ARES Workshop - Onshape

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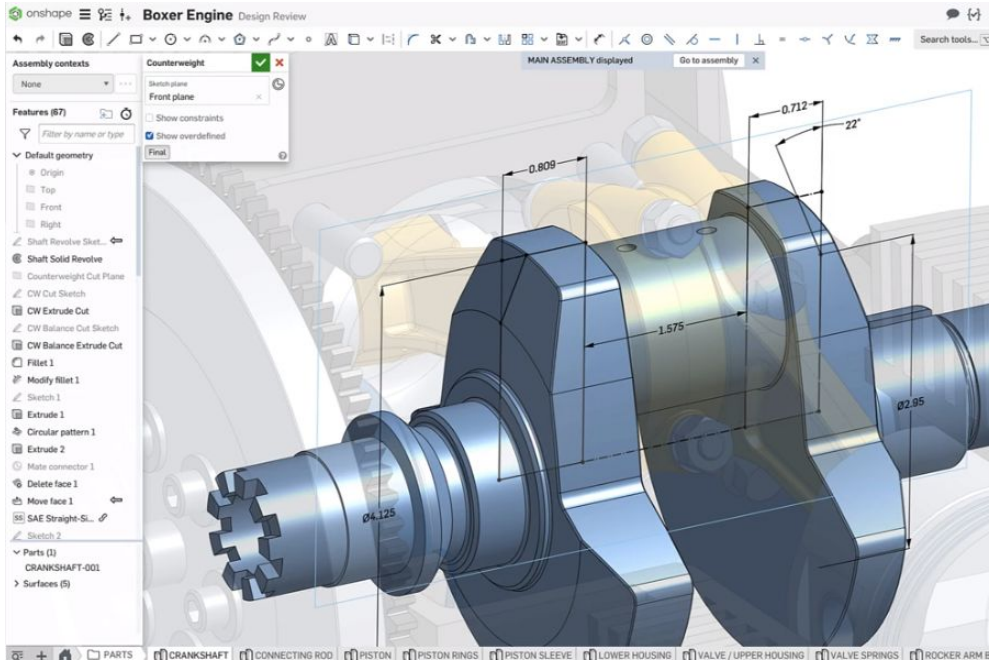
Onshape

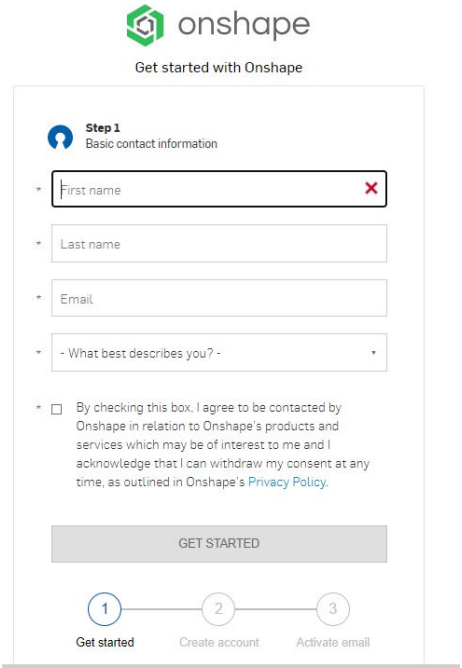
onshape®

- Cloud-based CAD system
- Allows multiple users to collaborate on one CAD document
- User-friendly interface
- Sketch in 2D, then model into 3D
- Can combine several parts into moving assemblies and see how the mechanism actuates
- Can export into various formats



