

TEXAS A&M UNIVERSITY
ROBOTICS TEAM & LEADERSHIP EXPERIENCE

SolidWorks Assembly

TURTLE Hatchling

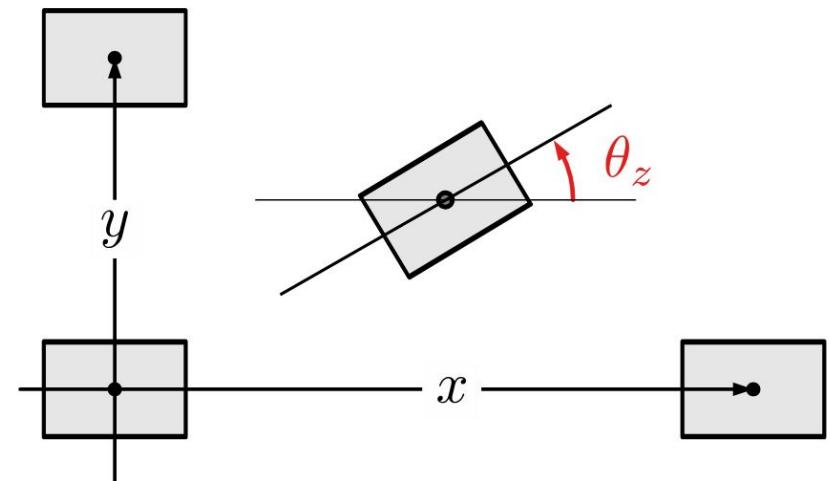
Sketches



Sketches are 2D “drawings”

A plane has 3 Degrees of Freedom:

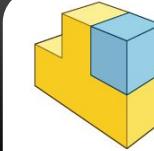
- X (Translational)
- Y (Translational)
- $\theta(z)$ (Rotational)



engineeringstatics.org

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

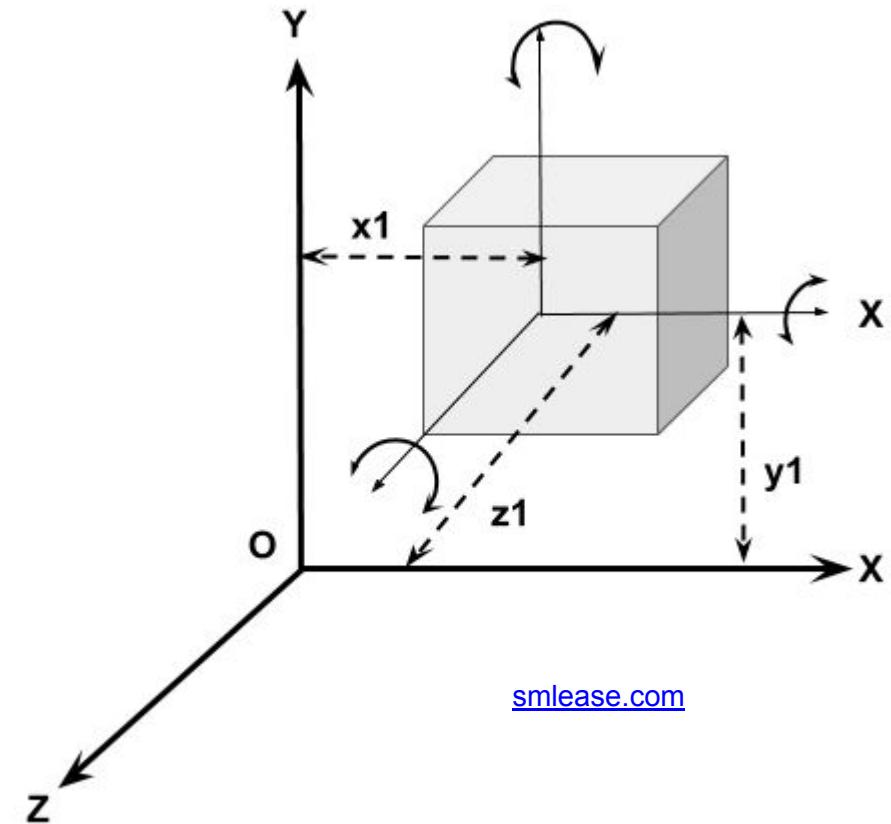
Assemblies



Assemblies are 3D

3D has 6 Degrees of Freedom

- X (Translational)
- Y (Translational)
- Z (Translational)
- $\theta(x)$ (Rotational)
- $\theta(y)$ (Rotational)
- $\theta(z)$ (Rotational)



smlease.com

Moving mechanism may result in under defined assemblies

Common Symbols



Defined States are shown in the feature tree

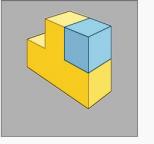
- (f) - Fixed component
 - Should have one of these in an assembly
- (-/+) - Under/Over defined
- (?) - Cannot solve
- Nothing - Fully defined
- - Education license tag

▶	SERVO_MG92B<1> (Pre
▶	(f) chassis<1> ->? (Defa
▶	SERVO_MG92B<3> (Pre
▶	SERVO_MG92B<4> (Pre
▶	Top plate<1> ->? (Defa
▶	Front Plate<1> ->? (Def
▶	SERVO_MG92B<5> (Pre
▶	(-) IMU representation<
▶	(-) PCB, Raspberry Pi Ze
▶	(-) IMU representation<
▶	(-) PCB, Raspberry Pi Ze
▶	(-) SCREWS
▶	(-) HEAT INSERTS
▶	FPlateAssem<2> -> (De
▶	Full Leg F24<1> (Default) <D
▶	Full Leg F24<3> (Mirror) <D
▶	Camera_module_3_wide
▶	MirrorFull Leg F24<1> (Mirro
▶	MirrorFull Leg F24<2> (Defa
▶	Mates

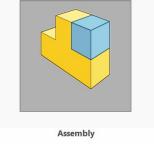
Creating an Assembly



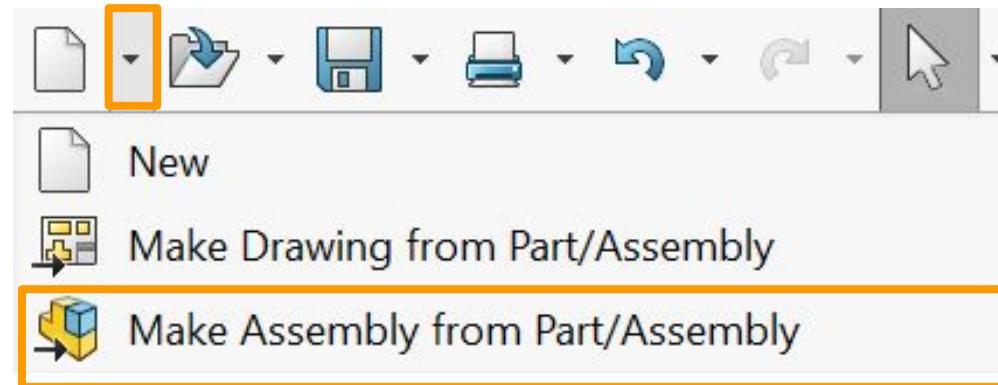
Method 1:

