



# Baja Assemblies in Solidworks

Advanced CAD Training

# Outline



- What our master assembly looks like
- Use case: Adding a part to an assembly
- Strategies for assembling parts
- Use case: Making a part's feature align with another in an assembly
- Concept: References
  - Summary: Do's and Don'ts for References
- Use case: Making an entire part defined by others in an assembly
- Do's and Don'ts for editing master assemblies
  - Assemblies in Kenesto
- Summary and conclusion

# When Presenting



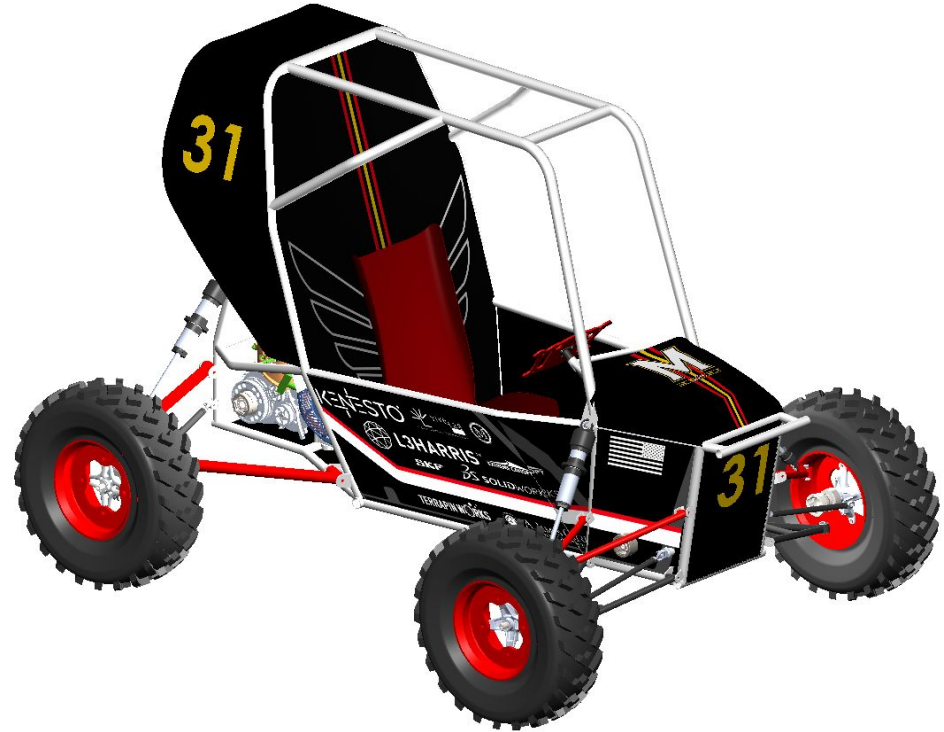
Have open in solidworks

- Master Assembly
- Parts to assemble into basic assembly (front shaft, see google drive)

# Inside Our Master Assembly



To SOLIDWORKS!



