

CAD Video - How to make an OnShape account (3:32)

- Look up OnShape free
- Click first link
- Press “Getting Started”
- Enter first and last name into login
- Use email address that you have easy access to
- In “What best describes you” make sure you put “Hobbyist/Maker”
 - Hobbyist enables you to have more capabilities than a student account
- Check the box that you agree
- In “companies name” make up a company name
 - Ex: “dog”
- In “industry” enter “other”
- In “number of CAD users” enter 1
- For “current CAD system” enter “Onshape”
- Enter no to the following two questions
 - interest/time frame
- Enter “CEO” for position
- Use either your phone number or your parents phone number
- Click create account
- Use code that was sent to your email to login to your account
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CA =D Video 1 (6:45)

- Go to your onshape account
- Click on “create” and “document”
 - Name your document
- Put two fingers down and you can move the planes around the origin
 - This will let you be able to see different angles
- Click on the sketch button the top left corner
- Select the front sketch plane
 - Press n on your keyboard
- Click on the line icon near the top left corner
 - Click once on the front plane to start your line
 - Click on another spot on the plane to end the line
- Press escape to avoid creating a new line
- Click on the line you just created and press the delete button
- Click on the square and select “center point rectangle”
 - Drag out the rectangle and then click
- Make a center-point circle
 - Make it inside of the rectangle

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CAD Video 2 (4:19)

- Continue working on the same project as last time
- Go to the panel on the far left and click on “sketch 1”
 - Right click and go to “edit”
 - Press the n key
- Click on the double sided arrow, the dimensions tool in the middle
 - Click on the length of the rectangle and move your mouse away
 - Set it to four
- Click on the width and set it to four
- Make the diameter of the circle 2
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CAD Video 3 (6:31)

- Start on the same project as last time
- Click the checkmark on “sketch 1”
 - This saves your sketch
- Move your sketch so it’s facing sideways
- Click on the extrude tool in the top left corner
- Click on your sketch
- Click on the box called “symmetrical”
- Edit the extrude and make the depth four inches
- Click on the right sketch plane on the cube
 - Press n
- Make a center rectangle
 - When the white box appears type 2
 - Repeat for all sides
- Make another box with dimensions of 1 and 1
- Click extrude
 - Make the depth 2 inches
- You have finished your first CAD project!

CAD Video 4 (2:38)

- Start by CADing the middle part
- Measure the depth
 - 0.183
- Click on sketch
- Change your view in the top right by clicking on the top of the cube
- Fill in the rectangle

- Click extrude
- Repeat with the two other squares
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[CAD Video 5 \(2:38\)](#)

[Challenge Video 1 - Fillets & Chamfers \(7: 14\)](#)

- Make a four by four cube
- Select fillet
- Select an edge of cube
- Press on the check mark
 - This will create a rounded corner
- Click on the dot at the top and bottom of the fillet
 - Select the measurement tool
 - The y and z distance both show the radius
- Click on Fillet previously created
 - Change radius by editing the radius dimensions
- Select Chamfer(next to Fillet tool)
- Select an edge of cube
 - This will create a slanted corner
- Change distance of Chamfer by editing the distance
- Press on check mark
- Click on dot at top and bottom of the chamfer
 - Select the measurement tool
 - The y and z distance both show the distance
- Click on Chamfer previously created
- Select “Two distances” from drop down menu by clicking on “Equal distance”
- Change only “Distance 1”
 - This changes the angle of the Chamfer
- Select “Distance and angle” from drop down menu by clicking on “Two distances”
- Change only “Angle”
 - This changes angle of Chamfer
- Change only “Distance”
 - This changes starting depth of Chamfer
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[Challenge Video 2 - Lofts](#)