1. Click “New” 
2. Click “Assembly” 

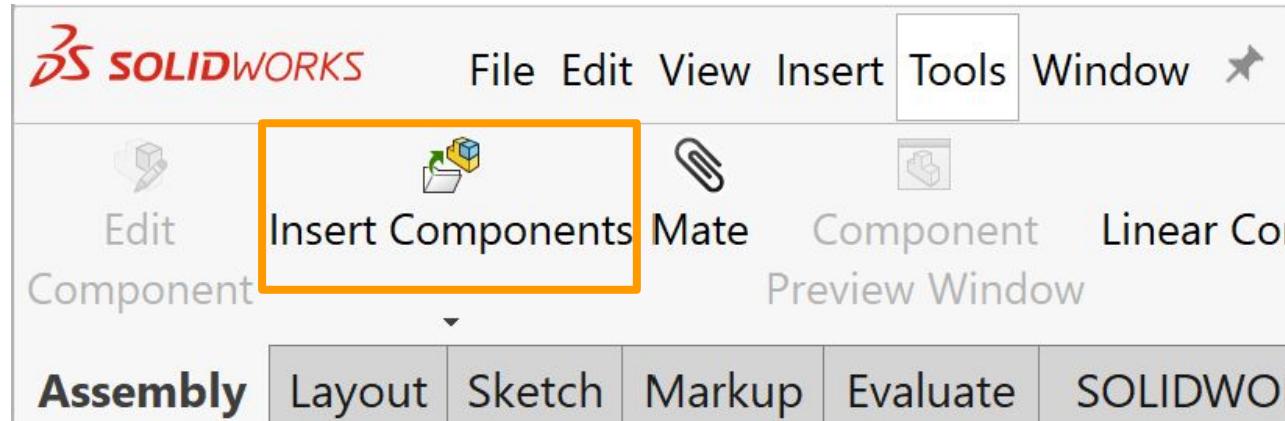
Method 2: From a part/assembly

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

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



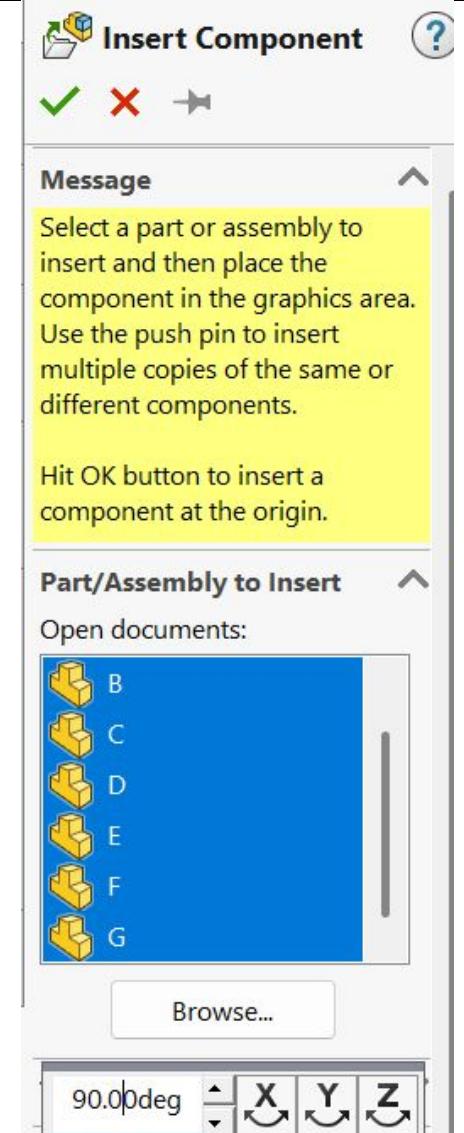
Inserting Components



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

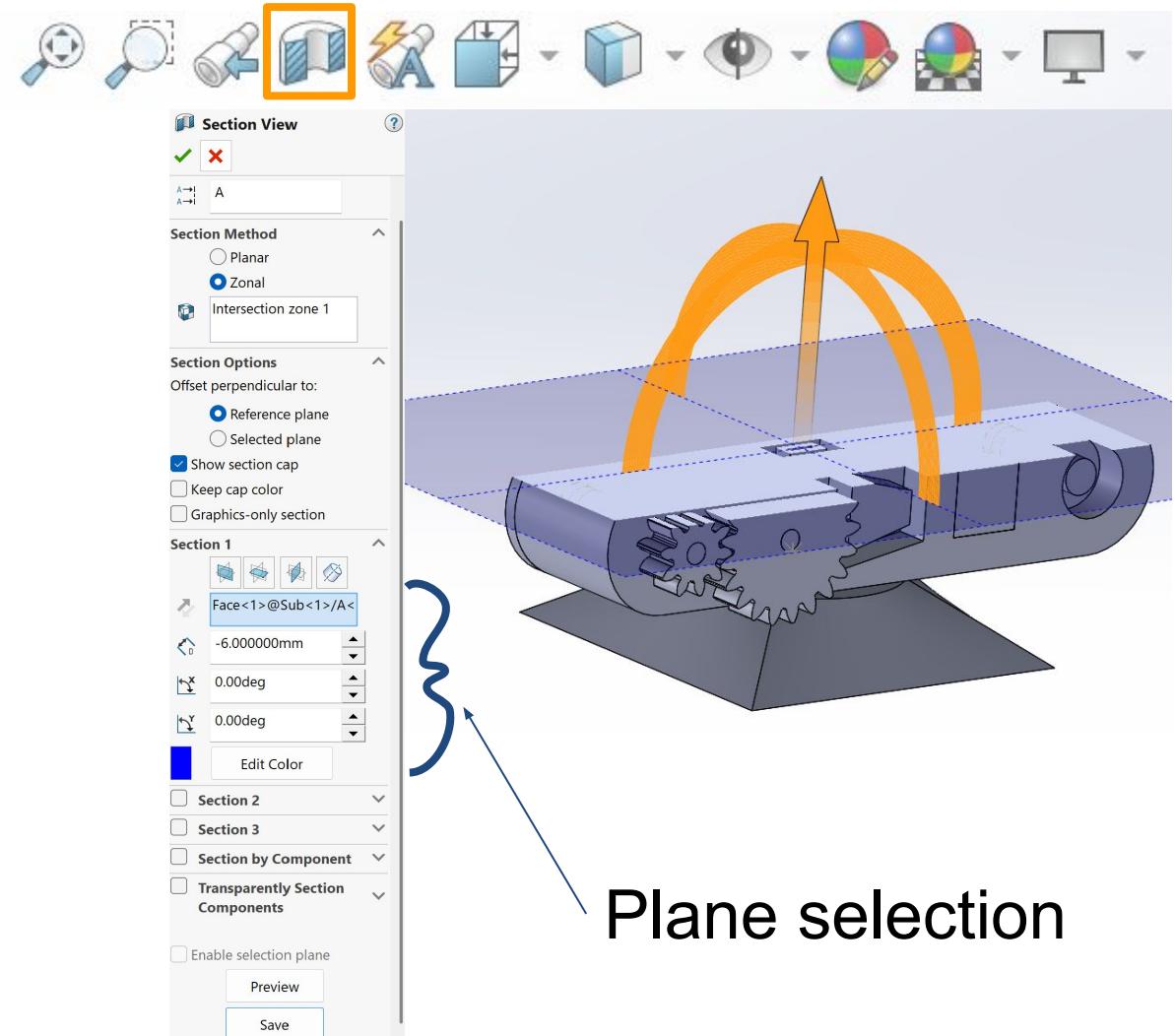


Section View



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



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

 Mates
○ Concentric1 (H)
↖ Coincident1 (H)
↖ Coincident2 (H)
○ Concentric2 (H)
○ Concentric3 (H)
↖ Coincident5 (H)
○ Concentric6 (2)
○ Concentric7 (H)
↖ Coincident8 (2)
↖ Coincident9 (2)
○ Concentric8 (H)
↖ Tangent3 (Hip)
↖ Coincident11 (H)
○ Concentric9 (H)
↖ Coincident12 (H)
○ Concentric10 (H)
○ Concentric11 (H)
↖ Coincident13 (H)

