 [Translated from French to English - www.onlinedoctranslator.com](https://www.onlinedoctranslator.com/en/?utm_source=onlinedoctranslator&utm_medium=doc&utm_campaign=attribution)

###### **Entrance to the workshop**

This task shows how to enter the Generative Shape Design workbench and open a wireframe part.

1. From the Start menu, select Shape -> Generative Shape Design.

The Generative Shape Design workbench opens.

2. Select File -> Open, then the document: GettingStartedShapeDesign.CATPart. in the documentation then French --> One line ->Sdgug then Samples

A wireframe part appears.

In the rest of the scenario, you will use the construction elements ofthis piece to design the following geometric representation.

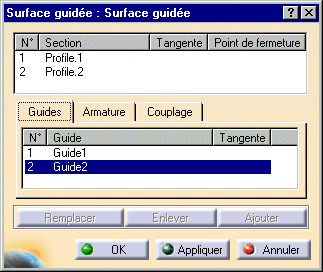
###### 

###### **Guidance and Offset**

This task shows how to create a guided surface and an offset surface.

1. Click the Guided Surface icon. 

The guide surface definition dialog box appears.



2. Select the two section curves.

3. Select the two guide curves.

4. Click OK to create the guided surface.

5. Click on the Shift icon: 

#### 6. Select the guided surface.

#### 7. Enter an offset value of 2mm.

The offset surface appears in the plane normal to the guided surface.

#### 8.Click OK to create the offset surface.

###### **Cutting, guiding and creating a fillet**

This task shows how to cut surfaces and then create a guided surface and two fillets.

#### 

#### 1. Click on the Intersection icon

#### 2. Select the offset surface and the first plane to create the intersection between these two elements.

3. Click on the Clipping icon

The Section Definition dialog box appears.

#### 4. Select the offset surface by clicking on the part to keep after cutting.

#### 5. Select the foreground as the cutting element.

#### 6. Click OK to cut the surface.

#### 7. Repeat the previous operation by selecting the guided surface then the second plane to first define the intersection, then to cut the surface.

#### 8. Click OK to cut the surface.

#### 9. Click the Guided Surface icon

The guide surface definition dialog box appears.

#### 10. Select the intersection edges of the two surfaces resulting from the splitting as sections.

#### 11. Click OK to create the guided surface between the two cut surfaces.

#### 12.Click on the Fillet icon

#### The dialog boxFillet is displayed.

#### 13. Select the first cut surface as the first support element.

#### 14. Select the guided surface as the second support element.

#### 15. Enter a fillet radius value of 3mm.

#### Arrows indicate the orientation of the surfaces.

#### 16. Ensure the correct orientation of the surfaces, then click OK to create the first fillet surface.

#### 17. Select the second cut surface as the first support element.

#### 18. Select the guided surface as the second support element.

#### 19. Enter a fillet radius value of 3 mm.

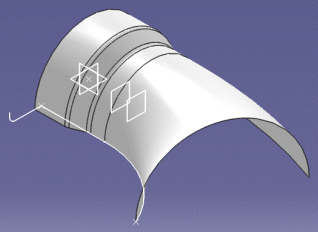
#### 20. Ensure the correct orientation of the surfaces, then click OK to create the second fillet surface

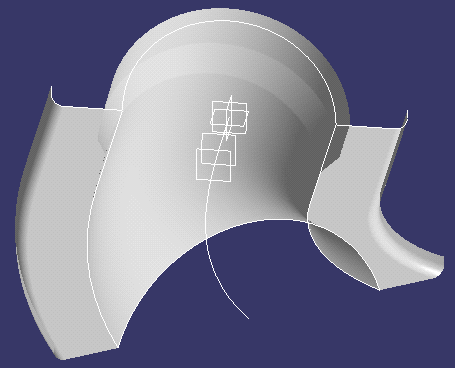
###### **Sweeping and creating fillets**

This task shows how to create sweep surfaces and fillets on both sides of the part.

To do this, you will use the profile element on the side of the part.

You will also create a symmetrical profile element on the opposite side of the part.





#### 1. Click the Scan icon.

#### The dialog boxScan Surface is displayed.

#### 2. Click the scan button: Explicit.

#### 3. Select the profile element (Corner.1).

#### 4. Select the guide curve (Guide.1).

#### 5. Select the central curve as the spine (Spline.1).

#### 6. Click OK to create the sweep surface.

#### 7. Click the Symmetry icon

#### The dialog boxSymmetry Definition is displayed.

#### 8. Select the profile element to be mirrored.

#### 9. Select the YZ plane as the reference element.

#### 10. Click OK to create the symmetrical profile element.

#### 11. Click the Scan icon again.

#### 12. Select the profile (Symmetry.3) and the guide curve (Guide.2).

#### 13. Select the central curve as the spine.

#### 14. Click OK to create the sweep surface.

#### 15. To create a fillet between the side part and the central part, click on the Fillet icon.

#### 16. Select the side sweep element and the central part of the part and enter a fillet radius value of 1 mm.

#### 17. Click Apply to preview the fillet, then click OK to create it.

#### 18. Select the other sweep element and the central part of the part, then enter a fillet radius value of 1 mm.

#### 19. Click OK to create the fillet.

###### **Using the History Graph**

This task shows how to use the history graph.