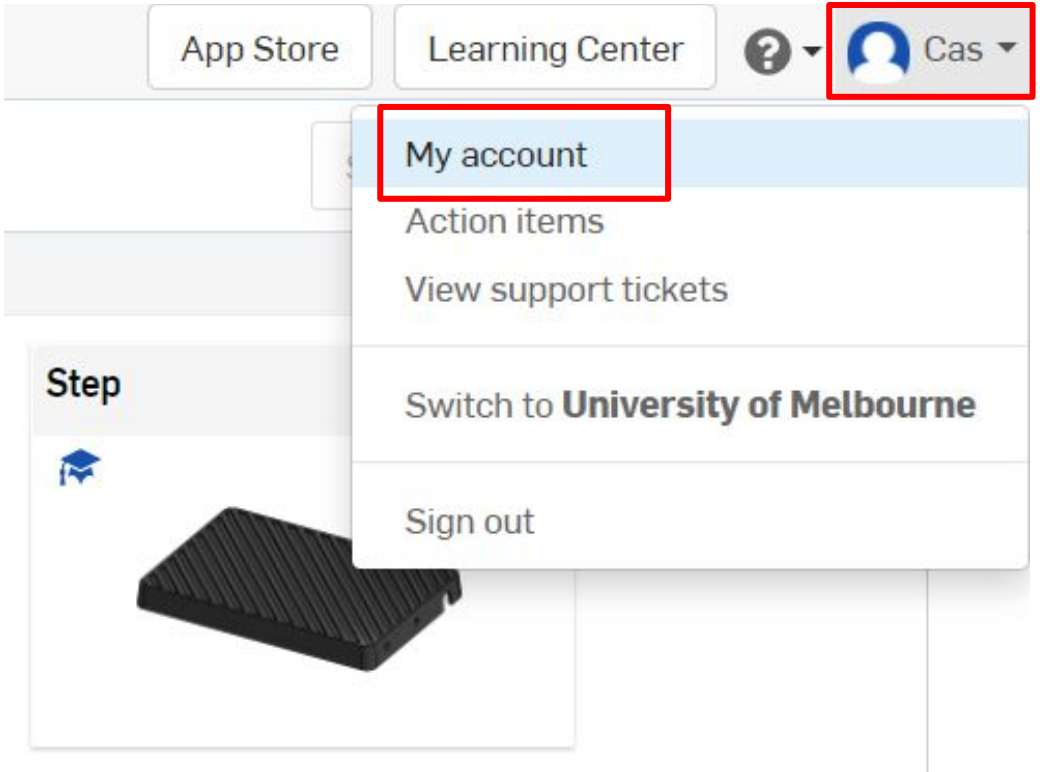
Creating an Account

- Sign up for a free trial: www.onshape.com/en/products/free
- Click **“Get Started”**
- Enter in your details
- Check your email and activate your account
- Set your password



Creating a new Document

- Sign in and go to Documents home page (**cad.onshape.com/documents**)
- First need to make sure your **units** are set to **metric**
- Go to Account Settings -> Preferences and set units to **mm** and **kg**
- Click the Onshape logo to go home
- Create a new Document by clicking **Create -> New Document**
- Make sure you've selected the **Part Studio** at the bottom before sketching your design



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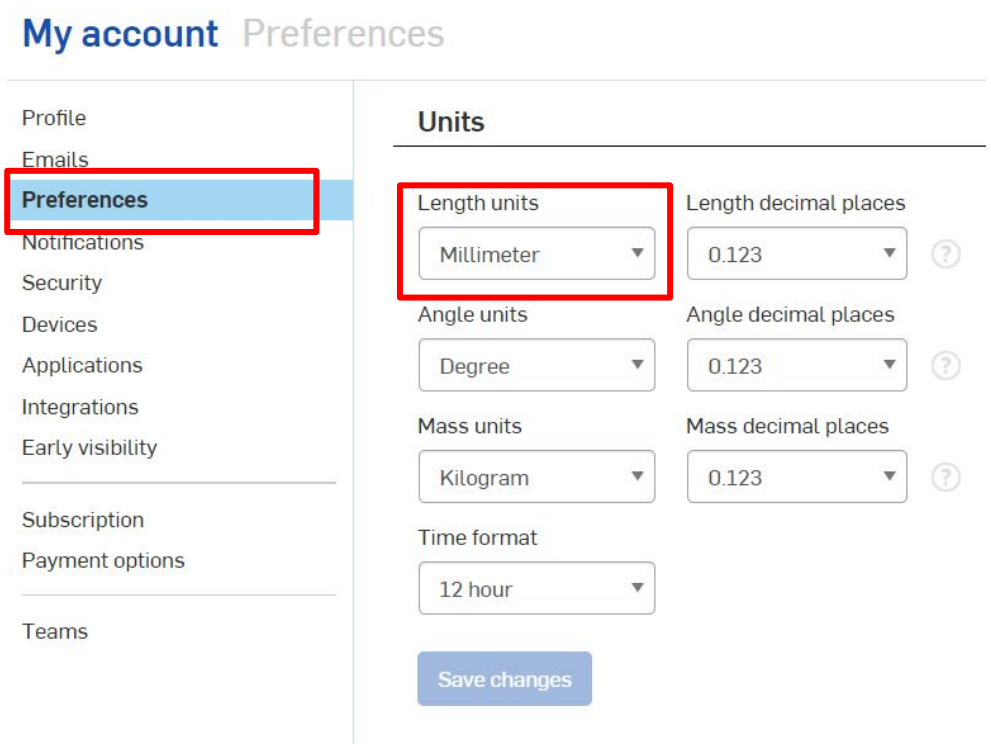

