

Siemens NX 9 Design Fundamentals

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Siemens NX 9

Design Fundamentals

A Step by Step Guide

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Preface

This textbook explains how to create solid models, assemblies and drawings using Siemens NX 9. NX is a three dimensional CAD/CAM/CAE software developed by Siemens PLM Software Inc., Germany. This textbook is based on NX 9. Users of earlier releases can use this book with minor modifications. We provide files for exercises via our website. Almost all files are in NX 6.0 so readers can open the files using NX 6.0 and later releases.

It is assumed that readers of this textbook have no prior experience in using Siemens NX for modeling 3D parts. This textbook is suitable for anyone interested in learning 3D modeling using Siemens NX.

Each chapter deals with the major functions of creating 3D features using simple examples and step by step, self-paced exercises. Additional drawings of 3D parts are provided at the end of each chapter for further self exercises. The final exercises are expected to be completed by readers who have fully understood the content and completed the exercises in each chapter.

Topics covered in this textbook

- Chapter 1: Basic components of Siemens NX 9, options and mouse operations.
- Chapter 2: Basic step by step modeling process of NX 9.
- Chapter 3 and 4: Creating sketches and sketch based features.
- Chapter 5: Usage of datums to create complex 3D geometry.
- Chapter 6: Additional modeling commands such as fillet, chamfer, draft and shell.
- Chapter 7: Modification of 3D parts to take advantage of parametric modeling concepts.
- Chapter 8: Copying features, modeling objects and bodies.
- Chapter 9: Additional modeling commands such as trim body, tube, sweep along guide, emboss and various commands in synchronous modeling.
- Chapter 10: Advanced sketch commands.
- Chapter 11: Measuring and verifying 3D geometries.
- Chapter 12 and 13: Constructing assembly structures and creating or modifying 3D parts in the context of assembly.
- Chapter 14 and 15: Creating drawings for parts or assemblies.
- Appendix A: Selecting Objects

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Chapter 6

Additional Modeling Commands - Part I

■ After completing this chapter you will understand

- the types and procedure of **Edge Blend**.
- how to create **Chamfer** on an edge.
- designer's requirements of **Draft**.
- various types and options of **Draft**.
- how to apply wall thickness by using the **Shell** command.

6.1 Detail Modeling

Recall the general modeling process introduced in “2.3 Summary of the Modeling Process” on Page 59.

1. Create a sketch.
 - ▶ Define the sketch plane.
 - ▶ Create the sketch curves and define their shape with constraints.
2. Create 3D geometry.
 - ▶ Create features using the **Extrude** or **Revolve** command to add or remove volumes.
3. Detail Modeling
 - ▶ Complete the model by applying commands such as Edge Blend and Shell that modify edges or faces

In this chapter, we will learn commands that are used in the third step.

Sample Chapter

There are four commands that can be classified in detail modeling: **Edge Blend**, **Chamfer**, **Draft** and **Shell**. Detail modeling commands do not require a sketch. They can be applied on edges or faces that have already been created. Refer to “A.4 Selecting Points, Lines and Faces” on Page 636 for selecting edges, faces and points.

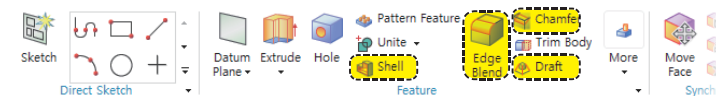


Fig 6-1 Icons for Detail Modeling

6.2 Edge Blend

Sharp edges can be rounded with the **Edge Blend** command which is generally called a fillet.

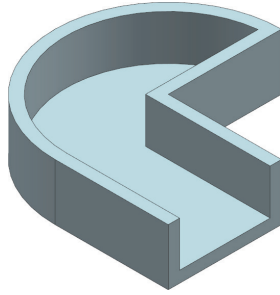


Fig 6-2 Before Edge Blend

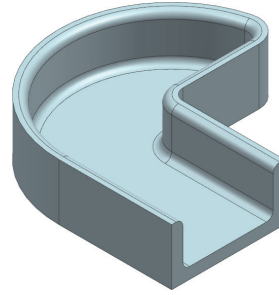


Fig 6-3 After Edge Blend

Edge fillets are applied to smoothen sharp edges. There are two types of sharp edges. Fig6-4 shows concave edges **A** and convex edges **B**.

When a part has been manufactured, convex edges can damage the parts, resulting in deterioration of product quality. They can even cause injury to people who deliver or handle the parts.

Sample Chapter

When a load is applied to a part, the stress will concentrate at the concave edges and a fracture may initiate from these edges. Using the **Edge Blend** command, you can create fillets on the concave or convex edges in the 3D model. Some types of fillets are created inevitably during the manufacturing process and some are required to improve the mechanical performance of the part.

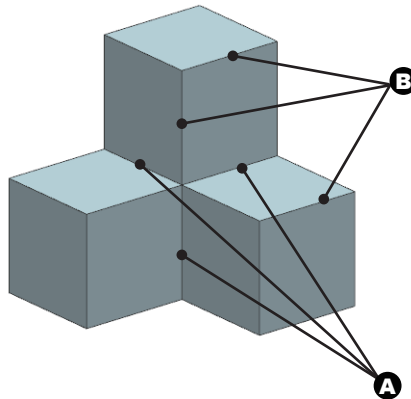


Fig 6-4 Types of Edges

6.2.1 Types of Edge Blend

In NX 8.5, there are four types of edge blends available as shown in Fig 6-5.

