TABLE OF CONTENTS

Introduction .		. 1
Wirefra	ame & Surfaces	. 2
Genera	tive Shape Design Workbench	. 3
Wireframe		15
Points		15
	Coordinate	15
	On curve	18
	On plane	22
	On surface	24
	Circle / Sphere / Ellipse center	26
	Tangent on curve	27
	Between	30
	Point Repetition	32
	Projecting points	37
	Intersection Points	41
	Extremum	45
	Polar Extremum	48
Lines.		50
	Point-Point	50
	Point-Direction	53
	Angle/Normal to curve	55
	Tangent to curve	60
	Normal to surface	63
	Bisecting	65
	Intersection lines	67
	Projecting lines	68
	Avis	69
	Polyline	71
Planes		75
1 failes	Offsat from plane	75
	Darallel through point	ר קר דר
	Angle/Normal to plane	// 70
	Through three points	70 90
	Through the points	00
	Through two lines	01
	Through point and line	02
		03
		84
		86
	Equation	8/
	Mean through points	89
	Between	90
~	Plane Between	92
Circles	~	95
	Center and radius	95
	Center and point	98

Two points and radius	99
Three points	101
Center and axis	103
Bitangent and radius	104
Bitangent and point	106
Tritangent	108
Center and tangent	110
Corners	111
Curves	116
Connect Curves	116
Conics	121
Splines	128
Helixes	134
Spirals	141
Project Curves	145
Combine Curves	. 148
Reflect Line Curves	152
Silhouette Curves	155
Intersection Curves	158
Parallel Curves	162
Rolling Offset Curve	168
3D Curve Offset	170
Curve comparison	173
Supports	175
Work on Support	175
Creation on the fly	181
Modifying	184
Datume	186
Object repetition	188
	100
Surfaces	189
Fytruded	189
Revolution	101
Snhere	103
Culinder	106
Offsat	108
Mid Surface	201
Swapt Surfaces	211
Swept Suffaces	
With reference surface	
With two guide curves	
With pulling direction	
	223
Lillear	227
I WO IIIIIIS	
With reference surface	
With reference curve	
with tangency surface	241
With draft direction	244

With two tangency surfaces	. 248
Circular	. 251
Three guides	. 252
Two guides and radius	. 254
Center and two angles	. 257
Center and radius	. 259
Two guides and tangency surface	. 261
One guide and tangency surface	. 263
Limit curve and tangency surface	. 265
Conical	. 268
Two guide curves	. 269
Three guide curves	. 271
Four guide curves	. 273
Five guide curves	. 275
Adaptive Sweep	. 277
Fill Surfaces	. 283
Multi-Section Surfaces	. 290
Blend surfaces	. 301
Spines	. 308
Isoparametric Curves	. 313
Laws	. 315
Operations	. 329
Joining Elements	. 329
Healing Surfaces	. 337
Curve smoothing	. 342
Splitting Elements	. 343
Trimming Elements	. 350
Untrimming Elements	. 354
Disassembling Elements	. 356
Disconnect	. 358
Sew Surface	. 360
Remove Face	. 362
Extracting Boundaries and Faces	. 364
Multiple Extract	. 369
Fillets	. 373
Shape Fillet	. 373
Edge Fillet	. 380
Face-Face Fillet	. 386
Tritangent Fillet	. 388
Styling Fillet	. 390
Automatic Filleting	. 397
Chamfer	. 402
Transformations	. 407
Translate	. 407
Rotate	. 410
Symmetry	. 413
Scale	. 414
Affinity	. 416

Axis to Axis	. 419
Patterns	. 420
Rectangular	. 420
Circular	. 423
User Defined	. 424
Extrapolating Curves and Surfaces	. 425
Inverting the Orientation	. 431
Creating the Nearest or Farthest Element	. 432
Multi-Selection	. 434
Analysis	. 439
Connect Checker Analysis	. 439
Light Distance Analysis	. 450
Draft Analysis	. 452
Surfacic Curvature Analysis	. 457
Porcupine Curvature Analysis	. 462
Geometric Information	. 472
Dress-Up	. 474
-	
Geometrical Set Management	. 477
Inserting Sets	. 477
Changing Sets	. 479
Hiding and Showing Components	. 481
Activating and Deactivating Components	. 482
Reordering Components	. 483
Groups	. 485
Isolating Geometry	. 487
Copying and Pasting Geometry	. 488
Ordered Geometrical Set Management	. 491
Inserting an Ordered Geometrical Set	. 491
Modifying Children	. 493
Reordering Components	. 495
Scanning Ordered Sets	. 497
Inserting in an Ordered Set	. 498
Switching to a Regular Geometrical Set	. 499
Miscellaneous	. 500
Parents/Children	. 500
Historical Graph	. 502
Quick Select	. 504
Inserting Elements	. 506
Sets of planes	. 507
Regular	. 507
Semi-Regular	. 509
Irregular	. 510
Keep and No Keep Mode	. 512
Keep Mode	. 512
No Keep Mode	. 512

