The Compass Alliance Pathways: Computer Aided Design (CAD)

Before you begin learning about CAD, first make sure you're in the right place: this pathway is specifically for FRC *modelling*, while there is another pathway (Design Pathway) that provides resources about general robot design. If you are hoping to learn about the tools available to help you model a design, you're in the right place. If you want to learn how to design an FRC robot, check out the other pathway.

In this pathway, we will explore the software of two CAD platforms, SolidWorks and Onshape. While there are many more platforms available, many of the same concepts transfer between platforms, and after learning one it will be much easier to pick up others.

Level 0: Getting Started

1. **What is CAD?**
    - Powerful tool that allows your to model and analyze your robot on a computer prior and during manufacturing
    - View interferences with parts and assemblies while it is still correctable (i.e. not wasting part material making a part to only later find out it interferes and needs to be changed)
    - Develop advanced designs (such as custom gearboxes)

2. **OnShape**
    - Why OnShape?
        - Easy to get started, no installation necessary
        - Great documentation available for beginning designers
        - Powerful collaboration and data management tools
    - Setup
        - Just create an account at https://www.onshape.com/
        - FRC teams can get a team-wide education account: https://www.onshape.com/edu/frc

3. **SolidWorks**
    - Why SolidWorks?
        - Commonly used in FRC
        - Many advanced features for experienced users
        - Commonly used in industry
            - Design students may end up taking design/engineering courses that use SolidWorks as their software of choice
    - Setup
        - Obtain a Solidworks License
        - Installation:
            - System Requirements

- [Download Link and Instructions](#)
- Disclaimer: computers must use some version of Windows operating system (or Mac computer must use Bootcamp to operate with Windows system for SolidWorks)

4. **Alternate CAD tools - if you choose a different software, TCA's Call Center is available to answer any questions regarding getting started.**
   - Autodesk Inventor
     - Multi-body parts (i.e. ability to design an assembly within a part)
   - Autodesk Fusion 360
     - Cloud saving to prevent unsaved work from being erased
     - Runs on both Windows and Mac computers!

Creating Your Team Workspace

1. **Team Workspace with OnShape**
   - Since OnShape is designed around the value of maximum collaboration, teams are already built into the features. Just check out this blog post to set up a new team: https://www.onshape.com/cad-blog/introducing-onshape-teams

2. **Team Workspace with SolidWorks**
   - Because SolidWorks itself isn't cloud-sharing like other softwares, it's necessary to have a different application enable your ability to have all files saved in one central location that is accessible to anyone who needs it.
   - Many teams use [GrabCAD](#) (available to download GrabCAD Workbench desktop app [here](#))
   - Getting started with GrabCAD:
     - Create your team's folders for what files you anticipate creating so you can keep everything organized
       - On 1678, they use some of these common folders:
         - CAD Training for 2017 (where every team member keeps their Training files)
         - Reference (for installing default templates)
         - 3D Printer (where every STL file is saved that will be used to print a part)
         - FRC Robots (folder for other team's robot CAD who choose to release it to the public)
     - Check out some sample parts within [1678 Public CAD Library](#) and Annual Robot CAD Releases
       - [2016 Robot (Stronghold)](#)
       - [2017 Robot (Steamworks)](#)
       - [2018 (PowerUp)](#)

3. **Setting up a COTS library**

- While designing your robot, you will use Components-Off-The-Shelf, or COTS parts. These are parts that you can purchase from vendors. Examples include: bolts, motors, sprockets, and more!
- Many teams have set up COTS libraries for other teams. This will make it much easier to quickly access the parts when you need them.
  - For SolidWorks, 3847 has set up a GrabCAD folder with many of the COTS components you will need: https://workbench.grabcad.com/workbench/projects/gczMZ2vZJYlzPJlo b9wznnuhi1uqUhYIRx23KRz0l5Gq_g#/space/gcmegFZmFlpCU6pC0lOik iembdYGx3tphi11JfHJ0THJkK/folder/1357747
  - For OnShape, team 1836 has a similar folder: https://www.chiefdelphi.com/forums/showthread.php?t=160516
- If you run into a situation where the library is missing a part you need, many FRC vendors include models of the parts in their website. Look for buttons like 'Download' or 'Tech Specs'. If they don't offer the file in the correct format, both you can always download it as a .step file, and both SolidWorks and OnShape will let you import it.

Level 1: Introduction to CAD

1. **Learning OnShape**

   – The best part about OnShape for a beginner is the extensive documentation available for getting started. The documentation is much more comprehensive than SolidWorks' equivalent. After creating an account, the documentation can be found here: https://learn.onshape.com/learn/dashboard
     – Start with the two unpaid fundamentals courses. After completing each, you can begin branching out to the more advanced topics. Note: some of the courses list a price, however these are meant to be instructor lead, and are not relevant in this context.

2. **Learning Solidworks**
   – Warning
     - After every large change to a Solidworks file, press Ctrl-S to save it. Solidworks is not cloud based, and frequently crashes. Don't lose your hard work.
   – Proprietary Solidworks Tutorials

     - With the software comes tutorials that will help any beginner understand how to use the software features.
     - They are not comprehensive, so hopefully this Pathway can fill in many of the gaps that the tutorials leave
     - Have every member who wishes to use SolidWorks complete all the tutorials (individual parts, assemblies, and part drawings) so that every student has a base level of understanding.

- 1114's Simbotics Solidworks Series
    - This 35-part series is a great way to learn with bite-size videos, which is way more informative than any text-based, step-by-step design tutorial
    - Have students walk through these tutorials video by video, and have students reference back to individual videos as needed.
- 973 Remote Assistance and Mentoring Program (RAMP) Tutorials
    - YouTube channel on various topics related to SolidWorks, Onshape, and just general design
    - When 1678 uses SolidWorks, all students walk themselves through both parts of designing and assembling a West Coast Drivetrain via the RAMP videos (Part 1 linked here and Part 2 linked here)
        - It's suggested that you watch these videos on 0.5x speed if you learn better visually (watching the mouse move and click) over audibly (listening to Adam Heard talk through what he's doing)

Information Not Covered in Tutorials:

1. **Good Design Practices**
    - Best practices are important for collaboration, accuracy, and most important resiliency. When done wrong, models can become unworkable messes. Implementing good practices from the start will save time in the long run.
    - Part Files (Solidworks and OnShape)
        - Always constrain your sketch features!
            - A sketch is fully defined when all the sketch entities (lines, circles, etc) are colored black.
            - If a sketch is unconstrained, it means the software does not have enough information to be sure where you want every line. If another person were to open the file and accidentally drag a line, it might move and mess up the part. Never leave a sketch unconstrained.
        - Design for symmetry
            - If you part has any planes of symmetry, always align the part so the symmetry is across the default planes (Front, Top, or Right). This way, it is easy to mirror features, or insert the part into a flipped position in an assembly
        - Design Tree Usage
            - Every time you create a new feature in a single part file, it will show up to the left of your feature display. The order of this tree is important; the features that come further down can reference the features further up. It's important to logically sequence your features so they do not become cumbersome later on.

- Correct ordering of the design tree requires looking ahead at the design and deciding what features are going to be most important to the design.
  - Any reference sketches or geometry should be kept at the top
  - The central, or 'core', features should always come next. These are the large, general features that you know will always exist with this part
  - Any holes, slots, or features that reference other parts should come next. At this point you should be able to insert your part into the assembly, even if it is incomplete.
  - Finally, all details can be added. This includes pockets, fillets, etc. If not done last, they will most certainly become an issue if you change any of the above features (holes, large features, etc.).
- In addition, you can name your features or add them to folders to further organize the parts
  - Right click your part in the design tree, select "rename part"
- Part Studios and Assembly files (OnShape)
  - Top Down Design
    - OnShape is built around the idea of top down design: you design mechanisms starting with a concept and working down to the details. On the other hand, when designing in SolidWorks you start with a bunch of parts and build them up into an assembly.
    - For this reason, OnShape adds more functionality to Part Studio files than Assembly files. In general, do as much of the design for a mechanism as you can in a single Part Studio. This way, all the parts are there, and can be easily changed later.
      - Any parts that reference each other should be built in a Part Studio, and Part Studios combined should create an assembly.
      - Assemblies are only there in order to add moving components, multiple instances of a part, and linked parts such as COTS components.
  - Mates
    - In OnShape, mates are very different than in SolidWorks. Use them as they are intended to be used - they define the degrees of freedom for a part. Most parts will only need one mate.
      - As an example, never add three planar mates when a single fastened mate will suffice. You can always add mate connectors to a part.
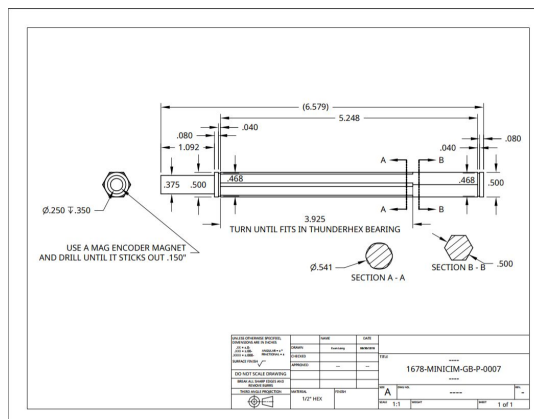
- Assembly files (Solidworks)
    - Base everything off the xyz planes
        - By placing the planes in smart locations, such as symmetry planes, mounting surfaces, or the ground, it can save you time later and make the model easier for others to understand.
        - In the same way, when mating an assembly use the xyz planes within the parts as much as possible. These planes can't be changed or deleted, so you will have less broken mates later.
    - Use the mirror feature wherever possible
        - In any assembly with a plane of symmetry, only develop half of the assembly, then mirror everything over. This will save time, and make sure that changes made on one side will also exist on the other.
    - Modular mating
        - If you know that components are meant to be stuck together, such as spacers and gears on a shaft, mate them to each other first, before mating to other components, such as a hole in a plate. This way, if the plate is deleted or changed, the shaft stays intact.
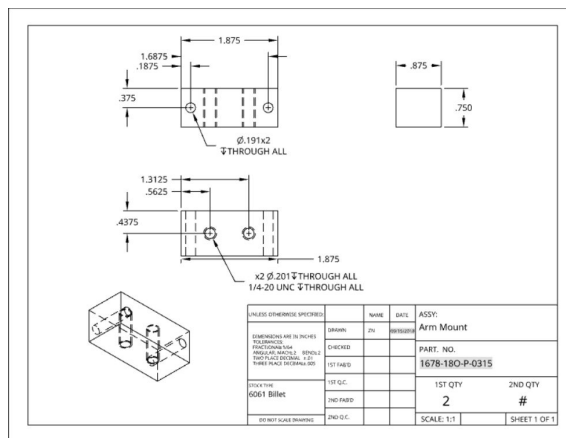
2. **Part Drawings**

- While there are several general part drawing tutorials included in the OnShape and SolidWorks built-in tutorials, these tutorials are not ideal. Part drawings are difficult to teach in a general context because every application is going to be different.

    - Note: Drawings in OnShape and SolidWorks are very similar, and knowledge of one translates very well to the other.

- In order to create successful drawings, it is necessary to have a good understand of machining. Drawings are a way to communicate how to make a part; if you don't know how to make it yourself, you won't be able to tell someone else how. It affects which views and dimensions you will need to include, and it's different for every machine.

    - Fore more information about manufacturing, check out the pathway found here: https://docs.google.com/document/d/1FxdnQ5w2kG_nVVp1DVH0lfQIUWBbNqq6M7LlAJeVzcY/edit?usp=sharing

- As a team, you will have the opportunity to decide for yourself what works best for your drawings. Just make sure there is open communication between the designers and machinists, so each is interpreting the drawings the same way.

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PATHWAYS

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FIND YOUR WAY FIRST

- – [This introductory video](#) does a good job explaining how to set up a drawing and create some initial dimensions.
  - – Note that the instructor says that the manufacturing industry is relying less heavily on drawings now. While is true, students operating manual machines in your own shop will still need detailed dimensions of your parts!
- – Here are some more videos on Solidworks Drawings. Note that there are many different ways to create drawings, and some are better than others.
  - – [This video](#) contains more information on creating and formatting your own dimensions.
  - – [This video](#) is much longer and more detailed. It will walk you through creating a drawing for a simple bracket.
- – As an example, here are a couple drawings that were used on team 1678:
  - – A complicated lathe part with a variety of features:

  - – A mill part:

3. **Parts Tracking**
   - – It can be helpful to create a naming system for all the parts on the robot. Some examples are:
     - – 1678-17-P-0231 (team-year-type-mechanism number + part number)

– Google sheets provides an excellent platform for keeping track of part statuses. Setting up a parts tracking spreadsheet will allow you to manage large quantities of parts. An example of 1678's spreadsheet:

Part Number	Assembly	Part Name	Designer	1st QTY	1st Status	2nd QTY	2nd Status	Comments
1678-18o-P-0201	Intake	Intake plate	Mingyo	4	Design in Progress			
1678-18o-P-0202	Intake	1/2 round shaft intake wheel	Mingyo	2	Needs Drawing			
1678-18o-P-0203	Intake	1/2 round shafts pivot	Mingyo	2	Needs Drawing			
1678-18o-P-0204	Intake	Hard stops	Mingyo	2	Abandoned			
1678-18o-P-0205	Intake	arm plates	Mingyo	4	Design in Progress			
1678-18o-P-0206	Intake	Shooter wheels shafts	Mingyo	2	Needs Drawing			
1678-18o-P-0207	Intake	60t shooter pulley	Mingyo	2	Complete			
1678-18o-P-0208	Intake	24t 30t dead axle pulley	Mingyo	2	Complete			
1678-18o-P-0209	Intake	36t space wheel plug pulley	Mingyo	2	Complete			
1678-18o-P-0301	Arm Mount	Arm Frame Tube	Zatara	2	Design in Progress			
1678-18o-P-0302	Arm Mount	Back Frame Tube	Zatara	1	Abandoned			
1678-18o-P-0303	Arm Mount	Sprocket Hub Spacer	Zatara	2	Needs Tool Path			Mill or Router?
1678-18o-P-0304	Arm Mount	Bent Arm Gusset	Zatara	2	Abandoned			
1678-18o-P-0305	Arm Mount	Arm Cross Beam	Zatara	4	Ready for Mill			
1678-18o-P-0306	Arm Mount	Arm Cross Beam Gusset	Zatara	4	Needs Tool Path			
1678-18o-P-0307	Arm Mount	Top Bucket	Zatara	1	Design in Progress			
1678-18o-P-0308	Arm Mount	Bottom Bucket	Zatara	1	Design in Progress			
1678-18o-P-0309	Arm Mount	GB Outside Plate	Sam	1	Abandoned			
1678-18o-P-0310	Arm Mount	GB Outside Plate	Sam	1	Abandoned			
1678-18o-P-0311	Arm Mount	GB Standoff	Sam	2	Abandoned			
1678-18o-P-0101	Drivetrain	siderail	Nathan	2	Needs Tool Path			
1678-18o-P-0102	Drivetrain	back frame	Nathan	1	Ready for Mill			
1678-18o-P-0103	Drivetrain	front frame	Nathan	1	Ready for Mill			
1678-18o-P-0104	Drivetrain	center frame	Nathan	1	Needs Tool Path			
1678-18o-P-0105	Drivetrain	frame gussets	Nathan	4	Needs Tool Path			
1678-18o-P-0106	Drivetrain	bearing support	Nathan	4	Needs Tool Path			
1678-18o-P-0107	Drivetrain	front shaft	Nathan	2	Ready for Lathe			
1678-18o-P-0108	Drivetrain	back shaft	Nathan	2	Ready for Lathe			
1678-18o-P-0109	Drivetrain	bellypan	Nathan	1	Needs Tool Path			
1678-18o-P-0210	Intake	3/8 Shooter Shaft	Mingyo	2	Needs Drawing			
1678-18o-P-0211	Intake	Shooter Spacers	Mingyo	2	Ready for 3D Print			
1678-18o-P-0212	Intake	Shooter Spacers	Mingyo	2	Ready for Mill			

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4. Using Chief Delphi as a resource
- Check out this Chief Delphi regarding just beginning to design and CAD
    - Pro tip: if you ever are stuck and have a question, try looking around different resources online to find your answer first. Most likely, other people on other teams will have had the same questions as you, and have already probably posted it in Chief Delphi. Look around Chief Delphi and google search results before asking a question within a forum like Chief.

Level 2: Forging New Paths

1. Advanced CAD concepts
– The following is a list of more advanced tools available that can greatly improve the efficiency and resiliency of modelling. These topics, however, are areas where different CAD platforms are rarely similar.
– Configurations (SolidWorks)
    – Configurations are a tool to allow multiple 'versions' of the same part to exist at once. They are excellent in cases where you need a very similar in multiple places, with slight changes between them.
    – 973 provides an excellent video on the use of configurations in FRC: https://www.greybots.com/videos.html (halfway down the page).
    – Note: because of the way SolidWorks handles suppression of features, parts with multiple configurations can quickly become messy and difficult to work. It is very important to order the design tree well (see previous section) and plan ahead at where you want the split in configurations to take place.
    – OnShape also has a feature named 'configurations', however it is not quite the same. It is a feature more used in libraries of parts.
– In-context design (OnShape and SolidWorks)

- In-context is a tool to let you edit a part in position in a large assembly. This is another place where the two platforms differ slightly. In SolidWorks, the contexts automatically update when the part is moved in the assembly, which leads to better version control but more common breaks. OnShape 'freezes' the context in the part, so you know it will never break.
  - OnShape tutorial: https://learn.onshape.com/learn/course/managed-in-context-design/what-is-top-down-design/what-is-top-down-design
  - Solidworks blog post: http://blogs.solidworks.com/tech/2014/06/an-introduction-to-in-context-part-editing.html
- Sheet metal (SolidWorks)
  - Sheet metal is a powerful design tool that lets you design complicated thin parts, then 'flatten' them at the end to allow for easy manufacturing.
  - For the SolidWorks documentation on the subject, take a look here: http://help.solidworks.com/2017/english/solidworks/sldworks/c_sheet_metal.htm
  - OnShape also has a sheet metal tool that is similar, however it is nowhere near as developed as the SolidWorks equivalent, and relatively limited in its ability.
- Multibody parts (SolidWorks)
  - Similar to configurations, multibody parts let you build up a series of highly related parts at the same time. They can share sketches and reference information.
  - http://help.solidworks.com/2016/english/solidworks/sldworks/c_multibody_parts_folder.htm
  - In OnShape, there is no need for a counterpart: the entire platform is built around the concept of top down design using the part studio feature.
- Global variables and equations (SolidWorks and OnShape)
  - Global variables and equations are tools available for the purpose of improving resiliency in part models. For example, rather than entering some critical dimension (size of wheels, for example) multiple places in a part, then needing to change the dimension later, you could create a single global variable and only change it once.
  - http://help.solidworks.com/2018/English/SolidWorks/sldworks/hidd_equation_manager.htm

2. **Templates (SolidWorks)**
   - Setting up a team template is a quick and easy way to standardize the base settings for every CAD file. Setting up a team template can help with collaboration and reduce CAD errors on your team.
   - For a quick guide of setting up templates check out the SolidWorks help section: http://help.solidworks.com/2017/english/SolidWorks/sldworks/t_creating_a_template.htm?id=2c7663562e4a4ce3bb656a7d8806a0d6#Pg0&ProductType=&ProductName=

3. **Mass Properties (SolidWorks)**
   – Mass properties are an excellent way to keep a rough estimate of the weight of the robot so there are no surprises a couple days before bag day. In addition, they can help your software team accurately control the robot or a specific mechanism by providing center of mass information.
   – In order for them to work properly, it is important to add materials to every part created. To add a material, right click on the 'Material <not specified>' item in the feature tree and click 'Edit Material'.
   – For COTS parts or parts without a uniform material, you can override the component material by searching for the 'Mass Properties' command, and selecting the 'Override Mass Properties' button
   – The mass of an assembly can be viewed by finding the 'Evaluate' toolbar and selecting 'Mass Properties'.

4. **Tips for training a CAD team**
   - <u>Pinpoint your team members</u> who are going to be committed to learning the software and participating during season.
     - On 1678, the number usually hovers between 4-10 people year-to-year. It's generally a smaller subteam given the amount of work it takes to learn a software and design year-round.
     - Don't be afraid to air on the smaller size, as it creates more freedom to communicate clearly, make sure everyone adheres to deadlines, and makes sure that there is enough work during season for everyone to be (and feel) productive.
   - <u>Pinpoint a leader or organizer</u> who can be in charge of keeping the team communication and progress on track
     - This can be a mentor or a student, however your team wishes to organize your design subteam.
   - Like in any efficient work setting, <u>good communication is key</u>.
     - Group chats or slack channels, as oppose to direct messages, keep everyone in the loop
     - Hold in-person meetings when students are training to make other members more available to ask/answer questions.
   - <u>Set manageable deadlines</u>
     - Be realistic. Have the students' abilities in mind to challenge them, but do not create deadlines that will overwhelm them and place quantity or completion over quality of design.
   - <u>Check each other's work</u>
     - Using GrabCAD enables everyone to be able to view each other's progress and correct each other's mistakes.

Appendix A - Revision History

Revision #	Revision Date	Revision Notes
1.0	Sept. 2018	Initial Release