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



Mate Type

- Coincident
- Parallel
- Perpendicular
- Tangent
- Concentric
- Do NOT use

Distance: 1.000000mm

Angle: 30.00deg

Mate alignment:

Direction

These should look familiar

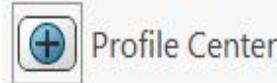
Identical to sketch relations but can be used in 3 dimensions

Switching the mate direction may flip other mate directions as well

Advanced Mates



Mate Types



Profile Center



Symmetric



Width



Path Mate



Linear/Linear Coupler

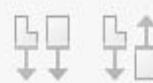


Distance
Range



Angle
Range

Mate alignment:



Fancy but often computational complex

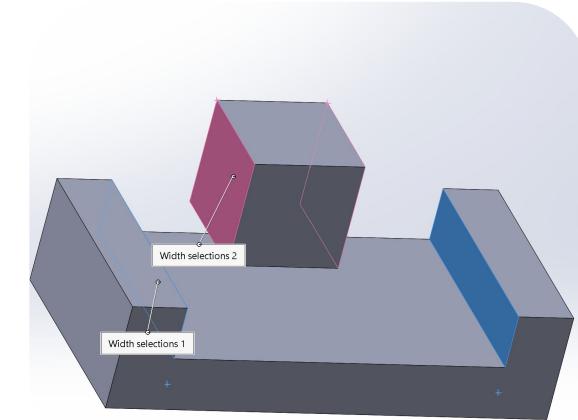
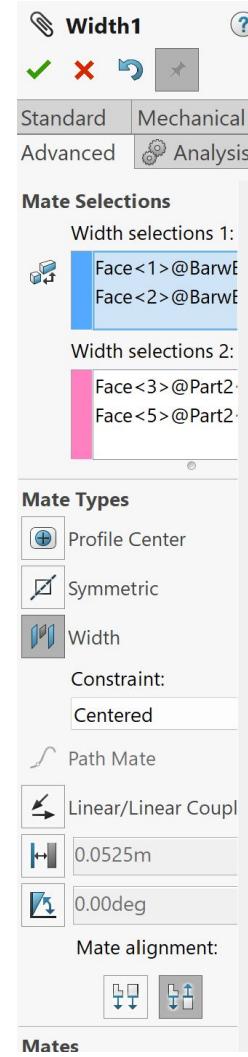
SolidWorks has no collision detection

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

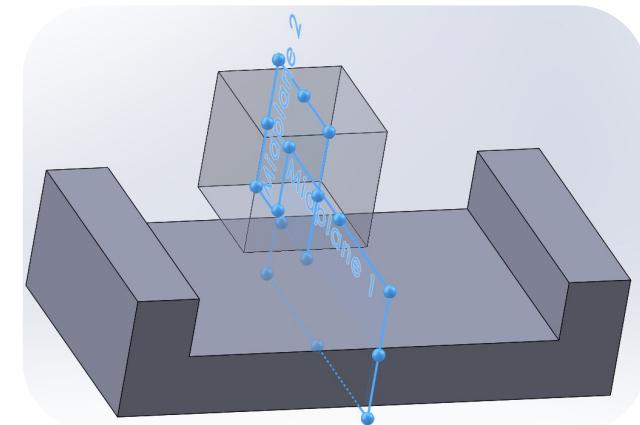
Width Mate



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



Same thing as



Path Mate



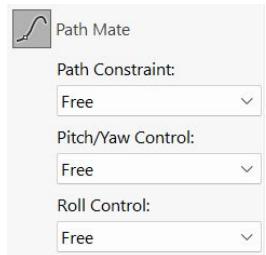
Requires 2 Sketches:

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

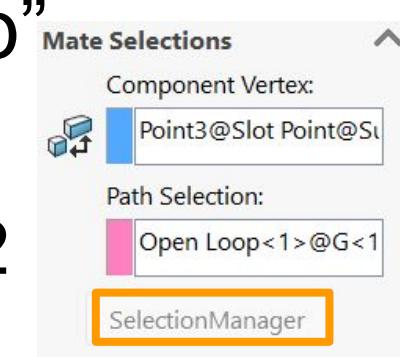


Open Loop

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

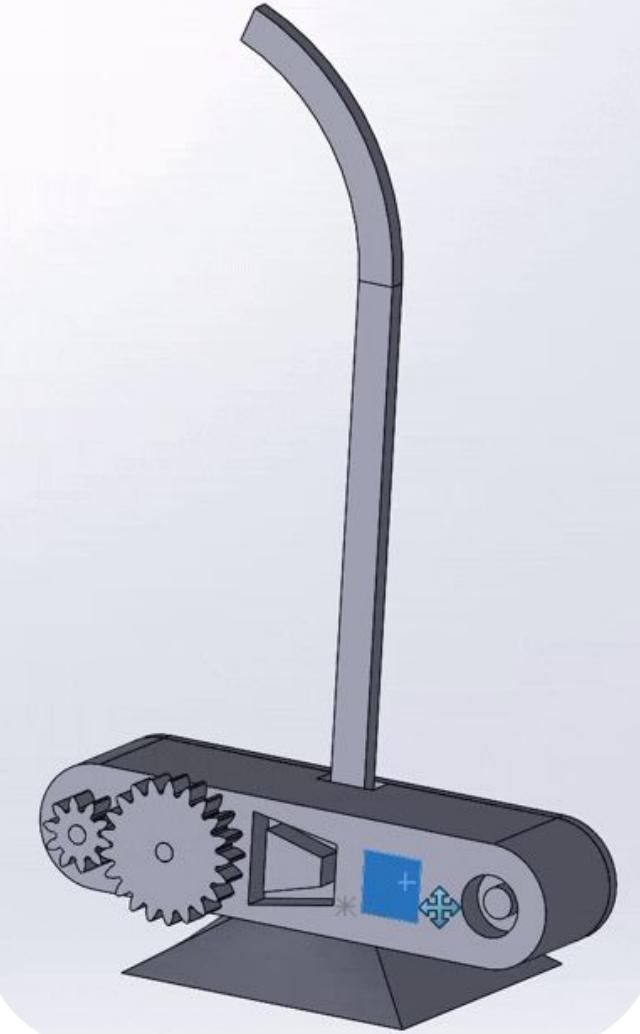


Can control location and orientation within mate



1
2

SelectionManager



Mechanical Mates



Mate Type



Cam



Slot



Hinge



Gear



Rack Pinion



Screw

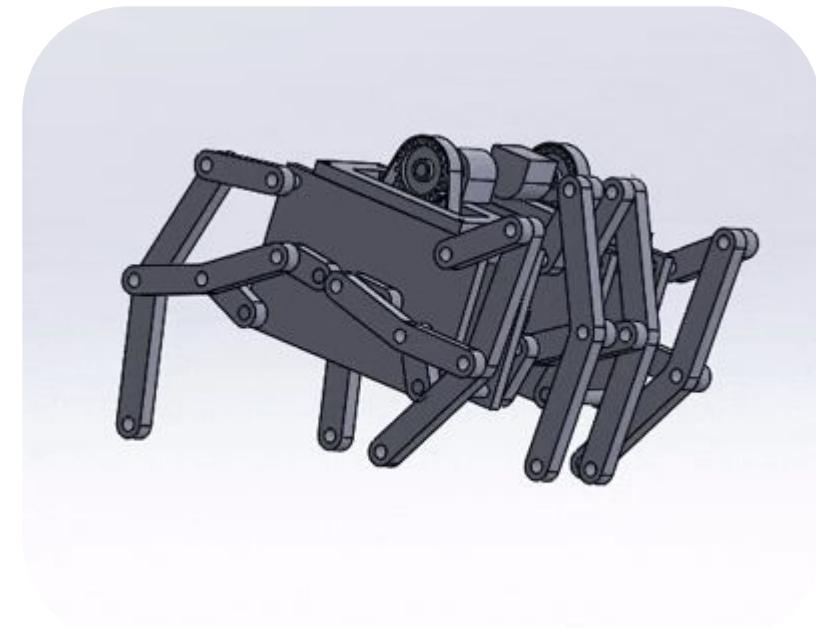


Universal Joint

Mate alignment:



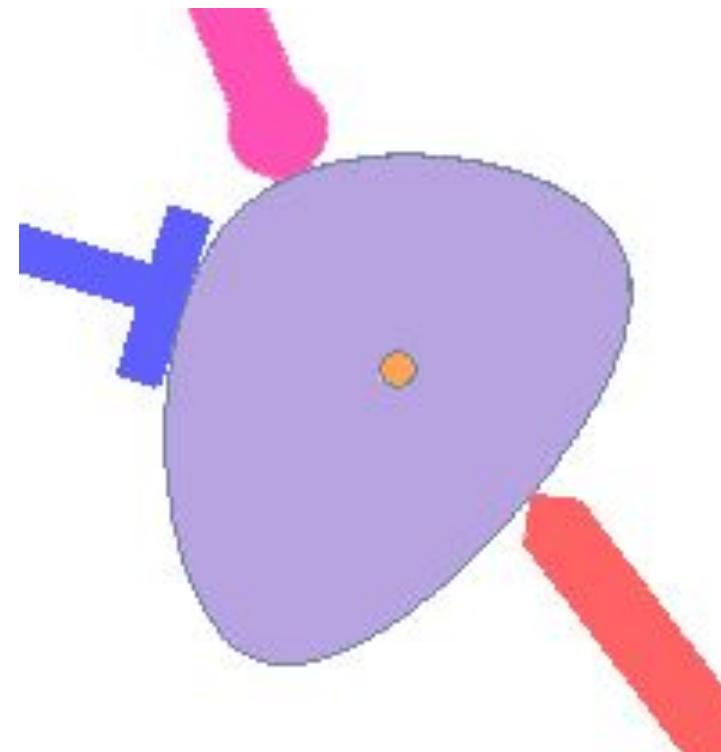
These allow you to simulate real-world mechanical movements between components



Cam Mate



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



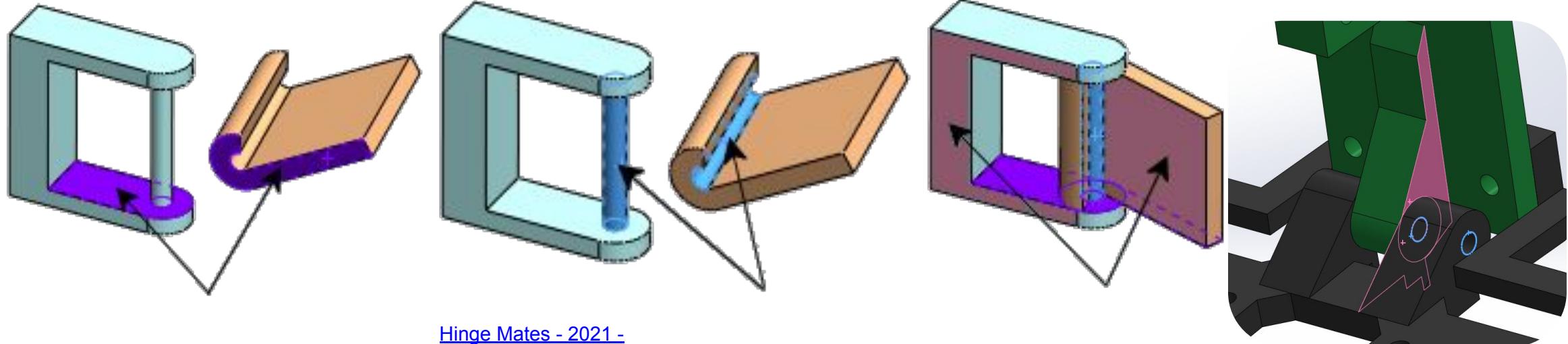
[Video:
Cam-Follower
Mate - 2021 -
SOLIDWORKS
Help](#)

Hinge Mate



Combines a “Coincident” and
“Concentric mate into one

Can also add an “Angle Range” mate



[Hinge Mates - 2021 -
SOLIDWORKS Help](#)

Mate Selections

Concentric Selections:



Face<1>@F<1>
Face<2>@A<1>

Coincident Selections:



Face<3>@A<1>
Face<4>@F<1>

Specify angle limits

Gear Mate



Requires two sketches

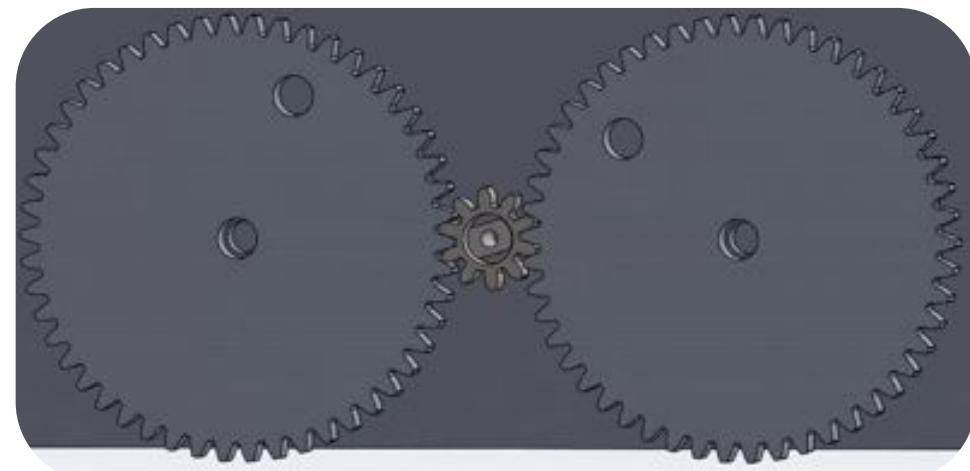
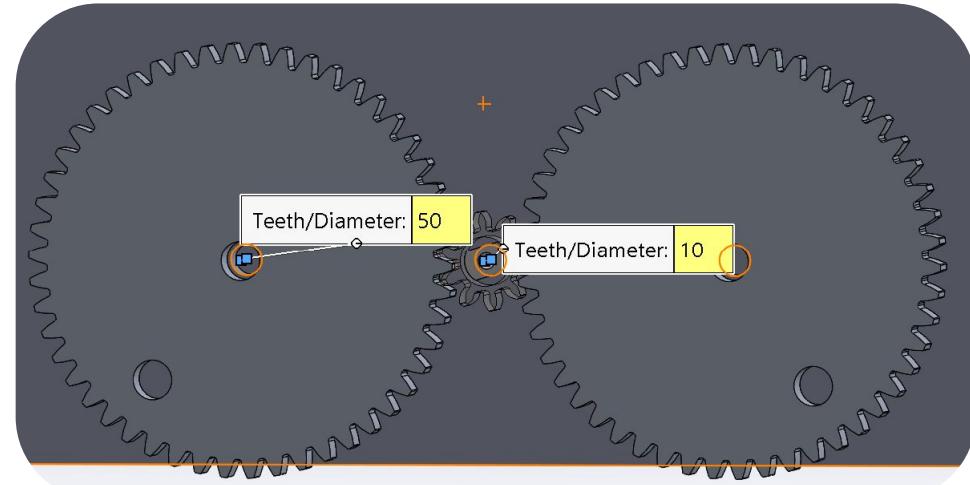
1. Construction Circle on Gear 1
2. Construction Circle on Gear 2

*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



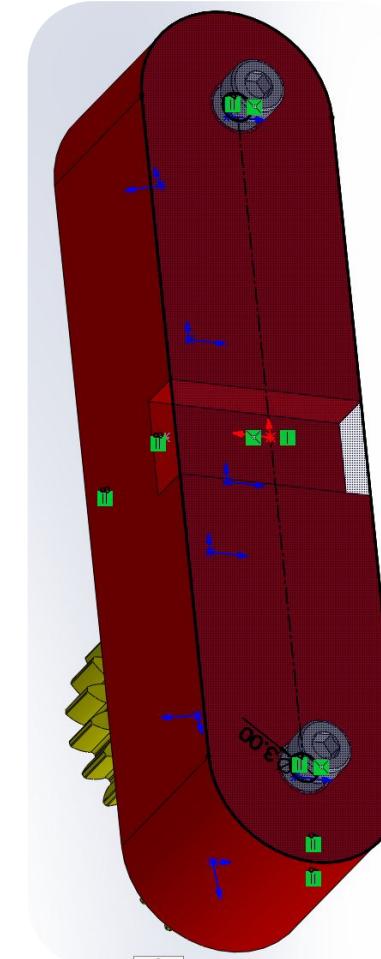
Purpose:

- 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

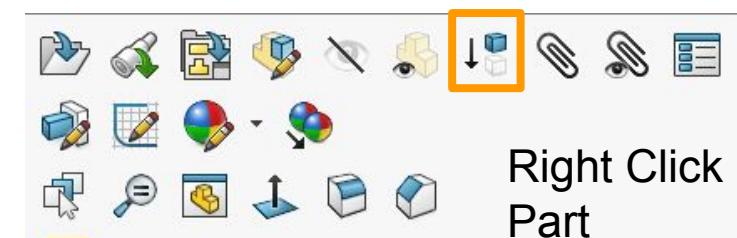
Advantages:

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

The screenshot shows a list of mates for a component in a CAD application. The list includes:

- Concentric7 (CyberDeckBc)
- Concentric8 (CyberDeckBc)
- Coincident26 (CyberDeckBc)
- Coincident27 (CyberDeckBc)

**Also suppresses the component's mates

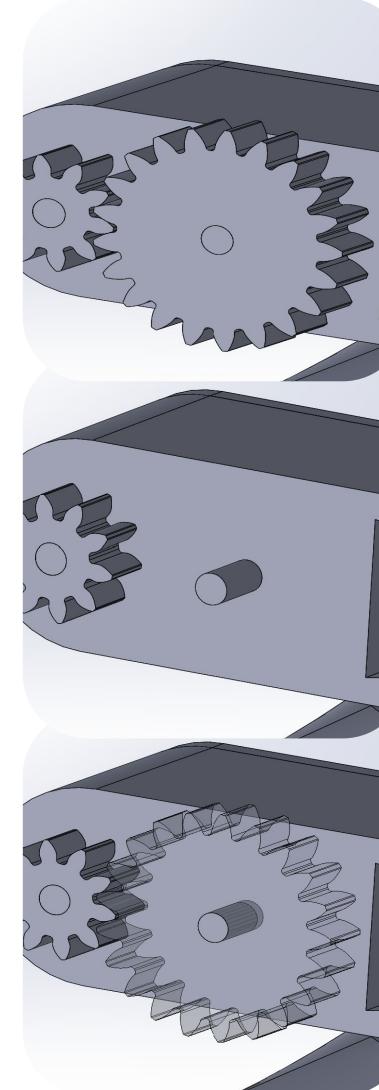




Hidden vs Transparent

Hidden:

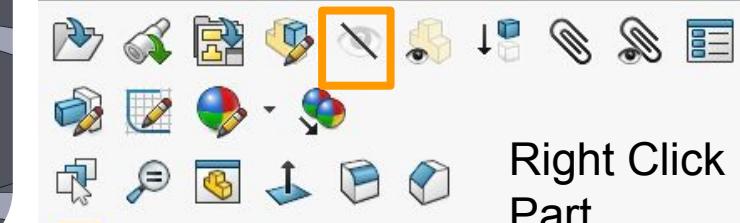
- Makes the component invisible
- Must unhide from feature tree to use again
- Used to focus on internal components



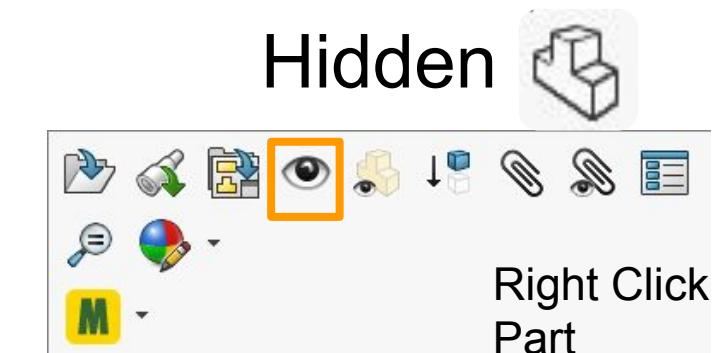
Transparent:

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

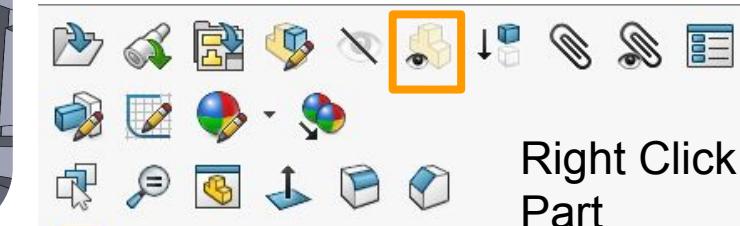
Normal



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

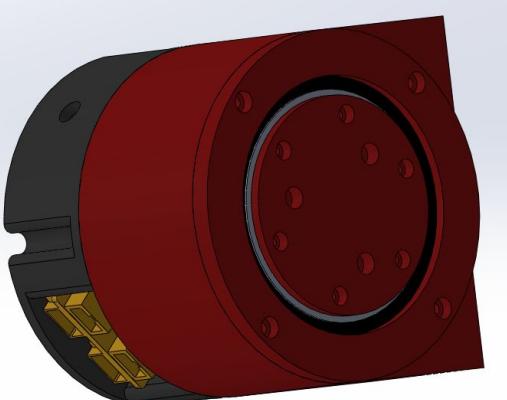
Think about screws. One CAD model works for every length variation

More on Configurations

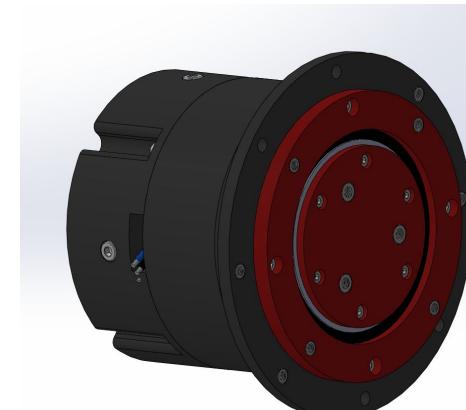


How to Use:

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



Configurations work on both parts and assemblies



- ↳ Tables
- ↳ Lower-C1 [Motors] ✓ Lower-C1-Fasteners [Motors]
- ↳ Upper-C1 [Motors] ✓ Upper-C1-Fasteners [Motors]
- ↳ Upper-R4 [Motors] ✓ Upper_R4_Fasteners [Motors]

*Green Check is the configuration you are in

- Tables
- ✓ Default
- Lower

- Add Configuration...
- Configuration Publisher...
- Rebuild All Configurations

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

Design Table for: Involute Gear, Metric																			
	Tooth Width@Sketch2	Root Fillet@Root Fillet	Tip Radius@Tip Radius	\$STATE@Hole	HoleDia@Sketch6	SUBRARY: 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 Hole	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 Hole	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	SOLIDWORKS Materials: 6061 Alloy	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 Hole	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	SOLIDWORKS Materials: AISI 316 Annealed Stainless Steel	25.0001	20.480	17.600	1.440	- none -	14	0.8	25	Stub	20.480	17.600	-0.00015	-0.53

Configurations

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

Collision Detection

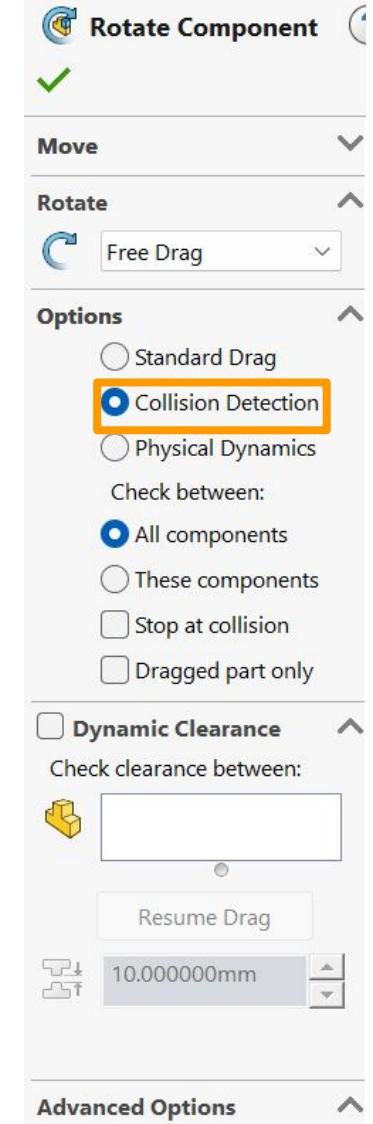


Checks if components touch or interfere with motion

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

Selection

Replace these component(s):

91292A109_18-8 Stainless

Instances to replace:

Only selected

All in same parent assembly

All

With this one:

91292A110_18-8 Sta

Isolate Missing Entities : 0

Mate Entities

> Face of 91292A110_18

> Edge of 91292A110_1



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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

The screenshot shows a search results page for "screws" on the McMaster-Carr website. The top navigation bar includes a search bar and links for "About Socket Head Screws" and "More". The main content area displays a product listing for "18-8 Stainless Steel Socket Head Screws". It features two product images: "Fully Threaded" and "Partially Threaded". Below the images is a detailed description: "Made from 18-8 stainless steel, these screws have good chemical resistance and may be mildly magnetic. Length is measured from under the head. Black-oxide stainless steel screws have a matte-black finish. Metric screws are also known as A2 stainless steel screws. Coarse threads are the industry standard; choose these screws if you don't know the pitch or threads per inch. Fine threads are closely spaced to prevent loosening from vibration. They are not compatible with coarse threads. Screws that meet ASME B1.1, ASME B18.3, ISO 21269, and ISO 4762 (formerly DIN 912) comply with standards for dimensions." A "CAD" link is provided for technical drawings and 3-D models. The table below lists product details for M1.4 x 0.3 mm screws.

Lg., mm	Threading	Thread Spacing	Head Dia., mm	Head Ht., mm	Drive Size, mm	Tensile Strength, psi	Specifications Met	Pkg. Qty.	Pkg.	
M1.4 x 0.3 mm										
18-8 Stainless Steel										
2	Fully Threaded	Coarse	2.6	1.4	1.3	70,000	DIN 912, ISO 4762	5	91292A328	\$16.30
3	Fully Threaded	Coarse	2.6	1.4	1.3	70,000	DIN 912, ISO 4762	5	91292A329	12.43
4	Fully Threaded	Coarse	2.6	1.4	1.3	70,000	DIN 912, ISO 4762	5	91292A330	13.08
5	Fully Threaded	Coarse	2.6	1.4	1.3	70,000	DIN 912, ISO 4762	5	91292A331	13.20
6	Fully Threaded	Coarse	2.6	1.4	1.3	70,000	DIN 912, ISO 4762	5	91292A332	13.29

*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 $\frac{1}{2}$ "Installed Length"



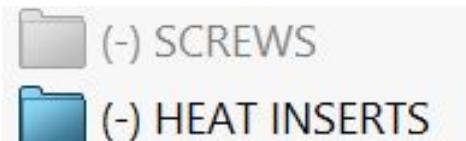
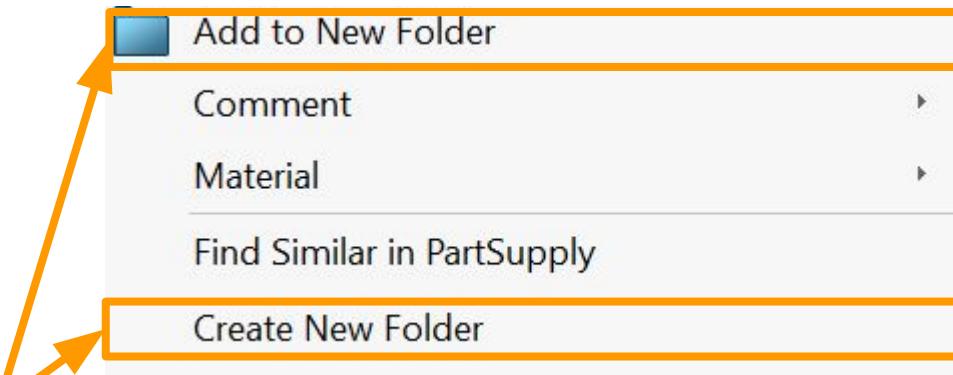
SolidWorks Folders



Necessity when organizing large assembly feature-trees
Please use for your fasteners

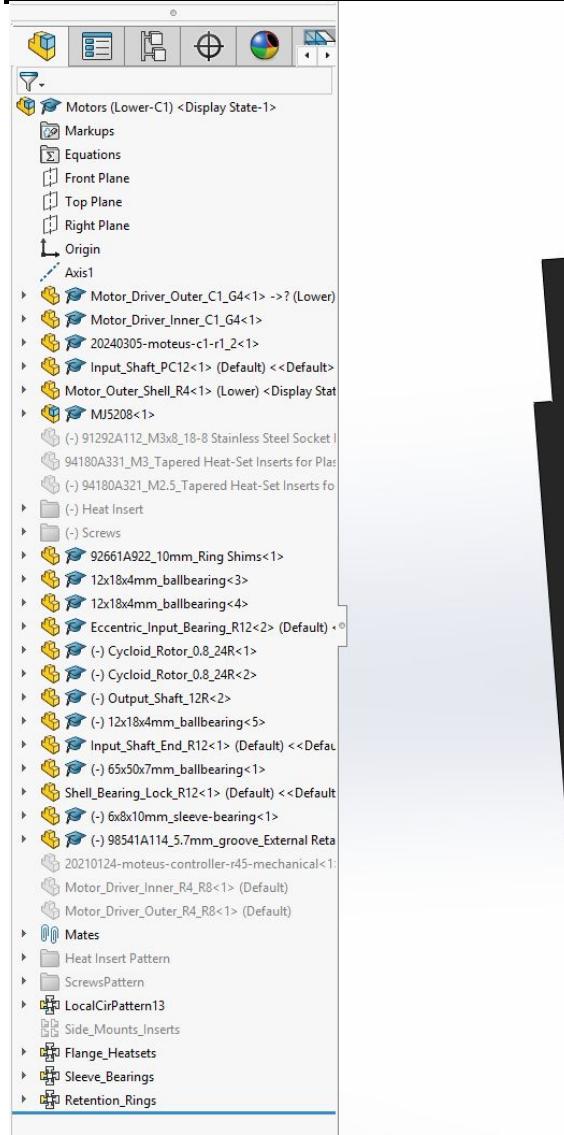
To use:

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



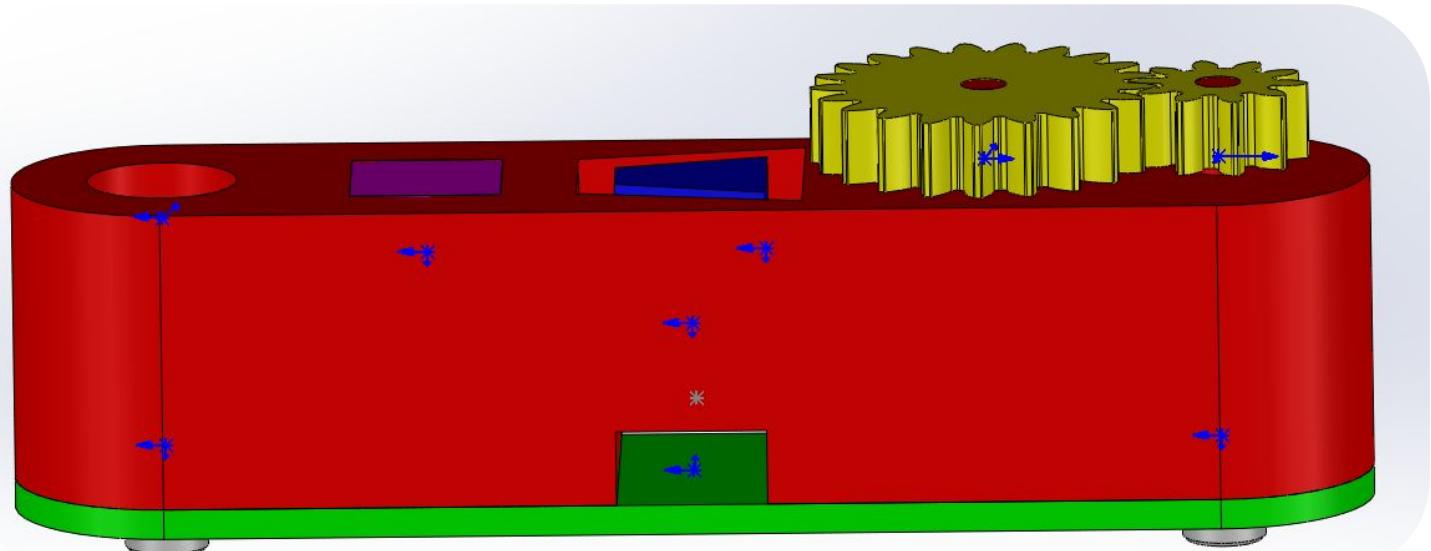
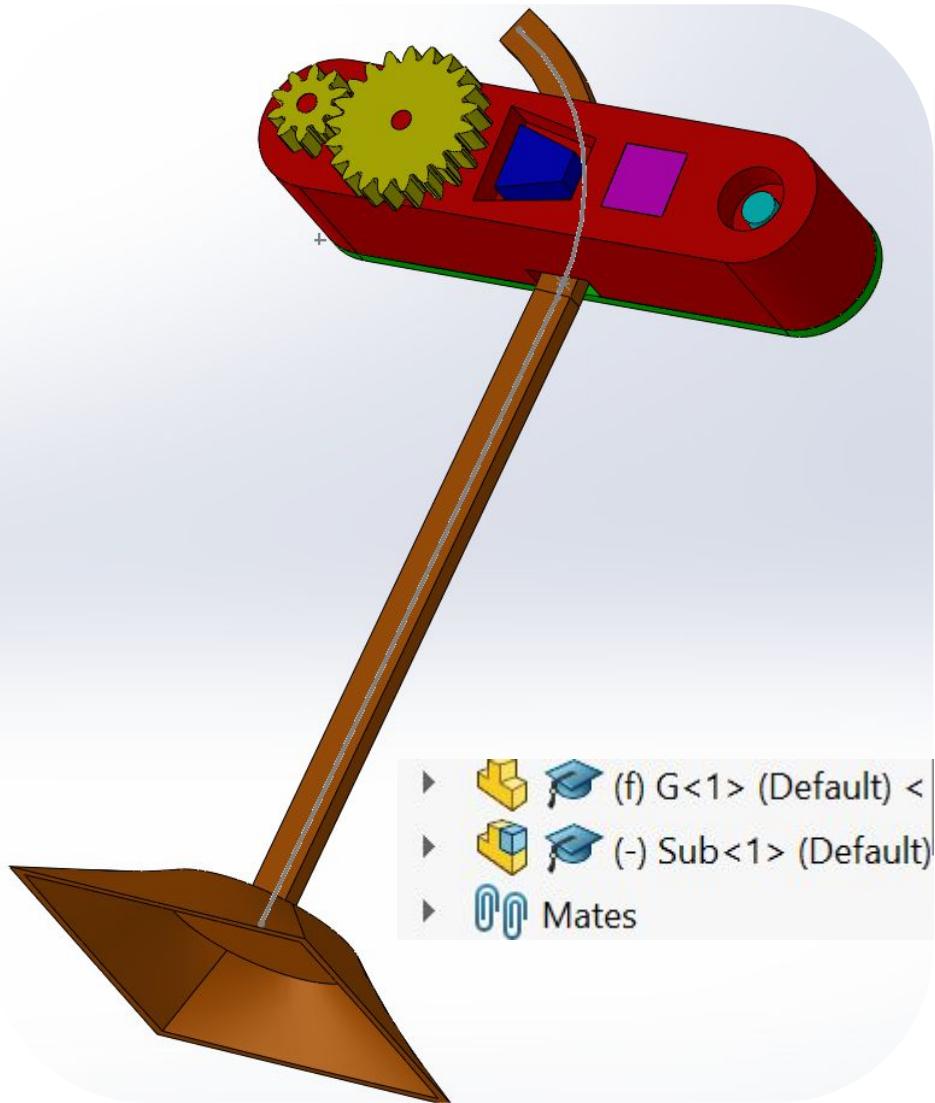
112 total components
- 38 unique

