

**TABLE OF CONTENTS**

Introduction ..... 1

    Wireframe & Surfaces ..... 2

    Pull Down Menus ..... 3

        Edit ..... 3

        Insert ..... 4

        Tools ..... 6

    Generative Shape Design Workbench ..... 7

    Bottom Toolbar ..... 9

        Tools ..... 9

        Analysis ..... 10

    Volumes ..... 11

    Advanced Surfaces ..... 12

    Developed Shapes ..... 12

    BiW Templates ..... 13

Wireframe ..... 15

    Points ..... 15

        Coordinate ..... 15

        On curve ..... 18

        On plane ..... 22

        On surface ..... 23

        Circle / Sphere / Ellipse center ..... 25

        Tangent on curve ..... 26

        Between ..... 29

        Point Repetition ..... 31

        Projecting points ..... 35

        Intersection points ..... 38

        Extremum ..... 41

        Polar Extremum ..... 44

    Lines ..... 46

        Point-Point ..... 46

        Point-Direction ..... 49

        Angle/Normal to curve ..... 50

        Tangent to curve ..... 55

        Normal to surface ..... 58

        Bisecting ..... 60

        Intersection lines ..... 62

        Projecting lines ..... 63

        Axis ..... 64

        Polyline ..... 66

Planes	70
Offset from plane	70
Parallel through point	72
Angle/Normal to plane	73
Through three points	75
Through two lines	76
Through point and line	77
Through planar curve	78
Normal to curve	79
Tangent to surface	80
Equation	81
Mean through points	82
Plane Between	83
Circles	85
Center and radius	85
Center and point	87
Two points and radius	88
Three points	90
Bitangent and radius	92
Bitangent and point	93
Tritangent	94
Center and tangent	95
Corners	96
Curves	100
Connect Curves	100
Conics	105
Splines	111
Helixes	117
Spirals	123
Project curves	126
Combine curves	129
Reflect Line curves	133
Intersection curves	135
Parallel Curves	139
3D Curve Offset	145
Curve comparison	148
Supports	151
Work on Support	151
Creation on the fly	156
Modifying	159
Datums	161
Object repetition	163

Surfaces	165
Extruded	165
Revolution	167
Sphere	169
Cylinder	171
Offset	173
Variable Offset	176
Rough Offset	178
Offset with Multiple Sub-elements	179
Sweep	182
Explicit	182
With reference surface	182
With two guide curves	190
With pulling direction	194
Linear	196
Two limits	196
Limit and middle	200
With reference surface	201
With reference curve	203
With tangency surface	205
With draft direction	207
With two tangency surfaces	211
Circular	213
Three guides	213
Two guides and radius	215
Center and two angles	217
Center and radius	219
Two guides and tangency surface	221
One guide and tangency surface	223
Limit curve and tangency surface	225
Conical	227
Two guide curves	227
Three guide curves	229
Four guide curves	231
Five guide curves	233
Adaptive Sweep	235
Fill surfaces	241
Multi-section surfaces	245
Blend surfaces	255
Spines	261
Isoparametric Curves	266
Laws	268

Operations	279
Joining Elements	279
Healing Surfaces	286
Curve smoothing	290
Splitting Elements	291
Trimming Elements	297
Untrimming Elements	300
Disassembling Elements	302
Extracting Boundaries and Faces	304
Multiple Extract	309
Creating the Nearest Element	312
Fillet	314
Shape Fillet	314
Edge Fillet	320
Variable Radius Fillet	322
Chordal Fillet	324
Face to Face Fillet	326
Tritangent Fillet	327
Styling Fillet	329
Transformations	334
Translate	334
Rotate	336
Symmetry	337
Scale	338
Affinity	339
Axis to Axis	342
Patterns	343
Rectangular	343
Circular	346
User Defined	347
Extrapolating Curves and Surfaces	348
Multi-Selection	353
Analysis	359
Connect Checker Analysis	359
Draft Analysis	369
Surfacic Curvature Analysis	373
Porcupine Curvature Analysis	377
Geometric Information	386
Dress-Up	388

Geometrical Set Management . . . . .	391
Inserting Sets . . . . .	391
Changing Sets . . . . .	392
Hiding and Showing Components . . . . .	394
Activating and Deactivating Components . . . . .	394
Reordering Components . . . . .	395
Groups . . . . .	397
Isolating Geometry . . . . .	398
Copying and Pasting Geometry . . . . .	399
Ordered Geometrical Set Management . . . . .	402
Inserting an Ordered Geometrical Set . . . . .	402
Modifying Children . . . . .	404
Reordering Components . . . . .	406
Scanning Ordered Sets . . . . .	408
Inserting in an Ordered Set . . . . .	409
Switching to a Regular Geometrical Set . . . . .	410
Miscellaneous . . . . .	411
Parents/Children . . . . .	411
Historical Graph . . . . .	412
Quick Select . . . . .	414
Inserting Elements . . . . .	416
Sets of planes . . . . .	417
Regular . . . . .	417
Irregular . . . . .	418
Semi-Regular . . . . .	419
Keep and No Keep Mode . . . . .	421
Keep Mode . . . . .	421
No Keep Mode . . . . .	421
Current Body . . . . .	425
Masks . . . . .	427
2D Visualization Modes . . . . .	431
Deleting Useless Elements . . . . .	435
Review . . . . .	437
Problems . . . . .	463
Problem #01 . . . . .	463
Problem #02 . . . . .	464
Problem #03 . . . . .	465
Problem #04 . . . . .	466
Problem #05 . . . . .	468
Problem #06 . . . . .	470
Problem #07 . . . . .	472

Appendix A	475
Shape - Generative Shape Design - General	475
Shape - Generative Shape Design - Work On Support	476
Appendix B	477
Part Design Using Surfaces	477
Split	477
Thick Surface	479
Close	480
Sew	481
Pad/Pocket	483
Boolean Operations	484
Appendix C	487
Generative Shape Optimizer	487
Bump Surfaces	488
Wrap Curve Surfaces	491
Wrap Surface Surfaces	493
Shape Morphing	495
Automatic Filleting	499
Appendix D	503
Developed Shapes	503
Unfold Surfaces	503
Transfer	507
Develop Wires	510
Appendix E	513
BiW Templates	513
Junction Surfaces	513
Diabolo	515
Hole	517
Hole Curve	521
Mating Flange	525
Bead	528
3D Working Supports	530
Appendix F	533
Surface Machining Tools	533
Appendix G	537
Volumes	537

## **Introduction**

### **CATIA Version 5 Wireframe & Surfaces**

Upon completion of this course, the student should have a full understanding of the following topics:

- Creating wireframe geometry
- Creating surfaces
- Performing operations on surfaces
- Modifying wireframe and surfaces
- Analyzing curves and surfaces
- Utilizing wireframe and surfaces in Part Design

## **Wireframe & Surfaces**

Many parts can be created using just the Part Design tools; however, there are times when surfaces need to be used in order to get the desired shape for your part. Wireframe geometry is also necessary at times to define support geometry for the various Part Design tools as well as the surface tools. Surfaces provide the ability to create complex contours that are often necessary in your design. There are a few workbenches in CATIA V5 that have wireframe and surface options. This class will focus on the Generative Shape Design workbench. The Generative Shape Design workbench has all of the tools that are available on the Wireframe & Surfaces workbench and more. This course will cover all of the options found in the Generative Shape Design workbench.

