Getting Autodesk Products:

Getting Started:

1. Go to www.autodesk.com
2. Make an Autodesk account with your school email
3. Go to https://www.autodesk.com/education/edu-software/overview
4. Then apply as a student.
    ○ You might have to send in a picture of your student ID at some point
5. You'll have to re-do part of this process every year
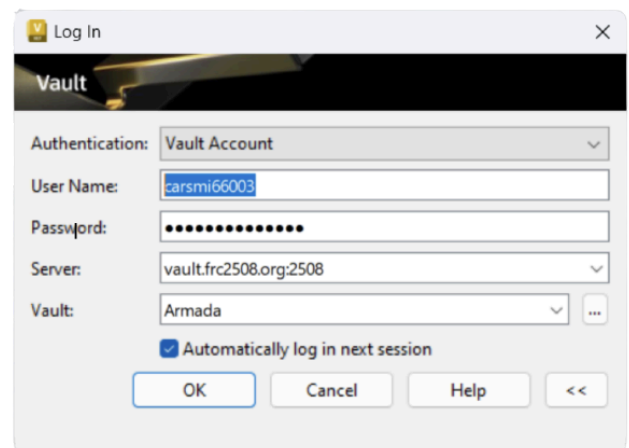
Getting Inventor & Vault:

6. Make sure your account has been verified
7. Go to https://manage.autodesk.com/products
8. Scroll down a bit until you see either Inventor or Vault
9. Then click download on the Vault Client 2024
    ○ If that doesn't work, contact Carter, Ben, Fredy, Jaxon, or Ian
10. Then click download on the Inventor 2024 (should be the same as vault)
    ○ If that doesn't work, contact Carter, Ben, Fredy, Jaxon, or Ian

Using Vault:

Logging on to the Vault: (IP, vault name)

1. You'll be using the training account until you've been trained in how to use Inventor & Vault
2. (Open Vault and you should be prompted to login)
3. Select **Vault Account** accounts under Authentication
4. Server info:
   - **Vault Server IP**: vault.frc2508.org:2508
   - **Vault Name**: Armada
5. Name and password for Training account
   - **User Name**: Trainee1
   - **Password**: n/a
6. Contact **Carter Smith**(/the person running the vault server [Carter, Ben, Fredy, Jaxon, or Ian]) to get a **permanent user account** with our Vault server. After you get your own account use that login and not the training account
7. You can change your password in the bottom right by selecting you, then 'change vault password'

Using Inventor Projects (Don't forget!!):

1. Open Vault and once you're logged in, right click on the Armada.ipj file and click 'Get' in Inventor.
2. Click the 3 dots to the side of the inventor home page to access the settings. Browse to C:/Armada, and select the Armada.ipj file
   a. All of our files are made using this project, and **will not work otherwise**

Our Folder Structure:

Inside of Vault Explorer we have the Admin and Designs folders. Admin files are things like the project file, configurations, and settings you should be using.

All of our projects are inside of the Designs folder, with each subsequent folder being for a different sub-project

Inside of the year folders are the robots for each competition. They have their own folder structure for each subassembly on the robot (i.e. Drivebase, Intake, Tower, etc)

- Part naming conventions
  - Our part names and numbers are in this format:
  Project/Year_Subassembly_PartType#
    - For example; 23_Claw_Plate1
    ^ The first example is for the 2023 robot > inside the claw subassembly > and is the first plate

    - Another example; ButtonBoard_Frame_Panel1
    ^ This is for the Button Board project > is part of the frame > and is the first panel
  - You can also look at the preview to get a better idea without opening the part

- How to use vault (check in/out, comments on parts, revisions)

- What is the common folder, and what's in it?
  - The common folder is for any parts that we know we'll use for many projects from year to year
  - Examples include pulleys, gears, bearings, extrusion, motors, electronics and more

- Template/settings folder (WIP)
  - Ian has made a setting folder that is in the Admin folder in Vault
  - This file includes basic settings that every student should be using
  - To use these types of files in Inventor, go to Tools, then Application Options, and import. Then select the .xml file

- When to use Vault vs Drive
  - All major projects should be in Vault
  - Any files that we would need quicker access to, or is a very minor project, it should be uploaded to the Google Drive   (ex. .cps file for the CNC router, or battery plugs)

- How to get drawings per part (this part is tied to what drawing)

Basic questions

- Who to contact for help
  - If you're having issues with logging in or interacting with Vault talk to Carter or Ben, and if the problem is with Inventor talk to Carter or Freddy

- Videos to watch
  - Contact Carter for access to Solidprofessor, which is a website with video tutorials about almost anything in Inventor
  - Feel free to do your own research for other ways to learn, Youtube is a great resource

- Where to get files
  - The two places often used for common parts are McMaster-Carr and GrabCAD

- What materials we use
  - We primarily use aluminum box section for the frame with 3D printed filament for smaller technical parts, aluminum gusset plates, and acrylic for covers on the robot.

