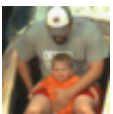


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Tutorial: SolidWorks Mold Tools

[SOLIDWORKS mold](#)

By [Stephen Nyberg](#) on 22 Mar 21:37 19 answers 22223 views 12 comments

Here is a simple Tutorial on using the Mold Tools in SolidWorks. For this tutorial we will make a simple basket and the corresponding mold. I used SolidWorks 2012 and finished files made in the tutorial as well as the screen shots can be found on my profile.

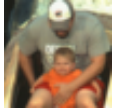
This will not allow you to create a production mold but rather will introduce how to use the tools in SolidWorks.

If you are interested in learning more about injection molding (not SolidWorks specific) you should check out www.protolabs.com as they will send you a bunch of free stuff that will really help if you want to learn about

injection molding. There is a picture of some the free stuff they sent me at the end of this tutorial.

Question unclear or needs more details before answering? [Comment \(12\)](#)

19 answers



[Stephen Nyberg](#) over 1 year ago

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This Tutorial was written "Barney Style" so even the most novice of SolidWorks users should be able to follow easily.

The files I created making this tutorial can be found here:

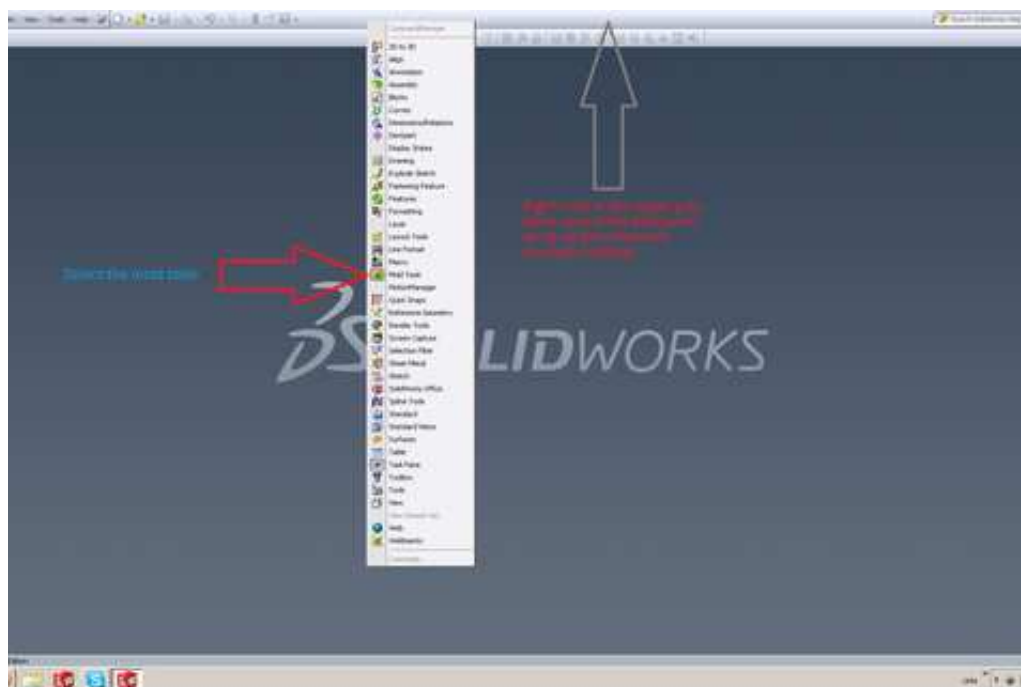
[Tutorial: SolidWorks Mold Tools](#)

1. Step 1

Open up SolidWorks.

To turn on the tool bar with the mold tools we will be using in this tutorial, right click in blank portion of the tool bar and this will open the "CommandManagers" - about half way down you find an icon labeled "MoldTools".

Click this and it will add the mold tools toolbar to your window and this can be moved wherever you find convenient.



2. Step 2

Start a new part

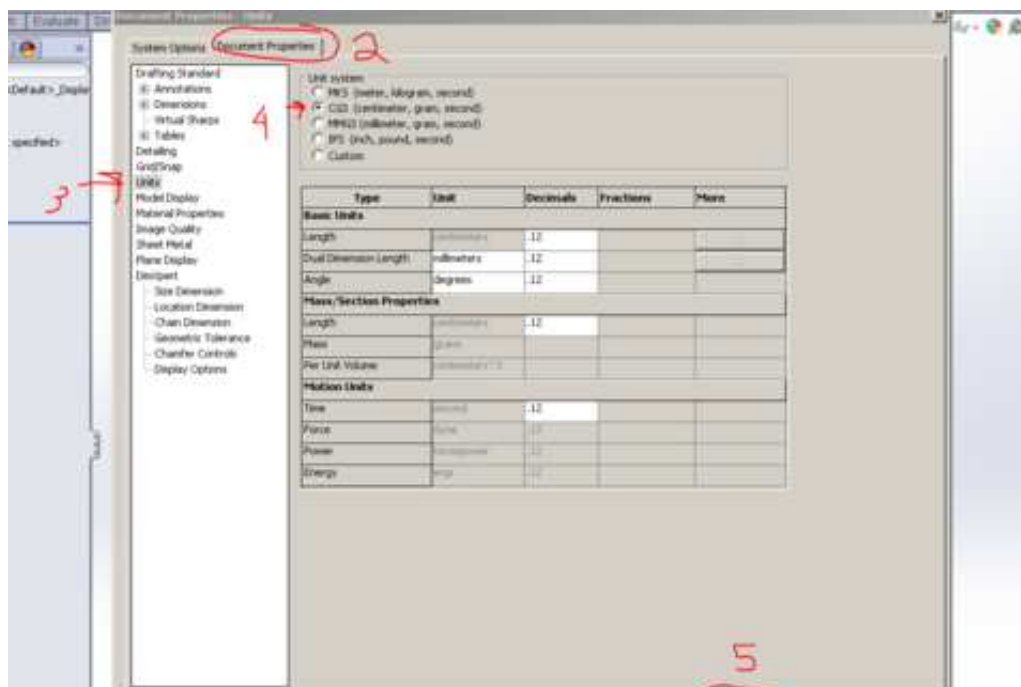


3. Step 3

This part is made in centimeters.