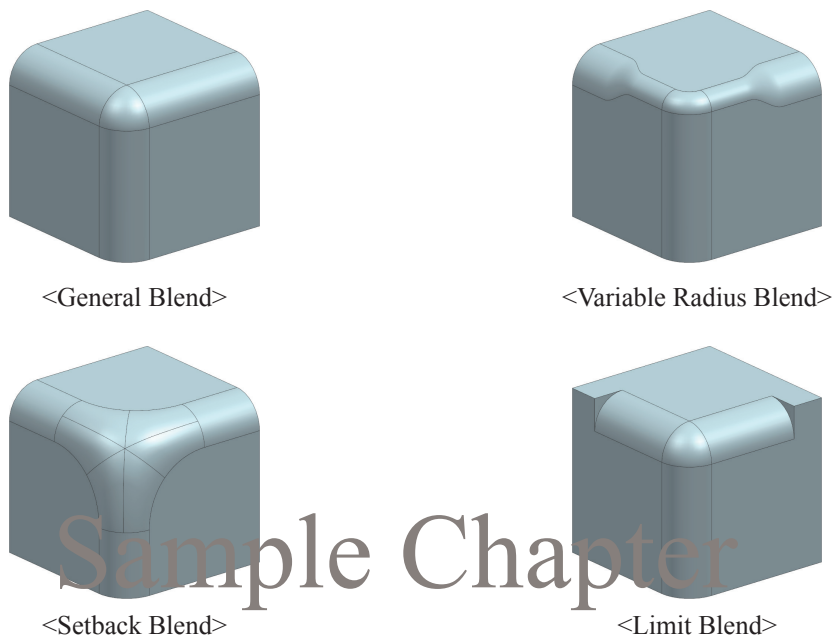


Fig 6-5 Types of Edge Blends

A setback blend can be applied on vertices where three or more edges meet. You can create a smoother blend on vertices.

A limit blend is applied on edges where you cannot create a complete fillet on the edge due to the complexity of the geometry. An edge blend is limited to a portion of the edge by specifying the distance from one end of the selected edge.

You can also apply blends on an edge with various radiuses at the specified points.

This exercise explains how to create general edge blend.

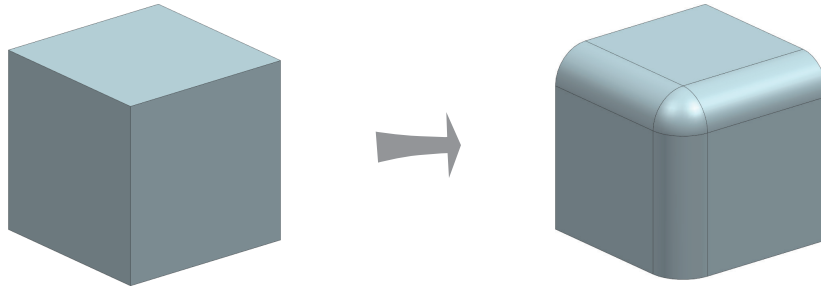


Fig 6-6 Before and After Edge Blend

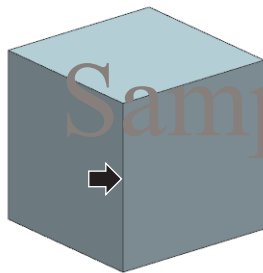


Fig 6-7 A Block

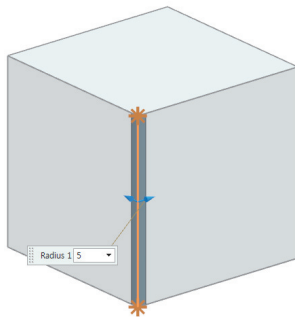


Fig 6-8 Preview of Edge Blend

1. Create a new part file and create a block as shown in Fig 6-7.
2. Click the **Edge Blend** icon in the **Feature** icon group.
3. Reset the dialog box.
4. Select the vertical edge designated by the arrow in Fig 6-7. You can see the preview of the 5 mm radius fillet as shown in Fig 6-8.
5. Delete the **Radius 1** value in the **Edge Blend** dialog box shown in Fig 6-9 and press the **Enter** key. You can see the preview of Radius 1 = 20 fillet in the graphics window. Press the **Apply** button in the dialog box.

The sharp edge is rounded as shown in Fig 6-9.

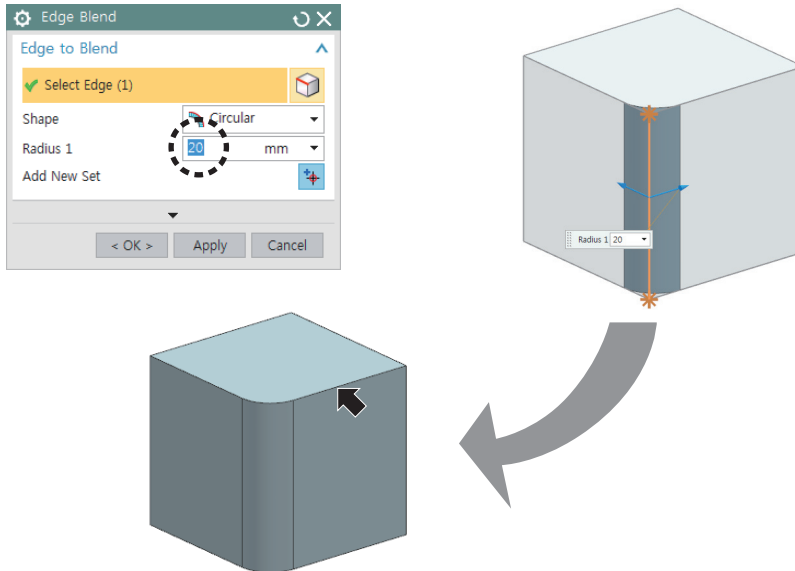


Fig 6-9 Creating R20 Edge Blend

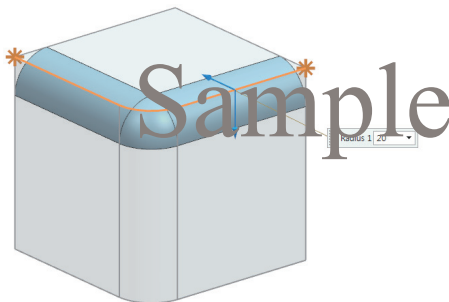


Fig 6-10 Preview of Edge Blend

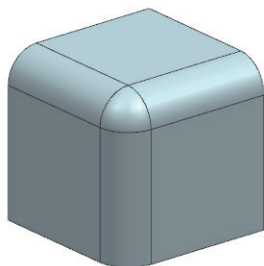


Fig 6-11 Edge Blend Created

6. Ensure that the **Radius 1** value is 20 and select the edge designated by the arrow in Fig 6-9.

All three edges are selected together as shown in Fig 6-10.

7. Press **OK** in the dialog box. A 20mm radius edge blend is applied on the three tangent connected edges as shown in Fig 6-11.

8. Close without saving the file.

END of Exercise

! Curve Rule

The Edge Blend is a command that can be applied on edges. Therefore, the Curve Rule is available at the selection step. The reason why the three edges are selected at the same time in Fig 6-10 is that the default curve rule for Edge Blend is Tangent Curves.

