CATIA V5 Training
Foils

Aerospace Sheet Metal Design

Version 5 Release 19
January 2009
EDU_CAT_EN_ASL_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- **Manage Sheetmetal parameters**
- **Create and modify the design of a Hydro-formed Sheetmetal Part**
- **Generate and draw a flattened part**
- **Create a Knowledge Expert Check using characteristic curves**

Targeted audience

Aerospace Designers

Prerequisites

Students attending this course should have knowledge of Part Design, Assembly Design, and Wireframe & Surface Design

8 hrs

Instructor Notes:

Table of Contents (1/5)

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Instructor Notes:

Table of Contents (5/5)

Instructor Notes:

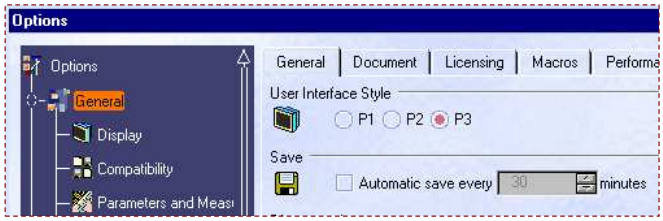
Introduction to Aerospace Sheet Metal Design

In this lesson you will learn about main features of Aerospace Sheet Metal Design workbench

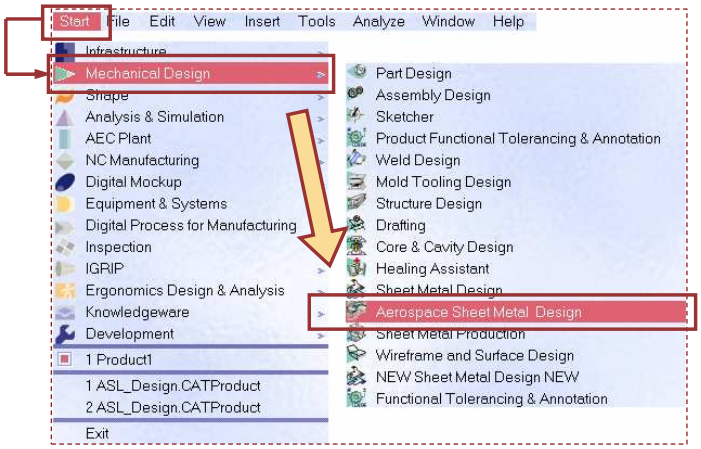
Instructor Notes:

ASL Workbench Presentation (1/4)

- Modifying CATIA environment
  - Tools > Option > General
    - General
    - User Interface Style: CATIA P3



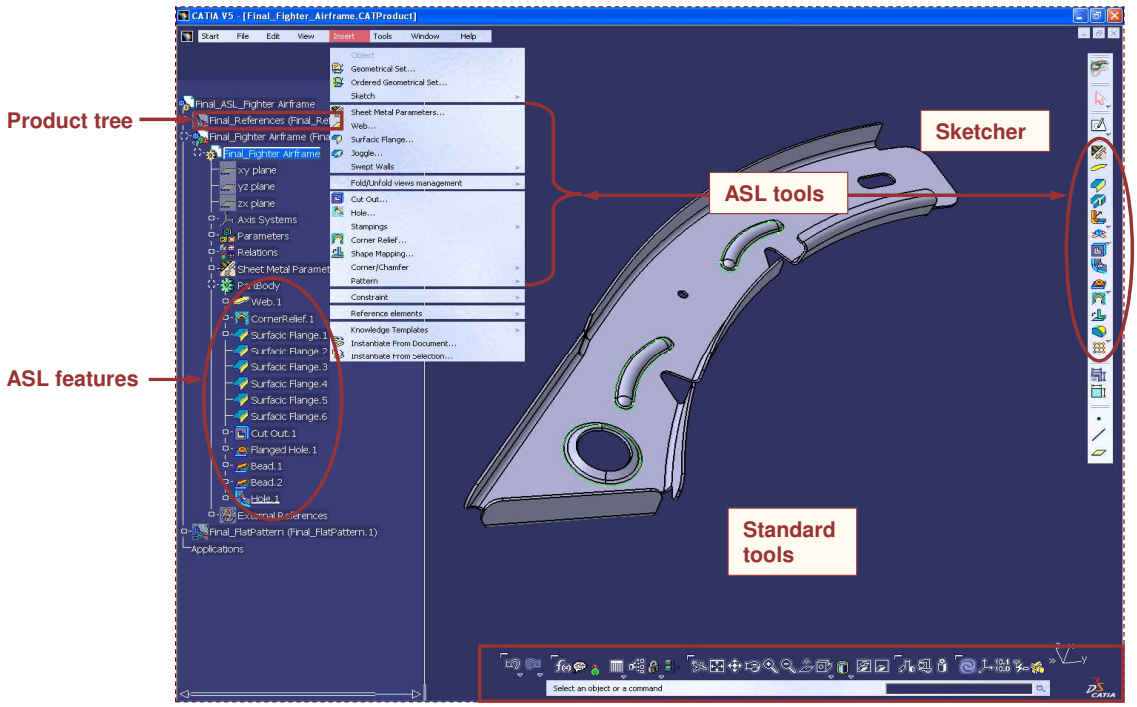
- Accessing the Workbench



Instructor Notes:

ASL Workbench Presentation (2/4)

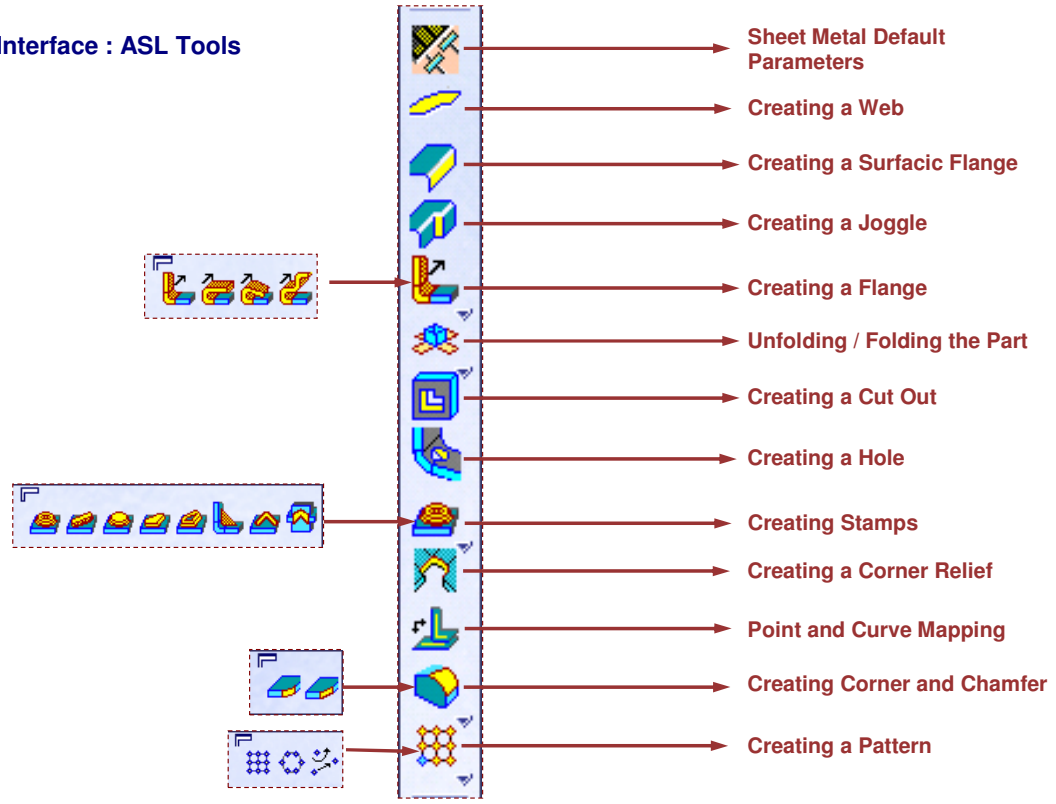
User Interface : General Presentation



Instructor Notes:

ASL Workbench Presentation (3/4)

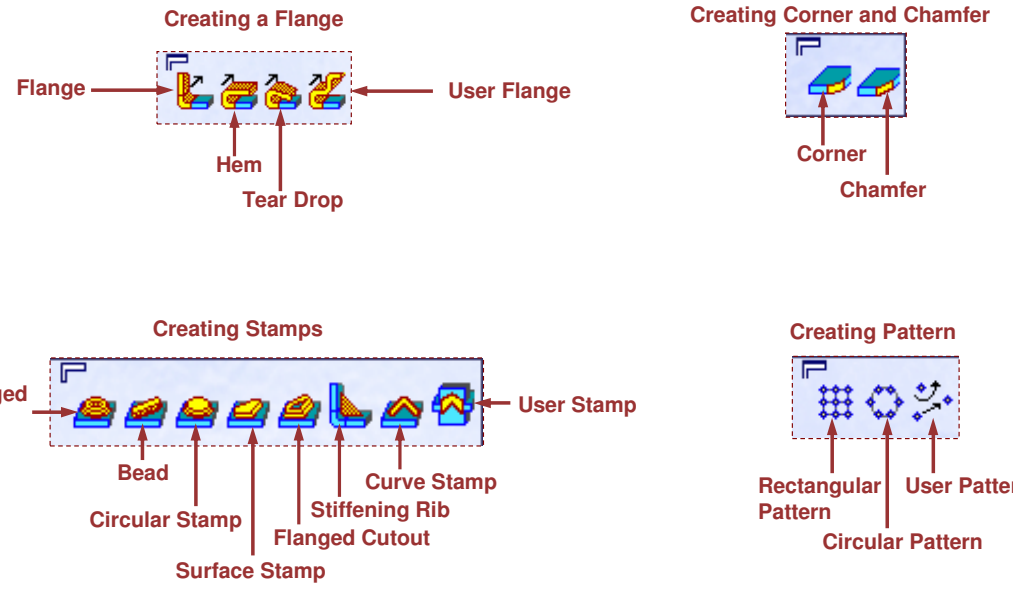
User Interface : ASL Tools



Instructor Notes:

ASL Workbench Presentation (4/4)

User Interface : ASL Tools



Instructor Notes:

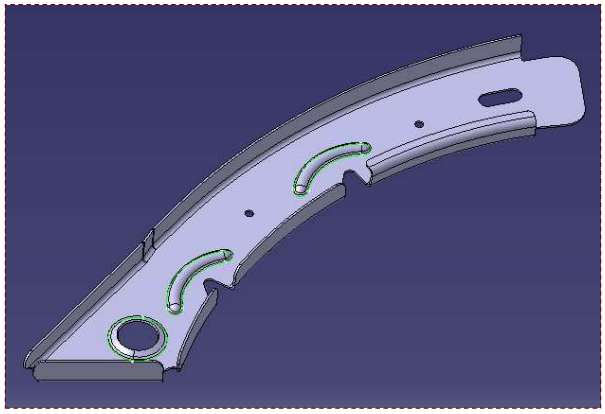
Master Exercise: Fighter Airframe

Exercise Presentation

50 min

In this step you will see :

- **Design Intent: Fighter Airframe**
- **Design Process: Fighter Airframe**



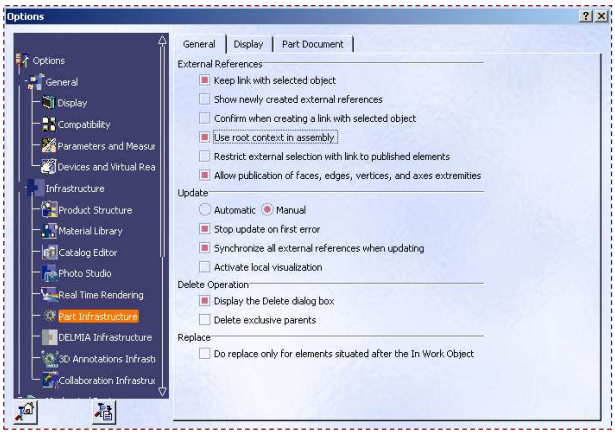
Instructor Notes:

Master Exercise: Fighter Airframe

Step 1: Setting an Environment

5 min

In this step we will set the CATIA environment



Instructor Notes:

ASL Parameters

In this lesson you will learn the setting parameters of an Aerospace Sheet Metal part

SheetMetalStandard	Thickness (mm)	RadiusTable	ExtrudedHoleStd	BeadStd	IndexHoleStd	ManufacturingHoleStd	ClearanceHoleStd	FastenerHoleStd	CircularStampStd	JoggleCompensationMethod1	JoggleCompensationMethod2	JoggleCompensationOnFlange
Aluminum	1.2	RadAl12.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls	Jog_Comp_Method1.xls	Jog_Comp_Method2.xls	Flg\Method2Thk1.xls
Aluminum	1.6	RadAl16.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk16.xls
Aluminum	2	RadAl20.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk2.xls
Aluminum	2.5	RadAl25.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk25.xls

Instructor Notes:

Loading Data (1/2)

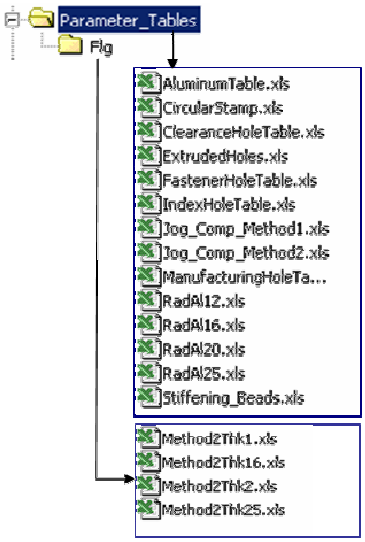
- You can link parameters table files such as Extruded Hole Std or Bead Std for example as well as Joggle Compensation Methods to the Sheet Standard . xls file, just by indicating their name in the corresponding column.

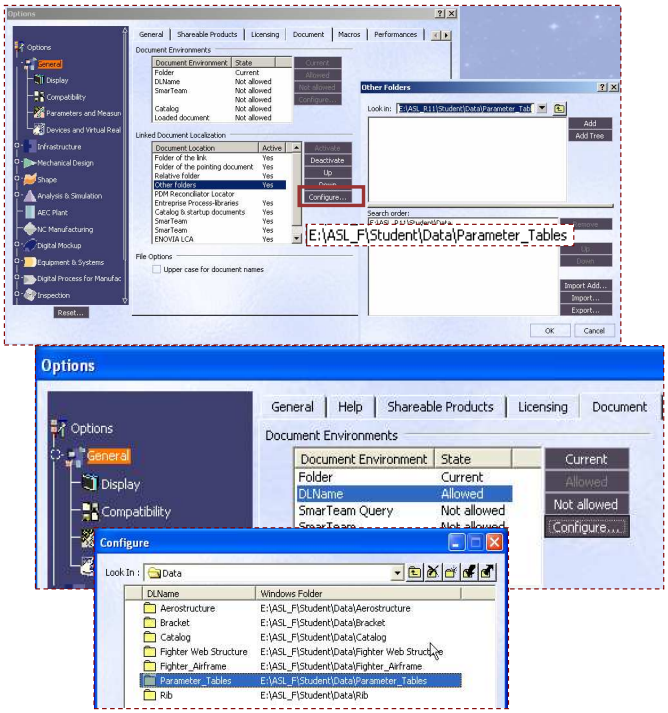
SheetMetalStandard	Thickness (mm)	RadiusTable	ExtrudedHoleStd	BeadStd	IndexHoleStd	ManufacturingHoleStd	ClearanceHoleStd	FastenerHoleStd	CircularStampStd	JoggleCompensationMethod1	JoggleCompensationMethod2	JoggleCompensationOnFlange
Aluminum	1.2	RadAl12.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls	Jog_Comp_Method1.xls	Jog_Comp_Method2.xls	Flg\Method2Thk1.xls
Aluminum	1.6	RadAl16.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk16.xls
Aluminum	2	RadAl20.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk2.xls
Aluminum	2.5	RadAl25.xls	ExtrudedHoles.xls	Stiffening_Beads.xls	IndexHoleTable.xls	ManufacturingHoleTable.xls	ClearanceHoleTable.xls	FastenerHoleTable.xls	CircularStamp.xls			Flg\Method2Thk25.xls

Instructor Notes:

Loading Data (2/2)

- **Prior to utilizing such design table you should declare correct path where parameters tables are stored**
  Tools > Options > General > Document
  Other Folders / Configure ,or
  DL Name Allowed and Configure





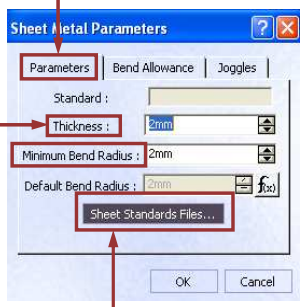
Instructor Notes:

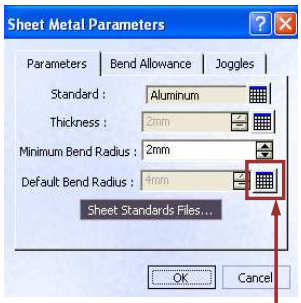
Modifying Default Thickness and Bend Radius

Click the Sheet Metal Parameters icon

Before starting to generate the part it is necessary to set default parameters such as Thickness and Bend Radius

Select the Parameters Tab





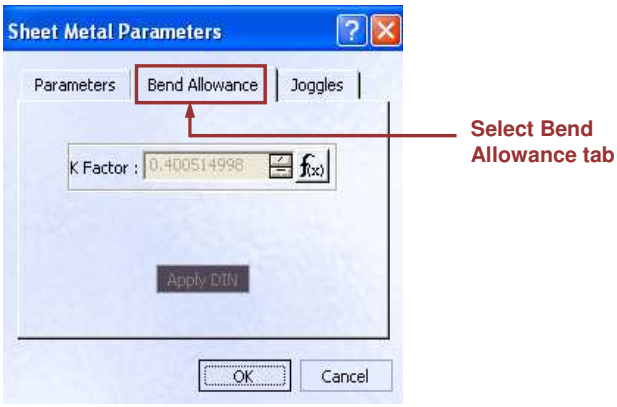
Set Thickness and Bend Radius entering directly the values

Another way to set parameters is to open a Sheet Standards Files and accessing to the design table by the corresponding button

Instructor Notes:

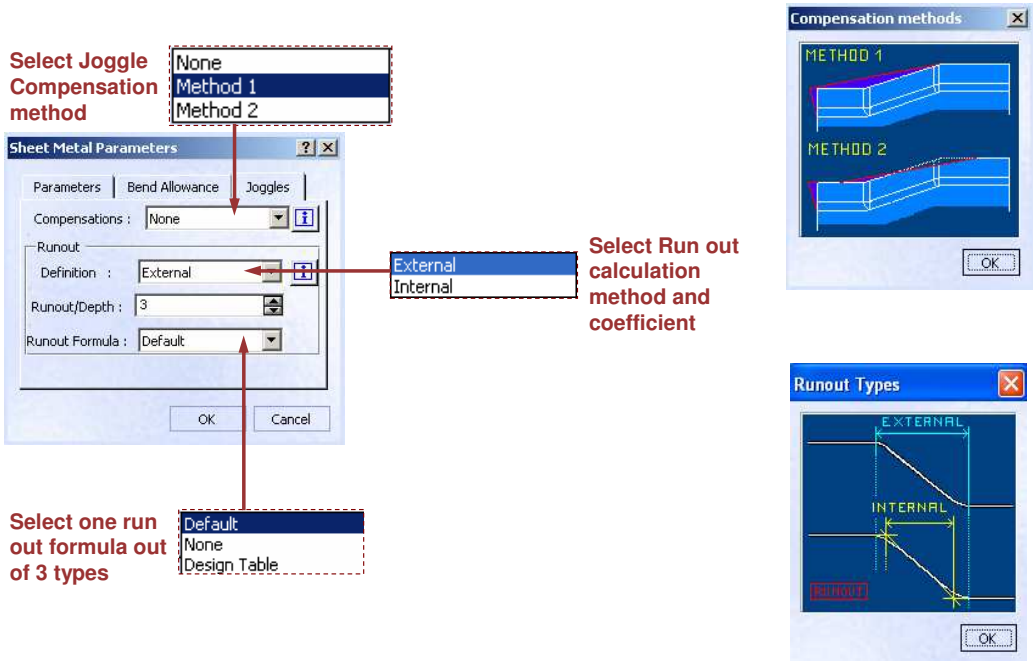
Modifying Default Bend Allowance

- A formula is associated to the K factor and could be deactivated (Right Button Formula Deactivate)



Instructor Notes:

Choosing Joggle Compensations



Instructor Notes:

Runout Formulas (1/2)

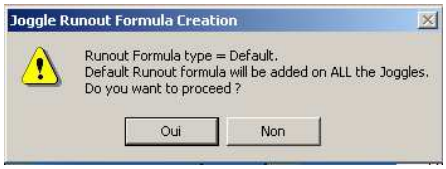
Default Formula :

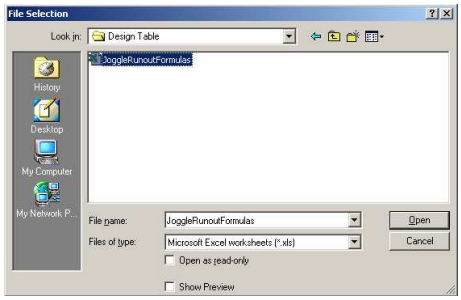
Runout = Coeff * Depth

No Formula : erases formulas on ALL part joggles

Design Table Formula :

choose a path to an .xls or .txt file





Instructor Notes:

Runout Formulas (2/2)

	A	B	C	D	E
1	JoggleRunoutFormulaName	JoggleRunout_a1	JoggleRunout_a2	JoggleRunout_a3 (mm)	JoggleRunoutFormulaBody
2	Formula 1	1.5	2.5	0.5	a1*DEPTH+a2*THICKNESS+a3
3	Formula 2	2	0.5	0	a1*DEPTH+a2*(STARTRADIUS+ENDRADIUS)
4	Formula 3	3	0.5	0	a1*THICKNESS+a2*DEFAULTBENDRADIUS
5	Formula 4	0.5	0	4.5	a1*DEPTH+a3

- There is a particular syntax to use in order to create the design table.
- As you can see on the picture, it can be described as follows :
  - The A Column (Formula Names) contains String Parameters
  - The B and C Columns contain Real Parameters (no unit)
  - The D Column contains Length Parameters (unit)
  - The E Column (Formula Bodies) uses previous Real & Length parameters (a1, a2, a3) for its own

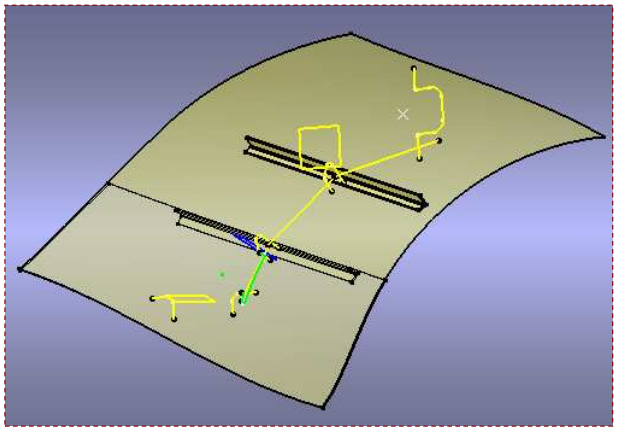
  formulas (as you can see, not all existing parameters must be used in the formula)

Instructor Notes:

Master Exercise: Fighter Airframe

Step 2: Setting default part parameters

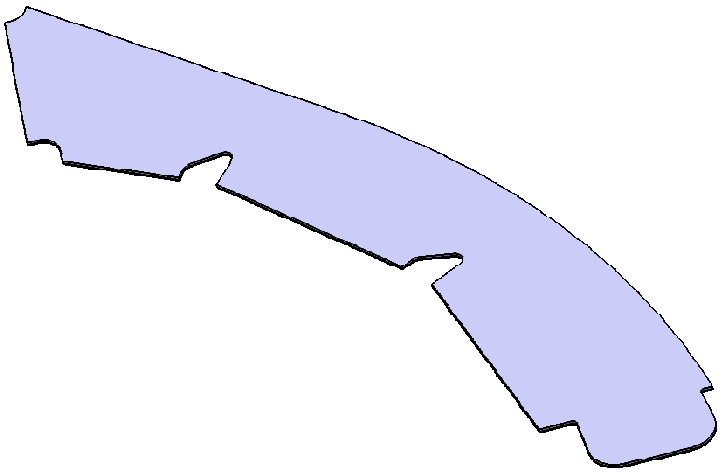
5 min



Instructor Notes:

Creating the Web

In this lesson you will learn creating the web.