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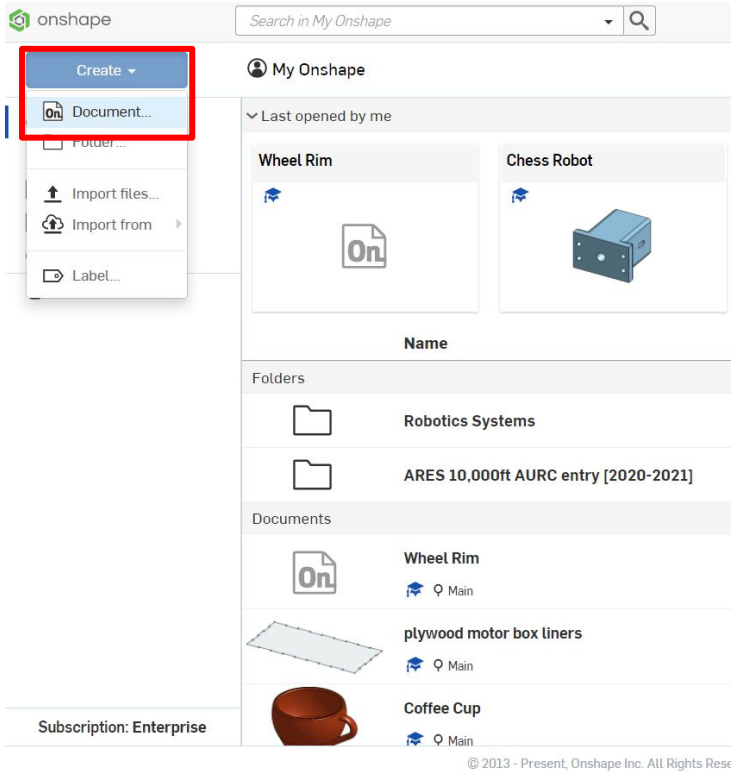

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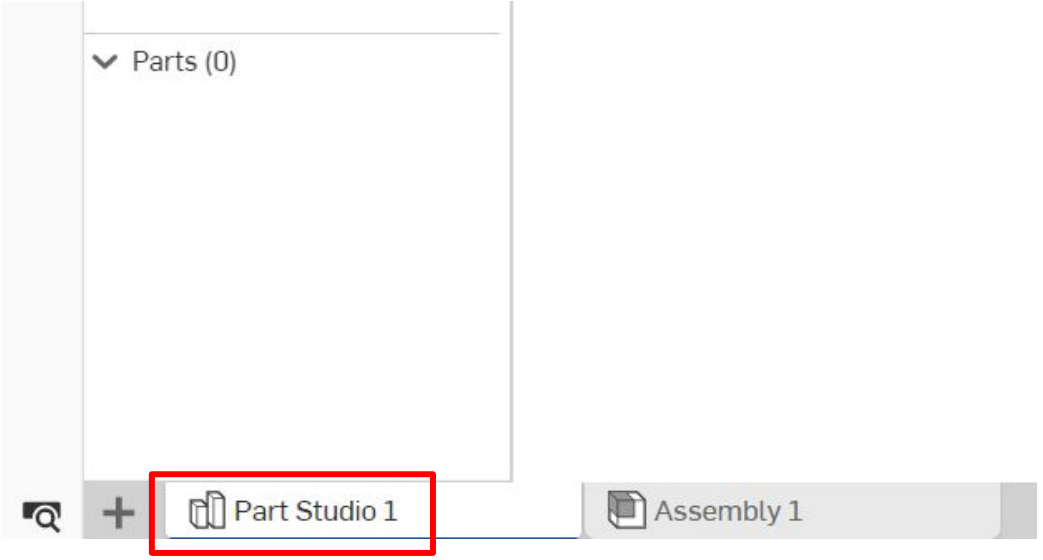