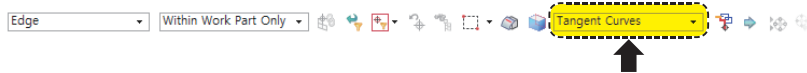


Fig 6-12 Curve Rule

Add New Set **Exercise 02**

You can define edge blends for several radiuses in a single **Edge Blend** feature. Let's learn how to add a new set of blend radiuses in the **Edge Blend** dialog box.

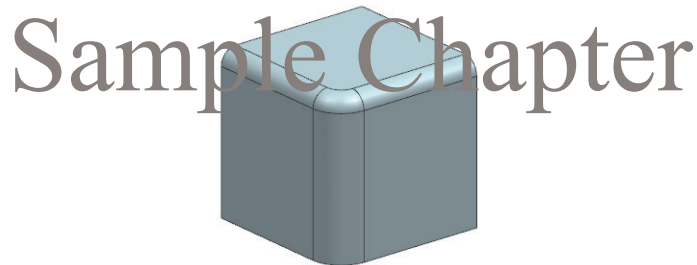


Fig 6-13 Edge Blend to Create

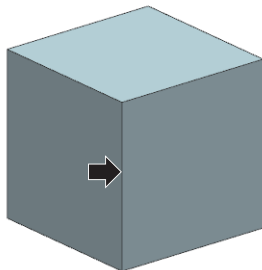


Fig 6-14 A Block

1. Create a new part file and create a block as shown in Fig 6-14.
2. Click the **Edge Blend** icon in the **Feature** icon group.
3. Reset the dialog box.
4. Enter 20 in the **Radius 1** input box and press the **Enter** key.
5. Select the vertical edge designated by the arrow in Fig 6-14.

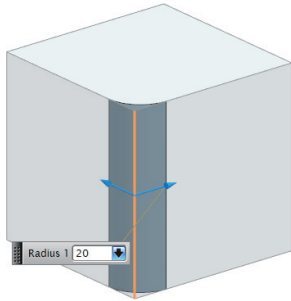


Fig 6-15 Preview of Edge Blend

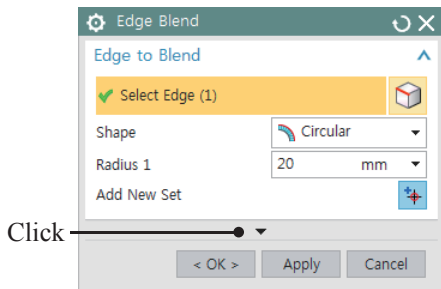


Fig 6-16 More Button

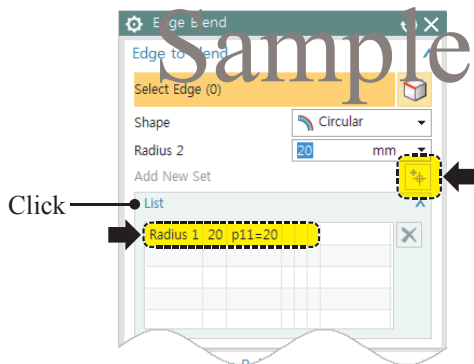


Fig 6-17 Edge Blend Defined

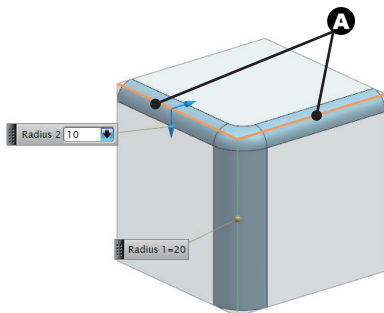


Fig 6-18 Edges to Select

A 20 mm radius edge blend is previewed as shown in Fig 6-15.

6. Click the **More** button to expand the dialog box as shown in Fig 6-16.
7. Expand the **List** area of the dialog box by clicking the title as shown in Fig 6-17.
8. Click the **Add New Set** icon as specified by the upper arrow in Fig 6-17.

The edge blend is defined in the **List** area and the title of the **Radius 1** input box changes to **Radius 2**.

9. Enter 10 in the **Radius 2** input box and press the **Enter** key.
10. Select the two edges designated by **A** in Fig 6-18.
11. Press **OK** in the dialog box.

The edge blend is created as shown in Fig 6-13. Only one edge blend feature is recorded in the **Part Navigator** as shown in Fig 6-19.

12. Close without saving the file.

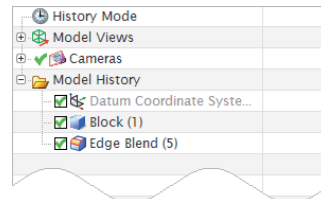


Fig 6-19 Part Navigator

END of Exercise

! Quiz!

Why is the **Tangent Curves** curve rule not applied when selecting edges in Fig 6-18?

Answer: The edge blend of radius 20mm is not created yet but it is just a preview. Therefore, the two edges are not tangent connected.

6.2.2 Shape Option

The shape option defines the sectional shape of the blend surface. While the **Circular** type generates a constant curvature surface along the section line, the **Conic** type generates a variable curvature surface.

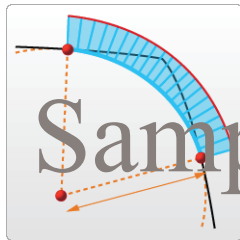


Fig 6-20 Curvature of Circular Type

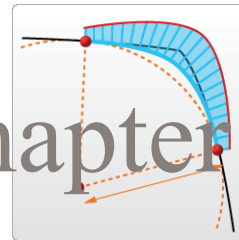


Fig 6-21 Curvature of Conic Type

If you choose the **Conic** type in the **Shape** option, other options are changed to define the variable curvature fillet.