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>

- ▶ (f) A<1> (Default) <
- ▶ B<1> (Default) <<D
- ▶ C<1> (Default) <<D
- ▶ (-) D<1> (Default) <
- ▶ E<1> -> (Default) <
- ▶ (-) F<2> (1m, 25 Deg, 2
- ▶ (-) F<3> (1m, 25 Deg, 2
- ▶ (-) McMaster
- ▶ Mates
- ▶ MirrorComponent1

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

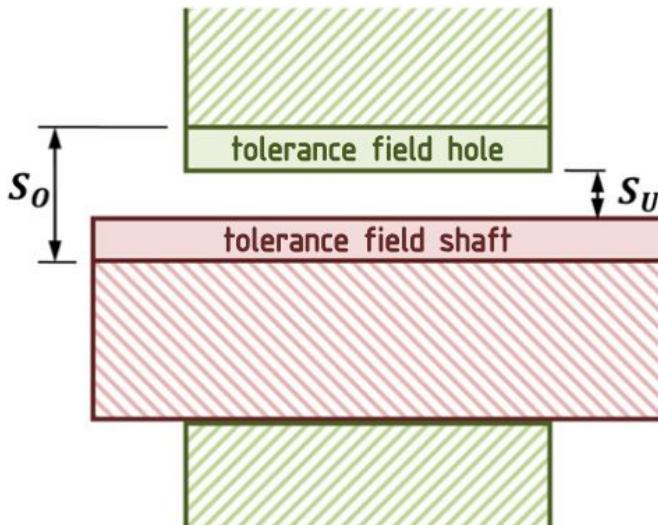


Tolerance -> Maximum allowed deviation from nominal design

Clearance -> The gap between two components

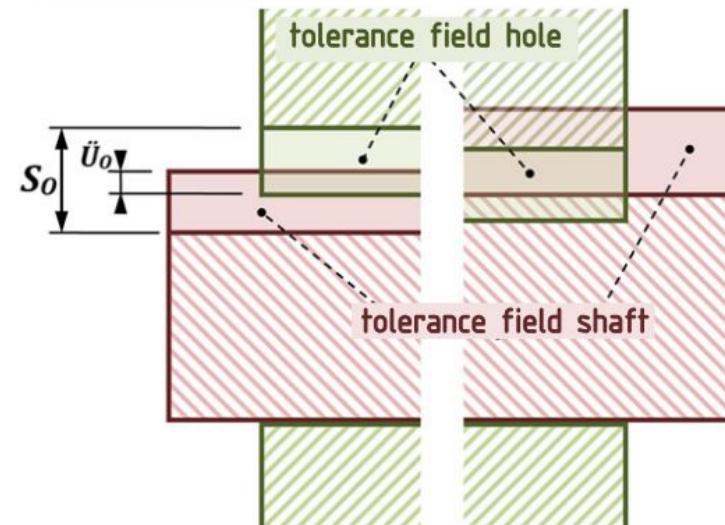
Types of fits / 3D Printing Design

clearance fit **0.5 mm +**



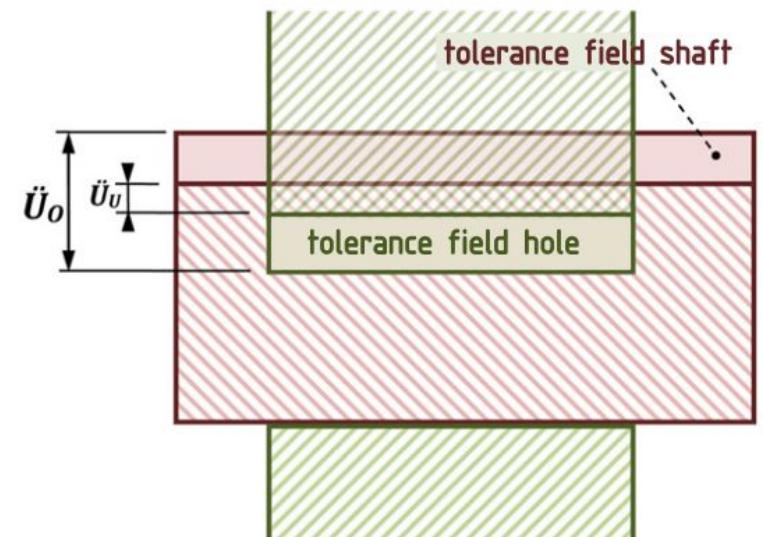
upper play: S_0
lower play: S_U

transition fit **0.2 - 0.4 mm**
two examples



upper play: S_0
upper interference: \ddot{S}_U

interference fit (press fit) **Exact**



lower interference: \ddot{S}_0
upper interference: \ddot{S}_U

[Taken from Wikipedia](#)

Gear Ratios

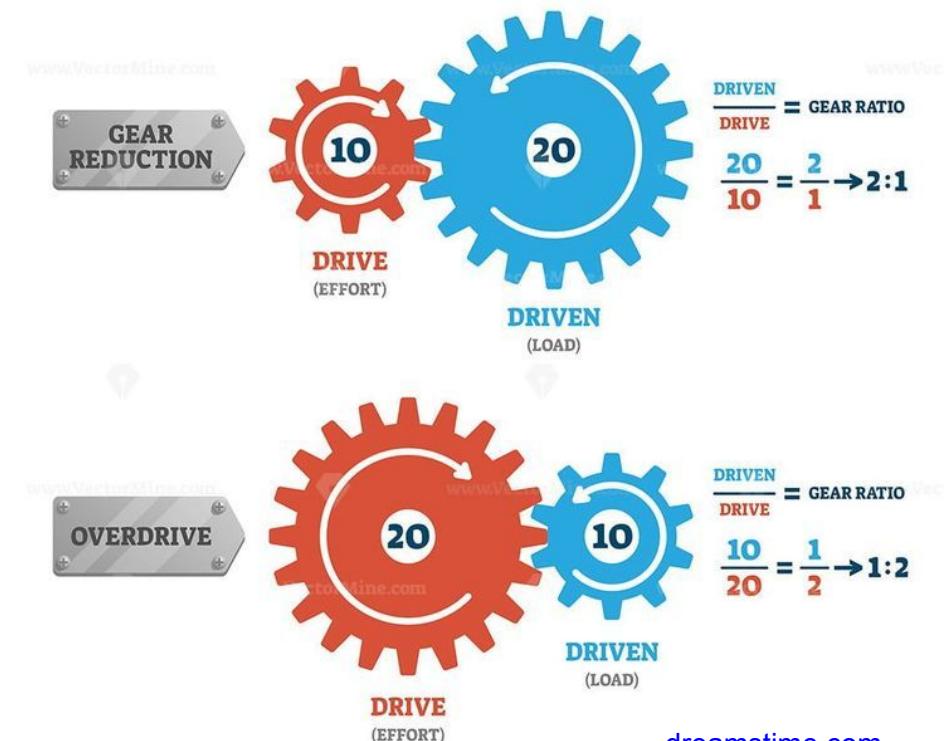


Gear ratio: The ratio of the number of rotations of a driver gear to the number of rotations of a driven gear

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

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



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