FRC Getting Started

- **www.OnShape4FRC.com**
  - An excellent resource for getting started in OnShape for FRC
- **Education Plan Upgrade**
  - If you already created a “free” OnShape account you can upgrade to educational by going to this site and registering with the same email you used for your free account. You will keep all of your files.
- **How to onboard an FRC Team**
  - How to create an OnShape Team and get your team setup
  - https://www.onshape.com/en/resource-center/articles/how-to-onboard-your-first-robotics-team
- **OnShape FRC Modeling and Design Tutorials by Nick Aarestad(AKA CADandCookies)**
  - Really great tutorials on making FRC robots in OnShape
  - https://www.chiefdelphi.com/t/onshape-modeling-and-design-tutorials/385871?u=allengregoryiv
- **OnShape Collaboration Features Explained**
  - https://www.youtube.com/watch?v=jI3ZQBUJkzM

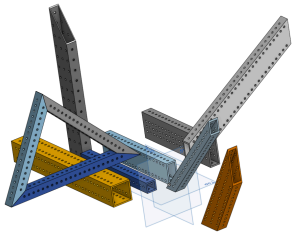
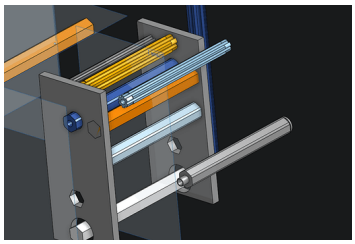
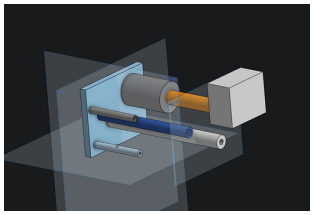
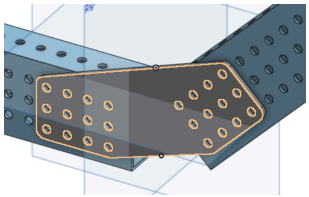
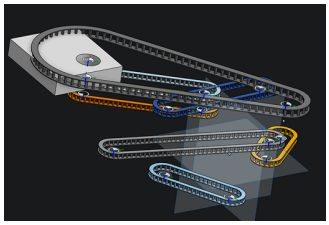
MKCad

- App Link: https://appstore.onshape.com/apps/Manufacturers%20Models/2ZT7X5D646R3LM3ZND7LGBTYRVM4SVH6CDDGM6I=/description
- Link: https://cad.onshape.com/documents?nodeId=3&resourceType=filter&q=type:document%20name:%22MKCad%22%20description:%22Official%22
- Video: https://www.youtube.com/watch?v=Fwjbc5_c3V4
- 2021 CD Thread - https://www.chiefdelphi.com/t/mkcad-2021-requests-and-updates/390651

Feature Scripts

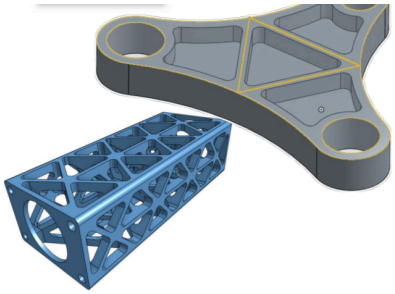
Julia’s Featurescript

- Numerous awesome FRC specific feature scripts will save you lots of time.
- CD Post: https://www.chiefdelphi.com/t/some-helpful-featurescripts/361652
- https://cad.onshape.com/documents/95c00401c440b44ad8799ef5/w/1f1ebce01a3b8eb6fa102975/e/763aab7b63ad3c30e7177daa
- Extrude Individual
  - How To - https://onshape4frc.com/blog/using-extrude-individual/
  - Video - https://www.youtube.com/watch?v=Fwjbc5_c3V4
- Tube Converter
  - How to - https://onshape4frc.com/blog/using-tube-converter/
  - Video - https://www.youtube.com/watch?v=Fwjbc5_c3V4