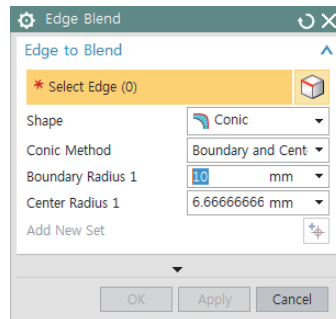


Fig 6-22 Options for Conic Blend

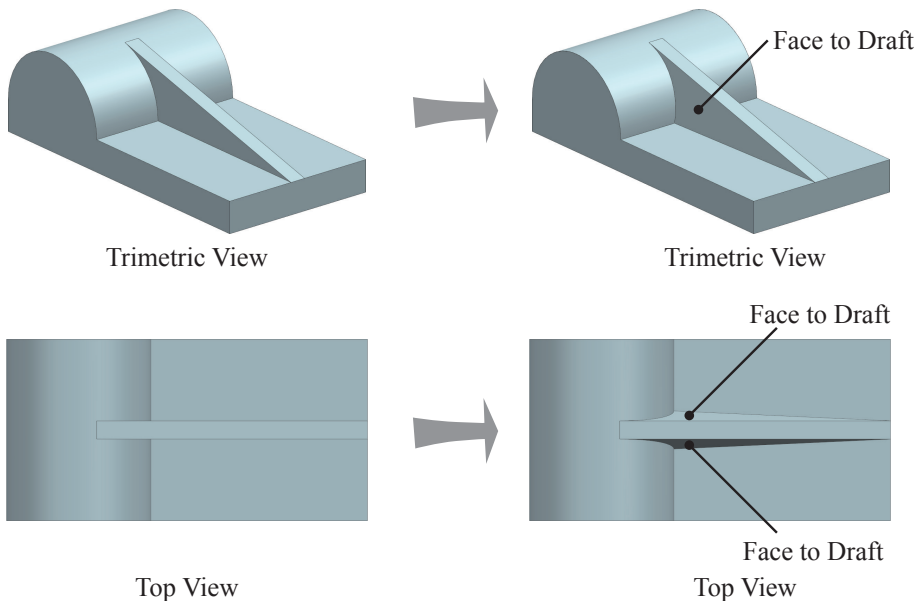


Fig 6-23 Completed Model

END of Exercise

Sample Chapter

! Variable Draft Points

The **Variable Draft Points** option is activated when you apply draft with the **From Edges** type. You can define various draft angles for the specified points.

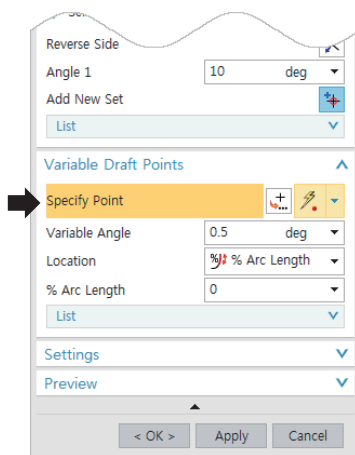


Fig 6-24 Variable Draft Points Option

Faces **A** and **B** in Fig 6-99 require draft. If you choose **Tangent to Faces** as the type of draft, the **Draw Direction** and the **Tangent Faces** option become available. Tangent face at the specified draft angle is created tangent to the selected face.

Open the given file and learn how to apply the **Tangent to Faces** type draft.

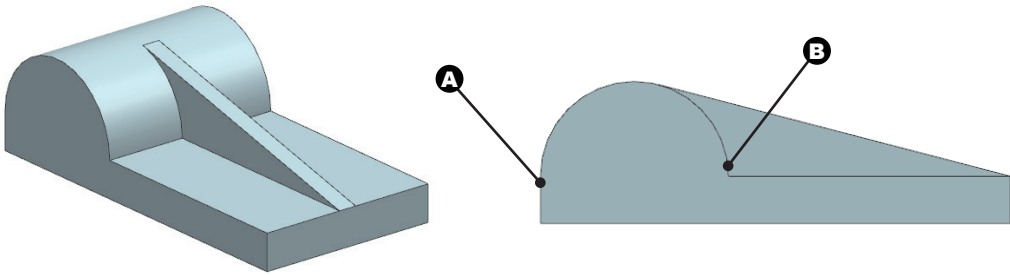


Fig 6-25 Face to Apply Draft

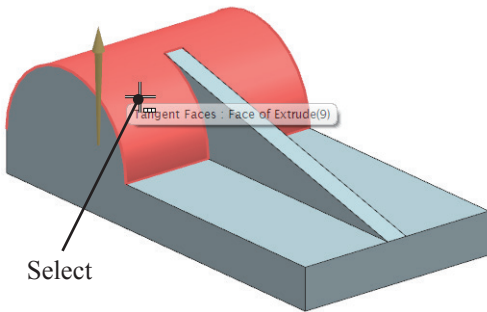


Fig 6-26 Selecting Tangent Face

Sample Chapter

1. Open the given file ch06_ex11.prt.
2. Click the **Draft** icon in the **Feature** icon group.
3. Reset the dialog box.
4. Select **Tangent to Faces** in the **Type** dropdown list. Make sure that the draw direction is set to +ZC direction of WCS and press MB2.
5. Make sure that the **Tangent to Faces** option is highlighted. Note that the **Face Rule** option is set to **Tangent Faces**.
6. Select the cylindrical face specified in Fig 6-100.

7. Enter 20° in the **Angle 1** input box and press the **Enter** key. A preview is shown as Fig 6-101.

8. Press **OK** in the dialog box. Align the view to Front and confirm that the draft angle is applied correctly.

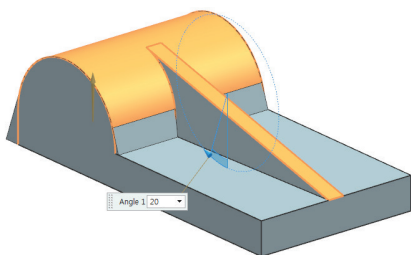


Fig 6-27 Preview

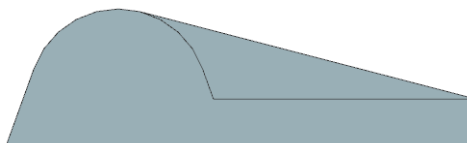


Fig 6-28 Front View

END of Exercise

! Important Sample Chapter

The key point in applying the Tangent to Faces type draft is that you have to select the faces (a) and (b) at the same time. Selecting face (c) does not affect the result because the face is normal to the draw direction.