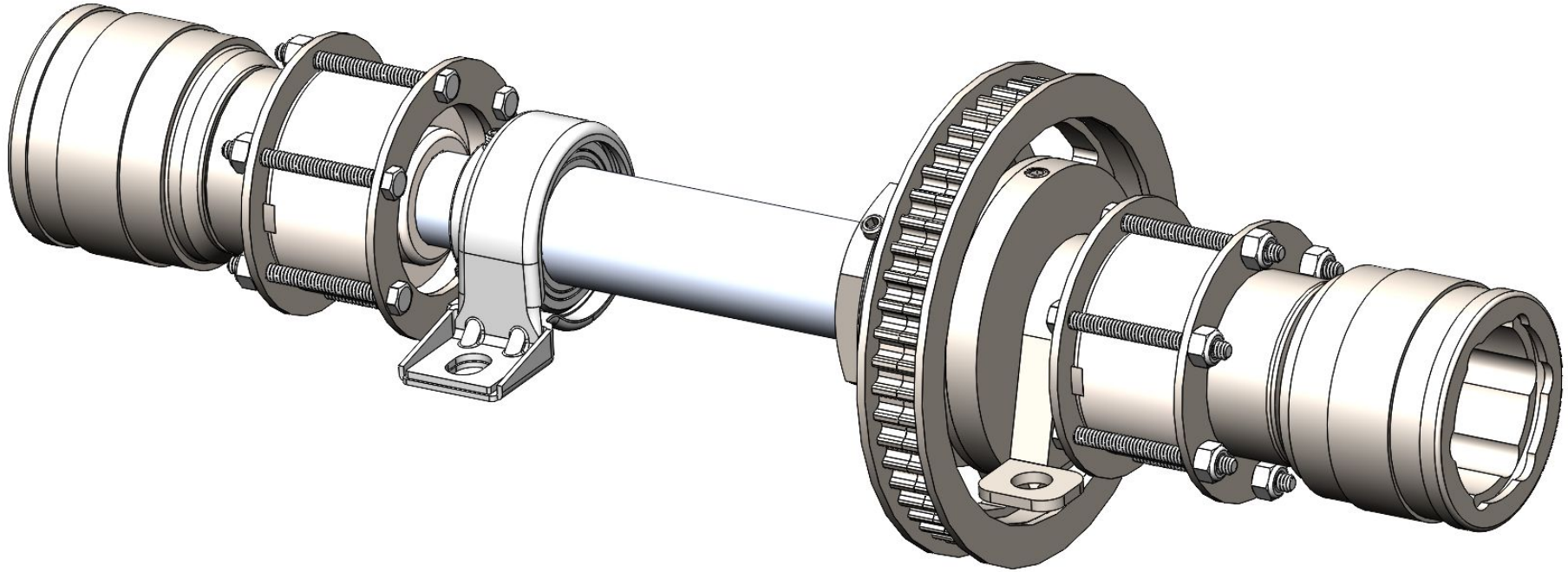
# The Fundamentals

To follow along, download the zip folder on Google Drive:

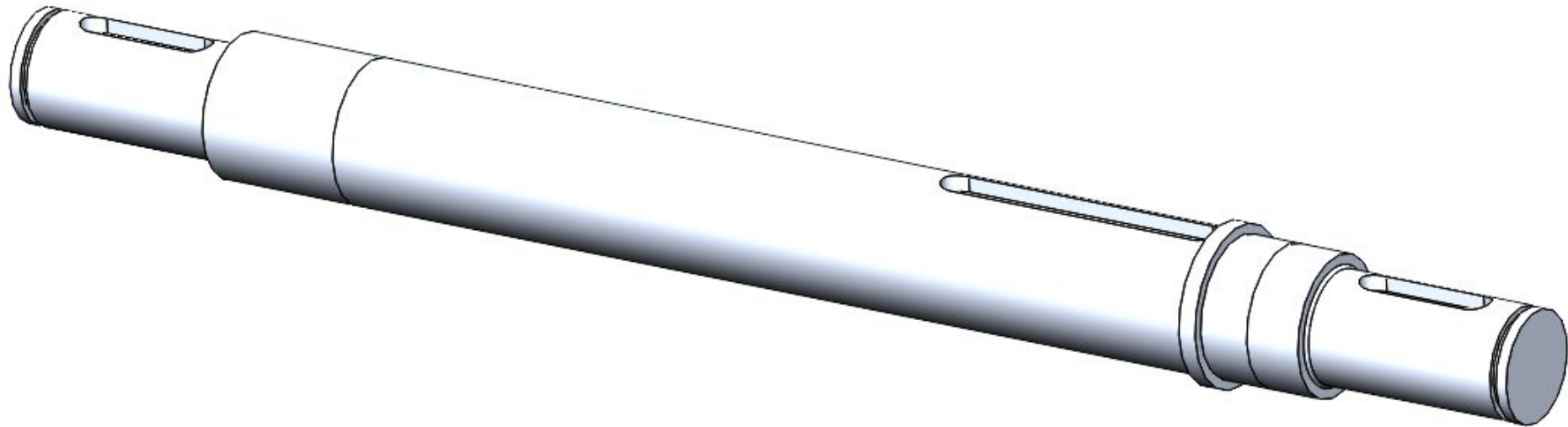
Folder: General Resources, CAD/FEA/Topology

