Fundamentals: Generative Shape Design
Master up to high complexity shapes
Course Objectives

This course provides CATIA Generative Shape Design knowledge in a V6 environment. The goal is to give you an overview of different tools of CATIA Generative Shape Design workbench.

Upon completion of this course you will be able to:

- Create simple reference and Wireframe geometry
- Use the reference wireframe elements to create simple surfaces
- Create clean topology from a set of surfaces and smooth sharp edges
- Detect and correct the discontinuities on curves and surfaces
- Create solids from surfaces

Targeted audience

Students pursuing education in engineering and design.

Prerequisites

CATIA V6 Fundamentals
## Table of Contents

### Introduction to Generative Shape Design
- Introduction to surface design ........................................ 6
- The Generative Shape Design Workbench ......................... 7
- Why create 3D Wireframe geometry? .............................. 9
- Surface Design Workbench User Interface ....................... 10
- Surface Design Workbench Terminology ....................... 11
- Surface Design Workbench General Process ................... 12

### Creating Wireframe Geometry
- Concept of Curve Continuity ......................................... 14
- The tension Concept ................................................... 15
- Creating Points in 3D ................................................ 16
- Creating Lines in 3D .................................................. 17
- Creating Planes in 3D ................................................ 18
- Creating Curves in 3D ................................................ 19
# Table of Contents

- **Creating Basic Surfaces**  
  - What about Basic Surfaces? 21  
  - Surfaces tools 22  
  - Extrude and revolute surfaces 23  
  - Surface of Revolution and Cylindrical 24  
  - Creating a Surface Offset from a Reference 25  
  - Creating a Swept Surface 26  
  - Creating a Fill Surface 27  
  - Creating a Multi-Section Surface 28

- **Performing Operations on the Geometry**  
  - Why are Operation on Geometry needed? 30  
  - Joining/ Splitting/ Trimming Elements 32  
  - Difference between a Split and a Trim 35  
  - Fillet Tools 36  
  - Extrapolating Elements 37  
  - Introduction to Connect Checker tool 39  
  - Methodology 40

- **Conclusion** 42
Here are the steps to be followed:

1. Introduction to surface design
2. The Generative Shape Design Workbench
3. Why create 3D Wireframe geometry?
4. User interface
5. Workbench Terminology
6. Workbench General Process
**Introduction to Surface Design**

- The creation of wireframe and surface geometry is often needed to define the complex shapes of parts. Ultimately, we want to create a solid to best capture our design intent, but this model may include surface geometry integrated into the solid part.

- Below are listed some key points of methodology:
  - Wireframe and surface geometry are used to define more complex 3D shapes in the design process.
  - Wireframe, surface and solid geometry form an integrated set of modeling capabilities that allow us to capture the design intent.
The Generative Shape Design Workbench

- Generative Shape Design workbench has a wide functional set. It is a complete surfacing tool used to create complex shape parts.
  - With Generative Shape Design workbench, designers can easily design plastic parts’ surfaces or shells.
The Generative Shape Design Workbench

- CATIA - GSD provides a comprehensive set of features for shape design, including wireframe elements like:
  - Wireframe simple (Point, Line, Plane, Curves, Circle).
  - Wireframe advanced (Parallel curves, Combine, Intersection and Projection).

- Standard and advanced surface features including:
  - Basic surface (Extrude, Revolute, Sweep and Fill).
  - Operations, such as Symmetry, Scaling, Translation, Affinity, Extrapolation and Fillet.
  - Analysis (Connect checker analysis, Porcupine curvature analysis, Surfacic curvature analysis).
  - Replication.
Why create 3D Wireframe geometry?

- In many design situations, there is a need to create geometry that is defined using the entire 3D space. This geometry is not limited to a single plane and therefore cannot be defined using the Sketcher workbench.

