

# **EP1000** 3D Models



### **Essential Tools in Fusion 360**

- From Autodesk Training
  - Introduction to Fusion 360
  - User Interface Overview
  - <u>Open, close, export,</u> <u>upload, and save</u> <u>designs</u>
  - <u>Set Preferences</u>
  - <u>Adjust Display settings</u>
  - Use the Marking Menu
  - <u>Use the Toolbox</u>

- <u>Create A Project</u>
- <u>Open a Design created</u> <u>in another CAD system</u>
- <u>Components and Bodies</u>
- <u>Parametric vs. direct</u> <u>modeling</u>
- <u>Working with Design</u> <u>versions</u>
- <u>Sketch Constraints</u>



# Fast Track for Engineers

- Kevin Kennedy <u>Product Design Online</u>
  - Recommended: Learn Fusion 360 in 30 days
  - Highlighted Topics
    - <u>Navigating the Fusion 360 User Interface (sections explained) -</u> <u>REVISED 2019</u>
    - Default settings for Fusion 360
    - <u>How to Manually Add Sketch Constraints Learn Autodesk</u> <u>Fusion 360 in 30 Days: Day #16</u>
    - How to Create text in Fusion 360
    - How and Why to Fully Constrain Your Sketches



# Fusion 360 Building Blocks

#### Sketch

- Created in a 2D plane
- Sketches should be constrained and closed
- Forms the building block of all models
- Body
  - Usually created from a sketch(s)
  - Is a SOLID
  - Can combine to form other bodies
- Component
  - Made up of bodies and sketches
  - Usually "joined" or "combined"
  - Can be used to form other components





#### Constraints

- Why constrain a sketch?
  - A constrained sketch cannot be changed (accidentally).
  - Each segment is locked by a dimension or a constraint.
  - Constrained segments are drawn in **BLACK**

Kevin Kennedy: How and Why to Fully Constrain Your Sketches



## **Types Of Constraints**

Horizontal/Vertical

\_\_ Coincident

🔿 Tangent

- = Equal
- 🥢 Parallel



- 🔒 Fix/UnFix
- 🛆 MidPoint
- O Concentric
- 🏏 Collinear
- [] Symmetry

🕆 Curvature

- Dimension
- Horizontal, Vertical
- Coincident (constrains a point to another point, line, arc, or curve.)
- Tangent
- Equal
- Parallel
- Perpendicular
- Fix / UnFix
- MidPoint
- Concentric
- Colinear ( constrains a line to another line, so that both lines fall onto the same line)
- Symmetry
- Curvature



#### Saving Designs

BROWSER      BROWSER      D     D     D     D     Named	New Component     Create Drawing     Create Selection Set	Export	×
D ⊕ m si		Name: test v1 Type:	
	Save Copy As Display Detail Control	Autodesk Fusion 360 Archive Files (*.f3d)	•
	Show/Hide V     Show All Components     Show All Bodies	Location: D:/Users/Rodney/Documents	
250	Opacity Control     Find in Window     Find in Timeline     M: Do not capture Design History		
		Cancel	ort

- Fusion 360 saves all files into the cloud
  - You can share your files within the cloud
  - You can export your design file to the local storage using export.
  - Output format is .f3d



#### Parametric Design

Parameter		Name		Unit E	Expression	Value	Comments
	Favorites						
~	User Parameters +						
	☆ User Param	p_diame	ter	mm	40 mm	40.00	Polyvon Diameter
	☆ User Param	p_sides			8	8	Number of polygon sides
	☆ User Param	p_angle		deg	( 360 / p_sides ) * 1 deg	45.0	Polygon angle between vert
1	Model Parameters						
			Name Unit Expression Value Comment	Enter I mm Enter I	Name  Expression  Comment	adding a new	,
				OK	Cancel	parameter	

- Enter variables as parameters
- Use parameters in your design
- Design becomes very flexible



# E.g. Parametric Polygon

- A fully configurable polygon with parametric sides and size.
- Try changing the parameters





# Methods of Creating 3D Models

#### Extrusion

- Use a 2D plane profile
- Extend into the 3rd plane
- Rotation
  - Use a 2D plane profile
  - Rotate the plane around an axis
- Sculpting
  - Start with a 3D object
  - Add, remove 3D objects
  - Subdivide the surface into sections
  - Push, pull, extend, contract sections



#### Extrusion

- Start with a 2D closed profile in plane
- Stop Sketch
- Create > Extrude in 3<sup>rd</sup> axis





# Extrude – Join / New Body

• You can build new bodies by adding/creating the original.





#### Extrude – Cut

• You can cut holes using subtraction to the original..





### Extrude - options

#### Start

EXTRUDE	
100	
Section 1	A contract of
Start	→ Profile Plane
	Profile Plane I → Offset I → Object
Taxa Arapa	1.1.00
Operation	New Body 🔹
0	OK Cancel

Direction

EXTRUDE	
10	
100 C	providence of the local sectors of the local sector
Direction	💫 One Side
international International	One Side
_	
Operation	New Body 🔹
0	OK Cancel

#### Extent Type

EXTRUDE	
100	<u> </u>
_	a second at the
Extent Type	H→ Distance
	H Distance 귀 To Object 남 All
Operation	New Body 🔹
0	OK Cancel





# Exercise 1: Name Tag

- Let's make a name tag
  - dimensions: 30mm x
     70mm x 4 mm
  - rim of 1.5mm thickness around the edges, height 2.5mm
  - key-ring hole of 4mm, reenforced with 1.5mm rim
  - name or design/pattern
     0.5mm below surface
  - base of name tag 1.5mm thick





# Modify > Shell

- Makes a shell of the solid object
- Starts with the face that was selected
- The shell thickness must be specified





### Create > Loft

• Create a solid object from profiles on different planes





# Exercise 2: A Lego brick

• This is Kevin Kennedy's video tutorial on the drawing of a Lego brick. <u>https://youtu.be/6yPKMSb6ja8</u>





### Exercise 3: Extrudes & Planes

• This object is made up of 20 cubes (20mm) glued together and then shelled to a thickness of 4mm





# EP1000 3D Models End