File: [Assembly and Reference Tutorial - TR24 Front Shaft Files.zip](#)

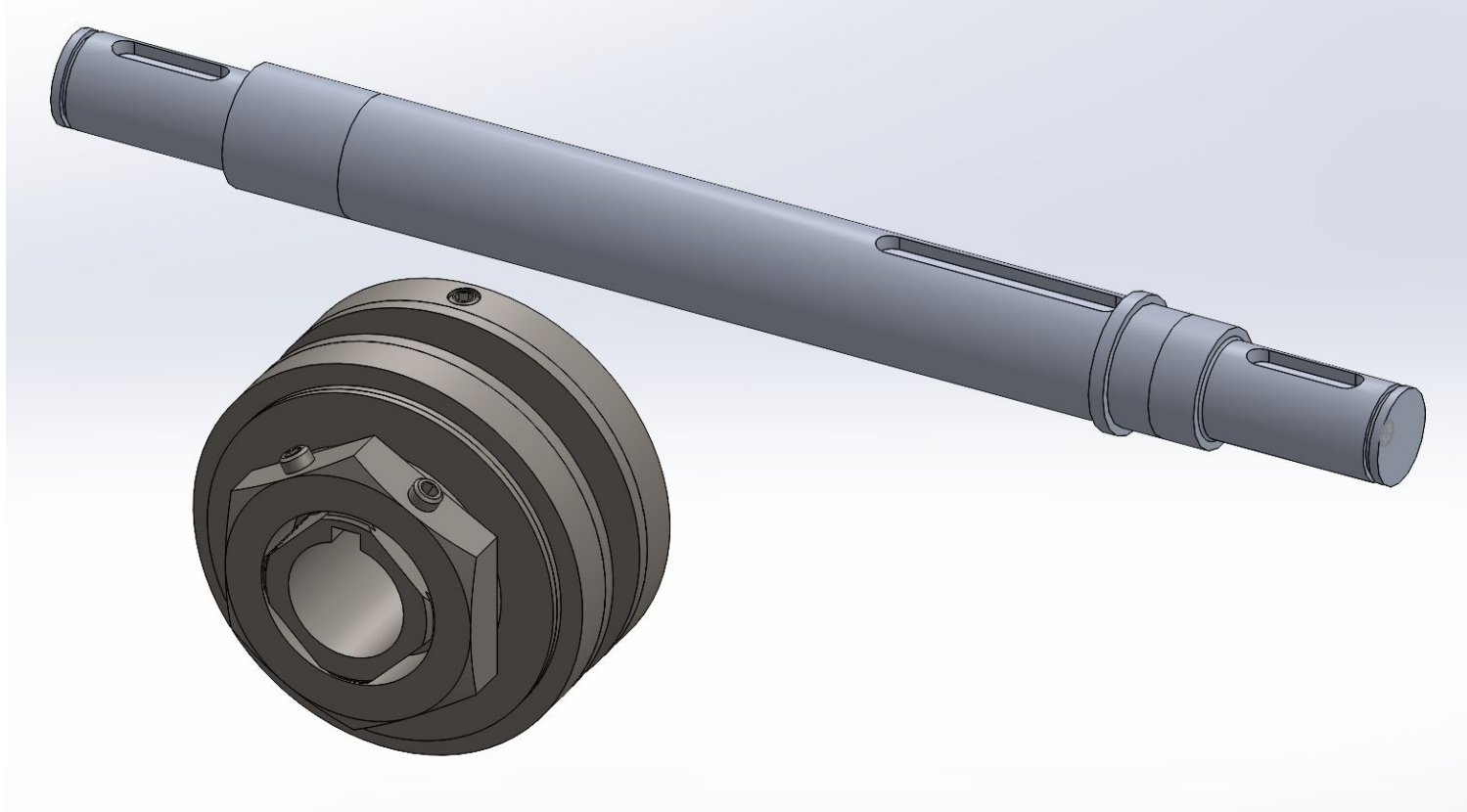
# Fundamentals: TR24 Front Shaft



# Adding Part to Assembly - Fixed at Origin



# Adding Part to Assembly - Free Floating





# Basic Mates - For Reference

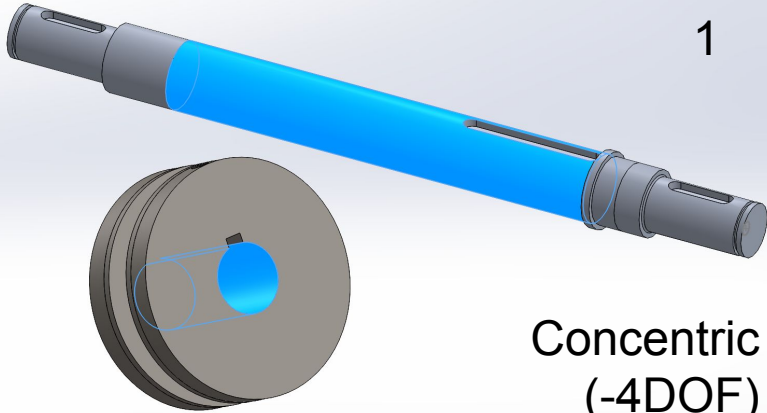


Mate Type	Works on Geometry •point,—line,○circular,■face	What it does, put simply
Coincident	• — ○ ■	Makes things touch
