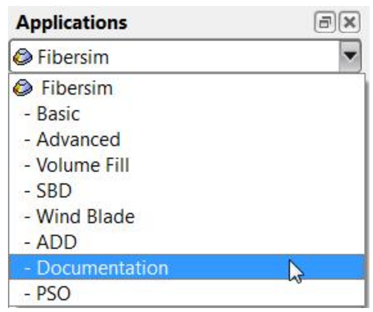


Figure 8–4

3. In the *Application Tree*, select **3D ross Section** under **3D Documentation**.
4. Click (Create New).
5. Specify the following parameters as shown in Figure 8–5:
   - Laminate: **Hood**
   - Cross Section Curve: **CROSS SECTION 1**
   - Ply/Layer Offset Scale: **10**

The Cross Section Curve determines where on the Tool Surface the cross-section will be created.

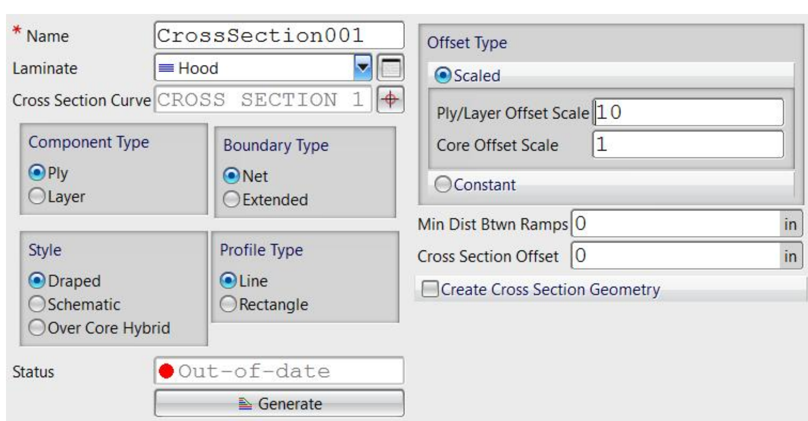


Figure 8–5

6. Click (Generates the 3D cross section) and click **OK**.

7. Review the 3D Cross Section curves generated on the model. The Cross Section tool adds color to the cross-section curves based on ply orientations for each material.

8. Click (Create New).

9. Create a second cross-section using the following parameters:

   - Laminate: **Hood**
   - Cross Section Curve: **Cross Section 2**
   - Ply/Layer Offset Scale: **10**
   - Boundary Type: **Extended**
   - Profile Type: **Rectangle**

10. Click (Generates the 3D cross section) and click **OK**.

11. Review the 3D Cross Section curves generated on the model.

In this task, you created the cross-sections as display geometry. To display the cross-section you must select it in Fibersim. If you want to create the cross-sections as actual CAD Geometry, select the **Create Cross Section Geometry** option in the create 3D Cross Section dialog box.

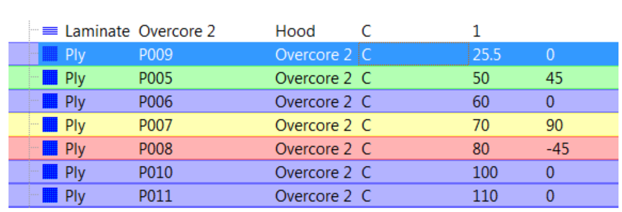
Task 3 - Re-sequence the plies in the Hood laminate and update 3D cross sections.

1. In the *Application Tree*, select **Ply** under **Source.**

2. Click (Composite Sequence Manager).

3. In the Laminate drop-down list, select **Hood**.

4. In the list of objects, select **P009** and drag it under **Overcore 2**, as shown in Figure 8–6.



Laminate	Overcore 2	Hood	C	1	
Ply	P009	Overcore 2	C	25.5	0
Ply	P005	Overcore 2	C	50	45
Ply	P006	Overcore 2	C	60	0
Ply	P007	Overcore 2	C	70	90
Ply	P008	Overcore 2	C	80	-45
Ply	P010	Overcore 2	C	100	0
Ply	P011	Overcore 2	C	110	0

Figure 8–6

5. Drag and drop the plies as described in Table 8–1 and shown in Figure 8–7:

Ply	Place after:
P013	P009
P010	P005
P014	P010
P011	P006
P015	P011
P012	P007
P016	P012

Table 8–1

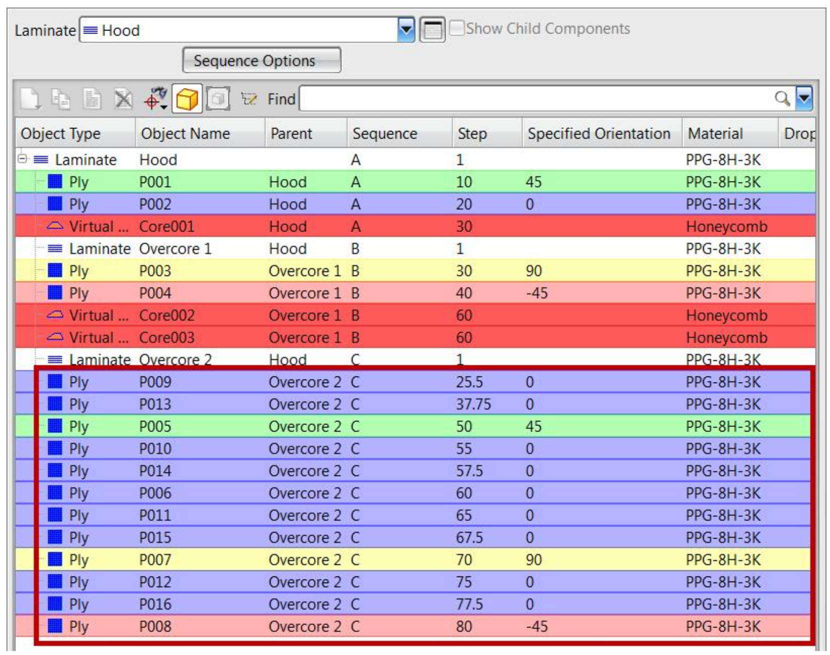


Figure 8–7

6. Click **OK**.

7. In the *Application Tree*, select **3D Cross Section** under **3D Documentation**.

8. Press <Ctrl> and select both **CrossSection002** and **CrossSection001**. The cross-sections display on the model.

9. Zoom in on the cross-section at the location shown in Figure 8–8.

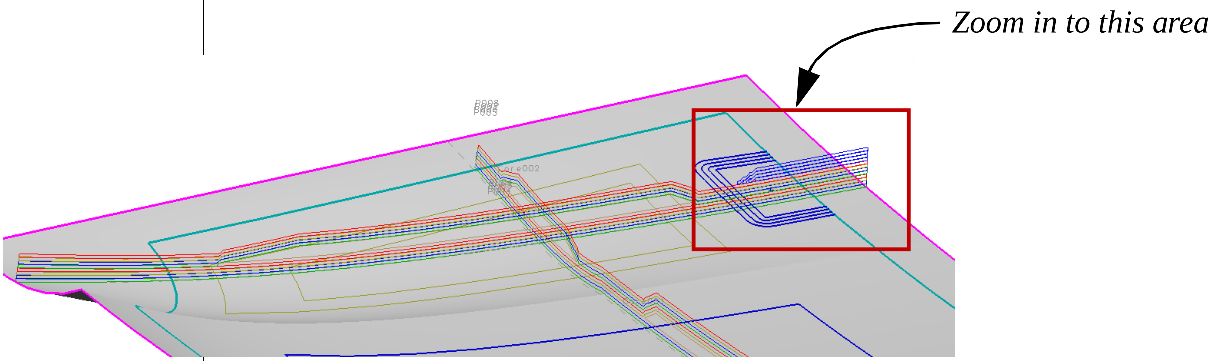


Figure 8–8

10. Activate the Fibersim window.

11. Move the Fibersim window so that the highlighted cross-section displays in the CAD window.

12. With both cross-sections highlighted, click (Generates the 3D cross section). The cross-sections update to reflect the changes made to the stackup.

13. Close the Fibersim window.

Task 4 - Create a Core Sample annotation.

In this task you will create a core sample annotation. Core Sample annotations enable you to display design station results in a 3D text object. Since Design Stations have not been created in the model, you will also need to create one on-the-fly. The model used in this exercise already contains annotations. To create a Fibersim annotation, you must first create an annotation in the 3D model to be populated with the Fibersim data.

1. In the NX Layer Settings window, show the **DESIGN_STATION_ORIGIN** and **ANNOTATIONS**; and hide **CROSS_SECTION_CURVE_1** and **CROSS_SECTION_CURVE2** as shown in Figure 8–9.

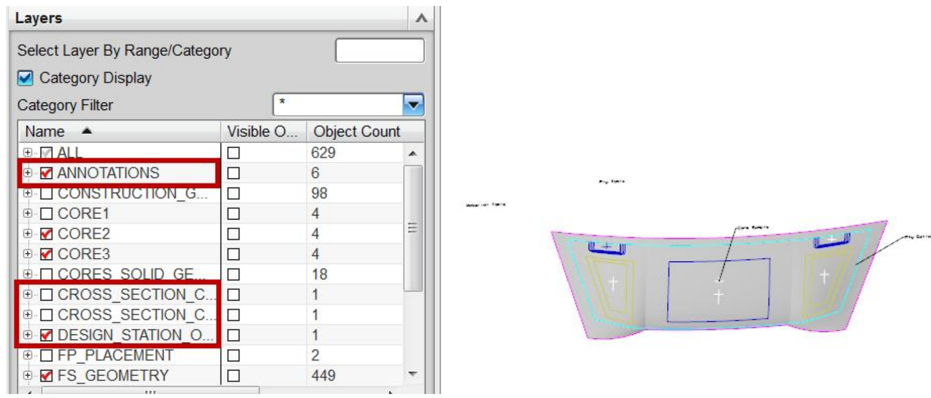


Figure 8–9

2. Click Close.

3. Press <F9> to start Fibersim.

4. In the *Application Tree*, select **Core Sample** under **3D Documentation > Annotations**.

5. Click (Create New).

6. Next to the *Design Station* field, click (Link with Link Dialog).

*You can also right-click in the main Fibersim window and select **Create New.***

7. Click (Create New).

8. In the Laminate drop-down list, select **Hood**.

9. Click (Link Geometry) next to *Origin*.

10. Select **DESIGN STATION ORIGIN** as shown in Figure 8–10 and click OK.

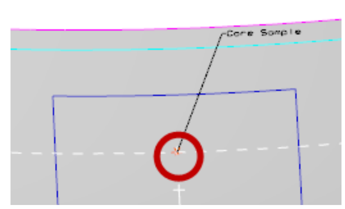


Figure 8–10

11. Click (Core Sample). In the message window, click **OK**.

12. Click **OK** to save the Design Station definition.

13. Ensure that the **DS001** design station that was just created is linked and click **OK** to link this station to the Core Sample.

An annotation must already be created in the model.

14. Next to the *Annotation Leader* field, click (Link Geometry).

15. In the NX window, select the **Core Sample** note and click **OK**.

16. The Fibersim window re-activates. Mouse over the NX window again to update the note, as shown in Figure 8–11.

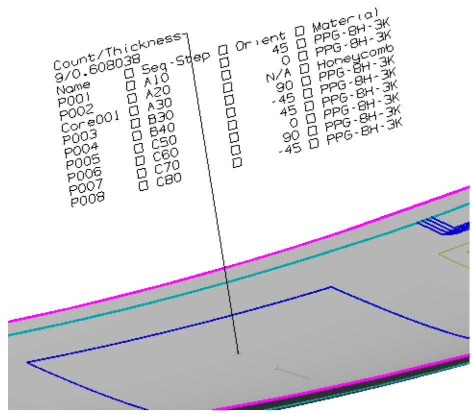


Figure 8–11

17. Click **OK** to save the Core Sample note.

18. Close Fibersim.

If necessary, reposition the note in the NX model to better format the text.

Task 5 - Create a Ply Table annotation.

In this task, you will create an annotation that will display ply and core information on the Hood laminate in a pre-existing 3D text annotation.

1. Press <F9> to start Fibersim.

2. In the *Application Tree*, select **Ply Table** under **3D Documentation > Annotations**.

3. Click (Create New).

4. In the Laminate drop-down list, select **Hood**.

*If the Status of the Ply Table is **Out-of-date**, click*  *(Generates the ply table).*

5. Link *Annotation Leader* to the **Ply Table** annotation. The model updates as shown in Figure 8–12.

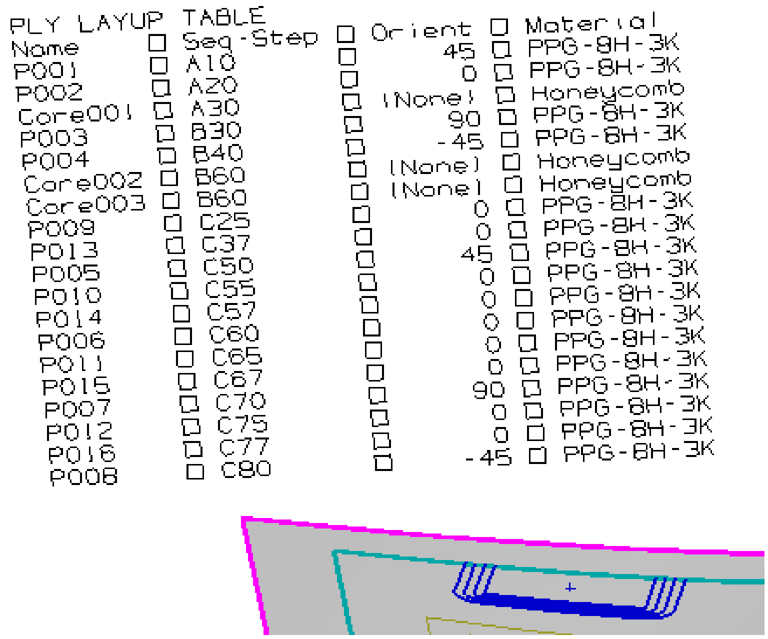


Figure 8–12

6. Click **OK** to save the Ply Table annotation.

Task 6 - Create a Ply Callout annotation.

In this task, you will create an annotation that will display all of the ply and core objects at a specified point on the Hood laminate.

1. In the NX Layer Settings window, show the **PLY_CALLOUT_ORIGIN** layer category.
2. Click **Close**.
3. Press <F9> to start Fibersim.
4. In the *Application Tree*, select **Ply Callout** under **3D Documentation > Annotations**.
5. Click  (Create New).
6. In the Laminate drop-down list, select **Hood**.

The callout point and annotation leader must already exist in the model.

7. Link the following geometry:
   - Annotation Leader: **Ply Callout**
   - Callout Point: **Ply Callout Origin** (See Figure 8–13.)

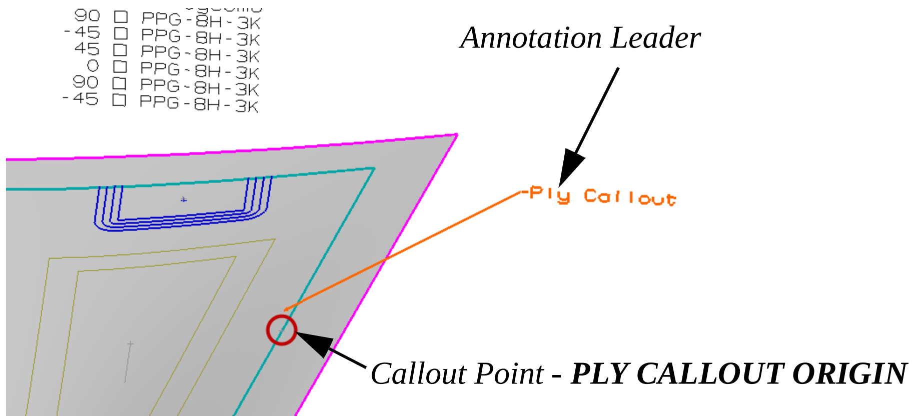


Figure 8–13

8. Enter [3] for *Number Of Names Per Line*.

9. Click  (Generates the callout). In the message window, click 

. The model updates as shown in Figure 8–14.

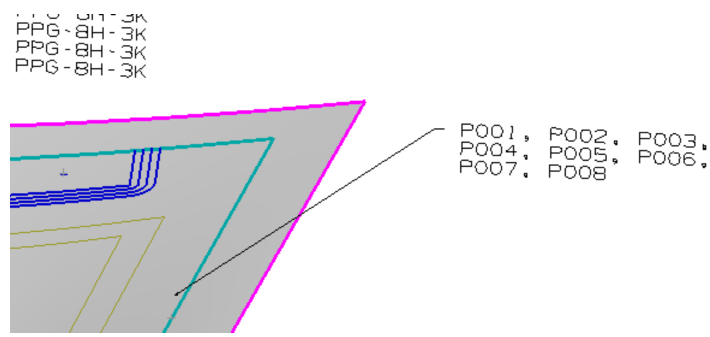


Figure 8–14

10. Click 

to save the Ply Callout annotation.

Task 7 - Create a Material Table annotation.

In this task, you will create an annotation to display all of the materials used in a given laminate(s).

11. In the *Application Tree*, select **Material Table** under **3D Documentation > Annotations**.

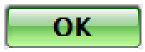
12. Click  (Create New).

13. Next to the *Laminates* field, click  (Link with Link Dialog).

14. Press <Ctrl> + <A> to select all three laminates.

You can also right-click on one of the highlighted laminates and select ***Link*** *or press <Ctrl> + <L>.*

15. Right-click and select  (Link).

16. Click 

.

17. Link the *Annotation Leader* to the **Material Table** annotation. The model updates as shown in Figure 8–15.

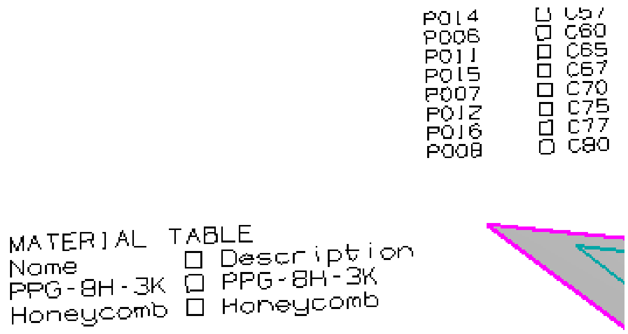


Figure 8–15

18. Click 

to save the Material Table annotation.

19. Close Fibersim and save the model.

Exercise 8b Generating a Ply Book and Ply Table

User Guide Reference:
5.4 Flat Pattern Layout
5.6.1 Ply Book
5.6.2 Ply Table

In this exercise, you will generate a 2D drawing of a composite part. 2D drawings are useful as a method of passing on the necessary information for the manufacturing the part. A sheet from the ply book that is generated in the drawing is shown in Figure 8–16.

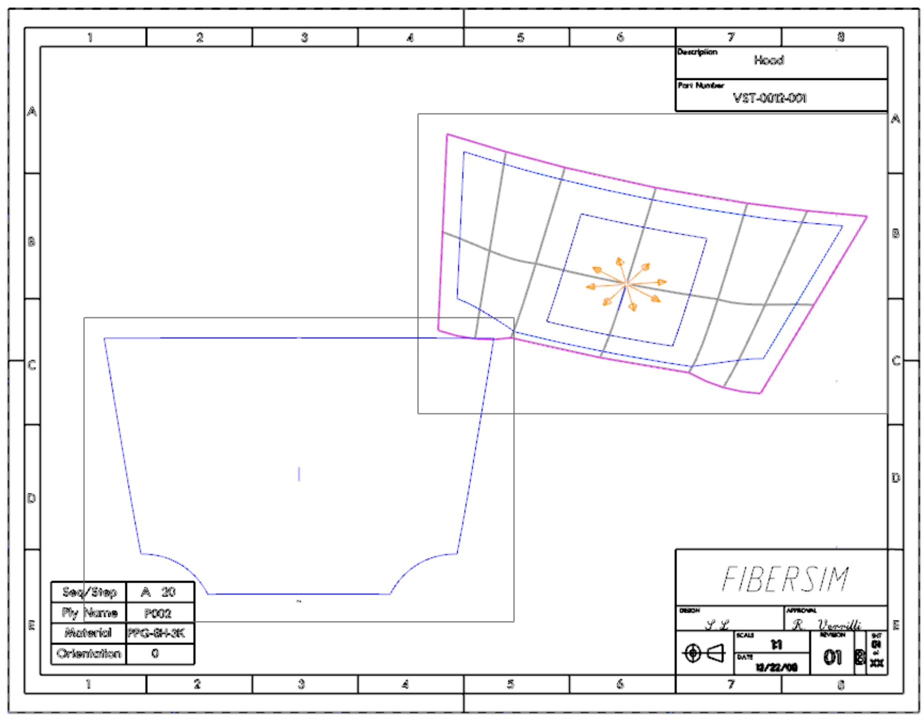


Figure 8–16

Goal

After you complete this exercise, you will be able to:

- ✓ **Utilize Flat Pattern Layout tool**
- ✓ **Create a Ply Book**
- ✓ **Place a Ply Table on a drawing sheet**

Estimated Time

25 min

Task 1 - Open a part.

1. Select **File > Open** and select **DESIGN_AND_ DOC_B.prt**.

   If you completed Exercise 8a, you can continue working with **DESIGN_AND_DOC_A.prt** instead. The model displays as shown in Figure 8–17.

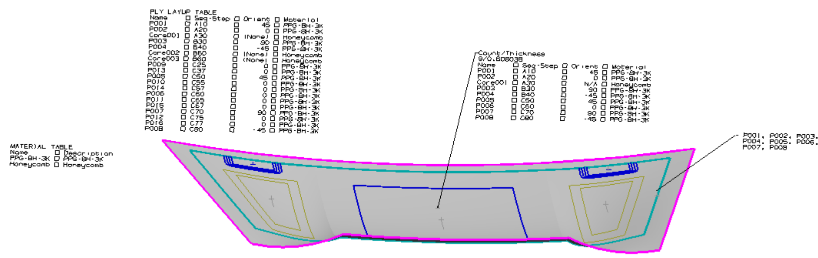


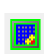
Figure 8–17

2. For clarity, hide **ANNOTATION** layer category in the Layer Settings window.

Task 2 - Create a Flat Pattern layout.

In this task, you will use the flat pattern layout utility to manipulate how the flat patterns display in the CAD window.

*You can also right-click on one of the highlighted plies and select **Modify** or press <Enter>.*

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Ply** under **Source**.
3. Press <Ctrl> + <A> to select all of the plies.
4. Right-click and select **Modify**.
5. Select the *Simulation Options* tab.
6. In the *Fiber Spacing Factor* field, enter [0.6].
7. Click  (Net Producibility). In the message window, click .
8. Click  (Generates the Net Flat Pattern).

9. Click **OK** to save the plies. The model displays as shown in Figure 8–18.

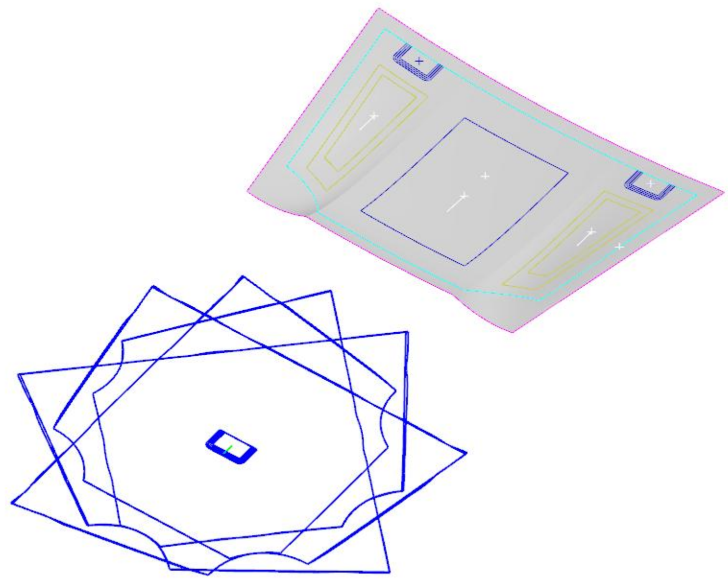


Figure 8–18

Flat patterns need to be created before the flat pattern layout utility is used.

10. In the ply list, highlight all of the plies by pressing <Ctrl> + <A>.

11. In the Ply Toolbar, click (Flat Pattern Layout).

*You can also find **Flat Pattern Layout** in the **Documentation** menu in the Application Tree.*

12. Enter the following parameters, as shown in Figure 8–19:
    - Grid Origin X: **-25 in (-635 mm)**
    - Cell Width: **50 in (1270 mm)**
    - Cell Height: **50 in (1270 mm)**
    - Row Direction: **-Y**
    - Column Direction: **X**

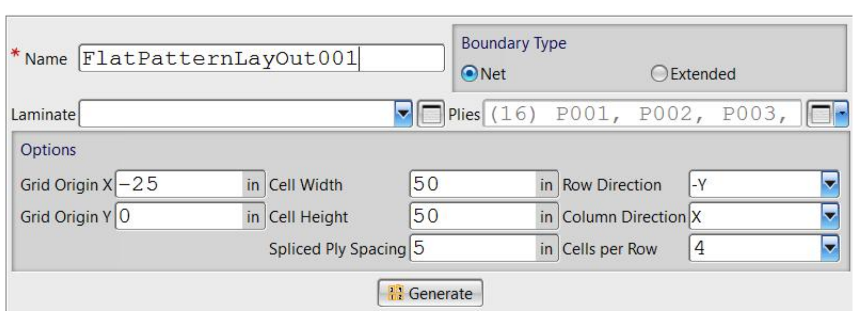


Figure 8–19

13. Click Generate.

14. Click OK and close Fibersim. The model displays as shown in Figure 8–20.

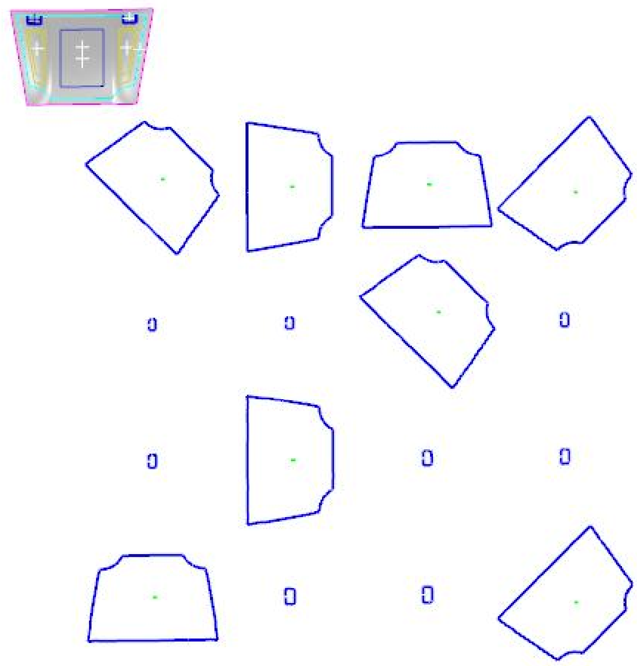


Figure 8–20

Task 3 - Prepare a drawing for ply book insertion.

In this task, you will add two views to an existing drawing. These views will be associated with the flat pattern and boundary diagrams in Fibersim in the next task. In the drawing, the tables, frame, and title block have already been created.

1. In NX, Click **Start > Drafting**.
2. In the *Part Navigator*, double-click on **Sheet "SH1"** under **Drawing**.
3. In NX, press <Ctrl> + <L> to open Layer Settings window.
4. Enter [1] in the field next to *Work Layer*.
5. Click Close .
6. Select **File > Import > Part**.
7. Select **Specify** for *Destination Coordinate System*.

8. Click **OK**.
9. Browse to **Drawing_template_A-Size.prt** in *TEMPLATES_FOR_DES_AND_DOC* folder.
10. Click **OK**.
11. Change the Coordinates to **Absolute - Displayed Part** as shown in Figure 8–21.

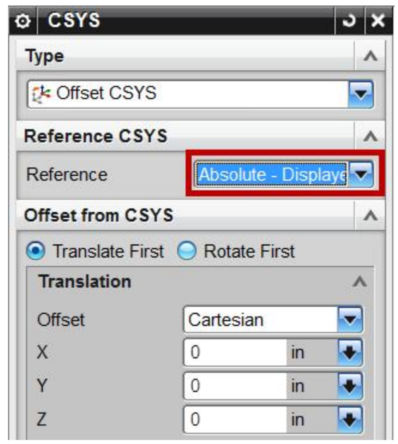


Figure 8–21

12. Click **OK** twice and OK to the warning message about the unit conflict.
13. Click **Cancel** to exit the Import command.
14. In NX, hide all layer categories except for **OML** and **0** layer category in the Layer Settings window, and click **Close**.
15. Press <Ctrl> + <Shift> + <M> to switch to Modeling.
16. Rotate the view to orient the model as shown in Figure 8–22.

