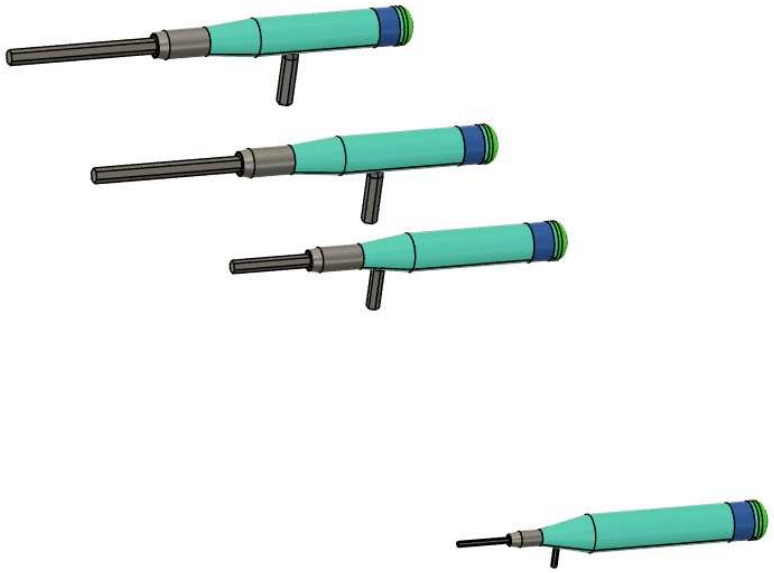
Mike's Fusion 360 Class

Module 3 – Working with parameters

Design a set of 3D printable handles
for your allen wrench set



Module 3 - Parameters

Parameters preamble

- Fusion 360 is a “Parametric Design System”.
- If you create you design using parameters, your can easily change one and the resulting change will rpiile through the rest of your design.
- If you have put a dimension on a sketch, you are already using parameters.

Real example for this module

- 3D printable Allen wrench Handles

Types of parameters we will cover in this class

These are all in the parameters window

- Basic dimensional parameters
- Simple and complex equations in parameters
- User parameters
- Model parameters
- Favorites