Ma-1	7
$\mathbf{Masks} \dots \dots$	1 1
2D Visualization Modes	ł
Deleting Useless Elements	5
Review)
Mouse Body)
Buttons and Wheel	Ĺ
Problems	3
Problem #01	3
Problem #02	1
Problem #03	5
Problem #04	5
Problem #05	3
Problem #06)
Problem #07	2
Problem #08	1
Problem #09	7
Problem #10)
Appendix A 582	2
Shape - Generative Shape Design - General 583	,
Shape - Generative Shape Design - General	ζ.
Shape - Generative Shape Design - Work On Support 584	3
Shape - Generative Shape Design - Work On Support	5
Shape - Generative Shape Design - Work On Support 585 Appendix B 587	3 5 7
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587	3 5 7 7
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587	3 5 7 7 7
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 589	3 5 7 7 7 7
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 589 Close 591	3 5 7 7 7 7
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592	357779L2
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 594	357779121
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 594 Boolean Operations 595	3577791215
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 592 Boolean Operations 595 Appendix C 597	35777912157
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 592 Boolean Operations 595 Appendix C 597 Developed Shapes 597	35 77791215 77
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 594 Boolean Operations 595 Appendix C 597 Developed Shapes 597 Unfold Surfaces 597	35 77791215 777
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 592 Boolean Operations 595 Appendix C 597 Unfold Surfaces 597 Transfer 602	35 77771215 7772
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 594 Boolean Operations 595 Appendix C 597 Unfold Surfaces 597 Transfer 602 Develop Wires 605	35 77771215 77725
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 592 Boolean Operations 592 Appendix C 597 Developed Shapes 597 Unfold Surfaces 597 Transfer 602 Develop Wires 602	35 77771215 77725
Shape - Generative Shape Design - Work On Support 585 Appendix B 587 Part Design Using Surfaces 587 Split 587 Thick Surface 587 Close 591 Sew 592 Pad/Pocket 592 Boolean Operations 592 Appendix C 597 Developed Shapes 597 Unfold Surfaces 597 Transfer 602 Appendix D 611 Volumes 612	3 5 7 7 7 7 7 1 2 1 5 7 7 7 2 5 L

Introduction

CATIA Version 6 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 V6 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 to 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 can be reviewed via the exercises located in Appendix B. It is also important that you understand how to work with 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 non-solid geometry. When you create new wireframe or surface geometry, you will need to be sure that the correct geometrical set is the in-work object in order to have an organized tree.

Wireframe

Wireframe geometry is critical to the creation of surfaces and is used as reference elements 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 that will be explored in the following exercises.

Coordinate

Open the WFAS - Points document. Remember, you will need to search for <u>WFAS - Points</u> and then open the document. You should see a surface and some wireframe geometry.

Switch to the Generative Shape Design workbench if are not already there. To change workbenches, you can select the My 3D Modeling Apps (West quadrant) section of the compass, then choose *Generative Shape Design*.

Select the Point icon.	^D The <i>Point Definition</i> window appears.	
	Point Definition ? X	
	Point type: Coordinates 🗸 🍿	
	X = Oin	
	Y = Oin	
	Z = Oin	
	Reference	
	Point: Default (Origin)	
	Axis System: Axis System.1	
	Robot Location	
	OK Cancel Preview	
Point type	Specifies what type of point you want to create: <i>Coordinates, On curve, On plane, On surface,</i>	
	Circle/Sphere/Ellipse center, Tangent on curve, or	

The coordinate values of the point to be created from the reference point

X=, *Y*=, *Z*=

Reference

Point	The point that the coordinates are based from; the default is the origin
Axis System	Defines the axis system that the point will be based off of
Robot Location	Creates the point at the location of the robot in the display

Press the third mouse button in the Axis System field. A contextual menu appears.