Concentric	• — ○	Makes things spin about the same axis
Parallel/Perpendicular	— ■	Parallel: face the same direction Perpendicular: face a different direction
Tangent	— ○ ■	Makes things touch, but for curvy faces (prefer coincident if possible)
Distance	• — ■	Spaces things a distance apart
Angle	— ■	Orients things at an angle from each other

# Adding Constraints

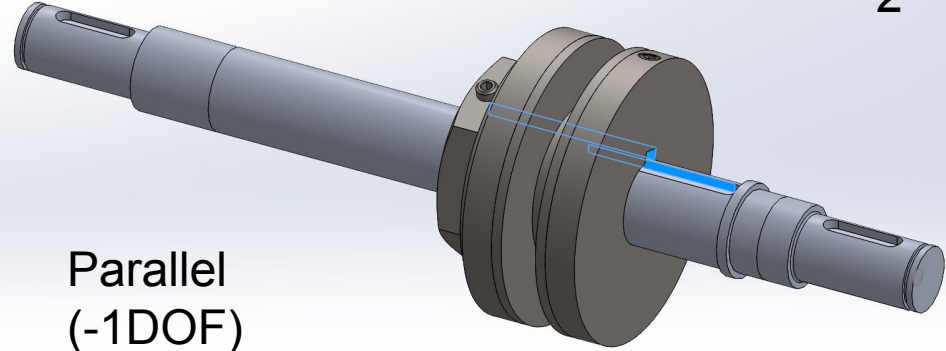


1



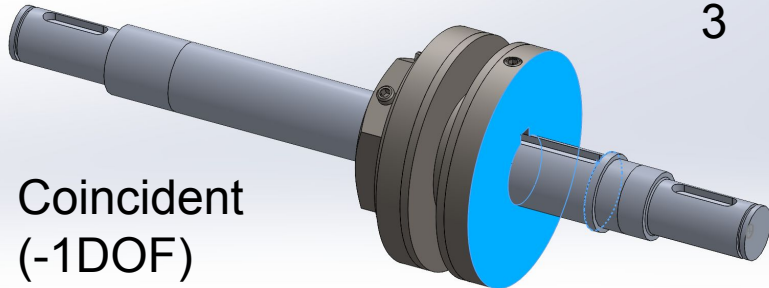
Concentric  
(-4DOF)