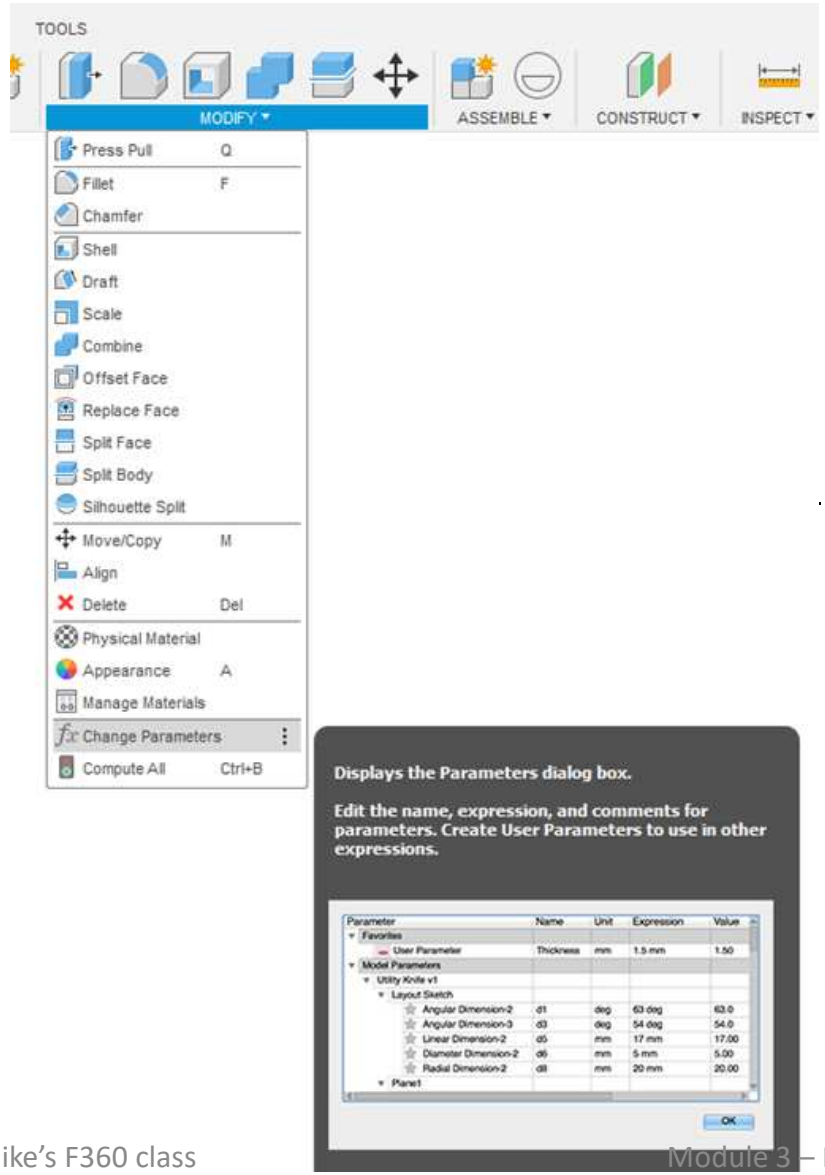
Equations in parameters

- Referencing other dimensions (parameters)
- Algebraic parameters

Functions in equations

- Functions built into Fusion 360

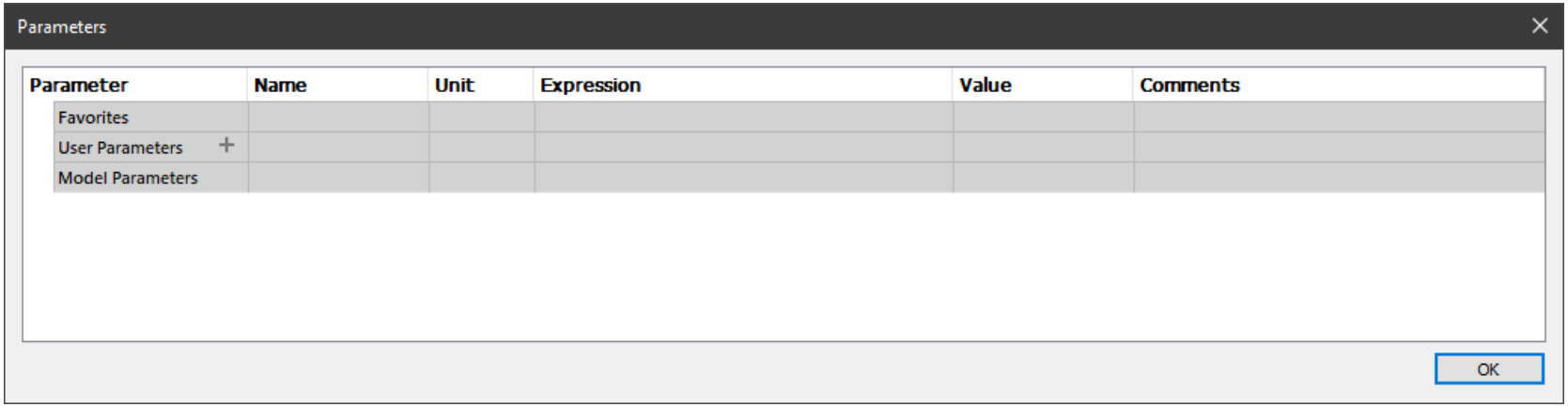
Accessing the Parameters Window



Expand the modify menu and find “Change Parameters”

Class-tip – Pin to toolbar (at least for this class)

The parameters window



Favorites – Parameters you select as favorites.