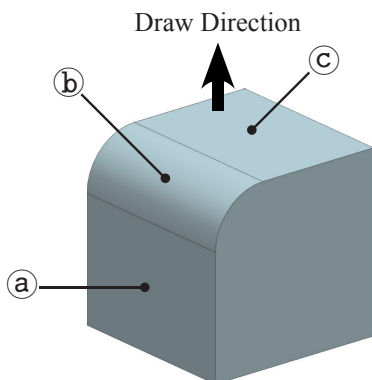


Fig 6-29 Faces in Model

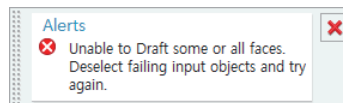
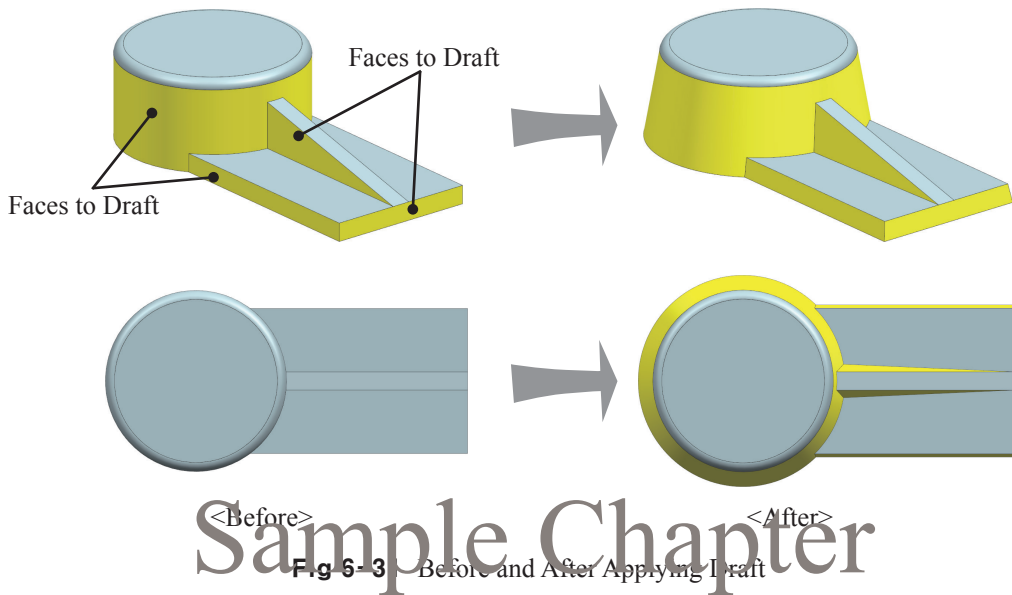


Fig 6-30 Alert Message

Note that if you select only face (a), an alert as shown in Fig 6-104 will appear.

Open the given file ch06_ex12.prt and apply 10° draft for the faces as specified in Fig 6-105. The draw direction is parallel to the ZC axis of WCS.



! Hint!

1. Apply draft for all surfaces that are parallel to the draw direction. The faces to apply draft are displayed in yellow in the file for this exercise.
2. You have to use at least two types of draft.
3. When the completed model is viewed from the Top, the yellow faces should not be visible.

END of Exercise

6.2.3 Other Options in Draft

If you use the **Parting Face** method as the **From Plane or Surface** type draft, you can apply draft with respect to a face, datum plane or a point. Selecting a point as the draft reference, you are specifying a normal plane to the draw direction that passes through the selected point. The reference of the draft is defined at the intersection between the parting face and the faces to draft.

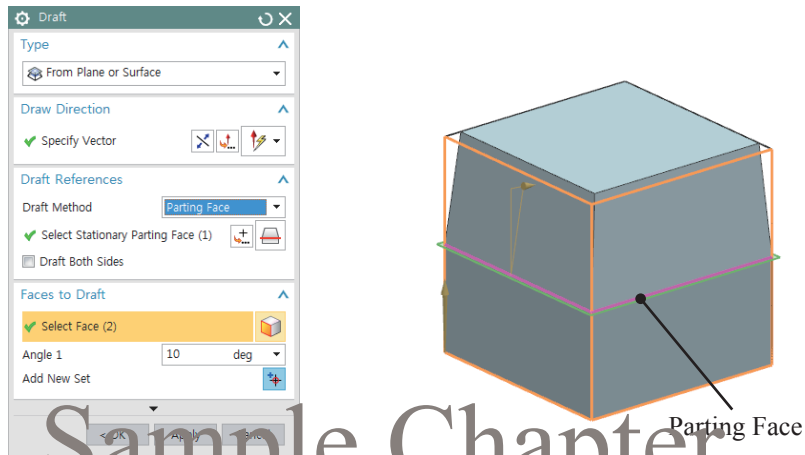


Fig 6-32 Parting Face Method

If you select the **Draft Both Sides** option, you can apply draft for both sides against the parting face. You can even apply a different draft angle for each side by unchecking the **Symmetric Angle** option.

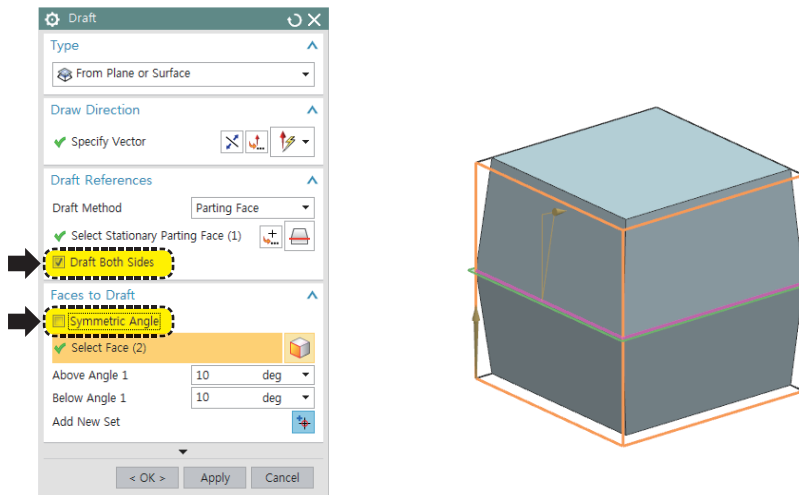


Fig 6-33 Draft Both Sides Option

If you use the **Stationary and Parting Face** method as the **From Plane or Surface** type draft, you can specify both the stationary face and the parting face. Note that the parting and the stationary faces are the same in the **Parting Face** method.