2



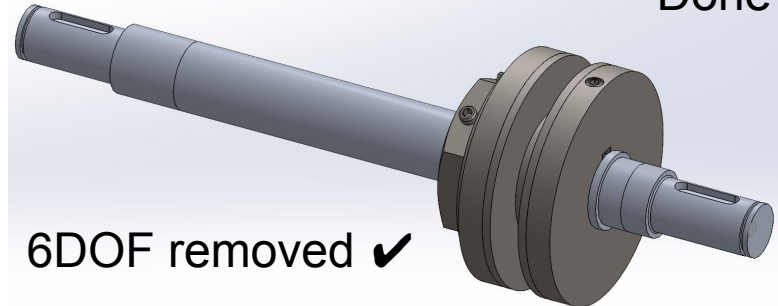
Parallel  
(-1DOF)

3



Coincident  
(-1DOF)

Done

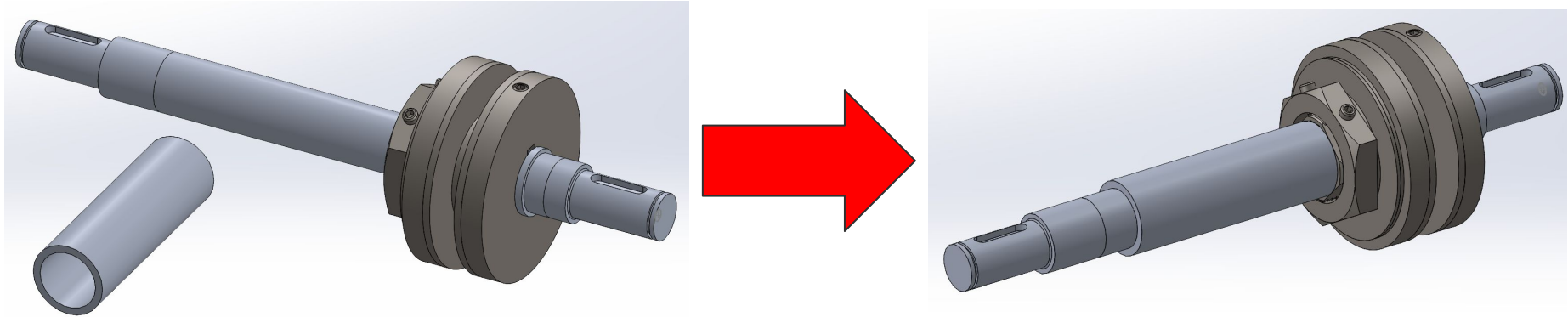


6DOF removed ✓

# Your Turn!



Add “TRB24-502-005.SLDPRT” and constrain it to be like this:



Can you do it with one constraint?

# This is cool, but what about real life?



How to make parts fit together and not interfere?

How to connect parts from different subassemblies?

How do I stop Solidworks from crashing?