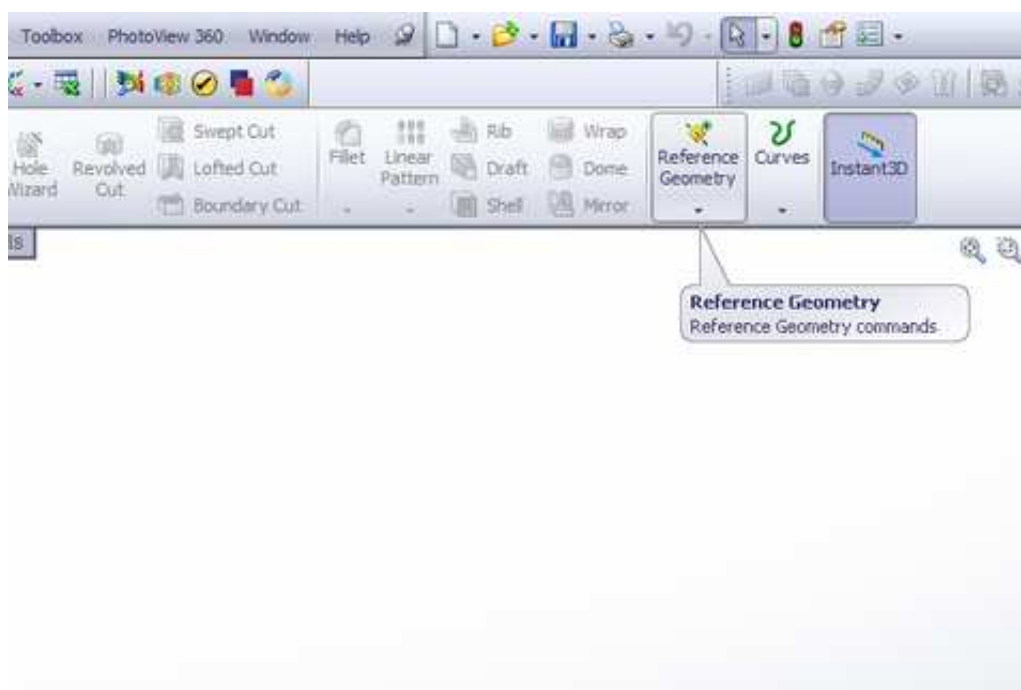
click on the options button then then select the Document Properties tab. from here select units and click the CGS bullet.

When finished select OK button on the bottom.