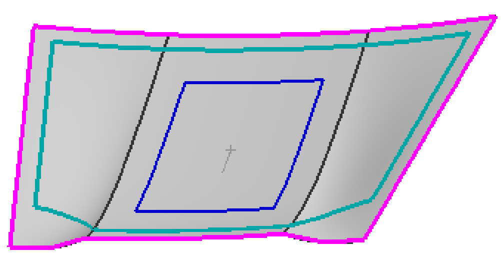


Figure 8–22

17. **[NX8.5]** Select **View > Operation > Save As**.

17. **[NX9/10]** Select **Menu > View > Operation > Save As**.

18. Enter [BOUNDARY-VIEW] in the Name field and click .

19. Press <Ctrl> + <Shift> + <D> to switch to Drafting

20. **[NX8.5]** Select **Insert > View > Base**.

20. **[NX9/10]** Select **Base View** on *Home* tab.

21. For *Model View*, select **BOUNDARY-VIEW** from the pull-down.

22. For *Scale*, select **Ratio** from the pull-down and enter [10] in the second field as shown in Figure 8–23.

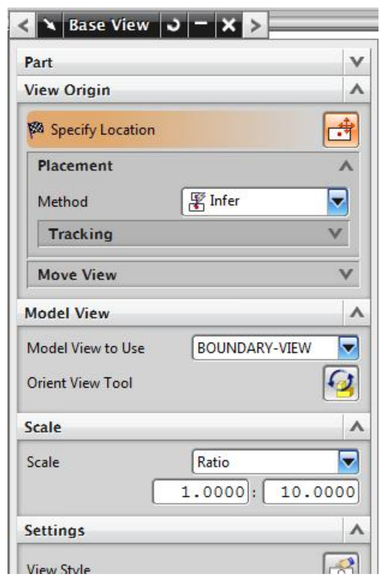


Figure 8–23

23. Place the view as shown in Figure 8–24 and press <Esc> to cancel the projection view creation.

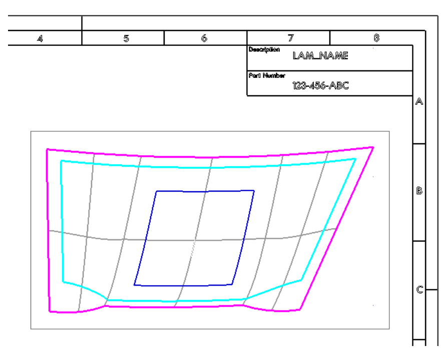


Figure 8–24

24. Back in NX, hide **OML** layer category in the Layer Settings window, and click **Close**.
25. **[NX8.5]** Select **Insert > View > Base**.
25. **[NX9/10]** Select **Base View** on *Home* tab.
26. For *Model View*, leave **TOP** view for *Model View to Use*.
27. For *Scale*, select **Ratio** from the pull-down and enter [10] in the second field.
28. Place the view as shown in Figure 8–25 and press <Esc> to cancel the projection view creation.

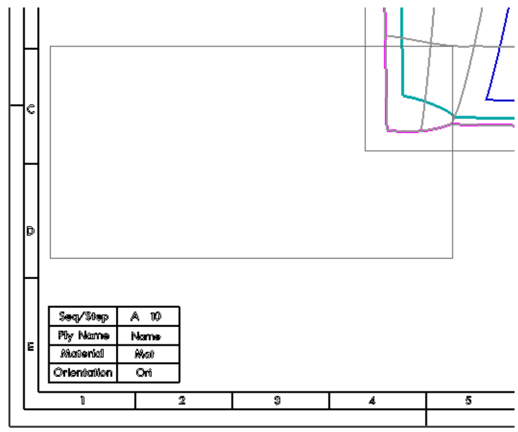


Figure 8–25

Task 4 - Generate the Ply Book.

1. Press <F9> to start Fibersim.
2. In the Application Tree, select **Ply Book** under **2D Documentation**.
3. Click (Create New).
4. In the Laminate drop-down list, select **Hood**.
5. Next to *Source Objects* field, click (Link with Link Dialog).
6. Press <Ctrl> + <A> to select everything.

*You can also right-click on one of the highlighted laminates and select **Link**.*

7. Click (Link).
8. Click **OK**.

9. Next to the *Template Sheet* field, click (Link Geometry).
10. Select the border of the Plybook Template sheet in NX, as shown in Figure 8–26, and click OK.

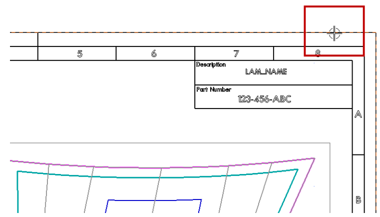


Figure 8–26

11. Select the *Text Setup* tab.
12. In the dialog box, in the *Text Associations* area, click (Create New with Geometry Selection).
13. Select the following text in the drawing, as shown in Figure 8–27:
    a. A
    b. 10
    c. Name
    d. Mat
    e. Ori
    f. LAM_NAME
    g. 123-456-ABC

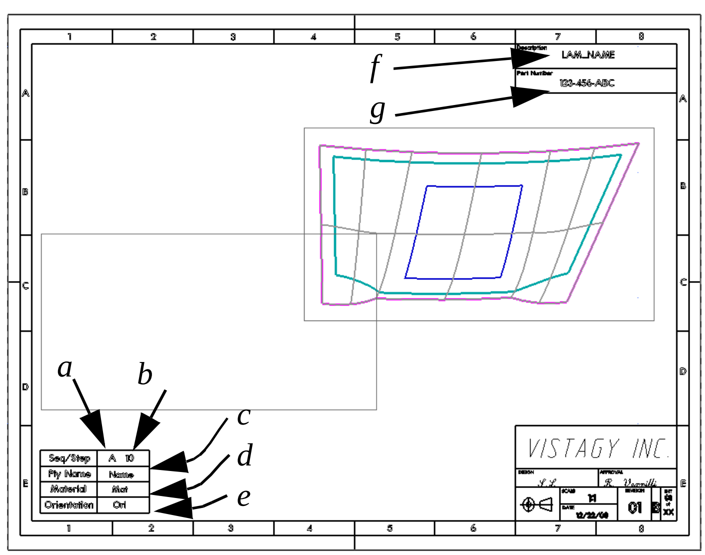
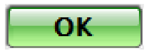


Figure 8–27

14. Click .

15. In the *Text.Seq* row, expand the Source Member drop-down list and select **Sequence**, as shown in Figure 8–28.

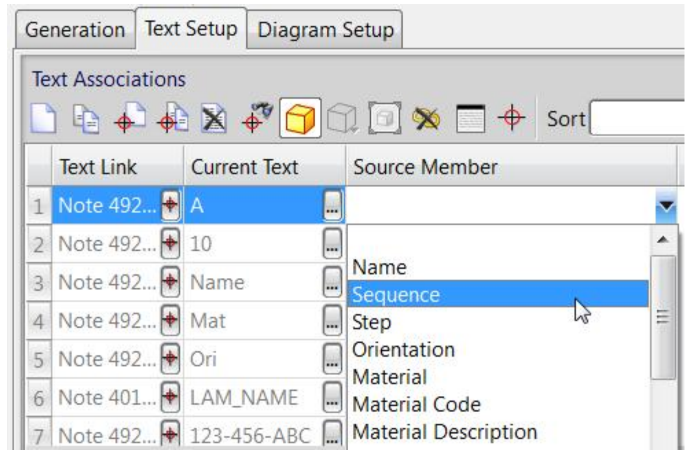


Figure 8–28

16. Enter the remaining Text Links as shown in Figure 8–29:

- 10: **Step**
- Name: **Name**
- Mat: **Material**
- Ori: **Orientation**
- LAM_NAME: **Laminate Name**
- 123-456-ABC: **Laminate Part Number**

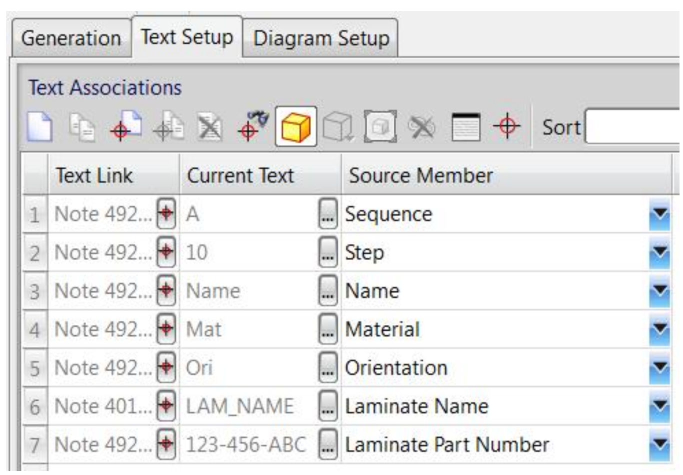


Figure 8–29

17. Select the *Diagram Setup* tab.

18. In the Flat Pattern Views window, select the **Ply Book** object type as shown in Figure 8–30.

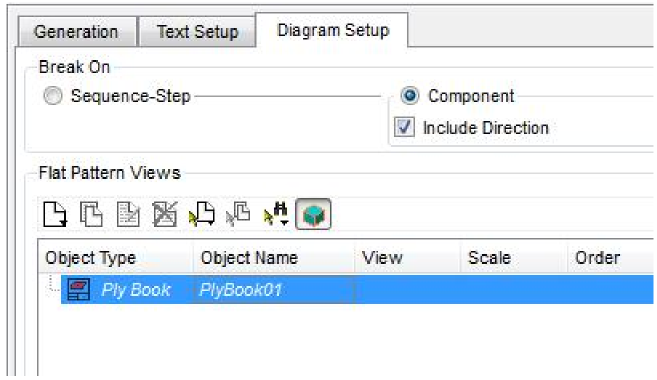


Figure 8–30

19. Click (Create New with Geometry Selection) and select **Flat Pattern Diagram**.

20. In the drawing, select the **Top** view and click OK.

21. Click OK.

22. In the Boundary Views window, select the **Ply Book** object type.

23. Click (Create New with Geometry Selection) and select **Boundary Diagram**.

Multiple flat pattern views can be created on one page if required. In this case, all ply boundaries will display in one boundary view.

24. In the drawing, select the **Boundary-View** and click OK.

25. Click OK.

26. Turn on the option for *Display Layup Information***.** The *Diagram Setup* tab displays as shown in Figure 8–31.

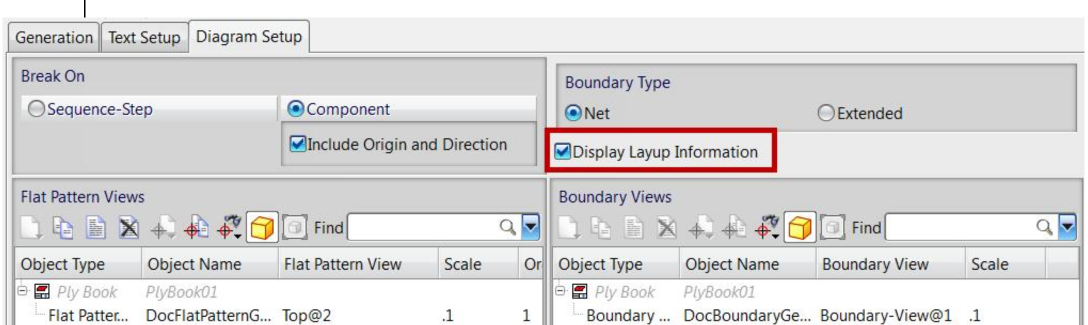


Figure 8–31

27. Click (Generates the sheets for the ply book).

28. Click OK to close the message window. When you generate the Ply book, some of the sheets generate an error. These sheets are views of the core. Since cores do not have a flat pattern associated with them, the flat pattern view cannot be generated.

29. Click OK.

30. Close Fibersim and review the generated sheets.

Task 5 - Create a Ply Table.

1. **[NX8.5]** Select **Insert > Sheet**.

1. **[NX9/10]** Select **New Sheet** on *Home* tab.

2. Select **Standard Size** option and **A** size if not already selected.

3. Enter [Table] in the *Drawing Sheet Name* field.

4. Click OK.

5. Select **File > Import > Part**.

6. Select **Specify** for *Destination Coordinate System*.

7. Click OK.

8. Browse to **Drawing_template_A-Size.prt** in *TEMPLATES_FOR_DES_AND_DOC* folder.

9. Click OK.

10. In the *Reference CSYS* area, select **Absolute - Displayed Part** for the *Reference* field.

11. Click OK.

12. Ensure that the Coordinates is set to **0,0**.

13. Click OK and OK to the warning message about the unit conflict.

14. Click Cancel to exit the Import command.

15. **[NX8.5]** Select **Insert > Table > Tabular Note.**

15. **[NX9/10]** Select **Tabular Note** on *Home* tab.

16. Enter [4] for *Number of Columns*.

17. Place the table in the drawing as shown in Figure 8–32.

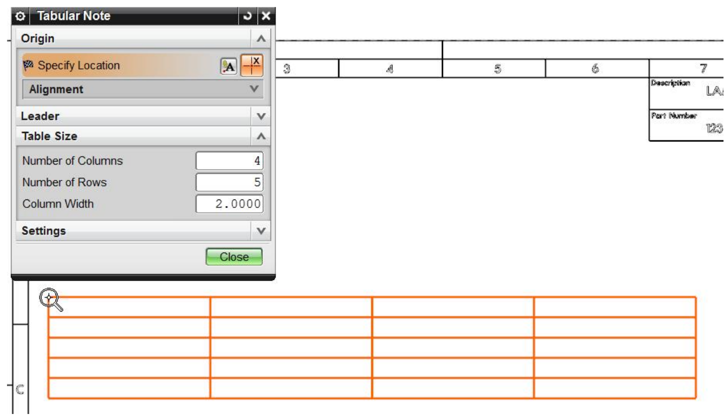


Figure 8–32

18. Click Close.

19. Press <F9> to start Fibersim.

20. In the *Application Tree* menu, select **Ply Table** under **2D Documentation**.

21. Click (Create New).

22. In the Laminate drop-down list, select **Hood**.

23. Next to *Source Objects* field, click (Link with Link Dialog).

24. Press <Ctrl> + <A> to select everything.

*You can also right-click on one of the highlighted laminates and select **Link**.*

25. Click (Link).

26. Click OK.

27. Next to the *Template Sheet* field, click (Link Geometry).

28. Activate the NX window and select the border of the drawing, as shown in Figure 8–33.

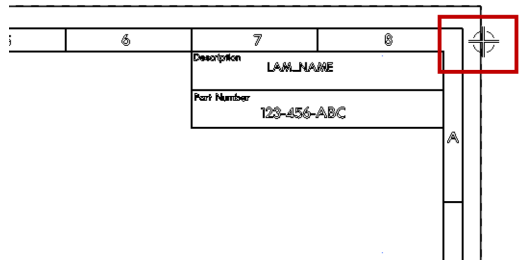
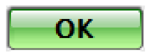


Figure 8–33

29. Click  .

30. Select the *Table Setup* tab.

31. In the *Table Associations* area as shown in Figure 8–34, click  (Create New with Geometry Selection).

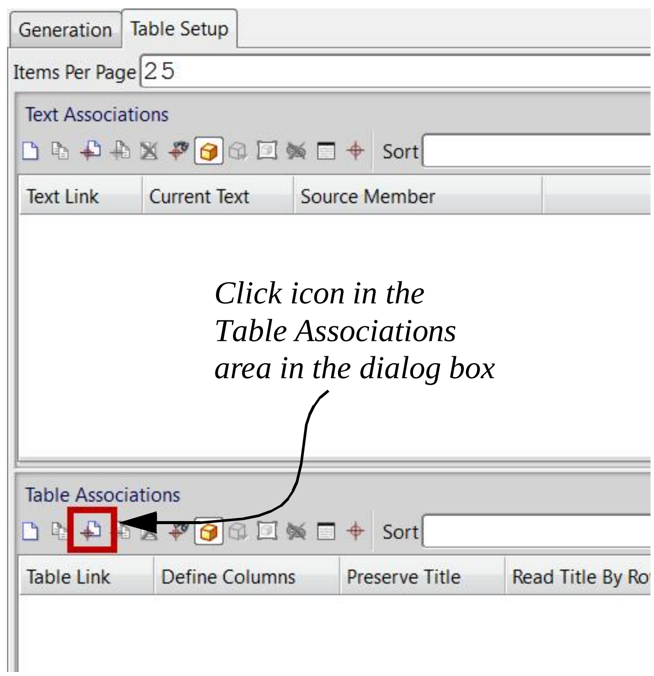
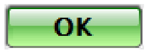


Figure 8–34

32. In the NX window, select the table as shown in Figure 8–35, and click  .

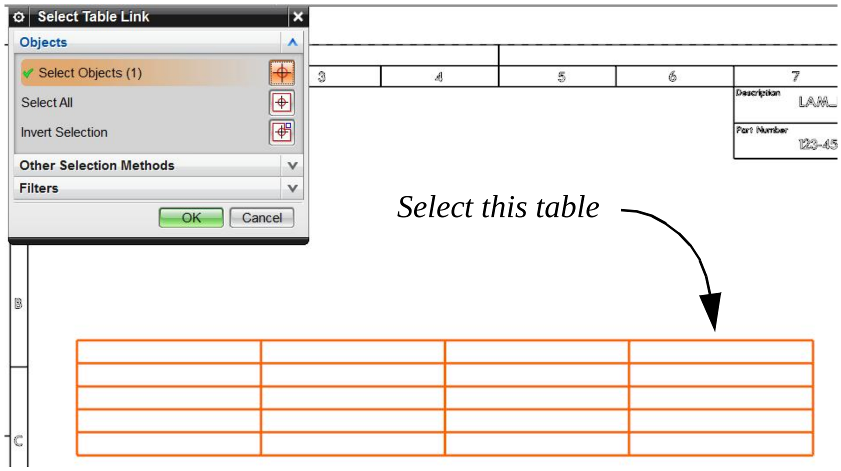


Figure 8–35

33. In the Table.1 row, click  as shown in Figure 8–36.

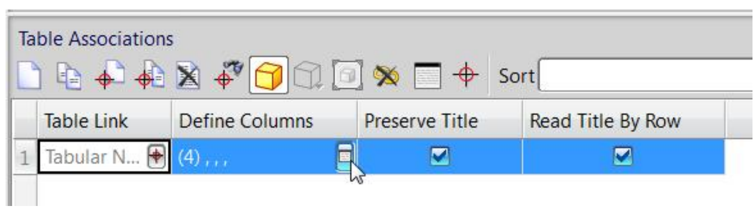


Figure 8–36

34. Double-click on the first row to modify it.

35. Enter [ORIENTATION] in the *Column Title* field.

36. In the Data Field drop-down list, select **Orientation** and click  .

37. Repeat Steps 34 through 36 for each of the other rows, as shown in Figure 8–37.

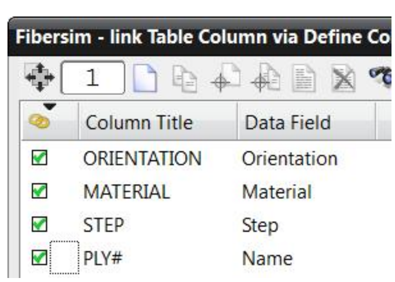


Figure 8–37

38. Click OK to close the Fibersim - link Table Column via Define Columns dialog box.

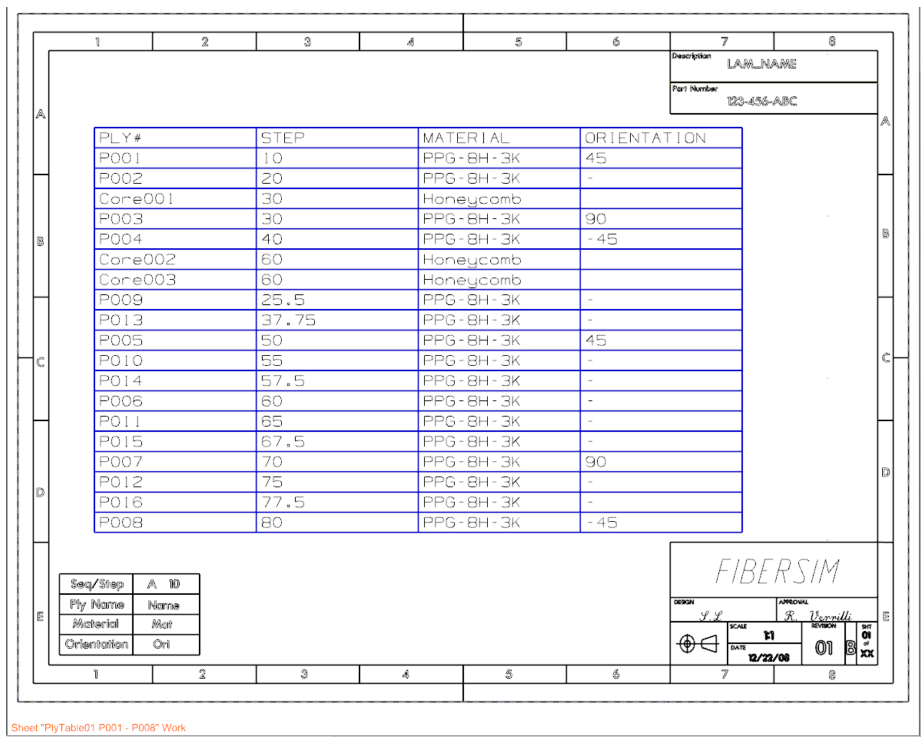
39. Click (Updates or generates the sheets).

40. Click OK in the message window.

41. Click OK to save the Ply Table.

42. Close Fibersim.

43. Activate the NX window. The last sheet of the drawing contains the Ply Table that was created, as shown in Figure 8–38.



PLY#	STEP	MATERIAL	ORIENTATION
P001	10	PPG-8H-3K	45
P002	20	PPG-8H-3K	-
Core001	30	Honeycomb	
P003	30	PPG-8H-3K	90
P004	40	PPG-8H-3K	-45
Core002	60	Honeycomb	
Core003	60	Honeycomb	
P009	25.5	PPG-8H-3K	-
P013	37.75	PPG-8H-3K	-
P005	50	PPG-8H-3K	45
P010	55	PPG-8H-3K	-
P014	57.5	PPG-8H-3K	-
P006	60	PPG-8H-3K	-
P011	65	PPG-8H-3K	-
P015	67.5	PPG-8H-3K	-
P007	70	PPG-8H-3K	90
P012	75	PPG-8H-3K	-
P016	77.5	PPG-8H-3K	-
P008	80	PPG-8H-3K	-45

Figure 8–38

44. Save and close the drawing and model.

Project 3

Core Panel

This chapter includes:

✓**Exercise P3: Core Panel**

Exercise P3 Core Panel

User Guide Reference:
2.9 Cores
2.5 Flat Pattern/
Producibility Simulation
B.3.7 Splice Ply

In this project, you will use both plies and modeled cores to complete a layup scenario. The completed model displaying the four cores is shown in Figure P3–1.

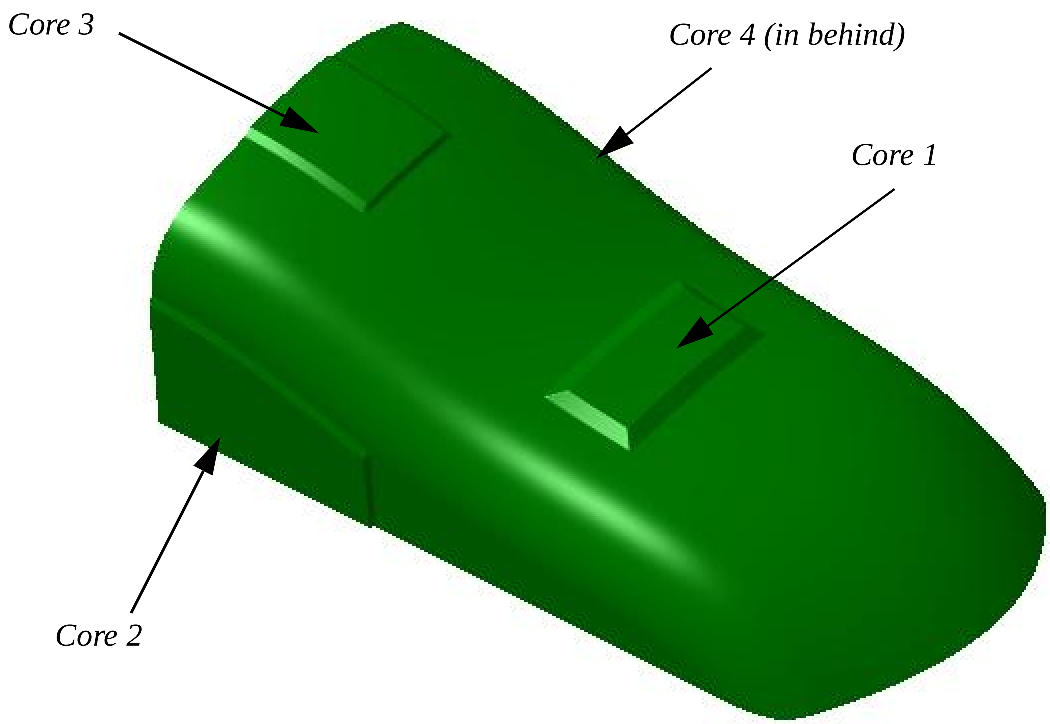


Figure P3–1

Goal

This project will test your knowledge on how to:

- ✓ **Create Ply and Modeled Core objects**
- ✓ **Define the Overcore laminate**
- ✓ **Run Net Producibility**
- ✓ **Splice plies to address material width producibility problem**
- ✓ **Create a Net Flat Pattern**

Given

The existing geometry and design details are:

- ✓ **Net and Extended Boundaries have been created**
- ✓ **OML laminate has been created**
- ✓ **Rosette geometry has been created**
- ✓ **Core CAD geometry has been created**
- ✓ **Overcore CAD geometry has been created**

Estimated Time

1 hr 30 min

Task 1 - Open a part.

1. Open **PROJECT3_CORE.prt**. The model displays as shown in Figure P3–2.

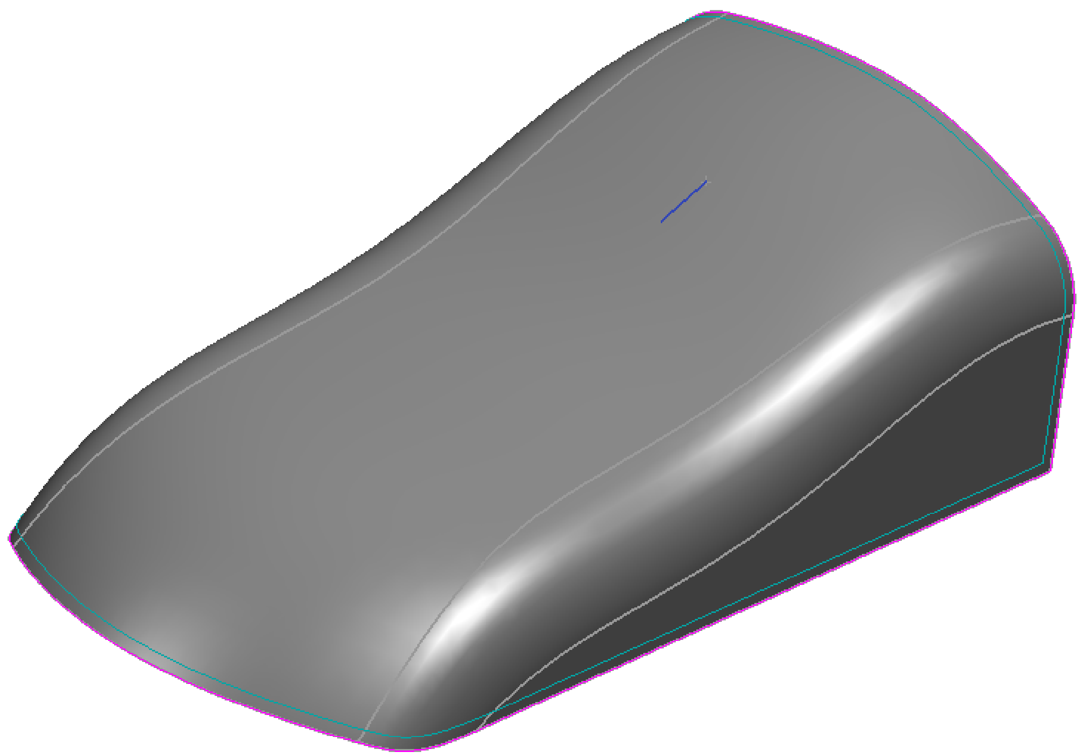


Figure P3–2

Task 2 - Review the model.

1. Review the setup of layers by showing/hiding each category as shown in Figure P3–3. (Make sure that the Category Display option is turned on.)