User Parameters – Parameters you create

- Will appear in popups
- Not tied to a component

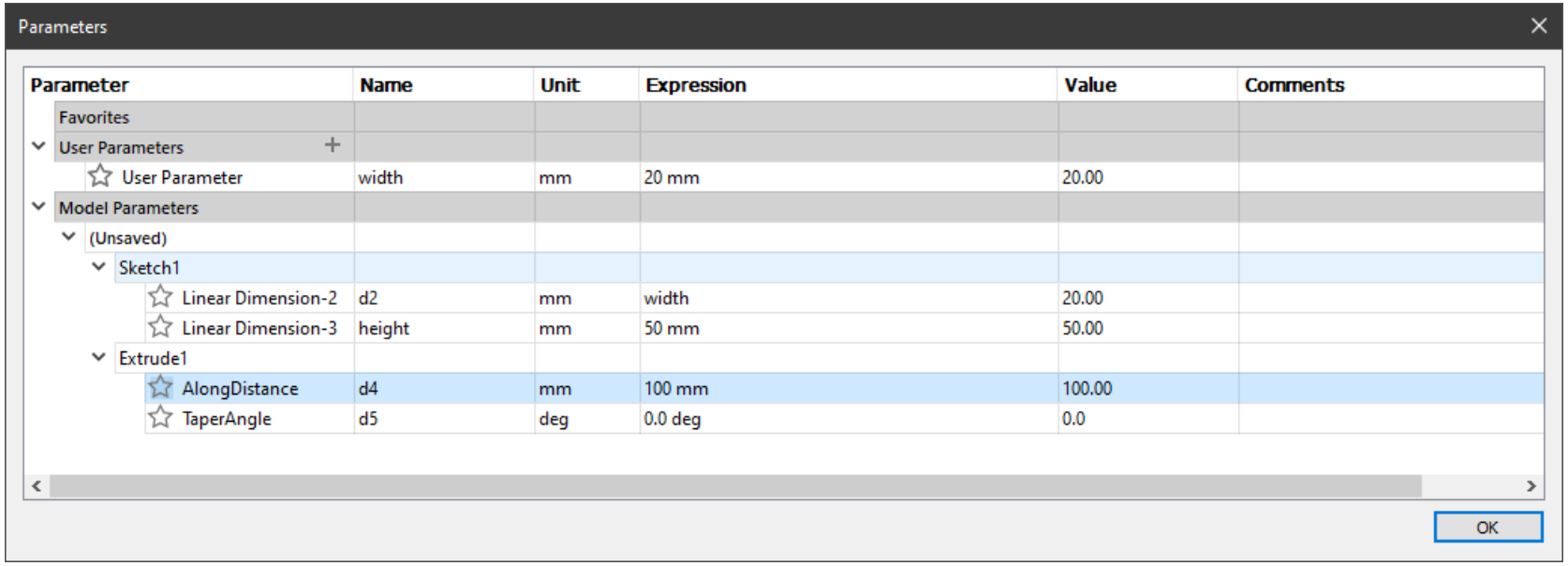
Model Parameters

- Captures the parameters you have not named.
- Sketches and Modeling steps

Watch and Do

1. Access the parameters window
    1. create a parameter called “width”
    2. Value = 20mm
    3. Close the window.
2. Create a sketch
    1. Draw a rectangle.
    2. Dimension the height with “height=50”
    3. Dimension the width - start typing “width” and note that the user parameter pops up in a selection box. Start typing “width” and note that width does NOT show up until you get the whole word.
3. Extrude the rectangle 100mm
    1. Rename the extrusion on the timeline to “My named Extrusion”
4. Radius two edges of the extruded shape
5. Return to the parameter window

Parameter window



- Note where each value showed up.
- Parameters you did not name are named by Fusion - “d1”, “d2”, “d3”, etc.
- Change any value and note the change in the object. (Try using different units)
- Change a name in the window (and try to reuse an existing name)
- Add an equation in the window.

Equations and functions

Equations

- Just type in the equation you want, no "=" needed unless you are assigning a name to a model parameter.
- RED means it's not legal (often a units issue)

Algebraic operators

+	addition
-	subtraction
%	floating point modulo
*	multiplication
/	division
^	power
(	expression delimiter
)	expression delimiter
;	delimiter for multiargument functions

Read about it: https://help.autodesk.com/view/fusion360/ENU/?guid=GUID-76272551-3275-46C4-AE4D-10D58B408C20