	Center tree on Preselected Objects
	<u>R</u> eframe on
۲	<u>H</u> ide/Show
	<u>P</u> roperties
	Other Selection
	Clear Selection
	Insert Wireframe

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

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



Select the Point icon again. The *Point Definition* window appears. This time, you will 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, then 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* field of the window.

Enter 0.0, 2.0, and 1.0 for the *X*, *Y*, and *Z* values respectively, then select *OK*. The new point is created off of the previous point rather than the origin.

Select the Point icon again. The *Point Definition* window appears. This time, you will use the axis system that has been created instead of using the absolute axis system.

Enter 1.0, 2.0, and 0.0 for the X, Y, and Z values respectively, then select OK. The point is created off of 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 change as shown here.

Point Definition ? \times		
Point type: On curve 🗸 🍿		
Curve: No selection		
Distance to reference		
Distance on curve		
O Distance along direction		
○ Ratio of curve length		
Length: Oin		
Geodesic O Euclidean		
Nearest extremity Middle point		
Reference		
Point: Default (Extremity)		
Reverse Direction		
Repeat object after OK		
OK Cancel Preview		

Curve	Specifies the curve on which the point will be created
Distance to reference	Determines the mode to use for the point creation
Distance on curve	The distance along the curve from the reference point
Distance along direction	The distance along the curve in a particular direction
Ratio of curve length	The ratio between the reference point and the extremity
Length/Offset/Ratio	A user-defined value to specify either the <i>Length</i> for the <i>Distance on curve</i> option, the <i>Offset</i> for the <i>Distance along direction</i> option, or the <i>Ratio</i> for the <i>Ratio of curve length</i> option
Geodesic	Forces the length to be measured along the curve

CATIA Wireframe & Surfaces	3DEXPERIENCE® R2019x
Euclidean	Corresponds to <i>Distance on curve</i> 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 in the middle of the curve
Reference	
Point	Allows you to specify a reference point if you do not want to use an extremity for the reference
Reverse Direction	Reverses which side of the reference the point is created on, or which extremity is used
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 a red arrow.

Select the *Distance on curve* option, then change the *Length* to 5.0 and make sure it is set to *Geodesic*. The point appears at five inches along the curve.

Select the *Euclidean* option instead of *Geodesic*. The point is still on the curve, but it is now five inches from the reference point instead of five inches along the curve.

Select OK. The point is created.

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

Set the *Reference Point* to be the origin of the axis system at the end of the curve.

Select the *Distance along direction* option, then select the *yz plane* from the specification tree for the *Direction*. This will be where the offset is measured from.

Change the *Offset* to 1.0 and select *OK*. The point is created and is measured normal from the yz plane along the curve.

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

Select the *Ratio of curve length* and *Geodesic* options, then 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. Only the *Distance on curve* option can use a *Euclidean* measurement.

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



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

Select the *Distance on curve* option, then select the *Nearest extremity* button. The point appears at the nearest endpoint of the curve.

Select the *Middle point* button and click *OK*. The point appears in the middle of the curve.

Select the Point icon again and ensure the *Point type* is set to *On curve*, then select the curve on the right. This time, you will use a reference point other than an extremity.

Select Distance on curve, then choose Geodesic and change the Length to 3.0.

Select the *Reference Point* field and choose the point shown below. Notice the direction of the arrow.



Select the *Reverse Direction* **button.** The arrow points to the opposite direction. If using an extremity, reversing the direction causes the reference point to switch to the other end of the spline.

Select OK. The point is created.

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

Choose Distance on curve and Geodesic, then change the Length to 1.0.

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



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

Points Repetition ? ×		
First Point:	Point.23	
Curve:	Default (Spline.1)	
Parameters:	Instances \sim	
Instance(s):	2	
Spacing:	0.039in	
Second point:	Default (Extremity)	
Mode for Repe	etition	
💿 Absolute 🕻	🗅 Relative	
Reverse Direction		
With end points		
Create normal planes also		
Create Axis System		
Axis System Repetition		
Туре:	Clones 🗸 🗸	
Axis System: No selection		
Create in a new editable Body		
OK Cancel Preview		

Select *Cancel.* These options will be covered in detail later in the book. Only the one point is created. This completes the options for creating a point on a curve.

Select in space to release the point.

Surfaces

Surfaces are extremely important for defining contours. With the use of wireframe geometry, surfaces can be created to represent any contour needed. Once the surfaces are created, they can then be used in Part Design to make a solid model. There are a variety of options for creating surfaces. Some are straightforward, while others are much more complex.