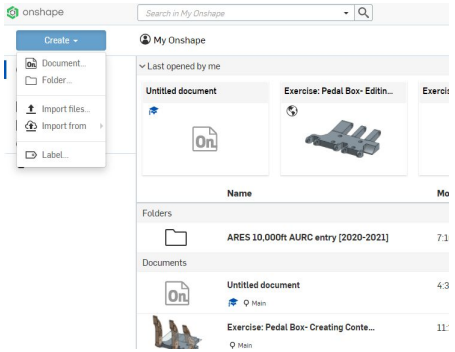
Onshape Geometry

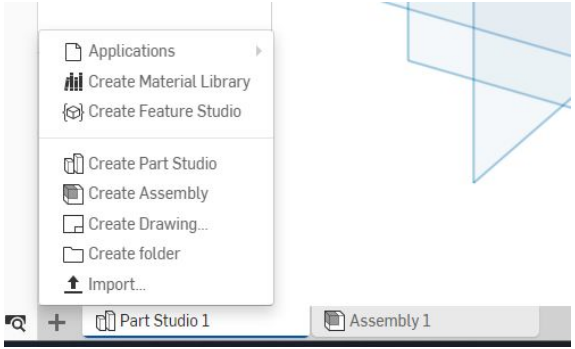
Projects are contained within a folder

Parts and assemblies are created in documents

In a document there are a number of different elements that can be inserted

- **Part Studio:** A space where a single part or multiple parts are designed together. **Multiple parts** which are **geometrically related** should be created within **one part studio**. In a new part studio you will need to consider the correct plane orientation and the location of the **origin** of the part you are creating.
- **Sketch:** a 2D design created in a part studio, which may describe geometry of multiple parts.
- **Feature:** used to add 3D geometry to a 2D sketch
- **Assembly:** A make up of multiple parts which may interact with each other. May include many instances of a part or part studio.





Creating a New Sketch

Select the appropriate plane orientation and select the sketch icon on toolbar **(Shortcut: Shift + S)**

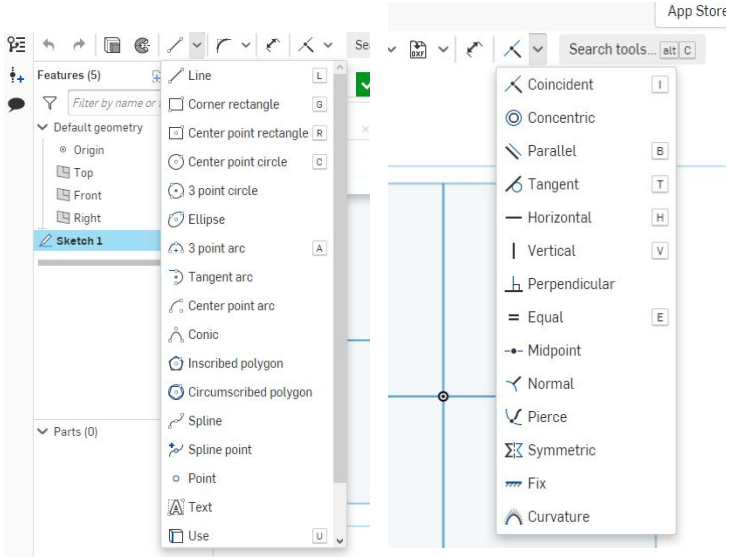
To view normal to the plane right click “View Normal to sketch plane” **(Shortcut: N)**