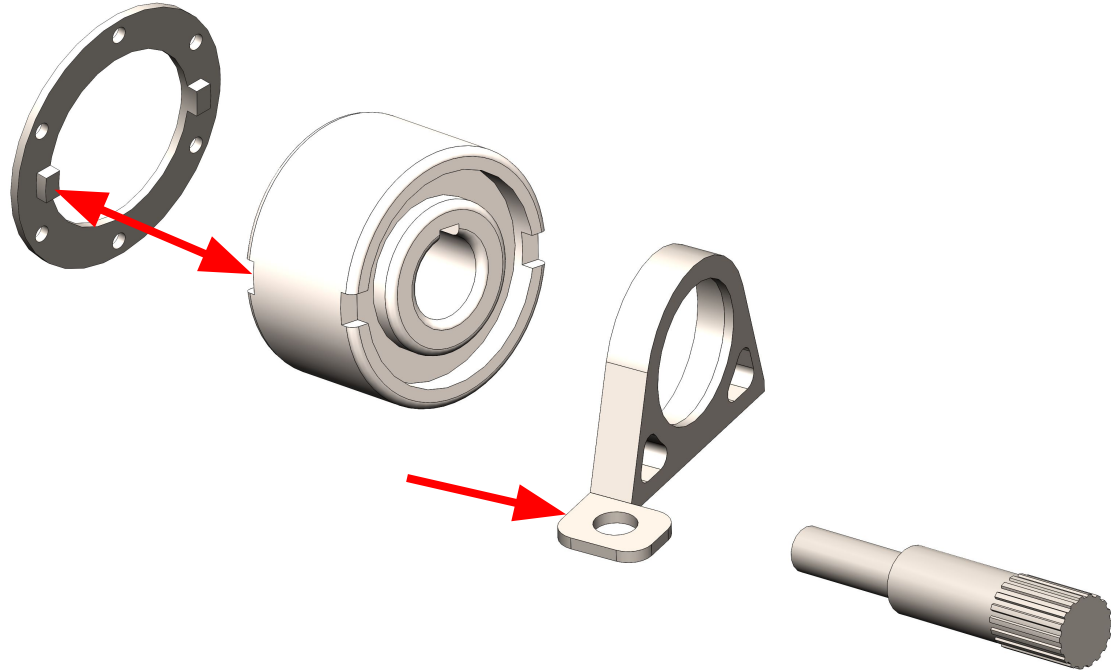


# Making Parts Fit Together



Many strategies

- Bolt Holes
- Locating geometry
- Tabs
- Splines
- etc...

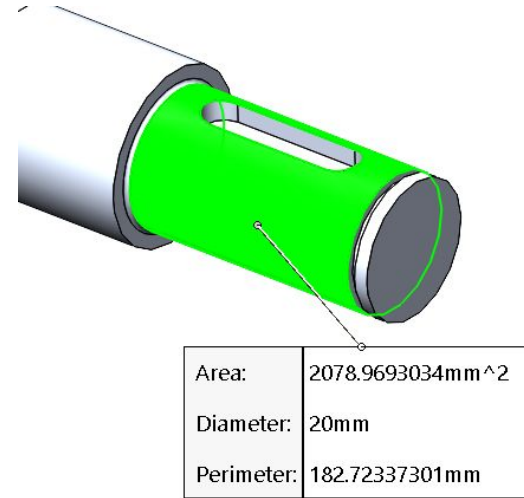
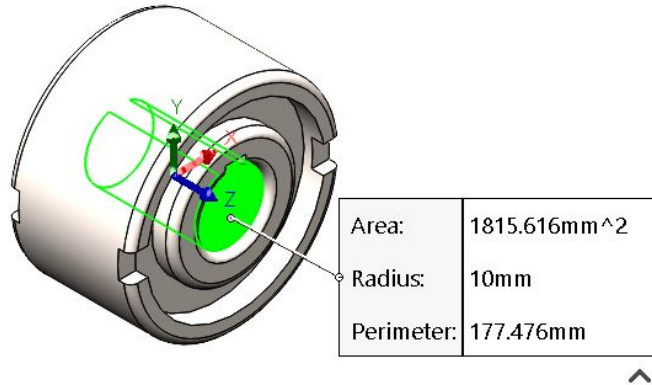


# Preferred Strategy: Manual



Literally copy the size/position of features where parts fit together

Example: Shaft diameter to fit clutch



Less likely to cause problems with Solidworks



Requires manually updating all fitted parts to change

# “Alternative” Strategy: References



A haiku, about references:

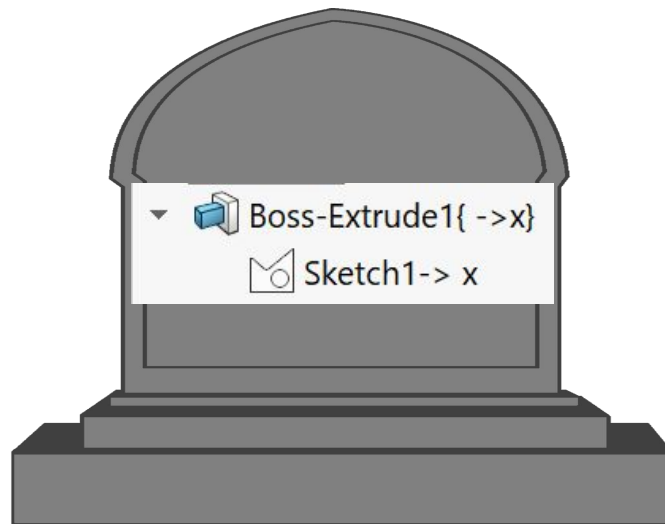
Avoid using them

Dark art causing suffering

Seriously don't

Anyway, here's how to use them

they're pretty useful sometimes

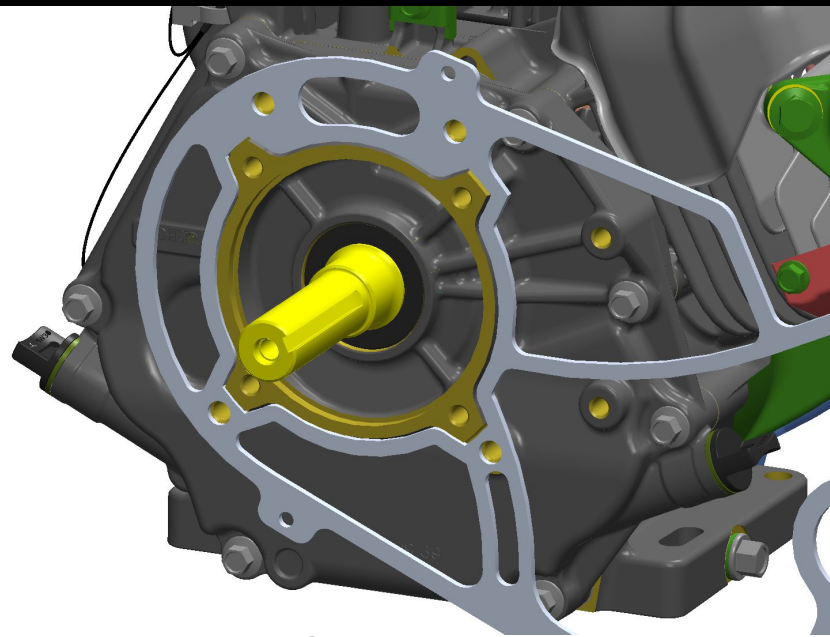


# Common Uses of References



- Copy a bolt pattern
- Copy a sketch from another part
- Make a part update with another when design changes
- Fit around complex geometry
- Aligning one part to multiple others
- Aligning part features with a layout sketch
- To annoy your colleagues

Tempting, right?



TR24 CVT case engine mount uses references



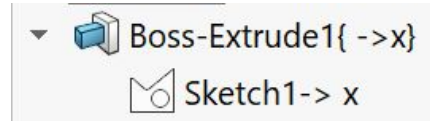
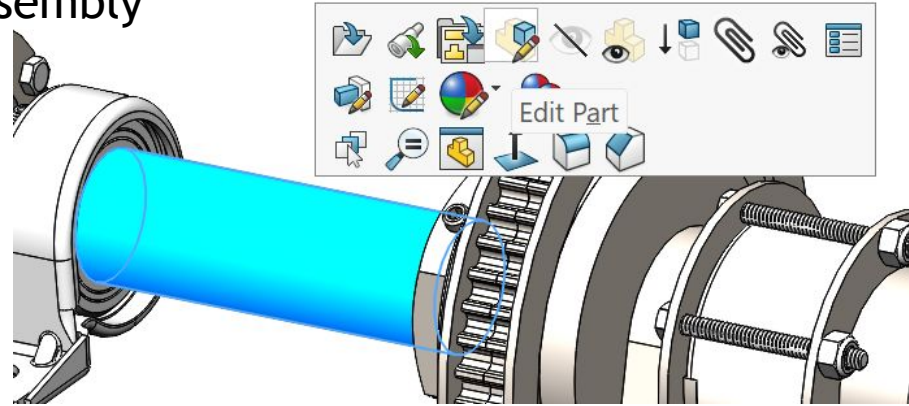
# Basics of References



1. Insert part into assembly
2. Fully constrain its motion
3. Right click on part → “Edit part” in assembly

What does this do?