Extruded

Extruded surfaces are created by extending an element in a linear direction. The resulting object is called an *Extrude* in the specification tree.

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

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

Extruded Su	rface Definition	?	×
Profile:	No selection		
Direction:	No selection		
Extrusion	Limits		
Limit 1			
Туре:	Dimension		\sim
Dimension:	0.787in		-
Limit 2			
Туре:	Dimension		\sim
Dimension:	0in		-
Mirrored Extent			
Reverse Direction			
OK.	Cancel	Prev	view

Profile

Specifies the shape to be extruded

Defines the direction of the extrusion

Direction

Extrusion Limits

Limit 1/2

Туре

Defines a distance or a limiting element

Two options available: *Dimension* and *Up-to element*

Dimension

Specifies a distance for the extrusion to extend

Up-to element

Specifies an element that the extrusion will stop at

Mirrored Extent	Forces <i>Limit 2</i> to be the same as <i>Limit 1</i> ; only available when the <i>Type</i> is set to <i>Dimension</i>
Reverse Direction	Reverses the direction of the extrusion

Select the curve shown below. This curve was created in a sketch. Whenever a sketch is selected for an extrusion, the direction will automatically be normal to the sketch plane.



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

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

Select the curve and plane as shown below. When a plane is selected for the direction, the surface will extend normal to the plane.



Change both *Dimension* fields to 1.0 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. It is located within the sub-toolbar of the Extrude icon. The *Revolution Surface Definition* window appears.

Revolution Surface Defini ? \times				
Profile: No selection				
Revolution axis: No selection				
Angular Limits				
Limit 1				
Type: Dimension \checkmark				
Angle 1: 180deg				
Limit 2				
Type: Dimension \checkmark				
Angle 2: Odeg				
OK Cancel Preview				

Profile	Specifies the shape that will be revolved
Revolution axis	Defines the axis around which the profile will revolve; if the profile is a sketch that has an axis defined within it, CATIA will use that axis for the revolution
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 0.0 and Angle 2 to 180, then select OK. The surface is created.



Sphere

Spherical surfaces are created by defining a center point and a radius. The resulting object is called a *Sphere* in the specification tree.

Select the Sphere icon. It is located within the sub-toolbar of the Extrude icon. The *Sphere Surface Definition* window appears.

Sphere Surface Definit	tion	?	×
Center: No selec	ction		
Sphere axis: Default	(Absolute)		
Sphere radius: 0.787	in	-	
Sphere Limitations			
	< ●		
Parallel Start Angle:	-45deg		-
Parallel End Angle:	45deg		-
Meridian Start Angle:	0deg		-
Meridian End Angle:	180deg		÷
OK	Cancel	Prev	iew

Center	Specifies the center point of the sphere
Sphere axis	Determines the orientation of the <i>Parallel</i> and <i>Meridian</i> curves
Sphere radius	Defines the radius of the sphere
Sphere Limitations	
Angles	Creates a partial sphere
Whole Sphere	Creates a full sphere
Parallel Start Angle	Defines the starting angle in the parallel direction; only available when theAngles icon is selected
Parallel End Angle	Defines the ending angle in the parallel direction; only available when theAngles icon is selected

CATIA Wireframe & Surfaces	3DEXPERIENCE® R2019x		
Meridian Start Angle	Defines the starting angle in the meridian direction; only available when theAngles icon is selected		
Meridian End Angle	Defines the ending angle in the meridian direction; only available when theAngles icon is selected		

Select *Point.6* either graphically or from the tree and enter 1.5 for the *Sphere radius*, then select *Preview*. Since you do not have an axis to select, you will use the default. The *Parallel* limits have a range of -90 degrees to 90 degrees, while the *Meridian* limits have a range of -360 degrees to 360 degrees. Basically, the *Parallel* limits are up and down, while the *Meridian* limits are side to side. This will depend on the axis selected, however.



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, then select *Preview*.



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

options became unavailable.

KOK. All the



Cylinder

Cylindrical surfaces are created by defining a point, a direction, a radius, and a length. The resulting object is called a *Cylinder* in the tree.

Select the Cylinder icon. It is located within the sub-toolbar of the Extrude icon. The *Cylinder Surface Definition* window appears.