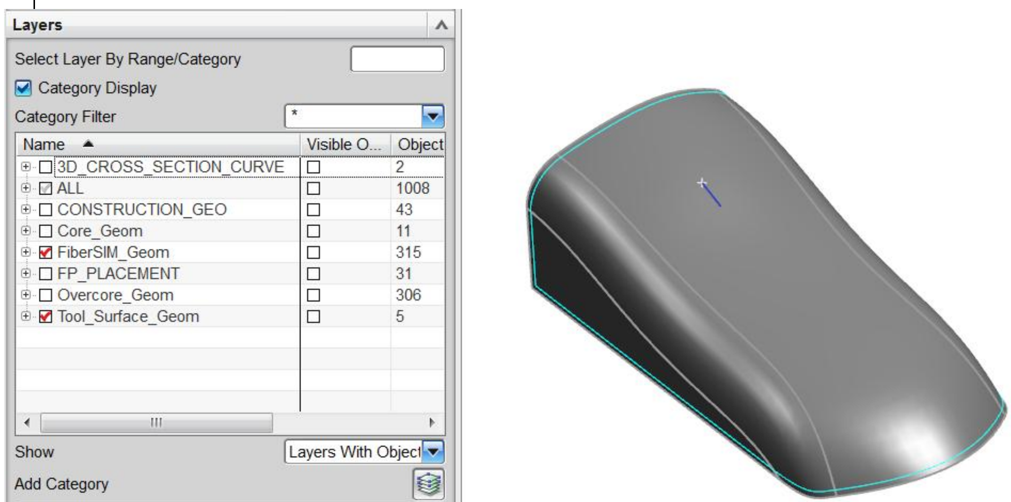


Figure P3–3

Task 3 - Create Ply and Core objects

1. When creating an OML laminate, select the sheet edge of the model for the extended boundary.
2. The ply details on the OML surface are:
   - Four full plies with a material of PPG-PL-3K with Specified Orientation of 0, +45, 90, and -45.
   - One adhesive ply with a material of Film-Adhsv
3. The Core material details are:
   - Core 1: **Honeycomb**
   - Core 2: **Foam-3.1**
   - Core 3: **Honeycomb**
   - Core 4: **Foam-3.1**

Task 4 - Cross section details

1. The cross section for Core 1 is shown in Figure P3–4.

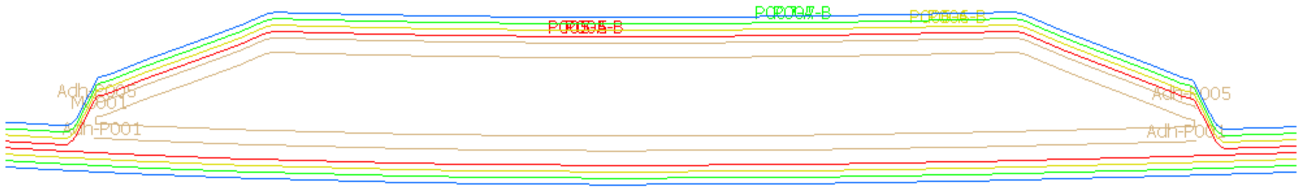


Figure P3–4

2. The cross sections for Core 2 and 4 are shown in Figure P3–5.

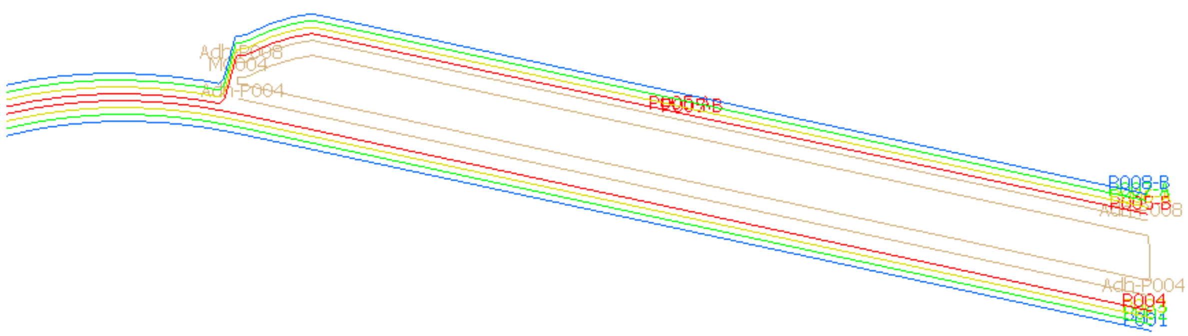


Figure P3–5

3. The cross section for Core 3 is shown in Figure P3–6.

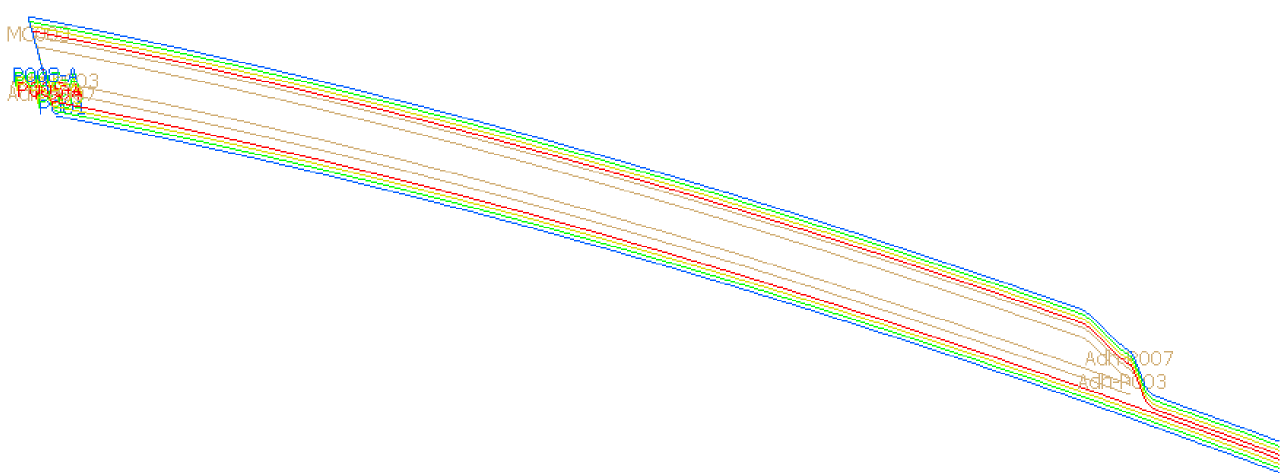


Figure P3–6

Task 5 - Define the Overcore laminate.

Create the Overcore laminate using the Overcore CAD geometry.

- Select the sheet edges of **Overcore_Surface** for the extended boundary of Overcore laminate.

Task 6 - Create four plies on the Overcore surface.

Create one adhesive ply and four full plies with Specified Orientation of -45, 90, +45, and 0 on the **Overcore** surface.

Task 7 - Confirm layup.

Create cross sections using the provided CAD geometry to confirm that the layup matches the design requirements.

Chapter 9

Introduction to Multi-Ply Design

This chapter includes:

✓**Exercise 9a: Basic Multi-Ply Design**

Exercise 9a Basic Multi-Ply Design

User Guide Reference:
3.3.2 Overlay Zone
3.7 Offset Specifications
3.8 Laminate Specifications
3.9 Material Specifications

In this exercise, you will learn the basics of Multi-Ply design on a simple composite part.

Goal

After you complete this exercise, you will be able to:

- ✓ **Define simple Overlay Zones**
- ✓ **Generate Layers from Overlay Zones**
- ✓ **Create Material Specification**
- ✓ **Create Laminate Specification**

Estimated Time

20 min

Task 1 - Open a part.

1. Open **MULTI_PLY_A.prt**. The model displays as shown in Figure 9–1.

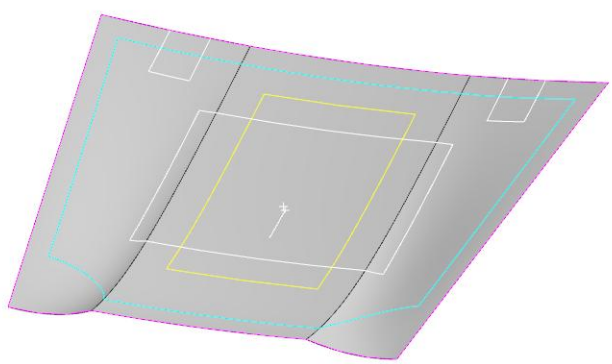


Figure 9–1

2. Press <F9> to start Fibersim.

3. From the Application Browser, select **Advanced** as shown in Figure 9–2.

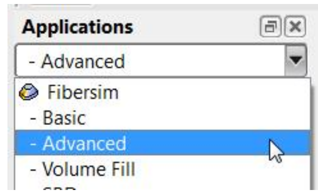


Figure 9–2

Task 2 - Review the existing Laminate.

1. In the *Application Tree*, select **Laminate** under **Design** as shown in Figure 9–3.

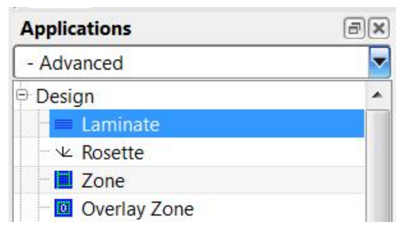


Figure 9–3

2. Double-click **Hood** from the object list to modify.
3. Note that the Design Boundary is set to **Extended** as shown in Figure 9–4.

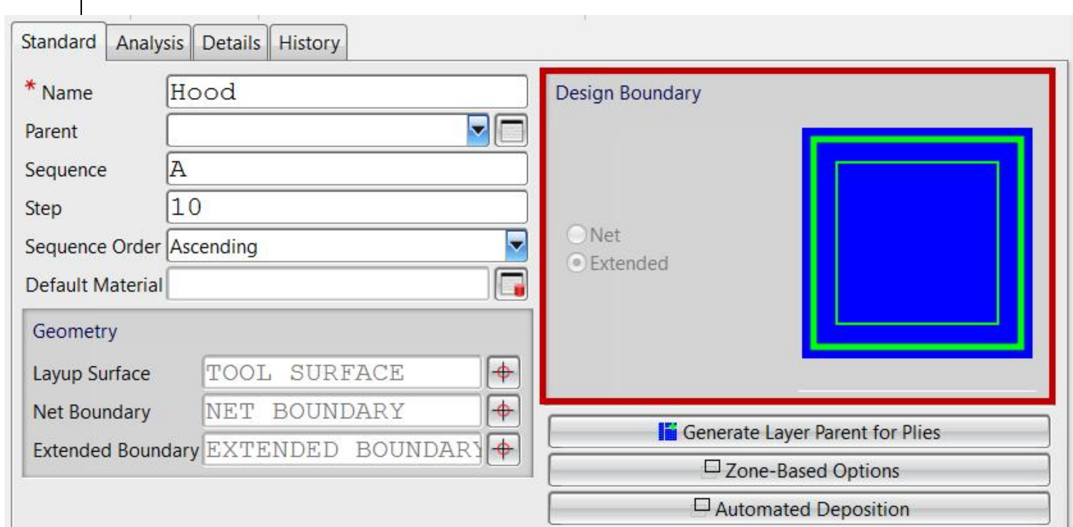
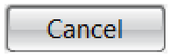


Figure 9–4

4. Click 

to close without saving.

Task 3 - Define Full Body Overlay Zone.

1. In the *Application Tree*, select **Overlay Zone** under **Design** as shown in Figure 9–5.

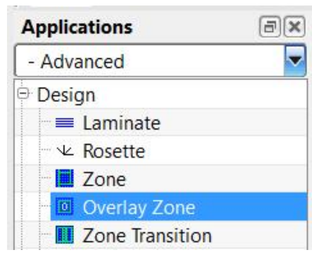


Figure 9–5

2. Click  (Create New).
3. *Create Overlay Zone* window is displayed as shown in Figure 9–6.

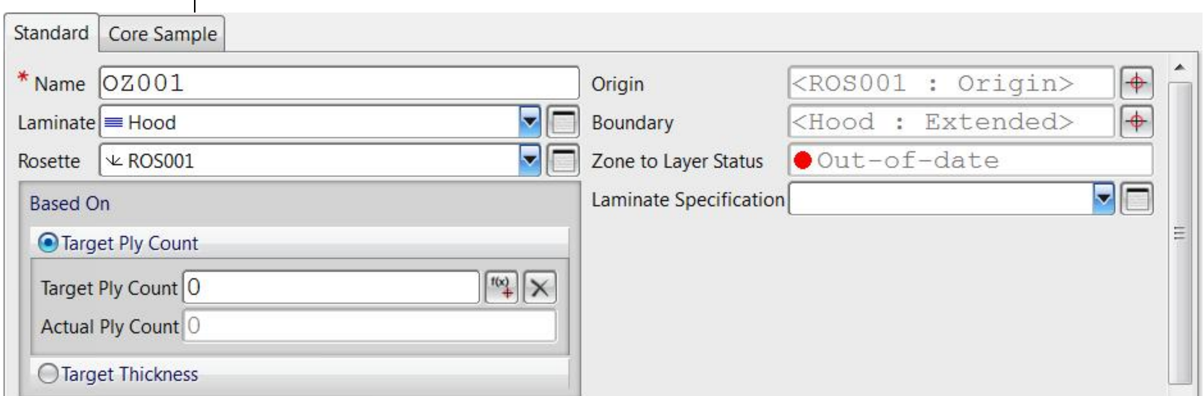
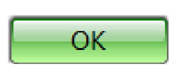


Figure 9–6

4. Enter [4] for *Target Ply Count*.

Task 4 - Create Material Specifications for OZ001.

1. Under *Material Specifications*, click  (Create New).
2. Click  (Link with Database Link Dialog) for *Material*.
3. Link **PPG-PL-3K** and click 

.
4. Select [+/-45] from the pull-down list for *Orientation*.

5. Enter [2] for *Ply Count*.
6. Right-click on **MS005** and select Create Based On as shown in Figure 9–7.

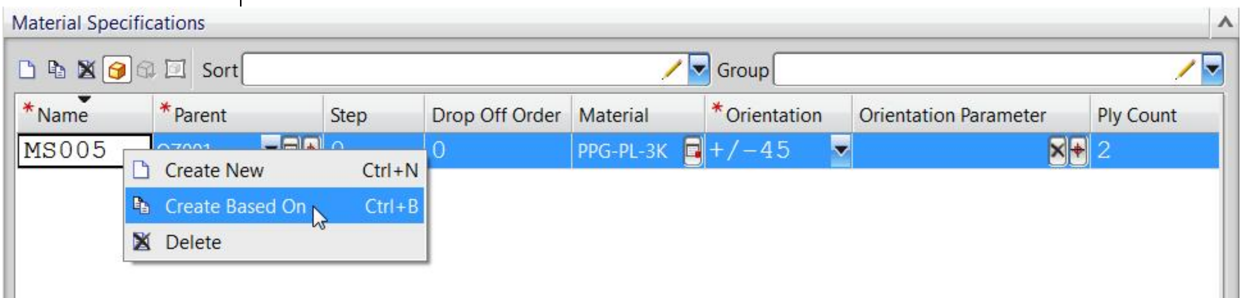


Figure 9–7

7. Select [0/90] from the pull-down list for **MS006**'s *Orientation*.
8. Click  to save **OZ001**.

Task 5 - Define Core Overlay Zone.

1. Click  (Create New).
2. Enter [Core] for *Name*.
3. Link the **Core** curve for *Boundary* as shown in Figure 9–8.

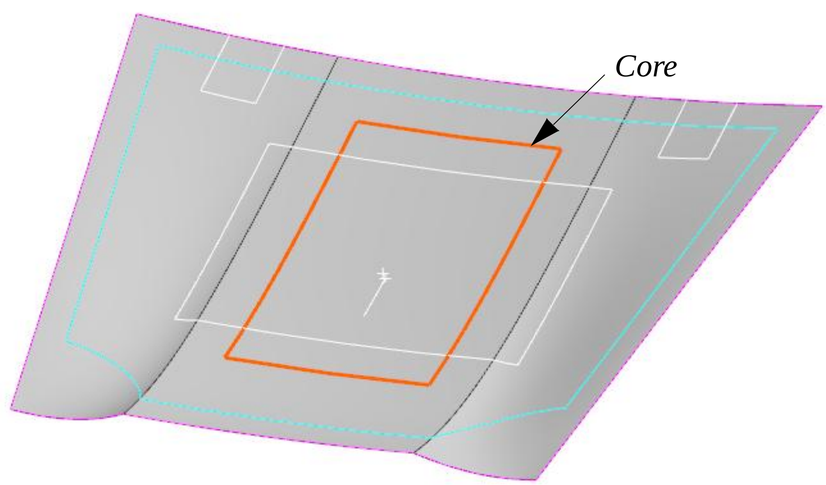


Figure 9–8

4. Enter [1] for *Target Ply Count*.

Task 6 - Create Material Specifications for Core.

1. Under *Material Specifications*, click (Create New).
2. Click (Link with Database Link Dialog) for *Material*.
3. Link **Foam-3.1** and click OK.
4. Enter [.5] (12.7 mm) for *Core Material Thickness*.
5. Click OK to save **Core**.

Task 7 - Define Center Pad Overlay Zone.

1. Click (Create New).
2. Enter [Center Pad] for *Name*.
3. Link **CENTER_PAD** for *Boundary* as shown in Figure 9–9:

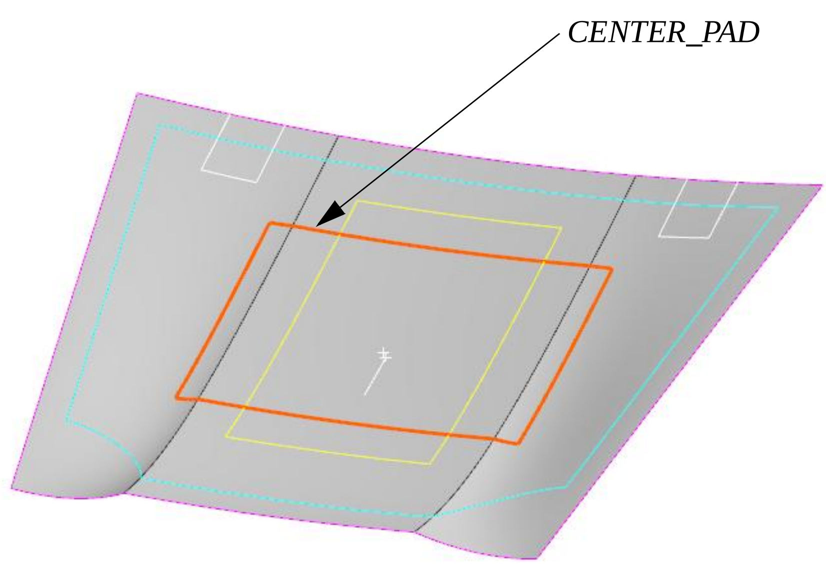


Figure 9–9

4. Enter [3] for *Target Ply Count*.

Task 8 - Create Laminate Specification for Center Pad.

1. For *Laminate Specification*, click (Link with Link Dialog).
2. Click (Create New).
3. Enter [3p] for *Name*.
4. Select **Red** for *Color*.
5. Click (Create New) to create a new Material Specification.
6. Click (Link with Database Link Dialog) for *Material*.
7. Link **T-24-in** from the list.
8. Click OK.
9. Create two more Material Specifications **based on** the first Material Specification.
10. Enter [45] and [-45] for the newly created Material Specifications *Orientation* as shown in Figure 9–10.

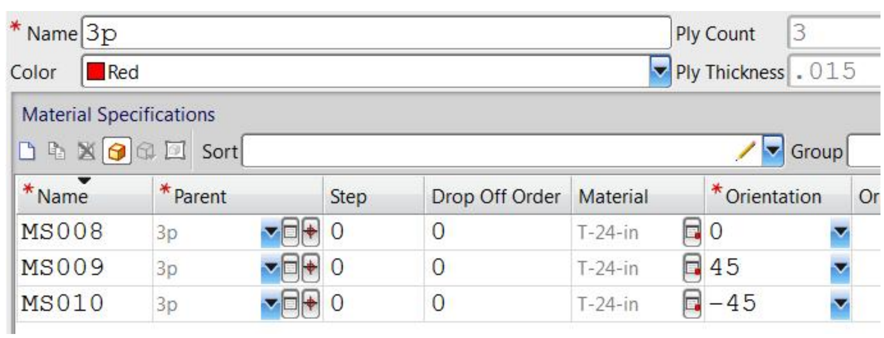


Figure 9–10

11. Click OK to save **3p**.
12. Make sure that **3p** is linked and click OK.
13. Click OK to save **Center Pad**.

Task 9 - Generate Layers.

1. Click (Zone to Layer Analysis) in the Overlay Zone toolbar as shown in Figure 9–11.

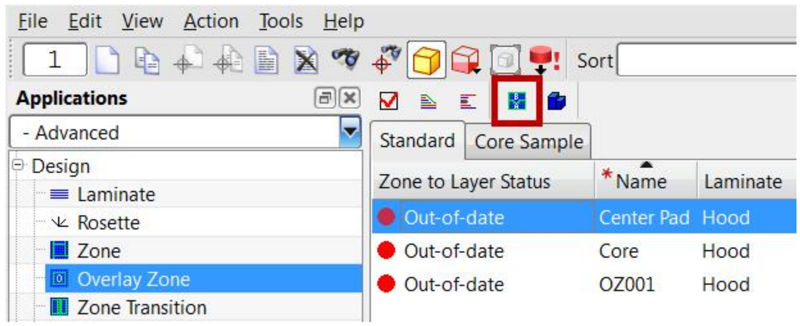


Figure 9–11

2. Click Generate.
3. Click OK to close the message window.
4. Review the report.
5. Click OK to close the Zone to Layer Analysis window.
6. In the *Application Tree*, select **Layer** under **Design**.

This part is designed to the extended boundary.

7. Click (Highlight Type) on the main toolbar as shown in Figure 9–12.

Figure 9–12

8. Make sure that **Extended Boundary** is checked on.

Press <Ctrl> + <A> to select all of the layers.

9. Multi-select all layers to highlight. The model displays as shown in Figure 9–13.

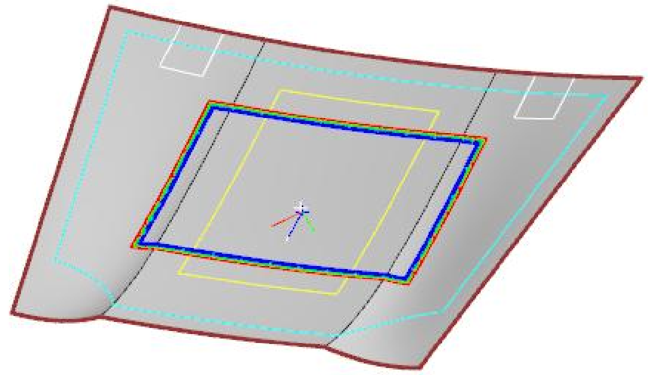


Figure 9–13

Task 10 - Generate Plies.

1. Activate Fibersim window, and make sure that all layers are selected.

2. Click (Generate Plies) in the Layer toolbar as shown in Figure 9–14.

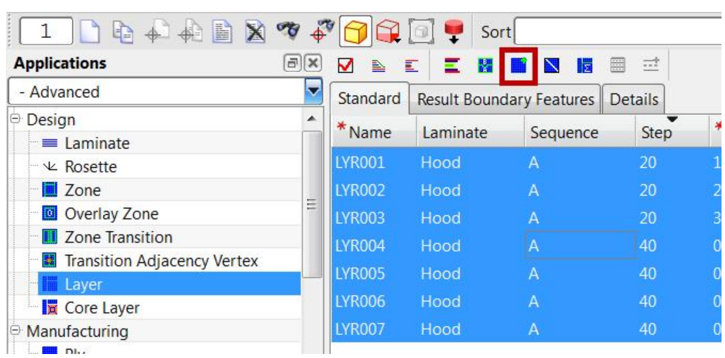


Figure 9–14

3. Click OK to close the message window.

4. In the *Application Tree*, select **Ply** under **Manufacturing**.

5. Review the generated plies.

Task 11 - Generate a Core.

1. In the *Application Tree*, select **Core Layer** under **Design**.
2. Select **CLYR001** from the list. The model displays as shown in Figure 9–15.

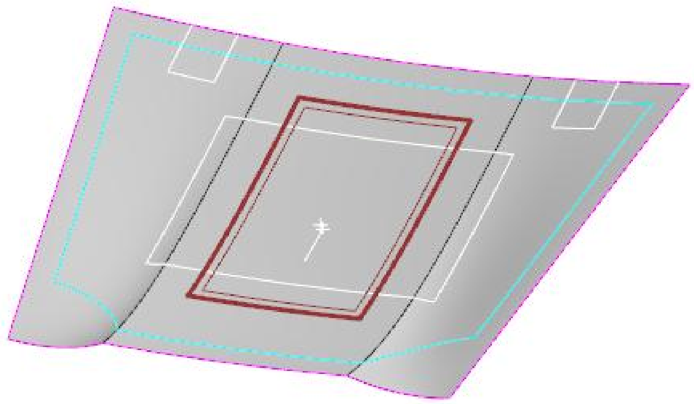


Figure 9–15

3. Click  (Generate Core) in the Core Layer toolbar.
4. Click 

to close the message window.
5. In the *Application Tree*, select **Core** under **Manufacturing**.
6. Review the generated core.

Task 12 - Define Latch1 and Latch2.

1. In the *Application Tree*, select **Overlay Zone** under **Design**.
2. Enter [2] in the object counter as shown in Figure 9–16.

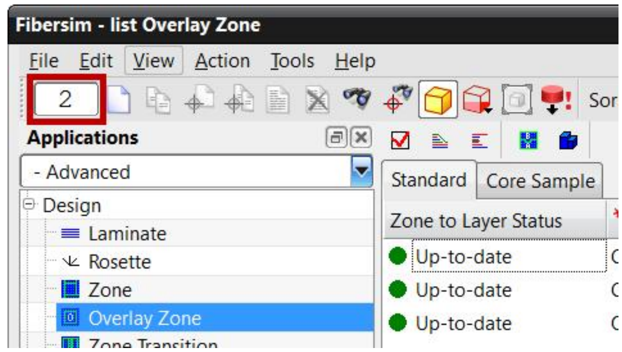


Figure 9–16

3. Click  (Create New).
4. For *Name*, enter [Latch1,1].
5. For Laminate Specification, select **4p** from the pull-down list.
6. Enter [4] for *Target Ply Count*.
7. Press <Ctrl> + <T> to switch to Table View.
8. Right-click on one of the columns and turn on **Origin** and **Boundary** columns as shown in Figure 9–17.

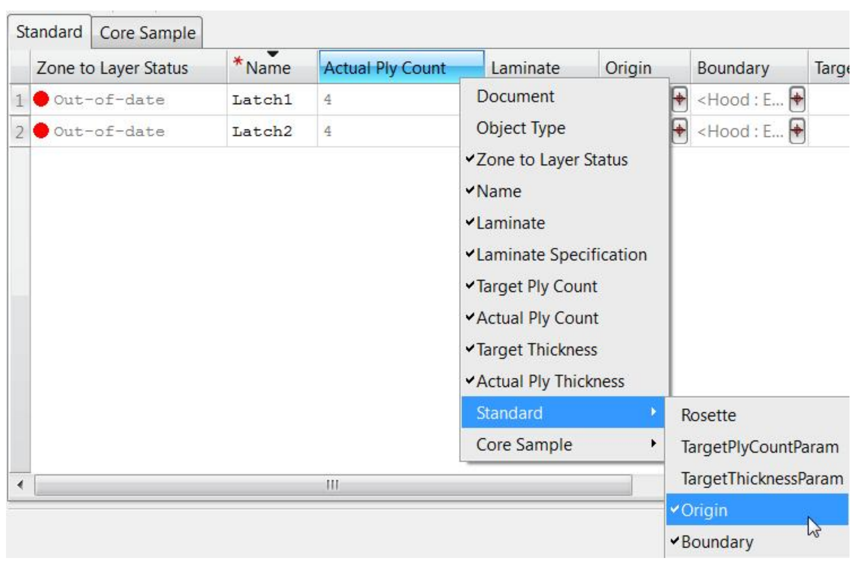


Figure 9–17

9. Click (Link Geometry) for Latch1’s *Origin*.
10. Turn on the **Point on Face** selection filter.
11. Select a point in the Latch1’s boundary as shown in Figure 9–18.

Figure 9–18

12. Click OK.
13. Click (Link Geometry) for Latch1’s *Boundary*.
14. Link **LATCH1** curve as shown in Figure 9–19.

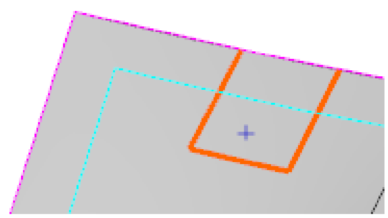


Figure 9–19

15. Click OK.
16. Repeat steps 9 through 15 to assign Origin and Boundary as shown in Figure 9–20.

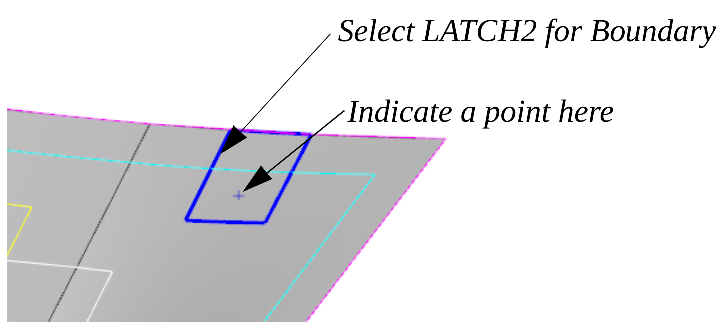


Figure 9–20

17. Click OK to save **LATCH1** and **LATCH2**.

Task 13 - Update Layers.