- These elements including points, lines, planes and curves created in 3D space are called “Wireframe geometry”.
  - Wireframe geometry is primarily used as construction geometry for creating more complex 3D elements such as curves and surfaces.
  - Wireframe geometry and Sketch geometry can be used together to define more complex 3D elements.
  - Even if Wireframe geometry is created in 3D, a support element (plane or surface) may be required to define the geometry.
Surface Design Workbench User Interface

1. Specification tree

2. Containers of type Geometric Set, Ordered Geometric Set and Body

3. Analysis

4. GSD Tools
Surface Design Workbench Terminology

- **Part Reference** is a combination of "PartBody" and "Geometrical Sets".
- **3D Representation** contains the features used to create a solid. It can contain surfacic and wireframe elements too.
- **PartBody** contains solid features.
- **Geometrical Sets** contains the features used to create surface and wireframe elements.
- **Ordered Geometric Sets (OGS)** contains surface and wireframe. The elements in this body are created in a linear manner. OGS can also contain "Body". Body allows creation of Part Design Solids within an OGS.

If you create Reference Elements (points, planes, lines) in Part Design Workbench, you have the option of directly containing them in Part body / Body, or you can insert a Geometric set and place these elements.
Surface Design Workbench General Process

1. Create the wireframe geometry
2. Create the surfaces
3. Trim and join the body surfaces
4. Create the part body
5. Modify the geometry
Creating Wireframe Geometry

In this step, you will learn what are curves and wireframes and how to create them.

Here are the steps to be followed:

1. Concept of Curve Continuity
2. The tension Concept
3. Creating Points in 3D
4. Creating Lines in 3D
5. Creating Planes in 3D
6. Creating Curves in 3D
A Curve is said to be Continuous when the vertices of two curves join to form a single curve.

The Continuities are of three types:

A. **Point Continuity**: If the distance between two vertices of the connecting curve is within (less than) specified CATIA V6 tolerance, then the curves are said to be *Point Continues*.

B. **Tangent Continuity**: If the angle between two normal curves at the connecting points is equal to zero or 180deg, then the curves are said to be *Tangent Continuities*.

C. **Curvature Continuity**: It is the ratio of the change in the angle of a tangent that moves over a given arc to the length of the arc.
The Tension Concept

In Generative Shape Design (GSD), you can apply different tension values to curves to achieve the required shape of the curve.

Let’s see the impact of tension on a curve with the example of a Connect curve:

A. The tension value on Curve 2 is kept constant at default value (T=1), and the tension value on Curve 1 varies.

B. The tension value on Curve 1 is kept constant at default value (T=1), and the tension value on Curve 2 varies.

Observations:

- When the tension value increases from T1 to T4, the connect curve (constrained with tangent continuity) changes its curvature. You can apply different tension values at each end to attain the desired shape of the curve.
- The curve still maintains the continuity constraints with its parent curve at different tension values.
Creating Points in 3D

1- ‘On curve’
   Curve = Spline.1
   Ratio = 0.5
   Geodesic

2- ‘On surface’
   Surface = Extrude.1
   Direction = zx plane
   Distance = 50mm

3- ‘Between’
   Point1 = Point.4
   Point2 = Point.5
   Ratio = 0.6

4- ‘Point repetition’
   Curve = Spline.1
   Instances = 5
   With end points
   Create in a new Body
Creating Lines in 3D

1- ‘Point-Point’
Point1 = line extremity
Point2 = surface vertex

2- ‘Point-Direction’
Point = line extremity
Direction = zx plane
Start-End = -30mm,0mm
Reverse Direction if needed

3- ‘Angle to curve’
Curve = surface edge
Support = zx plane
Point = surface vertex
Angle = 45 deg
Start= - 30mm
End = yz plane (UpTo)

4- ‘Normal to Surface’
Surface=Extrude.1
Point = surface vertex
Start-End = -30mm,0mm
Creating Planes in 3D

1- Creating an Offset Plane
   - Select the reference element (plane, face, etc…).
   - Define the offset value, either in the Offset field or using the graphic manipulators.

2- Creating a Plane through Three Points
   - Select three points
   - The plane passing through the three points is displayed.

3- Creating a Plane through a Point and a Line
   - Select a point and a line.
   - The plane passing through the point and the line is displayed.