Instructor Notes:

Creating the Web (1/2)

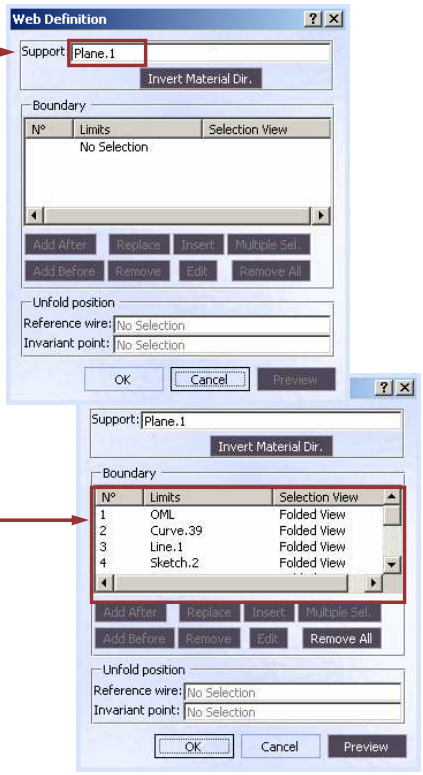
The web is the fix feature when we unfold the part

A. Click the Web icon 

B. Select a plane or a close sketch which support the web

C. Select curves, planes or surfaces to define the limits of the web.

The limiting elements must intersect.



Instructor Notes:

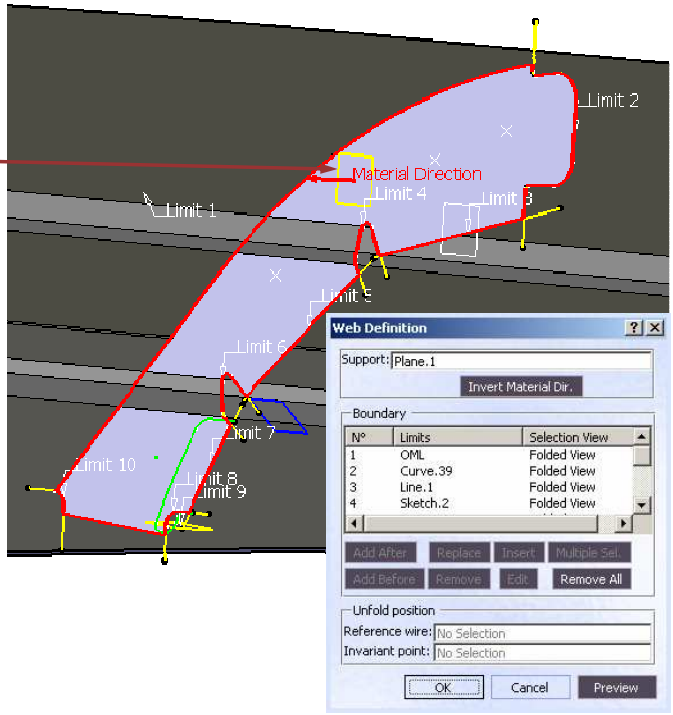
Creating the Web (2/2)

D. As soon as selected elements make a closed area, a preview of web is suggested

The contour must be selected in the logic sequence.

If necessary Select Web Material Direction.

Reverse by clicking on the red arrow which is perpendicular to the web support.



Instructor Notes:

Creating a Surfacic Web (1/2)

A. Click the Web icon 

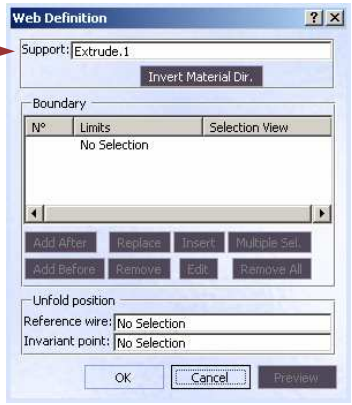
B. Select a surface that can be developed, even a curved one, which supports the web

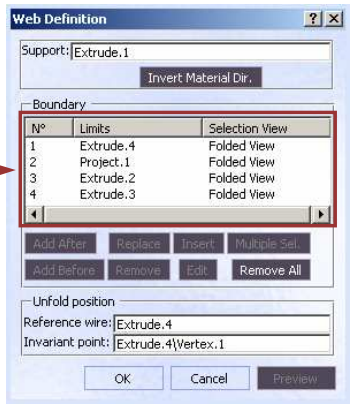
C. Select a reference wire and an invariant point as references for the unfolding of the web.

The reference wire must be located on one of the limits and the invariant point on the reference wire.

Select curves, planes or surfaces to define the limits of the web.
The limiting elements must intersect.

As soon as selected elements make a closed area, a preview of web is suggested
The contour must be selected in the logic sequence.



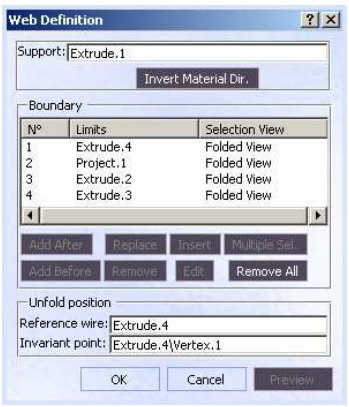


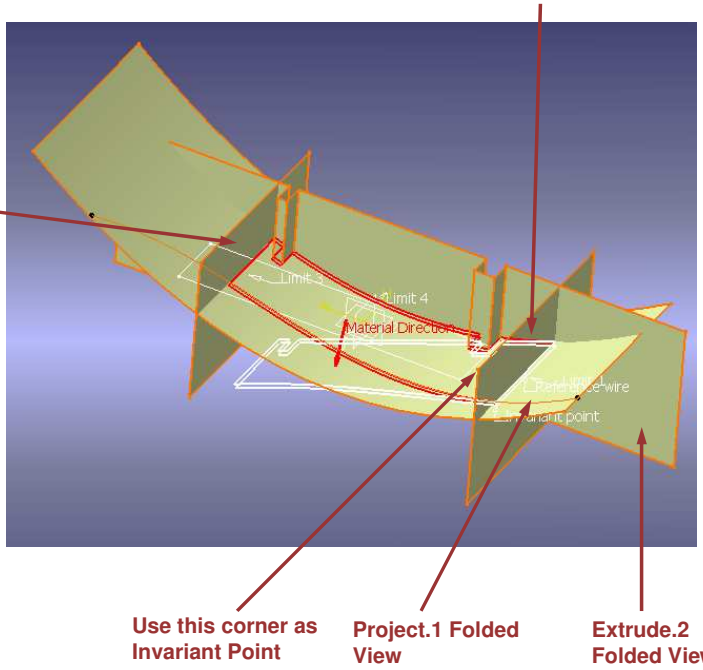
Instructor Notes:

Creating a Surfacic Web (2/2)

D. If necessary Select Web Material Direction.

Reverse by clicking on the red arrow which is perpendicular to the web support



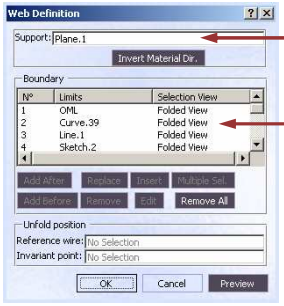


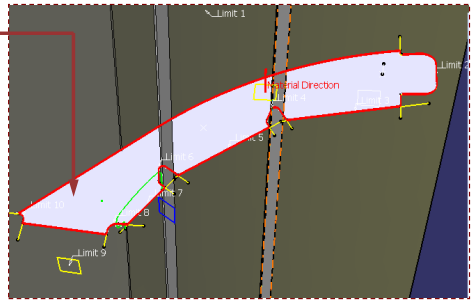
Instructor Notes:

Modifying the Web

Modify the web by accessing the Web Definition dialog box

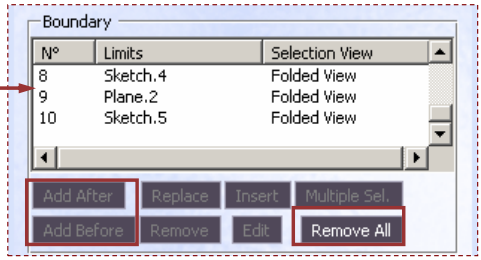
1. Double-click the Web feature in the tree
2. Change Support Geometry or Boundary Elements





3. Select a Boundary Element it becomes then a Reference Element and you can then:
   - Add After
   - Add Before
   - Replace
   - Remove
   - Insert

   You can also Remove All elements



Multiple Sel. option is available when Add After or Add Before are activated

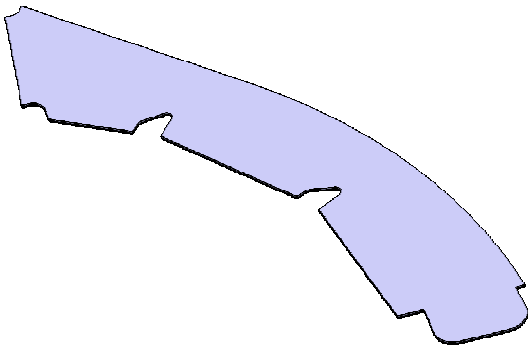
Instructor Notes:

Master Exercise: Fighter Airframe

Step 3: Creating the Web

2 min

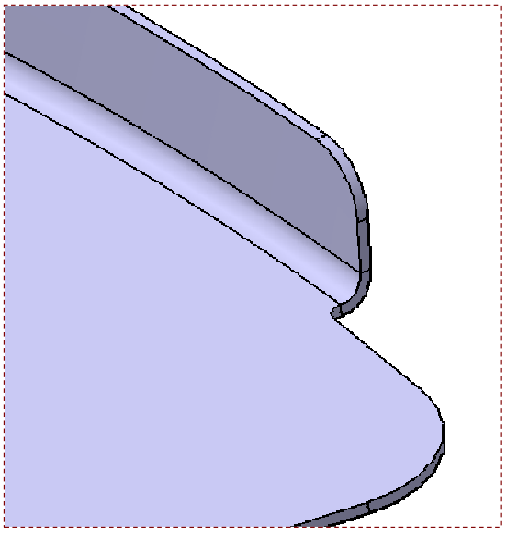
In this step we create the airframe web



Instructor Notes:

Creating a Surfacic Flange

In this lesson you will learn to create a Surfacic Flange.



Instructor Notes:

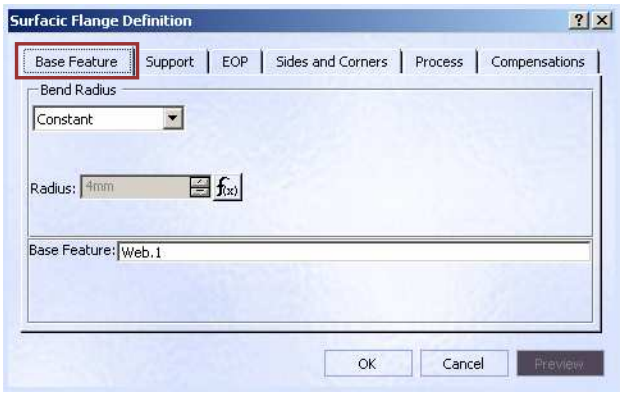
Creating a Surfacic Flange (1/7)

Selecting the Base Feature

A. Click the Surfacic Flange icon 

B. Click the Base Feature tab

C. Select the web or a flange as Base Feature



Instructor Notes:

Creating a Surfacic Flange (2/7)

Selecting the Support : Support Type

Support type :

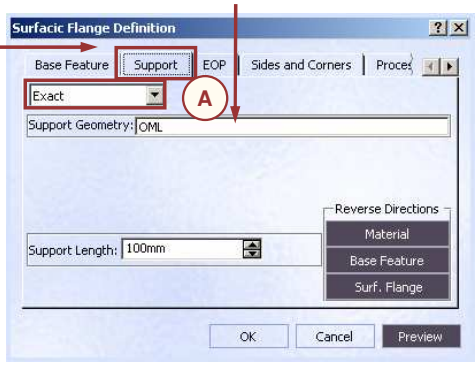
A. **Exact : the support is the surface geometry selected**

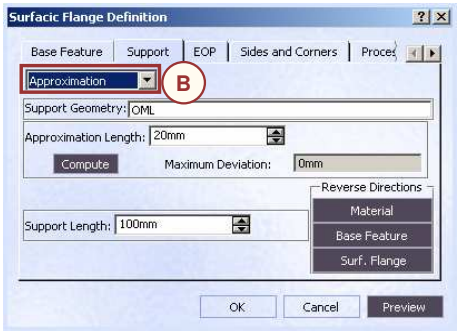
B. **Approximation : the selected surface is approximated into a ruled surface and the maximum deviation can be computed according to an approximation length**

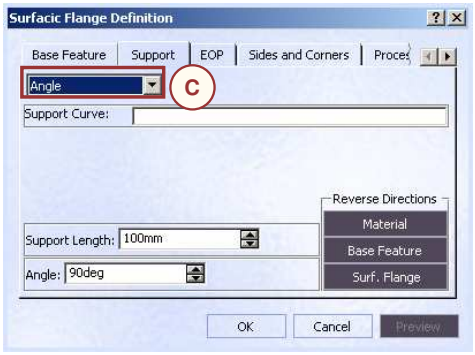
C. **Angle : the surface of the flange is defined by a curve an angle and a support length**

Click the Support tab

Select the support type







Instructor Notes:

Creating a Surfacic Flange (3/7)

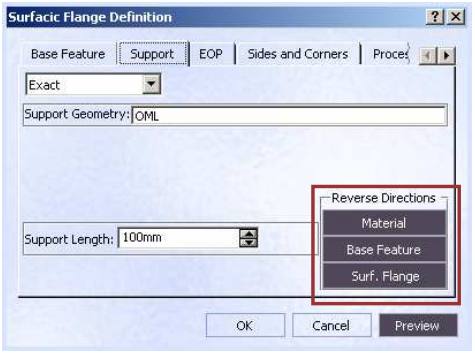
Selecting the Support : Flange Directions

Select Directions (Reverse by clicking on the corresponding red arrows) :

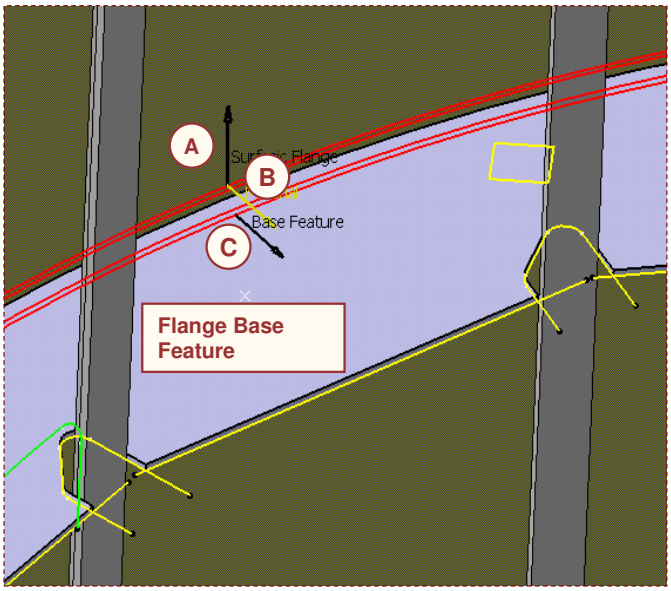
A. **Flange Direction**

B. **Flange Material direction**

C. **Flange Base Feature Direction**



Flange Support



Instructor Notes:

Creating a Surfacic Flange (4/7)

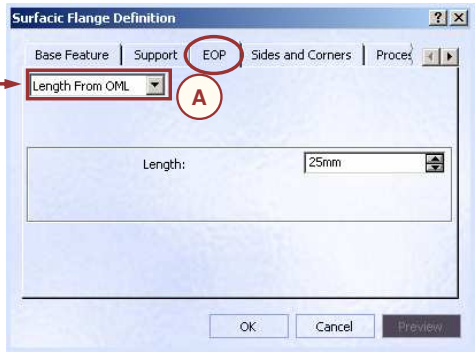
Define the Edge Of Part (EOP)

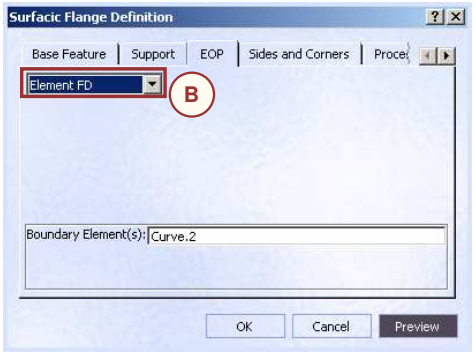
A. **Length From OML : length between the curve defining the top of the flange and the Outer Mold Line (OML)**

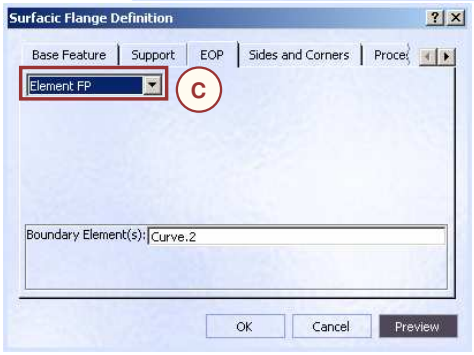
B. **Element FD (Folded) : Boundary element is a surface or a plane which intersects the flange surface or a wire projected on the flange surface**

C. **Element FP (Flattened) : Boundary element is a surface or a plane which intersects the flattened flange surface or a wire projected on the flattened flange surface**

Select the type flange length



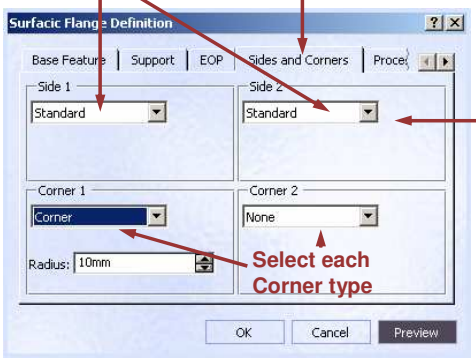




Instructor Notes:

Creating a Surfacic Flange (5/7)

Defining Sides and Corners :



SIDE TYPES

Standard: they are automatically defined at the web limit and the perpendicular plans are kept (the user doesn't have to define them)

None: no side computed (only the EOP is able to define them)

Element FD: they can be defined by a geometrical element FD (Curve or surface)

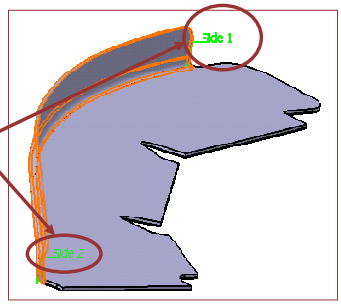
Element FP: they can be defined by a geometrical element FP (Curve or surface)

CORNER TYPES (angle defined between the EOP and the sides)

Corner: between the side and the EOP (defined with a radius value)

None: No corner computed (only the EOP is able to define the contour of the flange)