Cylinder S	ourface Definition	?	×	
Point:	No selection			
Direction:	No selection			
Parame	ters:			
Radius:	0.787in		×	
Length 1:	0.787in		-	
Length 2:	0.787in		-	
Mirror	ed Extent			
Reverse	Direction			
OK	Cancel	Pre	view	

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	Defines the length of the cylinder in the first direction	
Length 2	Defines the length of the cylinder in the second direction	
Mirrored Extent	Changes Length 2 to be the same as Length 1	
Reverse Direction	Reverses the direction of the cylinder	

Select the point and the line as shown below.



Change the *Radius* to 0.5, *Length 1* to 2.0, and *Length 2* to 0.0, then select the *Reverse Direction* button and click *OK*. A cylindrical surface is created.



Offset

Offset surfaces are created by offsetting an existing surface by a specified distance. Offsets are always extended normal to the original element. The resulting object is called an *Offset*.

Select the Offset icon from the <i>Transform</i> section.	T	The Offset Surface Definition
window appears.		

	Offset Surface Defi	nition	?	\times
	Surface: No select	ion		
	Offset: Oin			-
	Parameters S	Sub-Elements to	remove	
	Smoothing:	None	\sim	
	Regularization:		Global	
	Maximum Deviati	0.004in		
	Reverse Direction	n		
	Repeat object	after OK		
	Automatically C	Computes Sub-E	lements To Rer	nove
		OK Can	cel Prev	iew
Surface	S	pecifies the s	urface to be	offse
Offset	D	Defines the dis	stance of the	e offse
Parameters				
Smoothing		Creates assist i	s deviation i n creating th	n the
Maximum Dev	iation	Define can var	s the maxim	um ai origin
Reverse Direct	tion	Revers	es the direct	tion of
Both sides		Offsets	the surface	in bo
Repeat object o	after OK	Repeat	the offset n	umer
Sub-Elements To Rem	ove E aı ta fı tı	Excludes prob re added to a ab. Sub-elem rom it in orde he offset to fa	lematic surf list under th ents can be r to determi il.	aces f ne <i>Sub</i> addec ne wh

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



Select the *Reverse Direction* button and click *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 shown below and enter 0.5 for the *Offset*, then 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 in its definition window, you can use the Object Repetition icon to duplicate it, if desired.

Save and close your document.

Review

For this review exercise, you will create a computer mouse. The intention of the exercise is to demonstrate the process of building a solid model by utilizing wireframe and surface geometry.

Note: Set your view mode to Shading With Edges Without Smooth Edges in order to obtain the same results shown in the following images.

Mouse Body

You will first create the mouse body, followed by the buttons and wheel.

Create a new 3D part.

Insert a geometrical set named <u>Mouse Body</u>, then select the **Positioned Sketch** icon and set the options as shown below.

>		
🖉 Sketc	h Positioning	×
▼ Supp	ort	
Туре	Positioned	-
	xy plane	
- Origi	n	
Туре	Part origin	- /
• Orier	ntation	
Туре	Y Axis	-
	H Direction VI	Direction
Reverse	H Reverse V	Swap
	ОК	Cancel

Create the following sketch. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



Create the following sketch on the zx plane. The bottom of the arcs are coincident to the extremum points. Be sure the taller end of this sketch is towards the wider end of the first sketch.



Extract each curve from the sketch.

Create a *Point-Point* line between the top points of the arcs.



Create a plane through the line. Use the line for the *Rotation axis* and the zx plane for the *Reference*. The plane should be normal to the reference.



Select the Positioned Sketch icon and set the options as shown below.

Supp	ort	
Туре	Positioned	-
	Plane.1	
Origi	n	
Туре	Part origin	-
Orier	ntation	
Туре	Y Axis	-
	H Direction V	Direction
Reverse	H Reverse V	Swap

Create the following sketch. The top and bottom arcs in this sketch are coincident to the upper end points of the extracted arcs. All curves are tangent continuous. The geometrical constraints have been hidden for clarity.



Your model should look like this.



Create a spline between the two points at the top of each extracted arc. $\frown \Box$ The spline will be tangent continuous to both arcs with a tension of 0.375 at the first point, and

0.75 at the second point.



Create two geodesic points on the new spline. [□] The left point will have a ratio value of 0.2 from the left end of the spline, and the right point will have a ratio value of 0.3 from the right end of the spline.



Create a plane normal to the upper spline at both points.



Extrude the extracted spline two inches in both directions normal to the zx plane, then change the name of the extrude to <u>CHANNEL SURFACE</u> in the specification tree.



