FRC 6328 CAD Training

Week 2: Sketching

Course Outline

Week 1: Introduction to CAD and the training format

Week 2: Introduction to sketching

Week 3: Introduction to part design

Week 4: Effectively using Part Studios

Week 5: Assemblies and mate connectors in Onshape

Weeks 6-10: Using CAD to design for FRC and beyond

***We'll try to give opportunities for those who are more familiar with CAD to get ahead/do more advanced things in the first half of the training*

Week 1 Objectives

- Setup your Onshape account
- Link the CAD libraries and helpful scripts we'll be using
- Complete introduction training courses
  - Introduction to Parametric Feature-Based CAD
  - Navigating Onshape

Any questions from the week 1 materials?

Week 2 Objectives

- Learn all about sketching
  - Different shape types
  - Construction geometry
  - Sketch constraints
  - Why blue lines are bad
  - How *Use* is so powerful
- Watch the Onshape videos to prepare for next week

Where to do your work

- **You will be doing all your work in the provided document in our Onshape team**
  - You must have filled out the form from last week so we can add you to the folder
  - Document can be found under
    - Teams -> FRC6328 -> CAD Training -> Submissions -> <<Your Name>>
- Organize your files based on week by creating a new folder
  - Can be created from plus button in bottom left of document
  - You can create multiple part studios in the weekly folder for the different exercises

What you need to do this week

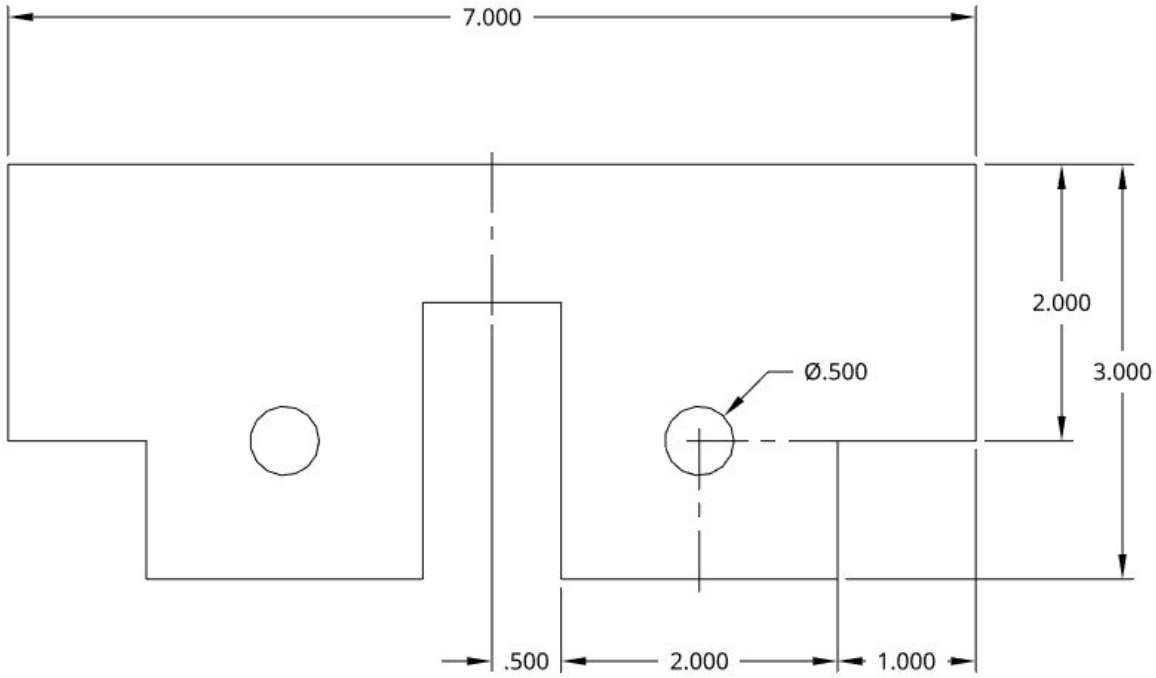
- Complete Onshape Learning Pathway - **Numbers 3 on sketching**
- Complete our sample sketches based on the provided drawings
  - 1 - Hatch intake component
  - 2 - Climber bearing block
  - 3 - Gusset
  - 4 - Climber hook
- Complete Onshape Learning Pathway - **Numbers 4 on part studios**
- Submit your results in the HW submission form
  - https://forms.gle/NytpYVaR1TfoiLPj6