- Why we use our standards
  - Our standards help make our systems organized and efficient
  - When we need to edit parts or assemblies, it saves a lot of time by designing something a different way so it's easier to modify if needed in the future
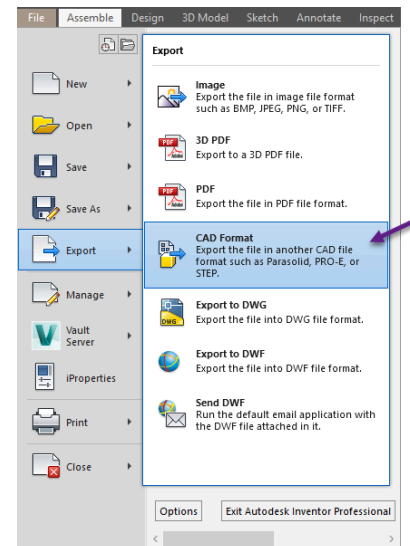
- How to export CAD models

  To export a CAD file you should make sure it is saved to a database, then you can go to: File > Export > CAD Format > and the file type you want to use
  For 3d printing: STL
  For CNC machining: STEP
  For public sharing: STEP

  498 × 602

CAD Standards

- CAD practices

  -Start your initial sketches on the origin (center using vertical + horizontal tools)
  -Everything should be constrained, if it's purple it isn't done

- Drawing standards (title block, ordinate set, date and revision, 11x17") (WIP)

- Equations (Fx)

- Bolts and Hole Sizes

  For our designs we use industry common bolt and screw sizes, the most common of which are below

Bolt		Close	Medium	Free	Tap Size
6-32		.144 / 27	.149 / 25	.169 / 18	.106 / 36
8-32		.169 / 19	.177 / 16	.193 / 10	.136 / 29
10-32		.196 / 9	.201 / 7	.221 / 2	.159 / 21
1/4-20		.257 / F	.266 / H	.281 / K	.201 / 7
3/8		.386 / T	.397 / X	.406 / 13/32	.332 / Q
1/2		.516 / 33/64	.531 / 17/32	.547 / 35/64	.453 / 29/64
M3		3.25 (.128) / 30	3.45 (.136) / 29	3.75 (.148) / 26	.098 / 40
M5		5.35 (.212) / 3	5.6 (.220) / 2	5.95 (.234) / A	.166 / 19
M6		6.45 (.254) / F	6.7 (.264) / 17/64	7.15 (.281) / K	.196 / 9
M8		8.45 (.333) / Q	9.1 (.358) / T	10.15 (.400) / X	.266 / 17/64
M10		10.6 (.417) / Z	11.15 (.439) / 7/16	12.2 (.480) / 31/64	.332 / Q
M12		13.1 (.516) / 33/64	14.15 (.557) / 9/16	15.2 (.598) / 19/32	.404 / Y

- Tolerances

  How to use
  -In a sketch, double-click a dimension or click the Dimension command in the Constraint panel and then click the dimension. In the Edit Dimension dialog box, click the arrow and choose Tolerance from the pop-up menu.
  Why use
  Tolerances are used to show room for error that may occur while the part is being made

- Filet everything as a Last feature
    - Before a part is considered done it should have few sharp edges. The most common way is to use filets
    - This eliminates any sharp edges that can be a potential hazard, and can help the part not crack at edges

How to

- Viewing weight and cg (center of gravity)

  !Every part must have a material set to have an accurate weight and cg!
  Right click on the assembly/part in the design tree, and go to iProperties;
  Go to the Physical tab, and click update. This will tell you the mass, as well as the coordinates for the cg
  You can also go the view on the ribbon, and click cg to see where it is, and measure it

- Making/using iParts

- Calculating belts
    -

- Pack and go

  Pack and go is used for when you need access to only a subassembly and the parts inside. You can create a folder with only the subassembly for easy transferring, and sending to others
  Go to file > save as > pack and go
  Select the destination folder, and check the box 'package as .zip'. This will make sure that it won't explode into your destination folder

Then click 'search now' and 'start'

- Configs?

- CAM
    - Feeds and speeds for materials, for different finishes
    - What end mills we have/should use

- Sketch blocks