As covered in previous courses, surfaces can be used within Part Design. This gives the capability of hybrid modeling. To review, you should remember that you can perform four operations with surfaces in Part Design. One option is to add thickness to a surface thereby creating a solid. A second option is to split your part with a surface. A third option is to sew a surface into your part, which will either add or remove material, or both. The last option is to close a surface with planar faces to form a solid. These options should be reviewed and there are review exercises located in Appendix B. It is also important that you understand how to work with the boolean operations in order to fully utilize all of the surface options. These are reviewed in Appendix B as well.

It is important to understand some of the terminology that CATIA uses when working with wireframe and surfaces. You should already be familiar with a PartBody and know that you can have more than one within your part. Wireframe geometry and surfaces are created within geometrical sets. You may also have more than one geometrical set in your part. Geometrical sets are used to organize your non-solid geometry. When you create new wireframe or surface geometry, you will need to be sure that the correct geometrical set is current in order to have an organized tree.



## Wireframe

Wireframe geometry is critical to the creation of surfaces and is used as reference geometry throughout CATIA.

## Points

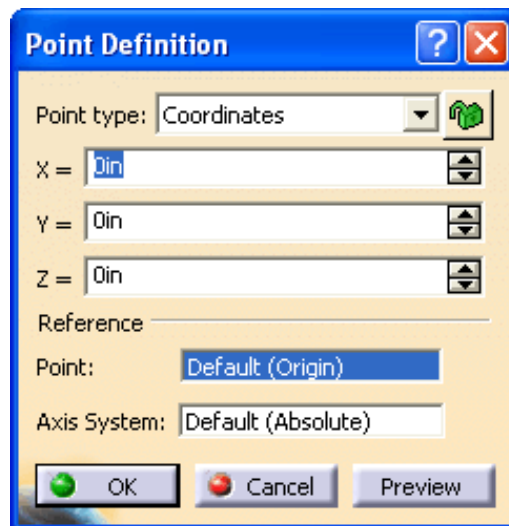
Points are useful to define specific locations and to assist in the creation of other wireframe geometry. You have a variety of options to define points. This exercise will explore those options.

## Coordinate

**Open the Points document.** You should see a surface and some wireframe geometry.

**Change to the Generative Shape Design workbench.** If you are already in the workbench then you will not need to change. If not, to change workbenches you can select pull down menu *Start, Shape, Generative Shape Design*.

**Select the Point icon.**  A *Point Definition* window appears.



*Point type* Specifies what type of point you want to create, either *Coordinates*, *On curve*, *On plane*, *On surface*, *Circle/Sphere/Ellipse center*, *Tangent on curve* or *Between*

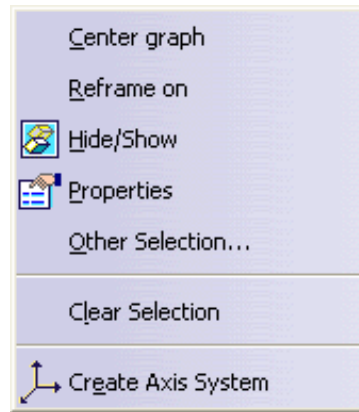
*X=, Y=, Z=* The coordinate values of the point that you want to create from the reference point

*Reference*

*Point* The point that the coordinates are based from. The default is the origin.

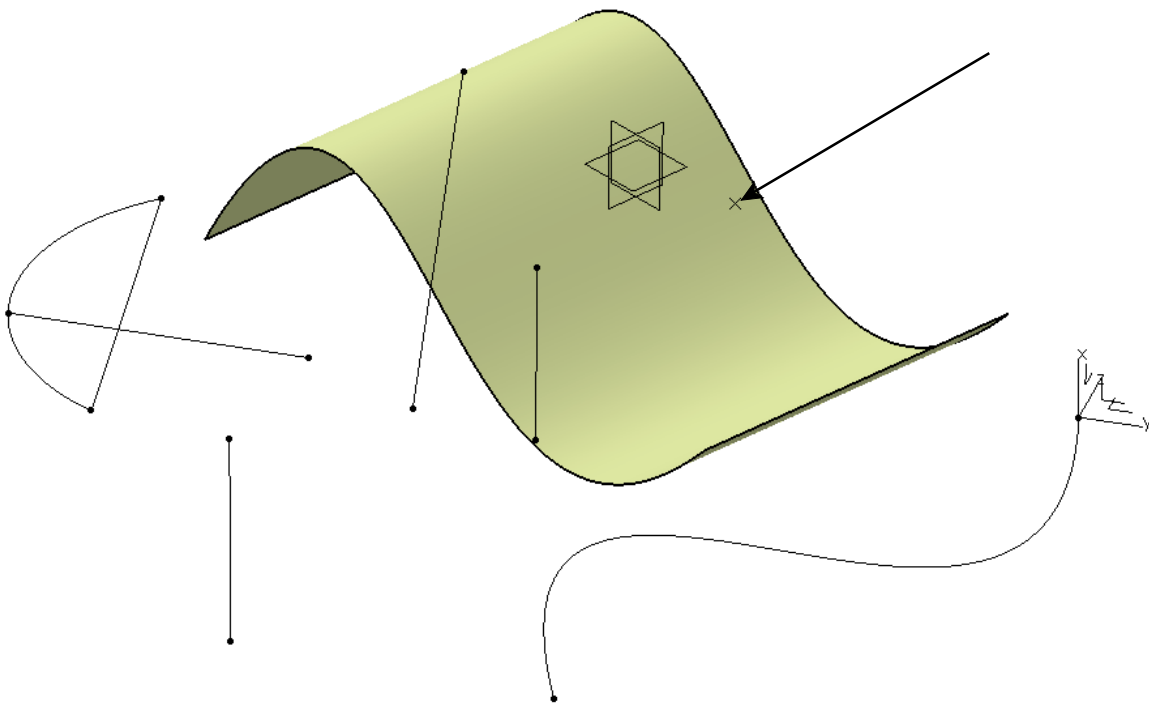
*Axis System* Defines the axis system that will be used to create the point

**Right select in the Axis System selection box.** A contextual window appears.



**Select Clear Selection.** By default, the active axis system is used to create a coordinate point. By clearing the Axis System selection box, the absolute axis system will be used to define the point.


**Key in 2.0, 4.0, 2.0 for the X, Y and Z values respectively and select OK.** You should have noticed a preview of the point as you were keying in the values before you selected *OK*. It should appear similar to the diagram shown below.



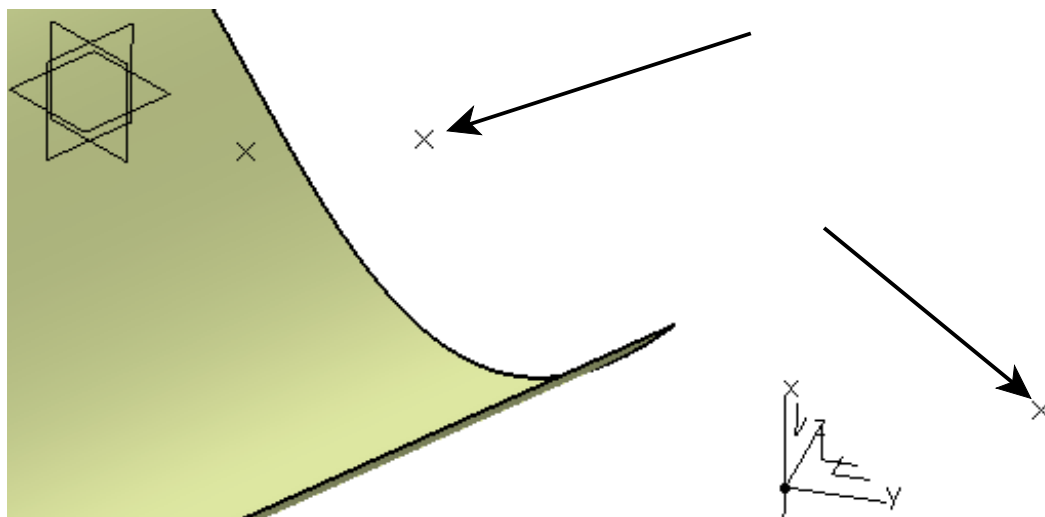
**Select the Point icon again.**  The *Point Definition* window appears. This time you are going to use a point as the reference instead of the origin.