General notes on exercises

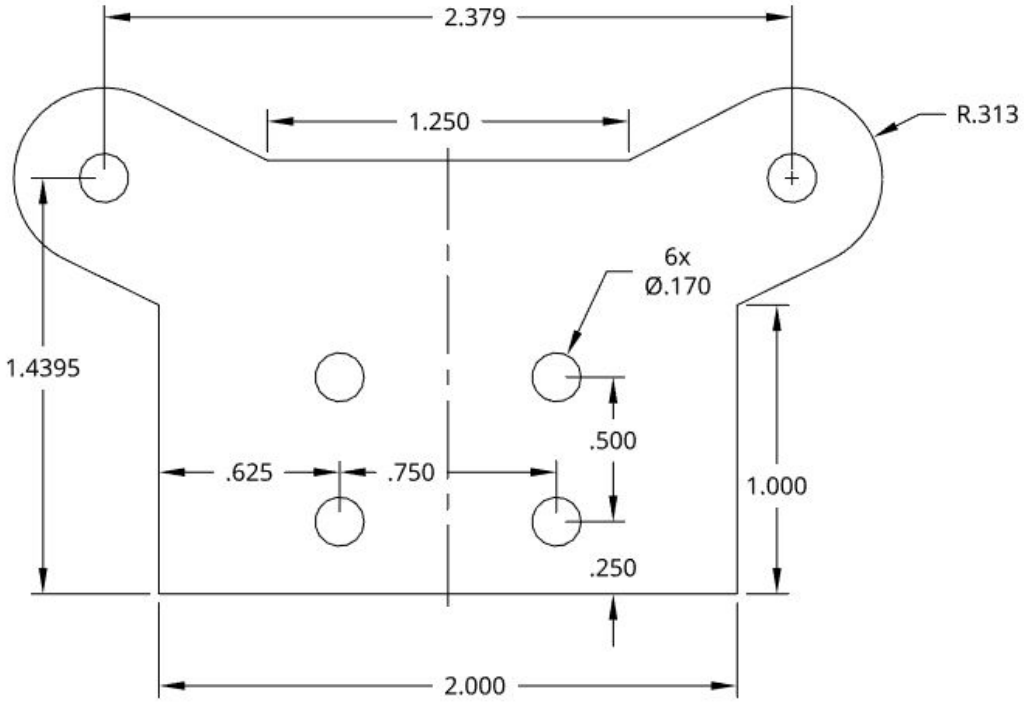
- Forgive us for not following ANSI standards in drawings. If you think some dimension is missing either reach out or make an educated guess
- You should have no blue lines when done
  - Blue means something isn’t fully constrained and you’re missing a dimension or constraint
- You can ignore any fillets (rounded corners)
  - These make sketches a lot harder to deal with and are normally added once in 3D

Part 1: 2019 Hatch Intake Component

Take advantage of automatically creating constraints as mentioned in ‘Using Constraints in Onshape’