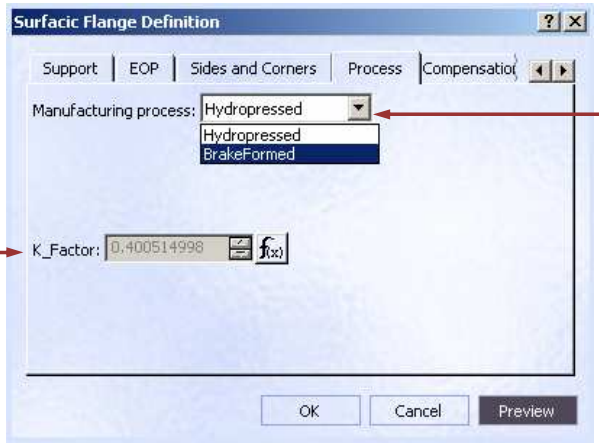
Two texts indicate Flange Sides (Side & corner 1 Side & Corner 2)



Instructor Notes:

Creating a Surfacic Flange (6/7)

Process tab: Manufacturing Process and specific K_ Factor



Manufacturing process

Hydro pressed or Break Formed

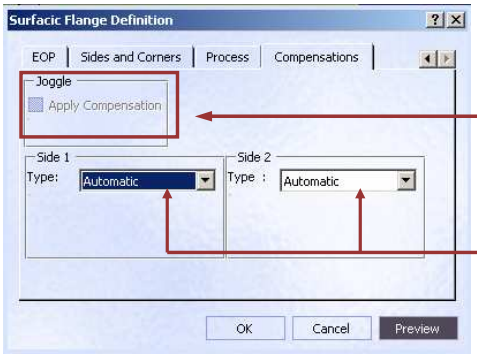
(information only for the current CATIA release)

General K_Factor can be locally altered for a specific Surfacic Flange

Instructor Notes:

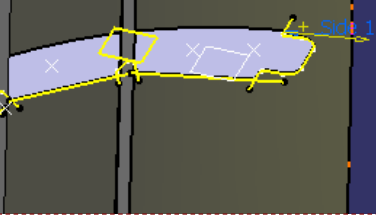
Creating a Surfacic Flange (7/7)

- **Compensation tab: Joggle and Flange Sides Compensations**



Apply or not Joggle Compensation method selected in Sheetmetal Parameters definition

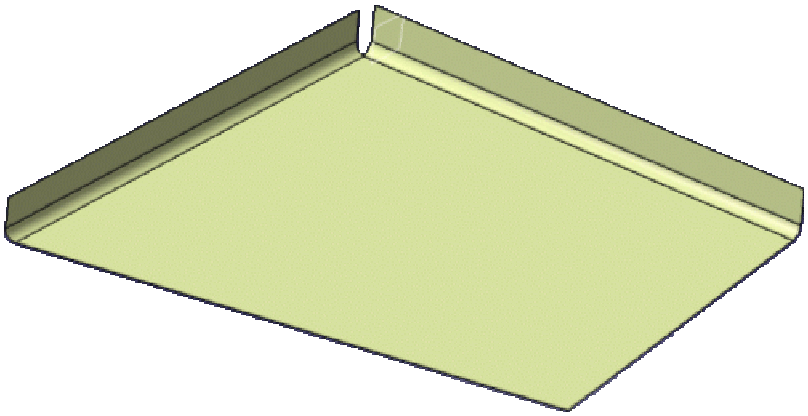
Choose Flange Sides Compensation method



Instructor Notes:

Intersecting Surfacic Flanges on a Web

- It is possible to intersect two surfacic flanges on a web. This means that you can choose as a support a web with an existing surfacic flange that will go through the new surfacic flange.
- The intersecting flanges are automatically detected, and the geometry of the first flange is relimited to enable the creation of the second flange; the unfolded view is computed accordingly. You can then remove the sharp vertex in the corner by creating a corner relief or a cutout at the intersection of the surfacic flanges.

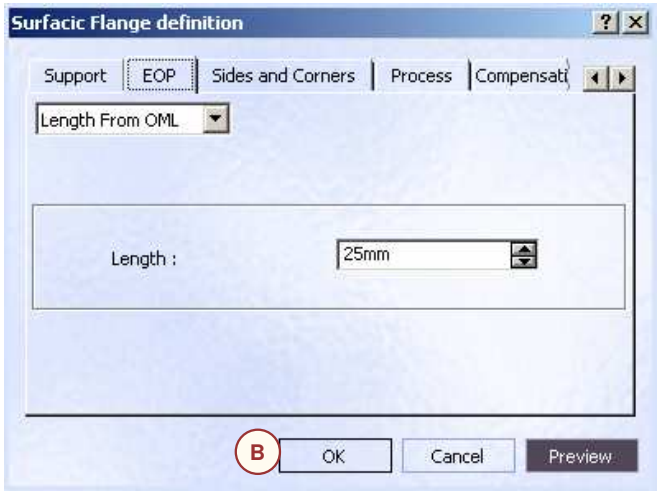


Instructor Notes:

Modifying a Surfacic Flange

A. Double Click the Flange in the part or in the tree and the Flange Definition dialog box is displayed

B. Modify desired parameters and click OK to validate



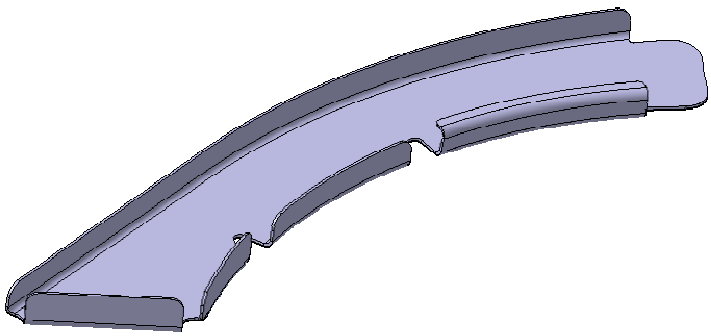
Instructor Notes:

Master Exercise: Fighter Airframe

Step 4: Creating Surfacic Flanges

10 min

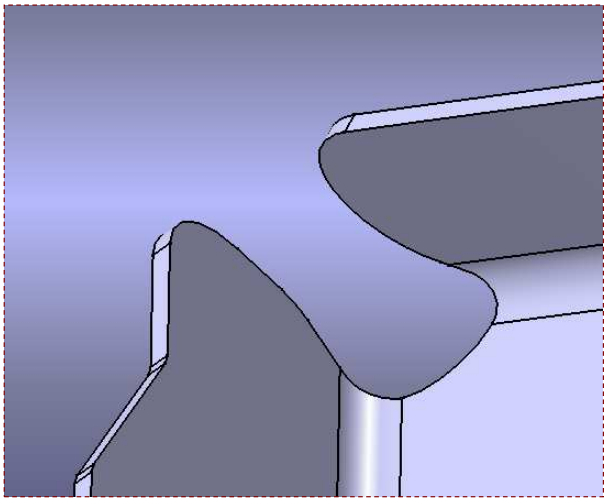
In this step we create 6 flanges using different definition types



Instructor Notes:

Creating a Corner Relief

In this lesson you will learn to create a Corner Relief



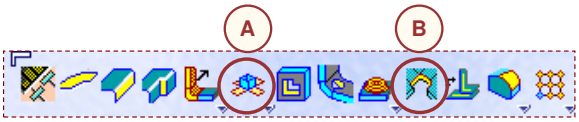
Instructor Notes:

Creating a Corner Relief (1/2)

- Setting corner relief parameters
  - Part must be flattened

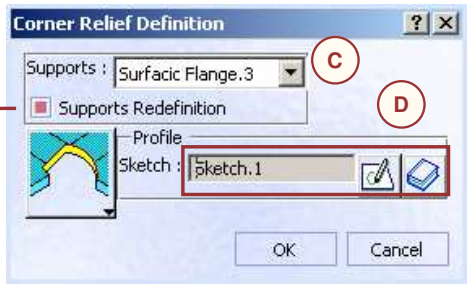
A. Click the Fold/Unfold icon to flatten the part

B. Click the Corner Relief icon



C. Select both involved flanges

D. Select the sketch of the corner relief profile or access to standard contour catalog or enter the Sketcher workbench to create the profile



Support Redefinition option is used when user wants flanges sides to be impacted (lengthened) by corner relief profile

Instructor Notes:

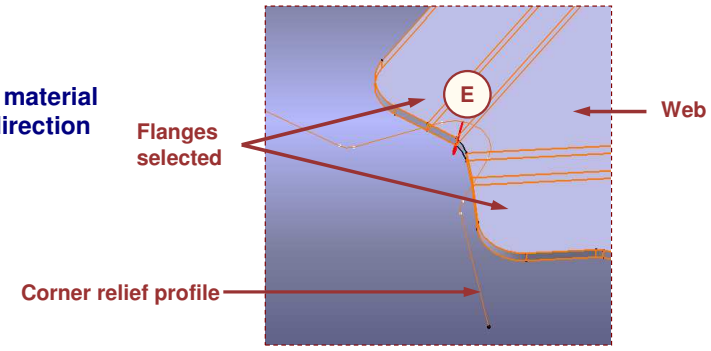
Creating a Corner Relief (2/2)

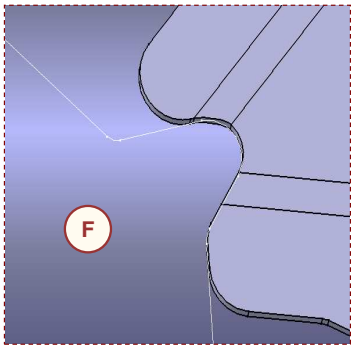
Setting corner relief parameters

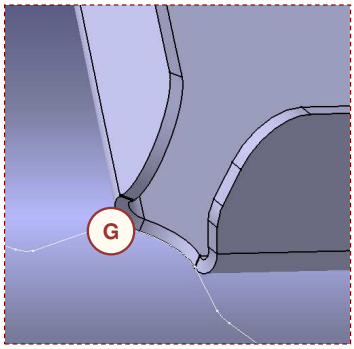
E. The red arrow determines the material side to be removed. Change direction if necessary

F. Update the part if necessary

G. Fold back the part





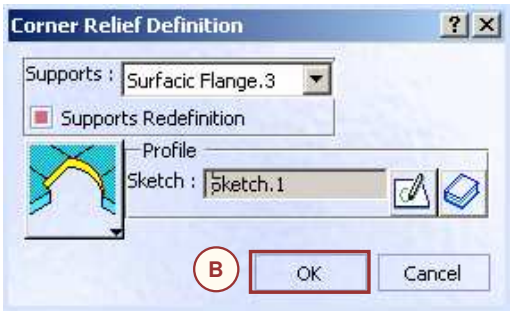


Instructor Notes:

Modifying a Corner Relief

A. Double Click the Corner Relief in the tree when the part is flattened and the Corner Relief Definition dialog box is displayed

B. Modify desired parameters and click OK to validate

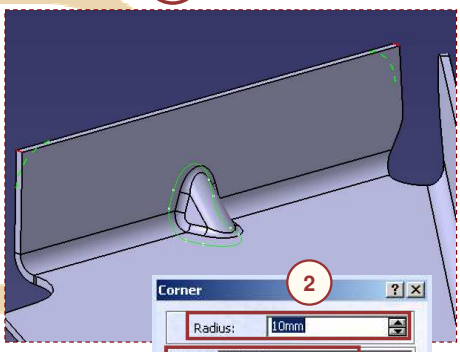


Instructor Notes:

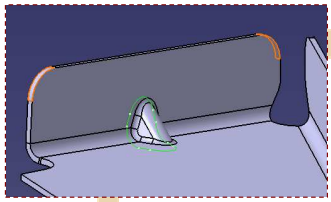
Create Corner or Chamfer on Sharp Edges

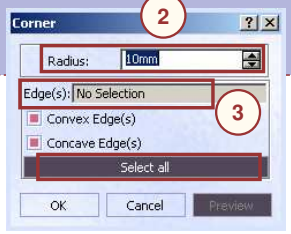
1 Select Corner or Chamfer icon

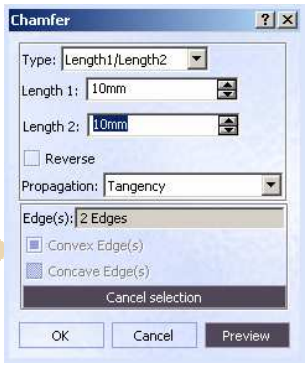
2 Key Radius value



3 Select Edges to be worked or Select all







In case of chamfer definition you will first have to choose the type Length1/Length2 or Length1/Angle

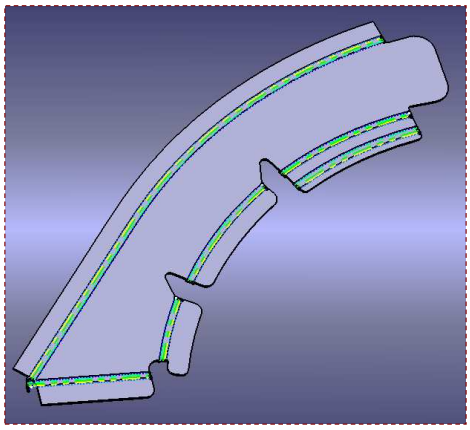
Instructor Notes:

Master Exercise: Fighter Airframe

Step 5: Creating a Corner Relief

10 min

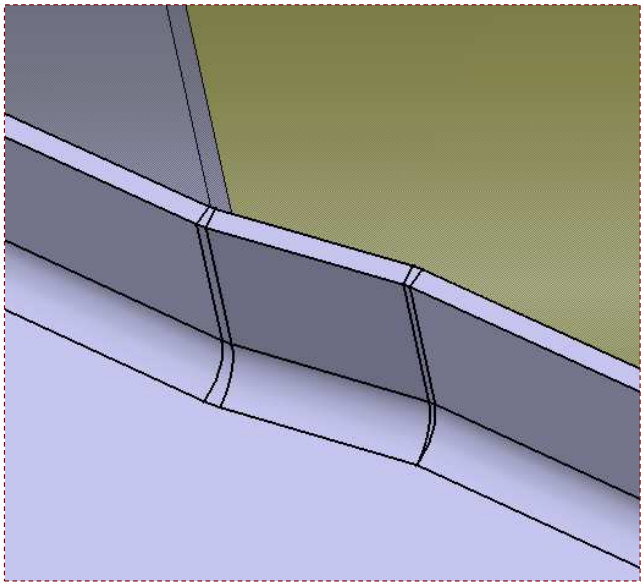
In this step we will create a corner relief



Instructor Notes:

Creating a Joggle

In this lesson you will learn about creating a Joggle



Instructor Notes:

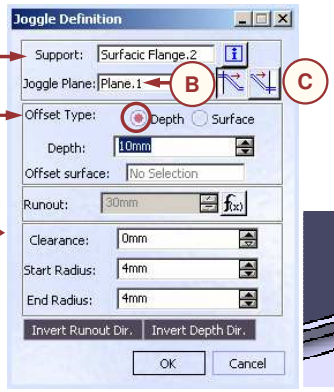
Creating a Joggle by Depth (1/3)

Using a flange

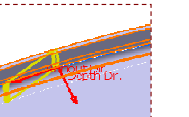
The vector normal to the joggle plane determines the side on which the joggle is to be created (run out)

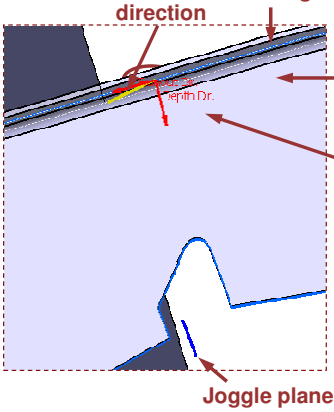
A. Select the flange as joggle support
B. Select the joggle plane
C. Select the role of the plane
D. Select Depth Type

Click the Joggle icon

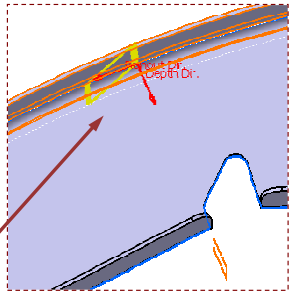


Result





The vector normal to the flange support determines the depth direction



The selected plane either start the joggle or end it.

Instructor Notes:

Creating a Joggle by Depth (2/3)

Setting joggle parameters

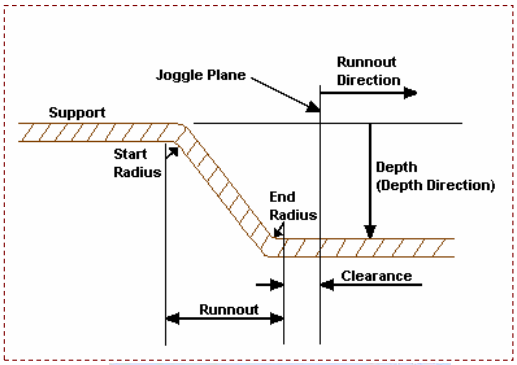
D. Depth : offset at the support surface

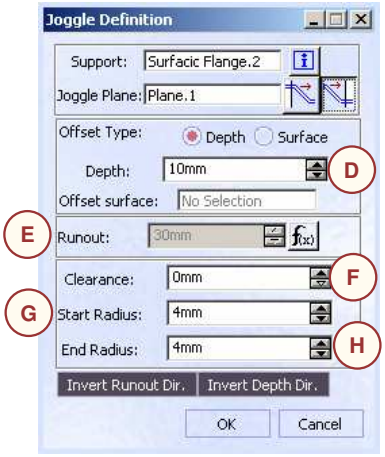
E. Run out : length between the original surface of the flange and the new surface.If desired you can deactivate formula

F. Clearance : offset between the Joggle Plane and the end radius

G. Start Radius : radius linked with the original surface

H. End Radius : radius linked with the new surface





Instructor Notes:

Creating a Joggle by Depth (3/3)

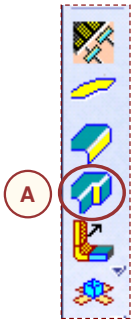
Using a web

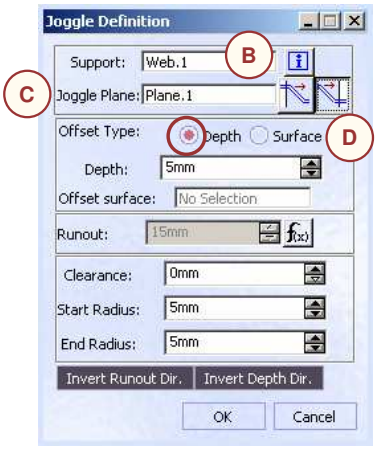
A. **Click the Joggle icon**

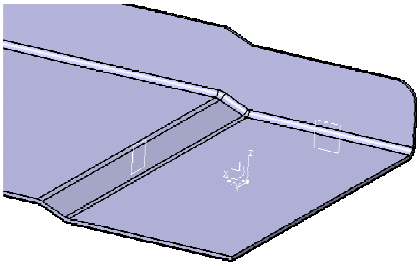
B. **Select the Web as joggle support**

C. **Select the joggle plane and the role of the plane**

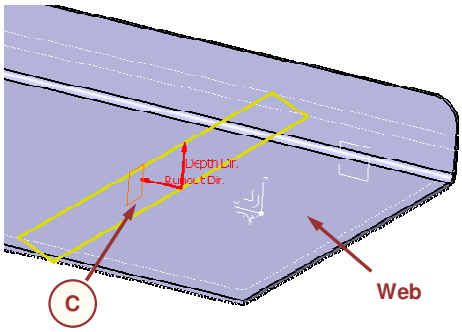
D. **Select Offset Type as Depth**







Result



Instructor Notes:

Creating a Joggle with an Input Surface (1/2)

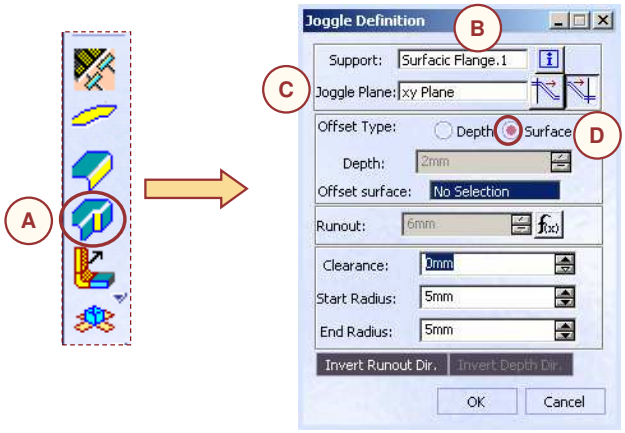
Using an offset surface

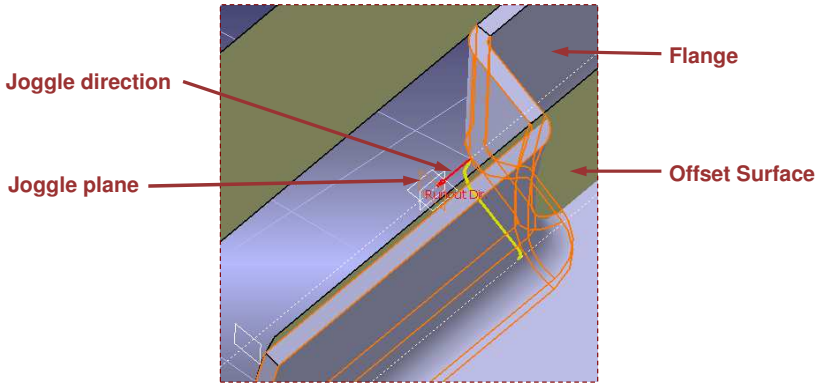
A. Click the Joggle icon

B. Select the flange as joggle support

C. Select the joggle plane and the role of the plane

D. Select Surface Type

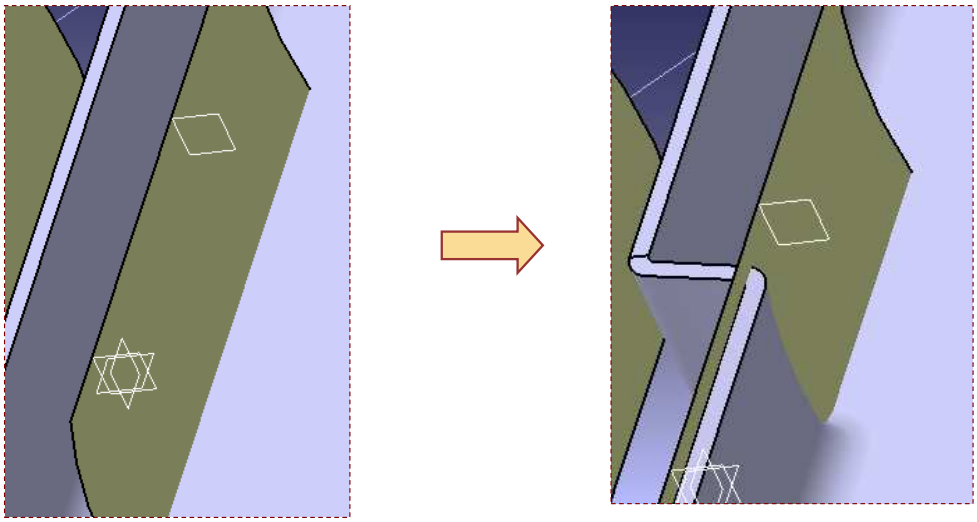




Instructor Notes:

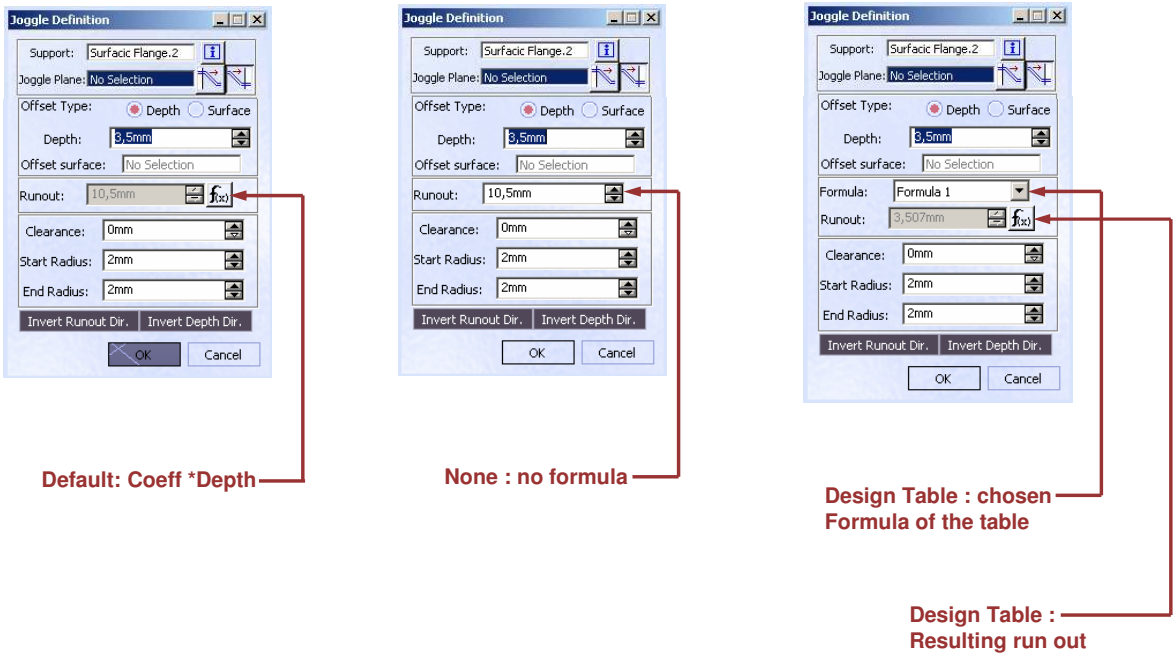
Creating a Joggle with an Input Surface (2/2)

The parameters work the same way as with a depth except for the offset surface which obviously replaces the depth parameter. We get this result at the end :



Instructor Notes:

Joggle Run out Formulas

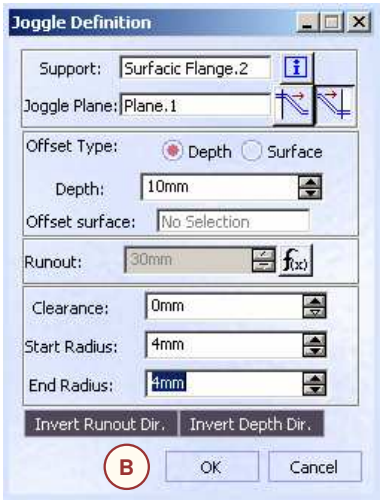


Instructor Notes:

Modifying a Joggle

A. Double Click the Joggle in the tree and the Joggle Definition dialog box is displayed

B. Modify desired parameters and click OK to validate



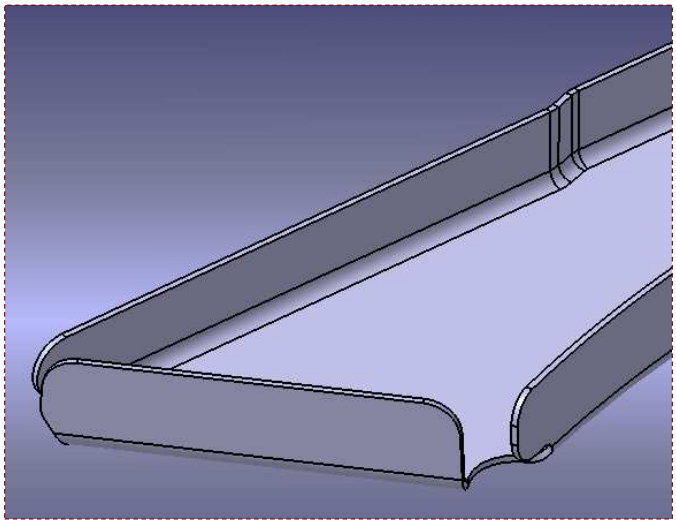
Instructor Notes:

Master Exercise: Fighter Airframe

Step 6: Creating a Joggle

2 min

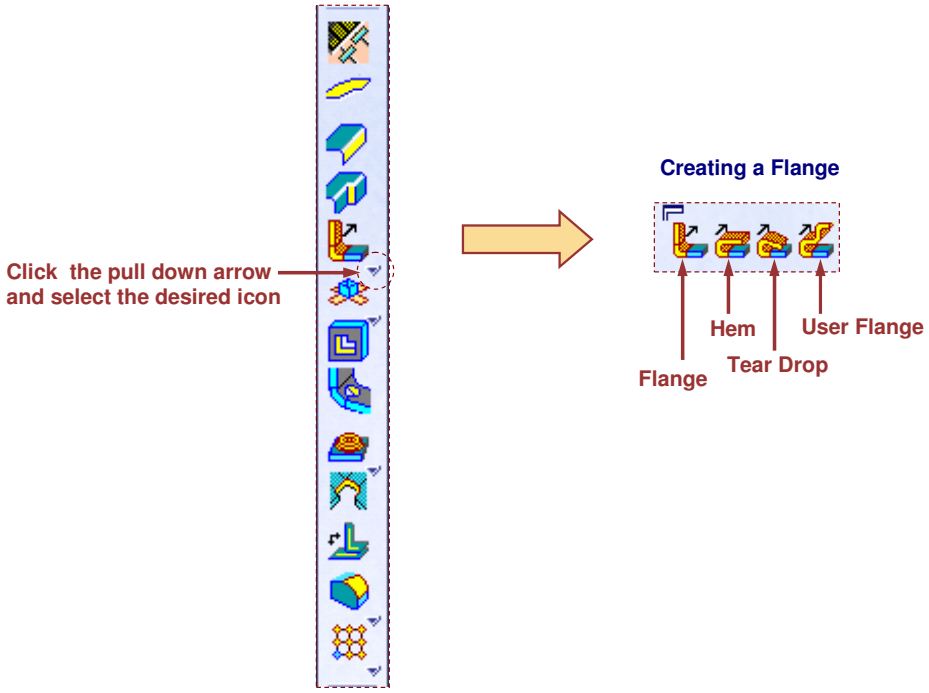
In this step you will create a joggle on a surfacic flange



Instructor Notes:

Creating Flanges other than Surfacic Flanges

In this lesson you will learn to create Flanges using various options

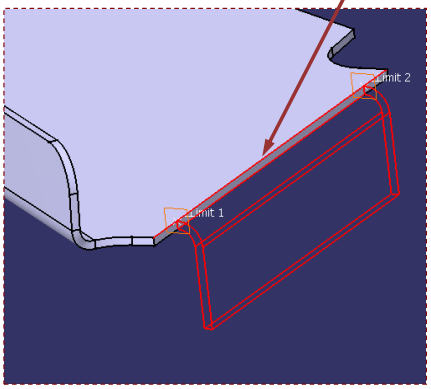


Instructor Notes:

Creating a Flange

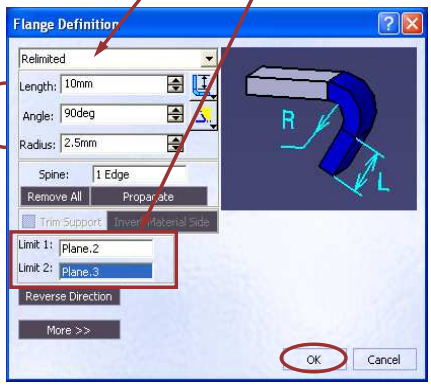
1 Click Flange icon

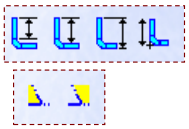
2 Select Spine



3 Choose Basic or Relimited option, If Relimited you will then select two relimiting elements

4





5 Select OK

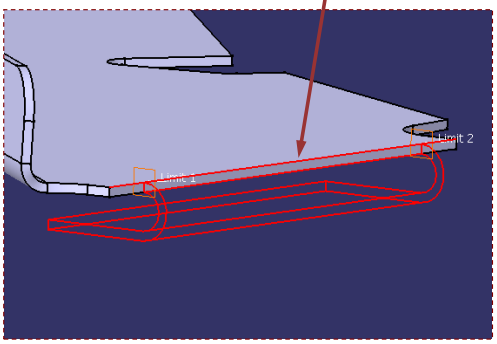
4 Key Flange parameters: Radius, Length, Angle

Instructor Notes:

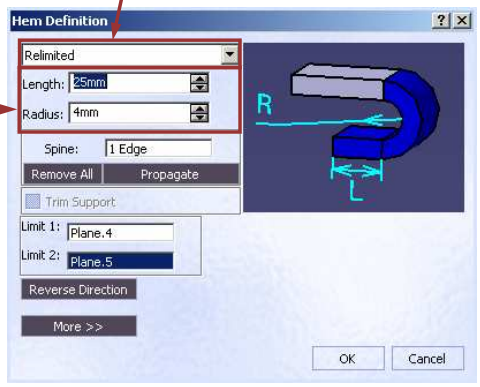
Create a Hem

1 Click Hem icon

2 Select Spine



3 Select Basic or Relimited option, If Relimited you will then select two relimiting elements



4 Key Flange parameters: Radius, Length

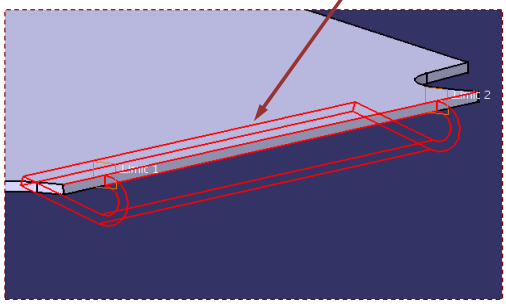
5 Select OK

Instructor Notes:

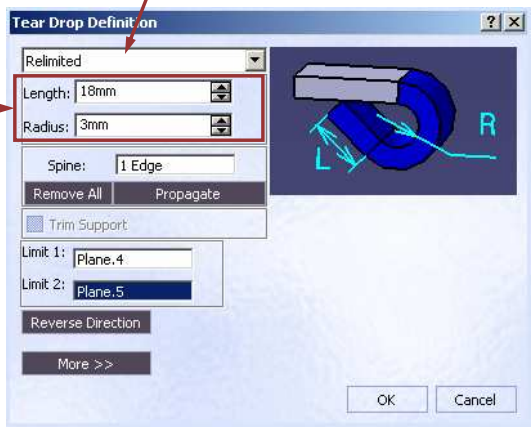
Create a Tear Drop

1 Click Tear Drop icon

2 Select Spine



3 Choose Basic or Relimited option, if you select Relimited option then you need to select two relimiting elements



4 Key Flange parameters: Radius, Length

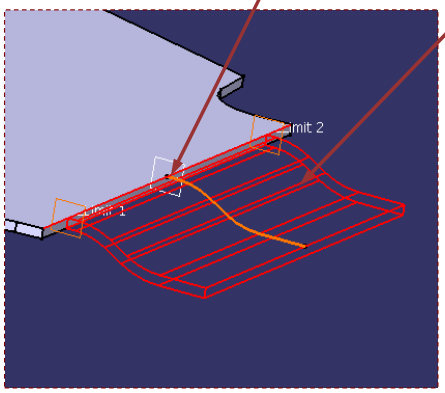
5 Select OK

Instructor Notes:

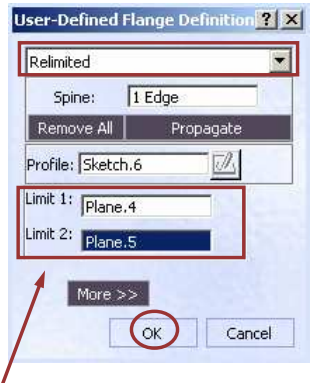
Create a User Flange

1 Click User Flange icon

2 Select Spine



3 Select Profile



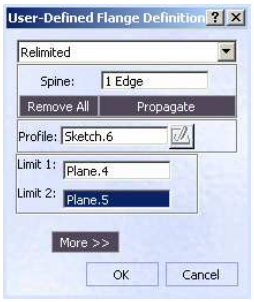
4 Choose Basic or Relimited option, if you select Relimited option then you need to select two relimiting elements

5 Select OK

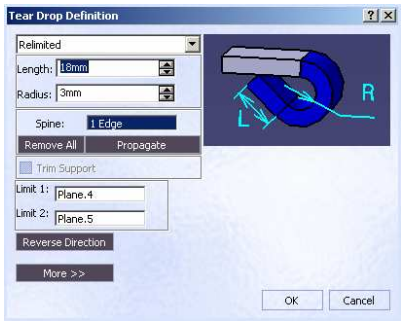
Instructor Notes:

Modifying Flanges other than Surfacic Flanges

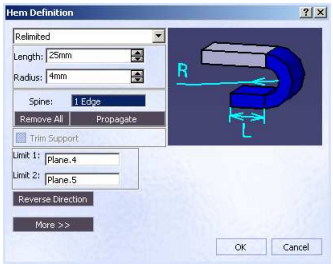
1. Double Click the Flange in the specification tree or on the geometry itself and the appropriate Flange Definition dialog box is displayed
2. Modify desired parameters and/or specification elements and then click OK to validate

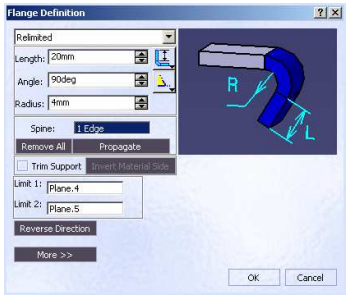


Step 1



Step 2

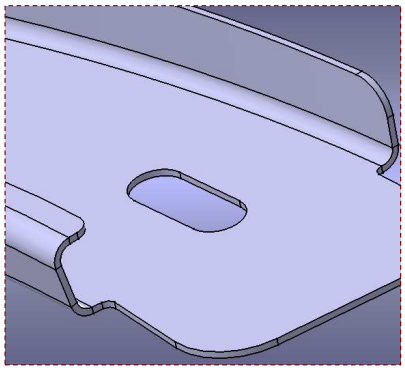


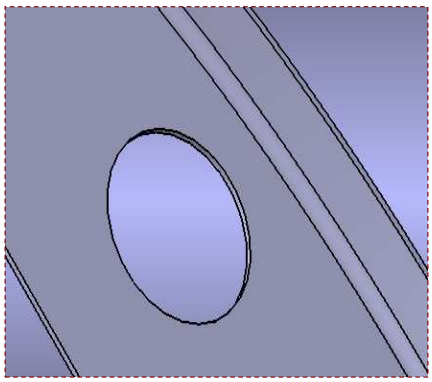


Instructor Notes:

Creating a Cutout

In this lesson you will learn to create a Cutout



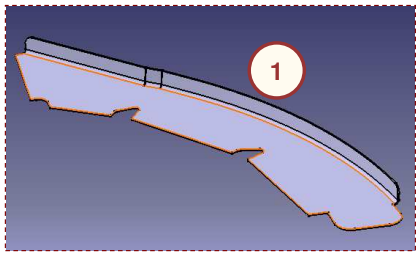


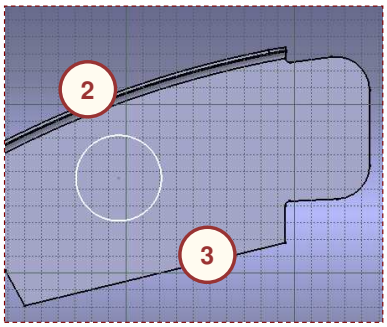
Instructor Notes:

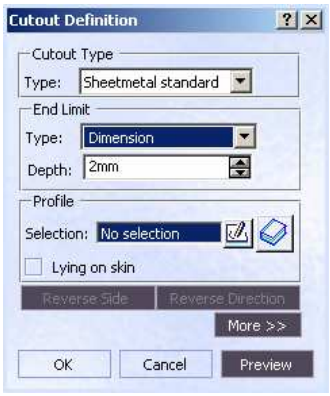
Creating a Cutout (1/5)

Creating a profile as Cutout profile

1. Create a sketch on the web surface or on a flange surface 
2. Define a closed contour
3. Exit the sketcher
4. Click the Cutout icon to display the Cutout Definition dialog box





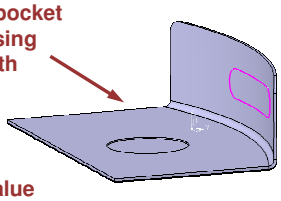


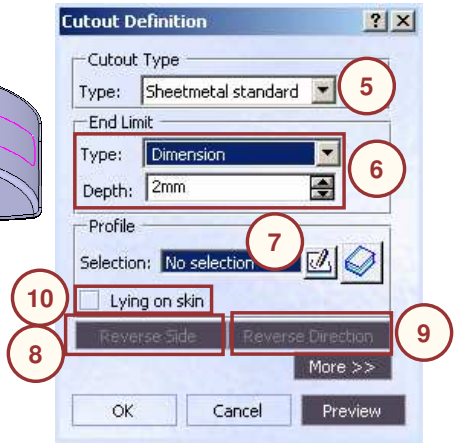
Instructor Notes:

Creating a Cutout (2/5)

Using a profile : Setting cutout Parameters

5 Select Sheetmetal standard or Sheetmetal pocket option. Sheet metal standard for sheet passing through cutout. Sheet metal pocket for depth smaller than sheet thickness





6 Three End Limit Types are available :

Dimension : depth is define by a specific value

Up to next : the limit is the first face the application detects while extruding the profile

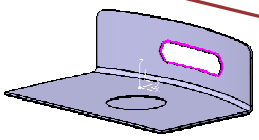
Up to last : the application will limit the cutout onto the last possible face encountered by the extrusion

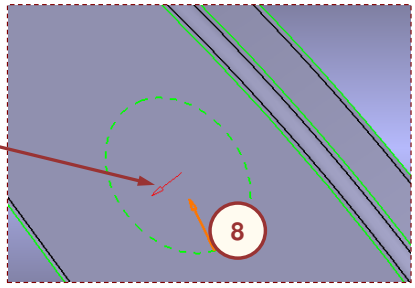
7 Select the profile previously created with the sketcher

8 Select the material part to keep : click on the Reverse Side button or on the orange arrow

9 Determine the cutout side : click on the Reverse Direction button or on the red arrow

10 Lying on skin allows selection of a 3D curve as profile definition



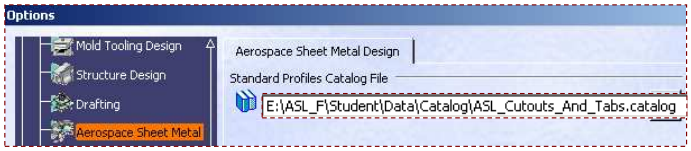


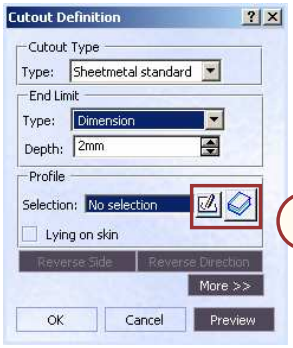
Instructor Notes:

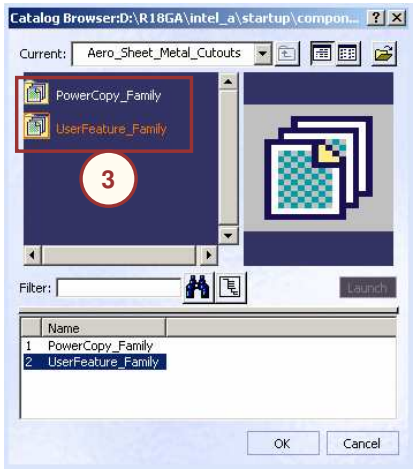
Creating a Cutout (3/5)