**Clear the *Axis System* selection box so that the absolute axis system will be used and select the point you just created to define the *Reference Point*.** The point is labeled as *Point* and the name of the point appears in the *Point* area of the window.

**Key in 0.0, 2.0, 1.0 for the X, Y and Z values respectively and select *OK*.** The new point is created based from the original point instead of the origin.

**Select the Point icon again.**  The *Point Definition* window appears. This time you are going to create a point using the axis system that has been created instead of using the absolute axis system.

**Key in 1.0, 2.0, 0.0 for the X, Y and Z values respectively and select *OK*.** The point is created based from the origin of *Axis System.1* instead of the absolute axis system. Your model should appear similar to the diagram shown below.

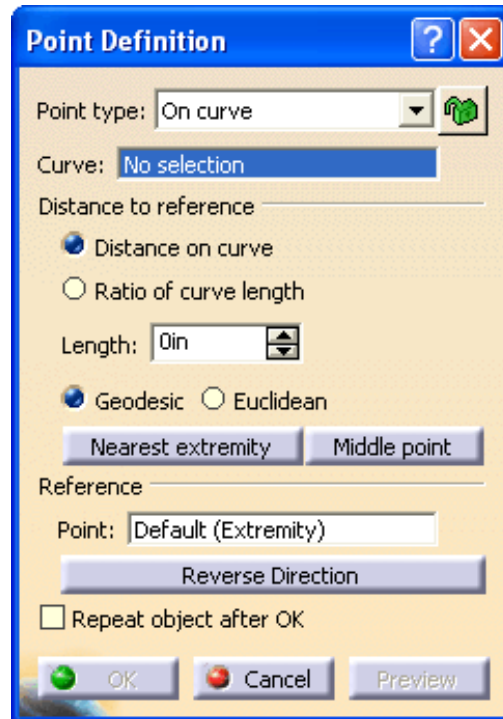


You can also create points on elements such as curves, planes and surfaces.

**On curve**

Select the Point icon.  The *Point Definition* window appears.

Change the *Point type* to *On curve*. The options in the window change.



*Curve* Specifies the curve on which you are going to create a point on

*Distance to reference*

*Distance on curve* Allows you to specify a distance along the curve from the reference point

*Ratio of curve length* Allows you to specify a ratio between the reference point and the extremity

*Length/Ratio* You can specify a *Length* if the *Distance on curve* option is used or a *Ratio* if the *Ratio of curve length* option is used

*Geodesic* The length is measured along the curve

*Euclidean* Corresponds to *Distance on curve* only, the length is the absolute value from the reference point

*Nearest extremity* Creates the point at the nearest extremity

*Middle point* Creates a point at the midpoint of the curve

*Reference*

*Point* Allows you to specify a reference point if you do not want to use an extremity as the reference

*Reverse Direction* Reverses which side of the reference is used or which extremity is used when creating the point

*Repeat object after OK* Allows you to create multiple equidistant points

**Select the curve on the right.** The curve is labeled *Curve* in the display and the extremity shows an arrow.

**Turn the *Distance on curve* option on, change the *Length* to be 5.0 and make sure it is set to *Geodesic*.** The point appears 5 inches along the curve.

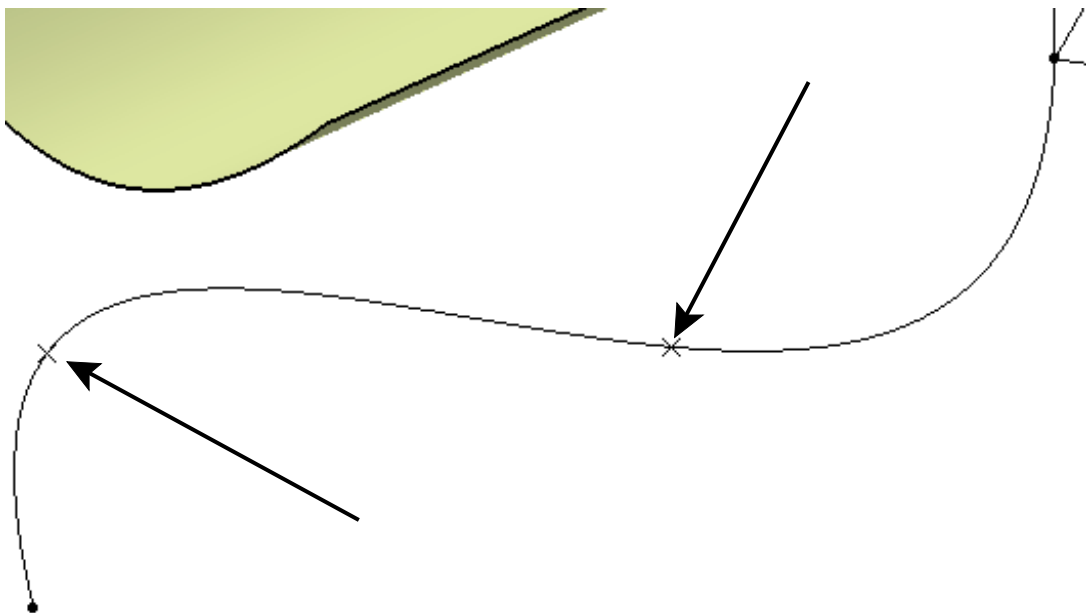
**Turn the *Euclidean* option on instead of the *Geodesic* option.** The point is still on the curve but it is 5 inches from the reference point instead of being 5 inches along the curve.

**Select *OK*.** The point is created.

**Select the Point icon again, make sure the *Point type* is set to *On curve* and select the curve on the right.**

**Turn on the *Geodesic* and *Ratio of curve length* options and change the *Ratio* to 0.25 and select *Preview*.** A point appears a quarter of the way along the curve. A ratio of 0.5 is the midpoint of the curve. You should note that only the *Distance on curve* option can use the *Euclidean* option.