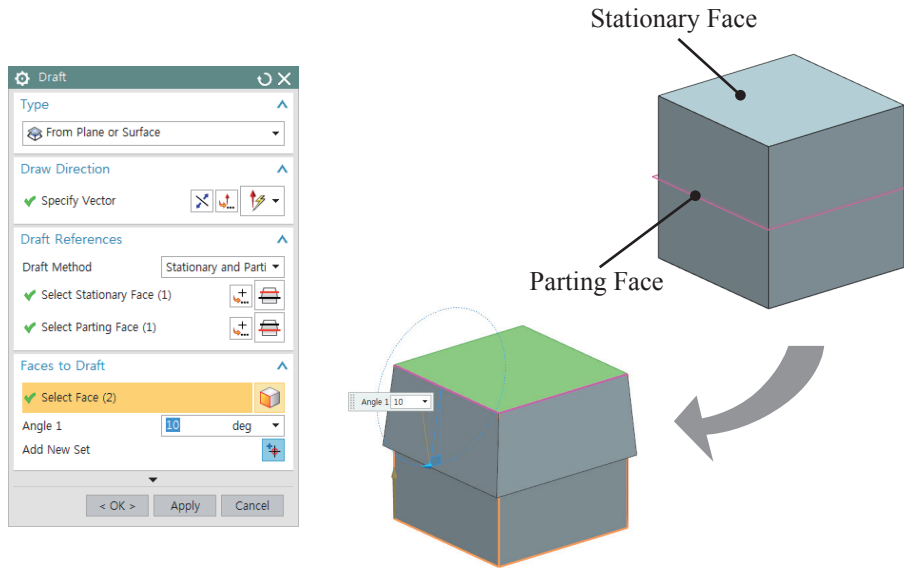


Fig 6-34 Stationary and Parting Face Method

If you use the **From Parting Edges** type draft, you can apply draft with respect to the pre-defined edges. The parting edges can be defined using the **Divide Face** command.

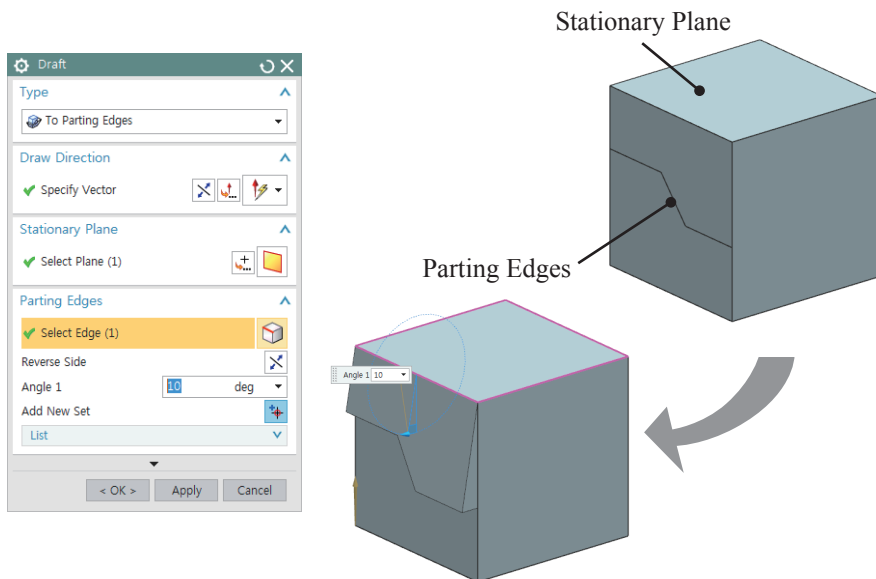


Fig 6-35 From Parting Edges Type

6.3 Shell

This command hollows out a solid body to create a thin wall.

Procedure

1. Click the **Shell** icon in the **Feature** icon group.
2. Reset the **Shell** dialog box.
3. Select the **Face to Pierce**.
4. Enter the thickness of the wall.
5. Press **OK** in the **Shell** dialog box.

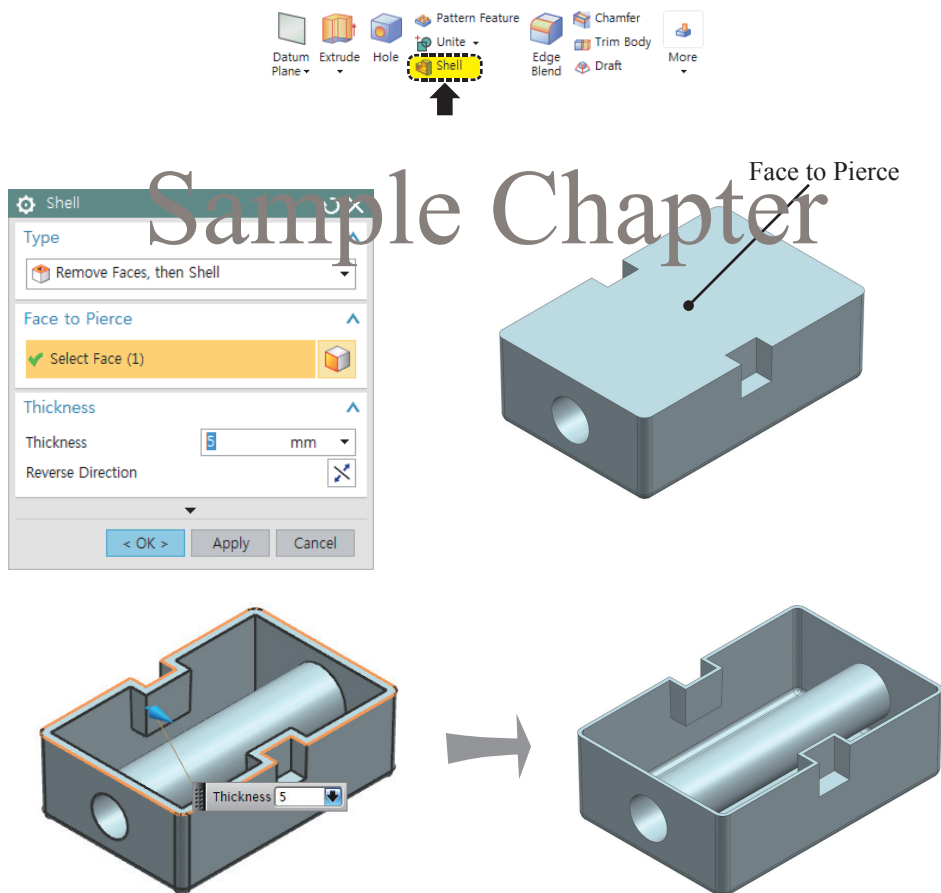


Fig 6-36 Procedure to Apply Shell

You can select as many faces to pierce as required provided that the original solid body is not divided into two or more solid bodies. Fig 6-107 shows an example of specifying several faces to be pierced.

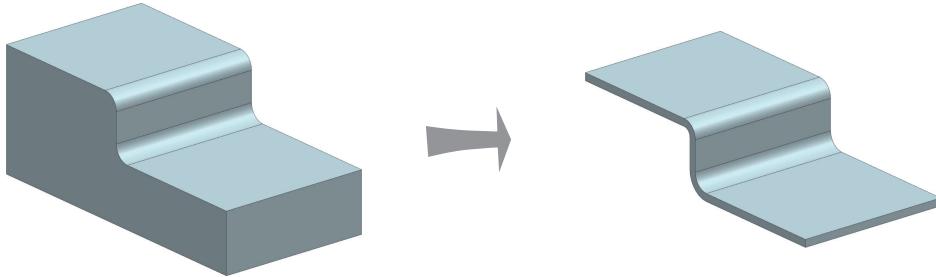


Fig 6-37 Specifying Several Faces to Pierce

6.3.1 Reverse Direction

Thickness of shell command is applied inward to the solid body by default. If you want to apply shell thickness outward, click the **Reverse Direction** option.

Fig 6-108 (a) shows the volume of liquid that will be filled in the bottle. If you want to create a bottle that has the inner volume maintained, you have to apply the **Shell** command by reversing the direction.

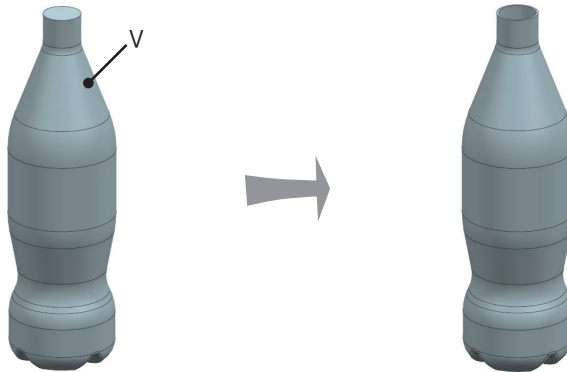


Fig 6-38 Reversing the Shell Thickness Direction

6.3.2 Alternate Thickness

You can apply various thicknesses for the specified faces.

Procedure (Step number corresponds to that in Fig 6-109 and Fig 6-110).

1. Click the **Shell** icon.
2. Reset the dialog box.
3. Select the face to be pierced.
4. Enter the thickness and press the **Enter** key.
5. Expand the **Alternate Thickness** option group (5 in Fig 6-113).
6. Click the **Select Face** option and select the face to apply a different thickness (6 in Fig 6-114).
7. Enter the thickness in the **Thickness 1** input box and press the **Enter** key.
8. Click the **Add New Set** button and apply a different thickness for other faces if required.
9. Press **OK** in the dialog box.

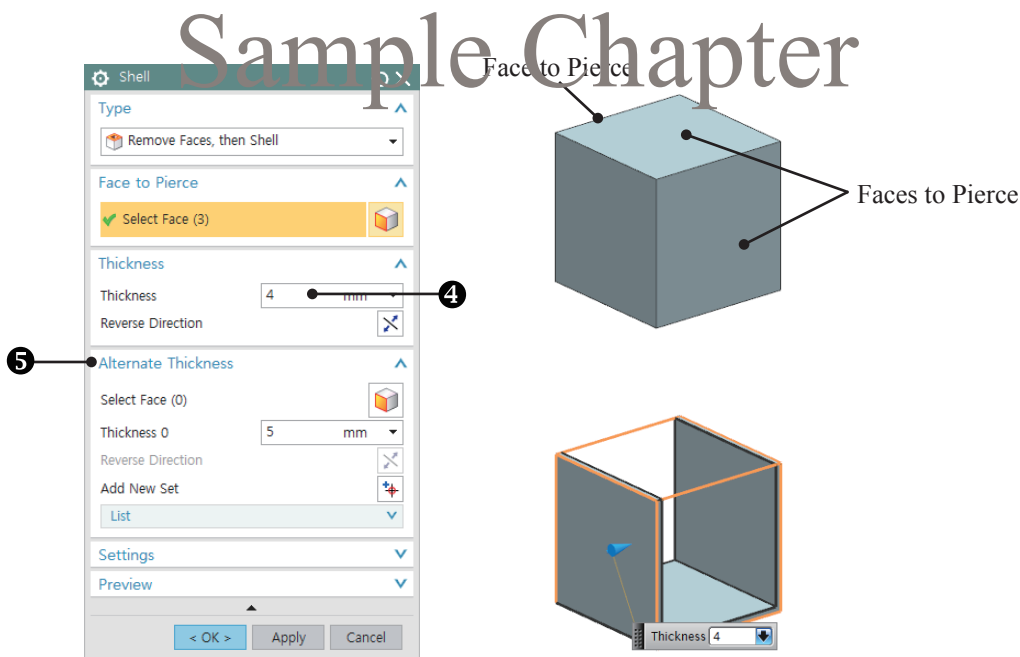


Fig 6-39 Procedure to Apply Alternate Thickness (1/2)

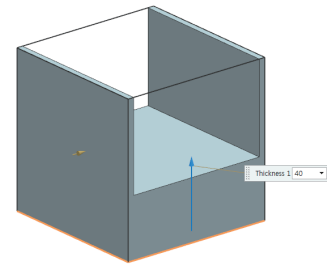
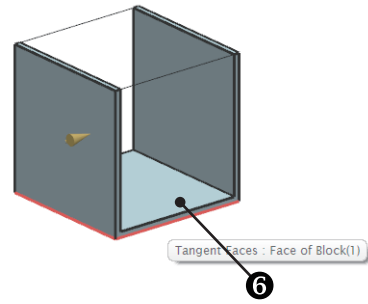
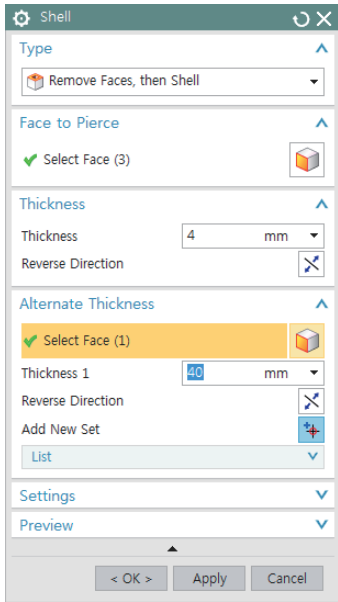


Fig 6-40 Procedure to Apply Alternate Thickness (2/2)

Note that you can apply the **Shell** command for the solid body even though it has already been applied by the **Shell** command. You can create the model shown in Fig 6-115 by applying additional **Shell** command to the resultant solid body shown in Fig 6-114.