4- Creating a Plane Normal to a Curve
   - Select a reference curve and a point.
   - A plane is displayed normal to the curve at the specified point.
Creating Curves in 3D

1- Creating a Projection
   - Select the projected element (line, spline, etc…) and select the support (surface).

2- Creating a Combine Curve
   - Select the two curves (line, spline, etc…) .

3- Creating a Parallel Curve
   - Select the curve (line, spline, etc…) and select the support (surface).
Creating Basic Surfaces

In this step, you will be introduced to the different types of Surface Creation tools available in Generative Shape Design.

Here are the steps to be followed:

1. What about Basic Surfaces?
2. Surfaces tools
3. Extrude and revolute surfaces
4. Surface of Revolution and Cylindrical
5. Creating a Surface Offset from a Reference
6. Creating a Swept Surface
7. Creating a Fill Surface
8. Creating a Muti-Section Surface
What about Basic Surfaces?

For certain designs, the geometry can not be completely defined using the tools in the Part Design workbench. Complex 3D shapes often need to be defined using surface geometry based on explicit wireframe construction geometry.

Surface geometry can then be integrated into the final solid part definition. If the industrial context does not require a solid, surfaces are kept as they are.

- Surface geometry can describe a more complex 3D shape.
- A surface element describes a shape, therefore it has no thickness.
- Surface geometry can be completely integrated into the solid part, meaning that surface modifications are reflected in the solid.
## Surface tools

<table>
<thead>
<tr>
<th>Category</th>
<th>Name</th>
<th>Icon</th>
<th>Description</th>
<th>Illustration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surfaces extruded in a direction</td>
<td>Extrude</td>
<td><img src="image" alt="Extrude Icon" /></td>
<td>Extrudes a user-defined profile in a specified direction.</td>
<td><img src="image" alt="Extrude Illustration" /></td>
</tr>
<tr>
<td></td>
<td>Cylinder</td>
<td><img src="image" alt="Cylinder Icon" /></td>
<td>Extrudes an implicitly circular profile in a specified direction.</td>
<td><img src="image" alt="Cylinder Illustration" /></td>
</tr>
<tr>
<td>Surfaces revolved around an axis</td>
<td>Revolve</td>
<td><img src="image" alt="Revolve Icon" /></td>
<td>Revolves a user-defined profile around an axis.</td>
<td><img src="image" alt="Revolve Illustration" /></td>
</tr>
<tr>
<td></td>
<td>Sphere</td>
<td><img src="image" alt="Sphere Icon" /></td>
<td>It works as a circular profile revolving around an axis implicitly. It is a Full or partial spherical surface.</td>
<td><img src="image" alt="Sphere Illustration" /></td>
</tr>
<tr>
<td>Surfaces connecting existing sections</td>
<td>Multi-sections</td>
<td><img src="image" alt="Multi-sections Icon" /></td>
<td>Surface passing through multiple sections.</td>
<td><img src="image" alt="Multi-sections Illustration" /></td>
</tr>
<tr>
<td>Surfaces sweeping a profile along a guide curve</td>
<td>Sweep</td>
<td><img src="image" alt="Sweep Icon" /></td>
<td>Sweeps a profile along a path (the profile is pre-defined or user-defined).</td>
<td><img src="image" alt="Sweep Illustration" /></td>
</tr>
<tr>
<td></td>
<td>Adaptive sweep</td>
<td><img src="image" alt="Adaptive sweep Icon" /></td>
<td>Sweeps a parametric profile along a path, allowing the parameters to evolve along the path.</td>
<td><img src="image" alt="Adaptive sweep Illustration" /></td>
</tr>
<tr>
<td>Surfaces filling a gap</td>
<td>Fill</td>
<td><img src="image" alt="Fill Icon" /></td>
<td>Creates a surface inside a closed boundary</td>
<td><img src="image" alt="Fill Illustration" /></td>
</tr>
<tr>
<td>Surfaces offset from an existing surface</td>
<td>Offset</td>
<td><img src="image" alt="Offset Icon" /></td>
<td>Creates a surface offset from an existing surface</td>
<td><img src="image" alt="Offset Illustration" /></td>
</tr>
</tbody>
</table>
The extruded surface is created from an open or closed profile, giving a direction and limits.