**Select *OK*.** The point is created and should appear similar to the diagram shown below.



**Select the Point icon again and make sure the *Point type* is set to *On curve*. Select the curve on the right.**

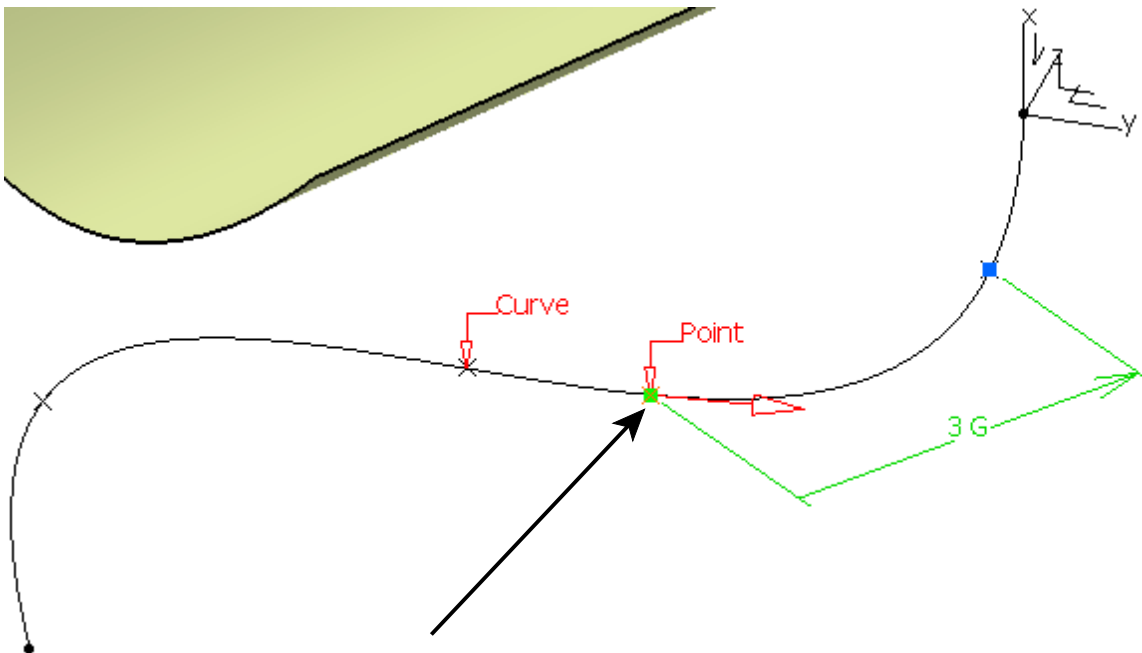
Select the **Nearest extremity** button. The point appears at the nearest endpoint of the curve.

Select the **Middle point** button and select **OK**. The point appears at the midpoint of the curve.

Select the **Point** icon again and make sure the **Point type** is set to **On curve**. Select the **curve on the right**. This time you are going to use a reference point other than an extremity. All the other options work the same except the distance and ratio, which are based from the reference point.

Turn on **Distance on curve, Geodesic** and change the **Length** to **3.0**.

Select in the **Reference Point** box and select the point shown below. Notice the direction of the arrow.



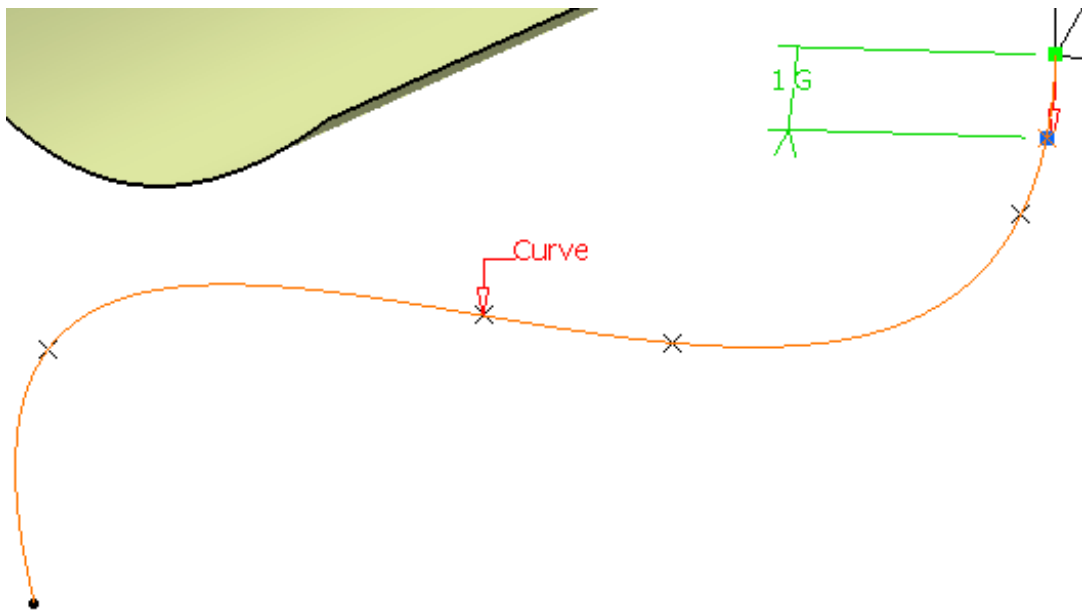
Select the **Reverse Direction** button. The arrow points the opposite direction. If you were using an extremity then reversing the direction would cause the reference point to switch to the other end of the spline.

Select **OK**. The point is created.

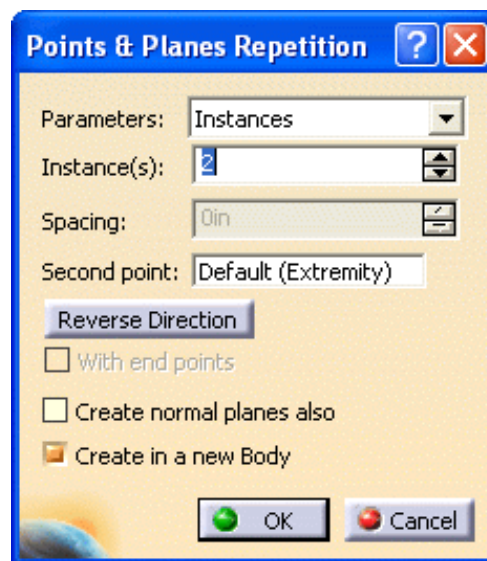
Select the **Point** icon again and make sure the **Point type** is set to **On curve** and select the **curve on the right**.

Turn on **Distance on curve, Geodesic** and change the **Length** to **1.0**.

Select the **Reverse Direction** button. This moves the reference to the other end.



Turn on **Repeat object after OK** and select **OK**. The *Points & Planes Repetition* window appears.



These options will be covered in detail a little later in the book. This was just meant to introduce a method for creating multiple points.

Select **Cancel**. Only the one point is created. This completes the options for creating a point on a curve.

Select off of the point in order to release it.


## Surfaces

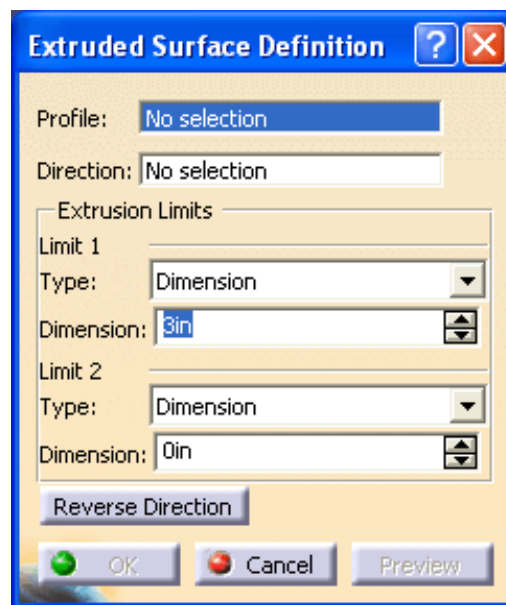
Surfaces are extremely important for defining any type of contour. Using wireframe geometry, you can create surfaces to represent any contour that you need. Once you have created the surface(s) that you need you can then use them in Part Design to contour your solid model. You have a variety of options to create surfaces. Some options are straightforward while others are much more involved.

### Extruded

Extruded surfaces are created by extruding an element in a linear direction. The resulting object is called *Extrude*.