- Inserting a profile instance from a catalogue
  - Prior to inserting profile instance from catalogue, you should set up catalogue access through Tools / Options / Mechanical Design / Aerospace Sheet Metal

1. Click the Cutout icon to display the Cutout Definition dialog box
2. Select the Open Catalogue icon
3. Select the Power Copy Family Object or User Feature Family Object







Instructor Notes:

Creating a Cutout (4/5)

Inserting a profile instance from a catalogue (cont'd)

4. Double click on Slot_contour

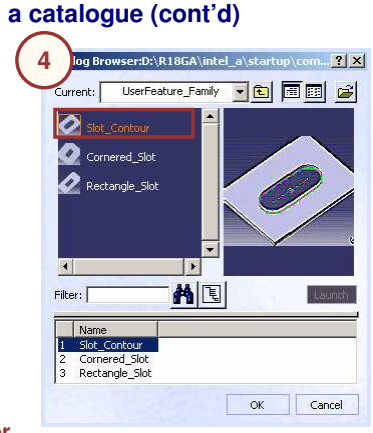
5. Select the web plane as Ref_Plane input, a point as Ref_Point and a line as Ref_Axis for orientation purpose

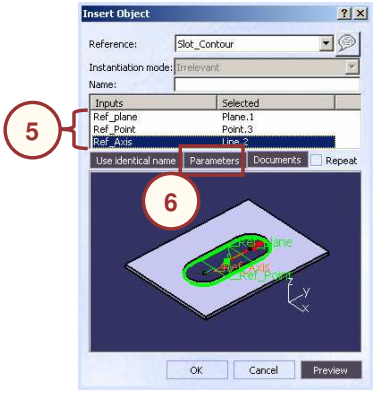
6. Select Parameters

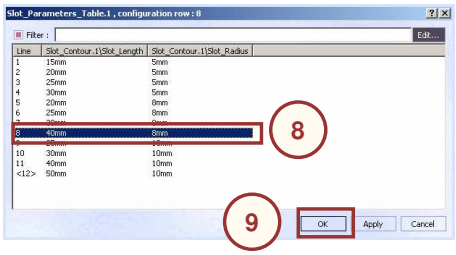
7. Select Design table icon and/or modify Angle value

8. Select 8th row

9. Click OK and Close all windows related to contour instantiation





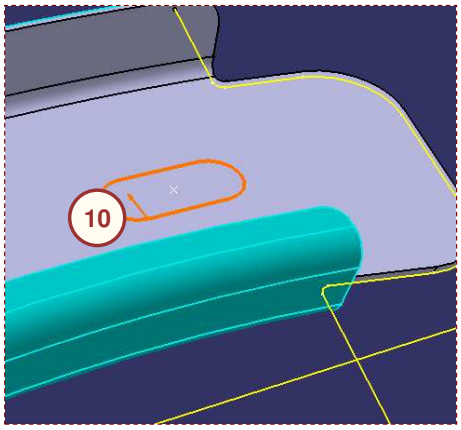


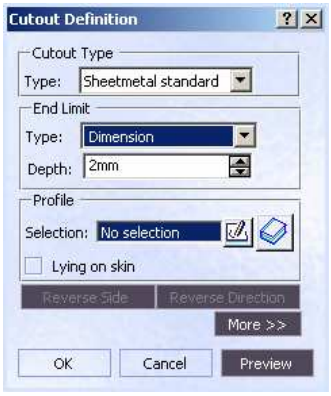
Instructor Notes:

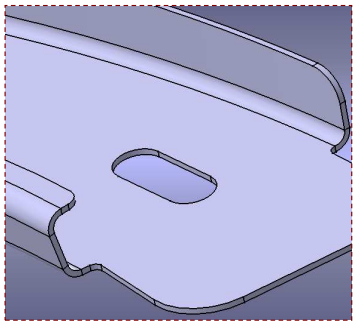
Creating a Cutout (5/5)

Inserting a profile instance from a catalogue (cont'd)

10 **Check that orange arrow is pointed inside the profile and click OK**





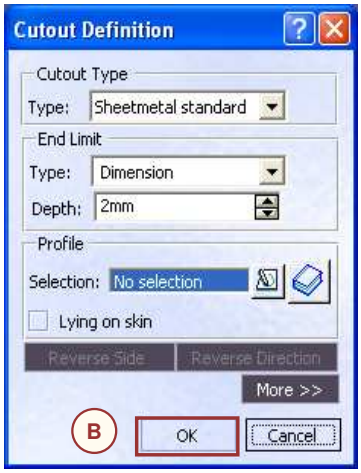


Instructor Notes:

Modifying a Cutout

A. Double Click the Cutout on the part or in the tree and the Cutout Definition dialog box is displayed.

B. Modify desired parameters and click OK to validate.



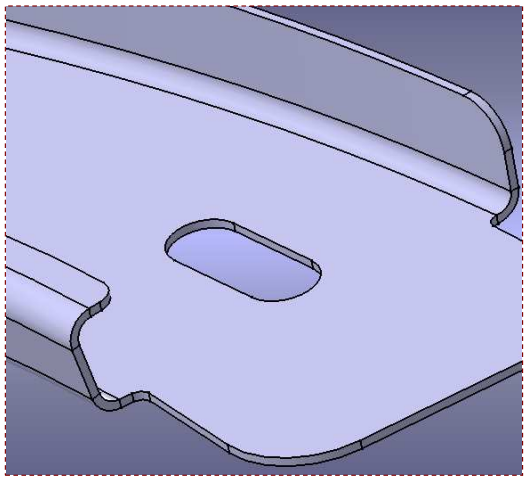
Instructor Notes:

Master Exercise: Fighter Airframe

Step 7: Creating a Cutout

5 min

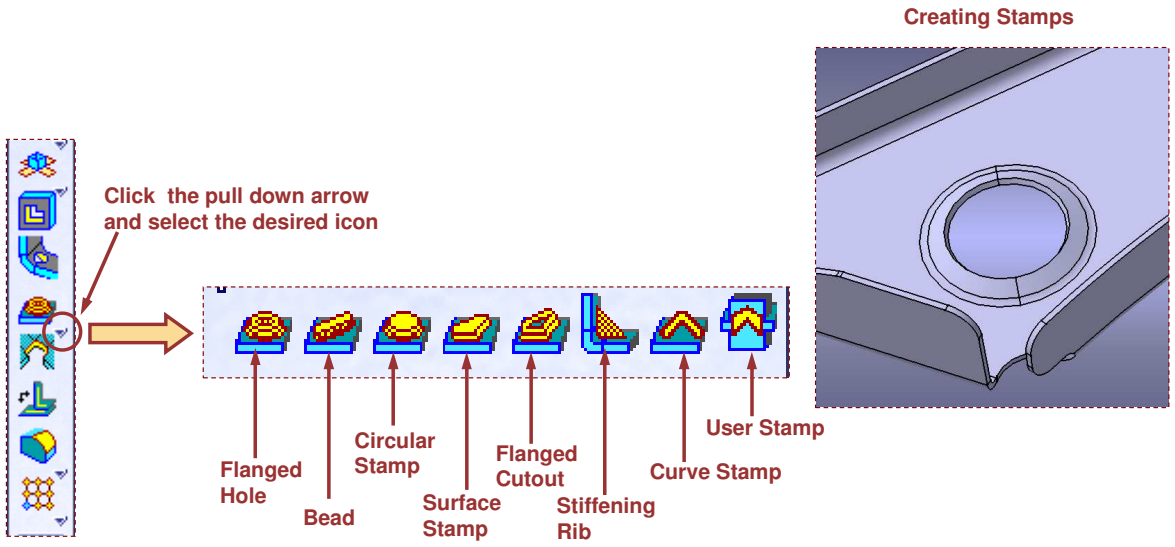
In this step we create a cut out on the web



Instructor Notes:

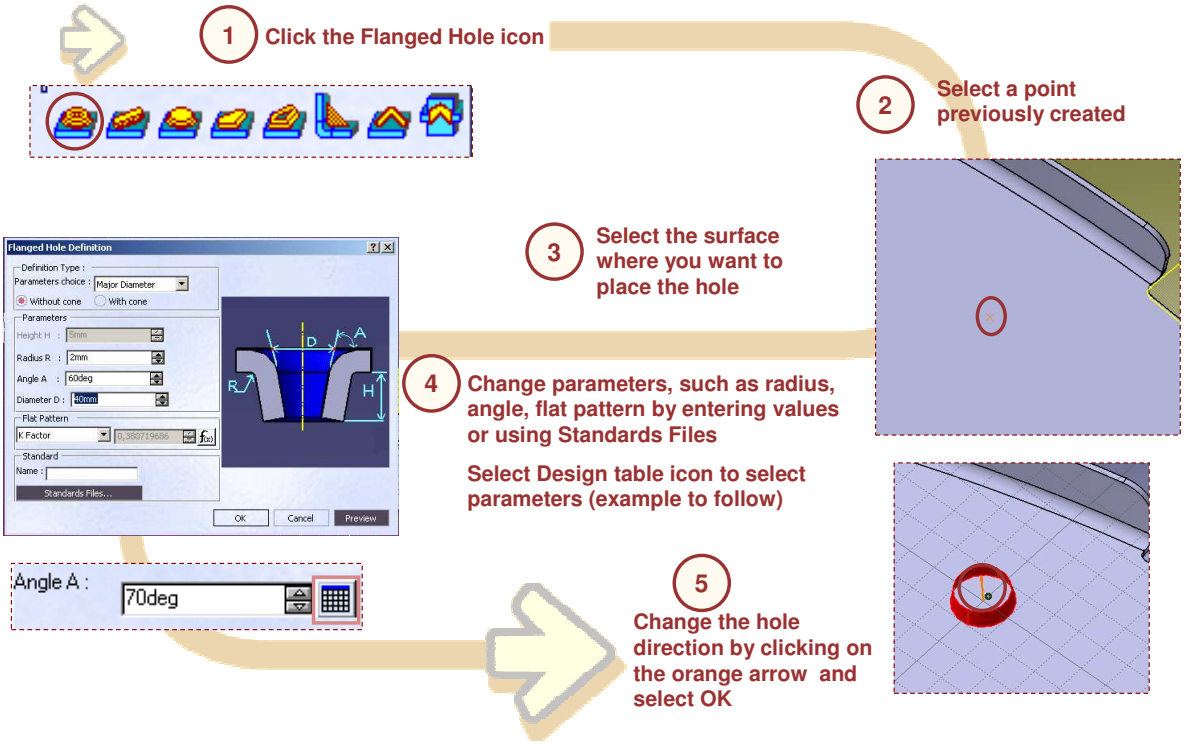
Creating Stamps

In this lesson you will learn to create and modify Stamps



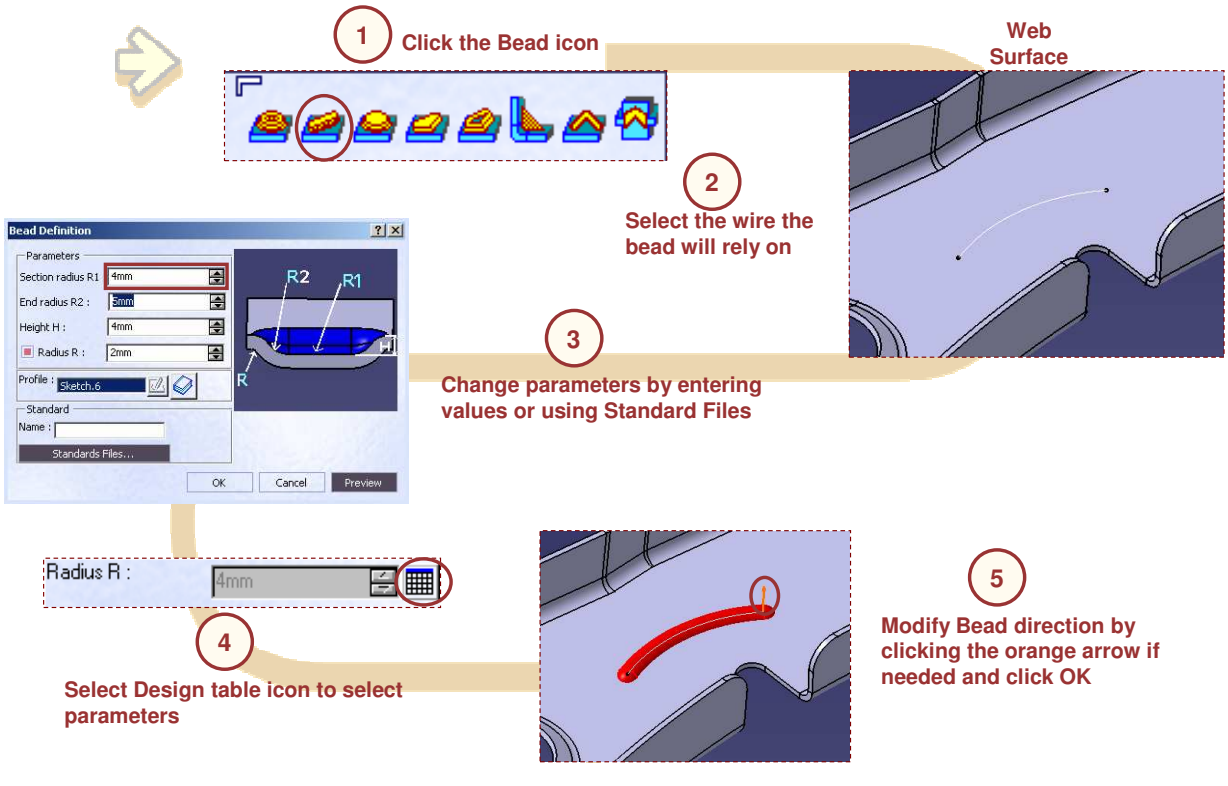
Instructor Notes:

Creating a Flanged Hole



Instructor Notes:

Creating a Bead

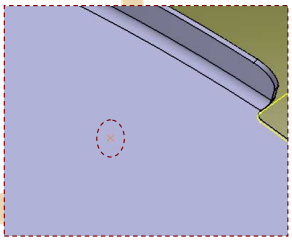


Instructor Notes:

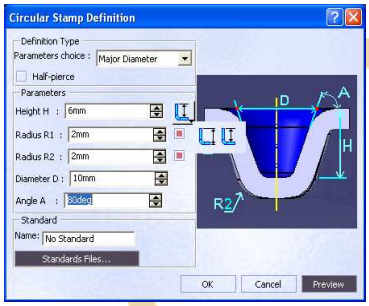
Creating a Circular Stamp

1 Click the Circular stamp icon

2 Select a point previously created



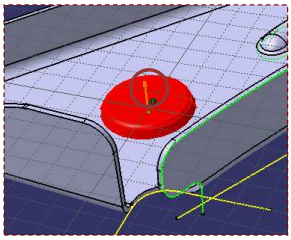
3 Select the surface where you want to place the hole



4 Change parameters by entering values or using Standards Files

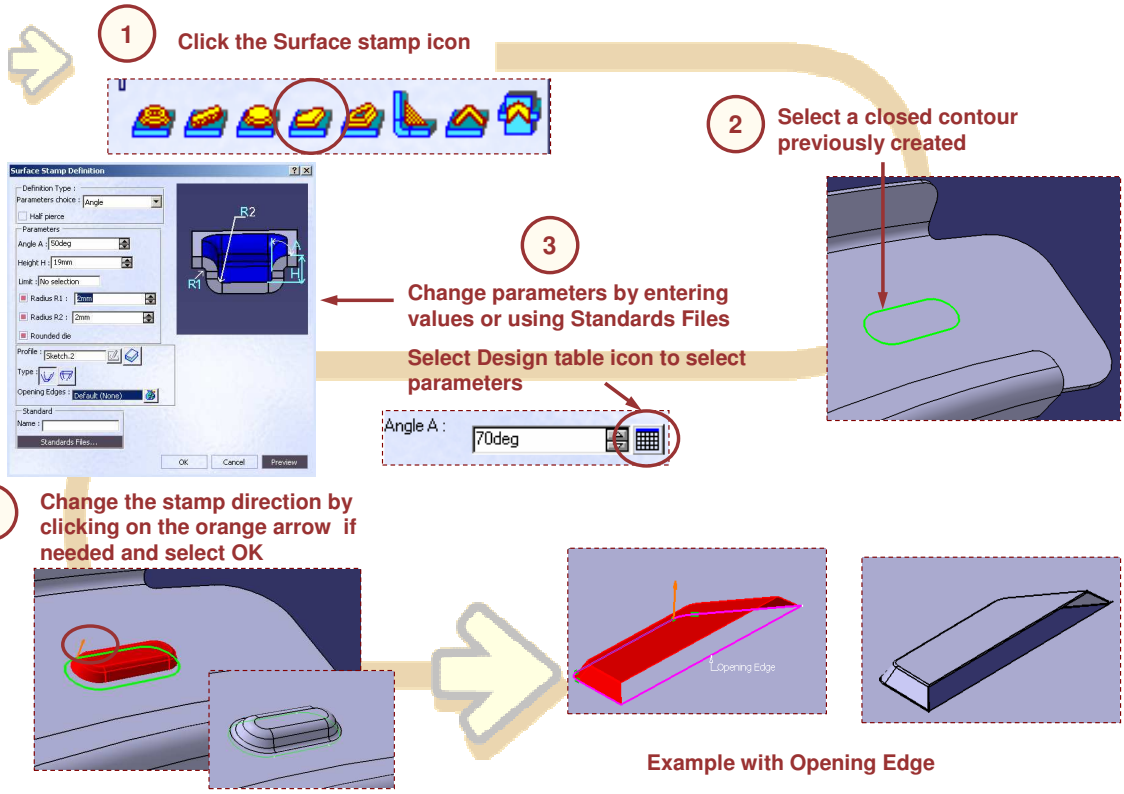
Select Design table icon to select parameters

5 Change the stamp direction by clicking on the orange arrow if needed and select OK



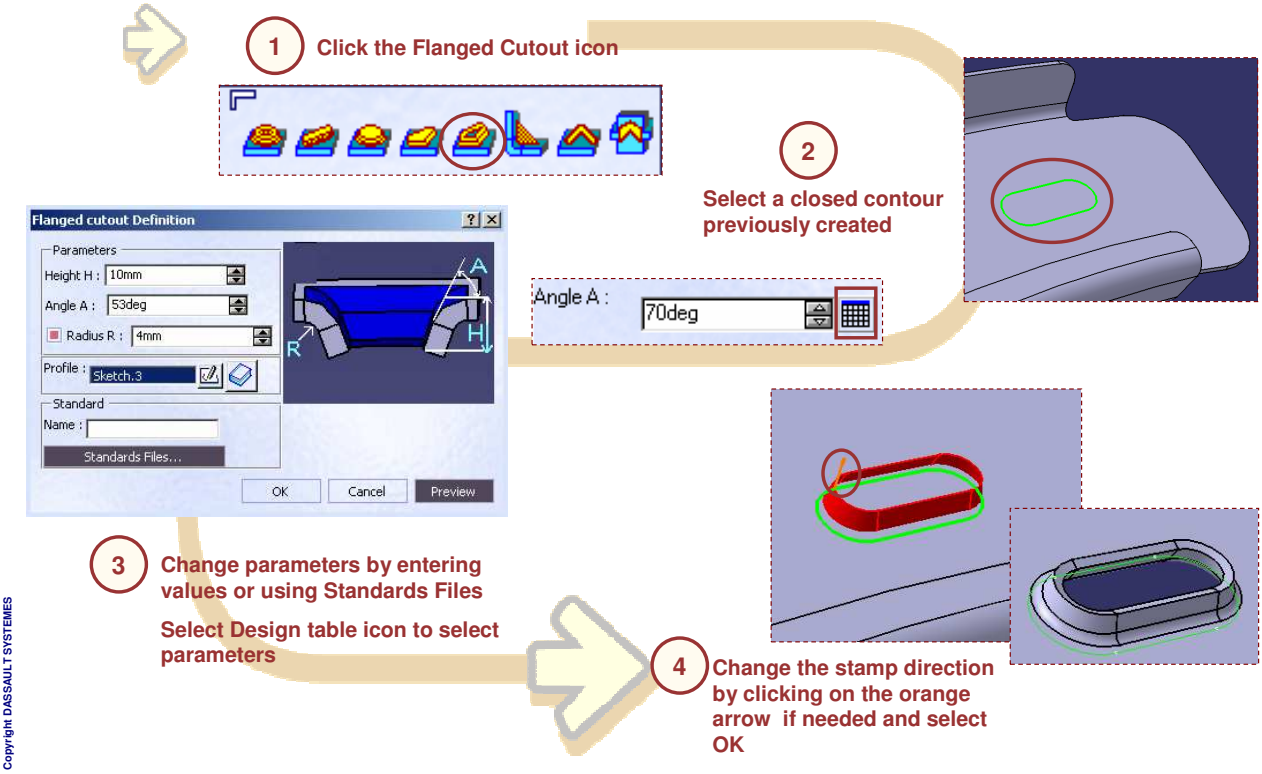
Instructor Notes:

Creating a Surface Stamp



Instructor Notes:

Creating a Flanged Cutout

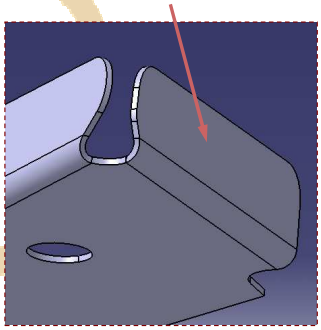


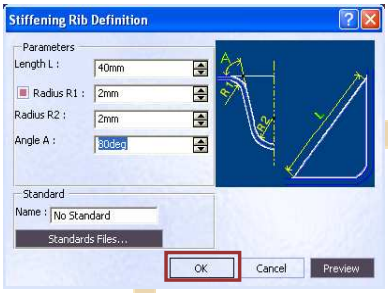
Instructor Notes:

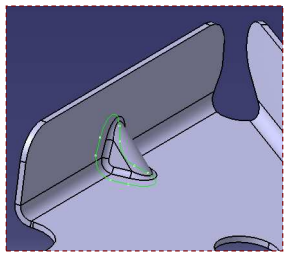
Creating a Stiffening Rib

1 Click the Stiffening Rib icon

2 Select the external face of a straight bend

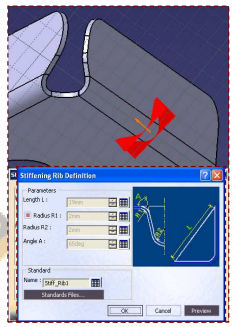






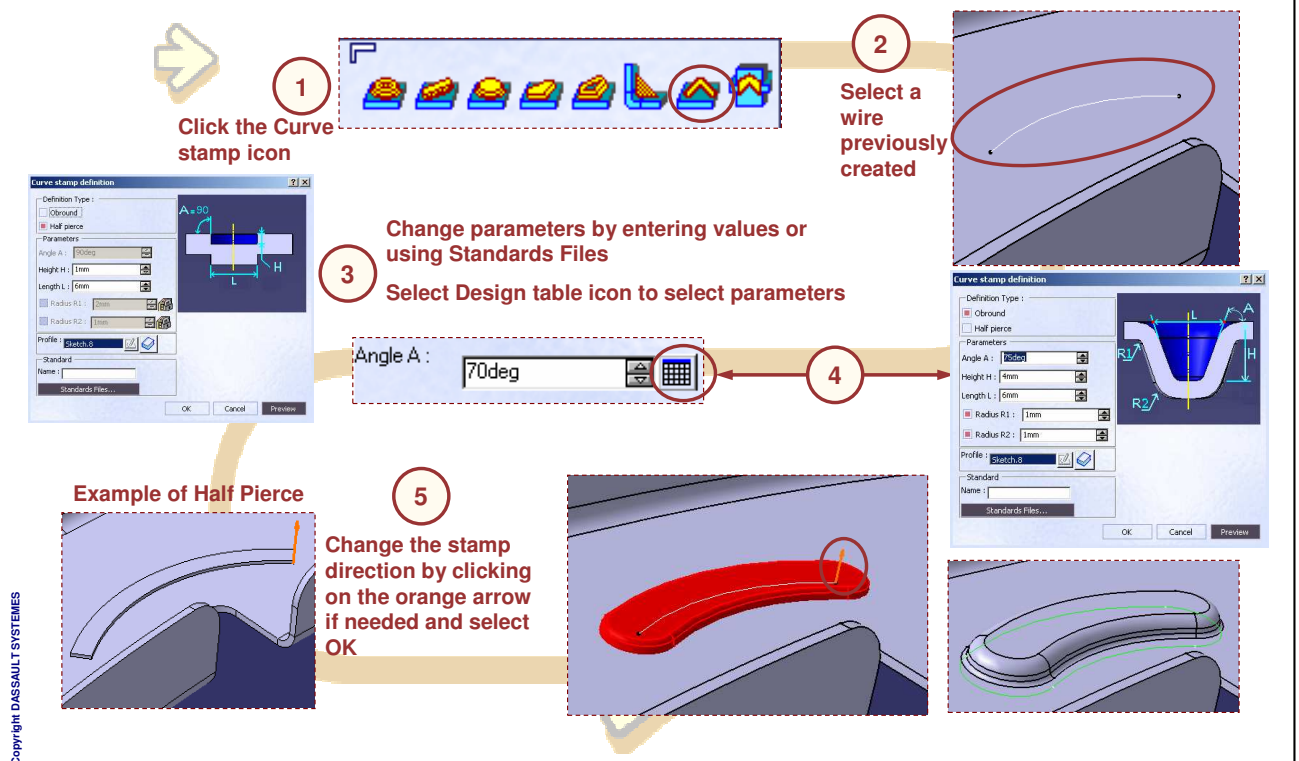
3 Change parameters by entering values or using Standards Files

Select Design table icon to select parameters



Instructor Notes:

Creating a Curve Stamp

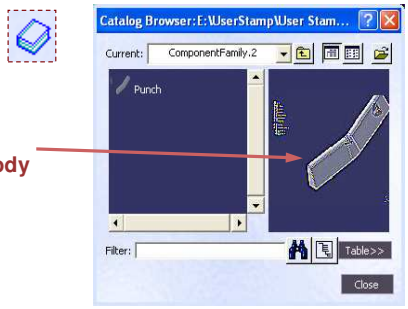


Instructor Notes:

Creating a User Stamps (1/2)

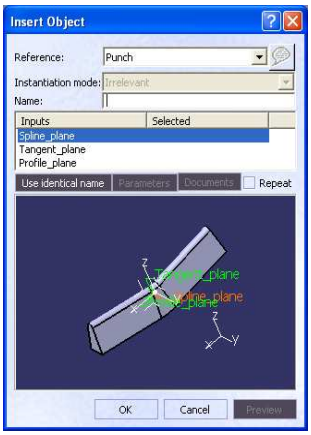
Create a User Stamp

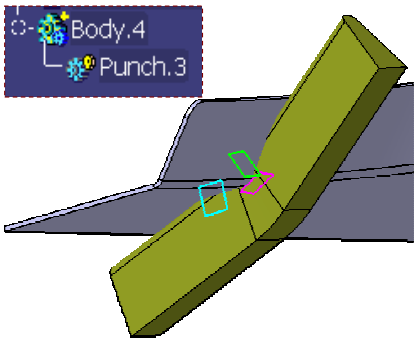
- Before actually creating the user stamp, you should first define or import the punch and eventually the die that will later be used for actual stamping



1

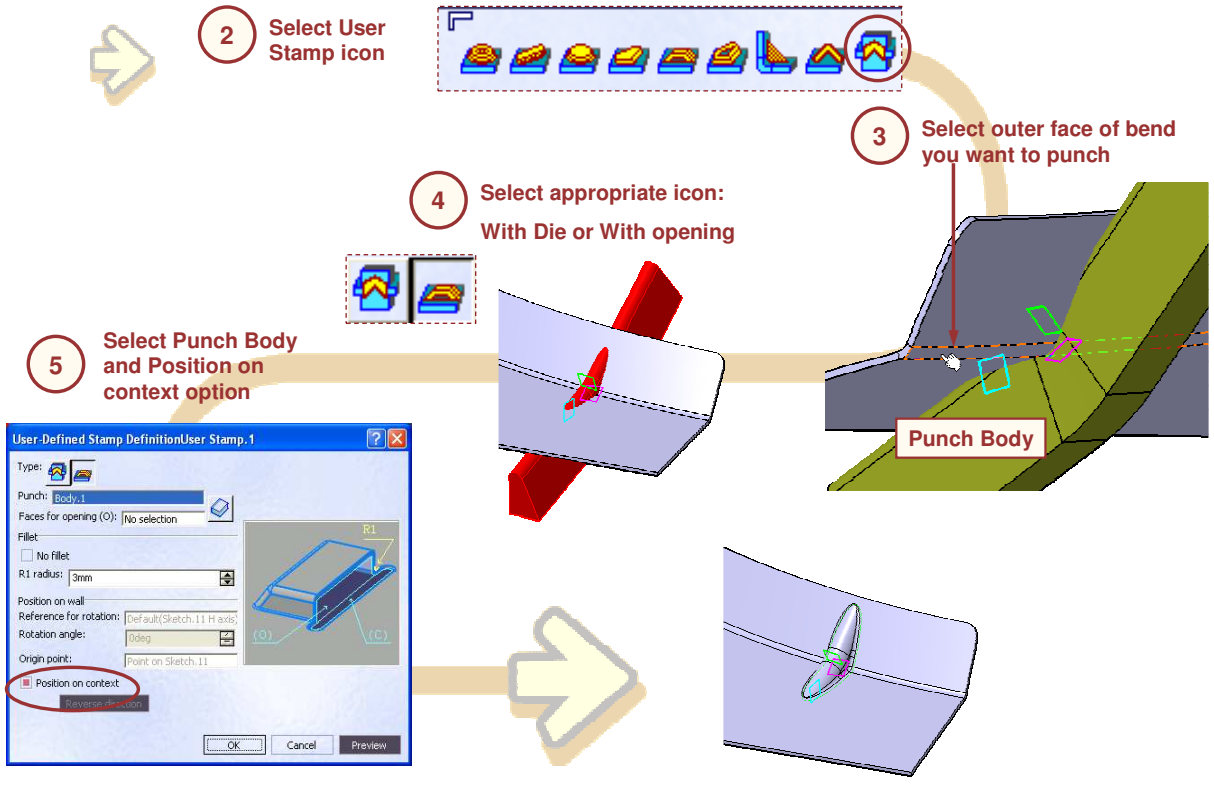
Import the Punch from a catalog into a specific Body





Instructor Notes:

Creating a User Stamps (2/2)



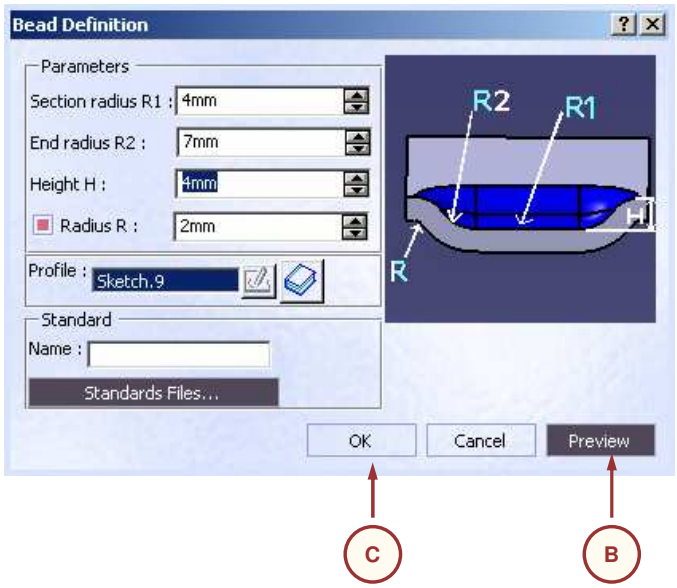
Instructor Notes:

Modifying a Stamp

A. Double Click the stamp to be modified

B. Modify desired parameters and click Preview if you wish

C. Click OK to validate



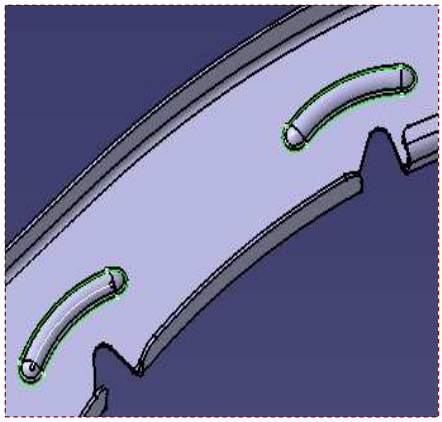
Instructor Notes:

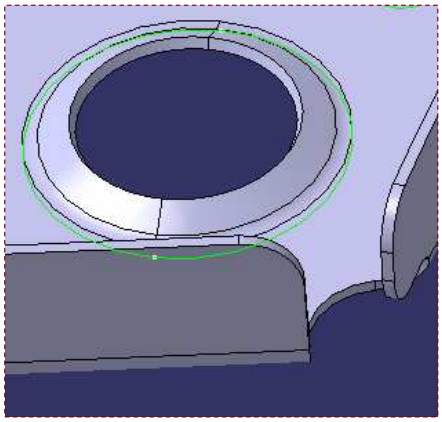
Master Exercise: Fighter Airframe

Step 8: Creating a Flanged Hole and Bead

2 min

In this step we create a flanged hole on the web

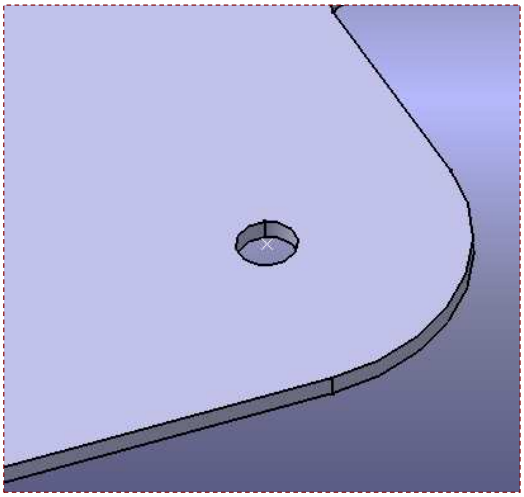




Instructor Notes:

Creating a Hole

In this lesson you will learn to create a Hole



Instructor Notes:

Creating a Hole (1/4)

A. Click Hole tool

B. Select a point on the web

C. Select the web

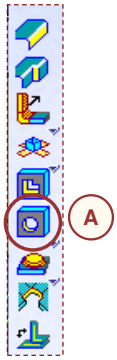
D. Choose a bottom limit for the hole

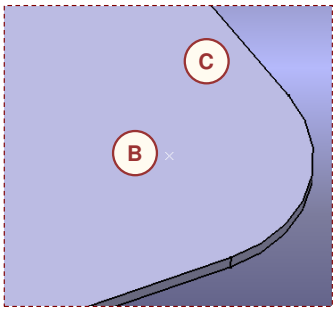
E. Specify the values as needed

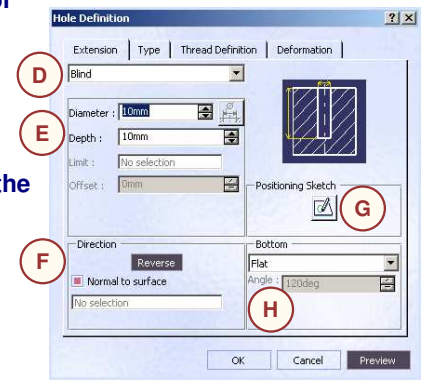
F. Specify the direction of the hole

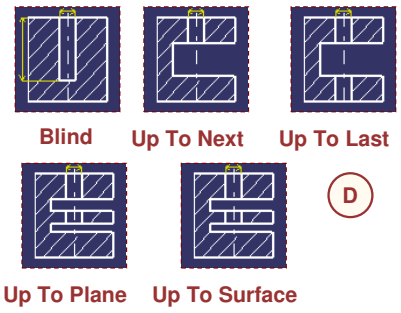
G. Position the hole on the web

H. Choose a bottom type for the hole





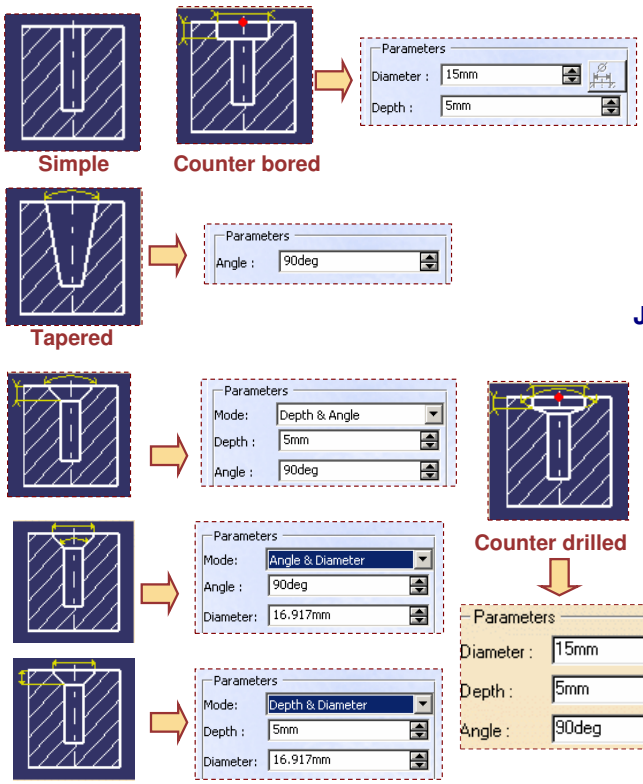


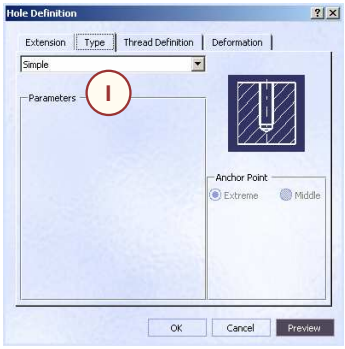


Instructor Notes:

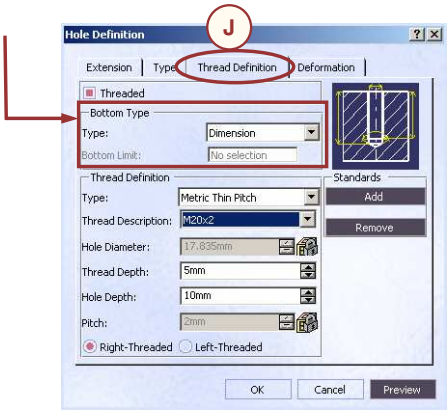
Creating a Hole (2/4)

I. Select the type of hole





J. Select the Bottom Type of a threaded hole

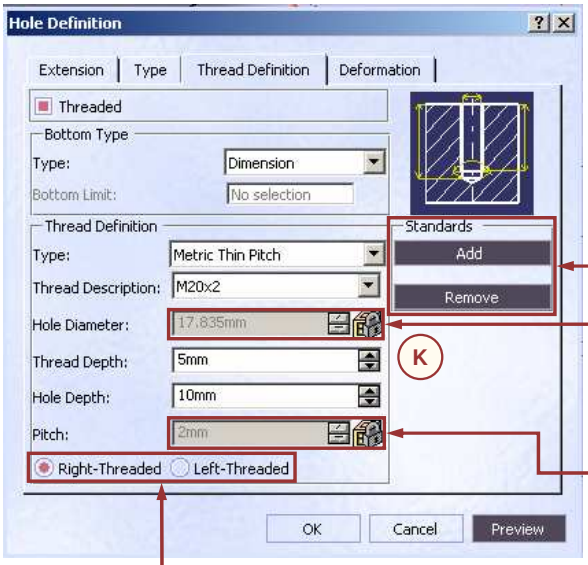


Instructor Notes:

Creating a Hole (3/4)

K. Specify other Thread Parameters

Hole Diameter, Pitch, Right or Left-Threaded, Add or Remove Standards



To add or remove one or several standards, you can use these two buttons.

By default, the Hole Diameter is automatically calculated in accordance with the Thread Diameter and the Standard. You can modify it to get a non-standard thread .

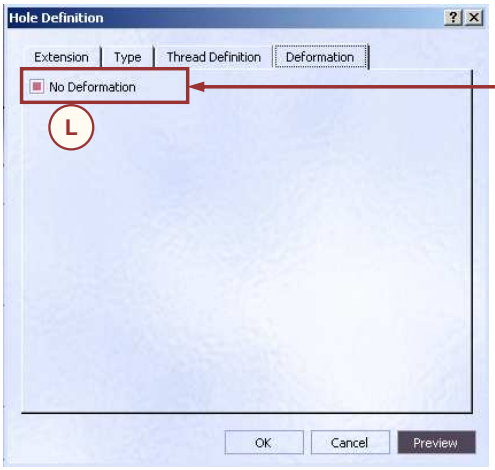
By default, the Pitch is automatically calculated in accordance with the Thread Diameter and the Standard. You can modify it to get a non standard thread.

You can choose a Left or Right-Threaded hole by selecting one of these two options.

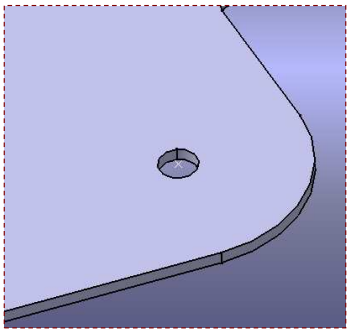
Instructor Notes:

Creating a Hole (4/4)

L. Specify Deformation



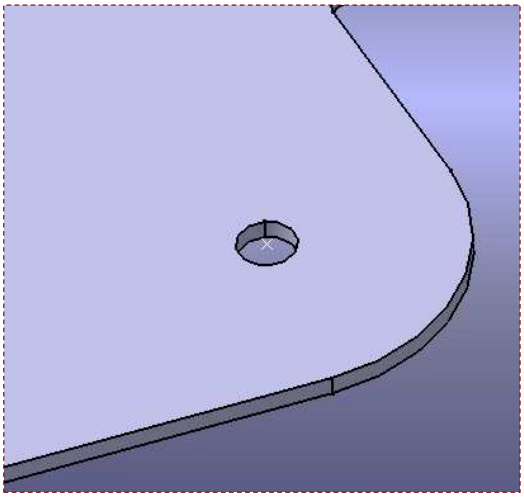
You can create a cylindrical hole in folded and unfolded view by clicking on No Deformation option in the Deformation tab.