Functions in equations

Syntax	Expected Types	Return Type
cos(expr)	angle	unitless
sin(expr)	angle	unitless
tan(expr)	angle	unitless
acos(expr)	unitless	angle
asin(expr)	unitless	angle
atan(expr)	unitless	angle
cosh(expr)	angle	unitless
sinh(expr)	angle	unitless
tanh(expr)	angle	unitless
sqrt(expr)	any	unit^1/2
sign(expr)	unitless	Any, Return 0 if negative, 1 if positive
exp(expr)	unitless	Any, Return exponential power. For example, returns 2.688E43 for 100.
floor(expr)	unitless	ul Next lowest whole number
ceil(expr)	unitless	ul Next highest whole number
round(expr)	unitless	ul, Closest whole number
abs(expr)	any	any
max(expr1;expr2)	any	any
min(expr1;expr2)	any	any
ln(expr)	unitless	unitless
log(expr)	unitless	unitless
pow(expr1; expr2)	any, and unitless	unit^expr2
random()	unitless	unitless

Paste vs Paste New

	Paste	Paste New
Basic behavior	Creates another instance of the copied component. Edits to either will change both.	Creates a new component based on the copied component. Changes to one do not affect the other.
Model Parameters		
User-Named or Fusion-Named parameters	Only one model parameter section representing all instances of the component.	New section created for new component. New name create with "_N+1" appended to name, where N is the last used numeral.
User parameters	Changing the parameter affects all instances	No new parameter created. Changing the parameter changes the copied from and the Paste New component.

All this is useful because it lets you have "copies" that have some parameters in common and others that are unique to the copy. Note: this is not because Model parameters can be named, that's just a convenience.

Tidbits

A warning

- Changing a parameter can alter a sketch if the dimension becomes invalid with a new parameter value. Same as if you change the dimension on a sketch. But this can change when the sketch is not displayed, so beware.
  - Lines can disappear
  - circles and arcs can disappear

Deleting User parameters

- F360 will not let you delete a user parameter if it is used somewhere. If it is, it will tell you the name of the dimension where it's used, but will not tell you where it is.

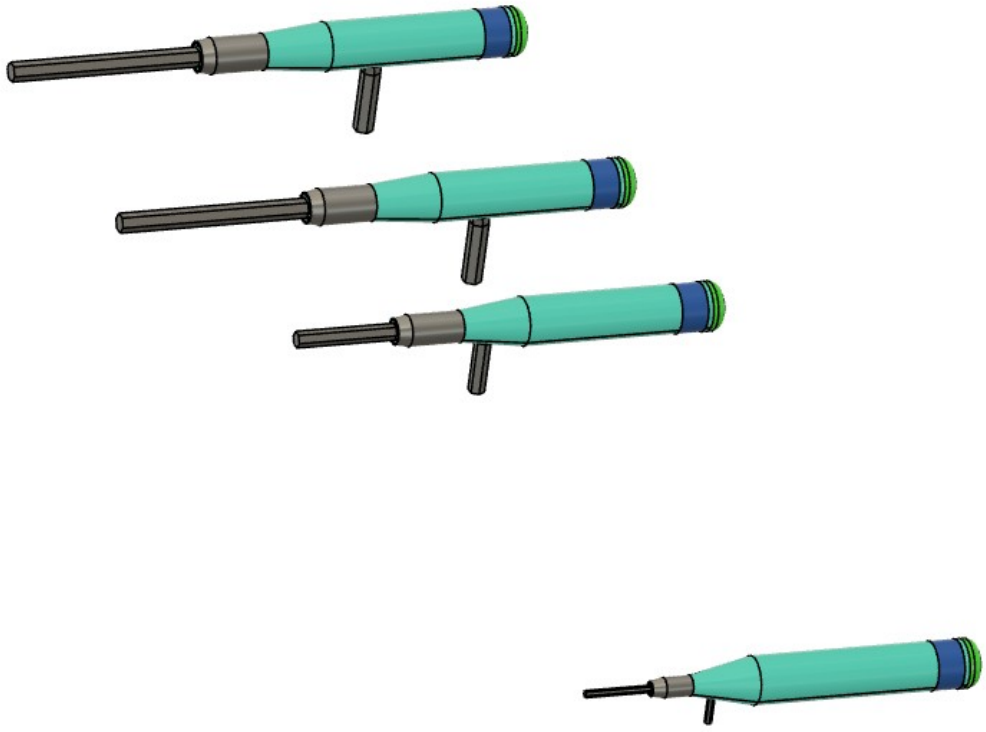
Named parameters and the timeline

- Once a named parameter is in the parameters table, it will stay there even if you roll the timeline back before when you named the parameter. With this knowledge you can "cheat the timeline" and use a dimension that has not "been created" yet.