A surface of revolution is created from an open or closed profile, giving an axis of revolution and an angle.
Surface of Revolution and Cylindrical

Spherical surface is created by defining the centre point, radius and an angle (in case a partial sphere is required).

Cylindrical surface is created by defining a point, direction and length of the cylinder.
Creating a Surface Offset from a Reference

If you want to create several surfaces separated by the same offset, check the option Repeat object after OK and click OK.
Creating a Swept Surface
Creating a Fill Surface

1. Select the boundaries of the fill surface and, if needed, the associated support(s) with one or more boundaries.
Creating a Multi-Section Surface
Performing Operations on the Geometry

In this step, you will be introduced with the different types of Operation tools available in Generative Shape Design.

Here are the steps to be followed:

1. Why are Operation on Geometry needed?
2. Joining/ Splitting/ Trimming Elements
3. Difference between a Split and a Trim
4. Fillet Tools
5. Extrapolating Elements
6. Checker Tool
7. Methodology
Why are Operations on Geometry needed?

- After the basic surface geometry is created, it may be composed of construction elements that do not describe the finished shape. Operations such as trim, join, extrapolate, and transform are then performed to produce the required finished geometry.

- Operations are used to produce the finished geometry shape.

- Elements involved in an operation are kept in the history of the operation but hidden.

- Healing is an important capability that can be used to repair the gaps that exist in surface geometry.

Surface fillet operation  Healing Operation
Why are Operations on Geometry needed?

Transformations like scaling and affinity help resizing up the part if required. Transformation operations like translate and rotate are required on the wireframe elements (lines and planes) to change the positioning of the part in the co-ordinate axis system.

- Affinity is an important operation to resize the part differently in different directions, according to a defined axis.
- Axis to Axis transformation is useful when we want to have more than one reference axis systems, and part elements are required to be moved from one axis to another.
Joining Elements

The **Join** tool is used to assemble wireframe and surfaces.

- Two adjacent splines
- Four adjacent surfaces
The **Split** tool is used to remove unwanted portions of wireframe and surface elements.
Trimming Elements

The **Trim** tool is used to trim two intersecting elements and keep only a part of those elements.
Difference between a Split and a Trim

Splitting geometry is breaking all the geometries at the intersection with the cutting element and then removing the unwanted portion. During splitting operation cutting element does not get affected.

Trimming geometry is cutting all the geometries with respect to one another to get the desired shape during trimming operation.
## Fillet Tools

The following table lists the different types of fillets in GSD workbench.

<table>
<thead>
<tr>
<th>Fillet</th>
<th>Icon</th>
<th>Description</th>
<th>Illustration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shape Fillet</td>
<td><img src="image" alt="Shape Fillet Icon" /></td>
<td>The Shape Fillet tool creates a smooth connection surface between two separated surfaces</td>
<td><img src="image" alt="Shape Fillet Illustration" /></td>
</tr>
<tr>
<td>Edge Fillet</td>
<td><img src="image" alt="Edge Fillet Icon" /></td>
<td>The Edge Fillet tool creates a transitional surface along a sharp edge of a surface</td>
<td><img src="image" alt="Edge Fillet Illustration" /></td>
</tr>
<tr>
<td>Variable Fillet</td>
<td><img src="image" alt="Variable Fillet Icon" /></td>
<td>The Variable Radius Fillet tool creates a fillet on a selected edge whose radius varies at a selected point.</td>
<td><img src="image" alt="Variable Fillet Illustration" /></td>
</tr>
<tr>
<td>Chordal Fillet</td>
<td><img src="image" alt="Chordal Fillet Icon" /></td>
<td>The Chordal Fillet tool creates a fillet on selected edges taking the chord length as the input.</td>
<td><img src="image" alt="Chordal Fillet Illustration" /></td>
</tr>
<tr>
<td>Face-Face Fillet</td>
<td><img src="image" alt="Face-Face Fillet Icon" /></td>
<td>The Face-Face fillet is used when there is no intersection between the selected faces or when there are more than two sharp edges between the faces.</td>
<td><img src="image" alt="Face-Face Fillet Illustration" /></td>
</tr>
<tr>
<td>Tritangent Fillet</td>
<td><img src="image" alt="Tritangent Fillet Icon" /></td>
<td>A Tritangent fillet creates a transitional surface by removing one of the three selected surfaces. The fillet surface is created tangent to the three selected faces.</td>
<td><img src="image" alt="Tritangent Fillet Illustration" /></td>
</tr>
</tbody>
</table>
Extrapolating Elements