1. Click (Zone to Layer Analysis) in the **Overlay Zone** toolbar.
2. Click Generate.
3. Close the message window and review the report.
4. Click OK to close the Zone to Layer Analysis window.
5. In the *Application Tree*, select **Layer** under **Design**.
6. Review the generated layers.

Task 14 - Generate Plies.

1. Multi-select all Layers in the list and click (Generate Plies) as shown in Figure 9–21.

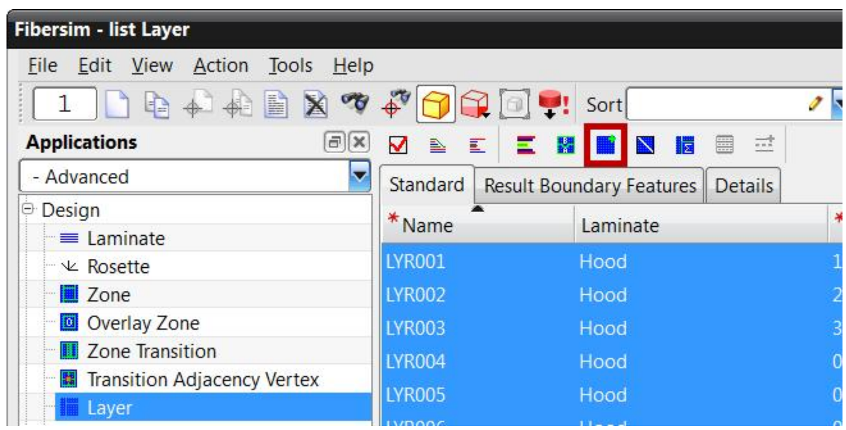


Figure 9–21

2. Wait until confirmation message opens and then click OK to close the message.
3. In the *Application Tree*, select **Ply** under **Manufacturing**. Note that 15 Plies have been created from the Layers.
4. Review the generate plies.
5. Close Fibersim and save the model.

Chapter 10

Transition Manipulation (Multi-Ply Based Design)

This chapter includes:

- ✓**Exercise 10a: Basic Offset Specification Options**
- ✓**Exercise 10b: Splitting and Extending Transitions**

Exercise 10a Basic Offset Specification Options

Userguide Reference:
3.7 Offset Specifications

In this exercise, you will move Zone Transitions by adding and modifying Offset Specifications to meet the design requirements below:

- A transitions must be centered about the boundary curve
- B transitions must be distributed along the surface
- C transitions must be offset from the tangent edges by 5 mm.

The completed model is shown below.

Figure 10–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Use Initial Offset to move the zone transitions**
- ✓ **Use Center Based Offset to center the zone transitions**
- ✓ **Use Offset Distance to modify the spacing of the zone transitions**
- ✓ **Use Fill to Curve option**

Estimated Time

15 min

Task 1 - Open a part.

1. Open **MULTI_TRANSITION_MAN_A.prt**. The model displays as shown in Figure 10–2.

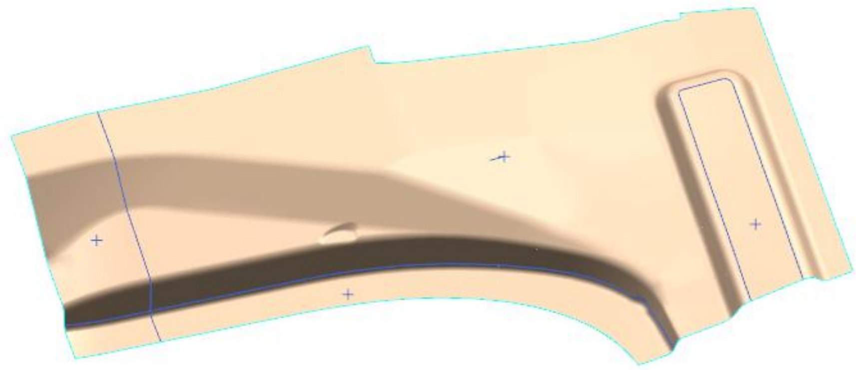
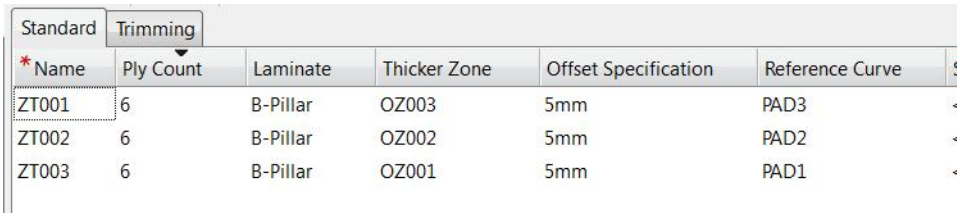


Figure 10–2

Task 2 - Review the Existing Geometry.

1. Press <F9> to start Fibersim.
2. In the *Application Tree*, select **Overlay Zone** under **Design**. Review the existing Overlay Zones in the model.
3. In the *Application Tree*, select **Zone Transition** under **Design**. Note that all zone transitions are currently using the **5 mm** as shown in Figure 10–3.



Standard | Trimming

*Name	Ply Count	Laminate	Thicker Zone	Offset Specification	Reference Curve
ZT001	6	B-Pillar	OZ003	5mm	PAD3
ZT002	6	B-Pillar	OZ002	5mm	PAD2
ZT003	6	B-Pillar	OZ001	5mm	PAD1

Figure 10–3

4. Multi-select all of the Zone Transitions using <Ctrl> + <A>.

5. Activate the NX window. The zone transitions display as shown in Figure 10–4.

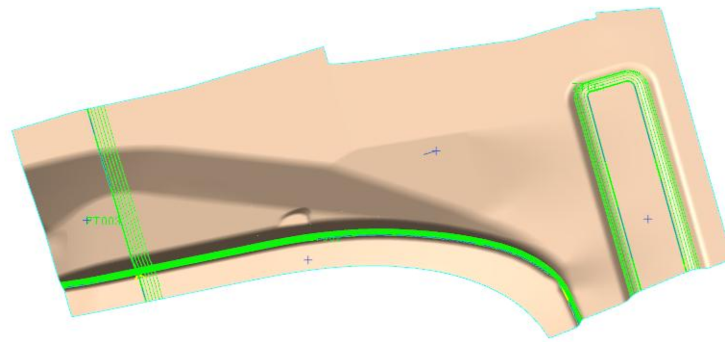


Figure 10–4

6. In the *Application Tree*, select **Layer** under **Design**.
7. Multi-select all layers.
8. Activate the NX window. The layers display as shown in Figure 10–5.

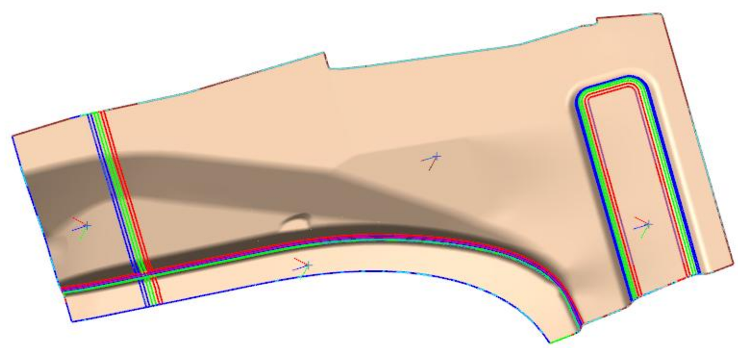


Figure 10–5

Task 3 - Create a new Offset Specification with Centered option.

In this task you will modify the zone transitions to center them by the OZ001 boundary.

1. In the *Application Tree*, select **Offset Specification** under **Specification**.
2. Right-click on **5mm** and select **Create Based On**.

3. Enter [5mm Centered] for *Name*.

4. Click Advanced Positioning in the *Transition Positioning* area.

5. Select **Center Based** as shown in Figure 10–6.

Figure 10–6

6. Click Close.

7. The Offset Specification form updates as shown in Figure 10–7.

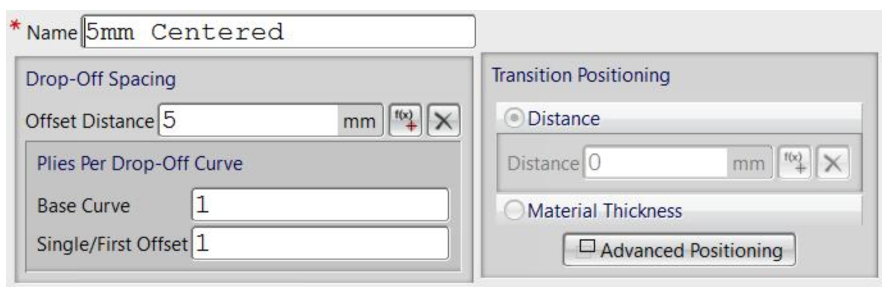


Figure 10–7

8. Click OK to save **5mm Centered**.

Task 4 - Apply the new Offset Specification to a Zone Transition.

1. In the *Application Tree*, select **Zone Transition** under **Design**.

2. Double-click on **ZT003** to modify.

3. Link **5mm Centered** for Offset Specification as shown in Figure 10–8. Close the confirmation message window.

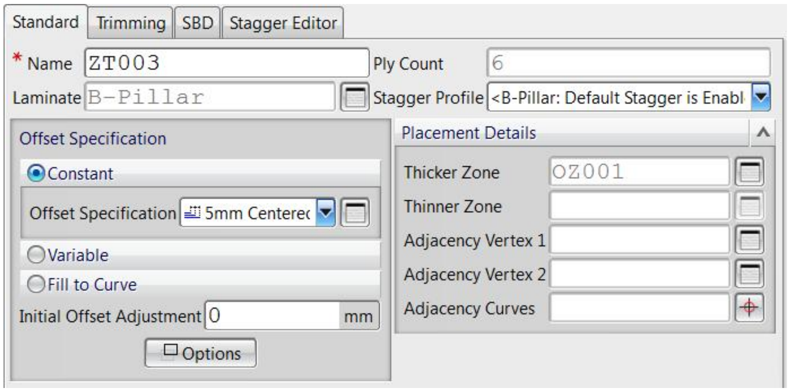


Figure 10–8

4. Click .

5. Highlight all layers and review the change in the NX window.

Task 5 - Modify ZT002.

1. In the Zone Transition list, double-click on **ZT002**.

2. Click  (Link with Link Dialog) next to the *Offset Specification* field.

3. Click  (Create New).

4. Enter [50mm Total] for *Name*.

5. Expand *Advanced Spacing* area as shown in Figure 10–9.

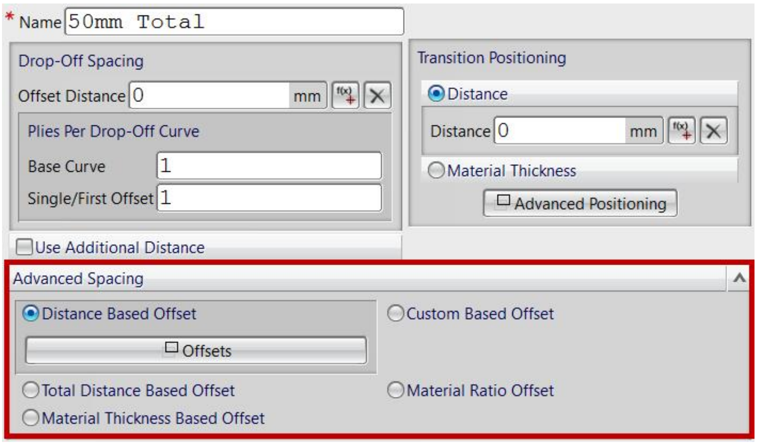


Figure 10–9

6. Click **Total Distance Based Offset**.
7. Enter [50] (1.968 in) for the *Total Distance* field.
8. Click OK.
9. Link **50mm Total** from the list as shown in Figure 10–10 and click OK.

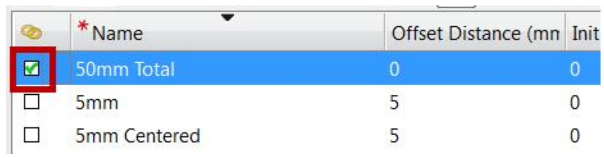


Figure 10–10

10. Click OK to close the message.
11. Click OK to save **ZT002**.
12. In the *Application Tree*, select **Layer** under **Design**.
13. Highlight all layers and review the model in NX. The model displays as shown in Figure 10–11.

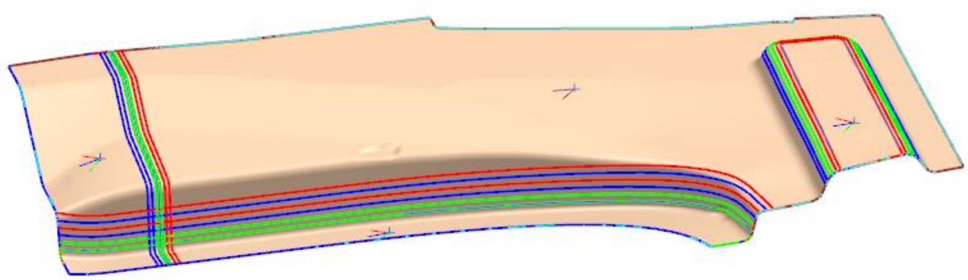


Figure 10–11

Note that the layer boundaries follow the zone transition's base curve shape.

Task 6 - Modify ZT003 - Fill to Curve option.

1. In the *Application Tree*, select **Zone Transition** under **Design**.
2. Double-click **ZT002** to modify.
3. In the *Offset Specification* area, select **Fill to Curve** as shown in Figure 10–12.

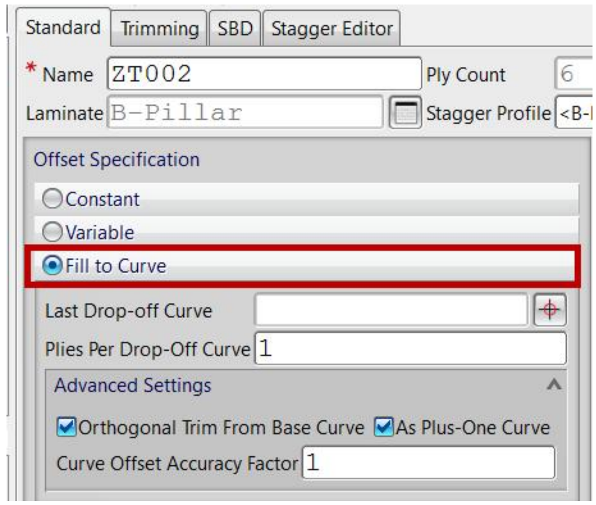


Figure 10–12

4. Click (Link Geometry) next to the *Last Drop-off Curve* field.
5. Change the *shading mode* to **Shaded with Edges**.
6. Select the highlighted edge as shown in Figure 10–13.

Make sure that all 26 curves are selected for the edge.

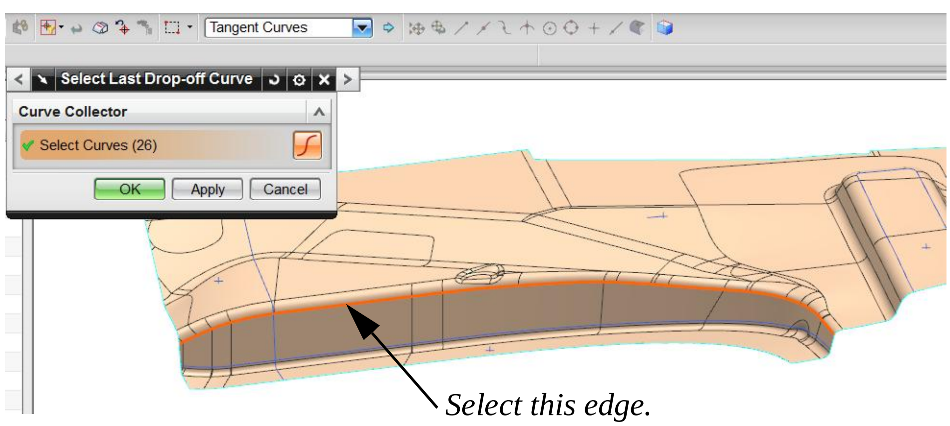


Figure 10–13

7. Click OK.

8. Expand *Advanced Settings* and clear the **Orthogonal Trim from Base Curve** option as shown in Figure 10–14.

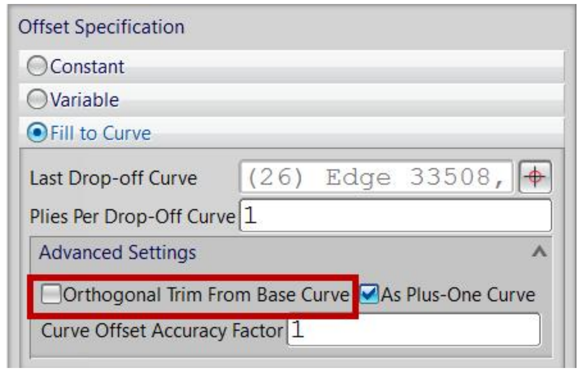


Figure 10–14

9. Click  to save **ZT002**.
10. In the *Application Tree*, select **Layer** under **Design**.
11. Highlight all layers and review the model in NX. The model displays as shown in Figure 10–15.

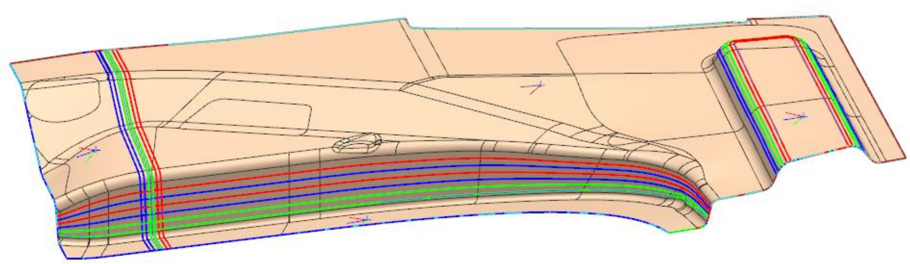


Figure 10–15

Task 7 - Modify ZT001.

1. In the *Application Tree*, select **Zone Transition** under **Design**.
2. Double-click on **ZT001** to modify.
3. Click  (Link with Link Dialog) next to the *Offset Specification* field.
4. Click  (Create New).
5. Enter [20:1] for *Name*.

6. Click **Material Ratio Offset** under *Advanced Spacing* as shown in Figure 10–16.

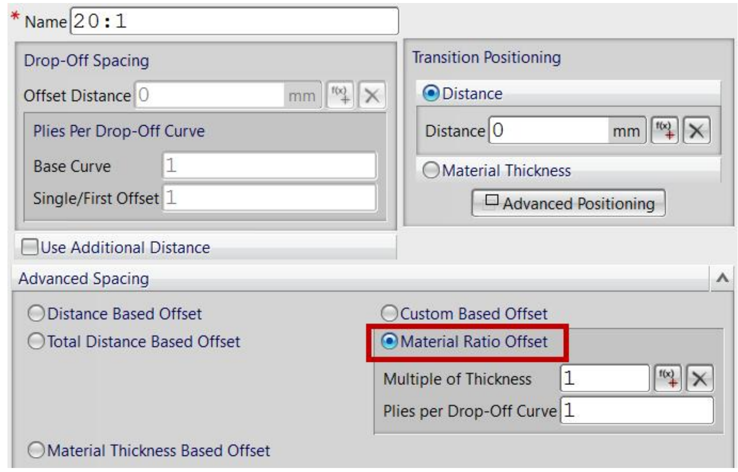


Figure 10–16

7. For **Multiple of Thickness**, enter [20].

8. Click OK.

9. Link **20:1** from the list and click OK.

10. Click OK to close the confirmation message.

11. Enter [5] mm (.197 in) for **Initial Offset Adjustment** as shown in Figure 10–17.

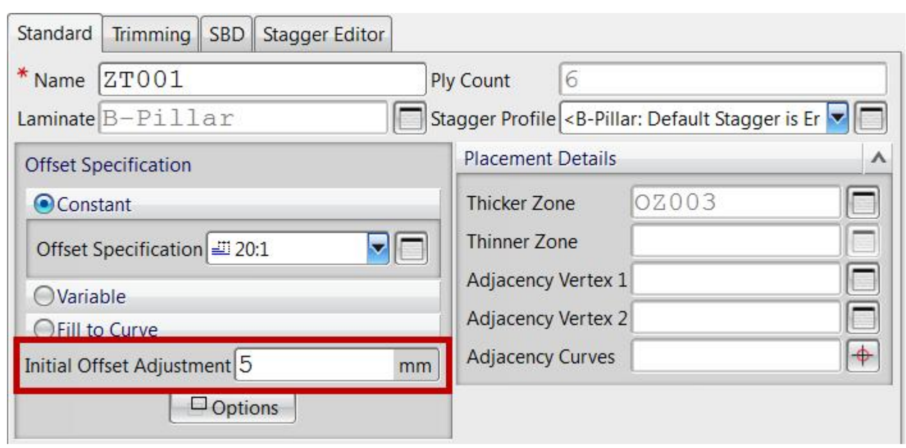


Figure 10–17

12. Click OK to save **ZT001**.

13. Select all layers and review the model in NX. The finished model is as shown in Figure 10–18.

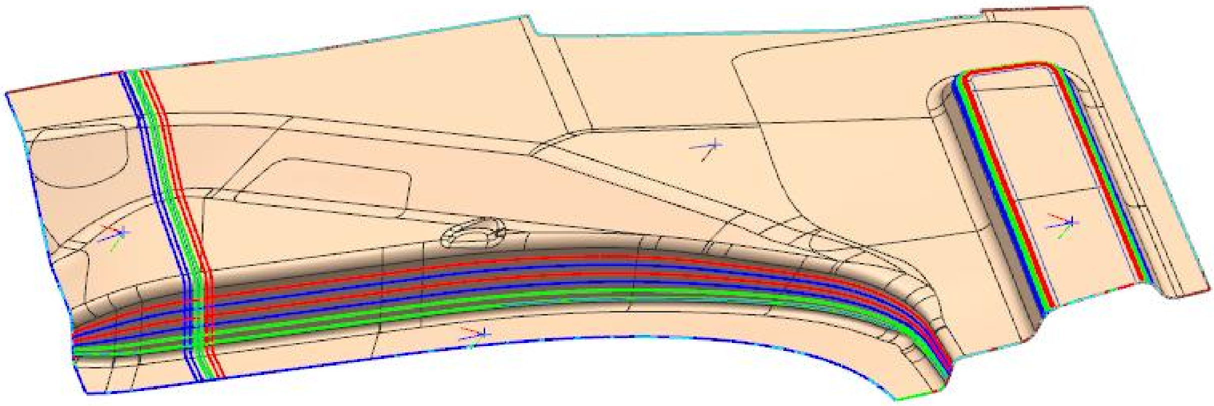


Figure 10–18

Exercise 10b Splitting and Extending Transitions

Userguide Reference:
3.7 Offset Specifications

In this exercise, you will use advanced Offset Specification tools and options to detail ply drop-off areas.

The completed model is shown in Figure 10–19.

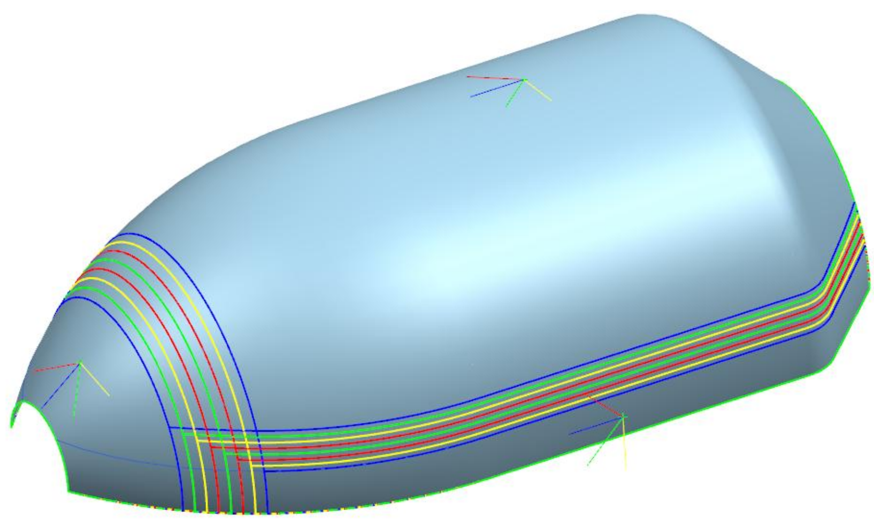


Figure 10–19

Goal

After you complete this exercise, you will be able to:

- ✓ **Use Zone Transition Splitter**
- ✓ **Extend Zone Transitions along a curve**

Estimated Time

15 min

The part is already populated with Laminate, Rosette, and Overlay Zones.

Task 1 - Open a part.

1. Open **MULTI_TRANSITION_MAN_B.prt**. The model displays as shown in Figure 10–20.

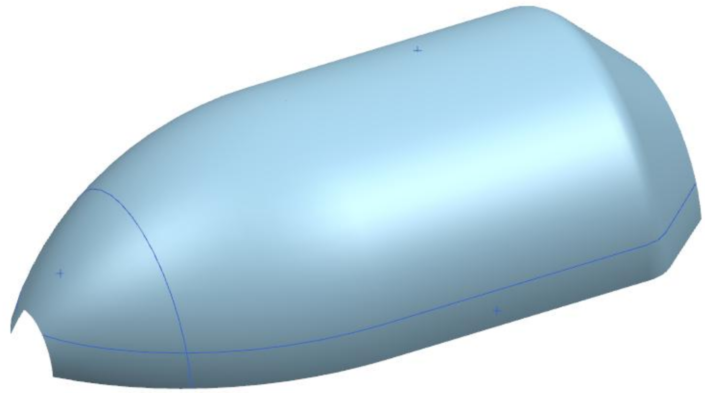


Figure 10–20

Task 2 - Review the existing geometry.

1. Press <F9> to start Fibersim.
2. Review the existing Laminate and Rosette in the model.
3. In the *Application Tree*, select **Layer** under **Design**.
4. Press <Ctrl> + <A> to highlight all layers.
5. Review the model in NX and zoom into the circled area in Figure 10–21.

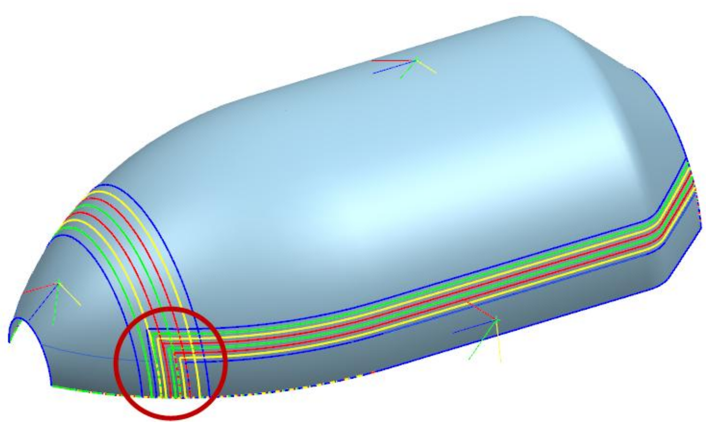


Figure 10–21

6. Note that the further you get from the baseline, the more misaligned the layer boundaries are as shown in Figure 10–22.

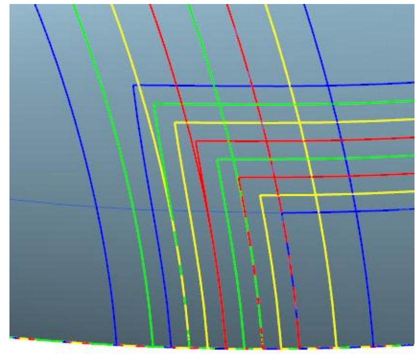


Figure 10–22

The design requirement is to have the right and left side layers butt up against the nose layers. In order to achieve this, the offset for the circumferential drop must be different than the horizontal drop. In the next task, you will split the zone transition to do this.

Task 3 - Split the Zone Transition on the left side.