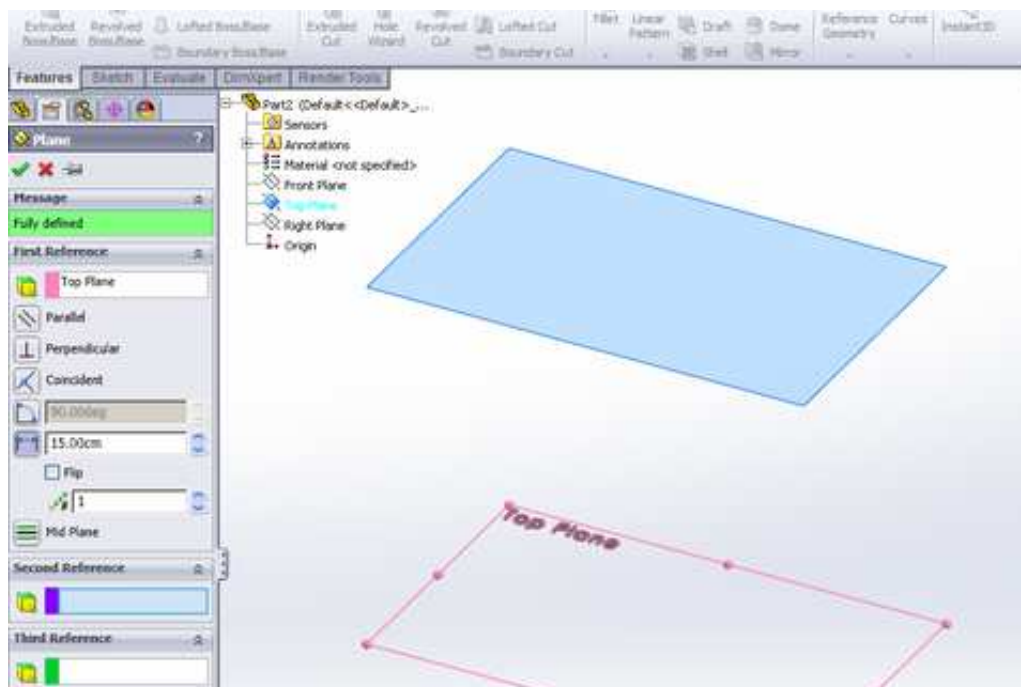
4. Step 4

Create a plane offset from the top plane by 15cm



5. Step 5

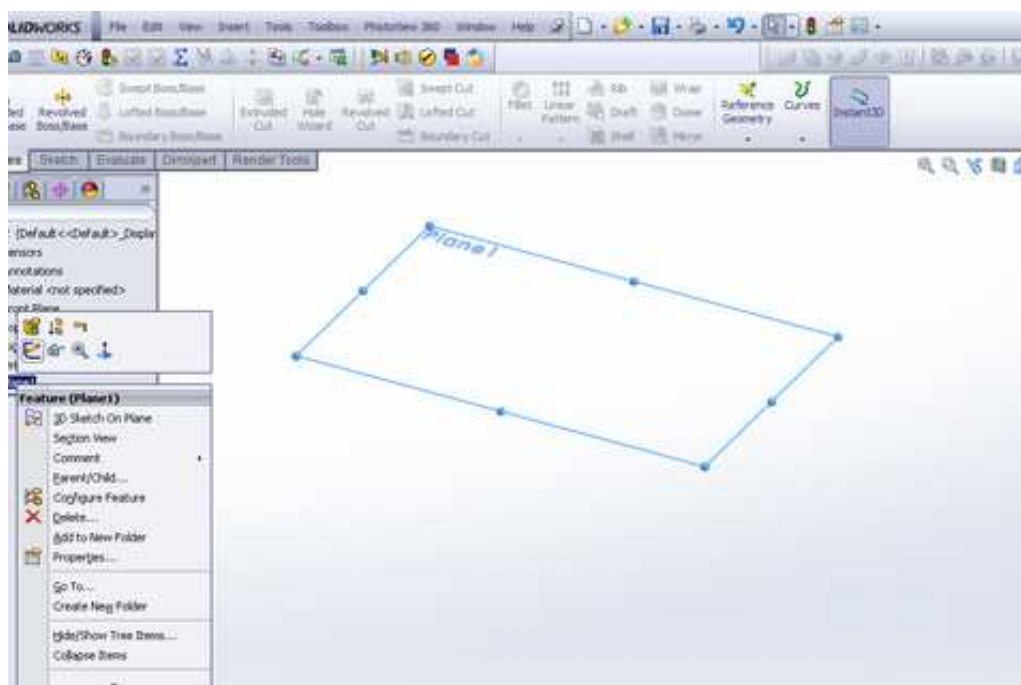
here is our new plane



6. Step 6

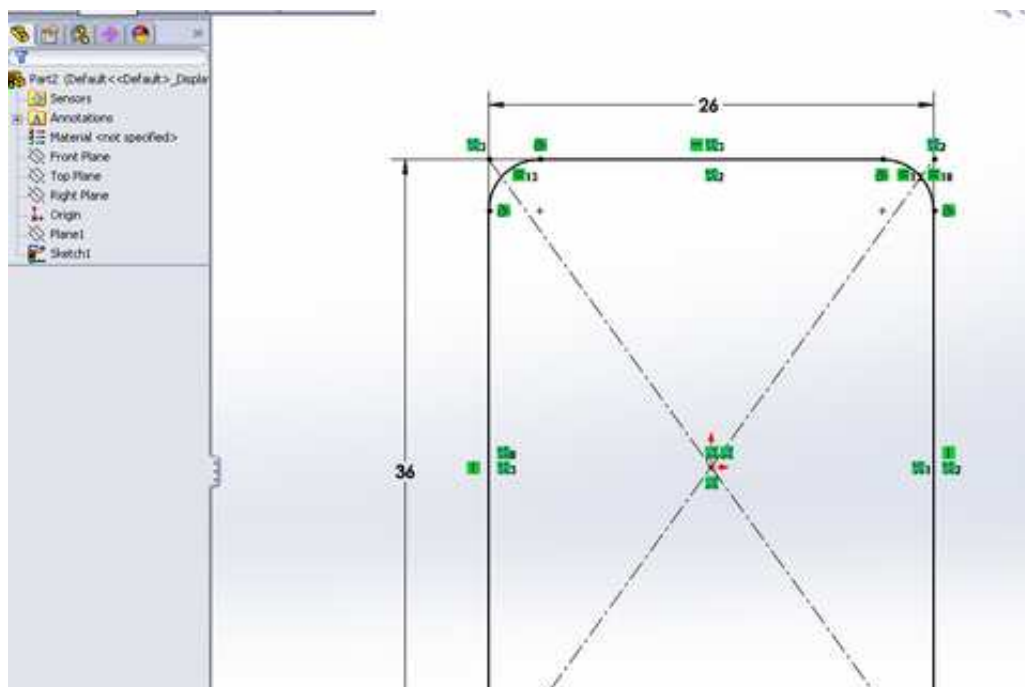
open a new sketch on this new plane.

(you can right click the plane in the feature tree and select the sketch icon to do this)



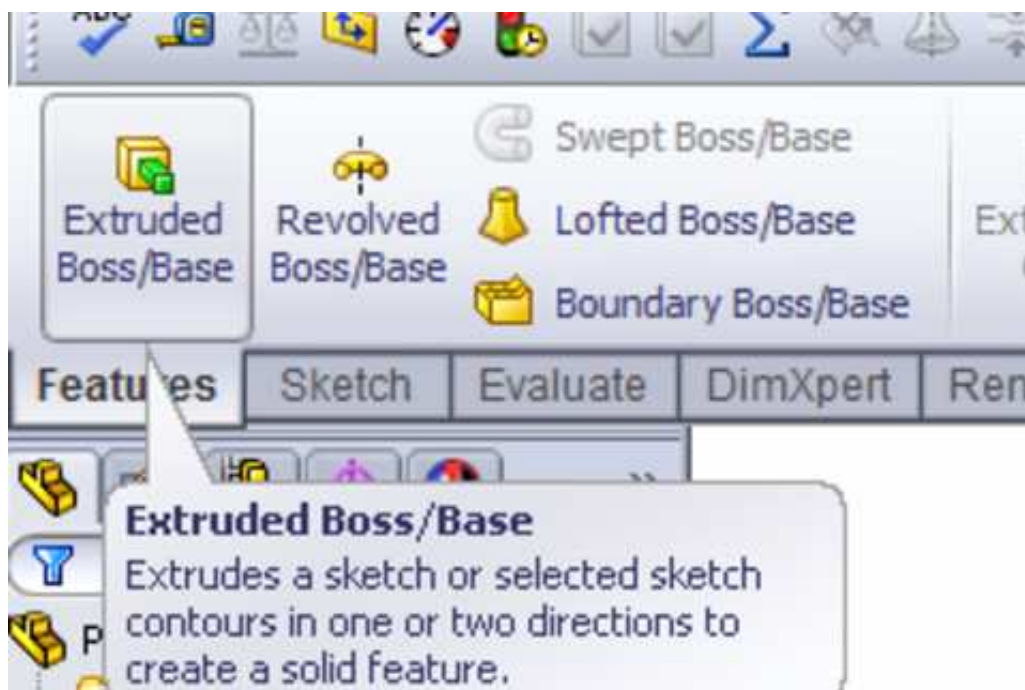
7. Step 7

Create the following sketch then exit the sketch.