The **Extrapolate** tool is used to extend a surface or a curve. It is often used to extend an element and not another so that these elements can later be trimmed, split, or intersected together.
Extrapolating Elements

- Extrapolating Wireframe
  - Curve to extrapolate
  - Boundary
  - Up to element
  - End model

- Extrapolating Wireframe
  - Curve to extrapolate
  - Boundary
  - Up to element
  - End model
## Introduction to Connect Checker tool

<table>
<thead>
<tr>
<th>Analysis Type</th>
<th>Description</th>
<th>Quick Analysis</th>
<th>Interpretation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Distance (G0)</td>
<td>The distance between the vertices of the surfaces is measured. If the distance is less than 1 micron, CATIA surfaces are considered continuous in Point.</td>
<td><img src="image1.png" alt="Image" /></td>
<td>The areas marked in RED highlight the gaps between 0.5 and Max value</td>
</tr>
<tr>
<td>Tangency (G1)</td>
<td>The angle between the surfaces at the connection is measured. If the angle is less than 0.5 deg, CATIA surfaces are considered continuous in Tangency.</td>
<td><img src="image2.png" alt="Image" /></td>
<td>The area marked in Yellow highlights the tangency discontinuities between 0.5 deg and Max value</td>
</tr>
<tr>
<td>Curvature (G2)</td>
<td>The Curvature (1/R) difference at the connecting edges of the surfaces is measured. The curvature difference is measured using a formula and is expressed in percentage. The discontinuity range is between 0 – 200 %. Lesser the percentage, better the surface connection.</td>
<td><img src="image3.png" alt="Image" /></td>
<td>The areas marked in Blue highlight the curvature discontinuities of 200%</td>
</tr>
<tr>
<td>Overlap of surfaces</td>
<td>Checks overlap between surfaces.</td>
<td><img src="image4.png" alt="Image" /></td>
<td>An information saying 2 overlaps is detected is displayed.</td>
</tr>
<tr>
<td>Internal Edges</td>
<td>Analysis can be done for the surfaces having internal edges (Two joined surfaces)</td>
<td><img src="image5.png" alt="Image" /></td>
<td>Analysis at the common edge in a single surface is displayed</td>
</tr>
</tbody>
</table>
Methodology

As you create complex geometries, it becomes important to manage your model. A good structured model having logical groups of geometries gives a better understanding of the design process.

An organized model has the following advantages:

A. All related geometries are clubbed together in groups or sets (like files stored in a folder on your computer). This helps the designers to easily interpret the steps used to design the model.

B. The size of the specification tree is reduced, what makes it easier to organize when the tree becomes too complex.

C. It is easier to reorder and replace features.

D. Problem solving becomes easier as the root cause of the defect can be easily identified.

The following tools are available to organize the model:
1. Geometrical Sets
2. Ordered Geometrical Sets
Geometrical Set (GS) is the default container for wireframe and surface elements. In GS, the features are not displayed according to the logical update order. It just contains the features.

- You can put any surfacic element in the GSG. The elements don’t need to be structured in a logical way. The order of these elements is not important as their access and visualization is managed independently without any rule.

- GS enables you to gather various features in a single set of different sub-sets and organize the specification tree.

- Multiple Geometrical Sets can be added to a model to contain a genre of surfacic elements.

- For example, one GS can be dedicated to contain only wireframes while the other can contain surfaces.
Generative Shape Design (GSD) plays an important role in the process of Forward and Reverse Engineering in a product design cycle.