#### 1. Select the item for which you want to display the historical graph.

#### 2. Click on the graph display icon

#### The History Graph dialog box appears.

#### In this case, you can browse the history of events related to the construction of the Guided Surface element.1. Each branch of the graph can be expanded or collapsed depending on the desired level of precision.

#### The available icons are as follows.

#### Addition to the graph;

#### Removal from the graph;

#### Center all;

#### Presentation SKIN / Presentation PART;

#### Settings ;

#### Constraints.

#### Just click on the icon

#### 3. Close to exit this mode.

###### **Room transformation**

#### This task demonstrates how to modify the part using an affinity operation.

#### 1.Click on the Affinity icon

#### The Affinity Definition dialog box appears.

#### 2.Select the end section profile to be transformed by the affinity.

#### 3. Indicate the characteristics of the axis system used for the affinity operation:

#### PT0 point at origin;

#### XY plane as reference plane;

#### horizontal edge of corner profile as X axis.

#### 4.Indicate the affinity ratios: X=1, Y=1 and Z=1.5.

#### 5. Click OK to create the new profile.

#### 6.Edit the lofted surface definition to replace the section profile with the new profile.

#### 7.If necessary, click the Update icon to update your creation.

#### 

Surface creation

CATIA Generative Shape Design lets you model both simple and complex surfaces using techniques such as guiding, sweeping, and filling.

Create Surfaces by Extrusion: Select a profile, indicate the direction of the extrusion and define the two extremities of the extrusion.

Creating Revolved Surfaces: Select a profile, a rotation axis and define the angle limits of the surface of revolution.

Creating Offset Surfaces: Select the surface to offset, indicate the value and the direction of the offset.

Sweep surface creation: Select one or more guide curves, the profile to be scanned, and optionally, a spine, a reference surface, and start and end values.

Creating fill surfaces: Select curves or edges of the surface forming a closed boundary and indicate the type of continuity

Creating Guided Surfaces: Select at least two planar section curves, and optionally guide curves and a spine, and specify the tangency conditions.

Creating Blend Surfaces:: Select two curves and, possibly their support, indicate the voltage, the continuity, the closing point and the coupling ratio, if necessary.

Geometry extraction: Select an edge or the face of a geometric element.

**Performing Operations on Geometric Shapes**

CATIA Generative Shape Design lets you modify your design using techniques such as trimming, extrapolation, and creating fillets.

 Joining geometries: Select at least two curves or surfaces to join.

 Adjusting geometries: Select at least two surfaces with a

space to adjust.

 Geometry clipping: Select the element to cut and an element of

cutting.

 Geometry trim: Select two elements to be trimmed and indicate the side

concerning.

 Creating fillets: Select two support surfaces, enter a radius value, specify fillet type and location.

 Creating Edge Fillets: Select an internal edge on a surface, the

surface itself, set the fillet type and propagation mode and enter

a radius value.

 Creation of variable holidays: Select an edge to transform into a fillet,

specify fillet end type, propagation mode, pick a point

on the edge where the radius will be variable and enter the radius value at that point.

 Create face-to-face fillets: Select a support surface, the two sides to

transform to county, specify the mode of trimming and enter a radius value.

 Creating tritangent fillets: Select a support surface, indicate the mode

of relimitation, the two faces to be transformed into a story and the one to be deleted.

 Geometry Translation: Select an element, a translation direction

#### (line, plane or vector) and indicate the distance of the translation.

 Achieving a symmetry on a geometry: Select an element, then a point, a line or a plane as a reference element.

 Transformation of a geometry by scale factor: Select an element, then a point, a plane or a flat surface as a reference element, and specify the ratio of the scale factor.

 Transformation of geometry by affinity: Select an element to transform,

specify the axis system characteristics and enter the affinity ratio values.

 Rotating geometry: Select an element, a line as axis of rotation and indicate the angle of rotation.

 Extrapolation of a surface: Select a boundary curve, then the surface

itself, indicate the extrapolation limit (value or limit surface/plane) and indicate

the end constraints (tangent/normal).

 Extrapolation of a curve: Select the end point of a curve, the curve

itself, indicate the limit of the extrapolation (length or limit surface/plane) and indicate the continuity constraints (in tangency/in curvature).