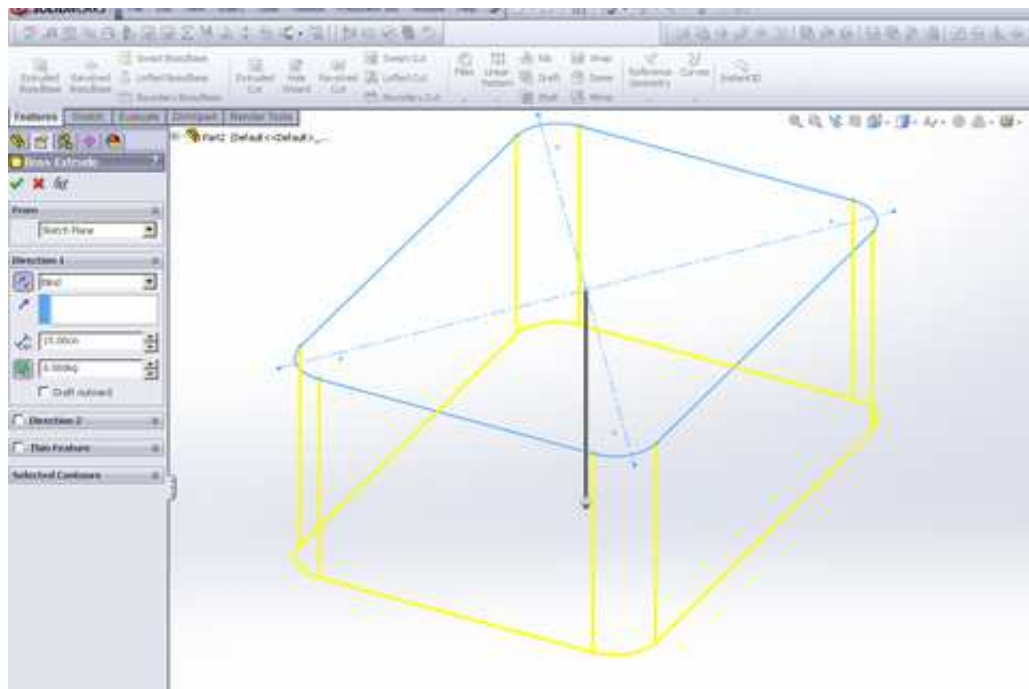
8. Step 8

With the sketch we just created selected, press the extruded boss/base button in the features menu.



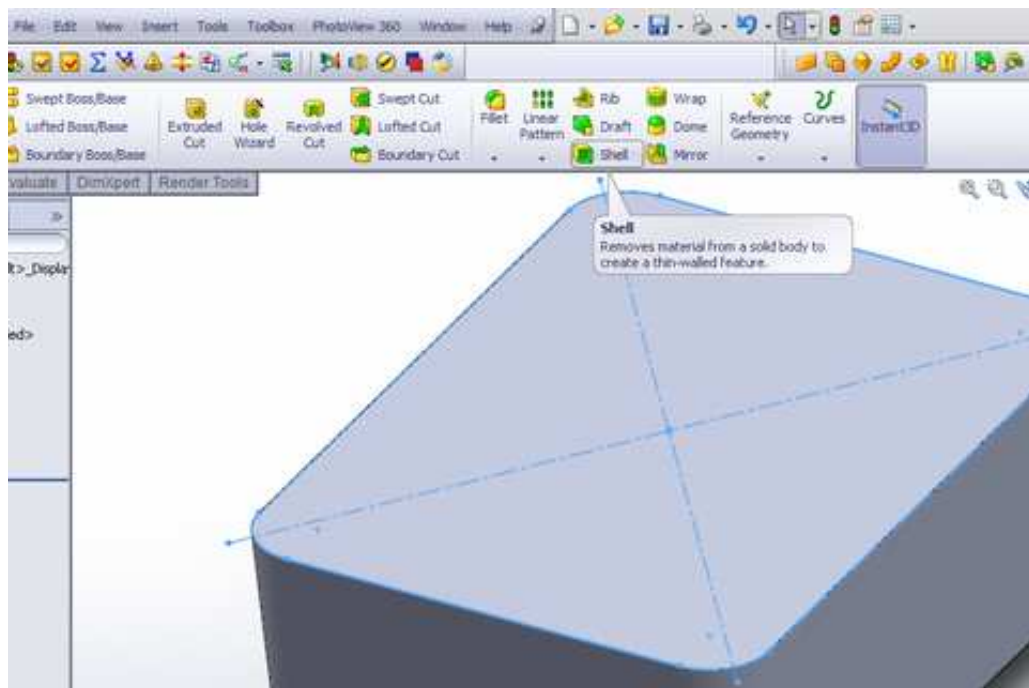
9. Step 9

Extrude down 15 cm with 6 degrees of draft angle



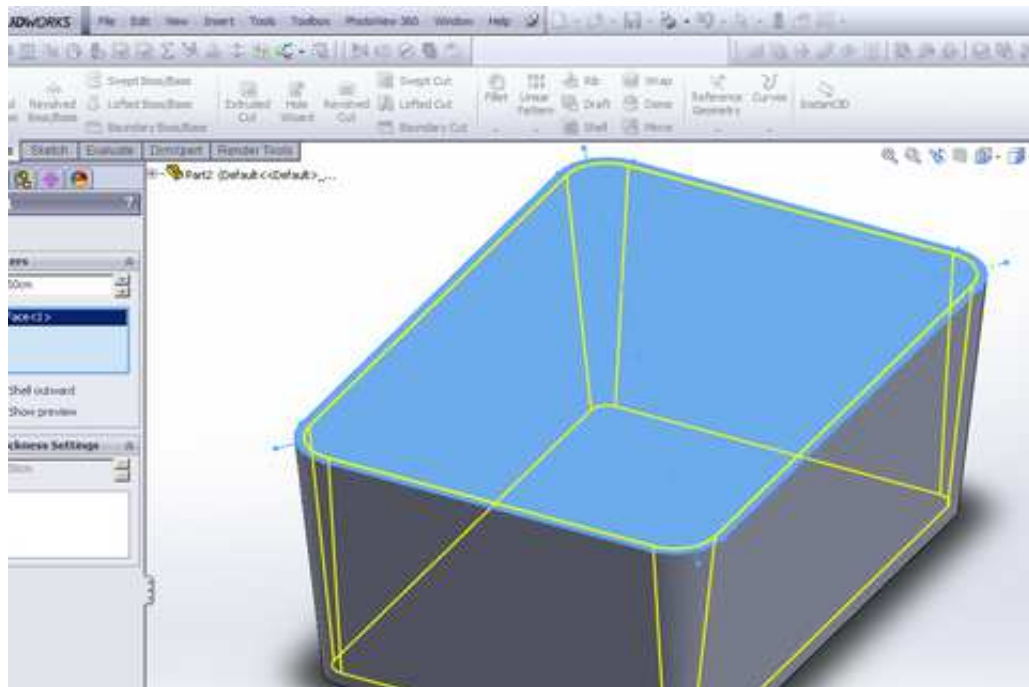
10. Step 10

Now We will use the shell comand



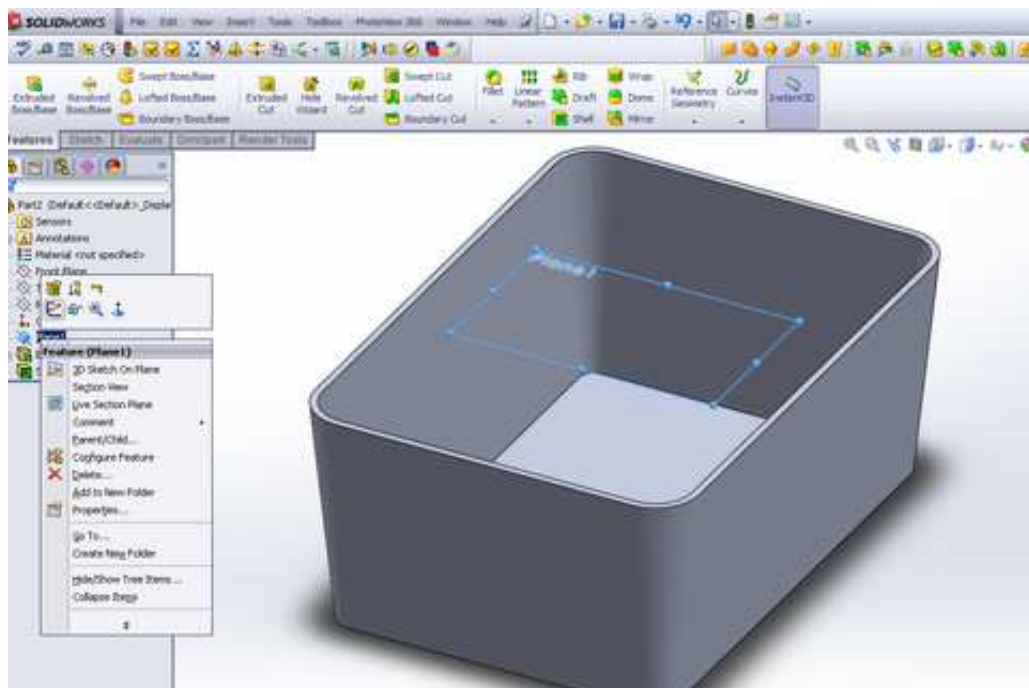
11. Step 11

Select the top face and make the wall thickness 0.5 cm.



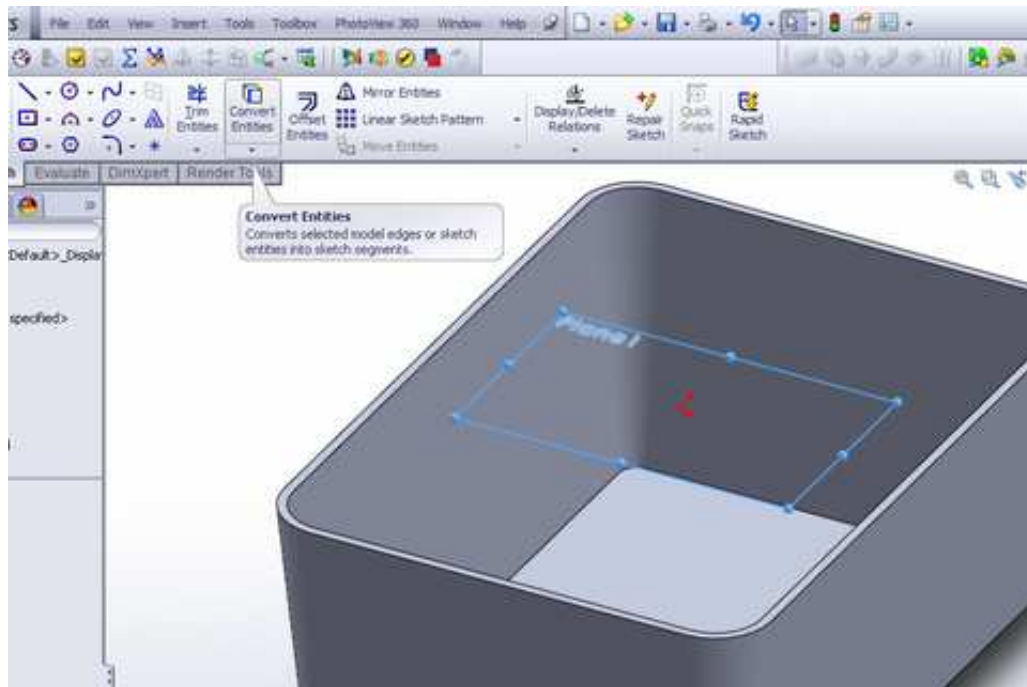
12. Step 12

Now that we have the basic basket, it needs a rim. open a sketch on our plane 1 again.



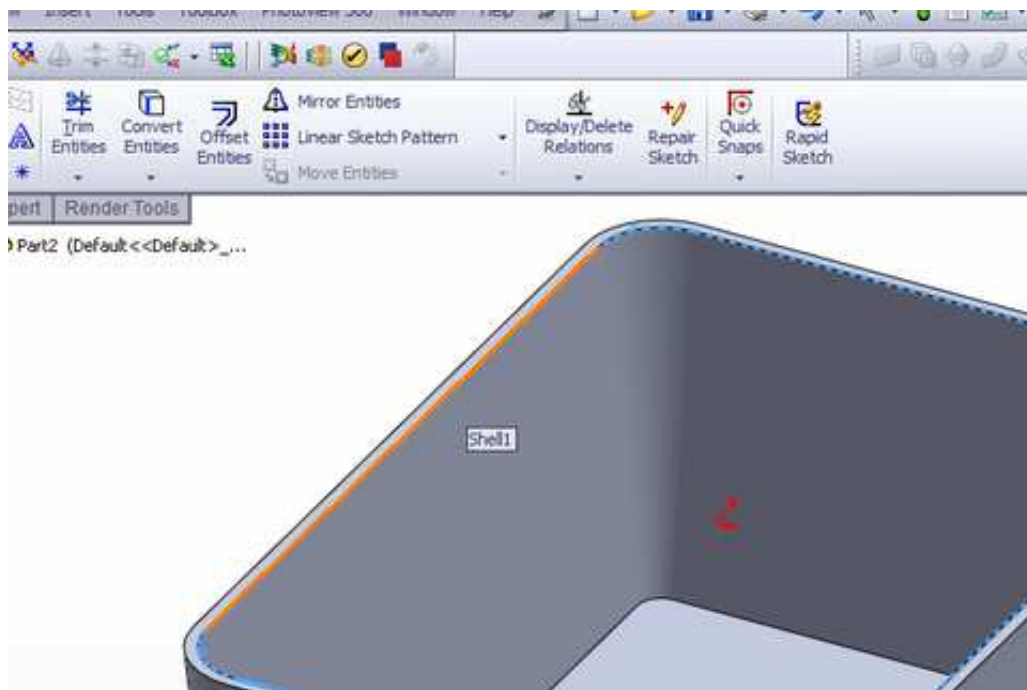
13. Step 13

use the convert entities button



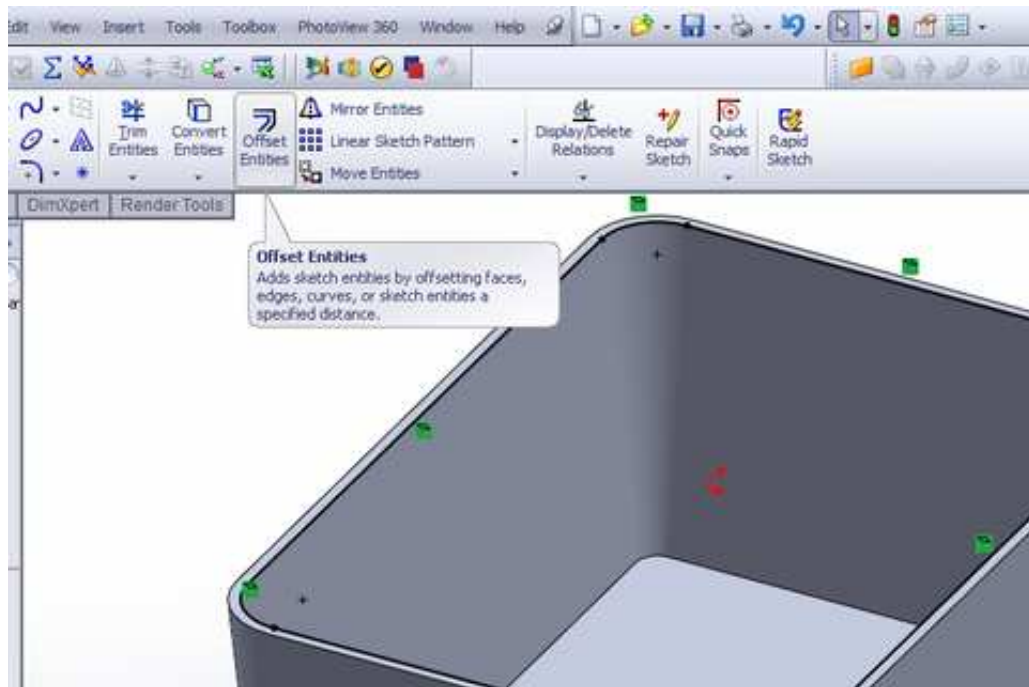
14. Step 14

Select the segments that make up the inner loop of the top edge of the basket. and press green check mark to accept.



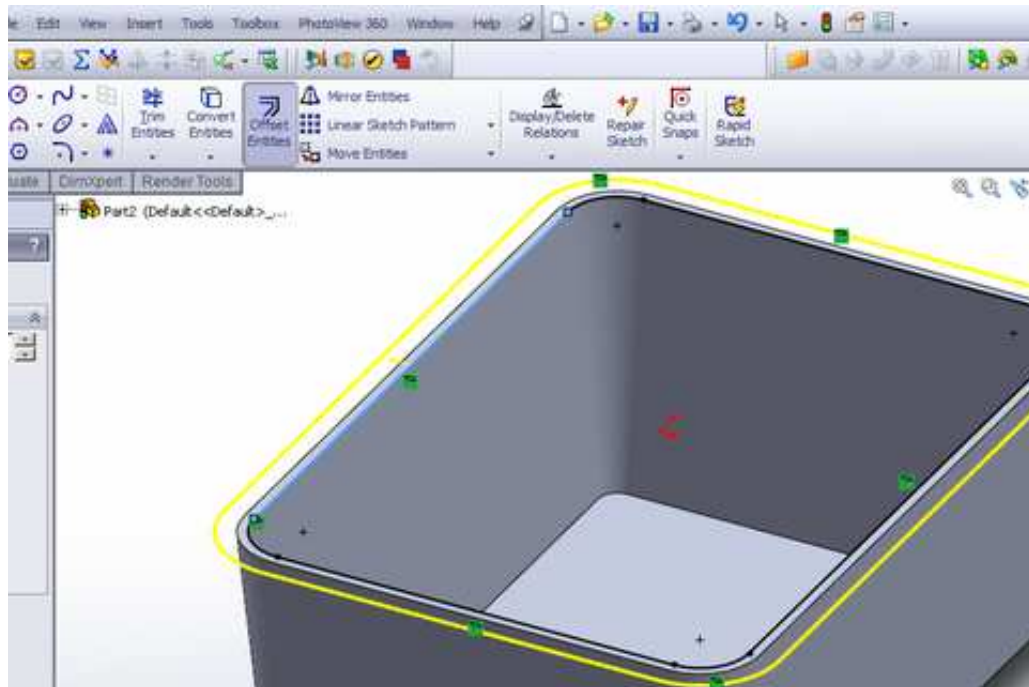
15. Step 15

now use offset Entities.