**Open the Basic Surfaces document.** You should see some wireframe geometry.

**Select the Extrude icon.**  An *Extruded Surface Definition* window appears.



*Profile* Specifies the shape that will be extruded

*Direction* Defines the direction of the extrusion

*Extrusion Limits*

*Limit 1/2* Defines the direction and limits for the extrusion

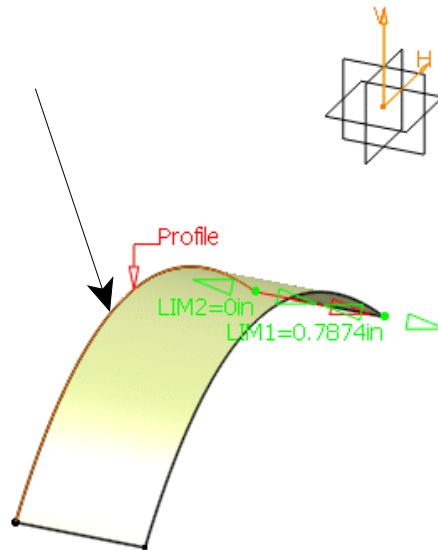
*Type* Specifies either a constant dimension or up to a selection

*Dimension* Specifies the limit distance

*Reverse Direction* Reverses the direction of the extrusion



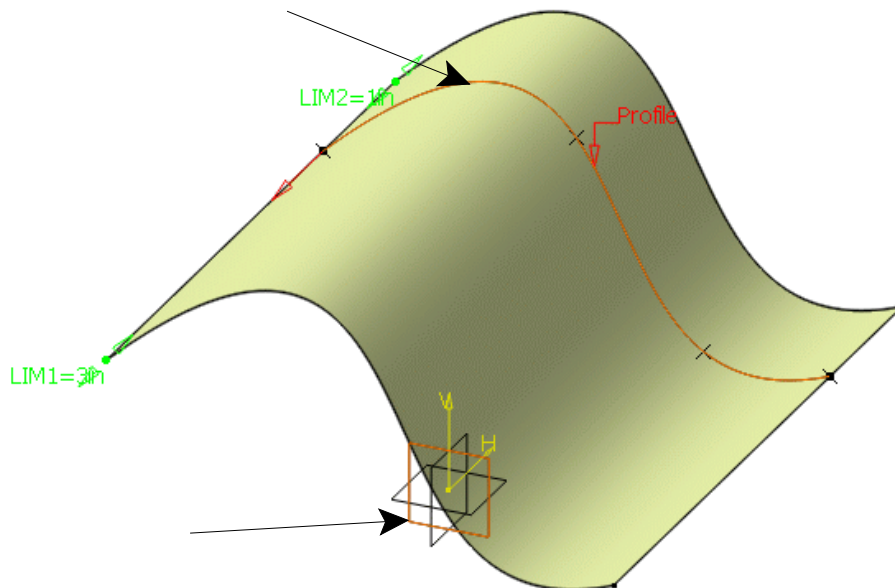
**Select the curve as shown below.** Since this curve was created in a sketch, the extrude option automatically assumes you want to go normal to the sketch.



**Key 3.0 for Limit 1, 1.0 for Limit 2, select the Reverse Direction button and select OK.** The surface is created.

**Select the Extrude icon again.** The *Extruded Surface Definition* window appears.


**Select the curve and plane as shown below.** The plane defines the direction to be normal to the plane.

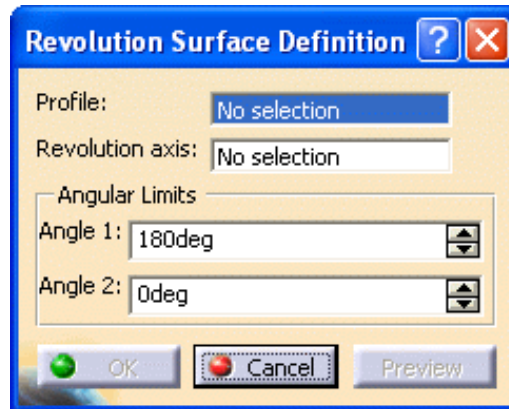


**Change the limits so that both are 1.0 inch and select OK.** The surface is created.

## Revolution

Revolution surfaces are created by rotating an element around an axis. The resulting object is called a *Revolute*.

**Select the Revolve icon.**  The *Revolution Surface Definition* window appears. The icon is located under the extrude icon.



*Profile* Specifies the shape that will be revolved

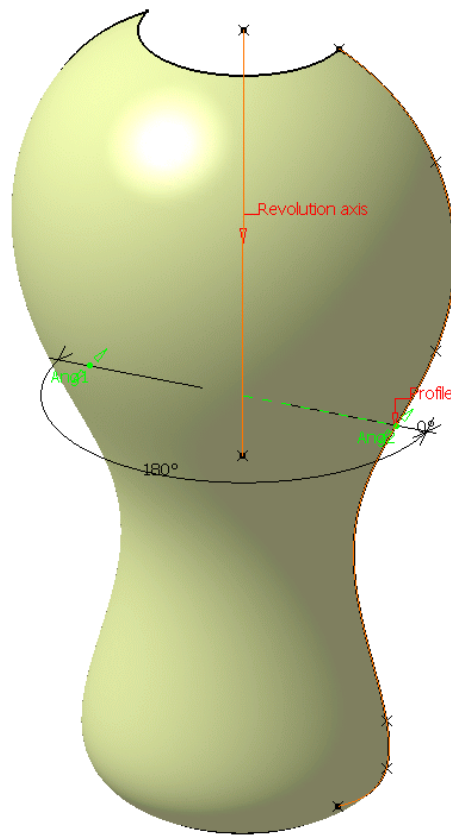
*Revolution axis* Defines the axis around which the profile will revolve. If your profile is a sketch and has an axis defined in it then that will be the default revolution axis.

### *Angular Limits*

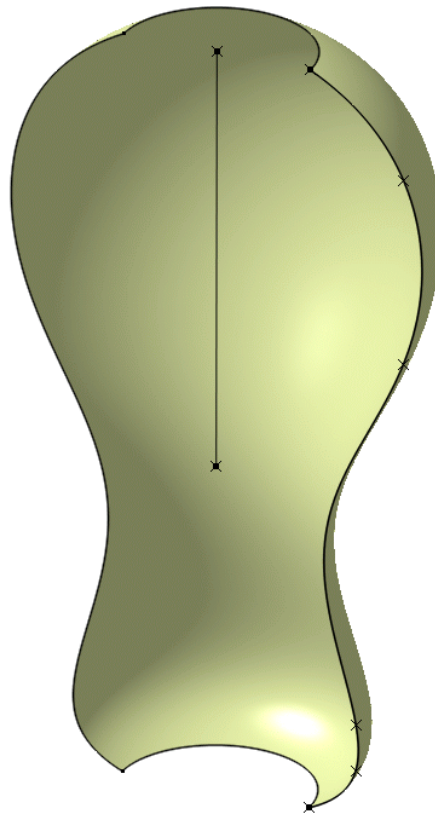
*Angle 1* Defines the starting angle for the revolution

*Angle 2* Defines the ending angle for the revolution

Select the profile and line as shown below.