Project the last sketch you created to *CHANNEL SURFACE* along the normal direction of the first plane created.

Split *CHANNEL SURFACE* with the projected curve, keeping the inner portion. The split surface is shown below. Much of the geometry has been hidden. Feel free to hide your geometry as necessary to reduce clutter.



Fill your first sketch with a surface. A This is the bottom profile of the mouse.



Create an intersection line between both planes and surfaces shown below. You should have four, separate intersection lines.



Create a 0.5 inch line normal to the intersection line indicated below that starts at the point and uses the plane as its support.



Create another line using the same geometry, but this time specify an *Angle* of -60. Ensure the line extends upward.

Repeat this process at the other point so that your model looks the same as below.



Create the two splines shown below. \frown The tension at all three points for both splines should be 1.0. Use the normal lines you just created for the tangent direction of the first and third points, and the zx plane for the tangent direction of the second points. Ensure each spline lies on the appropriate support plane indicated below.



Next, create the spline shown below. \frown This spline is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.5 at the bottom point. Ensure the spline lies on the support plane indicated below.



Create the spline shown below using the same method as the previous spline. It is tangent continuous to the spline above it and uses the angled line for the bottom point's tangent direction. The tension is 1.0 at the top point, and 1.25 at the bottom point. Ensure the spline lies on the support plane indicated below.



Mirror each of the last two splines created across the zx plane.

Join the three curves indicated below. Ensure they are tangent continuous.



Create another join for the three curves shown below. Ensure they are tangent continuous.



Create a boundary curve on the following edge.



Create another boundary curve on the edge shown below.



Create the same two boundaries on the opposite side of the surfaces. Only the boundary curves, the joined curves, the split surface, and the filled surface are shown below.



Split the upper spline at its normal planes, then hide all geometry except the boundary curves, the joined curves, and the new split.



Create a multi-section surface using the joined curves as sections, and the boundary curves and split curve as guides.

Next, create the two boundary curves shown below. You'll have to show the two surfaces.

Create a multi-section surface using the geometry shown below. The splines and the extracted arc are the sections, and the boundaries are the guides. Ensure the first and last sections are tangent continues to the surface shown below.

Your model should look like this.

Create the boundaries shown below.

Create a multi-section surface using the geometry shown below. The splines and the extracted arc are the sections, and the boundaries are the guides. Ensure the first and last sections are tangent continues to the surface shown below.

Your model should look like this. Only the three multi-section surfaces are shown below.

Split the upper spline at the plane shown below, keeping the left side. \checkmark

Create a fill surface from the curves shown below that passes through the split curve you just created. Ensure the fill surface is tangent continuous to the existing surfaces.

Your model should look like this. Only the multi-section surfaces and the fill surface are shown below.

Split the upper spline at the plane shown below, keeping the right side. 🖈

Create a fill surface from the curves shown below that passes through the split curve you just created. Ensure the fill surface is tangent continuous to the adjacent surfaces.

Your model should look like this. Only the multi-section and fill surfaces are shown.

Join all of the multi-section and fill surfaces together. arrow is pointing to the inside of the join. Do not forget to include the first fill surface you created for the bottom profile.

Create a 0.125 inch fillet along the bottom edge, then change the name of the fillet to **<u>OUTER SURFACE</u>** in the specification tree.

Thicken *OUTER SURFACE* 0.0625 inches to the inside, then hide the *PartBody*.

Buttons and Wheel

Insert a new geometrical set named **Buttons and Wheel**. For organizational purposes, the remaining reference geometry will be created in this new set.

Offset OUTER SURFACE inward 0.025 inches, then change the name of the new offset to <u>INNER SURFACE</u> in the specification tree and hide OUTER SURFACE.

Intersect *INNER SURFACE* with the zx plane.

Create a parallel curve from the intersection curve 0.0125 inches away on both sides. Ensure that *INNER SURFACE* is the support. AD

Intersect the yz plane with the intersection curve shown above, keeping only the top point. 🔊

Create a geodesic point on the intersection curve that is 9.75 inches away from the intersection point you just created.

Ensure that the point is between the nose of the mouse body and the intersection point. The parallel curves have been hidden in the following image.

Create a plane normal to the intersection curve at the last point created, then change the name to <u>CHANNEL PLANE</u> in the specification tree.