Instructor Notes:

Creating a Circular Cutout

In this lesson you will learn to create a Circular Cutout



Instructor Notes:

Creating a Circular Cutout

A. Create a point on the web

B. Select Circular Cutout icon

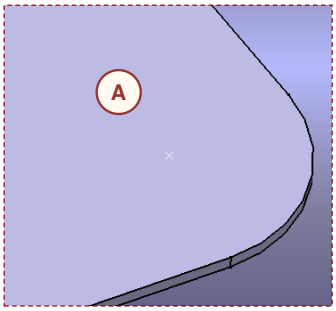
C. Select the point previously created

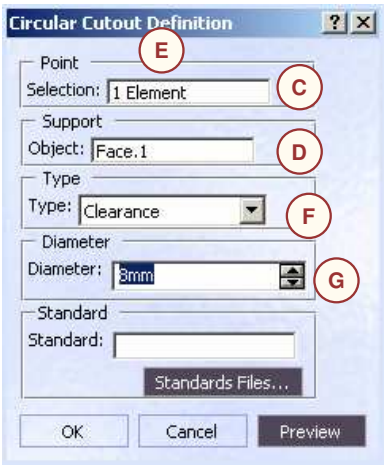
D. Select web as support

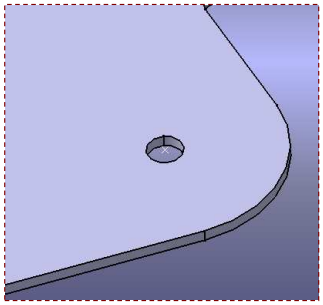
E. Select additional points if necessary

F. Select type of hole

G. Enter Diameter value and click OK







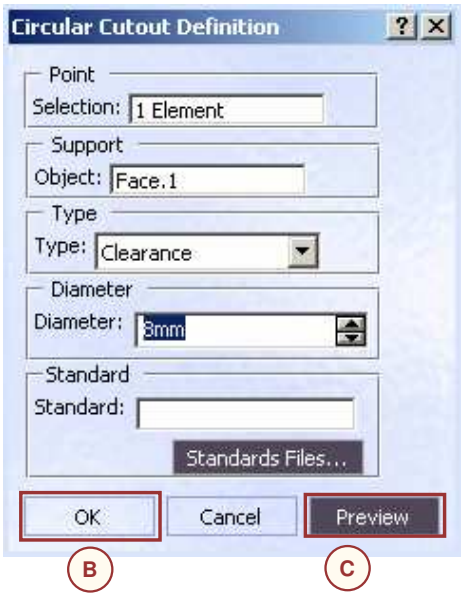
Instructor Notes:

Modifying a Circular Cutout

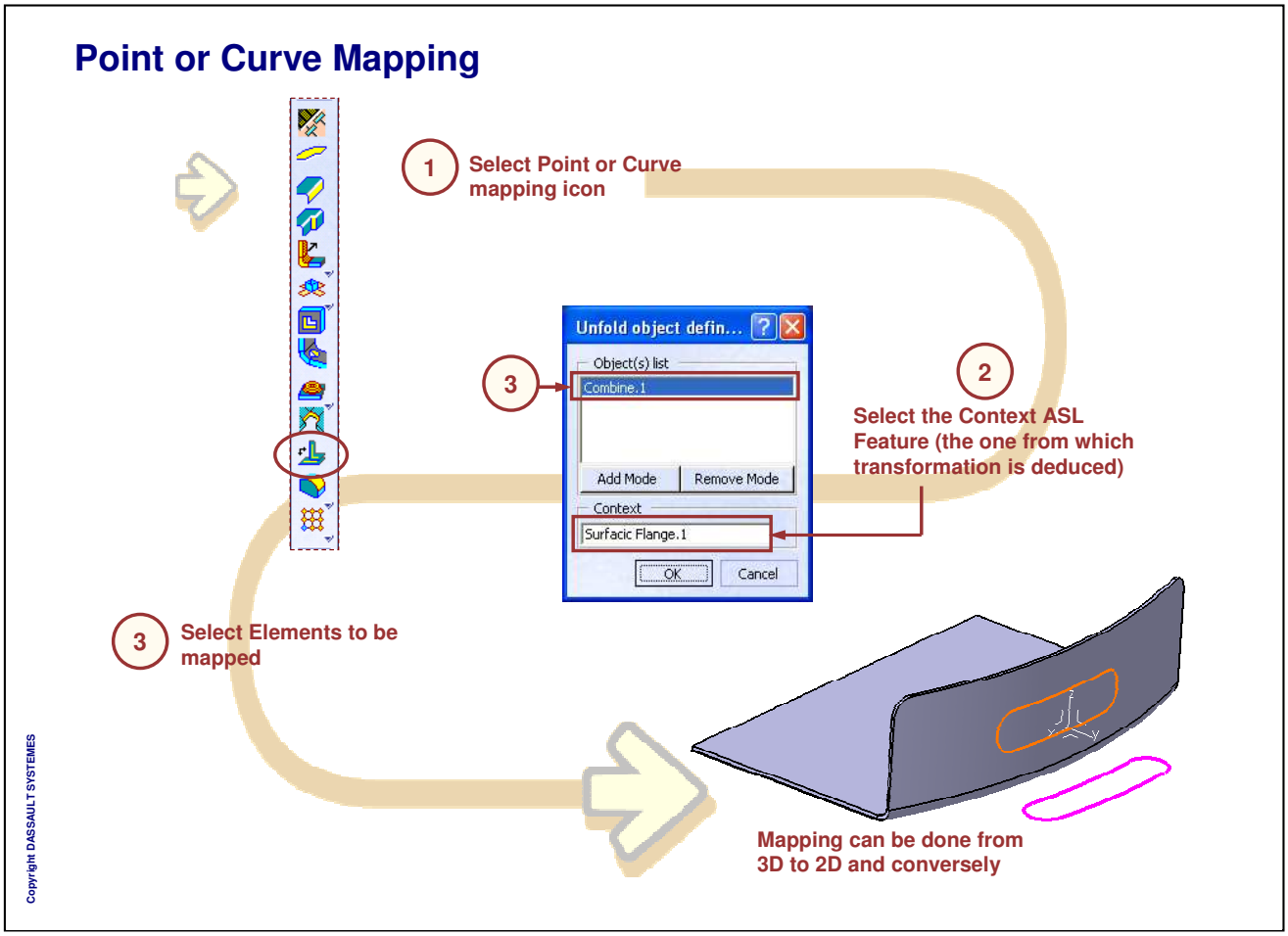
A. Double Click the Circular Cutout on the part or in the tree and the Circular Cutout Definition dialog box is displayed

B. Modify desired parameters and click OK to validate

C. Click Preview if you wish

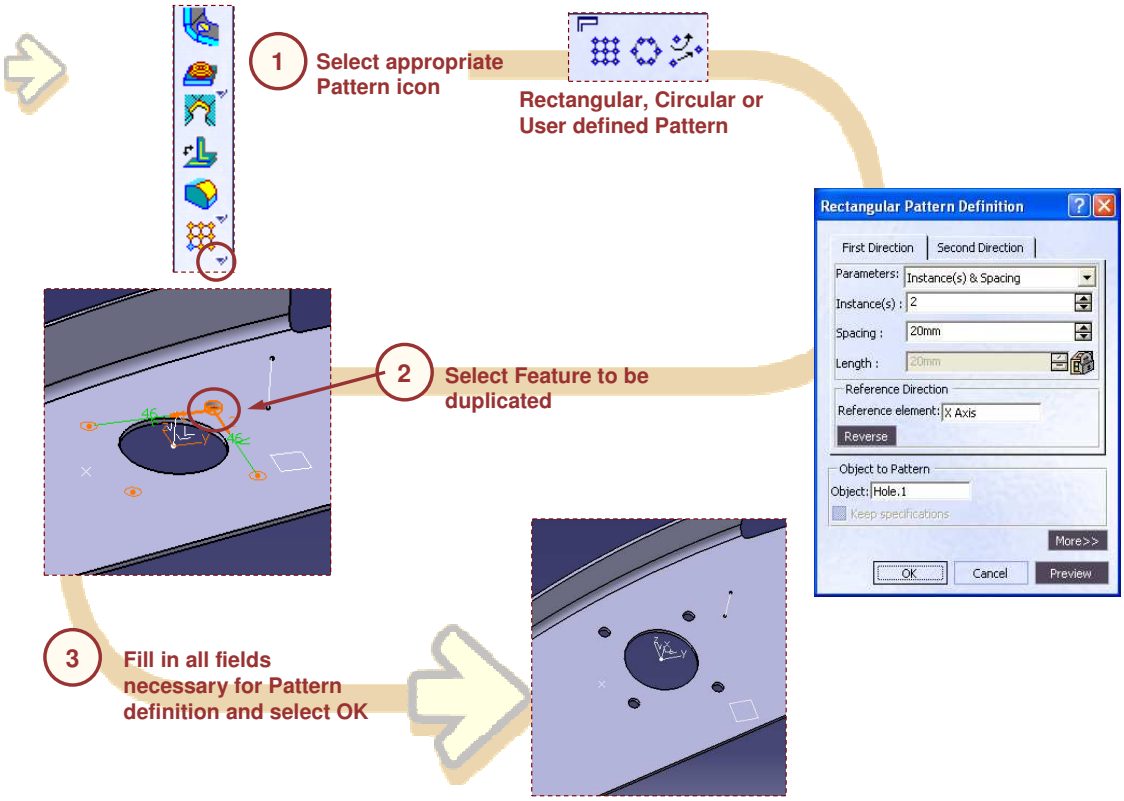


Instructor Notes:



Instructor Notes:

Duplicate ASL Feature by applying a Pattern



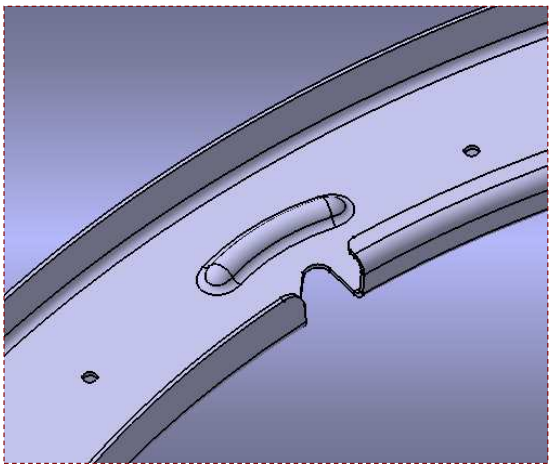
Instructor Notes:

Master Exercise: Fighter Airframe

Step 9: Creating Circular Cutouts

2 min

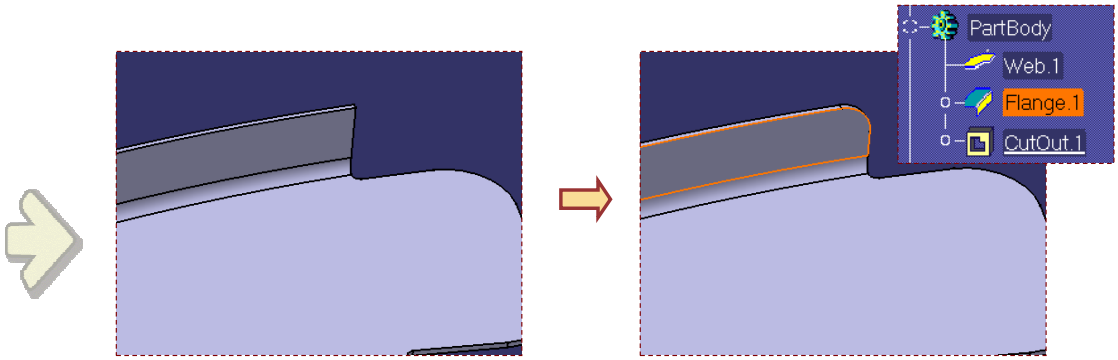
In this step we create circular cutouts on the web



Instructor Notes:

Modifying a Feature

In this lesson you will learn to modify a feature

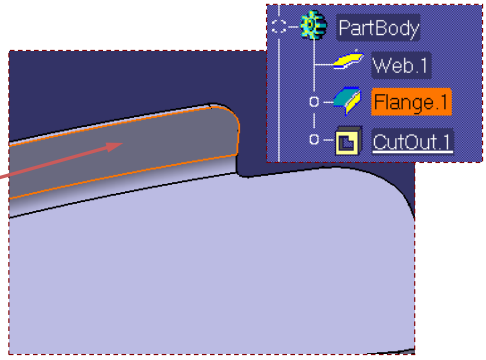


Instructor Notes:

Modifying a Feature (1/3)

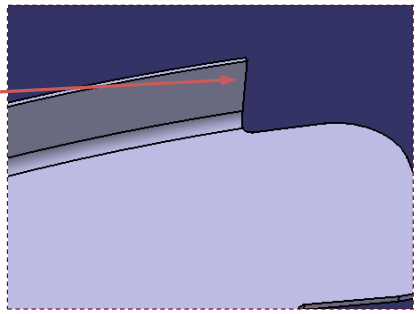
- Accessing feature definition window

A. Double Click the Feature to be modified on the part or in the tree and the Feature Definition dialog box is displayed (ex. : the first flange definition)



B. Modify desired parameters and click OK to validate (here we suppress the first corner of the flange).

   Update if necessary



Instructor Notes:

Modifying a Feature (2/3)

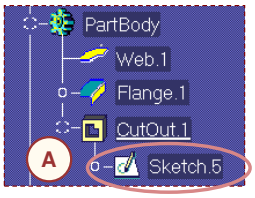
- **Modifying Sketcher Support**

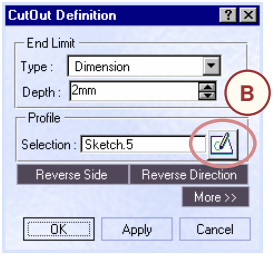
1. **Edit the sketch linked to the feature to be modified (ex. : the first Cutout definition)**

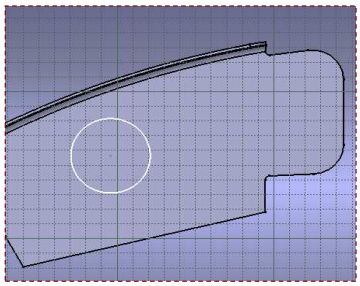
Two ways:

A. **Double-click on the sketch in the tree**

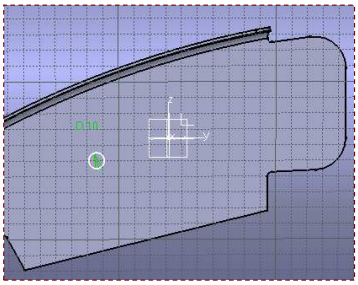
B. **Edit the feature definition window and click the Sketcher button**



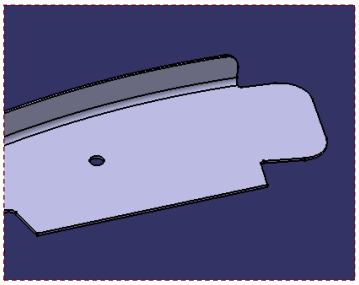




2. **Modify the closed contour**



3. **Exit the sketcher and update if necessary**



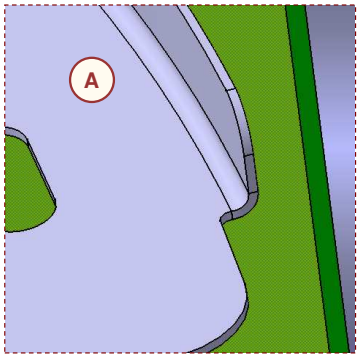
Instructor Notes:

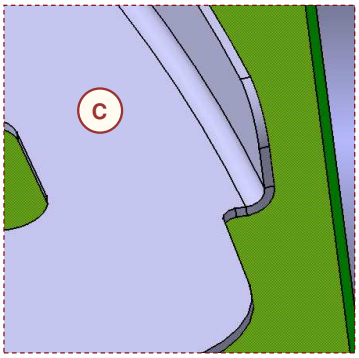
Modifying a Feature (3/3)

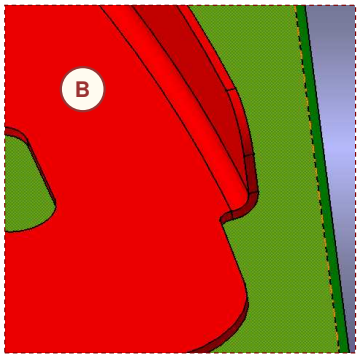
A. Change to WFS workbench 

B. Modify a support surface. Here we modify the offset of a flange support surface

C. Change to ASL workbench and update 







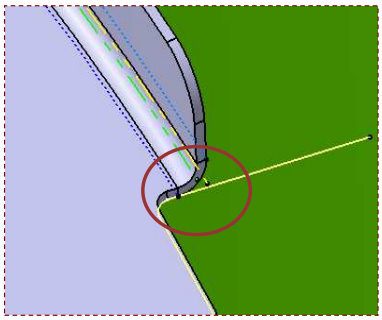
Instructor Notes:

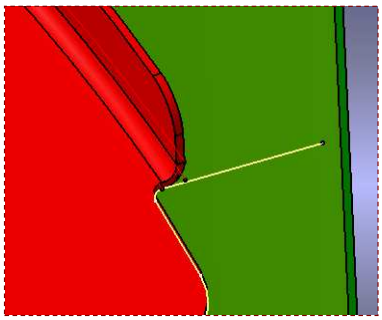
Master Exercise: Fighter Airframe

Step 10: Modifying Features

5 min

In this step we modify a flange surface support

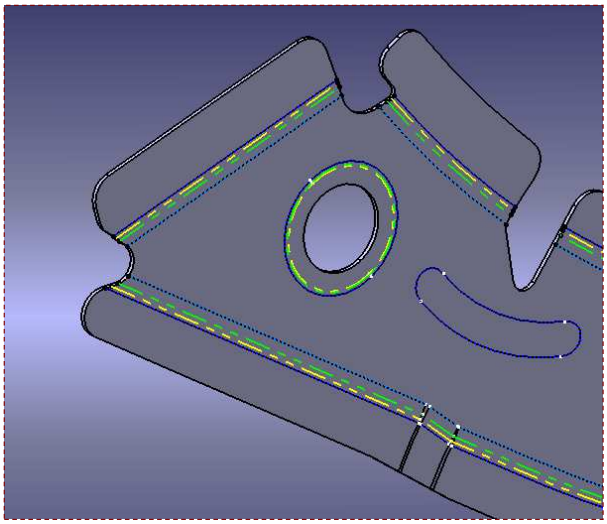




Instructor Notes:

Generating Folded and Flattened parts

In this lesson you will learn to Fold / Unfold a part.

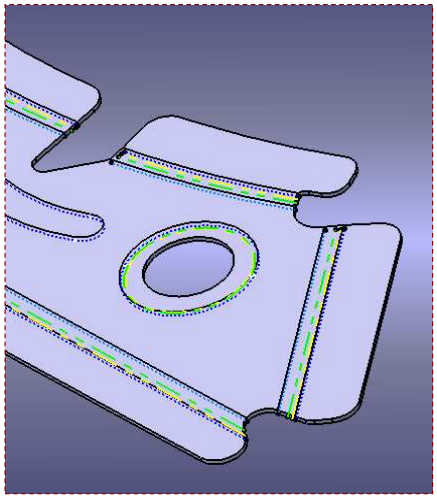


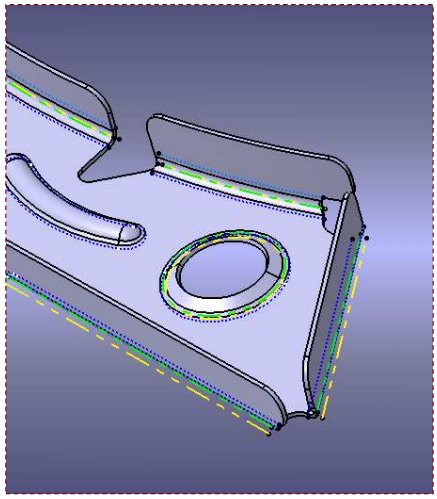
Instructor Notes:

Folding / Unfolding the Part

1. Select the Unfold icon to flatten the part according to the web

2. Select the Unfold icon again and the part is folded back



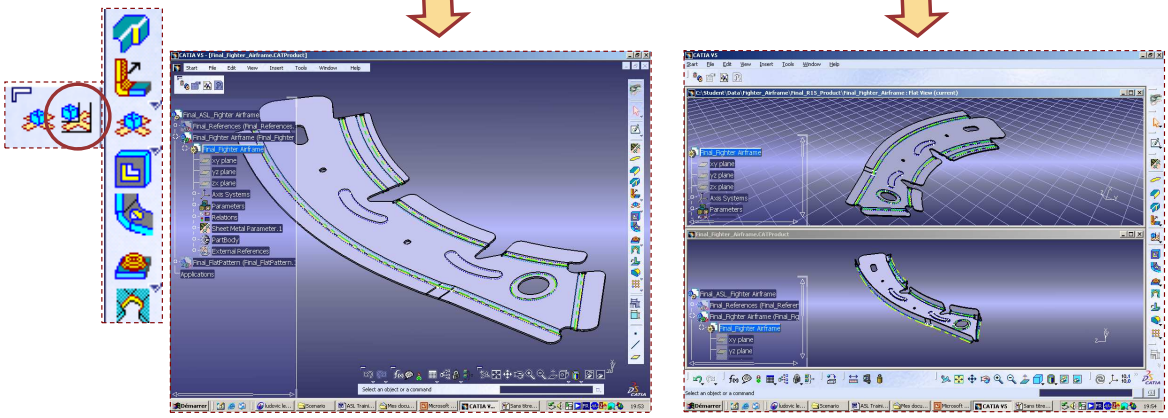


Instructor Notes:

Using Multi-View

1. Click the Multi-View icon to unfold the part in a second window

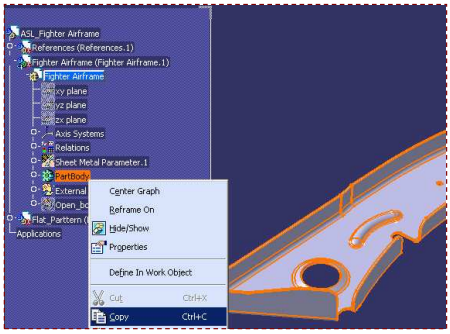
2. Select Window > Tile - Horizontally menu item to display both windows



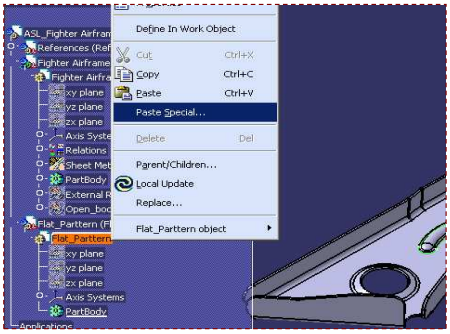
Instructor Notes:

Flat Solid linked to ASL Designed Part

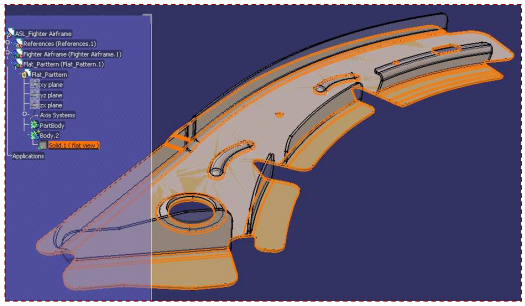
1. Copy the Part Body out of the ASL designed part



2. Paste As Result With Link Flat mode into another part or in another Body of the same part



You obtain a Solid (linked to initial ASL part) where you can add on Excess and Tabs

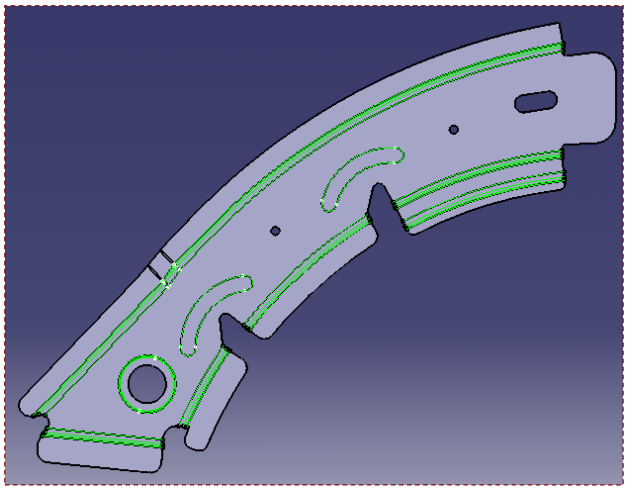


Instructor Notes:

Master Exercise: Fighter Airframe

Step 11: Flattening the Part

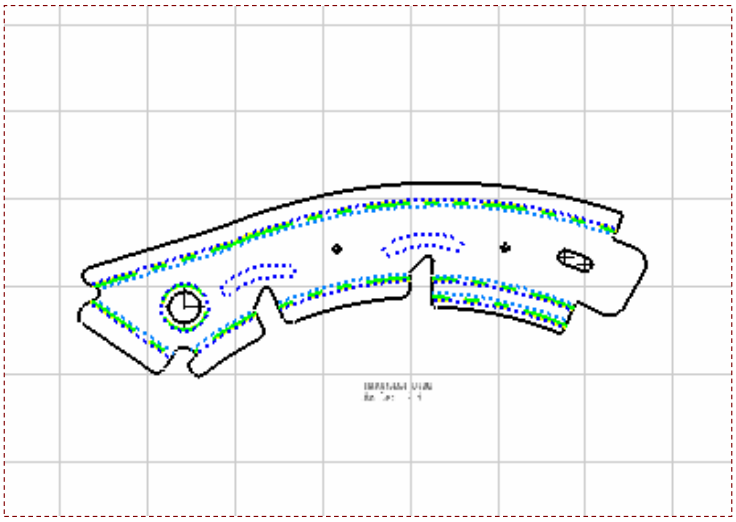
In this step you will flatten the part and use Multi-view



Instructor Notes:

Drawing Generation

In this lesson you will learn to generate a drawing of an ASL part.