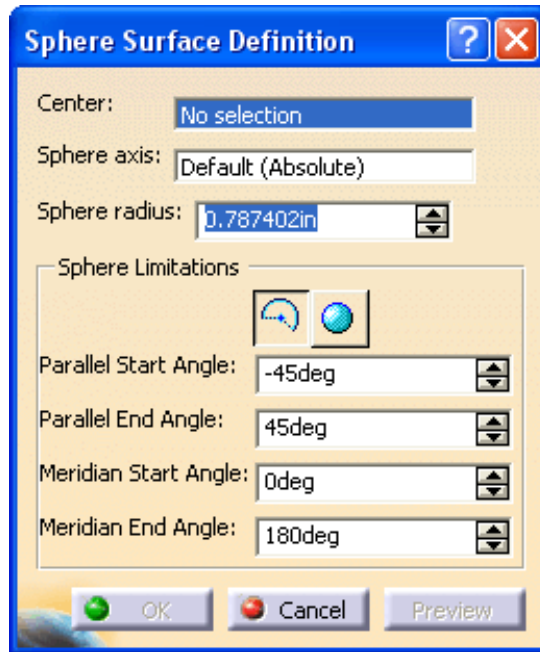
Change *Angle 1* to be 0.0 and *Angle 2* to be 180 and select *OK*. The surface is created.



## Sphere

Sphere surfaces are created by defining a center point and a radius. The resulting object is called a *Sphere*.

**Select the Sphere icon.**  The *Sphere Surface Definition* window appears. The icon is located under the extrude or revolve icon.



*Center* Specifies the center point of the sphere

*Sphere axis* Determines the orientation of the *Parallel* and *Meridian* curves

*Sphere radius* Defines the radius of the sphere

*Sphere Limitations*



With limits



Whole sphere

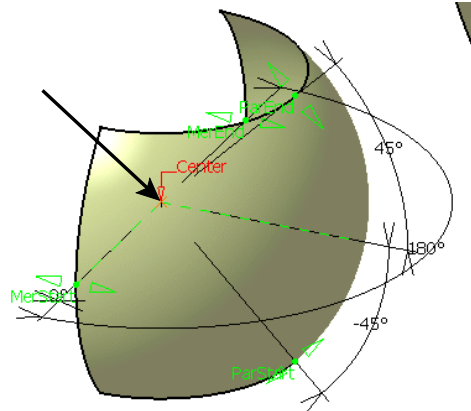
*Parallel Start Angle* Defines the starting angle in the parallel direction

*Parallel End Angle* Defines the ending angle in the parallel direction

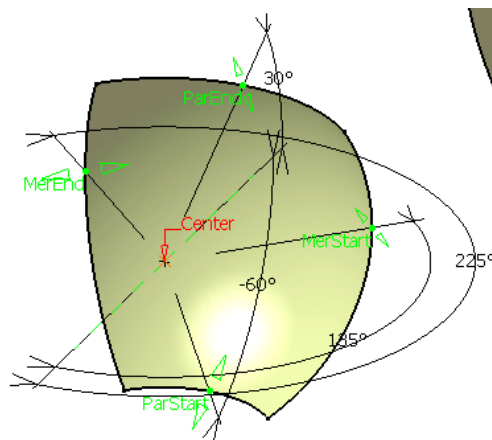
*Meridian Start Angle* Defines the starting angle in the meridian direction

*Meridian End Angle* Defines the ending angle in the meridian direction

Select the point as shown below, key **1.5** for the *Sphere radius* and select *Preview*. Since you do not have any other axis to select, you will use the default. The *Parallel* limits have a range of -90 to 90 while the *Meridian* limits have a range of -360 to 360. Basically, the *Parallel* limits are the up and down limits and the *Meridian* limits are the left and right limits. Of course, this depends on your axis.



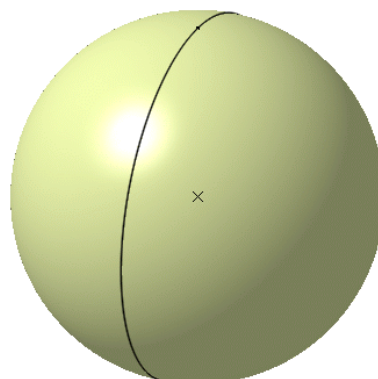
Change the *Parallel Start Angle* to -60, the *Parallel End Angle* to 30, the *Meridian Start Angle* to 135 and the *Meridian End Angle* to 225, and select *Preview*.



Select the **Whole Sphere** icon from the window and select **OK**.




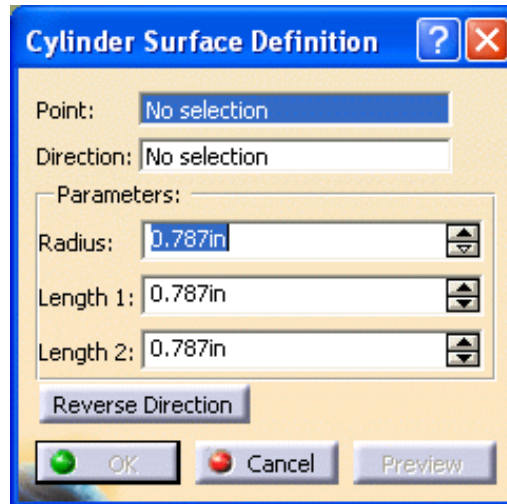
Notice that all the limit options were made unavailable.



## Cylinder

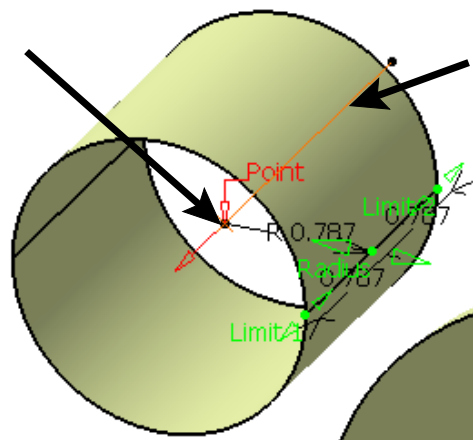
Cylindrical surfaces are created by defining a point and a direction, then you can specify a length and radius. The resulting object is called a *Cylinder*.

**Select the Cylinder icon.**  The *Cylinder Surface Definition* window appears. The icon is located under the extrude or sphere icon.



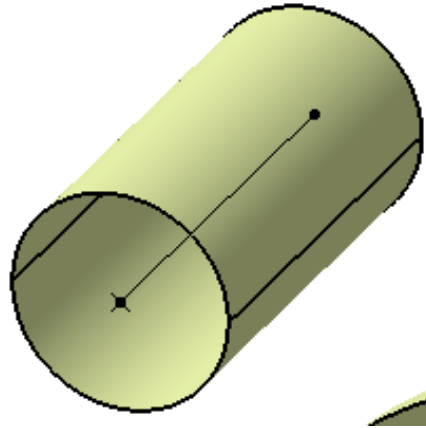
- Point* Specifies the center point of the cylinder
- Direction* Specifies the direction the cylinder will extrude
- Parameters*
- Radius* Defines the radius of the cylinder
  - Length 1,2* Defines the length of the cylinder in both directions
  - Reverse Direction* Reverses the direction of the cylinder

**Select the point and line as shown below.**



Change the *Radius* to 0.5, *Length 1* to 2.0, *Length 2* to 0.0. The cylinder changes size.