Fig 6-41 Applying Shell Twice

If the shell thickness is small enough with regard to the existing body, you can apply the **Shell** command to create the model as shown in Fig 6-116.

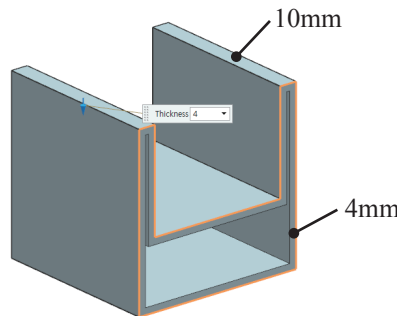


Fig 6-42 Applying Thickness

Exercise 13 Applying Fillet and Shell

ch06_ex13.prt

Apply a fillet and shell to the given model ch06_ex13.prt according to the following procedure.

- ① Apply R5 fillet to the specified fillet.
- ② Apply 3mm thickness uniformly by removing the bottom face.
- ③ Apply R3 Face Blend to the bottom edge.

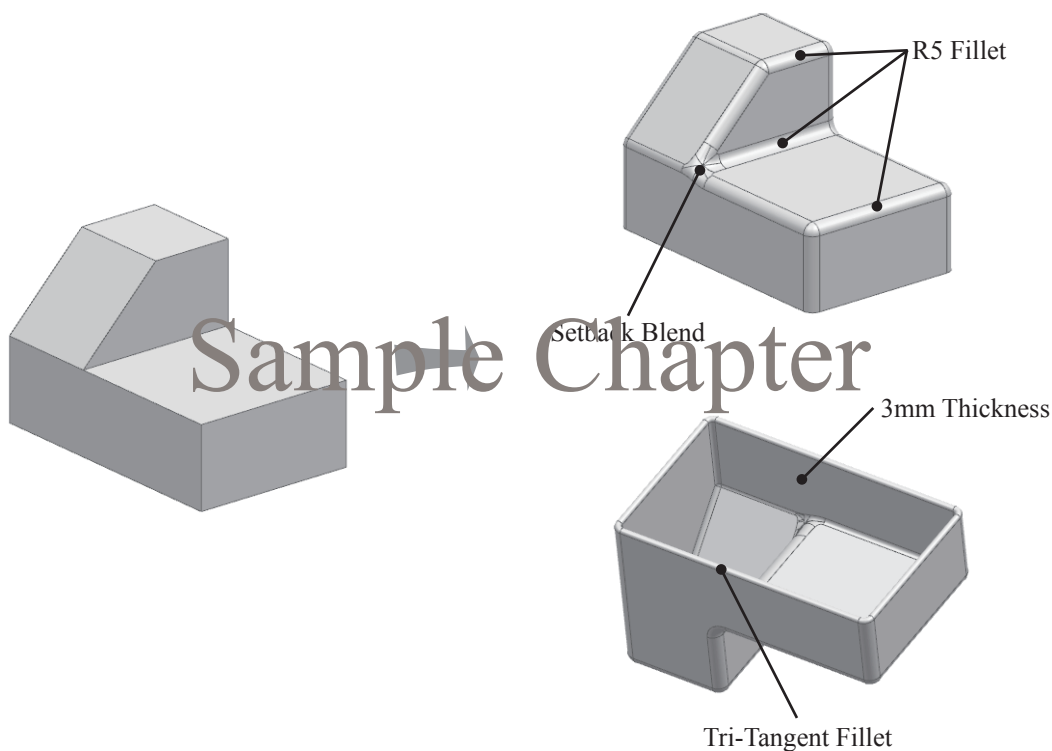


Fig 6-43 Applying Fillet and Shell



Tri-Tangent Fillet

Select **Insert > Detail Feature > Face Blend** in the **Menu** button and select **Three Defining Face Chains** as the type to apply the tri-tangent fillet.

END of Exercise

Apply draft, fillet and shell to the given model ch06_ex14.prt according to the following procedure.

- ① Apply 3° draft to all faces that are parallel to the draw direction which should be ZC axis of the WCS. The bottom plane and the face **A** should not be changed.
- ② Apply fillet for all edges except the edges on the bottom face.
- ③ Apply 2mm thickness shell by piercing the bottom face.

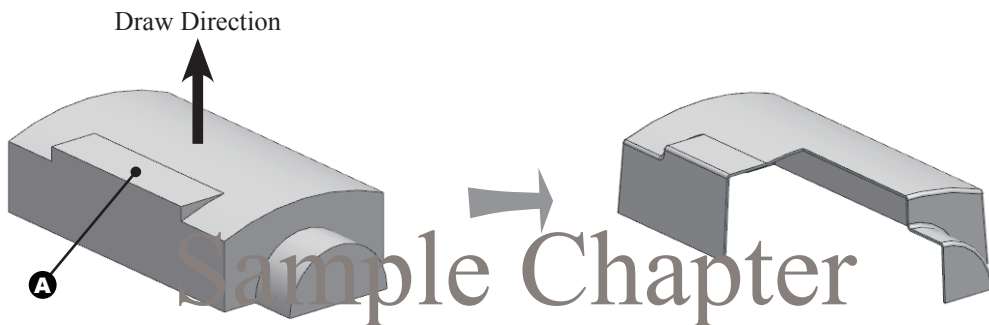


Fig 6-44 Applying Draft, Fillet and Shell

! *Hint!*

Types of draft to be applied are **From Plane** and **Tangent to Faces**.

END of Exercise