Top toolbar contains a selection sketching geometries (line, rectangle, circle, spline, point

Construction lines are useful to create geometric references e.g. symmetry axis **(Shortcut: Q)**

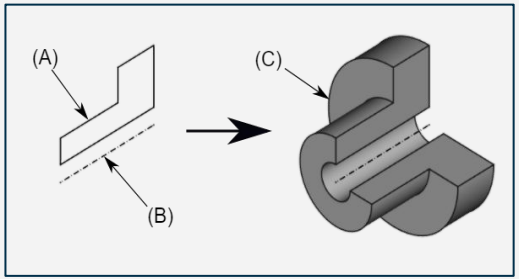
Automatic constraints (inferences) are created as you sketch. To view constraints hold the shift button. To create your own constraint left click the interested segment and use the constraint tools on the toolbar.

Select the dimension tool **(Shortcut: D)** then select the interested segment to edit its dimension.

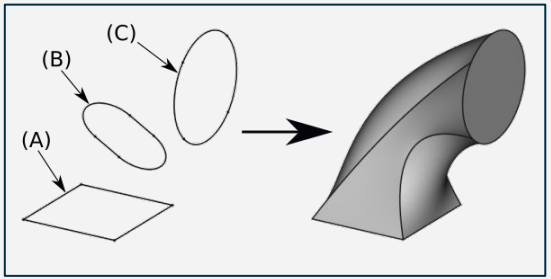


3D Tools

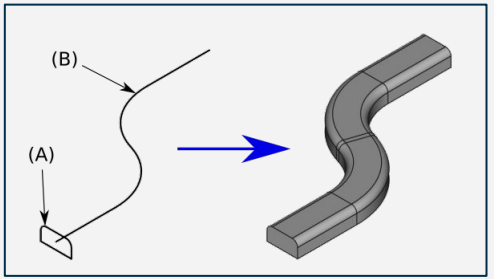
Revolution



Loft



Sweep



Exercise: Creating a Coffee Cup

Onshape Exercise: Coffee Cup

Tips:

- **Constraint tools** are very useful and can help avoid your sketch from blowing up when changing dimensions
- **Naming** your parts, planes and sketches is a good practice to maintain
- Use the **offset tool** to help create hollowed out shapes
- The **revolve tool** will require an **axis** (we can use the **construction tool** to make axes)
- A **sweep** will need a **path sketch** and a **face sketch**
- The cup should weigh 0.578kg

Exercise: Creating a Wheel Rim

Onshape Exercise: Wheel Rim

Tips:

- Create a new sketch with **Shift + S**
- **N** - move camera **normal** to the selected sketch or face
- **F** - zoom in to **fill** the screen with the part
- **P** - toggle whether **planes** are visible
- Don't worry about being precise doing your initial sketch - precision is added later (**sketch -> constrain -> dimension**)
- **Q** - toggle **construction lines** (virtual geometry to assist with sketching)
- Sketch should be entirely black after constraining

Exercise: Creating a Wheel Rim

Onshape Exercise: Wheel Rim

Tips:

- Hover over a tool for 3 seconds to get instructions for it
- "**Show constraints**" button in Sketch menu box
- Learn "**New/Add/Remove/Intersect**" option meanings for tools
- Feature Mirror vs. Part Mirror
- Can select a circular face **normal to the axis of rotation**
- For extrude, first direction has single arrow, **second direction** has double arrow
- **Shift + P** - **hide all construction** lines and sketches

Onshape Learning Resources

Onshape Learning Center - **learn.onshape.com/learn/dashboard**

Complete Onshape tutorials - **Onshape Fundamentals: CAD** and **CAD Basics** pathways

Next week - More Onshape!

See you next week! :)