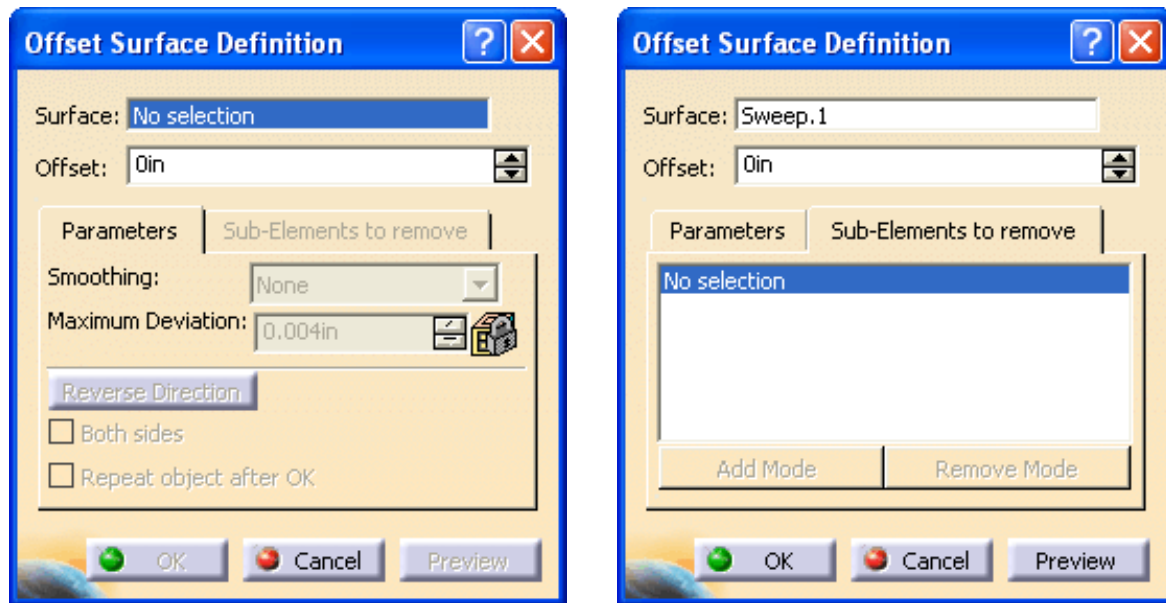
Select the *Reverse Direction* button and select *OK*. The surface is created.



## Offset

Offset surfaces are created by offsetting an existing surface a specified distance. The resulting object is called an *Offset*.

Select the Offset icon.  The *Offset Surface Definition* window appears.



*Surface* Specifies the surface that will be offset

*Offset* Defines the distance of the offset

### *Parameters*

*Smoothing* A smoothing is applied if a constant offset will not work, if it still will not work then a warning message is given

*Maximum Deviation* Defines the maximum amount the new surface can vary from the original

*Reverse Direction* Reverses the direction of the offset

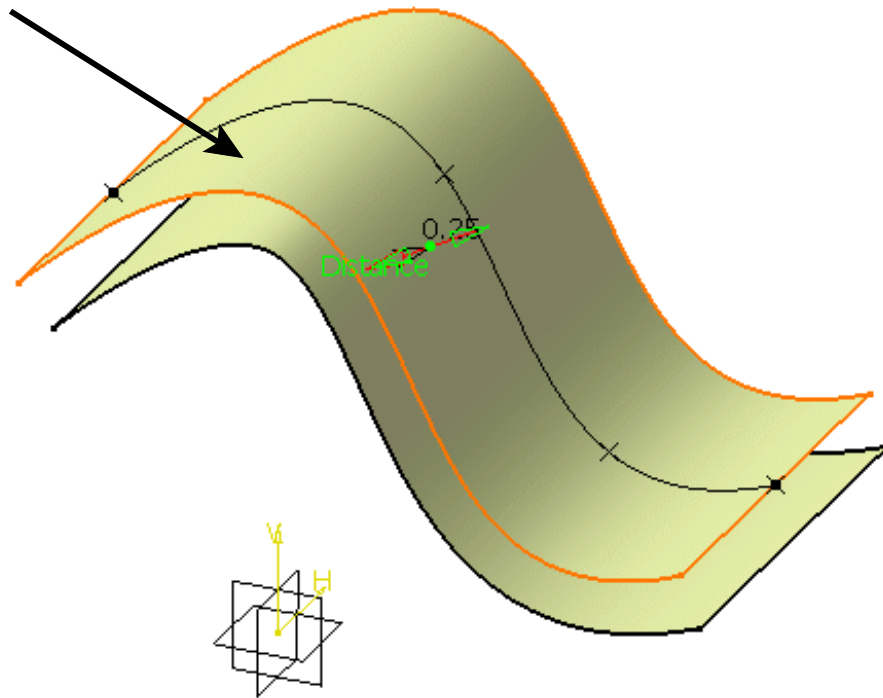
*Both sides* Offsets the surface both directions

*Repeat object after OK* Allows you to repeat the offset numerous times

*Sub-Elements To Remove* If an offset has problems then you can perform the offset without the sub-elements that have errors. The sub-elements will be listed in the *Sub-Elements To Remove* list. This is useful when trying to determine why an offset fails. You can *Add* or *Remove* sub-elements to the list.



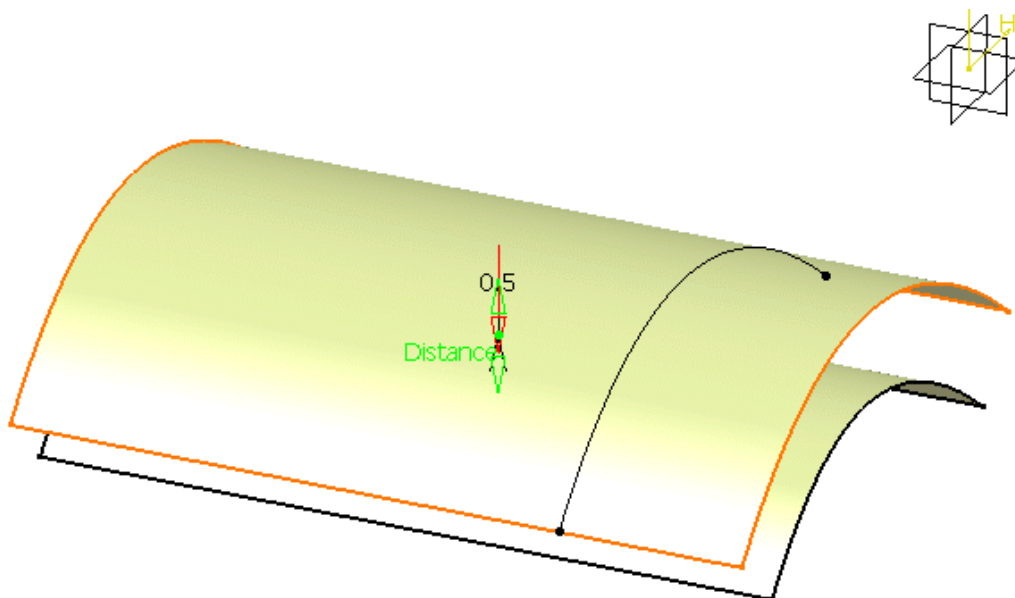
Select the surface as shown below, key **0.25** for the *Offset* and select *Preview*. An offset surface appears.