- Like editing a part on its own
- You can reference other the parts
  - In sketches, with relations
  - In features, like “extrude up to face”
- A cool arrow will show up next to features using references



# Let's do some references - follow along



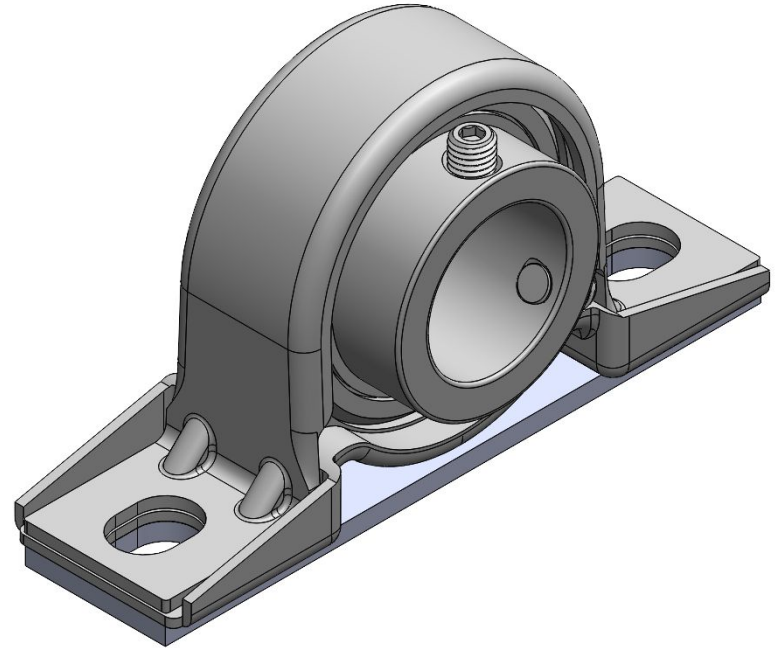
We want to bolt this part to something, but we don't know the dimensions of the slots

So we will use references!

Make a new part, a plate with dimensions:  
4.25" x 1" x 0.125"

Add "5913K64\_Low-Profile Mounted Sealed Steel Ball Bearing.SLDPRT" to new assembly

Need to save new part and assembly before making references!



# Properties of References



References linked to specific assembly

- Using part outside of original assembly results in a missing reference (“->?”)
  - Causes problems if the original assembly is lost
- One part can have references from different assemblies

Locking a reference (“->”) (Always do this after editing!!!)

- Will prevent the reference from updating
- Can be unlocked later

Breaking a reference (“->x”)

- Will prevent reference from updating
- Cannot be unlocked, permanent change

# Problems with References



- Changing a part might break references to it!!!
  - Fixing broken references is often slower than the manual method
  - Broken references in sketches are especially difficult to fix
- Couples a part to a specific assembly
  - Makes editing the part more difficult in the future
  - The assembly and the part itself need to be kept together



# Assemblies in Kenesto



Master assemblies are read-only by default, must be “checked out” to edit.

Checking in/out can be done with the right-click menu in file explorer

Opening a regular file or checking out a read-only file in Kenesto gives you *exclusive* write access until you close the file or “check in” respectively

**NOT USING THE KENSTO ADD-IN WILL BREAK PERMISSIONS FOR EVERYONE WHEN YOU OPEN ASSEMBLIES!!!**

- Not using it makes all parts in an open assembly read-only under your name!!!
- This halts CAD progress, which is really bad!
- You will be blamed!

# Summary



Assemblies are how we make sure parts on our car fit together

References allow us to design parts to fit together more easily

References should be used sparingly and with caution

References should always be left locked by default

Use the Kenesto Solidworks Add-In!

# Cool Trick



Q: What if you can't fully constrain a part before editing in the assembly?

A: Mate the part's origin with the assembly origin (coincident, locked axes), then use a reference plane for your first sketch

Use case? IDK, but it could come in handy.

Questions?