## Using AutoCad Inventor to Model your Trebuchet Arm

1. Using what you learned during the tutorial and what you can learn by exploring AutoCad Inventor create a model of a **Trebuchet arm** from your sketch.
   1. Include mounting holes for the pivot bearings. The bearing wheels are 0.75” diameter and 0.25” thick. The rod that threads through them is 0.18”. A 0.200” through hole is recommended
   2. Include an attachment method for the counter-weight. Screw eyes need a 0.0808” diameter hole 7/16” deep.
   3. Include a way to attach your release pin. The suggested nails are 0.060” diameter but they should fit snuggly in the hole. You are free to design your release pin any way you choose.
   4. The arm must fit inside a 1.5” x .75” x 16” profile.
2. Verify that the design does not exceed the 6.0” volume requirement
   1. Select **iProperties** in the **I-PRO** menu
   2. Select the **Physical** tab of the **iProperties** dialog box
   3. Choose **ABS Plastic** for **Material**.
   4. Click Update if necessary.
   5. Enter the values for Volume in your notebook (must be less than **6.0 in^3)**
3. Use AutoCad Inventor **iProperties** to calculate **mass**, **inertia** and **distance**
   1. The value for **mass** depends on the material. Make sure you have chosen **ABS Plastic**.
   2. The value for **inertia** about the center of gravity will be in **I3** if your design rotates about an axis parallel to the z-axis (**Rz**). Use I1 for rotation about Rx, I2 for Ry.
   3. Inventor calculates the **Center of Gravity** in X, Y and Z. The calculated value of inertia about the center of gravity, around the correct axis of rotation (I3 for z-axis rotation, etc.) is the value you should record and use in your simulation.  
      Enter a negative value for d if the center of gravity is on the short end of your trebuchet arm (not likely).
   4. Enter the values of your modeled arm into your notebook **and** into your simulation spreadsheet (create a new column for as-modeled). It would be good to re-simulate at this point to make sure you like the results.
4. Cut your part into two pieces.
   1. Consider how you will divide your arm into two pieces so it can be manufactured. Neither piece can be longer than 9”
   2. Create a working plane offset from an origin plane where you want to divide the part.
   3. Create a 2-d drawing in this plane that shows how your parts will be connected.
   4. Save your drawing.
   5. Save the drawing again with a new name (i.e. lefthalf.ipt)
   6. Save the drawing one more time with another name (i.e. righthalf.ipt)
   7. Delete the parts that are not part of each half from the respective drawings.
   8. Extrude connection features into the remaining parts.
   9. Save each part and verify that the total Volume remains less than 6 in^3
5. Create STL versions of your parts
   1. In the I-Pro menu select **Export->Cad Format**
   2. Change the Save as Type to **STL Files (\*.stl).**
   3. Under **Options**… verify that the Units are **inches**.
   4. Select OK to save the files
   5. Submit all four files (left/rightHalf.ipt, left/rightHalf.stl) in Canvas