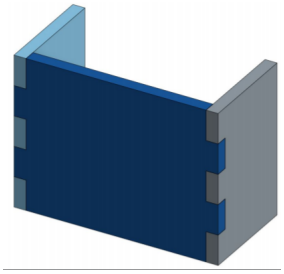
- 
- Shafts
  - How to - https://onshape4frc.com/blog/using-shaft-generator/
  - 
- Spacers
  - 
- Gusset Generator
  - 
- Chain Path
  - 
- And More

Lighten

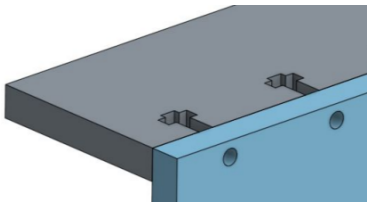
- “This custom feature removes material specified by sketch regions to make parts lighter. The sketch regions are extruded, offset, and filleted based on the desired remaining wall thickness and the radius of the cutting tool.”
- https://cad.onshape.com/documents/573f7d70e4b0fddafb52148c/v/85bb63ae5685ae3614116952/e/2dc22a6a95896a5243812d07
- How To- https://onshape4frc.com/blog/using-lighten/

- 

Laser Joint -

- https://cad.onshape.com/documents/578830e4e4b0e65410f9c34e/v/4a3153e5b4fb5385bd22a9b8/e/7af109b2f1cead90850525ae
- **Video** - https://www.youtube.com/watch?v=9JMZv3VPfvM
- 

T-Slot Joint -

- Description: https://forum.onshape.com/discussion/4498/updated-featurescript-t-slot-joints-for-laser-cut-parts
- https://cad.onshape.com/documents/5791a167e4b03c2aa6af3b35/v/7d9bb61358eb34c5bbcd9f2c/e/598fe63e4fe6846d3ebb2fbf
- **Video** - https://www.youtube.com/watch?v=9JMZv3VPfvM
- 

Auto Layout -

- Description - https://forum.onshape.com/discussion/3910/auto-layout-feature
- https://cad.onshape.com/documents/3b3bb87c95d03259328fdb1f/w/9828ddc941ddc2896ebeebdb/e/fcecc760e1bc713ee3aae876

Overcut/Dogbone -

- Exists as part of “laser joint” already but for any part
- https://cad.onshape.com/documents/b99915c0b73924ca981bc57f/w/fe0bd1c034e61bbcdb3e68eb/e/05dbdae41119d45d95588428

Awesome Featurescripts Github Page

- A more comprehensive list of Feature scripts

- https://github.com/dcowden/featurescript

OnShape Featured Featurescripts

- This is a bit out of date.
- https://www.onshape.com/featurescript

Measure Distance

- Measure Max and Min distance between two parts
- https://cad.onshape.com/documents/572b968ce4b07aad125dbaaf/v/08093fef6c4d4f330f24dd19/e/b40df94c5081948fe8195e81

Kerf Compensation

- https://cad.onshape.com/documents/57df36a88f63301089e8ac78/v/17d807acd78c9661ab3cbd3c/e/157268905828b0fc910e365b

Thin extrude

- Like Solidworks thin extrude, can use open shapes and add a thickness
- https://cad.onshape.com/documents/24819ddab7dc83c810eb8246

Surface Text

- Lets you configure text to be embossed or cutout of a part
- https://cad.onshape.com/documents/370197508187d5d8b361a414/w/64f8595ca2deaa8d5338db1a/e/fe4c2d2550f7c2d63b2704f9

Advanced Gear

- https://cad.onshape.com/documents/06467ec14c712fd27abe068d/w/e7be3ed1c164e01064645899/e/eb2e9443992685b24660527b
- Lets you make Helix and Double Helix gears

Shaped Flange

- https://cad.onshape.com/documents/602655eff016f183fc184978/w/c20bffc426bdb249281c9a38/e/5ff3b71883bf2073d7ceec14
- Lets you make a flange from a sketch profile, can also offset the flange easily so you can make “Bend Outside” parts nicely.

Part Number

- https://cad.onshape.com/documents/cb387c60a50e49e2a50a65da/w/38f957562c3238fb6debd3a3/e/c6024f1ff135baf098509c71
- Lets you rename parts based on variables and configurations

Part Color

- https://cad.onshape.com/documents/d997b0ffc30f659113b10c00/v/347f7240ed6eefd77e80907e/e/b6db0c9545b9a2df1d7ea546
- Allows you to configure part color from a configuration

Variable Library Feature Script

- Allows you to use the same variables across documents
- Items such as hole sizes, common additives, machine tolerances, can all be included in main document or your teams parts library.
- https://www.chiefdelphi.com/t/major-featurescript-release-plate-suite-variable-suite/395889

Own Sketch

- https://cad.onshape.com/documents/83e30bcaf55cf3dffeb60c71/w/9ee17983b16f012a3950b426/e/d6bcea4eaae4db001e3fb5ac
- Pulls out sketches from multiple sketches derive feature.

Realistic Chain

- https://cad.onshape.com/documents/1be886df2e36b431c92b2d61/v/2cc5a879879308d0d443846c/e/17a1b8e5fd0b4440efc1c3de
- Generates realistic chain on a path
- #35 chain is called #30-2

Thread Creator

https://cad.onshape.com/documents/6b640a407d78066bd5e41c7a/w/4693805578a72f40ebfb4ea3/e/f8aea9e5c33e02eab0854a4f

- Create Printable Threads in any size and pitch

Apps:

Kiri:Moto

- is an integrated cloud-based tool-path generator for *CAM*, *3D Printing* and *Laser cutting*.
- https://appstore.onshape.com/apps/CAM/EAAEWYIOMQKBENEMYW2N7MF253CT4WYL6SUJGEY=/description

LinkTab

- Create a tab linking to any website or service inside of Onshape Documents
- https://appstore.onshape.com/apps/Utilities/DFE73AMQ42NPMVAEQBQVP56QGLCWJ4ALJUBEBLA=/description

OneIPM

- “OneIPM provides easy-to-use, fully associative, project and task management completely within Onshape. It's also a powerful document browser helping users better visualize product structure and part versions.”
- CD Post: https://www.chiefdelphi.com/t/new-free-task-management-available-in-onshape/389982?u=allengregoryiv
- Sign Up Page: https://oneipm.com/first/
- How To - https://oneipm.com/help/

Features and Tips

Keyboard Shortcuts

- https://www.onshape.com/en/resource-center/tech-tips/tech-tip-keyboard-shortcuts

Configurations

- https://www.youtube.com/watch?v=pCyUPuDcVFk (good stuff starts at 4:45)

Custom Tables

- https://cad.onshape.com/help/Content/custom_tables.htm

Daily Checklist

- Starting: Update your linked documents and featurescripts, “pin reference” any linked documents that you don’t ever want to update.
- Ending: Version your documents, you can have infinate version. Especially if you are doing a multi-document robot project.
- Ending: Assign proper names to your parts
- Ending: Assign material properties to your parts

Workflows

Lasercut/routed box construction

- 2D Sketch -> Extrude Indvidual -> Laser Joint - > T-Slot Joint -> Auto Layout -> Kiri:Moto CAM

Multi-Part Part Studio vs In-contex vs Derived Parts

- A bit old (2017) looking for a newer one https://www.onshape.com/en/resource-center/videos/in-context-vs-derived-vs-a-multi-part-studio

Featurescript Workflows

- "If I were to model anything from rectangular tubing today, I would use the extrude individual and tube converter scripts."
- "If I were modeling something like a round tube for an intake or just something less common and more custom, I would use the custom length extrusion."
- "If I were modeling anything else (spacers, shafts, etc.) AND I wanted them in the Part Studio, I would use the custom length extrusion. Most of the time though, I would just use a configurable part imported straight from MKCad to my assembly."

“Sketch Block” style sketches

- Create a document with a part studio that has the sketches you want to use in other documents
- You can then insert those sketches into any assembly, and mate them where you want and then if that assembly is in context with any part studio you can use those sketches in sketches you create.

Mechanisms/Subsystems that move

- If part of your subsystem should move, such as a climber extending or an intake deploying,

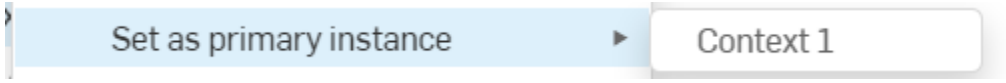
there should only be one revolute/slider mate for each of the degrees of freedom of the mechanism.
- Mate the main moving component with a revolute/slider.
- Mate everything else that moves with that part with a fixture mate to that moving part.

Derived Configurable Parts

- You can derive configurable parts and join them to other items
- We use this to combine multiple pulleys and attach it to a hub for a roller
- You can take a step further and use configurations to change that new part and the derived part in the part studio.

Multi-Document Robot

- Create a document for each subsystem (drive train, intake, etc) and a document for your main robot assembly.
    - We numbered them 0. Main Robot, 1. Drivetrain, etc so they sort nicely in a folder.
- In every document they will likely contain three tabs
    - 1 Assembly - this is the assembly used by outside documents, this includes all the parts and COTS parts for that subsystem and for main it has all the other subsystems inserted in to it.
    - 2 Part Studio - This is where the vast majority of parts for the subsystem will be created.
        - In main robot we use this part studio to create multiple sketches that each subsystem use to help start creating it's geometry.
        - Subsystems may be broken up in to multiple part studio depending on their complexity and how integrated they are.
    - 3. Construction Assembly - This assembly is used for adding a full copy of the main assembly that you will use to make your part studio in context so you can reference other parts of the robot.
- Ensure the origin of each part studio and assembly stays consistent so when you insert parts they are already in their correct locations and you can you use the group mate to simplify your process.
- In the subsystem part studios, Derive the sketches from the Main robot part studio, that will help you keep items consistent.
- Versioning and Updated Linked Reference will be the things you have to do the most to keep your documents synced. Version multiple times a session.
- "Pin Reference" lets you stop an item from being included in the Uplated All Linked document list, this is very useful for items you don't have ownership for like MKCAD items.
- "Set as Primary Instance" is very useful for allowing you to update the context in a part studio. The option is in the assembly though.



OnShape Sheet Metal

- Sketch your first face, use Sheet Metal Mode "Thicken" to create your base flange.
    - This is where you set your bend radius and k-factor
    - You can also change the bend relief method to tear if you plan to CAD your own or have

bends outside the main flange.
- Use the Shaped Flange Feature script to convert sketches on the side into flanges, you can use off-set to create bend outside flanges which the normal flange tool doesn't allow

OnShape ESP/DXF/AI/SVG to Part

- Convert the vector image to a DXF
    - Websites like this can help - https://cloudconvert.com/eps-to-dxf
- Import the DXF to Onshape the document
- Make a sketch in a part studio and use the import dxf button to place the DXF in the sketch
- Then extrude the sketch as you like to make the DXF into a part.

OnShape Wishlist

- Linear pattern based on a mate connector instead of a dimension
- Fix Replicate or add "Copy with mates"
- Mirror
- Off-line mode even if just view only
- Feature Scripts
    - VerasRoller style shaft generator, doesn't add anything and puts a hole for a 5/16-18 thread, can add snap rings in from the faces that you select.
    - Spacer generator that allows for ID to be set
    - Gusset Generator, allow no fillets on gussets, helps with sheet metal parts