Motion Study

Solidworks

Assemblies

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The CAD Academy
Launch Solidworks by locating the ICON and selecting it

We are going to make a small assembly to use for animation purposes. If at any time you want to CANCEL something press the Esc key twice to cancel..

Select the sheet of paper to begin a new PART

Select PART from the new screen and then select OK.

Select OPTIONS
Select the DOCUMENT PROPERTIES tab and then UNITS

Change the UNITS to IPS for this project and then select OK.

We are going to make the main part of our assembly.
Engineers start with a sketch or an idea so SELECT the SKETCH tab and then the TRIANGLE to begin our steel plate, which will be the base of our assembly.

Select the FRONT plane to begin the drawing. Selecting the correct plane is very important in making assemblies.

ROTATE the plane INTO the FRONT view.
Make two mouse picks as shown. It does not matter what size you make the rectangle as we will DEFINE the size through dimensions. Solidworks uses PARAMETRICS or dimension driven designs.

Select SMART DIMENSION and then SELECT the LINE as shown.

Next PICK right ABOVE the dimension line for the dimensions line location and where the existing value is replace with 6.00.
Smart dimension the left side by selecting the line and moving to the left for the dimension line location and changing the value to 3.500.

When completed check the GREEN OK checkmark to close the dimension dialog box.
Go to the FEATURES tab and select EXTRUDED BOSS/BASE

Change the value of D1 to be 1.25 which is the thickness of our part. Be sure and select the GREEN CHECK or OK to complete the extrusion.

USING VIEW ORIENTATION change the view back to the FRONT and plan (2D) view.
We are going to continue working on our design. SELECT the SKETCH tab and then SELECT CIRCLE.

PICK TWO POINTS AS SHOWN

TYPE in 1.00 for the RADIUS in the circle parameters dialog box

check GREEN

CHECK OK
SMART DIMENSION as shown at right.

First pick the top line and then the circle. Select the dimension line location away from the part as shown and change the value to 1.750.

Make a second smart dimension as shown and change the value to 1.500. Notice how the dimensions move our circle to the exact location required.
Select CIRCLE again and move the cursor over the center of the previous circle and you should see guide line for CENTER. Then move to the left.

draw a circle as shown and change the RADIUS to .5

Using SMART DIMENSION - dimension CENTER to CENTER and change value to 3.00.
CLOSE dimension by selecting the GREEN CHECK.

Select the FEATURES TAB and then EXTRUDED CUT

MAKE SURE TO CHANGE THE VALUE TO 1.25 FOR D1. GREEN CHECK OK.

Select DISPLAY and then ISOMETRIC

completed base
Select the RED SOLIDWORKS ICON. This allows you to select the FILE menu. Go to SAVE AS and save part as BASE.

Select the SHEET of PAPER to CREATE a NEW part.

Select PART and OK

SELECT the SKETCH tab and select FRONT PLANE to begin the part
Draw a circle and then SMART DIMENSION the circle to 1.75 (this is a diameter)

SELECT GREEN CHECK to close DIMENSION
SELECT FEATURE TAB
SELECT EXTRUDE
TYPE IN 1.25 FOR D1
SELECT GREEN CHECK OK.

SELECT EDIT APPEARANCE ball and change the cylinder to a different color.
Then select BROWSE a material or color of your choice.
SELECT the "F" key to HOME UP THE SCREEN

GOTO SOLIDWORKS ICON and select FILE MENU
and then SAVEAS.

SAVE AS - CYL1

REFER to STEPS above for questions.

GOTO SHEET of PAPER

START a NEW PART in the FRONT PLANE.

GO TO SKETCH

select CIRCLE. Draw a circle and SMART
DIMENSION to .75

GREEN CHECK DIMENSION
DIALOG BOX OK.

GOTO FEATURE

SELECT EXTRUDE

TYPE IN 1.25 FOR D1

GREEN CHECK OK.

CHANGE COLOR
GOTO FILE
SAVEAS - CYL2

GOTO sheet of paper. SELECT ASSEMBLY and then OK.

SELECT the BASE and move RIGHT into the drawing area.
Place the drawing by using the left mouse button. Go to display and change view to FRONT VIEW.

GOTO insert components

Insert CYL 1 and place approximately in center. Insert CYL2 and place approximately in center as shown.
If we wanted to we could move the cylinders all over the place. We are going to put CONSTRAINT on them so they move only the way we want them to.

SELECT MATE and then CONCENTRIC
Select the OUTSIDE circle and then the INSIDE circle and green check OK.

Perform a CONCENTRIC MATE on the second cylinder using the steps above as a guide.

Go to VIEW ORIENTATION icon and change to an ISOMETRIC view.
If your CONCENTRIC MATE was done correctly you should be able to SELECT a CYLINDER by just PICKING on it and dragging it. Try to move it any way but BACK and FORTH or CONCENTRIC to the hole in the base and you should not be able to.

We want others to UNDERSTAND how this ASSEMBLY WORKS. We can do an ANIMATION or MOTION STUDY and save it to a file. Someone who does not own Solidworks can then look at the part and understand how it works.

Select MOTION STUDY at the LOWER LEFT corner of the screen.
We want our customer to see an overall view of this assembly.

This can be AUTOMATED by SELECTING the ICON shown below.

press NEXT

We will do 1 clockwise rotation

press NEXT.
select FINISH

This macro selected the base and then is rotating over the specified time.

Press PLAY

SELECT CYL1

THE DIAMOND REPRESENTS CYL1. GOTO slide the diamond to the END of the ROTATIONS and then MOVE the CYLINDER away from the
base. MOVE the DIAMOND for the LENGTH OF TIME you want to have this move take. PRESS PLAY to make sure it looks OK. I showed mine moving in and then out.

Now select CYL2 MOVE the DIAMOND out to the END of the ROTATION and then MOVE the CYLINDER and then MOVE the DIAMOND for the LENGTH of time you want the move to take

It takes a little practice to make this work exactly how you would like it to.

You can also move the CYLINDERS as it ROTATES if you would like.

SAVE your ANIMATION to a FILE you can share with others. Make sure to put your ANIMATION in a place you can find again.

EXPERIMENT and have FUN

Lesson is complete