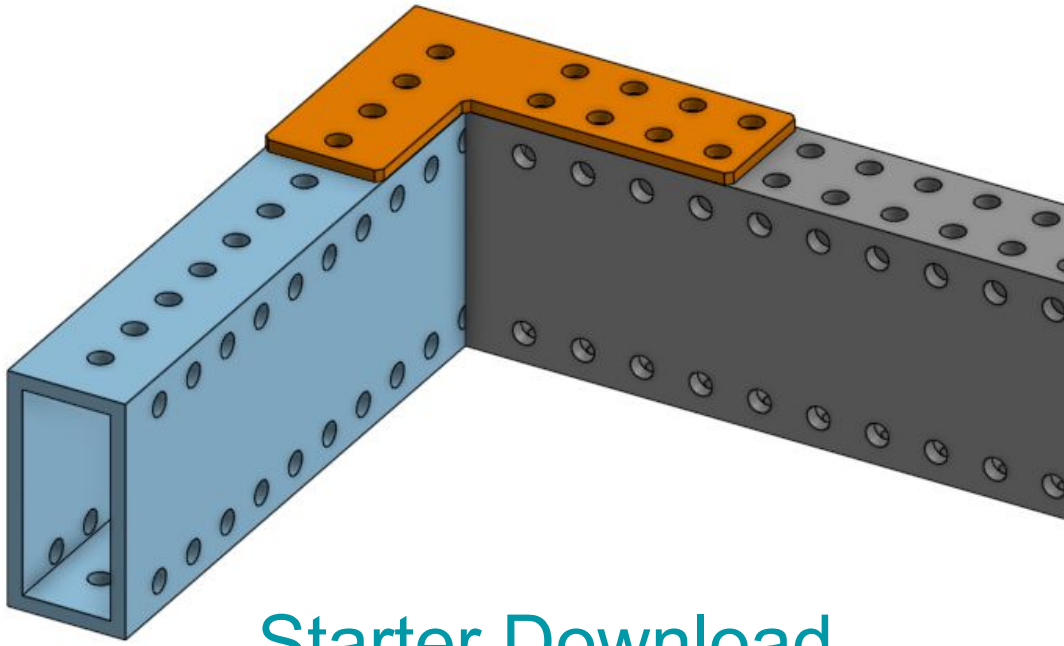


Part 2: 2020 Climber Bearing Block



Part 3: Gusset

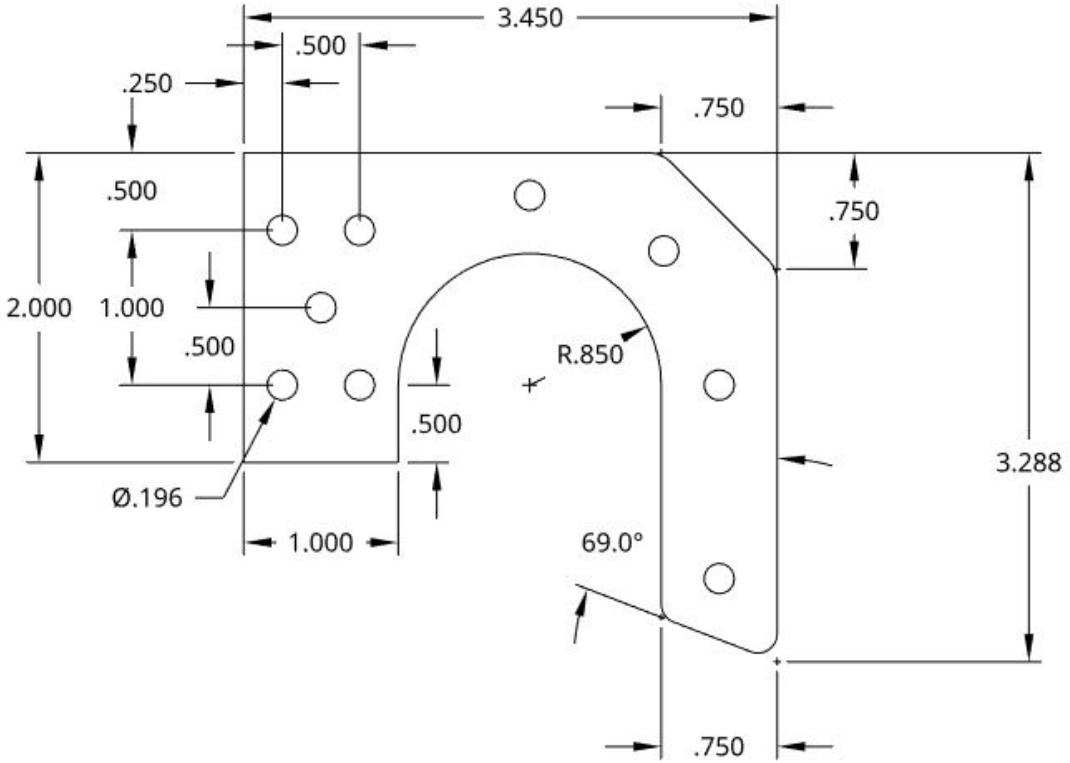
- This is all about learning to use the `use` command
- Download the starter file and import it into your document from the bottom left corner + button
- Start a sketch on the top face and design the gusset in orange
- It should cover 4 holes in each tube