Split the intersection curve with *CHANNEL PLANE* and *CHANNEL SURFACE*, then **change the name of the split to** <u>CHANNEL CURVE</u> in the specification tree. The split curve is shown below.

Create a midpoint on CHANNEL CURVE.

Create a plane that is tangent to *INNER SURFACE* at the midpoint on *CHANNEL CURVE*. The tangent plane, the midpoint, INNER SURFACE, and CHANNEL CURVE are shown below.

Make the PartBody the in-work object, then create the following positioned sketch onthe new plane.Image: Use the midpoint for the Origin and the y axis for the Orientation,then reverse the V Direction.

Pad the sketch upward 0.0625 inches and use *Up to next* for the *Second Limit*, then hide the *PartBody*.

Activate the Buttons and Wheel geometrical set.

Split INNER SURFACE with both parallel curves, CHANNEL PLANE, andCHANNEL SURFACE, keeping the smaller, inner portion.Image: Superior Content of the second secon

Create a boundary element on both edges indicated above.

Create a linear sweep with a zero degree angle that is 0.05 inches long on both

boundary elements, using the zx plane as the reference surface. Set Ensure the

direction of the sweep is outward. The split surface and both linear sweeps are shown below.

Join the split surface and the linear sweeps shown above, then change the name of the join to <u>TOP CHANNEL</u> in the specification tree.

Hide *TOP CHANNEL*, then show *INNER SURFACE* and intersect it with *CHANNEL PLANE*.

Create a parallel curve from the intersection curve that is 0.0125 inches away on both sides. Sides. Ensure that *INNER SURFACE* is the support. The intersection curve, both parallel curves, and *INNER SURFACE* are shown below.

Split INNER SURFACE with the new parallel curves and CHANNEL SURFACE, **keeping the top portion.** The result is shown here.

Create a boundary curve on both edges indicated above.

Create a linear sweep with a zero degree angle that is 0.05 inches long on both **boundary elements, using** *CHANNEL PLANE* **as the reference surface.** Sensure the direction of the sweep is outward. The split surface and both linear sweeps are shown below.

Join the split surface and the linear sweeps shown above, then change the name of the join to <u>CROSS CHANNEL</u> in the specification tree and hide it.

Show CHANNEL SURFACE, then offset it downward 0.025 inches.

Show INNER SURFACE, then trim CHANNEL SURFACE and its offset with INNER SURFACE, keeping only the inside portion of INNER SURFACE. The result is shown below.

Split the newly created trim with *OUTER SURFACE*, keeping the inside portion, then change the name of the split to <u>SIDE CHANNEL</u> in the specification tree.

Show TOP CHANNEL and CROSS CHANNEL from the tree.

Trim *SIDE CHANNEL*, *TOP CHANNEL*, and *CROSS CHANNEL* together, then change the name of the trim to <u>COMBINED CHANNELS</u> in the specification tree.

Split the *PartBody* **with** *COMBINED CHANNELS*, **keeping the inside portion of the solid.** \bigcirc Ensure the *Extrapolation type* is set to *Tangent*. The *PartBody* is shown here.

Create a plane that is offset downward from the last plane by 0.15 inches. Be sure to activate the *Buttons and Wheel* geometrical set.

Project the pad's sketch to the new plane.

Create an axis line through the minor axis of the projected, elongated hole.

Create a parallel curve to the inside of the projected, elongated hole that is 0.1 inches away.
Use the offset plane as the support.

Split the projected, elongated hole with the axis line. 🐬 The offset plane, the axis line, and the split projection are shown below.

Create a 360 degree groove with the split projection curve and the axis line.

Create a parallel curve to the inside of the split projection that is 0.025 inches away, then change the name of the parallel curve to WHEEL PROFILE in the specification tree. \bigcirc Use the offset plane as the support. The parallel curve is highlighted below.

Create a 360 degree shaft from *WHEEL PROFILE* and the axis line.

Use WHEEL PROFILE to create the following positioned sketch on the offset plane. The offset plane is the *Planar support*, the midpoint previously created is the *Origin*, and the y axis is used for the *Orientation*. Reverse the directions as necessary. In the image below, the *PartBody* is cut at the sketch plane for clarity.

Note: Show the PartBody if the sketch is not visible.

Create a groove that is two degrees in both directions using the new sketch and the axis line.

Create a 360 degree circular pattern of the groove with 50 instances. We the axis line for the *Reference Direction*.

Fillet the edges indicated below with the given values.

Your finished model should look like this.