1. In the *Application Tree*, select **Zone Transition** under **Design**.

2. Right-click on any column headers and select **Standard > Thicker Zone** as shown in Figure 10–23.

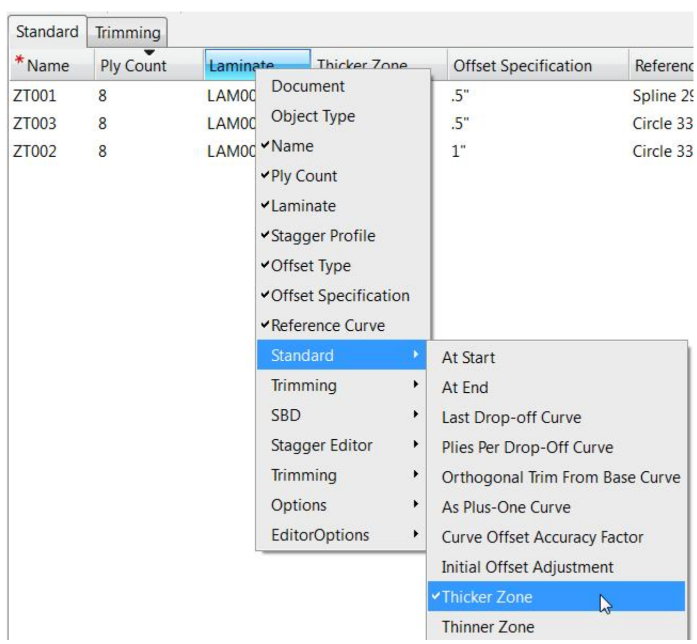


Figure 10–23

3. Select the Zone Transition with **Left Side** as Thicker Zone.

4. Click (Zone Transition Splitter) from the Zone Transition toolbar as shown in Figure 10–24.

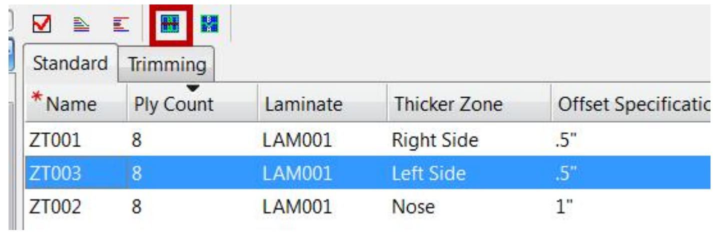


Figure 10–24

5. Select **Non-Tangency Angle** for *Splitting Method*.

6. Click Preview Split .

7. Switch to NX and note the red circle at the corner as shown in Figure 10–25. The red circle indicates the location of the split.

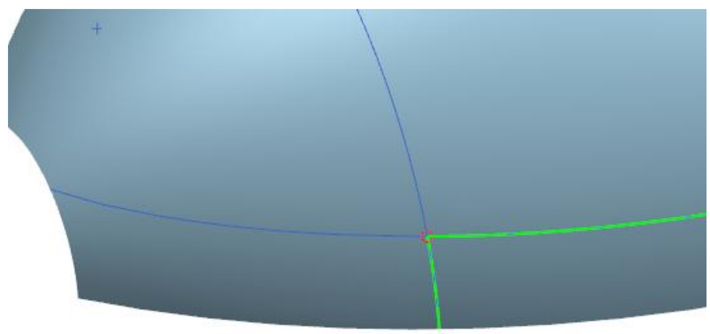


Figure 10–25

8. Back in Fibersim, click Split .

9. Click OK twice to save and close Zone Transition Splitter.

Task 4 - Modify the vertical section of the split zone transition.

1. From the Zone Transition list, double-click on **ZT003-B** (the vertical section) to modify.
2. For *Offset Specification*, select **1"** from the list.
3. Enter [-2] in (-50.8 mm) for Initial Offset Adjustment as shown in Figure 10–26.

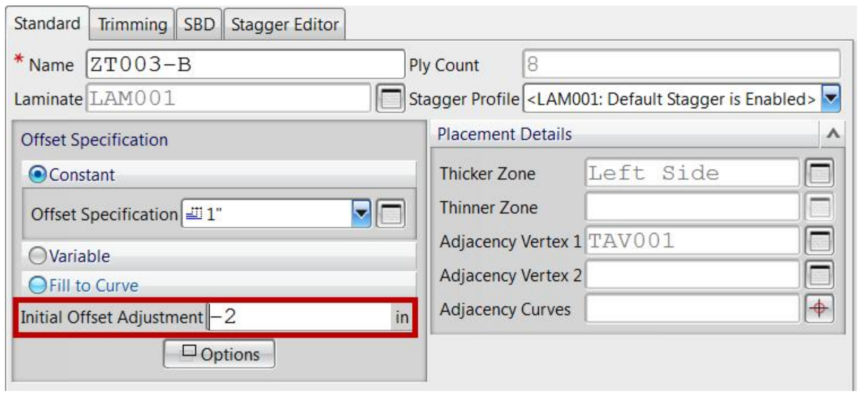


Figure 10–26

4. Click OK to save **ZT003-B**.

Task 5 - Review Layers.

1. In the *Application Tree*, select **Layer** under **Design**.
2. Highlight all layers. The model displays as shown in Figure 10–27.

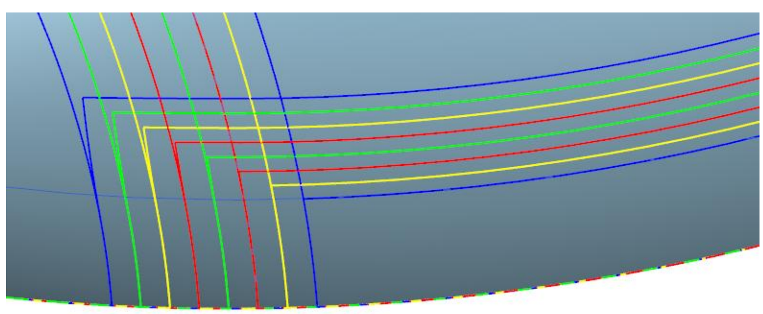


Figure 10–27

Note that the further you get from the corner, the more misaligned the layer boundaries are. This is because the vertical Zone Transition is extending tangentially to the original, rather than along the circumferential intersection curve.

Task 6 - Modify the trimming option.

1. In the *Application Tree*, select **Zone Transition** under **Design**.
2. Double-click **ZT003-B** to modify.
3. Select the *Trimming* tab.
4. Turn on the **Extend Along Reference Curve** option for *At Start* as shown in Figure 10–28.

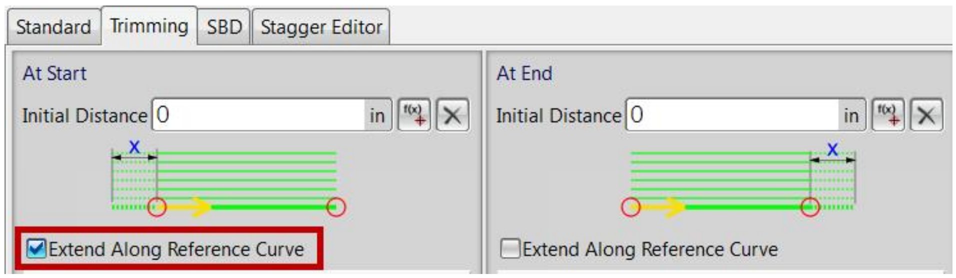


Figure 10–28

5. Click OK.
6. Highlight all layers and review the model. The model displays as shown Figure 10–29.

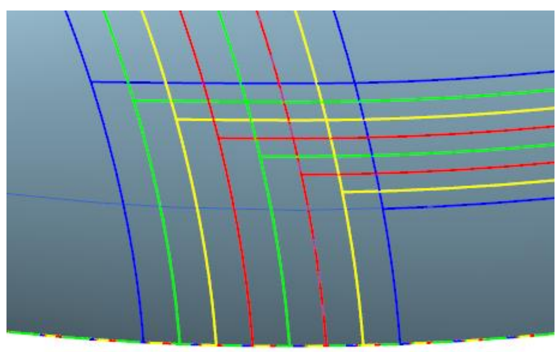


Figure 10–29

Task 7 - Modify the Zone Transition on the right side.

1. Repeat Task 3 - 6 for the right side zone transition to line up the boundaries to butt against the nose layers. Note that you will need to extend the zone transition along the reference curve at END.

Chapter 11

Layer Definition (Multi-Ply Based Design)

This chapter includes:

✓**Exercise 11a: Layer Sequencing (Multi-Ply Based Design)**

Exercise 11a Layer Sequencing (Multi-Ply Based Design)

Userguide Reference:
B.3.4 Step Utility
5.2 3D Cross Section
B.2.3 Composite Sequence Manager

In this exercise, you will learn how to sequence layers on a simple composite part.

The required layer sequence is shown in Figure 11–1.

Material	Orientation	Full Body	OZ001	OZ002	OZ003
PPG-PL-3K	+/-45				
T-12-in	0				
T-12-in	45				
T-12-in	-45				
PPG-PL-3K	0/90				
PPG-PL-3K	0/90				
T-12-in	-45				
T-12-in	45				
T-12-in	0				
PPG-PL-3K	+/-45				

Figure 11–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Interrogate Layer Sequence using 3D Cross Section utility**
- ✓ **Resequence Layers using Step Utility**
- ✓ **Resequence Layers using Composite Sequence Manager**
- ✓ **Resequence Layers using step-based Material Specifications**

Estimated Time

15 min

Task 1 - Open a part.

The part is fully populated with Laminate, Rosette, Overlay Zones, and Layers.

1. Open **MULTI_LAYER_DEFINITION_A.prt**. The model displays as shown in Figure 11–2.

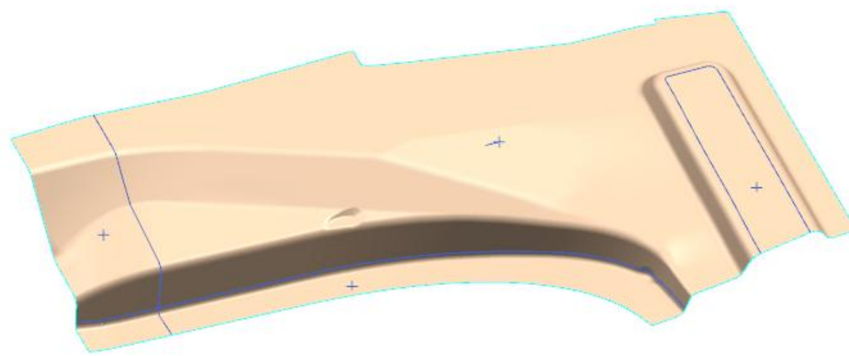


Figure 11–2

2. Press <F9> to start Fibersim.
3. In the *Application Tree*, select **Layer** under **Design** and multi-select all layers in the list.
4. The model displays as shown in Figure 11–3. The colored lines/curves represent Layer boundaries, color-coded accordingly to their material direction.

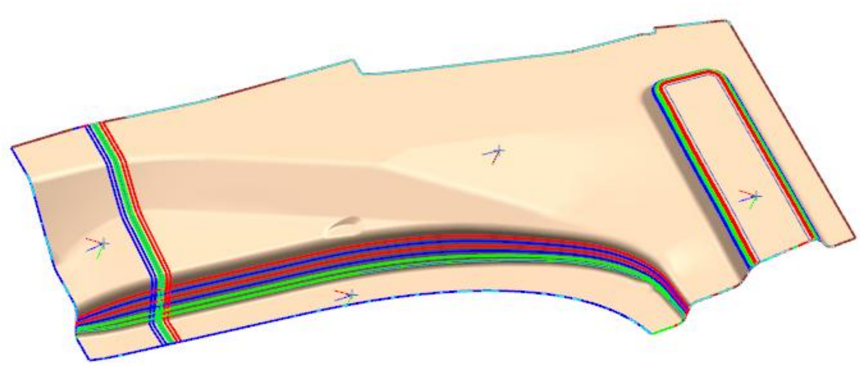


Figure 11–3

5. Review the layers in the Layer list. Note all layers within each Zone group have the same step value.

Task 2 - Create 3D Cross Section.

1. In the Layer list view, click (3D Cross Section) from the Layer toolbar.

2. In the form, click (Create New).

3. In the 3D Cross Section form, enter the following parameters as shown in Figure 11–4:

   - Component Type: **Layer**
   - Style: **Schematic**
   - Ply/Layer Offset Scale: **5**
   - Cross Section Curve: Show and link **XSection** from the Part Navigator

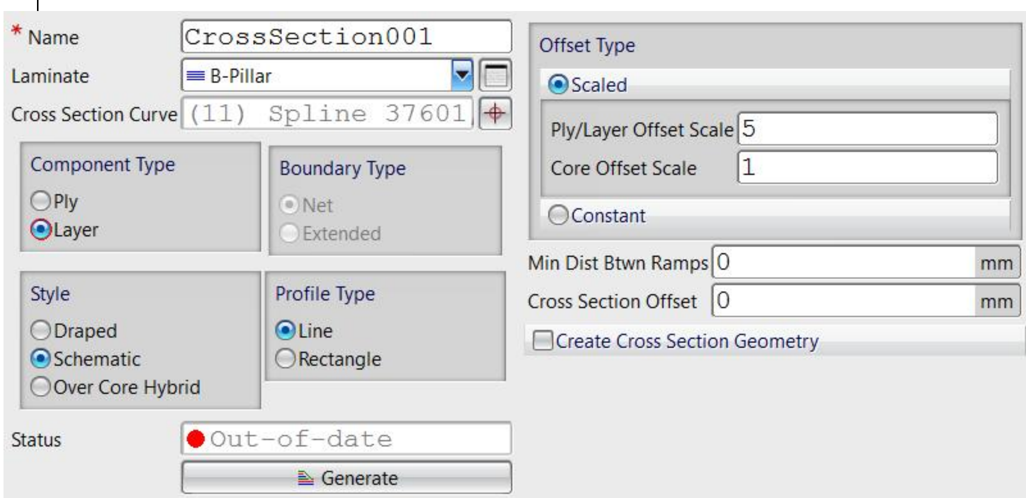


Figure 11–4

4. Click Generate and OK to close the 3D Cross Section form.

5. Activate NX window. The model displays as shown in Figure 11–5.

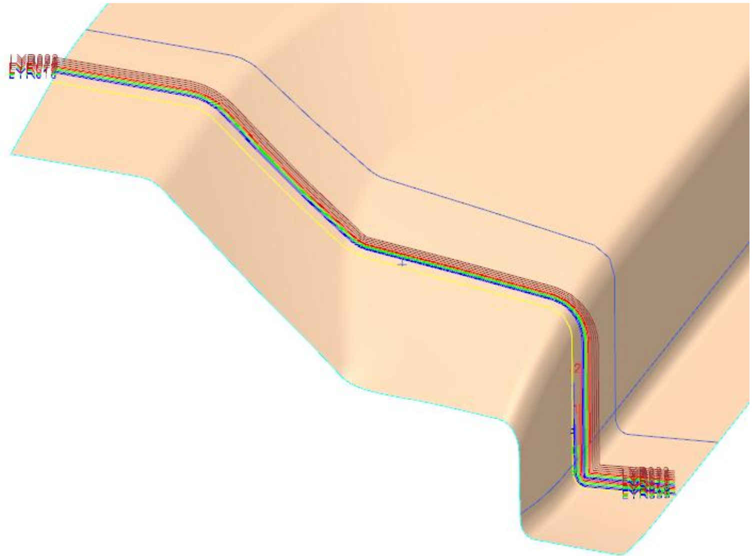


Figure 11–5

6. Zoom into the right side of the cross-section as shown in Figure 11–6. Note that the layer sequence Fibersim created by default is different from what is required.

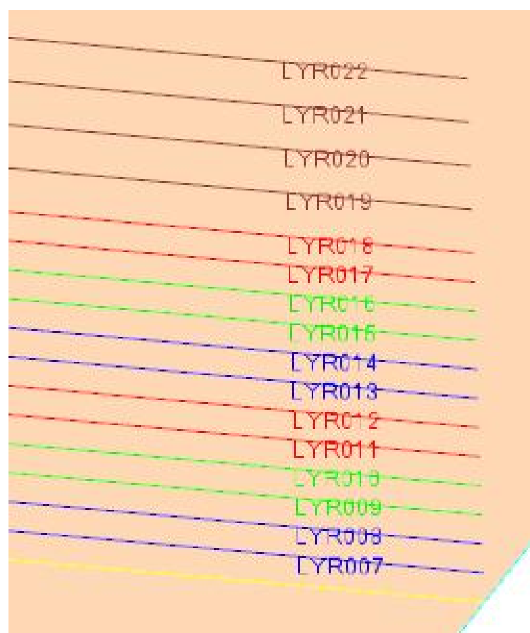


Figure 11–6

7. Click Close to close the 3D Cross Section form.

Task 3 - Sequence T-12-in Layers using Step Utility.

1. Click (Step Utility) as shown in Figure 11–7.

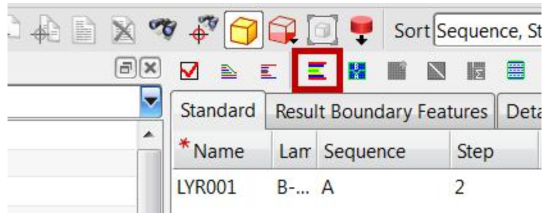


Figure 11–7

2. In the form that opens, click (Create New). The Step Utility form displays as shown in Figure 11–8.

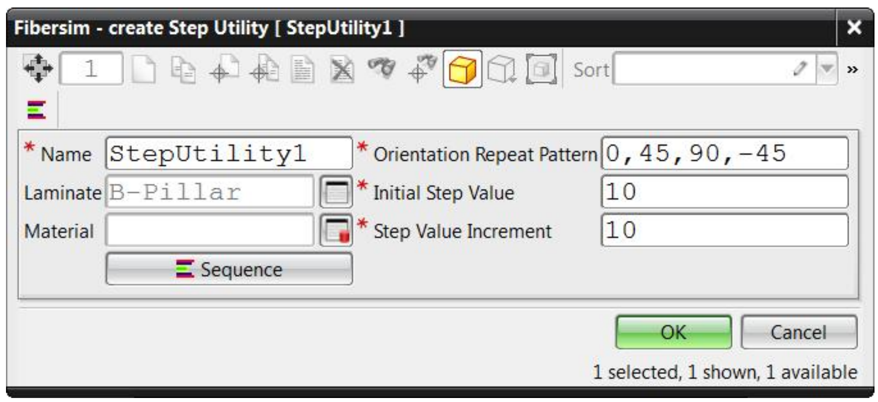


Figure 11–8

Only materials that exist in the part will be listed.

3. Next to the *Material* field, click (Link with Database Link Dialog) and in the link Material form, select **T-12-in** material as shown in Figure 11–9. Click OK.

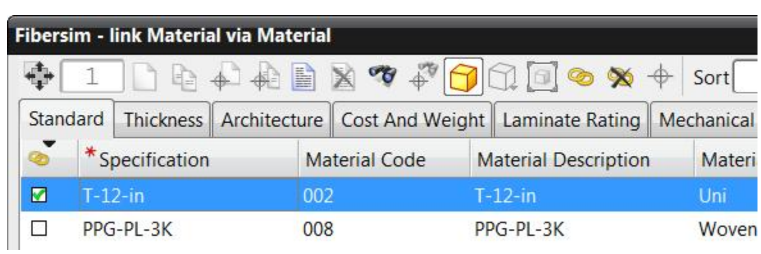


Figure 11–9

4. Click Sequence.

5. When the confirmation message displays, click **OK** to close the message.

6. Click **OK** and **Close** to close the Step Utility form.

7. Select **Sequence, Step and Name** from the Sort pull-down list as shown in Figure 11–10.

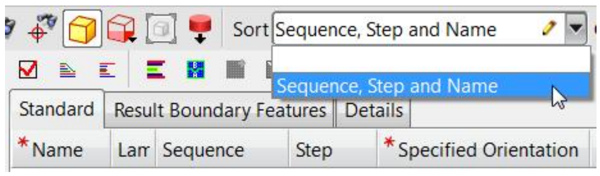
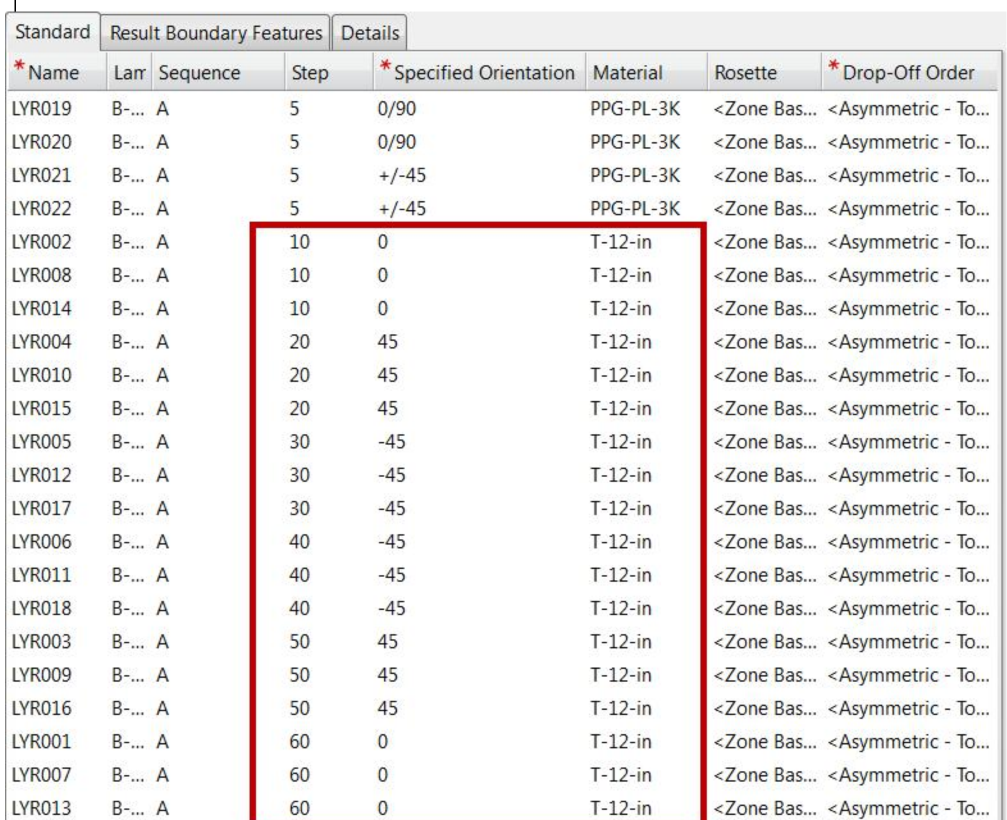


Figure 11–10

8. The list of Layers displays as shown in Figure 11–11.



*Name	Lan	Sequence	Step	*Specified Orientation	Material	Rosette	*Drop-Off Order
LYR019	B-...	A	5	0/90	PPG-PL-3K	<Zone Bas...	<Asymmetric - To...
LYR020	B-...	A	5	0/90	PPG-PL-3K	<Zone Bas...	<Asymmetric - To...
LYR021	B-...	A	5	+/-45	PPG-PL-3K	<Zone Bas...	<Asymmetric - To...
LYR022	B-...	A	5	+/-45	PPG-PL-3K	<Zone Bas...	<Asymmetric - To...
LYR002	B-...	A	10	0	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR008	B-...	A	10	0	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR014	B-...	A	10	0	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR004	B-...	A	20	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR010	B-...	A	20	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR015	B-...	A	20	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR005	B-...	A	30	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR012	B-...	A	30	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR017	B-...	A	30	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR006	B-...	A	40	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR011	B-...	A	40	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR018	B-...	A	40	-45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR003	B-...	A	50	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR009	B-...	A	50	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR016	B-...	A	50	45	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR001	B-...	A	60	0	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR007	B-...	A	60	0	T-12-in	<Zone Bas...	<Asymmetric - To...
LYR013	B-...	A	60	0	T-12-in	<Zone Bas...	<Asymmetric - To...

Figure 11–11

9. Note that all layers of **T-12-in** are now interleaved as required, but layers of **PPG-PL-3K** material are yet to be properly sequenced.

Task 4 - Update the 3D Cross Section.

1. In the Layer list view, click (3D Cross Section) from the Layer toolbar.
2. Select **CrossSection001** from the list.
3. Click (Generate 3D Cross Section).
4. Review the updated 3D Cross Section in NX.
5. Click Close to close the 3D Cross Section window.

Task 5 - Finalize the stackup using Composite Sequence Manager.

1. Click (Composite Sequence Manager) as shown in Figure 11–12.

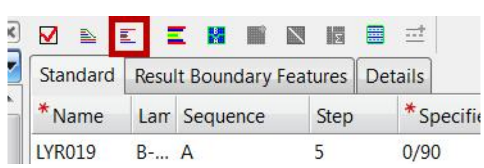
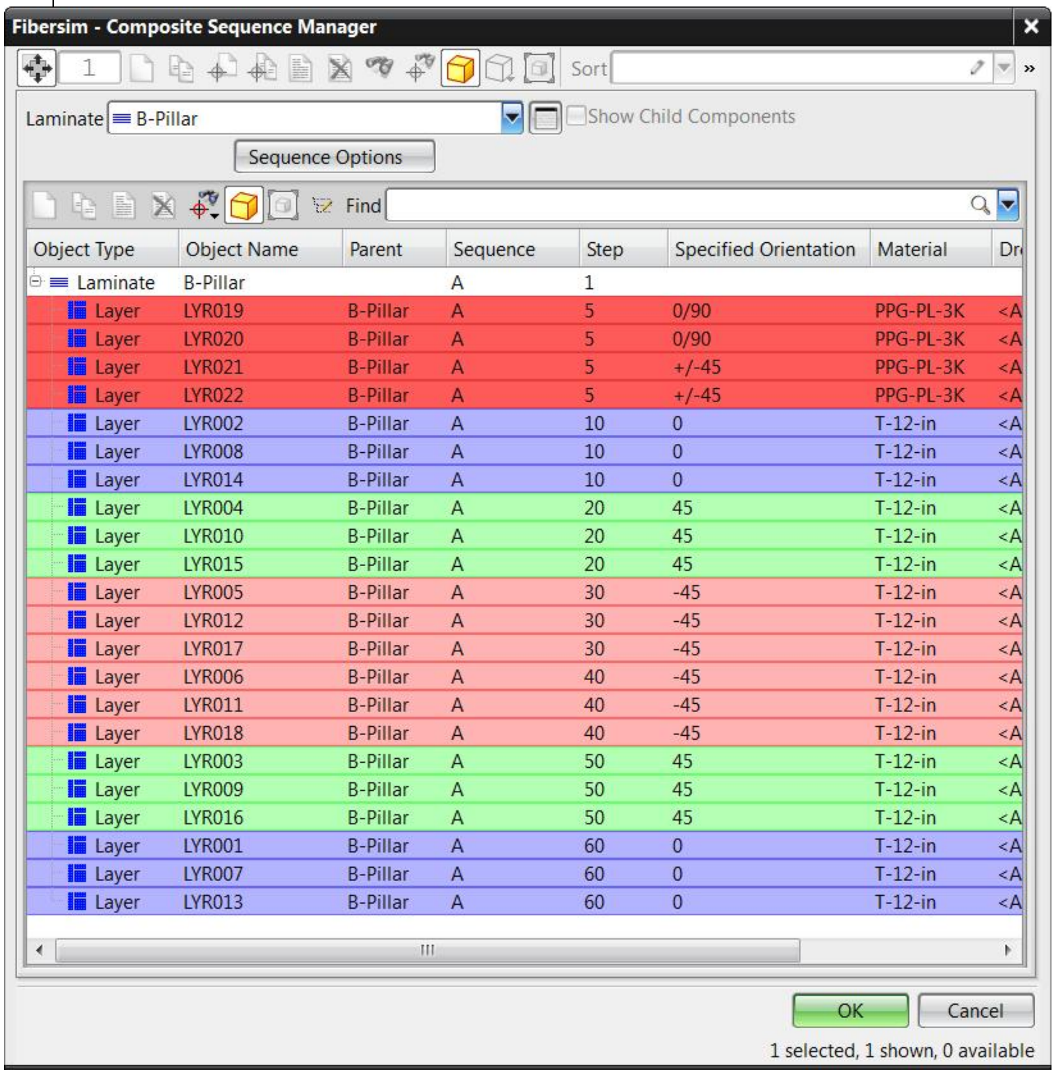


Figure 11–12

2. The Composite Sequence Manager form opens as shown in Figure 11–13.



Object Type	Object Name	Parent	Sequence	Step	Specified Orientation	Material	Dr
Laminate	B-Pillar		A	1			
Layer	LYR019	B-Pillar	A	5	0/90	PPG-PL-3K	<A
Layer	LYR020	B-Pillar	A	5	0/90	PPG-PL-3K	<A
Layer	LYR021	B-Pillar	A	5	+/-45	PPG-PL-3K	<A
Layer	LYR022	B-Pillar	A	5	+/-45	PPG-PL-3K	<A
Layer	LYR002	B-Pillar	A	10	0	T-12-in	<A
Layer	LYR008	B-Pillar	A	10	0	T-12-in	<A
Layer	LYR014	B-Pillar	A	10	0	T-12-in	<A
Layer	LYR004	B-Pillar	A	20	45	T-12-in	<A
Layer	LYR010	B-Pillar	A	20	45	T-12-in	<A
Layer	LYR015	B-Pillar	A	20	45	T-12-in	<A
Layer	LYR005	B-Pillar	A	30	-45	T-12-in	<A
Layer	LYR012	B-Pillar	A	30	-45	T-12-in	<A
Layer	LYR017	B-Pillar	A	30	-45	T-12-in	<A
Layer	LYR006	B-Pillar	A	40	-45	T-12-in	<A
Layer	LYR011	B-Pillar	A	40	-45	T-12-in	<A
Layer	LYR018	B-Pillar	A	40	-45	T-12-in	<A
Layer	LYR003	B-Pillar	A	50	45	T-12-in	<A
Layer	LYR009	B-Pillar	A	50	45	T-12-in	<A
Layer	LYR016	B-Pillar	A	50	45	T-12-in	<A
Layer	LYR001	B-Pillar	A	60	0	T-12-in	<A
Layer	LYR007	B-Pillar	A	60	0	T-12-in	<A
Layer	LYR013	B-Pillar	A	60	0	T-12-in	<A

Figure 11–13

3. Drag and drop **PPG-PL-3K** layers in the following sequence:

   - LYR019 after LYR017
   - LYR020 after LYR019
   - LYR022 after LYR013

4. The reordered list should update as shown in Figure 11–14.

Laminate	B-Pillar		A	1		
Layer	LYR021	B-Pillar	A	5	+/-45	PPG-PL-3
Layer	LYR002	B-Pillar	A	10	0	T-12-in
Layer	LYR008	B-Pillar	A	10	0	T-12-in
Layer	LYR014	B-Pillar	A	10	0	T-12-in
Layer	LYR004	B-Pillar	A	20	45	T-12-in
Layer	LYR010	B-Pillar	A	20	45	T-12-in
Layer	LYR015	B-Pillar	A	20	45	T-12-in
Layer	LYR005	B-Pillar	A	30	-45	T-12-in
Layer	LYR012	B-Pillar	A	30	-45	T-12-in
Layer	LYR017	B-Pillar	A	30	-45	T-12-in
Layer	LYR019	B-Pillar	A	35	0/90	PPG-PL-3
Layer	LYR020	B-Pillar	A	37.5	0/90	PPG-PL-3
Layer	LYR006	B-Pillar	A	40	-45	T-12-in
Layer	LYR011	B-Pillar	A	40	-45	T-12-in
Layer	LYR018	B-Pillar	A	40	-45	T-12-in
Layer	LYR003	B-Pillar	A	50	45	T-12-in
Layer	LYR009	B-Pillar	A	50	45	T-12-in
Layer	LYR016	B-Pillar	A	50	45	T-12-in
Layer	LYR001	B-Pillar	A	60	0	T-12-in
Layer	LYR007	B-Pillar	A	60	0	T-12-in
Layer	LYR013	B-Pillar	A	60	0	T-12-in
Layer	LYR022	B-Pillar	A	70	+/-45	PPG-PL-3

Figure 11–14

5. Click Sequence Options.

6. Enter [10] for both **Initial Step** and **Step Increment**.

7. Click (Resequence). The list updates as shown in Figure 11–15. Note that the Step values are ordered in regular increments as required.

Laminate	B-Pillar		A	1		
Layer	LYR021	B-Pillar	A	10	+/-45	PPG-PL-3K
Layer	LYR002	B-Pillar	A	20	0	T-12-in
Layer	LYR008	B-Pillar	A	20	0	T-12-in
Layer	LYR014	B-Pillar	A	20	0	T-12-in
Layer	LYR004	B-Pillar	A	30	45	T-12-in
Layer	LYR010	B-Pillar	A	30	45	T-12-in
Layer	LYR015	B-Pillar	A	30	45	T-12-in
Layer	LYR005	B-Pillar	A	40	-45	T-12-in
Layer	LYR012	B-Pillar	A	40	-45	T-12-in
Layer	LYR017	B-Pillar	A	40	-45	T-12-in
Layer	LYR019	B-Pillar	A	50	0/90	PPG-PL-3K
Layer	LYR020	B-Pillar	A	60	0/90	PPG-PL-3K
Layer	LYR006	B-Pillar	A	70	-45	T-12-in
Layer	LYR011	B-Pillar	A	70	-45	T-12-in
Layer	LYR018	B-Pillar	A	70	-45	T-12-in
Layer	LYR003	B-Pillar	A	80	45	T-12-in
Layer	LYR009	B-Pillar	A	80	45	T-12-in
Layer	LYR016	B-Pillar	A	80	45	T-12-in
Layer	LYR001	B-Pillar	A	90	0	T-12-in
Layer	LYR007	B-Pillar	A	90	0	T-12-in
Layer	LYR013	B-Pillar	A	90	0	T-12-in
Layer	LYR022	B-Pillar	A	100	+/-45	PPG-PL-3K

Figure 11–15

8. Click to close the Composite Sequence Manager form.
9. Press <Ctrl> + <A> to select all layers.
10. Right-click and select **Modify**.
11. Enter [LYR001,1] for **Name**.
12. Click .

Task 6 - Update the 3D Cross Section.

1. In the Layer list view, click (3D Cross Section) from the Layer toolbar.
2. Select **CrossSection001** from the list.
3. Click (Generate 3D Cross Section).
4. Activate the NX window. The model displays as shown in Figure 11–16.

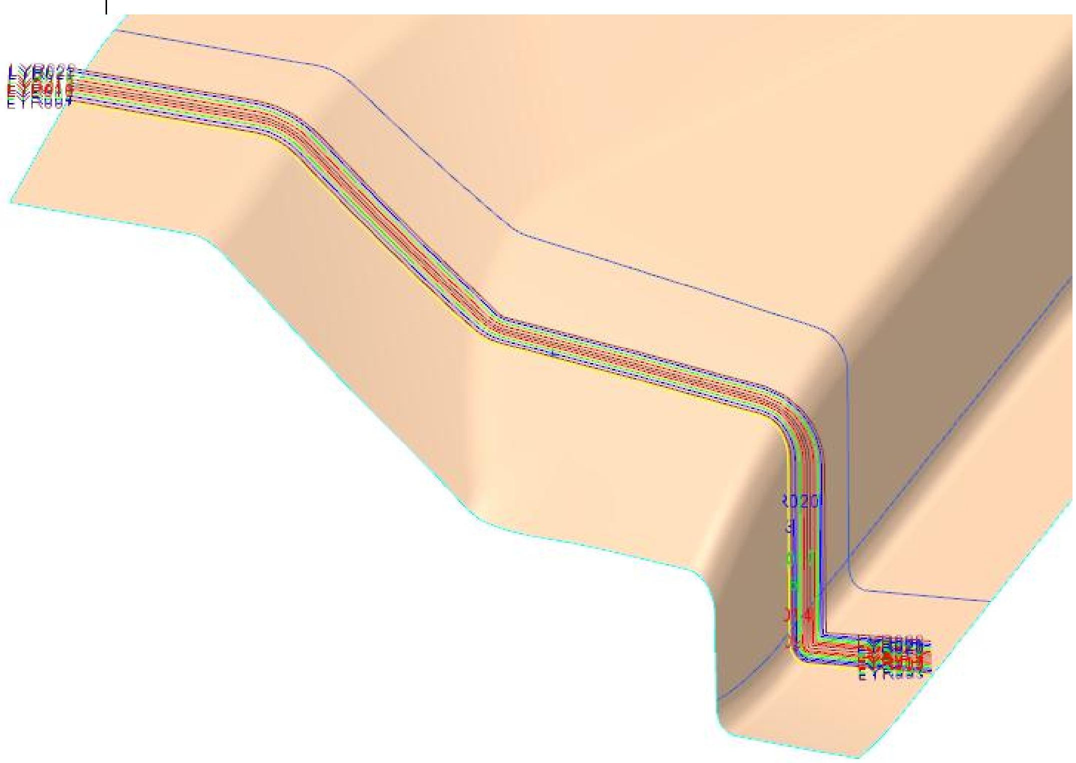


Figure 11–16

5. Click Close to close the 3D Cross Section window.

The same results can be made by assigning step values to Material Specifications as shown in the next couple tasks.

Task 7 - Assign step-based Material Specifications to Overlay Zones.

1. In the *Application Tree*, select **Overlay Zone** under **Design**.
2. Select **OZ001, OZ002**, and **OZ003** from the object list.
3. Right-click and select **Modify**.
4. For *Laminate Specification*, select **6p_Sequenced** from the pull-down list.
5. Click on the **Laminate Specification** link as shown in Figure 11–17.

Figure 11–17

6. Review the list of Material Specifications for **6p_Sequenced**. Note the step value for each Material Specification.
7. Click Cancel to close without saving any changes to **6p_Sequenced**.
8. Click OK to save changes to the Overlay Zones.
9. Double-click on **Full Body** in the Overlay Zone list to modify.
10. For *Laminate Specification*, select **4p_Sequenced** from the pull-down list.
11. Click OK to save changes to Full Body.

Task 8 - Regenerate the Layers.

1. Click (Zone to Layer Analysis) from the Overlay Zone toolbar.
2. Click Generate.
3. Review the report and note that Fibersim deleted all layers and create new ones.
4. Click OK to close the *Zone to Layer Analysis* window.
5. In the *Application Tree*, select **Layer** under **Advanced > Design**.
6. Review the generated layers.

Task 9 - Update the 3D Cross Section.

1. In the Layer list view, click (3D Cross Section) from the Layer toolbar.
2. Select **CrossSection001** from the list.
3. Click (Generate 3D Cross Section).
4. Activate the NX window and review the model.
5. Click Close to close the 3D Cross Section window.
6. Close Fibersim.

Chapter 12

Surface Offset (Multi-Ply Based Design)

This chapter includes:

✓**Exercise 12a: Creating Parametric Surface Offset with Ramps**

Exercise 12a Creating Parametric Surface Offset with Ramps

User Guide Reference:
8.5 Flat pattern Placement Options
10 Parametric Surface Offset (PSO) Elements

In this exercise, you will learn how to generate offset surfaces and solid models based on zones, layers, and plies using Fibersim's Parametric Surface Offset object.

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a Parametric Surface Offset**
- ✓ **Create an IML Surface and a final Solid with Ramps**
- ✓ **Update an IML Surface and Solid after design changes**

Estimated Time

20 min

Task 1 - Open a part.

1. Open **MULTI_SURFACE_OFFSET_A.prt**. The model displays as shown NX in Figure 12–1.

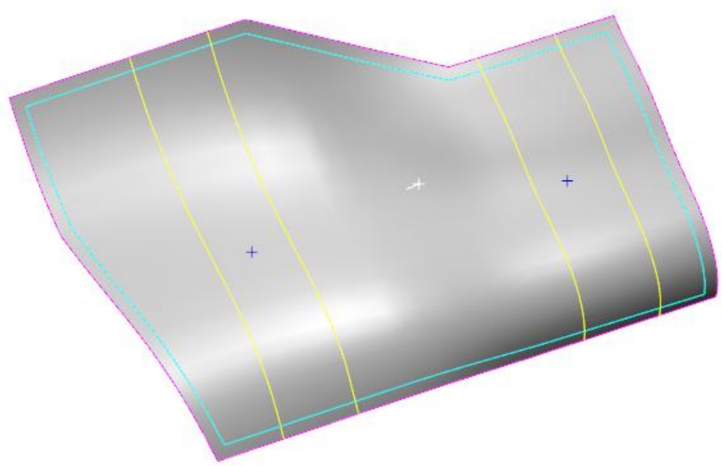


Figure 12–1

2. Press <F9> to start Fibersim.

3. In the *Application Tree*, select **Overlay Zone** under **Design**.

4. Multi-select all three Overlay Zones in the list. The model should display as shown in Figure 12–2.

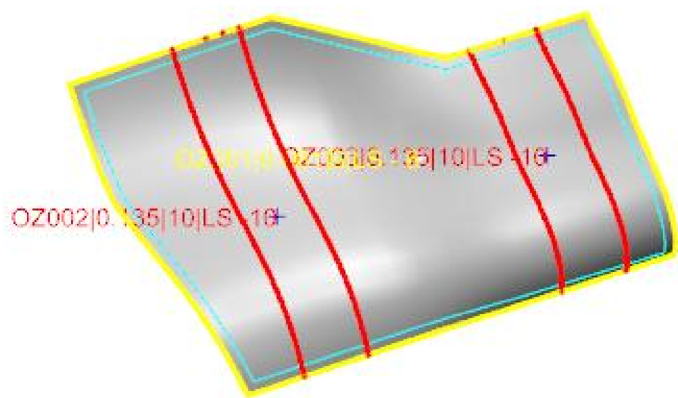


Figure 12–2

Task 2 - Create a Final Solid with Parametric Surface Offset.

1. Select **Tools > Surface/Solid Creation > Parametric Surface Offset** as shown in Figure 12–3.

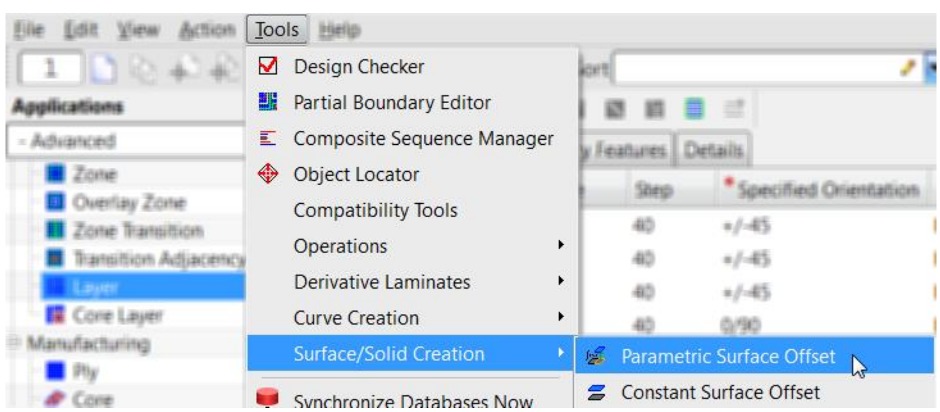


Figure 12–3

Make sure to check the Boundary Type before turning on the option for Solid creation.

2. In the form that opens, click (Create New).

3. Click the checkbox for **Extended** under *Boundary Types* and **Create Solid to Net** as shown in Figure 12–4.

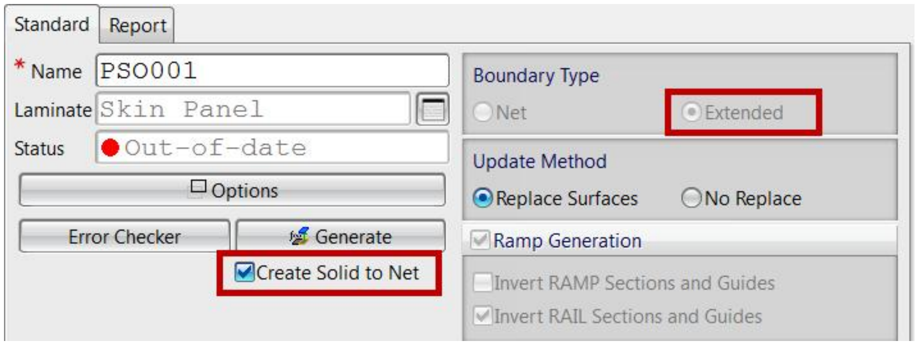


Figure 12–4

4. Click Generate to generate the surface.

5. A message window opens prompting you that the parametric surface offset was generated. Click OK twice to close the message and save **PS001**.

6. Click Close to return to the main Fibersim window.

7. Close Fibersim and switch to NX and review the model. The model should display as shown in Figure 12–5.

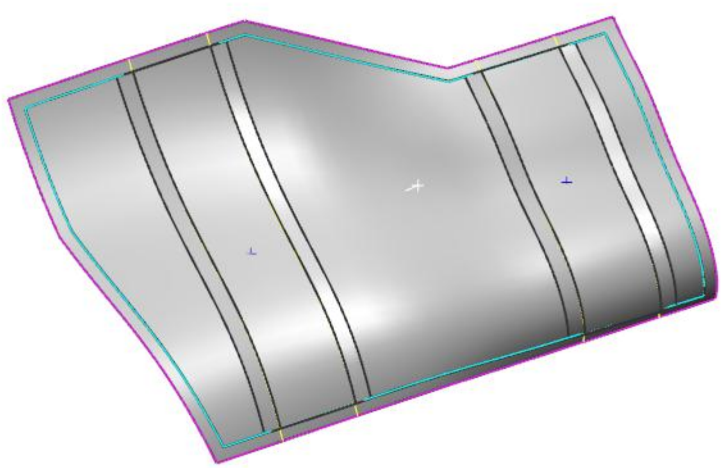


Figure 12–5

Task 3 - Change the Laminate thickness in one Overlay Zone.

1. Press <F9> to start Fibersim.

2. In the *Application Tree*, select **Overlay Zone** under **Design**. In the list, double-click **OZ003**.

3. For the *Target Ply Count*, enter [16]. In the Laminate Specification drop-down list, select **16p** as shown in Figure 12–6.

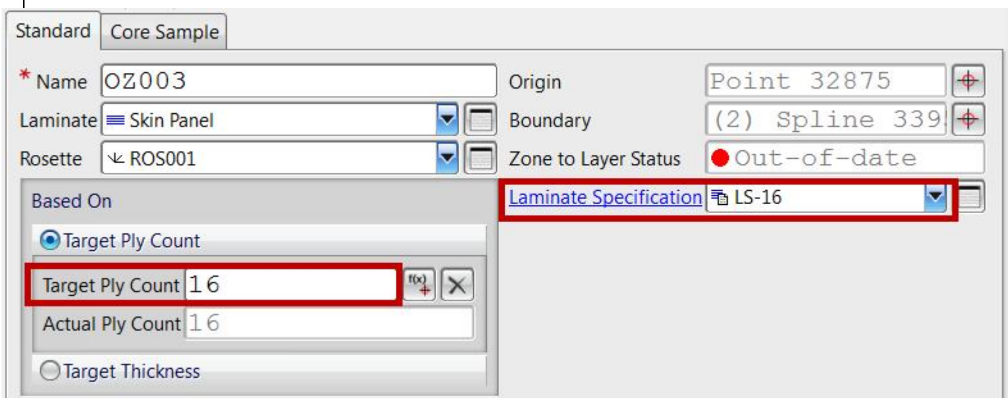


Figure 12–6

4. Click OK.

5. Click (Zone to Layer Analysis) to open Zone to Layer Analysis form.

6. Click Generate.

7. Click OK twice to close the message box and the Zone to Layer Analysis form.

Task 4 - Update the Solid.

1. Select **PSO** from the *Application Browser* as shown in Figure 12–7.

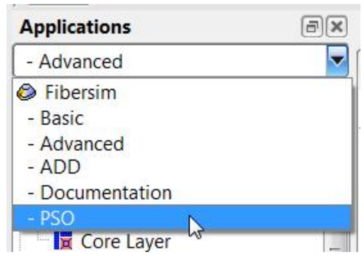


Figure 12–7

2. Select **Parametric Surface Offset** from the menu.
3. Select **PSO001** from the list of objects.
4. Click  (Generates a parametric offset).
5. Click  to close the message box.

6. The solid model updates according to the laminate definition when edges are displayed, as shown in Figure 12–8.

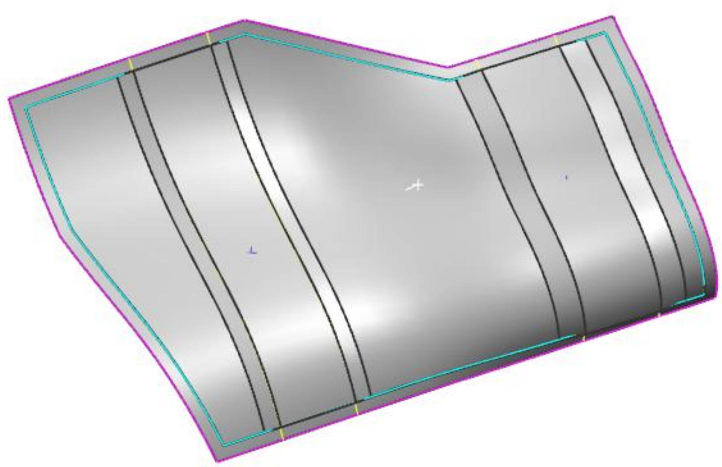


Figure 12–8

7. Close Fibersim.
8. Save and close the model.

Chapter 13

Manufacturing Features

This chapter includes:

✓**Exercise 13a: Layer-based Manufacturing Laminate, with Splice and Dart Groups**

Exercise 13a Layer-based Manufacturing Laminate, with Splice and Dart Groups

Userguide Reference:
3.4 Dart Group
3.5 Splice Group
3.6 Laminate Region
B.4.4 Manufacturing Laminate Creation

In this exercise, you will create a new product consisting of the layer-populated engineering model and the empty manufacturing model for concurrent work purposes. You will then create a manufacturing laminate and transfer the layer data from the engineering model and create splice groups and dart groups to resolve producibility issues. The completed manufacturing model displays as shown in Figure 13–1.

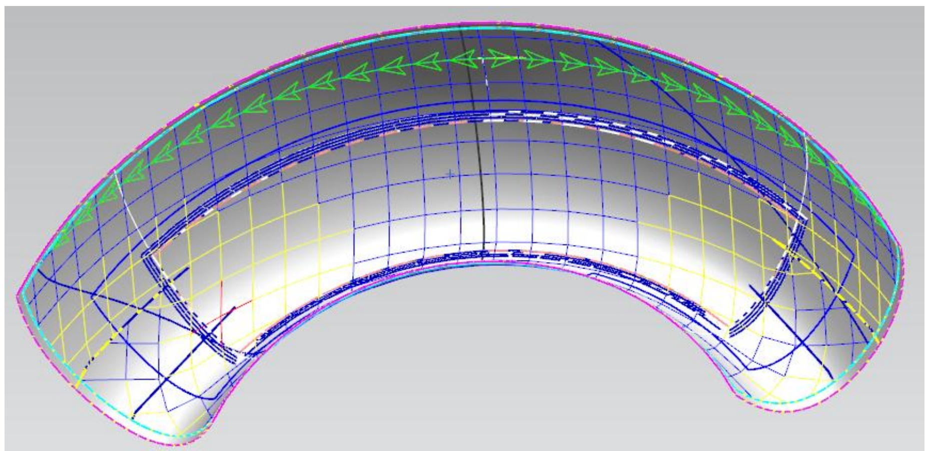


Figure 13–1

Goal

After you complete this exercise, you will be able to:

- ✓ **Create a new Product consisting of both the Engineering and Manufacturing models**
- ✓ **Create Manufacturing Laminate and transfer layers (no springback)**
- ✓ **Create Splice Groups for 0 degree plies**
- ✓ **Create Overlap Splice Laminate Region**
- ✓ **Create Splice Group for 45 degree plies**
- ✓ **Create Dart Group for 90 degree plies**
- ✓ **Run Producibility to show that all issues have been resolved**

Estimated Time

20 min

Task 1 - Open fully populated zone-based Engineering part.

1. Open **ENG.prt** in the **MULTI_MAN_FEATURES_A** folder. The model displays as shown in Figure 13–2.

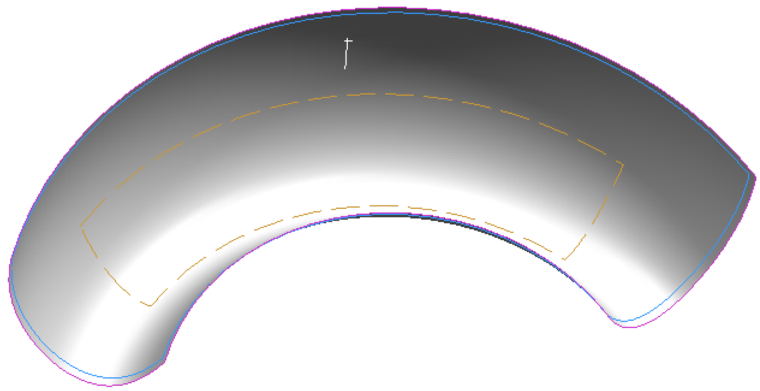


Figure 13–2

2. Press <F9> to start Fibersim.
3. Note the Fibersim objects that have already been created.

Note that the Fibersim objects were created with Fibersim Elite.

Task 2 - Open and examine empty Manufacturing part.

1. Open **MFG.prt**. The model displays as shown in Figure 13–3.

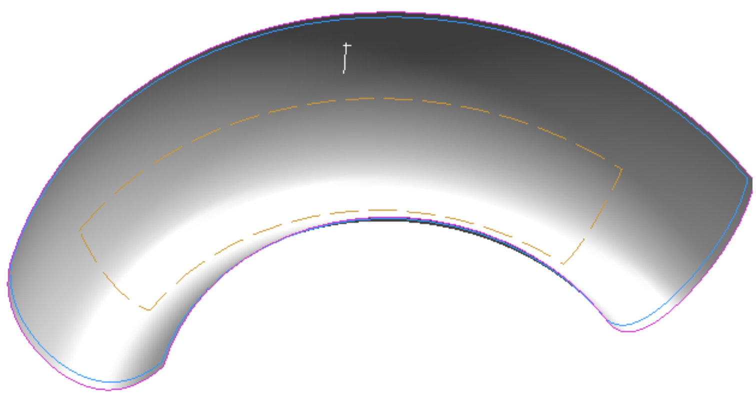


Figure 13–3

2. Press <F9> to start Fibersim.
3. Note that there are no Fibersim objects created.
4. Close Fibersim.

Task 3 - Create new Product consisting of both parts.

1. In NX, select **File > New**.

It is important to set the assembly's unit to match the other parts. NX will not let you make the the mismatched parts to be the working parts.

2. Select **Assembly** and make sure that unit is set to **Inch**.
3. Click .
4. Press <Shift> and select **ENG.prt** and **MFG.prt** from the Loaded Parts list.
5. Make sure that the *Positioning* field is set to **Absolute Origin**.
6. Click .
7. The model displays as shown in Figure 13–4.

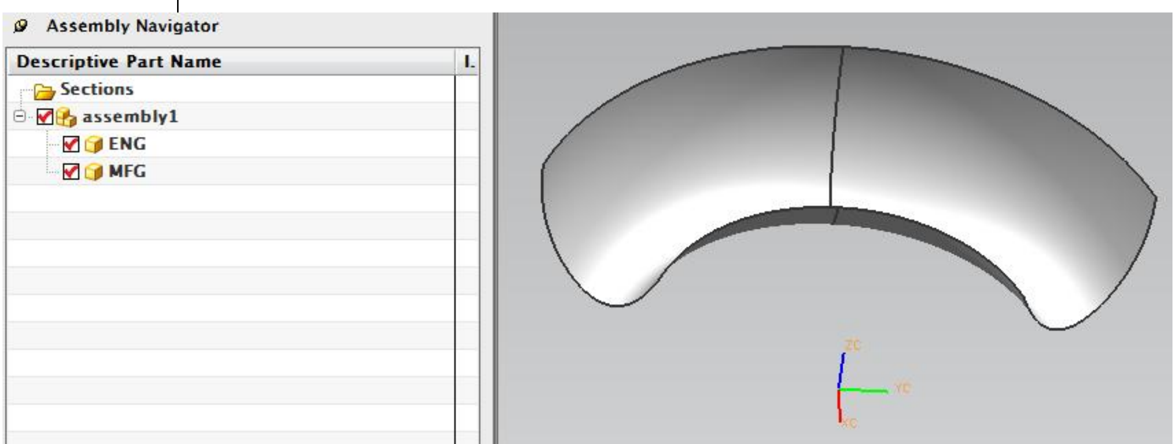


Figure 13–4

Task 4 - Create Laminate and Rosette in Manufacturing part.

1. In the *Assembly Navigator*, right-click on **ENG** and select **Hide**.
2. Right-click on **MFG**, and select **Make Work Part** as shown in Figure 13–5.

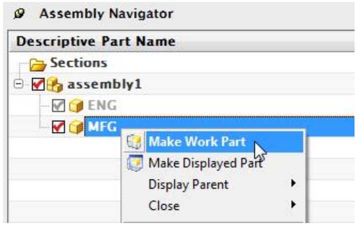


Figure 13–5

3. Press <F9> to start Fibersim.
4. In the *Application Tree*, select **Laminate** under **Design**.
5. Click  (Create New).
6. Specify the following parameters as shown in Figure 13–6.

Verify that the Parent field is empty.

- Name: **MFG-TOOL**
- Step: **1**
- Layup Surface: **MFG_Tool Surface**
- Net Boundary: **MFG_Net Boundary**
- Extended Boundary: **MFG_Ext Boundary**

Select the boundaries from "Select From the List..." to make ensure that you select the correct splines.