Parameter Names can't have spaces in them

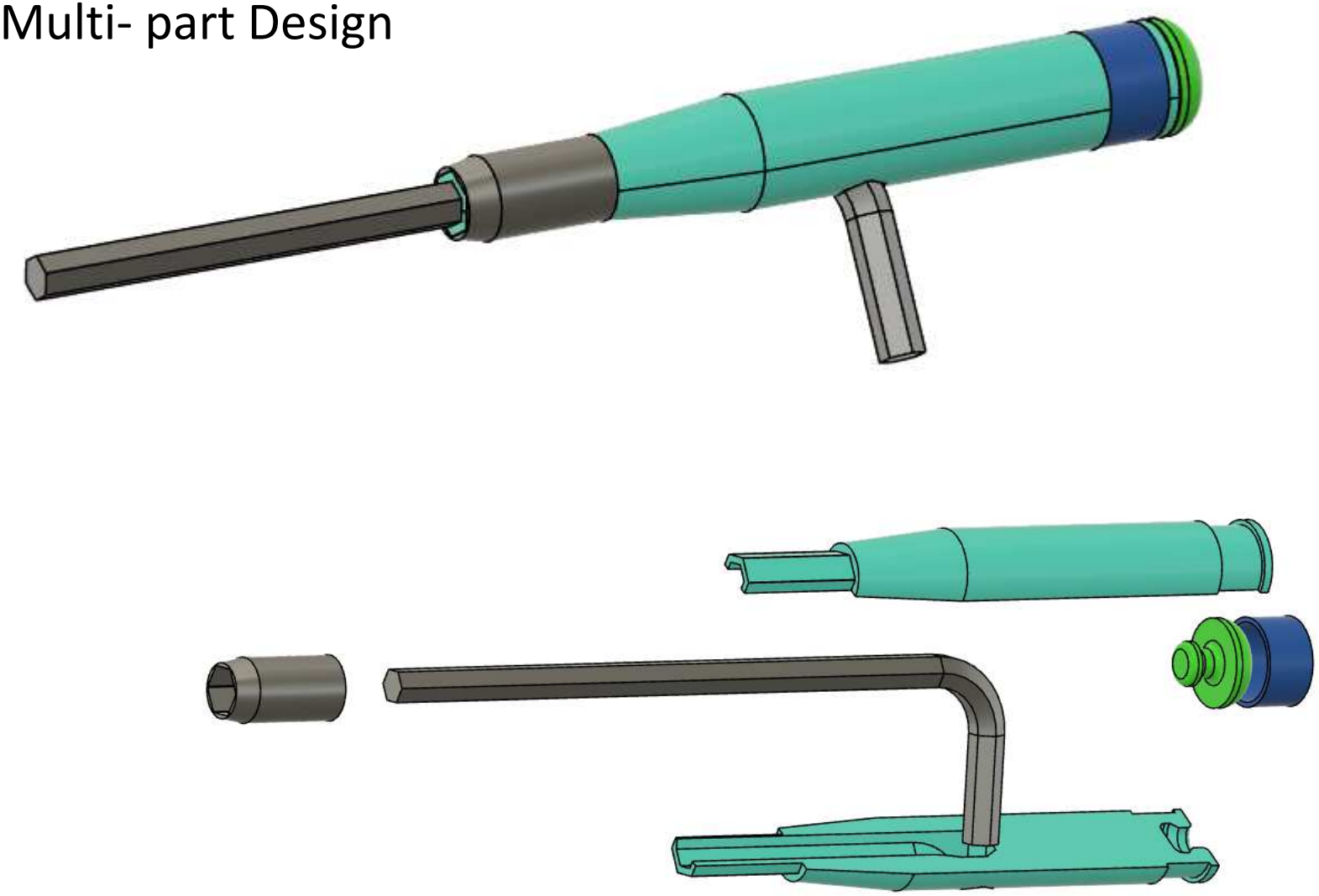
Design Example - Allen Wrench handles

Design goal – a set of 3D printable handles for cheapo allen wrenches. Make a family of parts.