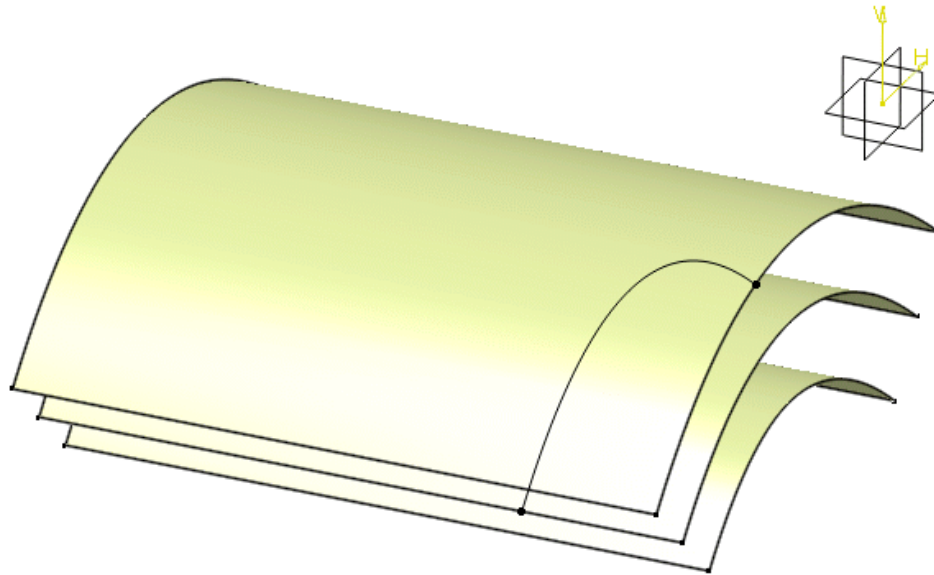
Select the *Reverse Direction* button and select *OK*. The offset surface appears above the original surface instead of below it.

Select the *Offset* icon again. The *Offset Surface Definition* window appears.

Select the surface as shown below, key **0.5** for the *Offset* and select *Preview*. An offset surface appears below the original.



Turn on the **Both sides** option and select **OK**. Offset surfaces appear above and below the original surface.



*Note: Since the offset surface has a Repeat object after OK option, you can use the object repetition icon on offset surfaces.*




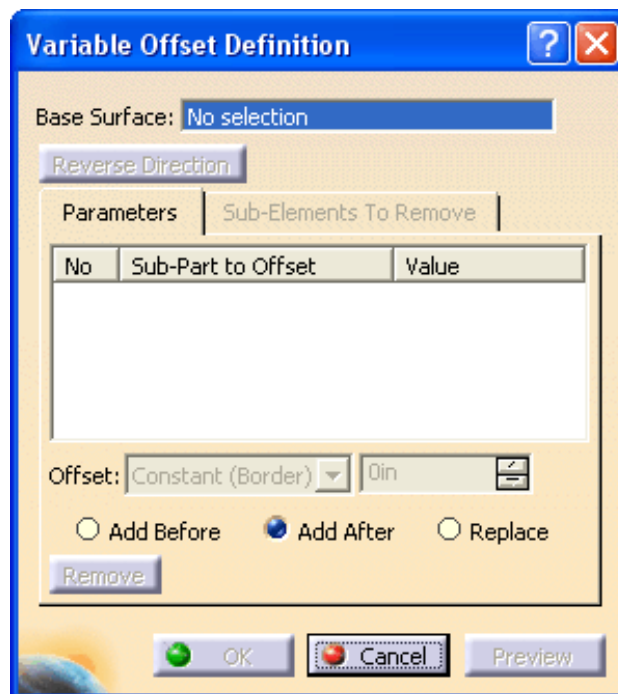
**Save and close your document.**

## Variable Offset

A variable offset surface is created by offsetting a multiple element surface and specifying different offset values for the sub-elements. The resulting element is called a *Variable Offset*.

**Open the Offset Surfaces document.** You should see a surface that is comprised of three sub-elements.

**Select the Variable Offset icon.**  The *Variable Offset Definition* window appears. The icon is located under the offset icon.



*Global Surface* Specifies the surface that is going to be offset, it should contain multiple sub-elements

*Parameters* Allows you to define each sub-element with a *Constant Offset* or a variable offset. If you choose a constant offset then a value must be specified.

*Add Before* Allows you to add a sub-part before the current one

*Add After* Allows you to add a sub-part after the current one

*Replace* Allows you to replace the current sub-part with another

*Remove* Removes the current sub-part

The other options are the same as offset.

**Select the surface.** This defines the *Global Surface*. In order to define the sub-parts you will have to select them from the specification tree.

**Select *Extrude.1* from the specification tree.** This defines a sub-part.

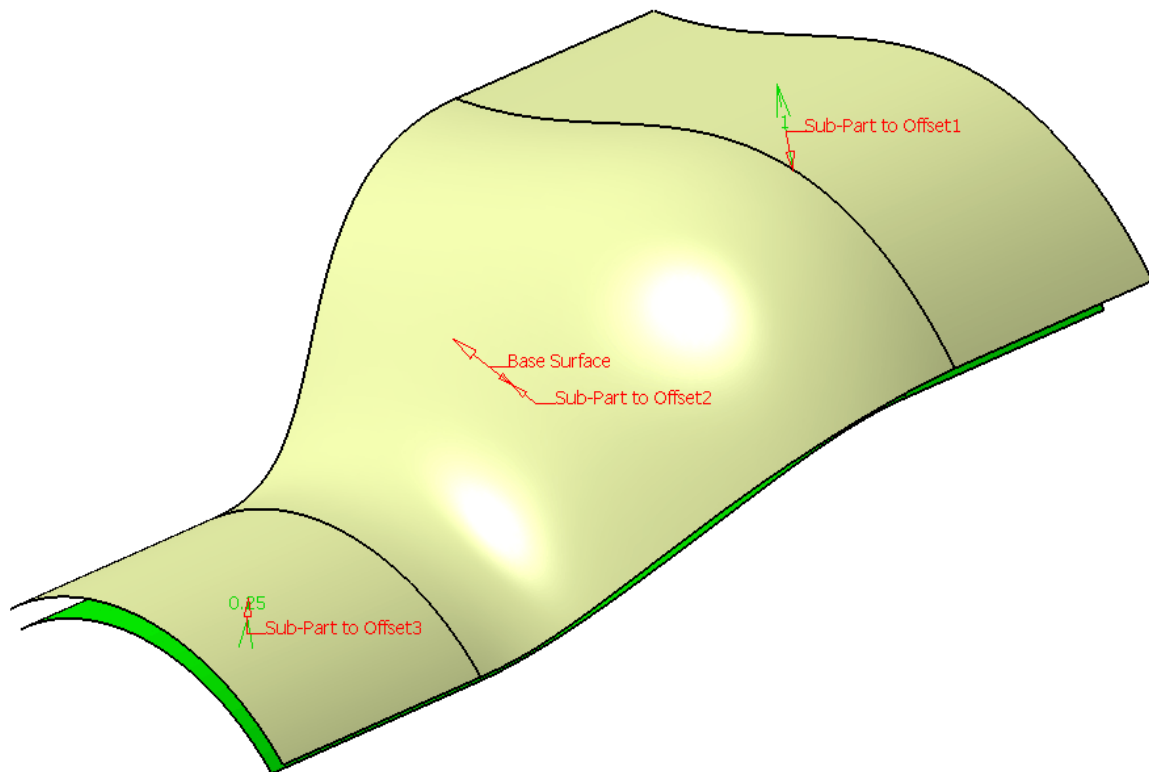
**Set the *Offset* option to be *Constant* and key **1.0** for the value.**

**Select *Blend.1* from the specification tree and set the *Offset* option to be *Variable*.** This makes this sub-part variable.

**Select *Extrude.2* from the specification tree.** This will be the last sub-part.

**Set the *Offset* option to be *Constant* and key **0.25** for the value.**

**Make sure the arrow is pointing upwards and select *Preview*.**



**Select *OK*.** The variable offset surface is created.