Starter Download

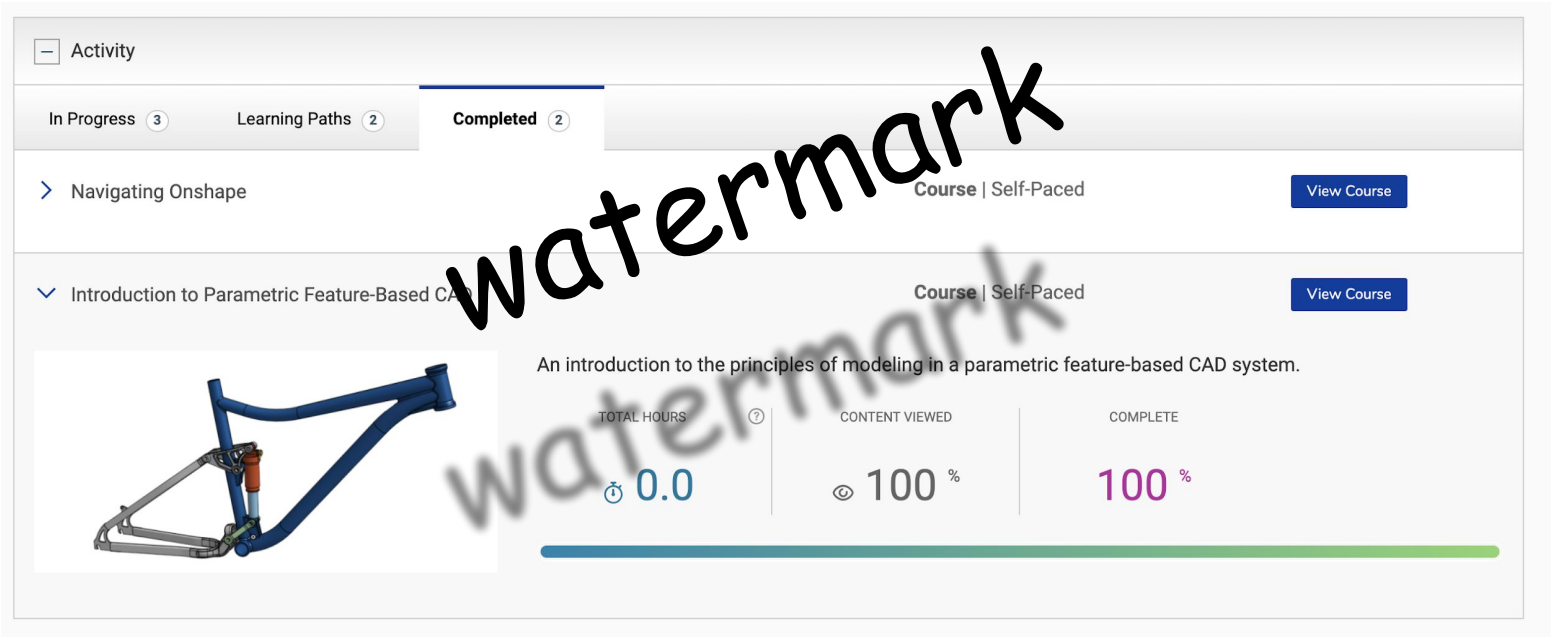
Part 4: 2020 Climber Hook

The right 4 holes are up to your judgement as to where exactly they go



Example Submission - Course Completion

After completing a course you can go to this link to view your completion status



Any Questions?