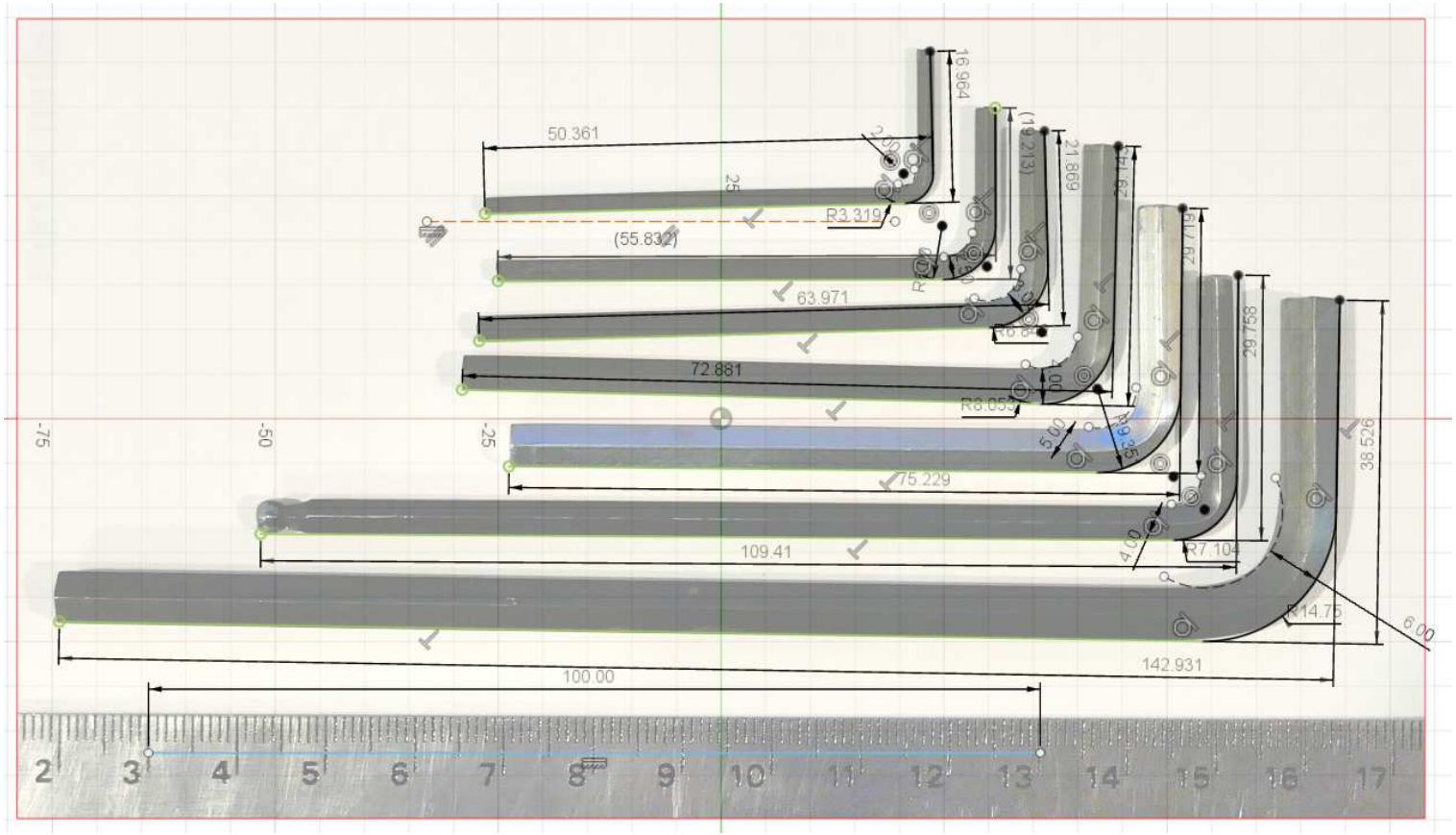
Design Example - Allen Wrench handles

Multi- part Design



Collection of wrenches from my bench

A Scaled Canvas and a sketch



After class practice

- Add handles for the remaining wrenches
- Make a set of handles for your own wrenches
- Add some grip features on the barrel of the handle
- Make the handle diameter and lengths functions of the wrench size
- Take a picture of your printed handle set and email it to me.