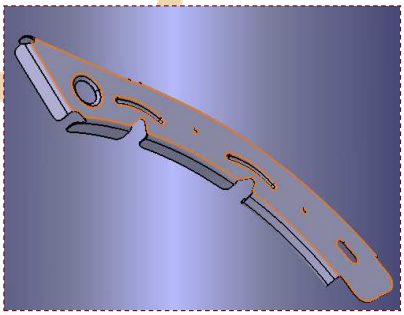
Instructor Notes:

Drawing Generation

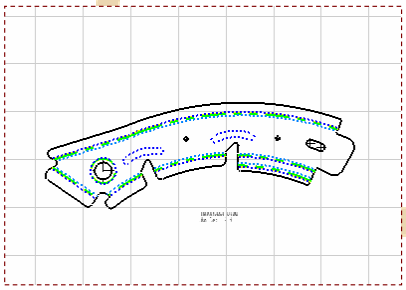
1 Change to Drafting Workbench and select an empty sheet

2 Select the Unfolded View icon

3 Select the web on the part



4 Select a place on the sheet where to locate the flattened view



NB : the characteristic curves are generated in the drawing

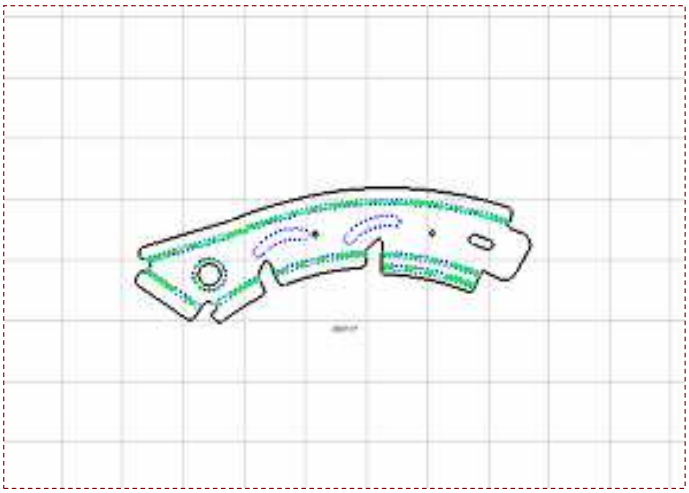
Instructor Notes:

Master Exercise: Fighter Airframe

Step 12: Drawing Generation

2 min

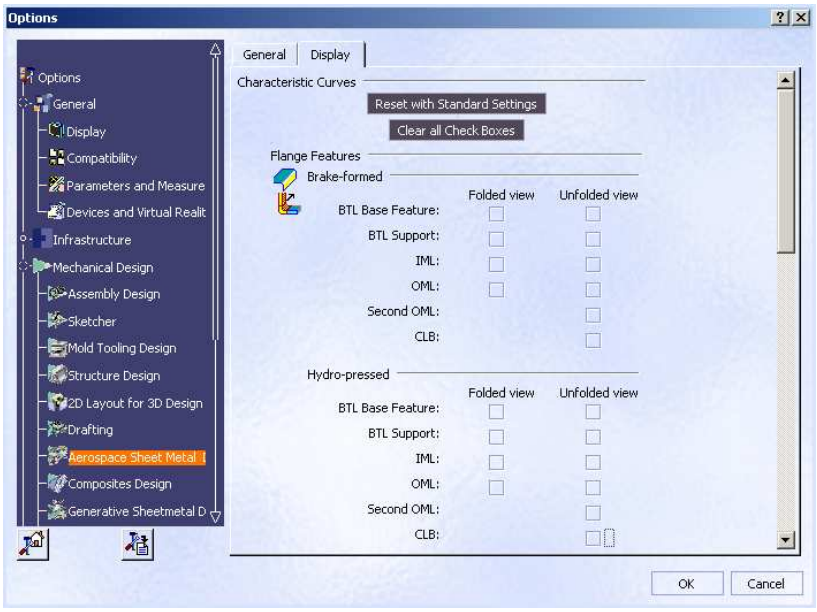
In this step we draw the flatten view of the part



Instructor Notes:

Administration

In this lesson you will learn Administration Tasks.

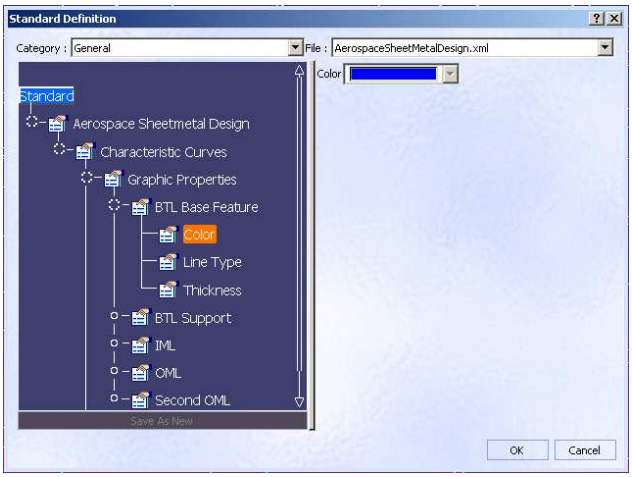


Instructor Notes:

About Standards

- Standards are embedded in the sheet metal part
- Standards are administrator-defined
- A standard file is available by default
- Editing the standard file

  The standard file can be edited using an interactive editor. This editor provides an easy-to-use graphic interface to let you customize the parameters included in the standard file.

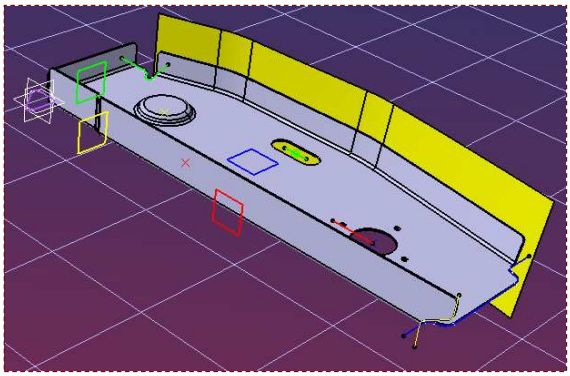


Instructor Notes:

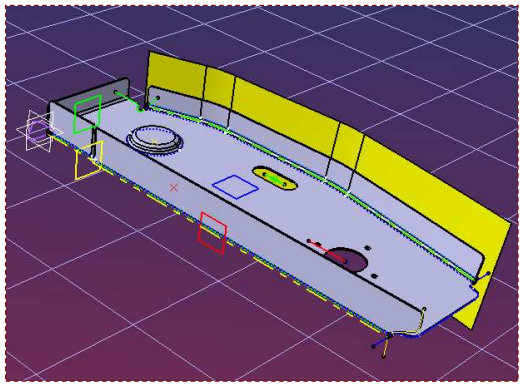
Setting Standard Parameters

- Administrators can for instance :
  - Customize the graphic properties of characteristic curves in both 3D (Aerospace Sheet metal Design workbench) and 2D views (Generative Drafting workbench).
  - Manage the visibility of all characteristic curves (BTL, IML, OML and 2nd OML) in folded and unfolded 2D views through standards.

Invisible curves



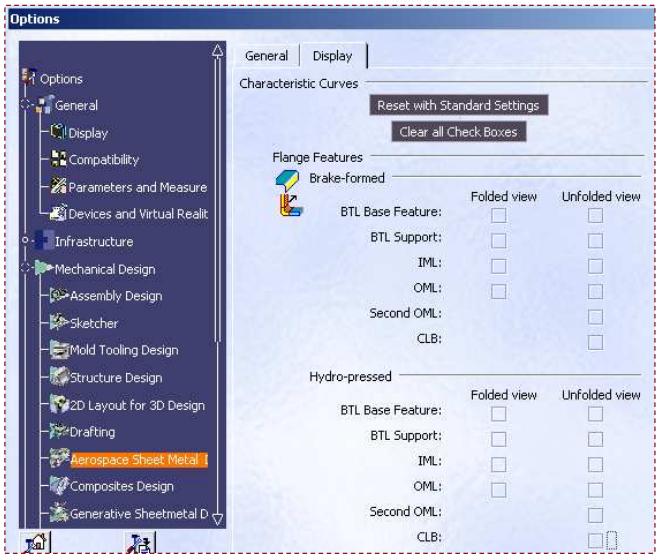
Visible curves

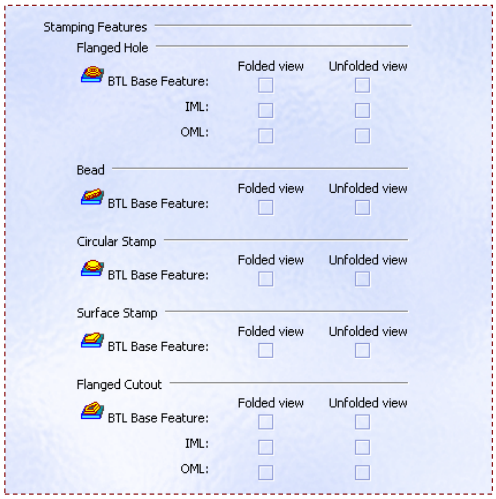


Instructor Notes:

Setting Parameters for a Designer

- The designer can also Manage the visibility of all characteristic curves (BTL, IML, OML and 2nd OML) in folded and unfolded 2D views through standards. in Tools/Options/Mechanical Design/Aerospace Sheet Metal Design with the display tab)

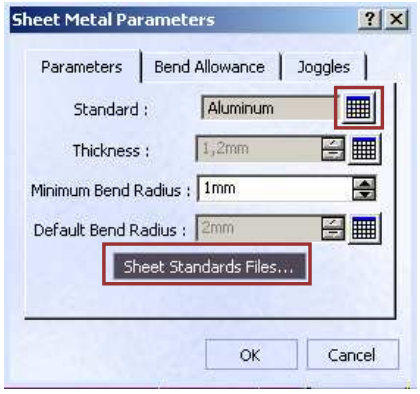


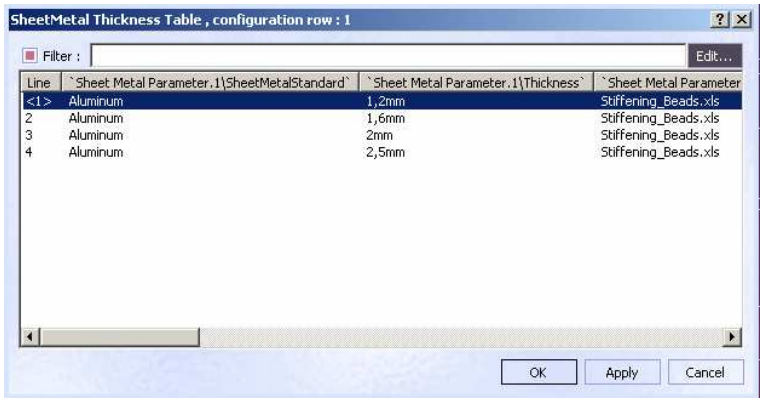


Instructor Notes:

Customizing Standards Files to define Design Tables

- In the Sheet metal Parameters Window, you can choose the sheet standard files and then select the values you want in the given design table.
- In the example below, you can see that the thickness and default bend radius are driven by design tables, hence they are grayed out (i.e. you can't modify the values)

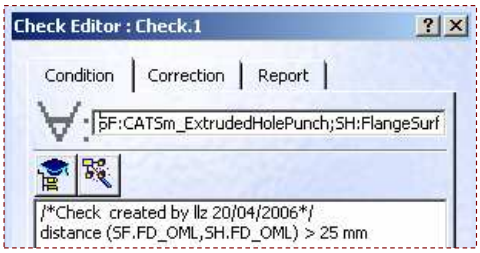




Instructor Notes:

How to use Knowledge Expert

- In order to perform a clearance check, you can use the characteristic curves like IML and OML in Check formulas, in the Knowledge Expert Workbench



- Thus you can see if the security clearance is verified or not

Instructor Notes:

To Sum Up

In this course you have learned how to:

- **Manage sheet metal parameters**
- **Create and modify the design of an Hydro formed Sheet Metal Part by defining its internal features:**
  - **Web**
  - **Surfacic Flanges**
  - **Joggles**
  - **Different kinds of flanges**
  - **Corner Relieves**
  - **Cutouts**
  - **Different kinds of Stamps**
  - **Holes**
  - **Points and Curves Mapping**
  - **Corners and Chamfers**
  - **Patterns**
- **Generate a flattened part**
- **Draw a flattened part**
- **Fulfill some administration tasks**
- **Create a Knowledge Expert Check using characteristic curves**

Instructor Notes:

Additional Exercises

You will perform the following additional exercises to reinforce the knowledge gained in this course:

- Additional Exercise: Aerostructure
- Additional Exercise: Bracket
- Additional Exercise: Fighter Web Structure
- Additional Exercise: Rib
- Additional Exercise: Fairing Linking

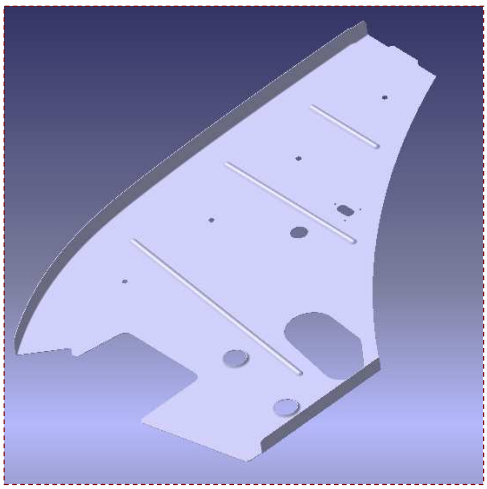
Instructor Notes:

Additional Exercise

Aerostructure

30 min

Design of a Sheet metal Aerostructure to demonstrate ASL functionalities



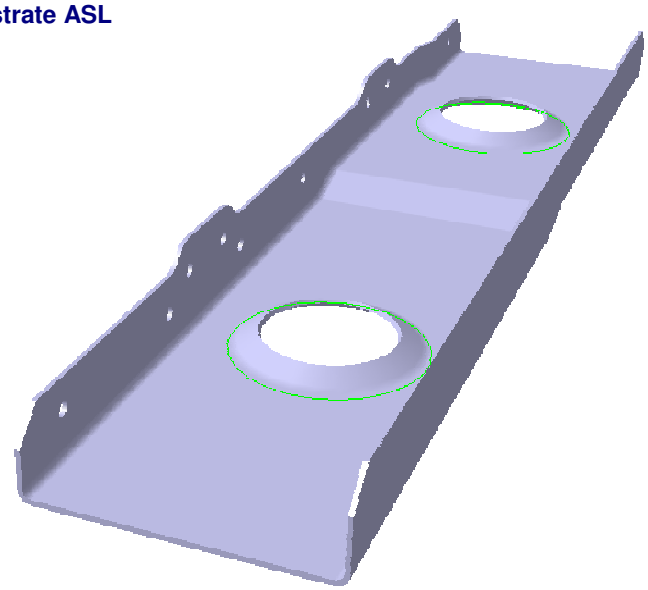
Instructor Notes:

Additional Exercise

Bracket

15 min

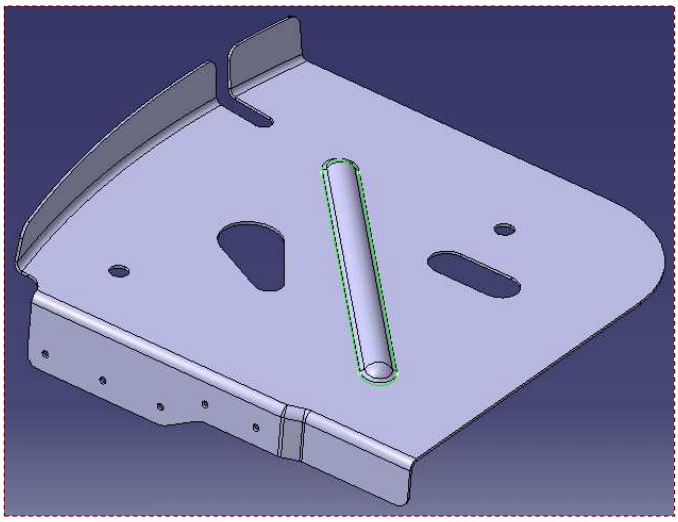
Design of a Sheet metal Bracket to demonstrate ASL functionalities.



Instructor Notes:

Additional Exercise

Fighter Web Structure

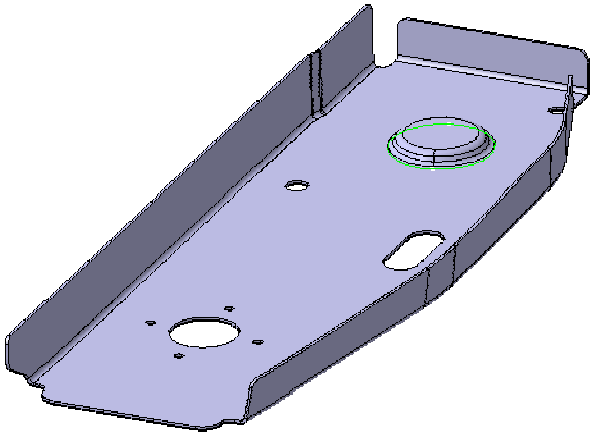


Instructor Notes:

Additional Exercise

Rib

20 min

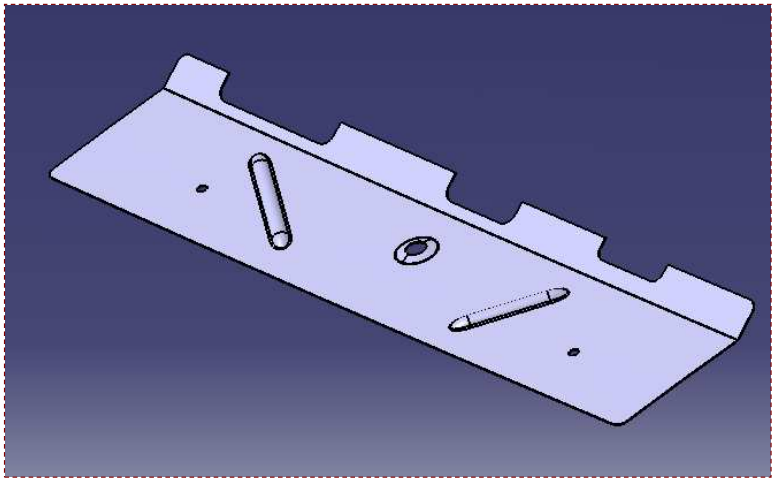


Instructor Notes:

Additional Exercise

Fairing Linking

20 min



Instructor Notes: