TEXAS A&M UNIVERSITY
ROBOTICS TEAM & LEADERSHIP EXPERIENCE

SolidWorks Assembly

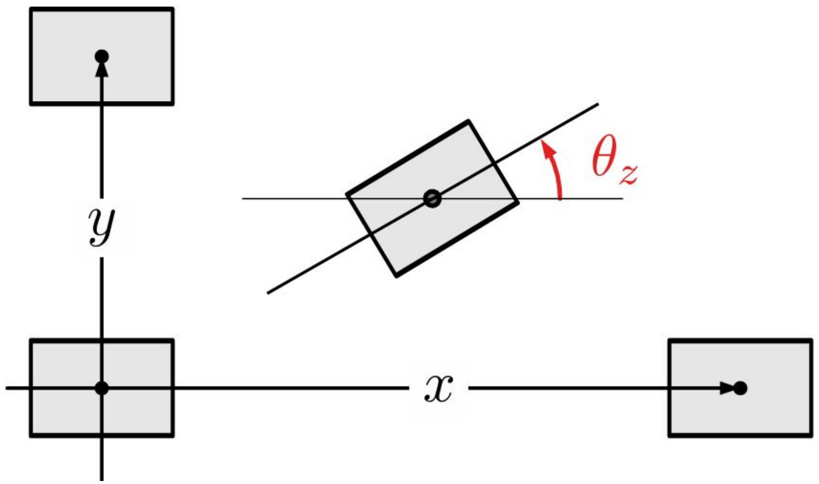
TURTLE Hatchling

Sketches

Sketches are 2D “drawings”

A plane has 3 Degrees of Freedom:

- X (Translational)
- Y (Translational)
- θ(z) (Rotational)



engineeringstatics.org

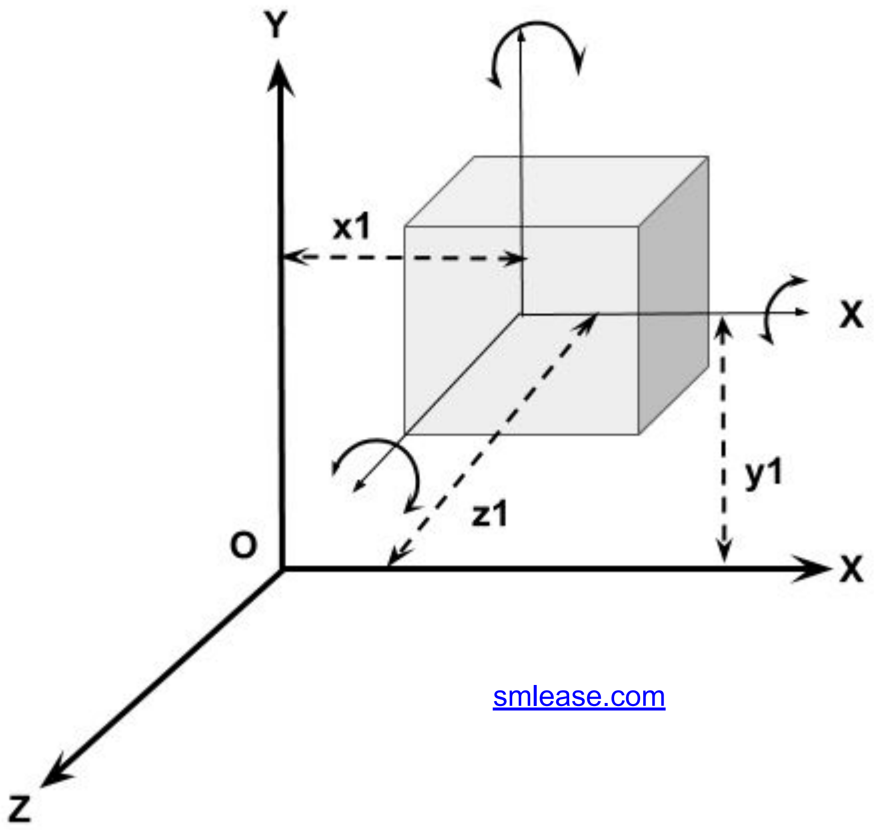
An ideal sketch will have one enclosed region and fully defined (cannot move) in space

Assemblies

Assemblies are 3D

3D has 6 Degrees of Freedom

- X (Translational)
- Y (Translational)
- Z (Translational)
- θ(x) (Rotational)
- θ(y) (Rotational)
- θ(z) (Rotational)



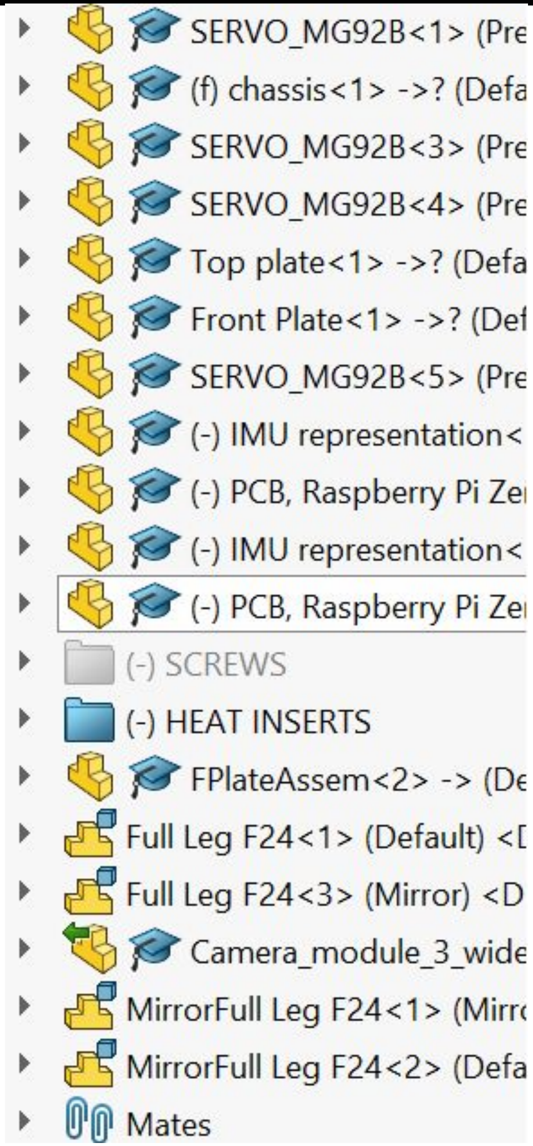
smlease.com

Moving mechanism may result in under defined assemblies

Common Symbols

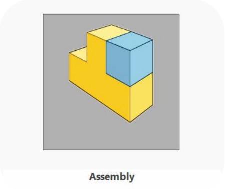
Defined States are shown in the feature tree

- (f) - Fixed component
  - Should have one of these in an assembly
- (-/+) - Under/Over defined
- (?) - Cannot solve
- Nothing - Fully defined
-  - Education license tag

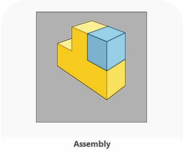


Creating an Assembly

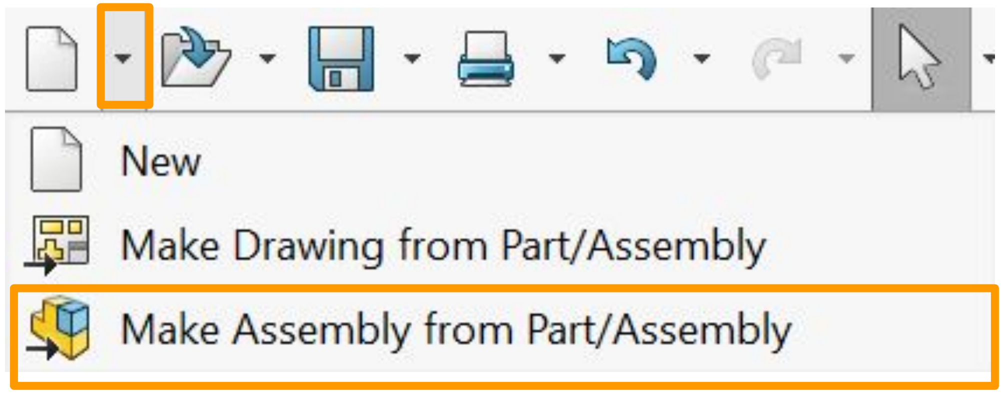
Method 1:

1. Click “New” 
2. Click “Assembly” 

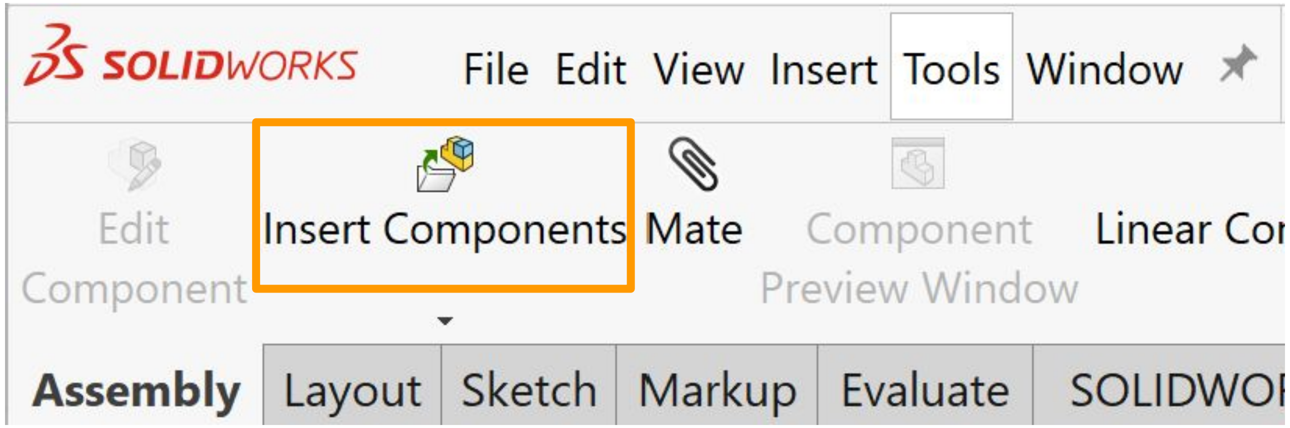
Method 2: From a part/assembly

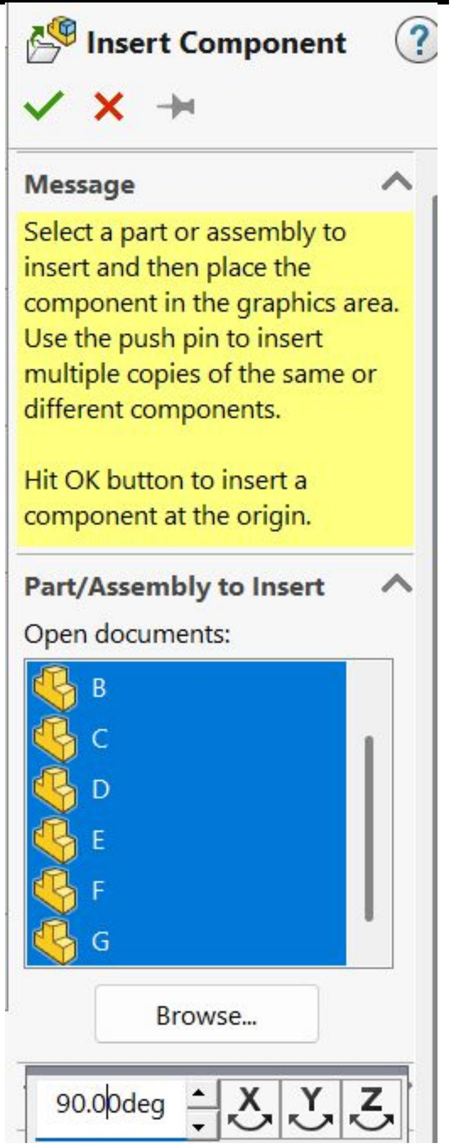
1. Click down arrow on “New”
2. “Make Assembly from part/assembly”
3. Click “Assembly” 

*Method 2 automatically aligns origin and “fixes” the part



Inserting Components





Go to *Assembly Tab > Insert Components*

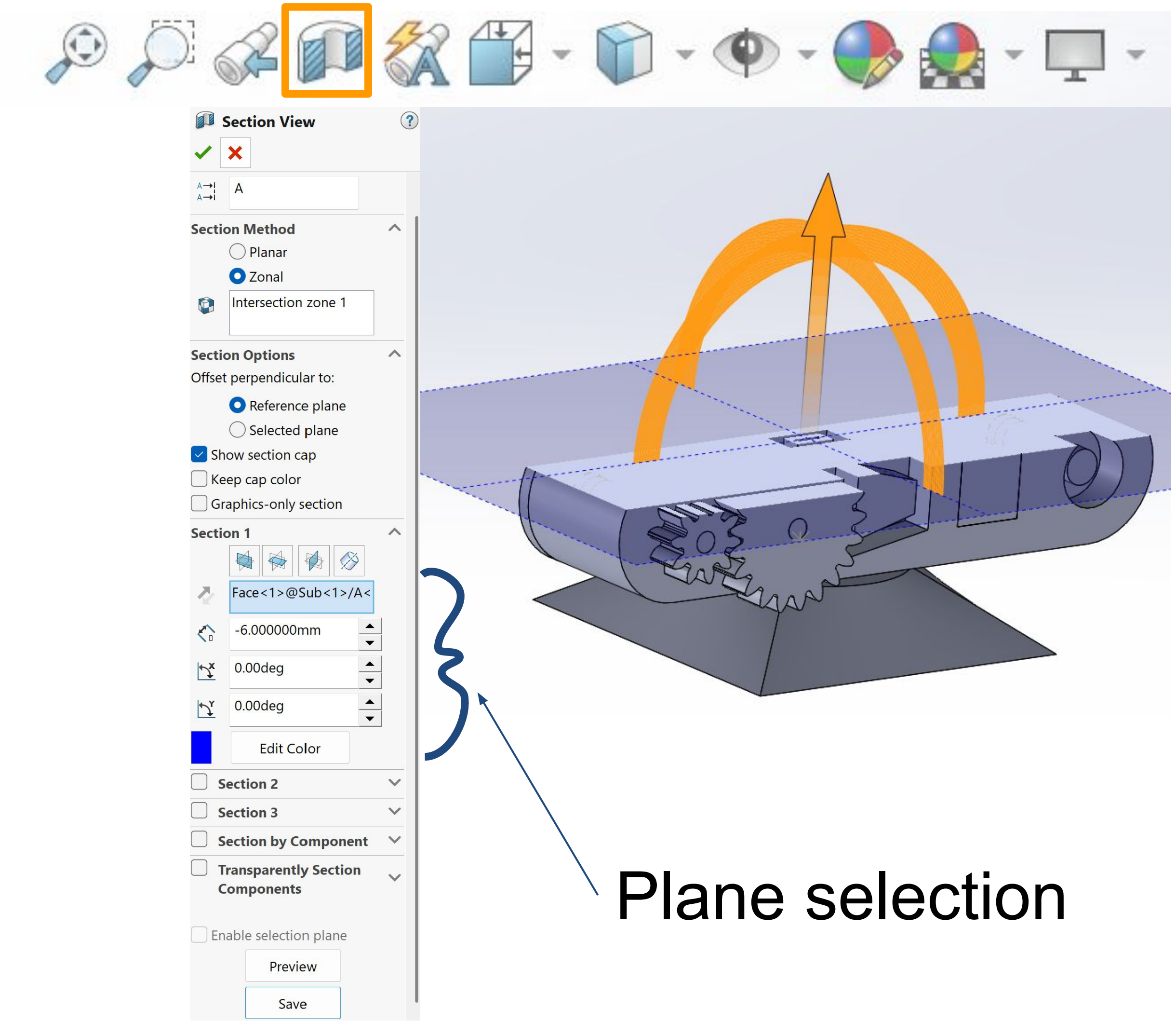
From here, open parts can be added or browse for saved files

From the drop down, new parts can be created, or parts can be inserted from Part Supply

Section View

Allows you to “cut” an assembly/part and look at difficult to reach places

*Faces visible through section view are temporary You cannot use them for functional purposes



Plane selection

Helpful Assembly Shortcuts

****You can customize shortcuts for your workflow****

“q” : Shows all planes
“f” : Orients view to fit the model
“Ctrl + t” : Flat Feature Tree
“Shift + c” : Fully Minimizes Feature Tree
Feature Tree Search “*” : Fully Expands Feature Tree
“Spacebar” : Allows Viewing Orientation Selection

Mates

Three categories of mates:

- Standard - Used a majority of the time
- Advanced
- Mechanical

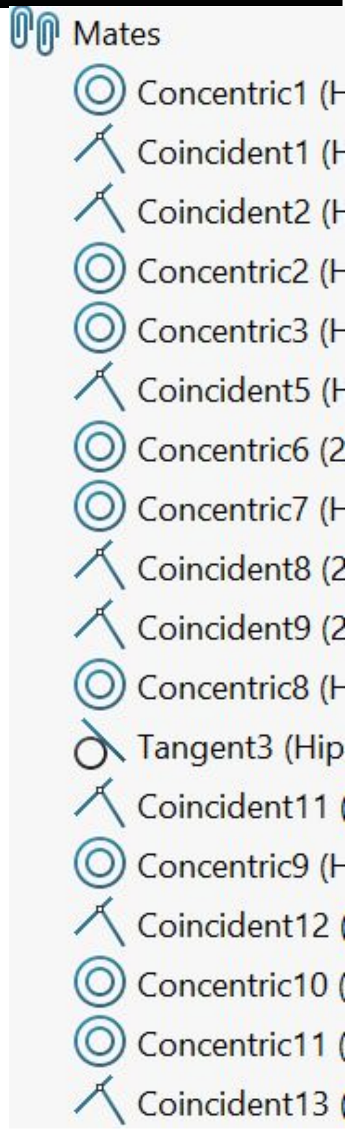
Entity Priority:

1. Plane/Face
2. Edge
3. Point

How to use:

1. Click “Mate” 

2. Select mate type
3. Select mating entities

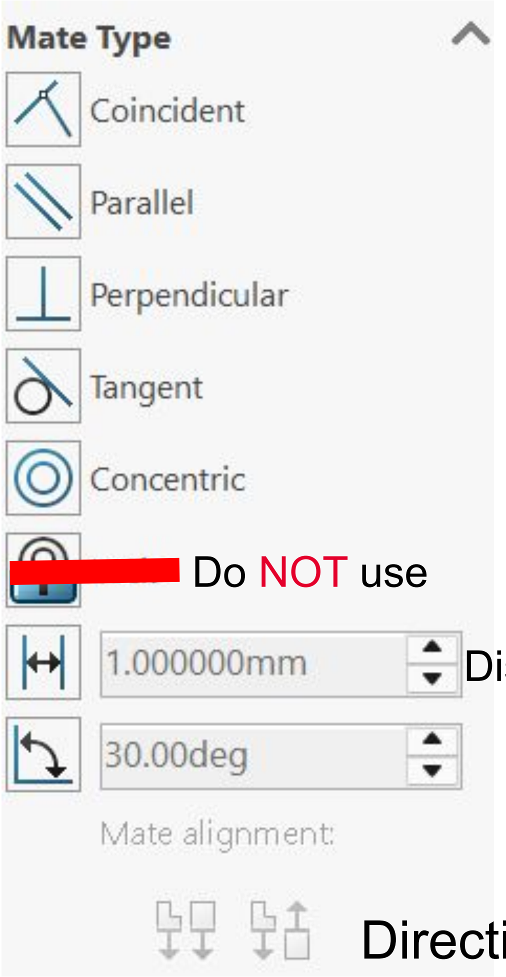


How should you add mates?

Technically best practice is to mate as you would assemble the parts in real life

It is much easier and faster to mate all of one component before moving on to the next component

Standard Mates

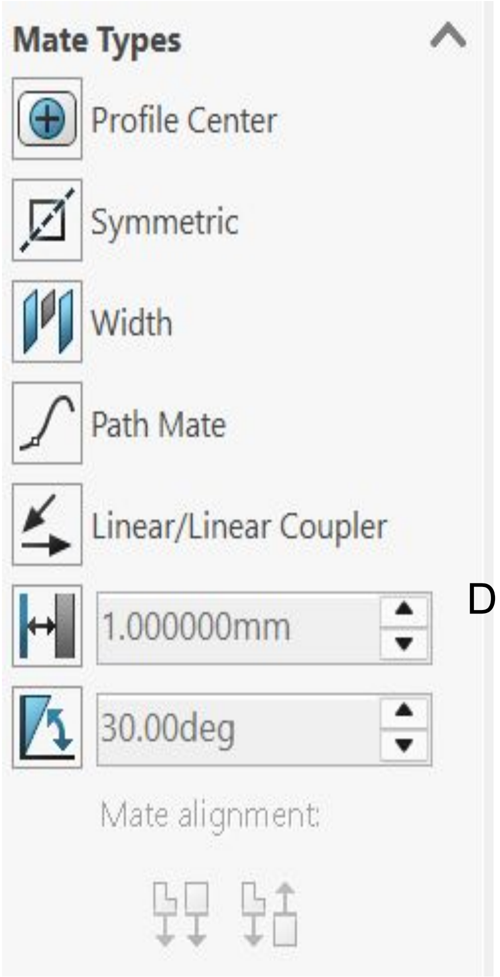


These should look familiar

Identical to sketch relations but can be used in 3 dimensions

Switching the mate direction may flip other mate directions as well

Advanced Mates



Distance Range

Angle Range

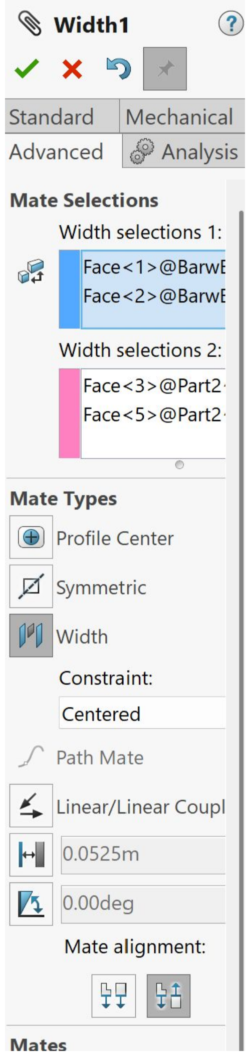
Fancy but often computational complex

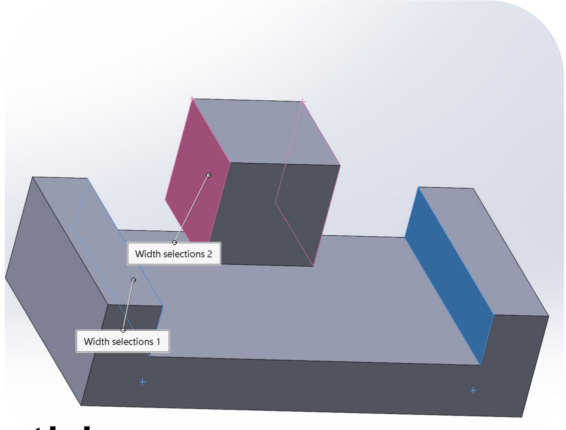
SolidWorks has no collision detection

Ranges can be used to simulate collision constraints, however, this can cause funky CAD. Would recommend using configurations for various positions.

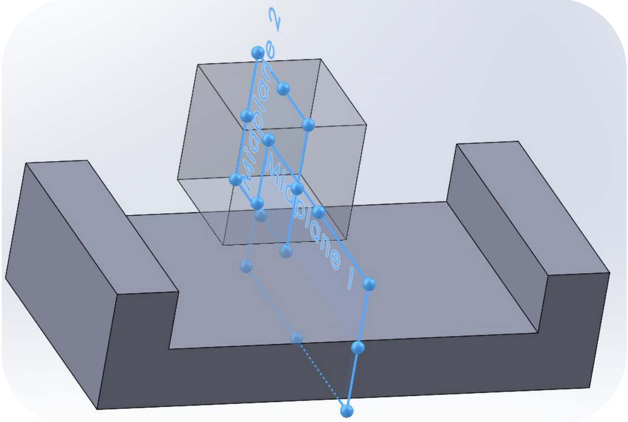
Width Mate

Creates an imaginary midplane between two selected faces on two parts and makes them coincident





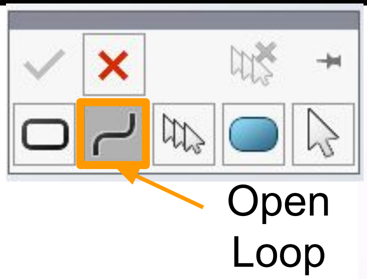
Same thing as



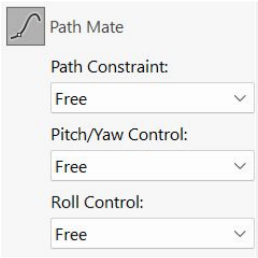
Path Mate

Requires 2 Sketches:

1. A sketch with a "following" point
2. Construction Guide Curve



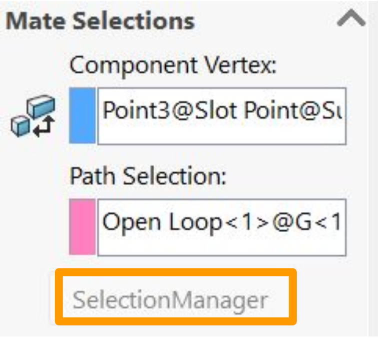
*If your path is multiple connected line segments. you must use "Selection Manager" and "Select Open Loop"



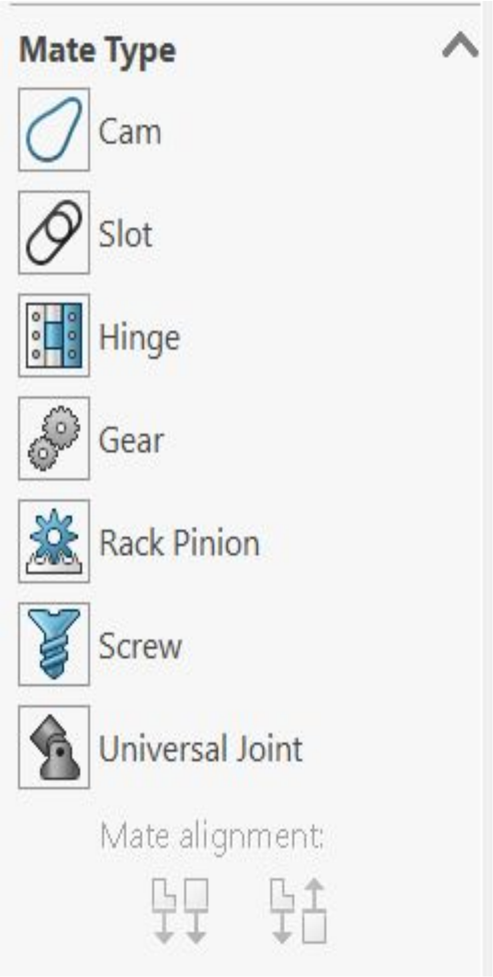
Can control location and orientation within mate

1

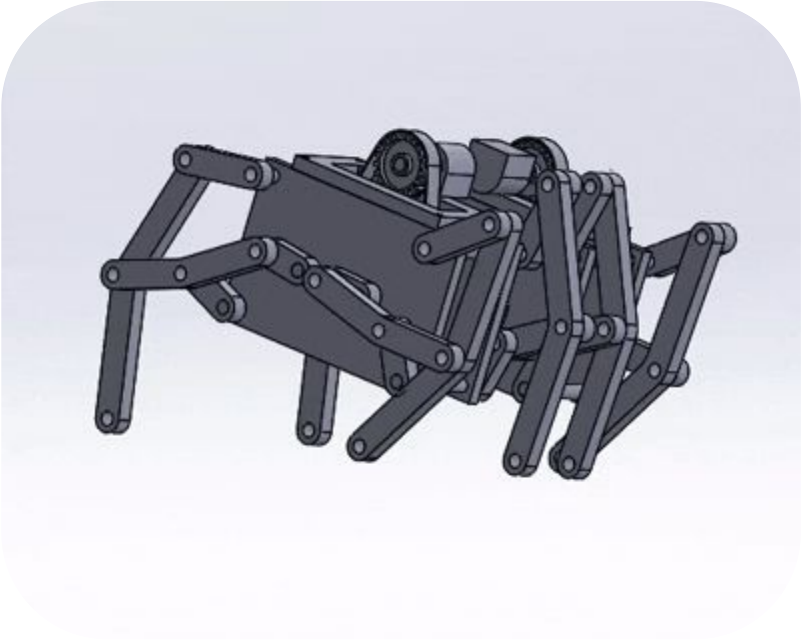
2



Mechanical Mates

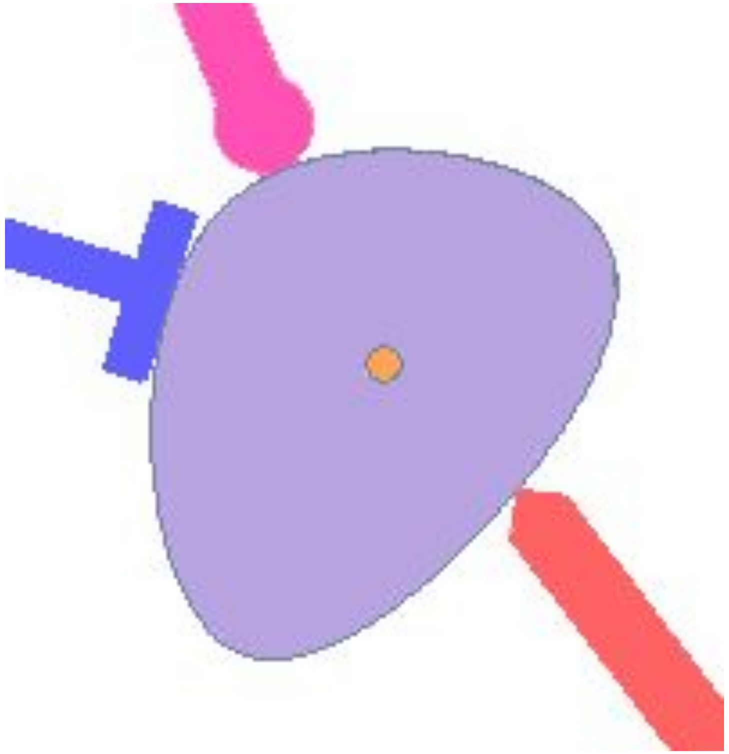


These allow you to simulate real-world mechanical movements between components



Cam Mate

This mate constrains a follower component to stay in contact with the cam component as it rotates



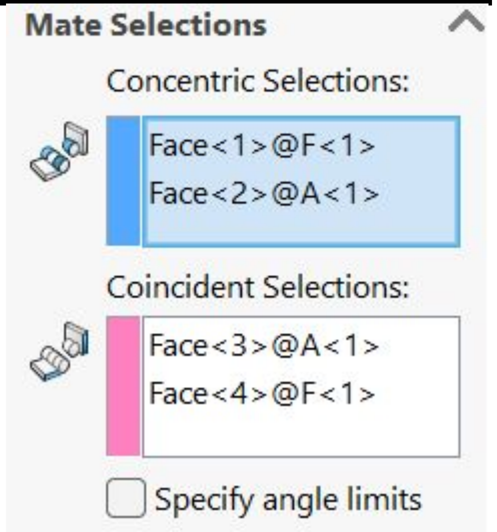
Video: Cam-Follower Mate - 2021 - SOLIDWORKS Help

Hinge Mate

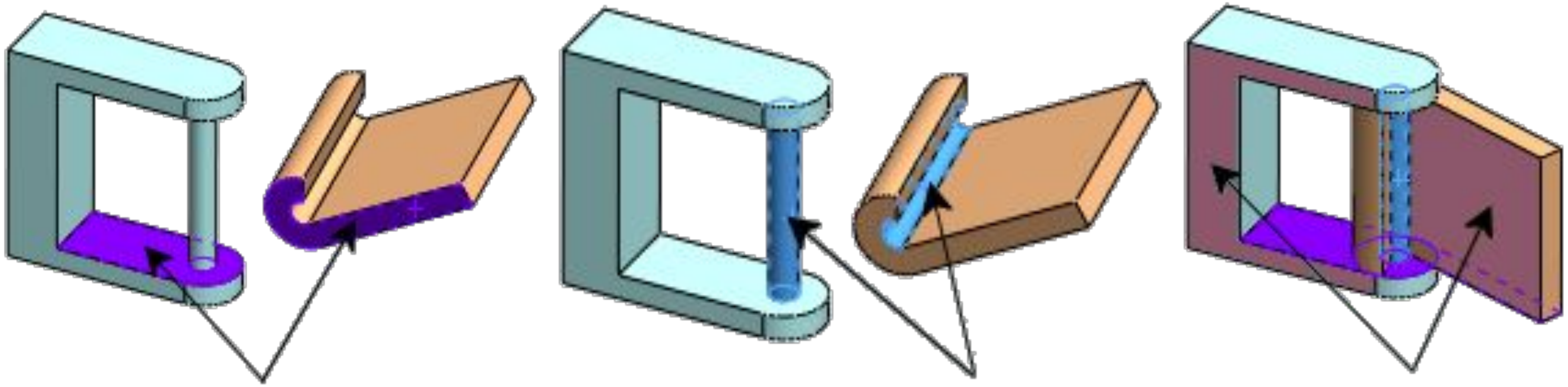
Combines a “Coincident” and “Concentric mate into one

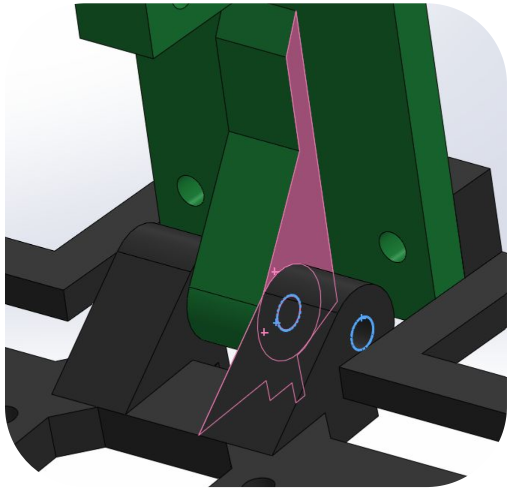
Same as “Concentric”

Same as “Coincident”



Can also add an “Angle Range” mate



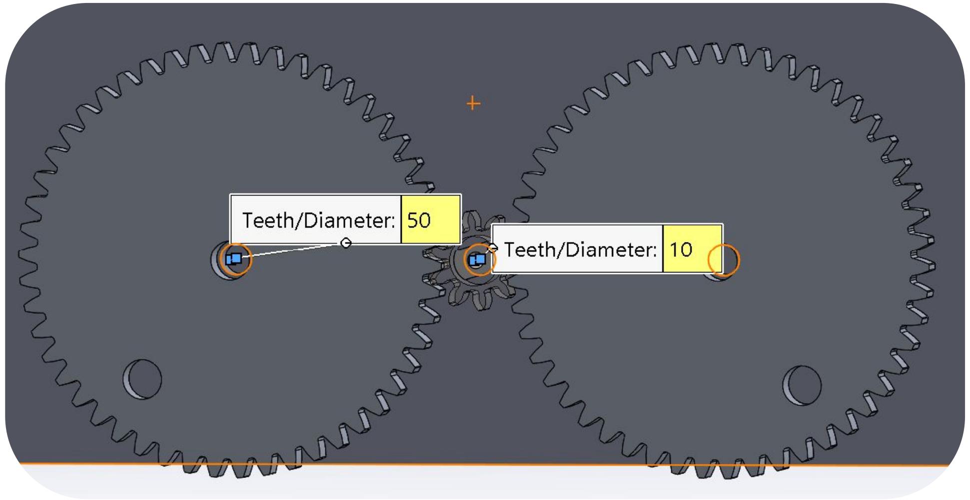


[Hinge Mates - 2021 - SOLIDWORKS Help]

Gear Mate

Requires two sketches

1. Construction Circle on Gear 1
2. Construction Circle on Gear 2

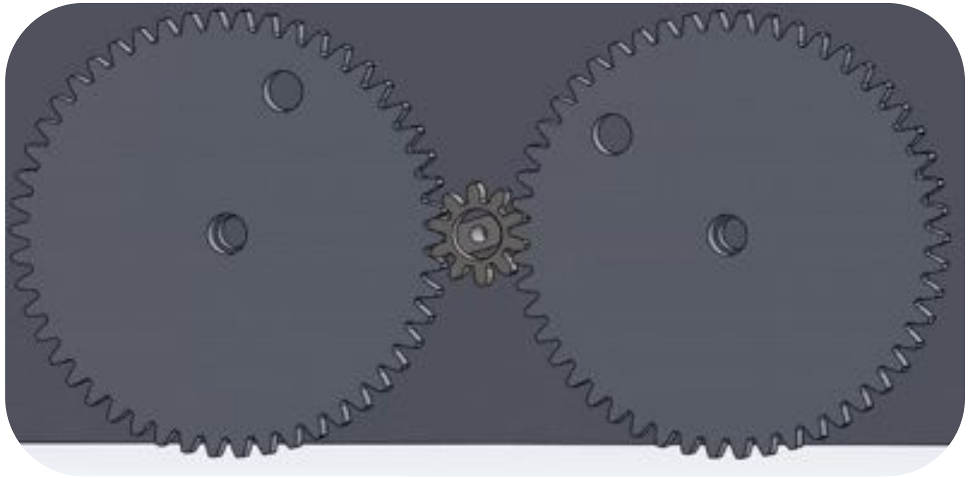


*Best practice is to use the corresponding pitch circle for mating

**Teeth will not automatically align

Aligning teeth options:

- Collision detection
- Temporary “For Position” mate



Sub-Assemblies

- You can add assemblies to assemblies
  - You cannot add an assembly to itself
- Mates within sub-assembly do NOT calculate
  - This is **essential** for large (100+) component assemblies
  - Think modularity. Every module should be a sub-assembly
- Flexible sub-assemblies do calculate internal mates
  - Only use if you absolutely must 

External References

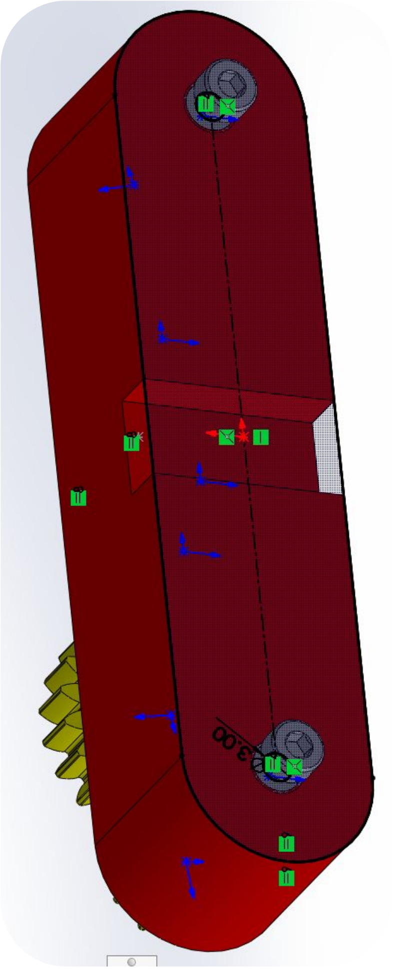
Purpose:

- Automatically update dependent parts when changes occur
- Relations between components based on geometry or features from other parts, assemblies, or drawings

Created by:

- Editing a component in an assembly and defining geometry based on another component

Note: These can break CAD models quickly, best practice is to remove external reference on finished parts*

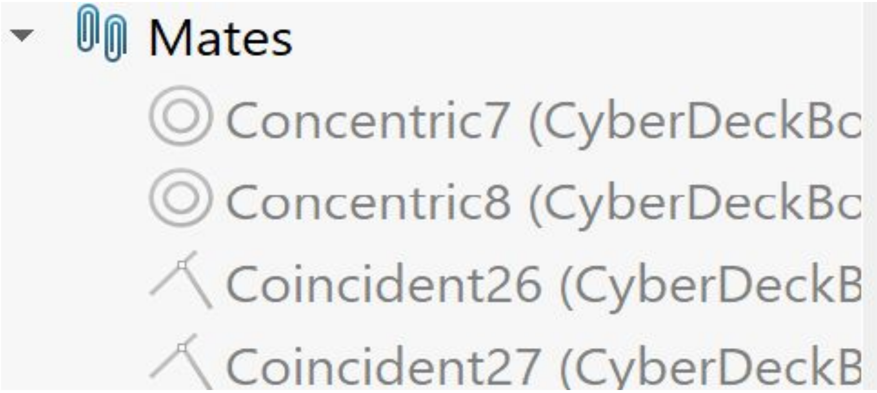


{ - >} indicates external reference

Suppressed State

Temporarily removes a component from the assembly without deleting it

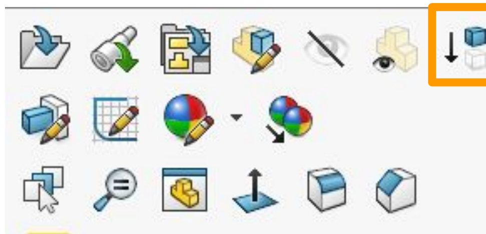
- No longer part of the parametric equations



**Also suppresses the component's mates

Advantages:

- Allows configurations with added features
- Simplifies assemblies and reduces computational load
- Easily reversible



Right Click Part

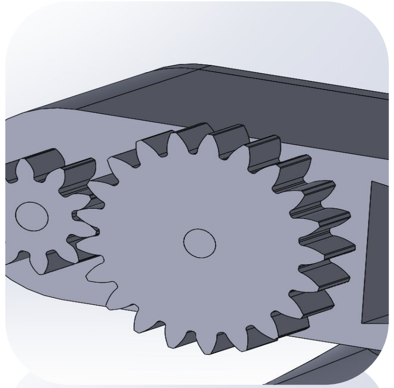
Hidden vs Transparent

Hidden:

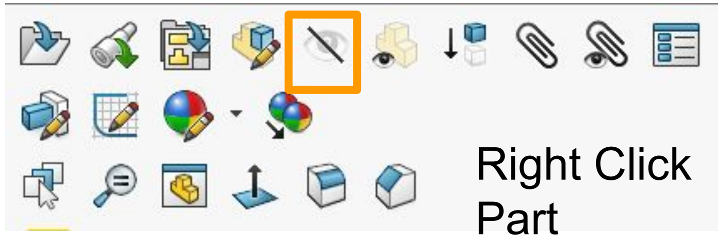
- Makes the component invisible
- Must unhide from feature tree to use again
- Used to focus on internal components

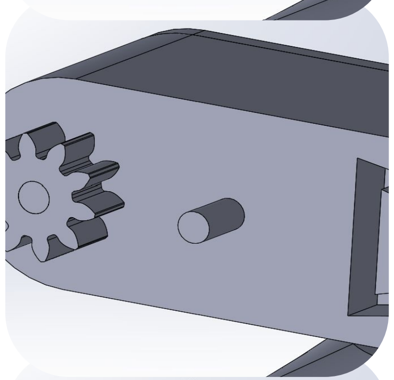
Transparent:

- Makes the component… well transparent
- Edges are still selectable
- Used to visualize or reach internal components

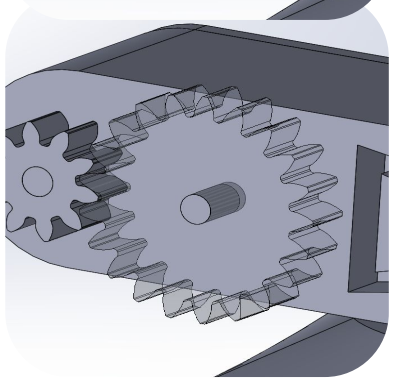


Normal

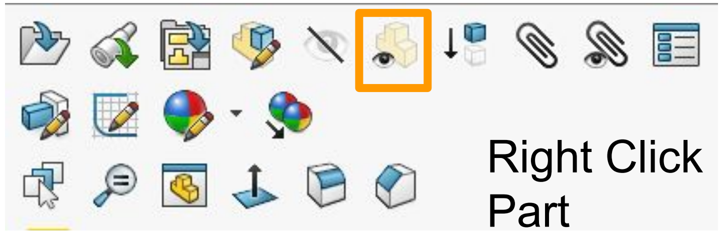




Hidden



Transparent



Configurations

Used to create a multiple variations of a part or assembly within a single file

Advantages:

- Prevents modeling same thing twice
- Allows design table parametric modelling
- Reduces the number of files you need

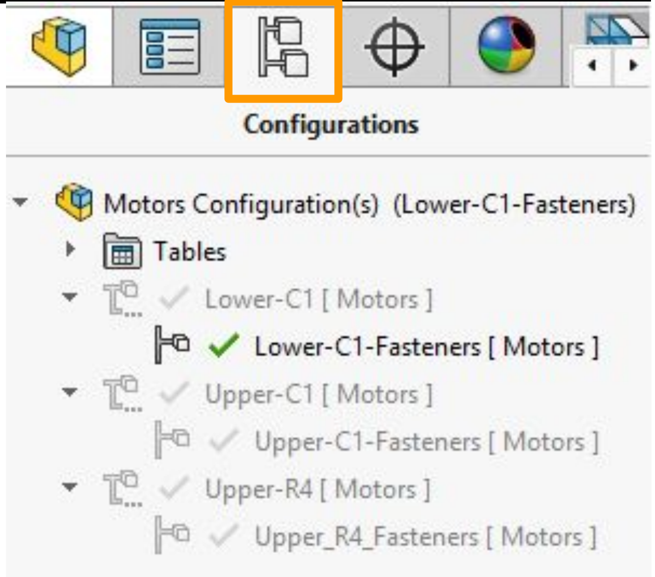
Mcmaster M3 Screw

Think about screws. One CAD model works for every length variation

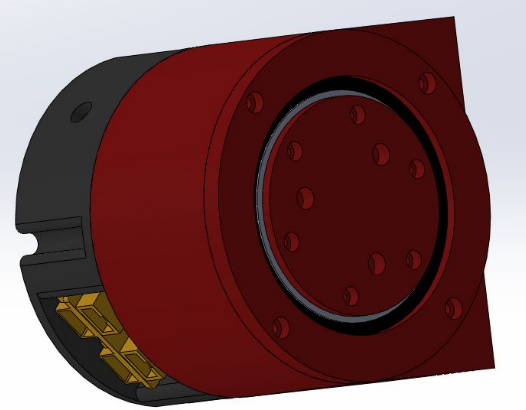
More on Configurations

How to Use:

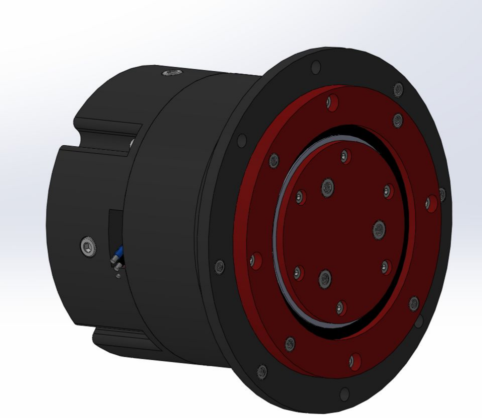
1. Switch to “Configurations” tab
2. Right click then “Add Configuration”
3. Double Click to switch between configurations

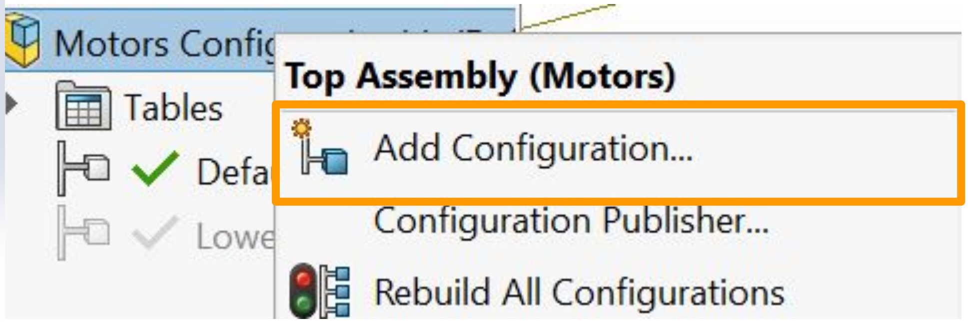


*Green Check is the configuration you are in



Configurations work on both parts and assemblies





Design Table

Implementation of Excel in SolidWorks for parametric modeling
More powerful version of “Global Variables”

Sketch Dimensions

*We do not have time to teach this but it is worth learning

Configurations

Design Table for: Involute Gear, Metri	Tooth Width@Sketch2	Root Fillet@Root Fillet	Tip Radius@Tip Radius	$STATE@Hole	HoleDia@Sketch6	$LIBRARY:MATERIAL@Involute Gear, Metric Spur, DT 2021	PressAngle_Measured@Sketch2	Outside_Diameter@Sketch2	Root_Diameter@Sketch2	Tooth_Depth@Sketch2	$PRP@UNIT_OF_MEASURE	Shaft Dia@Sketch5		Module	Pressure Angle	Tooth Form	Outside_Diameter@Sketch2	Root_Diameter@Sketch2	Pressure Angle Error, Decimal Degrees	Pressure Angle Error, Seconds
1m, 25 Deg, 10T, Standard, Center Ho	1.571	0.35	0.13	U	3	SOLIDWORKS Materials:ABS PC	25.0025	12.000	7.500	2.250	- none -	10		1.0	25	Standard	12.000	7.500	-0.00252	-9.07
1m, 25 Deg, 21T, Standard, Center Ho	1.571	0.35	0.13	U	3	SOLIDWORKS Materials:ABS PC	25.0001	23.000	18.500	2.250	- none -	14		1.0	25	Standard	23.000	18.500	-0.00007	-0.27
0.8m, 20 Deg, 24T, Stub	1.257	0.28	0.10	S	11	OLIDWORKS Materials:6061 Allo	19.9999	20.480	17.600	1.440	- none -	14		0.8	20	Stub	20.480	17.600	0.00011	0.39
0.8m, 20 Deg, 33T, Standard	1.257	0.28	0.10	S	15	SOLIDWORKS Materials:Brass	20.0002	28.000	24.400	1.800	- none -	20		0.8	20	Standard	28.000	24.400	-0.00016	-0.57
0.8m, 25 Deg, 24T, Standard, Center H	1.257	0.28	0.10	U	10	SOLIDWORKS Materials:AISI 304	24.9999	20.800	17.200	1.800	- none -	14		0.8	25	Standard	20.800	17.200	0.00014	0.50
0.8m, 25 Deg, 24T, Stub, Center Hole	1.257	0.28	0.10	U	10	aterials:AISI 316 Annealed Stainl	25.0001	20.480	17.600	1.440	- none -	14		0.8	25	Stub	20.480	17.600	-0.00015	-0.53

Green = Input
Orange = Secondary Input
Yellow = Calculated
Driven Value for Reporting in DT

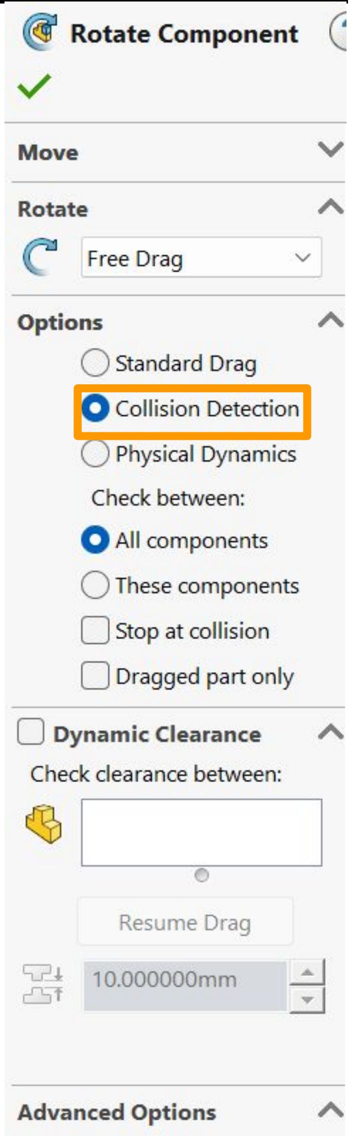
Collision Detection

Checks if components touch or interfere with motion

Uses:

- Ensures proper fit and movement of parts
- Prevents design errors
- Simulate realistic motion scenarios for mechanism

Accessed by clicking on “Move Component” or “Rotate Component”

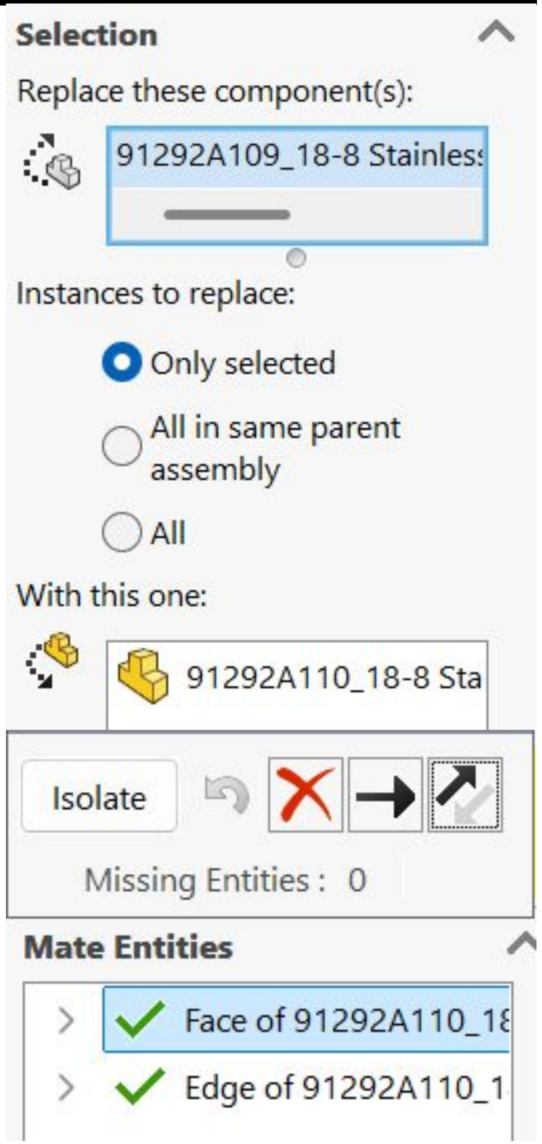


Replace Components

1. Use the “Replace Components” tool
   a. Next to “Insert Components”
2. Select the components
3. Confirm/Reassign mates

Only really useful if mating geometry is identical

**Often it is easier to delete the object and add new mates



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Commercial-off-the-Shelf (COTS)

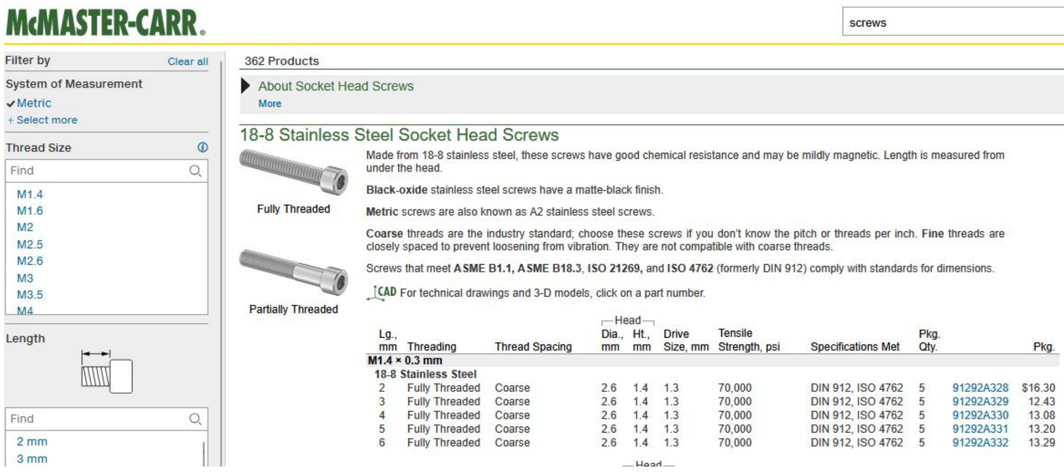
Work Smarter NOT Harder

Most item suppliers will provide a CAD file. Use them

- SolidWorks accepts most CAD file formats

Common suppliers:

- McMaster-Carr: Fasteners, Power Transmission, etc
- Misumi: Industrial components
- Adafruit: Electronics
- A lot more

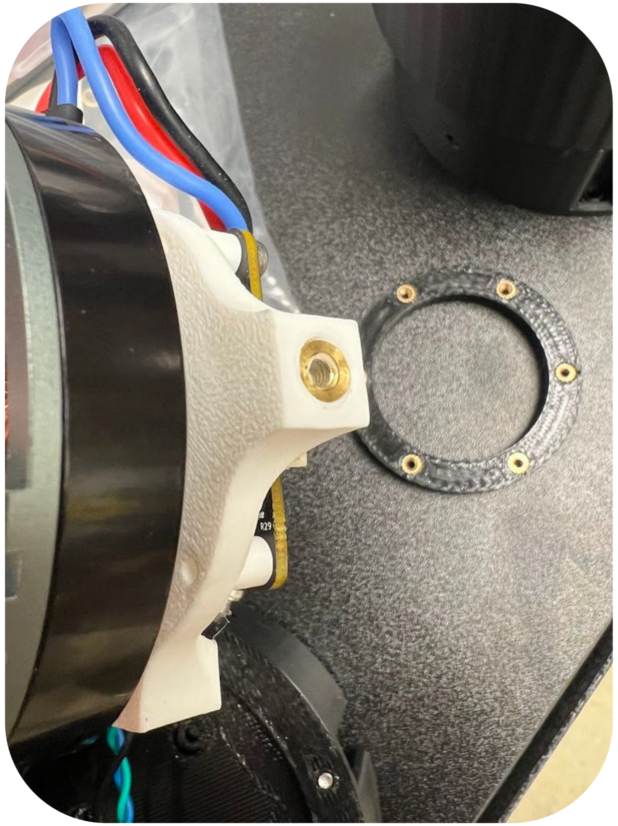


*We have M3, M2.5, or M2 screws in the lab

Heat Set Inserts

- Threaded inserts melted into 3D-printed parts (Use the Soldering tool with het set insert tip)
  - Let the insert cool before touching it (From experience)

- 3D-printed threads/self tapping are very easy to destroy



Important information: (M3 example)

Installed Length	3.8 mm
For Min. Material Thickness	4.57 mm
Drill Bit Size	No. 8
For Maximum Hole Diameter	0.199"

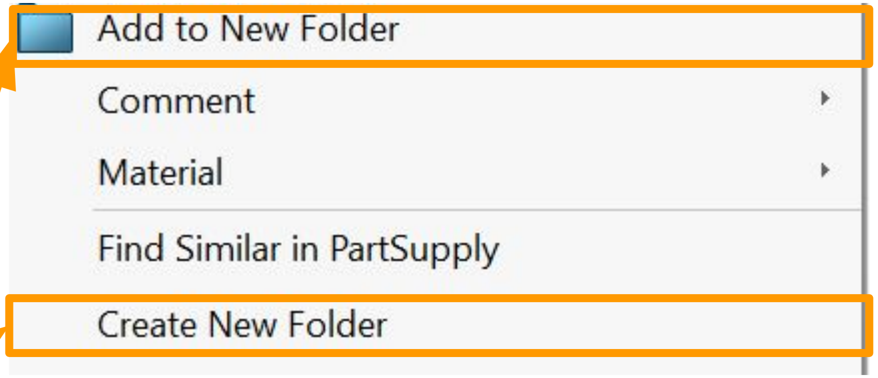
Hole cutout dimensions for CAD

Note: To make installation easy, add an 8° chamfer to ½ “Installed Length”

SolidWorks Folders

Necessity when organizing large assembly feature-trees

Please use for your fasteners

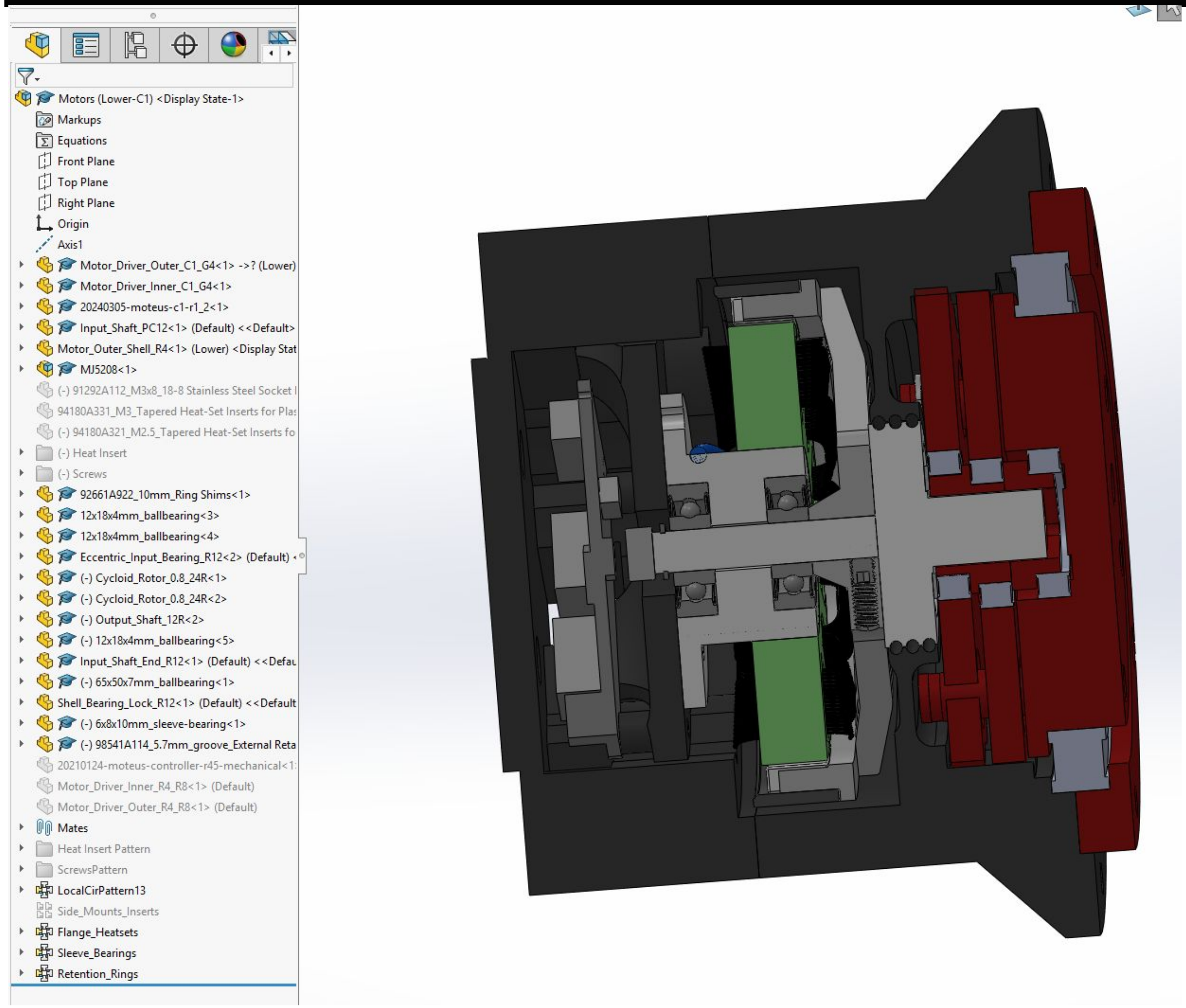


To use:

1. Right Click feature-tree
2. Select one of the two options
3. Click and drag components

You can suppress every component in one click

Example



112 total components

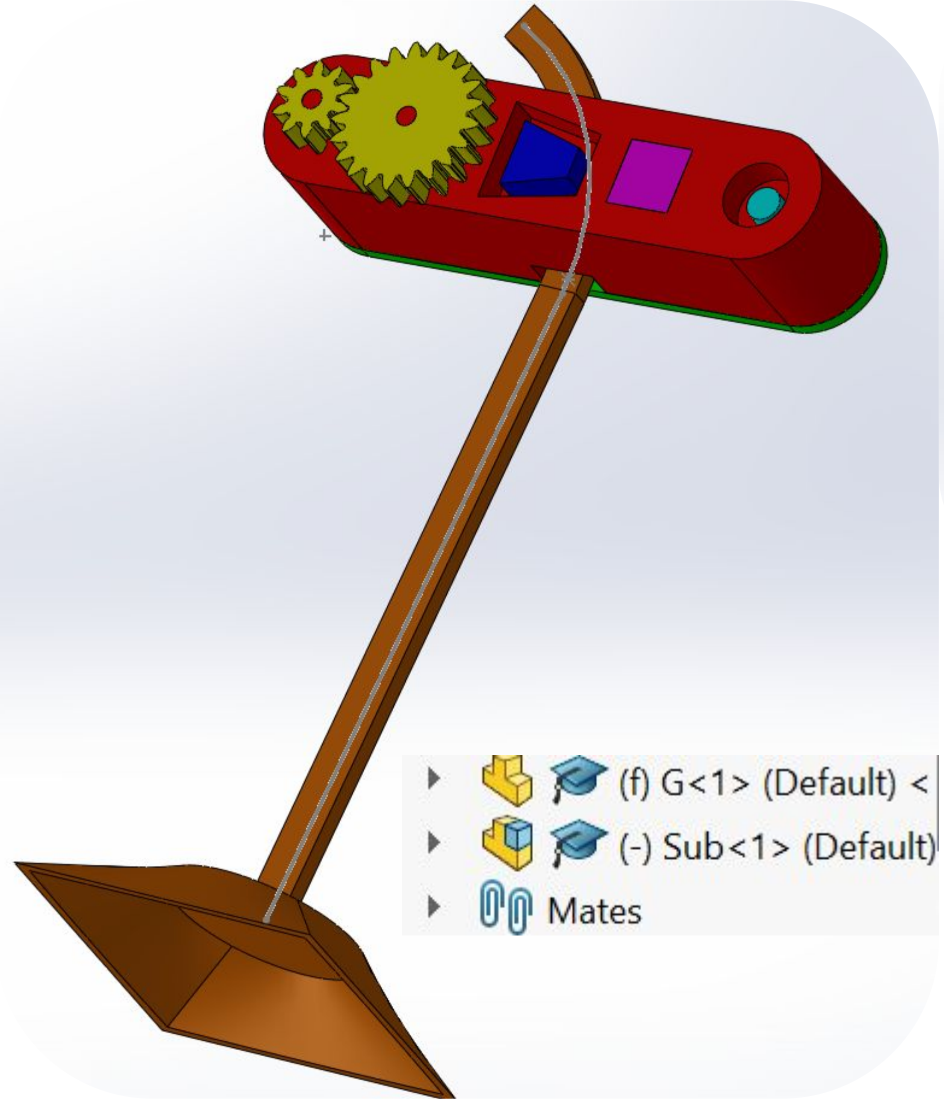
- 38 unique

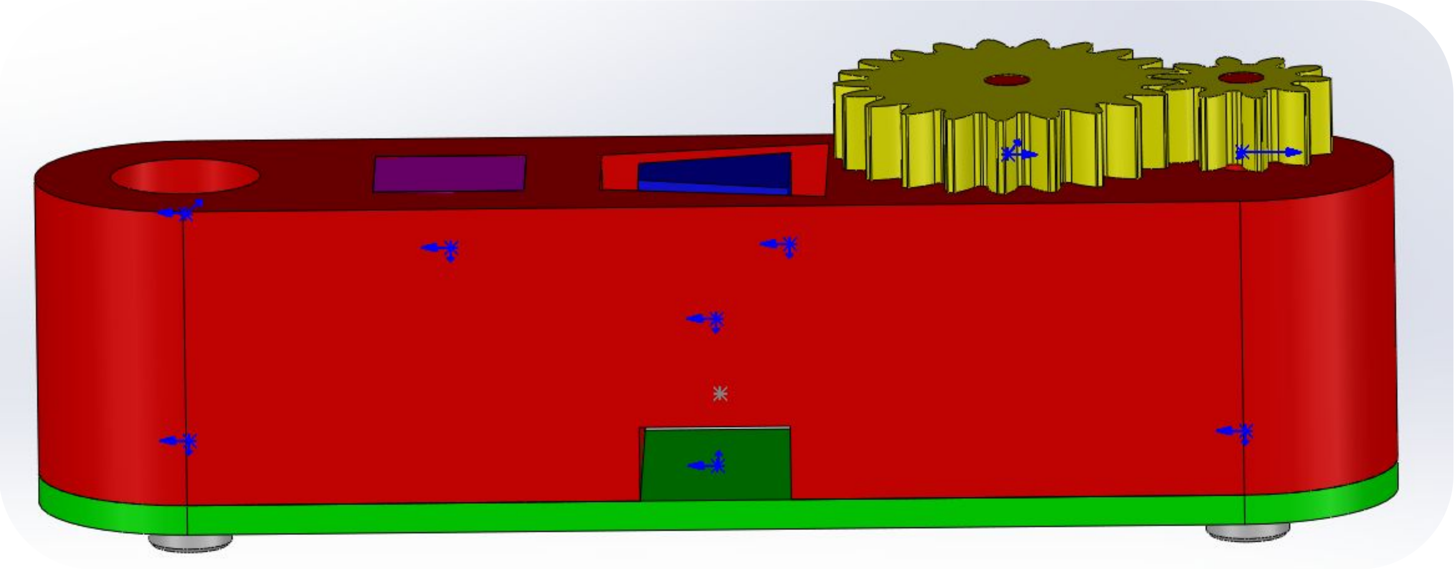
61 Screws/Heat Inserts

Still easy to navigate

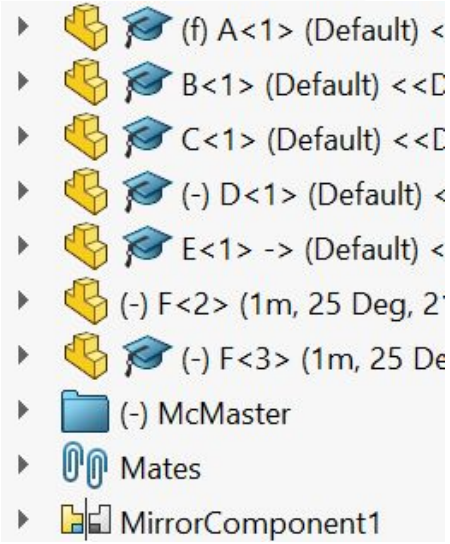
Suppressed components load significantly quicker

Let’s Practice





Files are available under “CAD Examples” at https://www.turtlerobotics.org/hatchling



Action Plan 1

Sub assembly

1. Insert parts A-F, Should have 2 F parts
2. 3 “Coincident” mates between A and B
3. 1 [2.5mm] “Distance” mate, 1 “Width” mate, and 1 “Coincident” mate between A and C
4. 1 “Concentric” mate and 1 “Coincident” mate between A and D
5. 2 “Concentric” mates and 1 “Coincident” mate between A and E. Use an external reference to define E

Action Plan 2

Sub assembly continued

6. 1 “Hinge” mate between A and F_21tooth
7. 1 “Concentric” mate and 1 “Coincident” mate between A and F_10tooth
8. Download M3X4 (91292A109) screws and M3 heat-set insert (94180A331) part files from McMaster Carr
9. Insert 2 M3X4 (91292A109) screws and 2 M3 heat-set insert (94180A331)
10. Repeat twice, 1 “Concentric” mate and 1 “Coincident” mate between A and heat-set insert
11. 1 “Concentric” mate and 1 “Coincident” mate between E and M3X4 screw

Action Plan 3

Sub assembly continued

12. Mirror the heat-set insert
    a. This is just to showcase how to use assembly mirror

Main assembly

13. Insert part G and the “Sub” assembly
14. 1 “Path” mate between G and “Sub”
    b. Pitch/Yaw control - “Follow Path” Y-axis
    c. Roll control - “Up Vector” [Front flat face of G rail] Z-axis

Take a little bit of time to clean up your assembly feature tree. This will make modifying the assembly a lot easier.

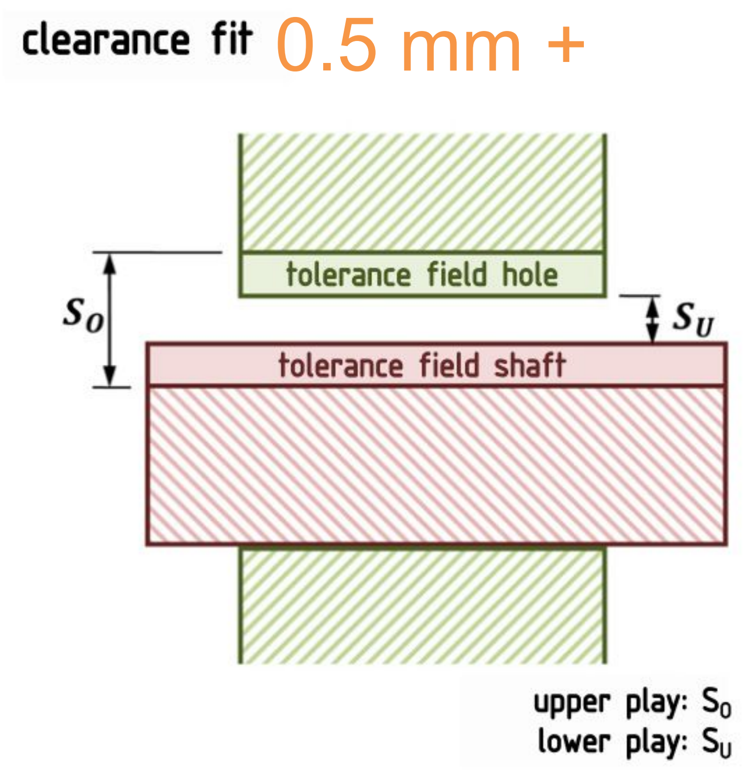
Tolerance vs Clearance

Tolerance -> Maximum allowed deviation from nominal design

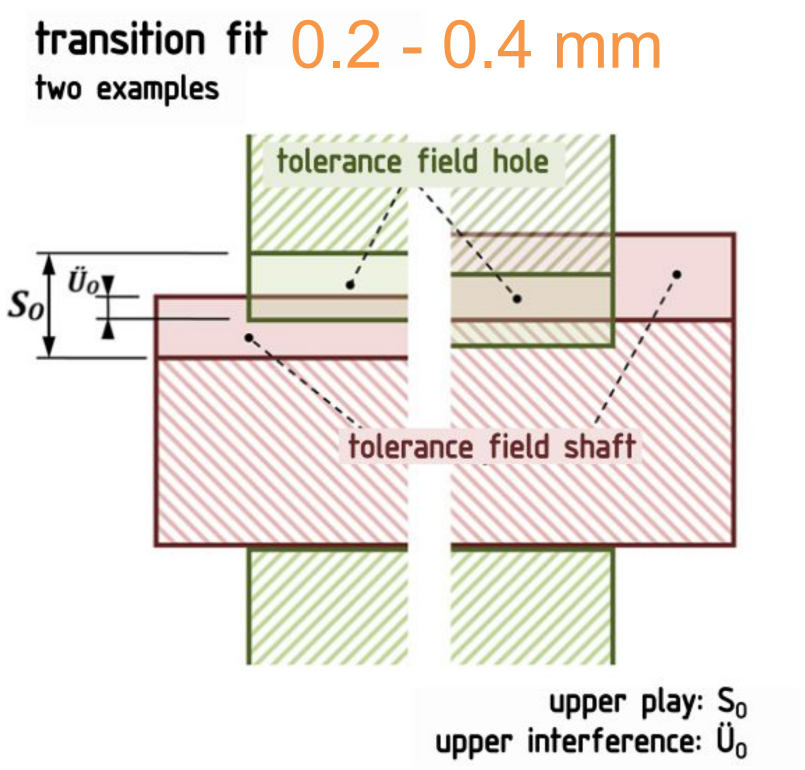
Clearance -> The gap between two components

Types of fits / 3D Printing Design

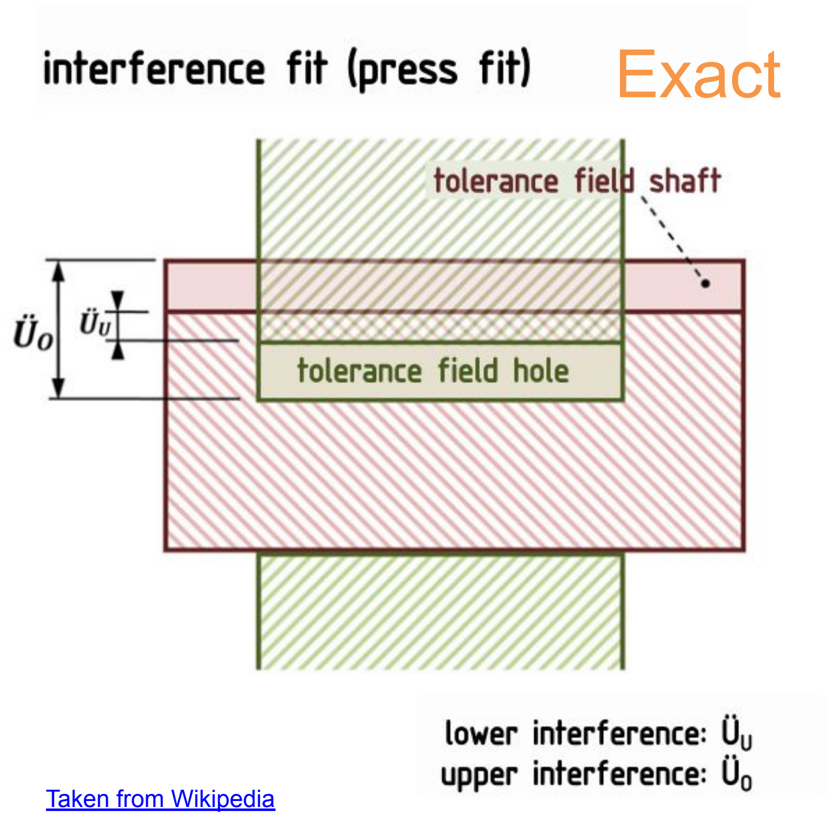
0.5 mm +



0.2 - 0.4 mm



Exact



[Taken from Wikipedia]

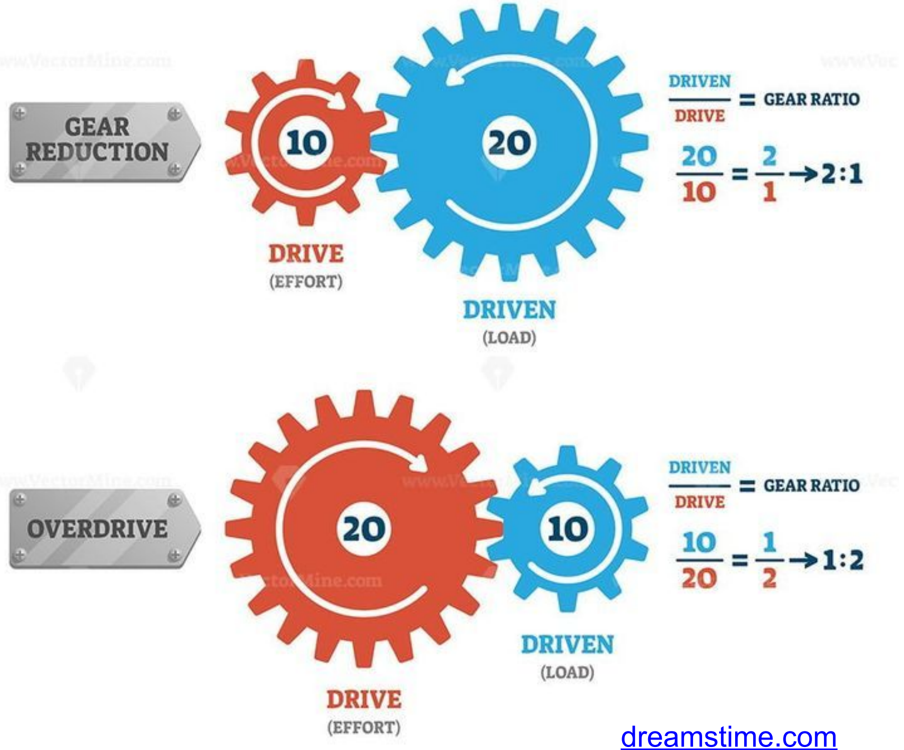
Gear Ratios

Gear ratio: The ratio of the number of rotations of a driver gear to the number of rotations of a driven gear

**Keep 3D Printed Gears to a minimum tooth count of 9 **

Use Case:

- Manipulate torque to speed ratio
  - Reduce torque by increasing speed
  - Increase torque by decreasing speed
- Keeping two axes synchronized
- Gears spin in alternating direction



dreamstime.com

Note: To distribute tooth wear, add 1 to your "friendly" gear ratio*

CSWA/CSWP Free Vouchers

These SolidWorks Certifications look great on your resume.

The information on our slides and the LinkedIn course is enough to pass both with some practice. The CSWA comes before the CSWP.

How to claim a voucher?

1. Create a Tangix TesterPro account with "@tamu.edu" email
2. Request a voucher on the FEDC website
3. Download the Tangix TesterPro Client or schedule a FEDC CAD Station
4. You will get an email when the voucher is loaded. The voucher expires in a week. (You will need SolidWorks installed)

Next Milestone

Milestone: Assembly Review

Date: Week 7: Programming and Git/GitHub (1 week from today)

Expectation: Have a detailed sketch and begin CAD of drive system. Decide on the electronics you will use.

Exceed Expectation: Have a CAD assembly of a drive system. Have a finished electronics wiring diagram.

Impact: We will review design viability and suggest improvements. Potential to prototype your mechanism.

TEXAS A&M UNIVERSITY
ROBOTICS TEAM & LEADERSHIP EXPERIENCE

Programming and Git/GitHub

Next Week

"Design is not how it looks like and feels like. Design is how it works"

Steve Jobs

Hatchling