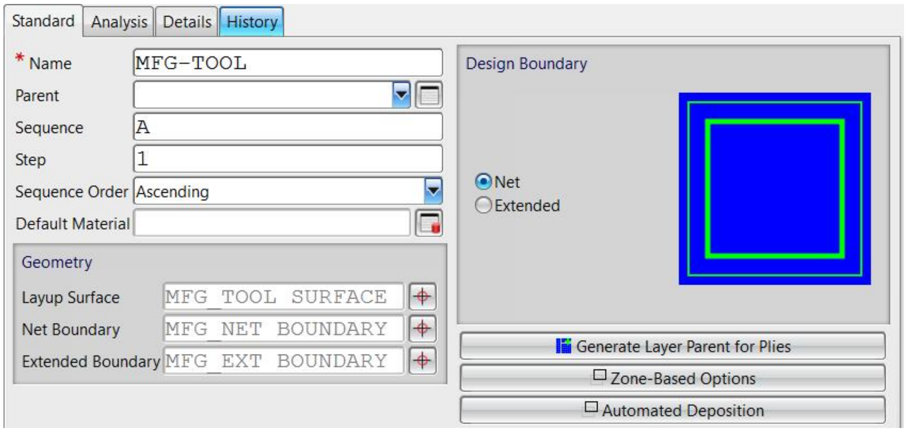


Figure 13–6

7. Click  to save.
8. In the *Application Tree*, select **Rosette** under **Design**.

9. Click (Create New).

10. Specify the following parameters as shown in Figure 13–7.

    - Name: **MFG-ROS**
    - Surface: **MFG_Tool Surface**
    - Origin: **MFG_Rosette Origin**
    - Direction: **MFG_Zero Direction**

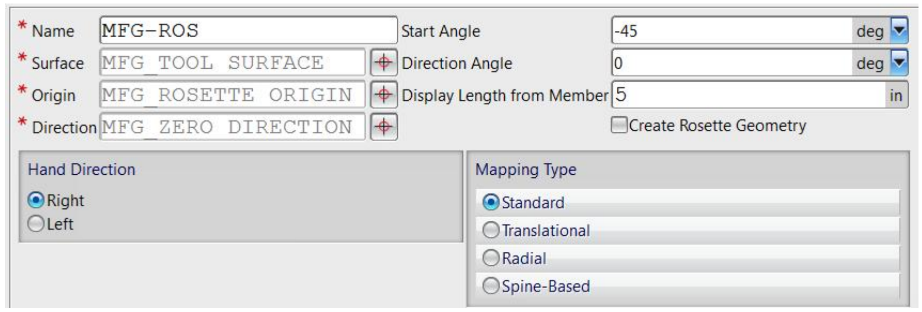


Figure 13–7

11. Click OK to save.

Task 5 - Create Manufacturing Laminate and transfer layer data.

You can also select (Manufacturing Laminate Creation) in the Laminate toolbar.

1. In the *Documents Browser*, right-click on **assmebly1** and select **Make Storage Document**.

2. In the Fibersim main menu, select **Tools > Derivative Laminates > Manufacturing Laminate Creation** as shown in Figure 13–8.

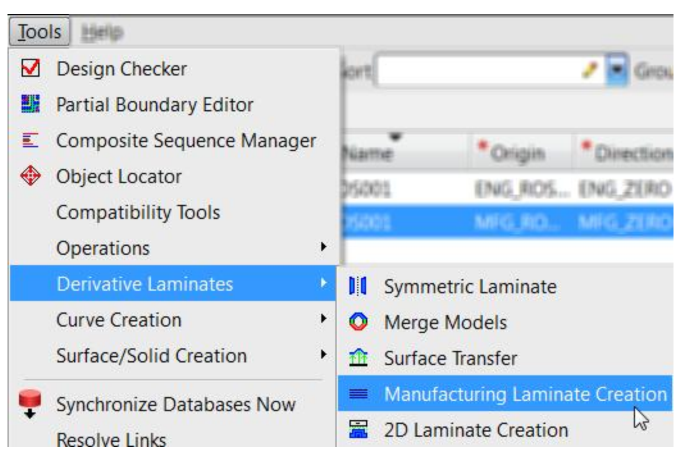


Figure 13–8

3. Click (Create New).

4. Specify the following parameters as shown in Figure 13–9. Close the Fibersim message that displays when selecting Engineering Laminate.

   - Engineering Laminate: **Shroud**
   - Engineering Rosette: **ROS001**
   - Manufacturing Laminate: **MFG-TOOL**
   - Manufacturing Rosette: **MFG-ROS**

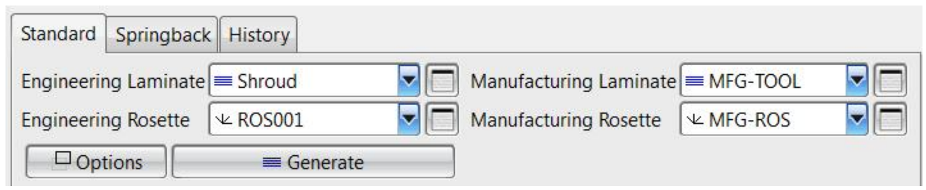


Figure 13–9

5. Click . Accept the default options as shown in Figure 13–10.

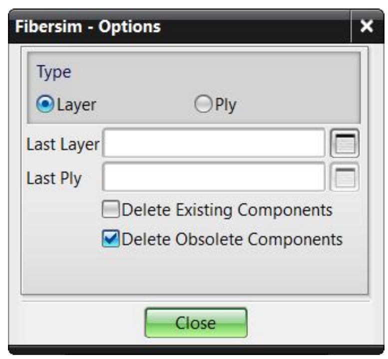
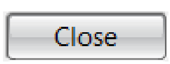
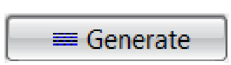


Figure 13–10

6. Click .

7. Click .

8. A message window opens prompting you that components were created as shown in Figure 13–11.

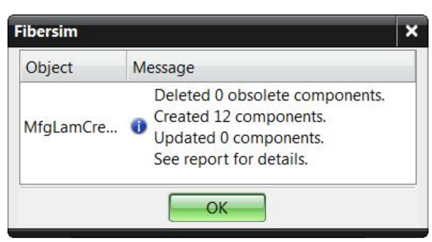


Figure 13–11

9. Click OK and view the report as shown in Figure 13–12.

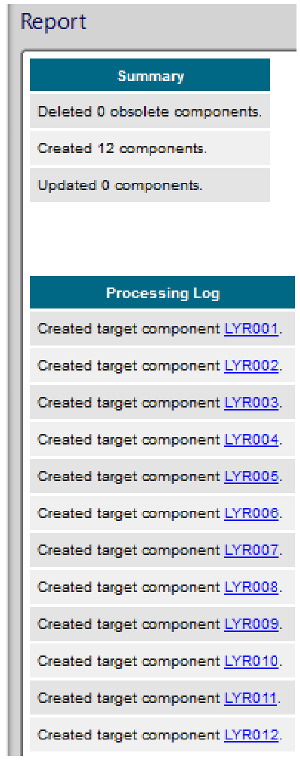


Figure 13–12

10. Click OK to save.

11. Click Close.

12. In the *Application Tree*, select **Layer** under **Design**.

Click » to see the Active Laminate field.

13. In the main toolbar area, select **MFG-TOOL** from the pull-down list for *Active Laminate* as shown in Figure 13–13.

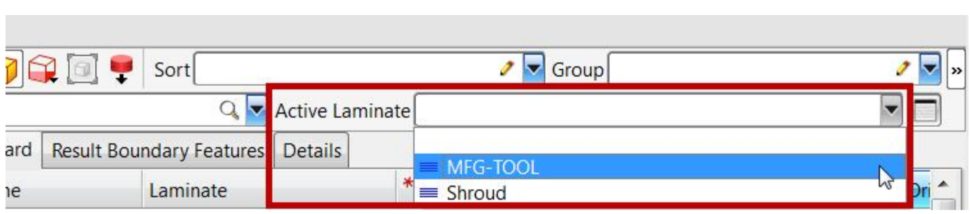
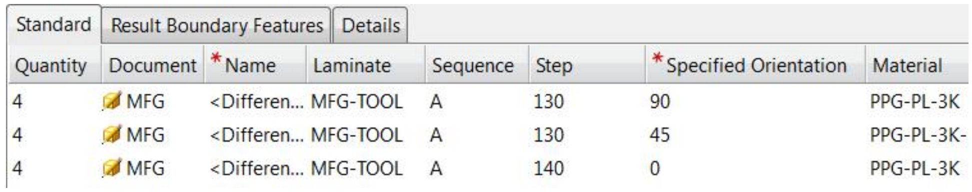


Figure 13–13

Note that the material for the 45 degree layers is PPG-PL-3K-36, that is different than others.

14. Review the list of layers as shown in Figure 13–14. Note that Figure 13–14 shows layers grouped by Specified Orientation.



Standard | Result Boundary Features | Details

Quantity	Document	*Name	Laminate	Sequence	Step	*Specified Orientation	Material
4	MFG	<Differen...	MFG-TOOL	A	130	90	PPG-PL-3K
4	MFG	<Differen...	MFG-TOOL	A	130	45	PPG-PL-3K-
4	MFG	<Differen...	MFG-TOOL	A	140	0	PPG-PL-3K

Figure 13–14

Task 6 - Create Offset Specification and Stagger Profile.

In this task, you create a constant offset specification and a stagger profile to use when creating splice groups.

1. In the *Documentation Browser,* right-click on **MFG** and select **Make Storage Document**.
2. In the *Application Tree*, select **Offset Specification** under **Specification**.
3. Click (Create New).
4. Enter [.25] (6.35 mm) for *Offset Distance*.
5. Click OK to save.
6. In the *Application Tree*, select **Stagger Profile** under **Specification**.
7. Click (Create New).
8. Leave *Stagger Profile Pattern* as **Linear Ascending** as shown in Figure 13–15.

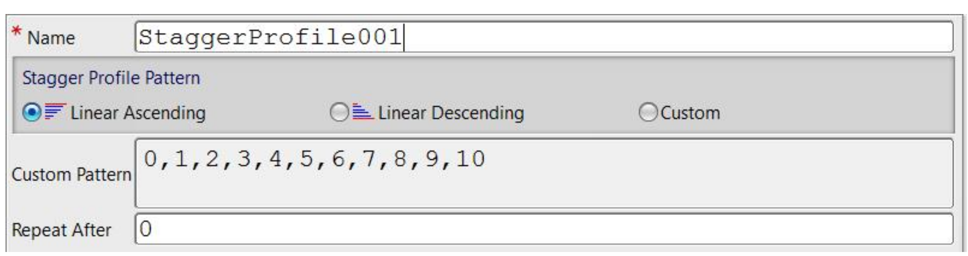


Figure 13–15

9. Click OK to save **StaggerProfile001**.

Task 7 - Create Splice Groups for 0 degree plies.

1. In the *Application Tree*, select **Splice Group** under **Manufacturing**.

2. Click  (Create New).

3. Specify the following parameters.

The Offset Direction is on the ply origin side of the splice curve which is the default direction. Use Reverse Offset Direction *to change direction if required.*

- Name: **SpliceGroup 0A**
- Laminate: **MFG-TOOL**
- Orientation: **0**
- Splice Curves: **Splice Curve 1** as shown in Figure 13–16.

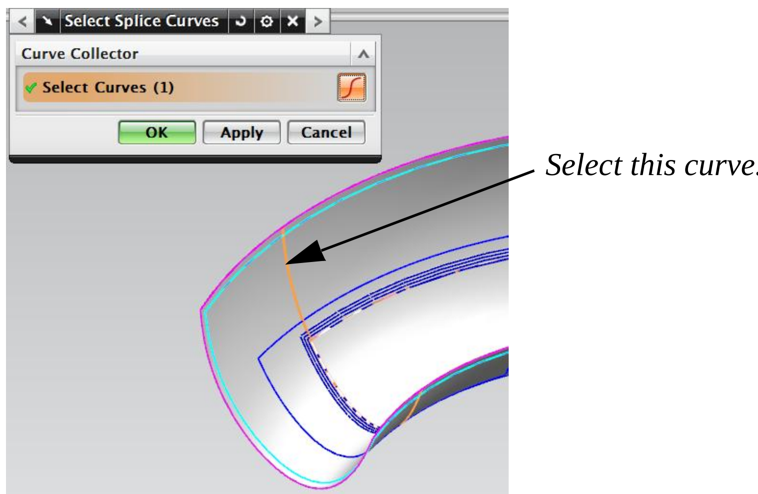


Figure 13–16

- Stagger Profile: **StaggerProfile001**

4. Click Options.

5. Clear the **Only Splice Wider Than Material Width** option as shown in Figure 13–17.

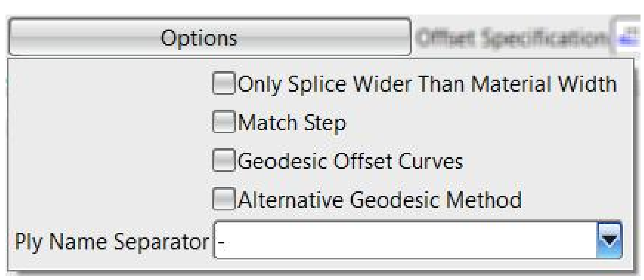


Figure 13–17

6. Click OK to save the splice group.
7. In the Splice Group list, select **SpliceGroup 0A**.
8. Right-click and select **Create Based On**.
9. Specify the following parameters as shown in Figure 13–18.

   - Name: **SpliceGroup 0B**
   - Splice Curves: **Splice Curve 2** (Remember to clear **Splice Curve 1** first)

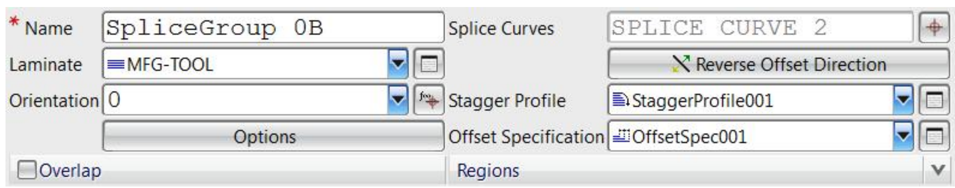


Figure 13–18

10. Click Reverse Offset Direction to indicate the offset direction on the ply origin side of the splice curve as shown in Figure 13–19.

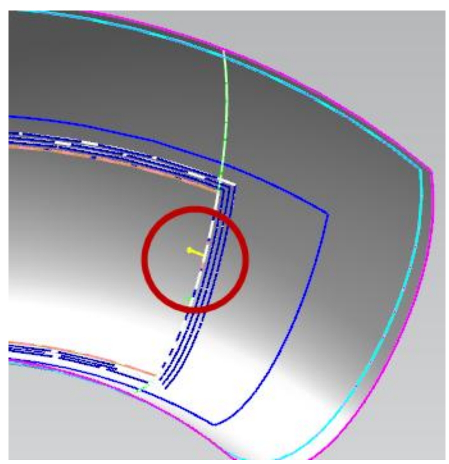
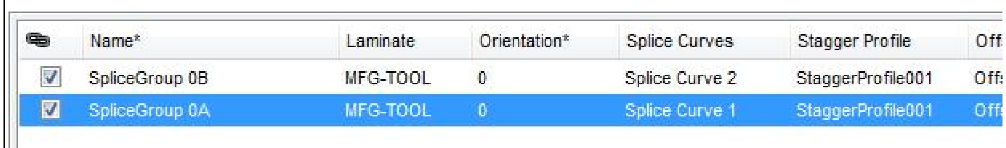


Figure 13–19

11. Click OK to save the splice group.

Task 8 - Assign Splice Groups to 0 degree layers and Splice Plies.

1. In the *Application Tree*, select **Layer** under **Design**.
2. Select **Specified Orientation** from the pull-down list for *Group*.
3. Select the 0 degree orientation layer group, right-click, and select **Modify**.
4. Next to the *Splice Groups* field, click (Link with Link Dialog).
5. Click the checkboxes to link both **SpliceGroup 0A** and **SpliceGroup 0B** as shown in Figure 13–20.



	Name*	Laminate	Orientation*	Splice Curves	Stagger Profile	Off
☑	SpliceGroup 0B	MFG-TOOL	0	Splice Curve 2	StaggerProfile001	Off
☑	SpliceGroup 0A	MFG-TOOL	0	Splice Curve 1	StaggerProfile001	Off

Figure 13–20

6. Click OK.
7. A message window opens prompting you that the Splice Group edit may have invalidated origins of spliced plies. Click OK.
8. Click Splice Plies.
9. A message window opens prompting you that splice plies were created successfully as shown in Figure 13–21.

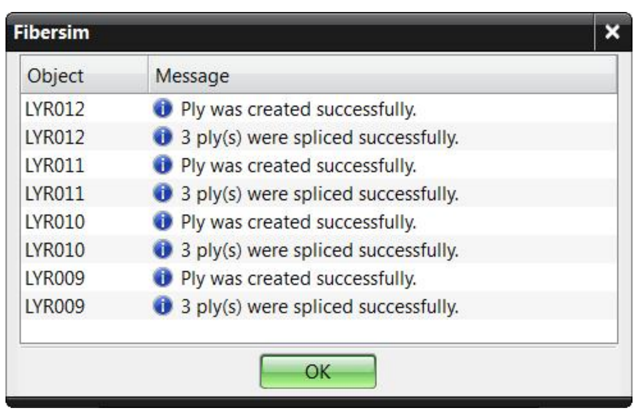


Figure 13–21

10. Click OK twice.

Task 9 - Rename Spliced Plies.

1. In the *Application Tree*, select **Ply** under **Manufacturing**. The created plies are displayed as shown in Figure 13–22.

Standard | Net Geometry | Extended Geometry | Analysis | Result Boundary Features | Det

Document	*Name	Parent	Sequence	Step	Material	Rosette
MFG	P009-A	LYR009	A	140	PPG-PL-3K	MFG-RO
MFG	P009-B-A	LYR009	A	140	PPG-PL-3K	MFG-RO
MFG	P009-B-B	LYR009	A	140	PPG-PL-3K	MFG-RO
MFG	P009-B-C	LYR009	A	140	PPG-PL-3K	MFG-RO
MFG	P010-A-A	LYR010	A	140	PPG-PL-3K	MFG-RO
MFG	P010-A-B	LYR010	A	140	PPG-PL-3K	MFG-RO
MFG	P010-A-C	LYR010	A	140	PPG-PL-3K	MFG-RO
MFG	P010-B	LYR010	A	140	PPG-PL-3K	MFG-RO
MFG	P011-A-A	LYR011	A	140	PPG-PL-3K	MFG-RO
MFG	P011-A-B	LYR011	A	140	PPG-PL-3K	MFG-RO
MFG	P011-A-C	LYR011	A	140	PPG-PL-3K	MFG-RO
MFG	P011-B	LYR011	A	140	PPG-PL-3K	MFG-RO
MFG	P012-A-A	LYR012	A	140	PPG-PL-3K	MFG-RO
MFG	P012-A-B	LYR012	A	140	PPG-PL-3K	MFG-RO
MFG	P012-A-C	LYR012	A	140	PPG-PL-3K	MFG-RO
MFG	P012-B	LYR012	A	140	PPG-PL-3K	MFG-RO

Figure 13–22

2. Note that the naming convention follows the order of splicing:

   - **SpliceGroup 0A** creates P009-A
   - **SpliceGroup 0B** creates P009-B-A, P009-B-B and P009-B-C

3. Multi-select the four **P009** plies and switch to the NX window to display as shown in Figure 13–23.

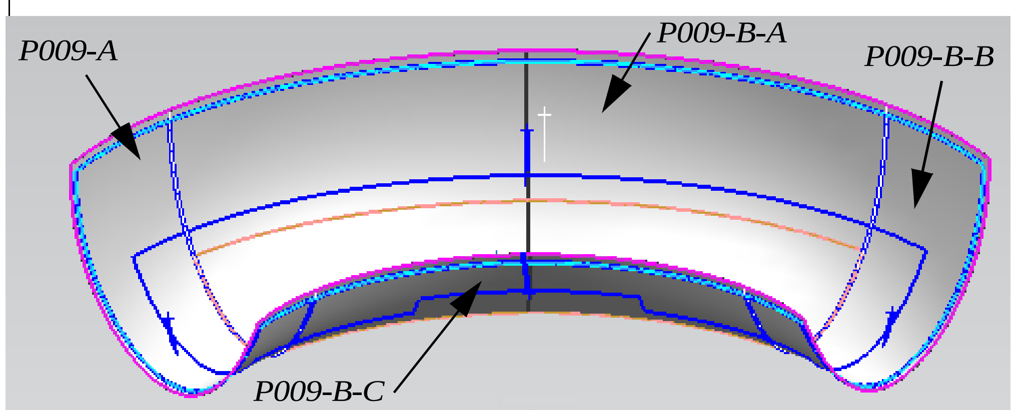


Figure 13–23

4. With the four P009 plies highlighted, right-click, and select **Modify** as shown in Figure 13–24.

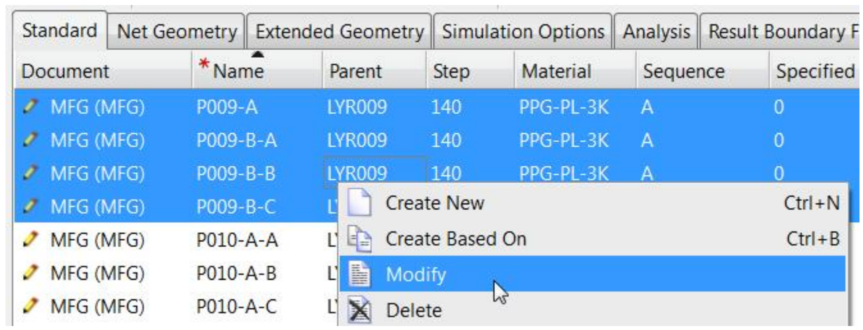


Figure 13–24

5. In the *Name* field, enter [P009-A,@] and press <Enter>.

6. Click OK to save.

7. The renamed P009 plies display as shown in Figure 13–25.

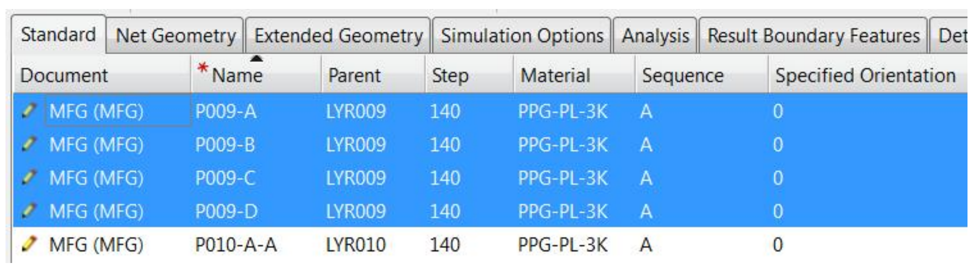


Figure 13–25

8. Use the same steps to rename the other spliced plies (P010, P011 and P012).

Task 10 - Generate Plies for all 45 Degree Layers.

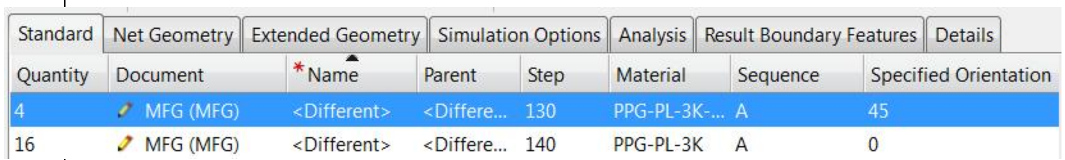
1. In the *Application Tree*, select **Layer** under **Design**.

2. Select the 45 degree layers and in the Layer toolbar, click (Generate Plies).

3. A message window prompting you that the four plies were created successfully opens. Click OK.

4. In the *Application Tree*, select **Ply** under **Manufacturing**.

5. Group by **Specified Orientation** and select the 45 degree plies as shown in Figure 13–26.



Quantity	Document	*Name	Parent	Step	Material	Sequence	Specified Orientation
4	MFG (MFG)	<Different>	<Differe...	130	PPG-PL-3K-...	A	45
16	MFG (MFG)	<Different>	<Differe...	140	PPG-PL-3K	A	0

Figure 13–26

6. Activate the NX window to view the new plies as shown in Figure 13–27.

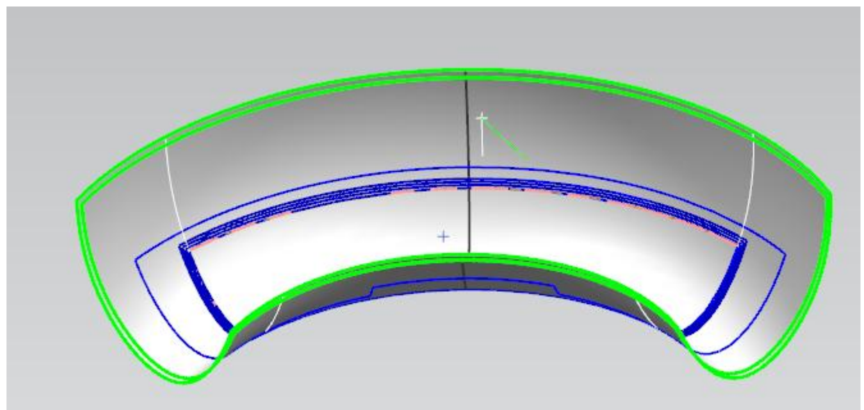


Figure 13–27

Task 11 - Create Overlap Splice Laminate Region.

7. In the *Application Tree*, select **Laminate Regions** under **Specification**.

8. Click (Create New) and select **Overlap Splice Region** as shown in Figure 13–28.

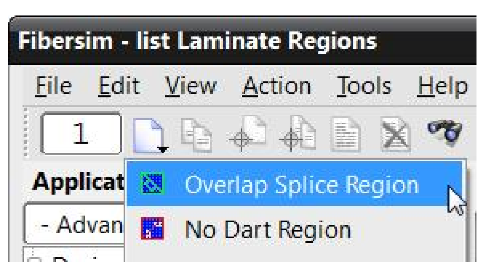


Figure 13–28

*An **Overlap Splice Region** is used to specify an area in which plies can be spliced with an overlap. Splices outside the overlap region will be performed as butt splices.*

9. Specify the following parameters:
   - Name: **OverlapSpliceRegion001**
   - Laminate: **MFG-TOOL**
   - Origin: **Overlap Origin** (See Figure 13–29)
   - Boundary: **Overlap Region** (See Figure 13–29)

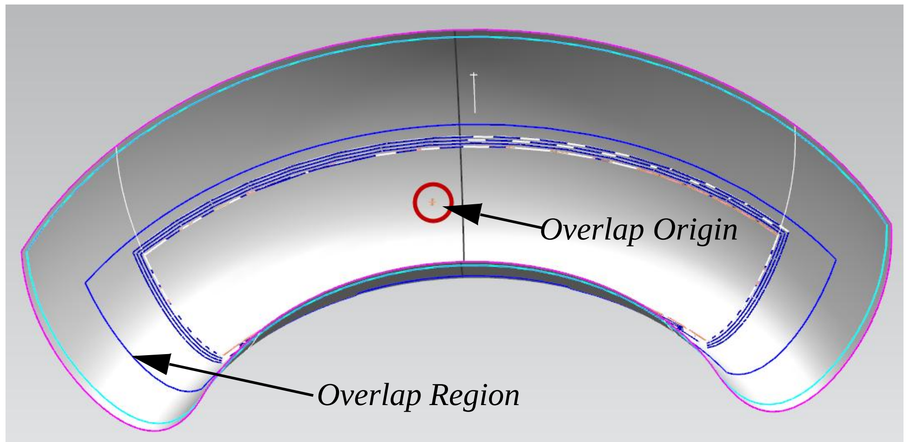
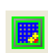
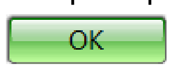


Figure 13–29

10. Click  to save.
11. In the *Application Tree*, select **Ply** under **Manufacturing**.
12. Ungroup the list and double-click **P006** to modify.
13. Click  (Net Producibility).
14. A message window opens prompting you that the material width is exceeded. Click .
15. Select *Net Geometry* tab.
16. Expand **Material Width Lines** area.
17. Specify the following parameters as shown in Figure 13–30.
   - Material Width Offset: **-10 in (-254 mm)**
   - Positioning Method: **Centered**

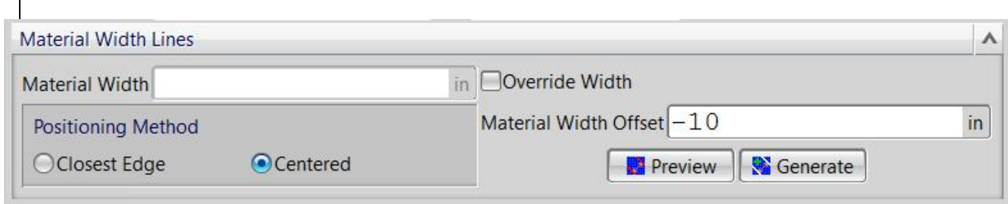
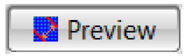


Figure 13–30

18. Click 

.

19. Activate the NX window to display the material width lines as shown in Figure 13–31.

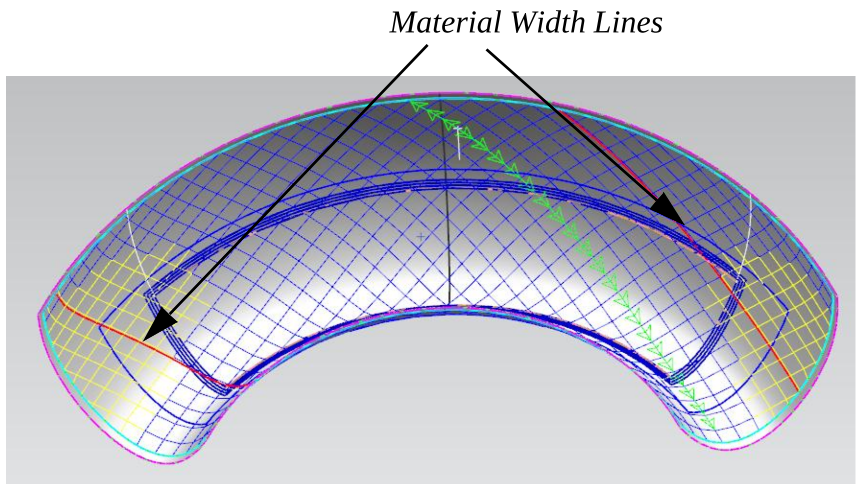
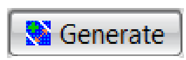


Figure 13–31

20. In Fibersim, click 

.

21. Click 

to save.

Task 12 - Create Splice Group for 45 degree plies.

1. In the *Application Tree*, select **Layer** under **Design**.
2. Double-click the 45 degree orientation layers to modify.
3. Next to the *Splice Groups* field, click  (Link with Link Dialog).
4. Click  (Create New).
5. Specify the following parameters as shown in Figure 13–33.
   - Name: **SpliceGroup 45**
   - Laminate: **MFG-TOOL**
   - Orientation: **45**
   - Splice Curves: select the generated material width lines

- Offset Direction: set pointing left to the net boundary from the first splice curve selected as shown in Figure 13–32.

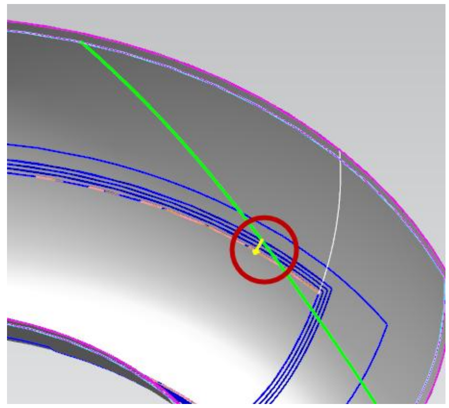


Figure 13–32

Recess Distance *shifts the beginning of the overlap splice inside the overlap region boundary by the specified amount*

- Overlap Distance: **1 in (25.4mm)**
- Overlap Region: **OverlapSpliceRegion001**
- Recess Distance: **0.1 in (2.54mm)**

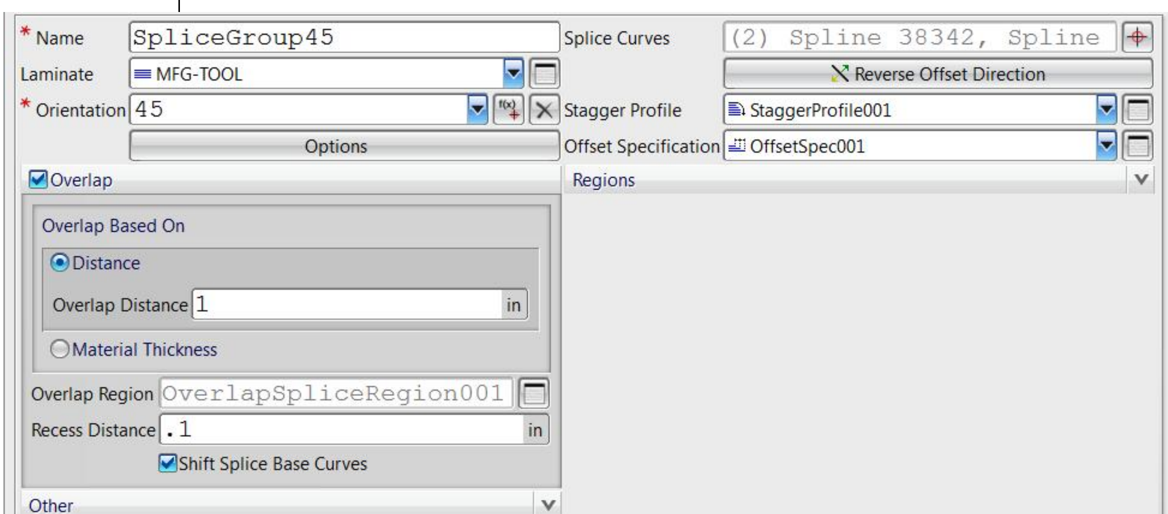
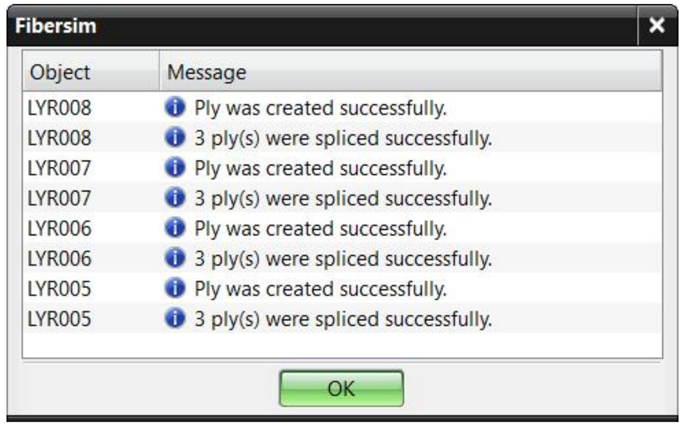


Figure 13–33

6. Click Options.
7. Clear the **Only Splice Wider Than Material Width** option.
8. Click OK twice to save and assign the SpliceGroup 45.
9. A message window prompting you that Splice Group edit may have invalidated origins of spliced plies. Click OK.

10. Click **Splice Plies**.

11. A message window opens prompting you that the plies were created successfully as shown in Figure 13–34.



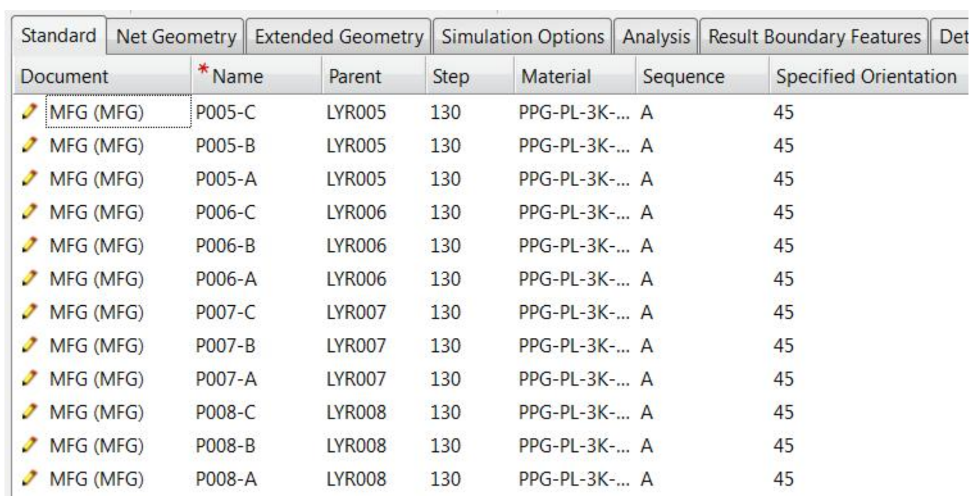
Object	Message
LYR008	Ply was created successfully.
LYR008	3 ply(s) were spliced successfully.
LYR007	Ply was created successfully.
LYR007	3 ply(s) were spliced successfully.
LYR006	Ply was created successfully.
LYR006	3 ply(s) were spliced successfully.
LYR005	Ply was created successfully.
LYR005	3 ply(s) were spliced successfully.

Figure 13–34

12. Click **OK**.

13. Click **OK** to save.

14. In the *Application Tree*, select **Ply** under **Manufacturing**. The created plies are displayed as shown in Figure 13–35.



Document	*Name	Parent	Step	Material	Sequence	Specified Orientation
MFG (MFG)	P005-C	LYR005	130	PPG-PL-3K-...	A	45
MFG (MFG)	P005-B	LYR005	130	PPG-PL-3K-...	A	45
MFG (MFG)	P005-A	LYR005	130	PPG-PL-3K-...	A	45
MFG (MFG)	P006-C	LYR006	130	PPG-PL-3K-...	A	45
MFG (MFG)	P006-B	LYR006	130	PPG-PL-3K-...	A	45
MFG (MFG)	P006-A	LYR006	130	PPG-PL-3K-...	A	45
MFG (MFG)	P007-C	LYR007	130	PPG-PL-3K-...	A	45
MFG (MFG)	P007-B	LYR007	130	PPG-PL-3K-...	A	45
MFG (MFG)	P007-A	LYR007	130	PPG-PL-3K-...	A	45
MFG (MFG)	P008-C	LYR008	130	PPG-PL-3K-...	A	45
MFG (MFG)	P008-B	LYR008	130	PPG-PL-3K-...	A	45
MFG (MFG)	P008-A	LYR008	130	PPG-PL-3K-...	A	45

Figure 13–35

Task 13 - Generate 90 degree plies and run producibility.

1. In the *Application Tree*, select **Layer** under **Design**.
2. Select the 90 degree layers and in the Layer toolbar, click (Generate Plies).
3. A message window prompting you that the ply was created successfully opens. Click OK.
4. In the *Application Tree*, select **Ply** under **Manufacturing**.
5. Highlight **P003**.
6. In the ply toolbar, click (Net Producibility).
7. Activate the NX window and note the problem areas as shown in Figure 13–36.

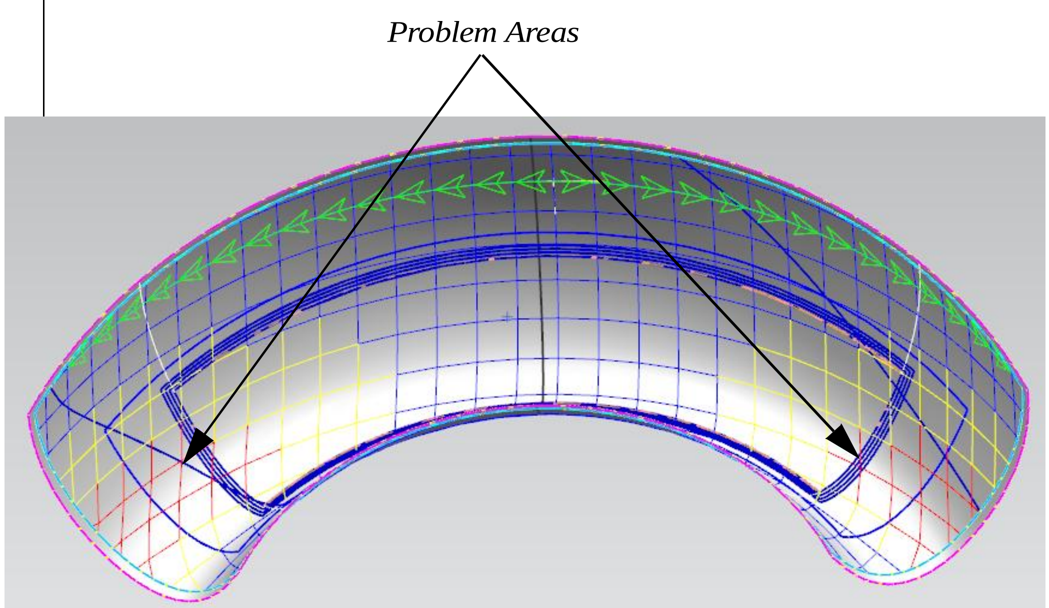


Figure 13–36

Task 14 - Create Slit Darts.

1. In the *Application Tree*, select **Darts** under **Manufacturing**.
2. Click (Create New) and select **Slit Dart**.

3. Specify the following parameters:
   - Name: **SlitDart001**
   - Laminate: **MFG-TOOL**
   - Base Curve Points: **IndicationPoint.1** and **IndicationPoint.2** as shown in Figure 13–37. (Make sure that the Point On Face selection filter is enabled.)

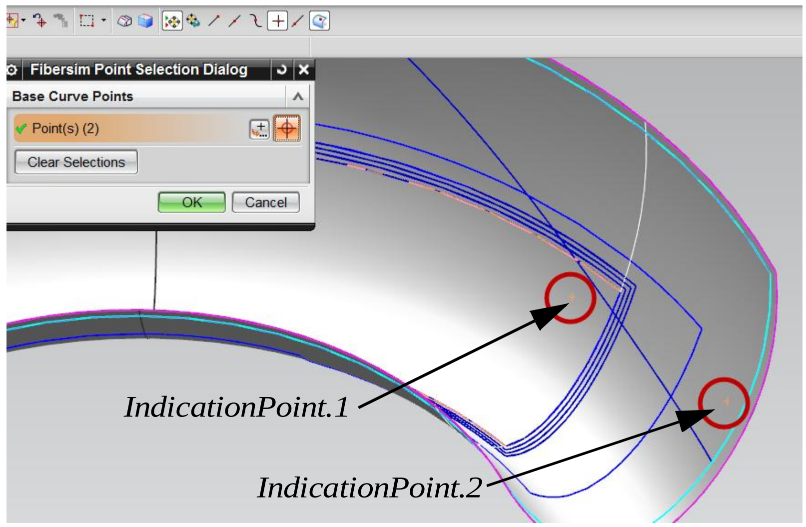


Figure 13–37

4. Click Generate Dart Feature.
5. Click OK to save.
6. Use the same procedure to create **SlitDart002**, **SlitDart003**, and **SlitDart004** in approximate locations as shown in Figure 13–38.

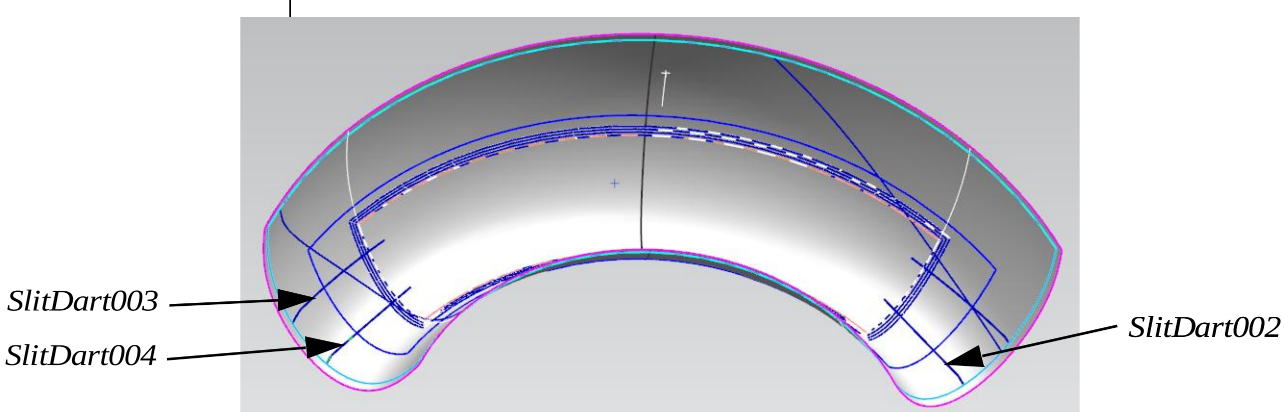


Figure 13–38

Task 15 - Create Dart Group.

1. In the *Application Tree*, select **Dart Group** under **Manufacturing**.
2. Click (Create New).
3. Specify the following parameters:
   - Name: **90 degree Dart Group1**
   - Laminate: **MFG-TOOL**
   - Orientation: **90**
4. Next to the *Darts* field, click (Link with Link Dialog).
5. Click the checkboxes for **SlitDart001** and **SlitDart002** as shown in Figure 13–39.

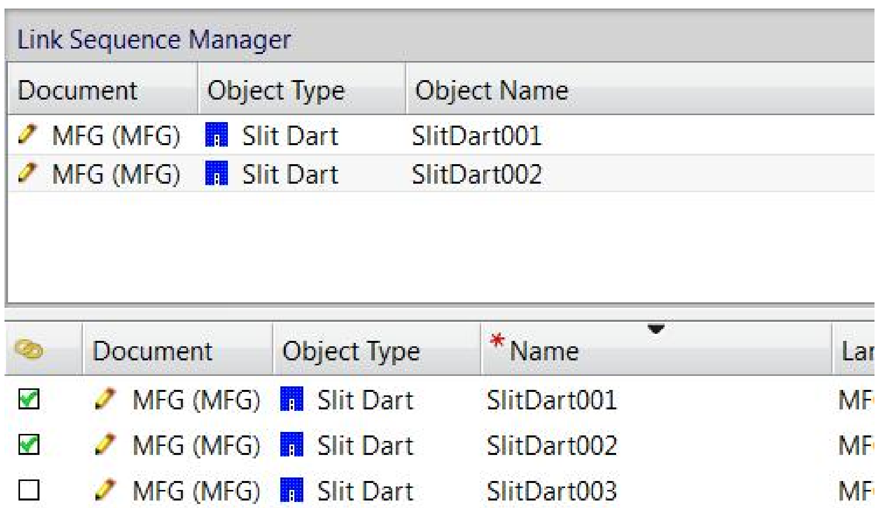
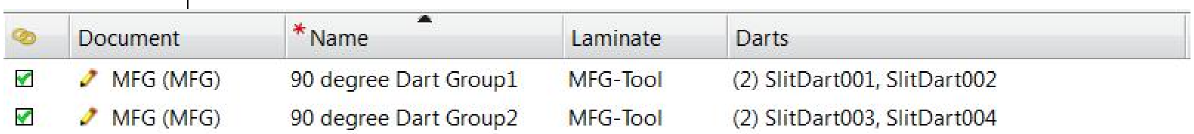


Figure 13–39

6. Click OK.
7. Click OK to save the dart group.
8. Use the same procedure to create **90 degree Dart Group2** using **SlitDart003** and **SlitDart004**.

Task 16 - Assign Dart Groups to 90 degree layers.

1. In the *Application Tree*, select **Layer** under **Design**.
2. Double-click the 90 degree orientation layer group to modify.
3. Next to the *Dart Groups* field, click (Link with Link Dialog).
4. Click the checkbox to link both the **90 degree Dart Group1** and **90 degree Dart Group2** to the layers as shown in Figure 13–40.



	Document	*Name	Laminate	Darts
☑	MFG (MFG)	90 degree Dart Group1	MFG-Tool	(2) SlitDart001, SlitDart002
☑	MFG (MFG)	90 degree Dart Group2	MFG-Tool	(2) SlitDart003, SlitDart004

Figure 13–40

5. Click OK
6. Click Dart Plies.
7. A message window opens prompting you that four plies were darted successfully as shown in Figure 13–41.

Figure 13–41

8. Click OK.
9. Click OK to save.

10. In the *Application Tree*, select **Ply** under **Manufacturing**.

11. Highlight **P003** and click  (Net Producibility) on the Ply toolbar.

12. Activate the NX window and note that the problems have been resolved as shown in Figure 13–42.

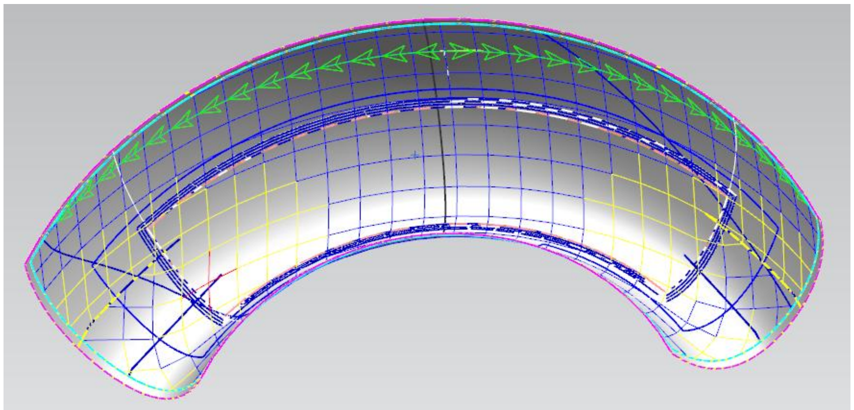


Figure 13–42

13. Close Fibersim.

14. Save and close the model.

Project 4

C Pillar

This chapter includes:

✓**Exercise P4: C Pillar**

Exercise P4 C Pillar

User Guide Reference:
2.5 Flat Pattern/
Producibility Simulation
B.3.7 Splice Ply

In this project, you will design a C Pillar by resolving producibility issues and following design guides given by CAE.

Goal

This project will test your knowledge on how to:

- ✓ **Create ply layup following CAE design guides**
- ✓ **Resolve any producibility issues in important areas**
- ✓ **Create flat patterns**
- ✓ **Create 3D and 2D documentation**

Given

The existing geometry and design details are:

- ✓ **Tool surface**
- ✓ **Plane that represents the zero direction of the part**
- ✓ **Hole geometry**
- ✓ **Special area geometry used in CAE design guides**

Estimated Time

2 hr 30 min

Task 1 - Open a part.

1. Open **PROJECT4_C_PILLAR.prt**. The model displays as shown in Figure P4–1.

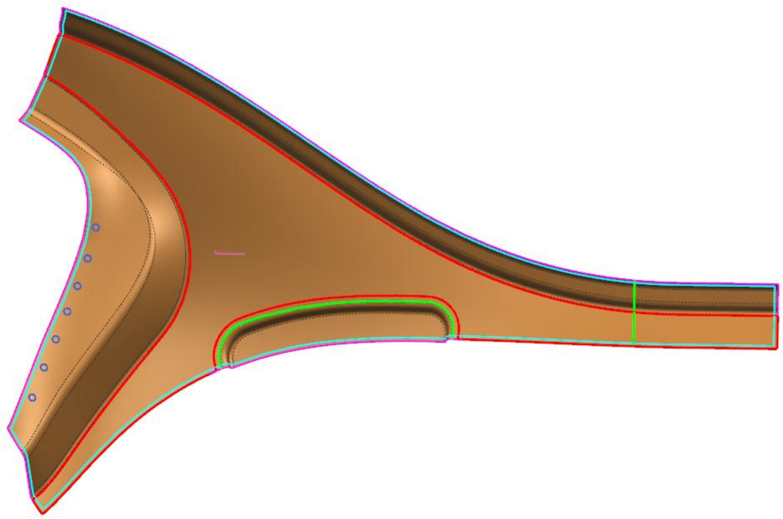


Figure P4–1

Task 2 - Review the Fibersim geometry.

1. The existing OML, Zone, 0 direction, Cutouts, and critical area geometry are organized in layers as shown in Figure P4–2.

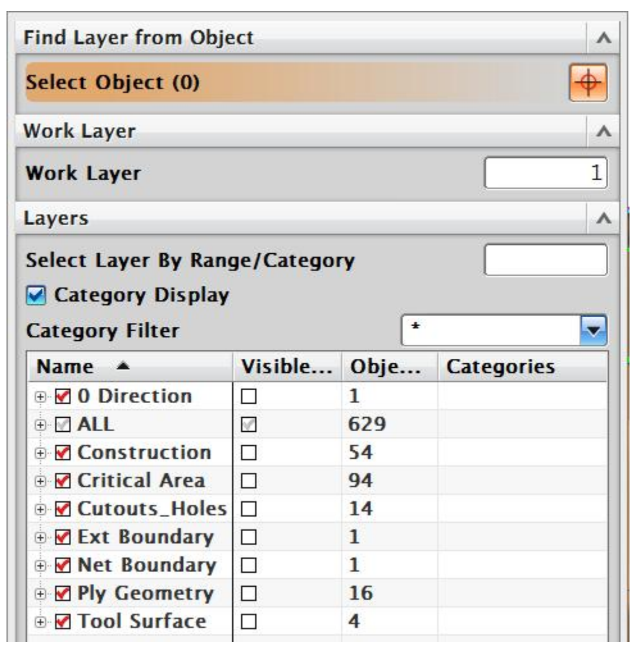


Figure P4–2

Task 3 - Create a Laminate and Rosette.

1. Using the geometry given, create a Laminate.
2. Create the necessary geometry for Rosette using the **0 Direction** curve, and create a Rosette.

Task 4 - Create Layup.

Analysis has determined that the layup must be:

- 4 Full body layers: two 0/90 deg and two +/-45 deg of DRY-FINE
- Within **Center Reinforcement** area as indicated in Figure P4–3:
  - Three 0 deg layers of T-24
  - One +/- 45 deg layer of DRY-5H-3K

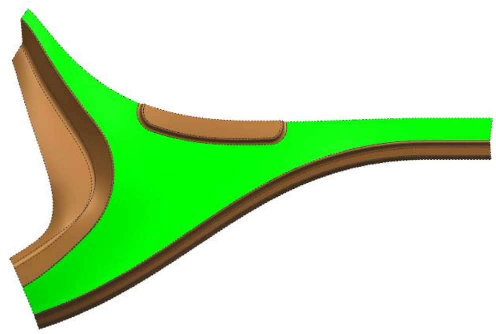


Figure P4–3

- Within **End Reinforcement** area as indicated in Figure P4–4:
  - Two 90 deg layers of T-24
  - One +/-45 deg layer of DRY-5H-3K

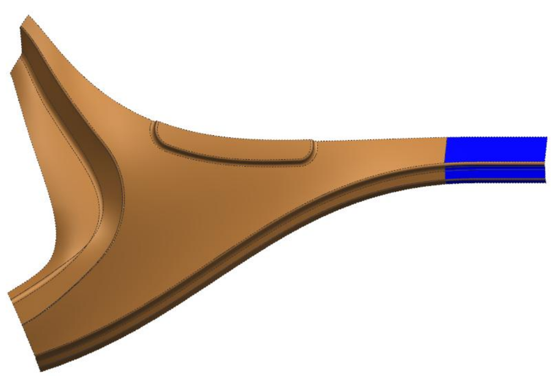


Figure P4–4

- All reinforcement layers must be dropped off 5 mm (0.1969 in) to avoid stress concentrations.
    - Drop-offs must occur on flat surface of the part to help manufacturing position plies.
- Reinforcements must be interleaved between full-body layers.

Task 5 - Create Cutouts.

Define cutouts in all plies using the given geometry.

Task 6 - Run Net Producibility.

Run Net Producibility on all of the plies and address any issues based on the CAE design guides below:

- Must resolve deformation problems within the area marked as critical in Figure P4–5.
- Fibers must not deviate past 8 degrees within the critical area
- No bridging/Tearing conditions must exist throughout the entire part.
- No darting is allowed within the critical area.
- Splicing is allowed but must be staggered at least 25.4 mm (1 in).
- Butting splices are allowed; overlaps in splicing are not necessary.

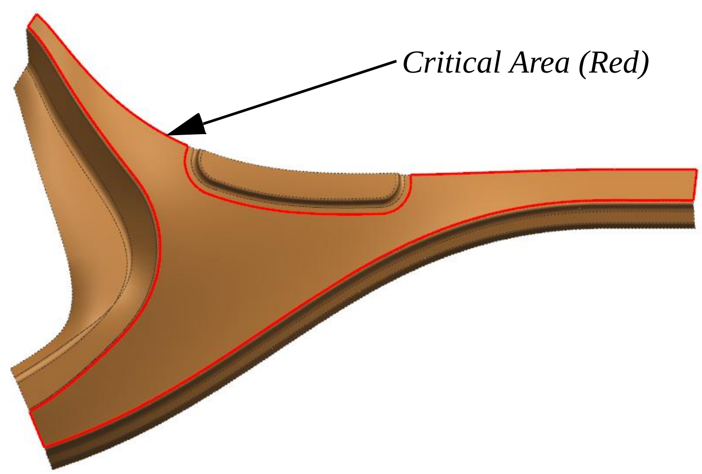


Figure P4–5

Task 7 - Generate Flat Patterns.

When all design rules (listed in Task 6) are met, create **extended flat patterns** for all plies.

Task 8 - Create 3D Documentation.

Create 3D Cross Sections, Core Sample, Ply Table, Ply Callout, and Material Table annotations.

Task 9 - Generate Weight Table.

Run weight table action and review the data given.

Task 10 - Generate Ply Book.

Setup and generate a ply book with appropriate template and text that you feel communicates the layup accurately to manufacturing.