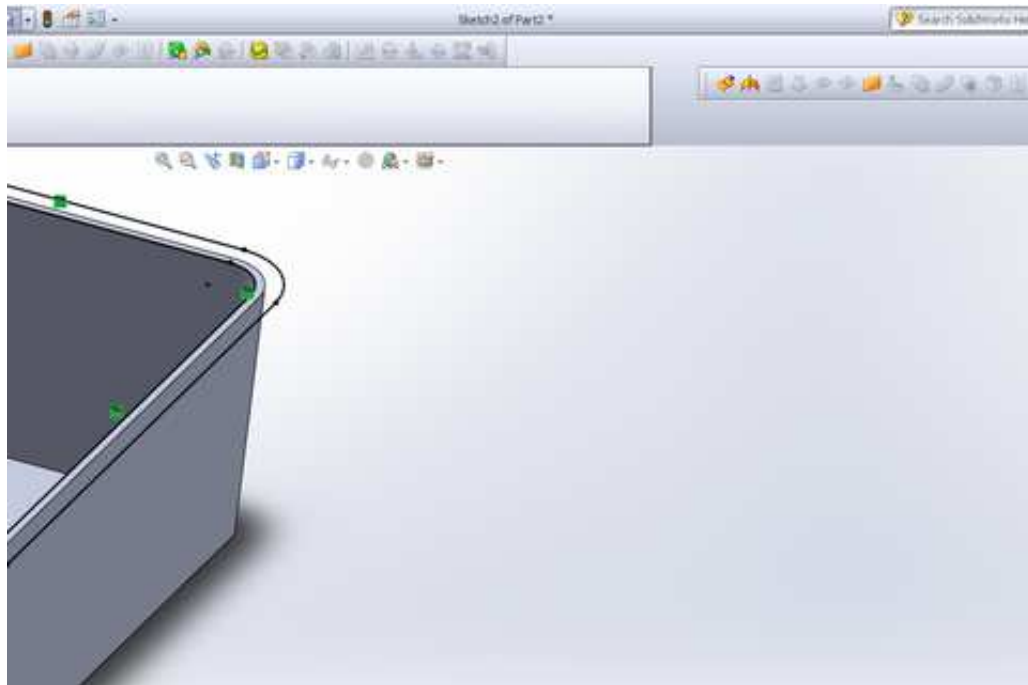
16. Step 16

select the same inside loop from before and offset it 1.5 cm



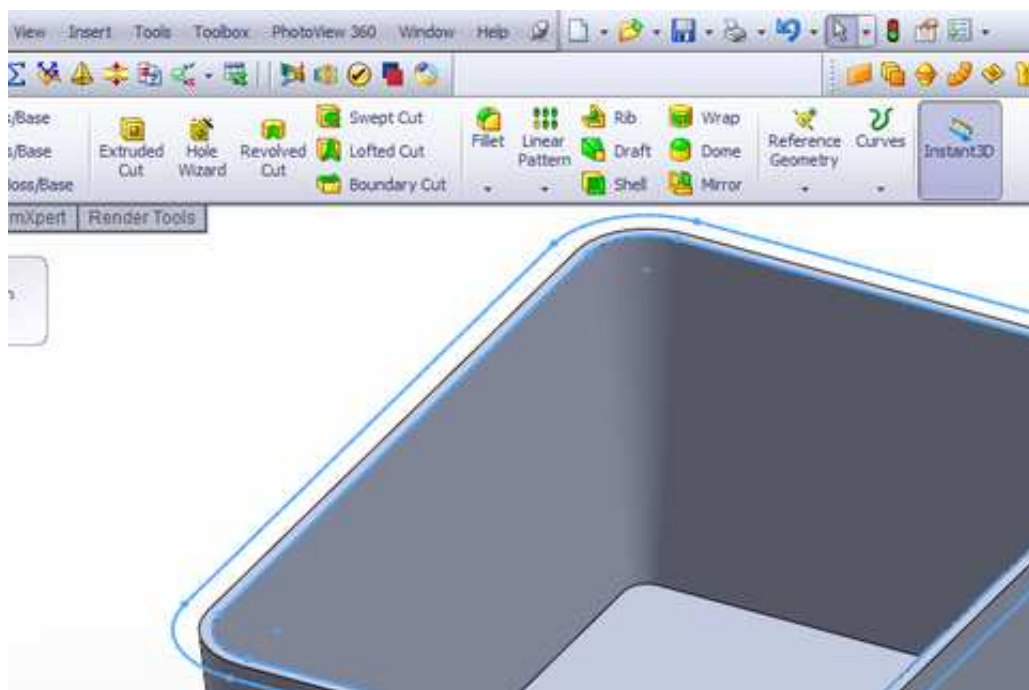
17. Step 17

exit this sketch



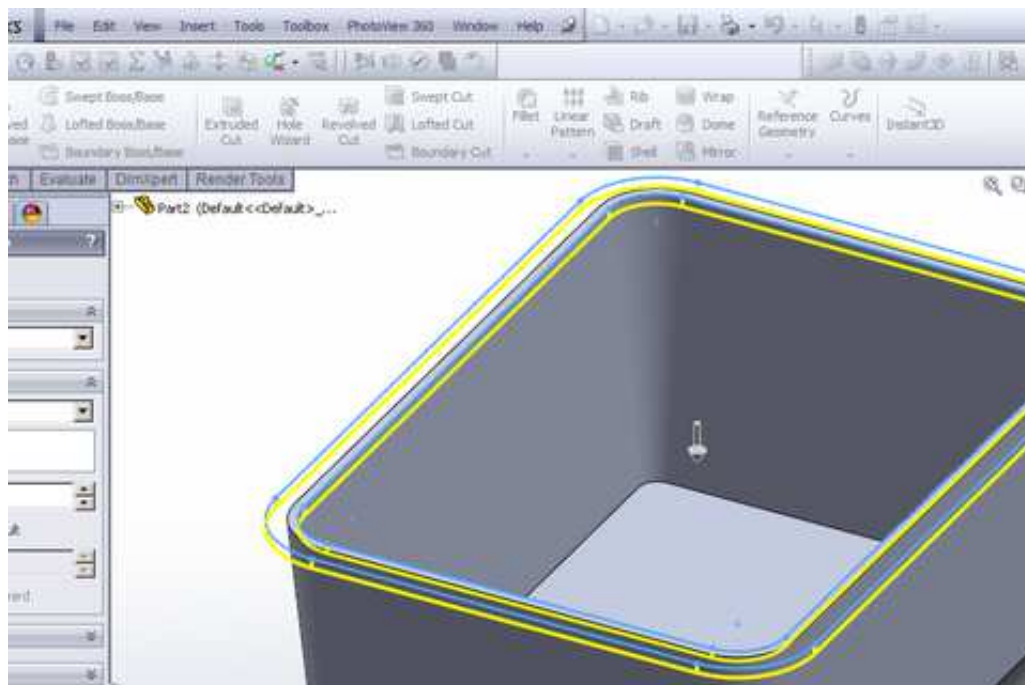
18. Step 18

With the last sketch still selected, choose the extruded boss/base button from the features menu ribbon.



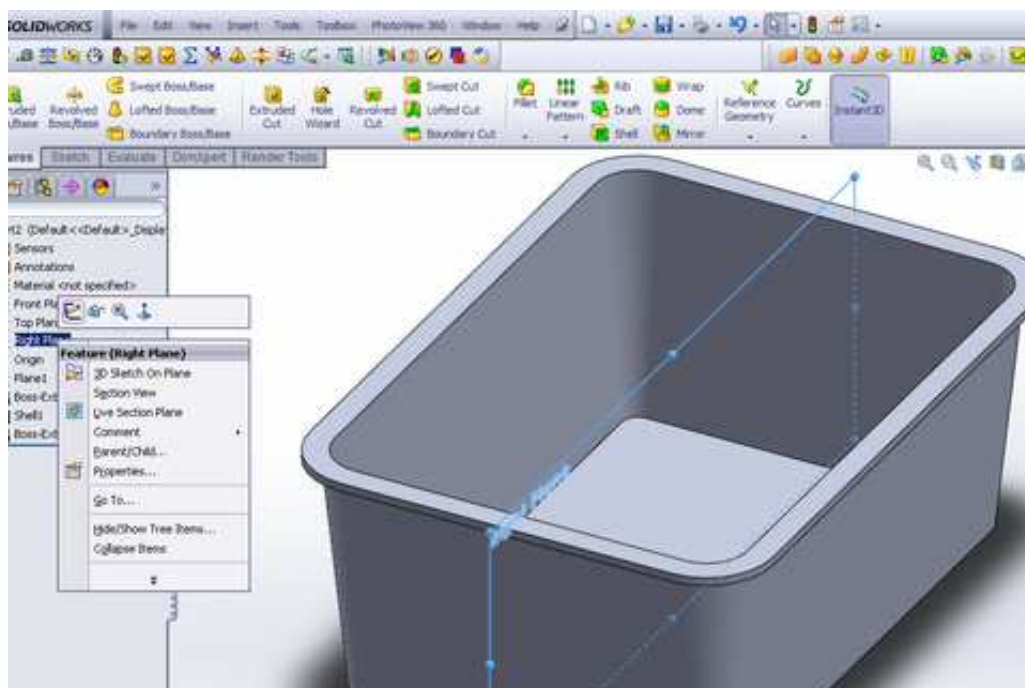
19. Step 19

extrude down .5 cm.



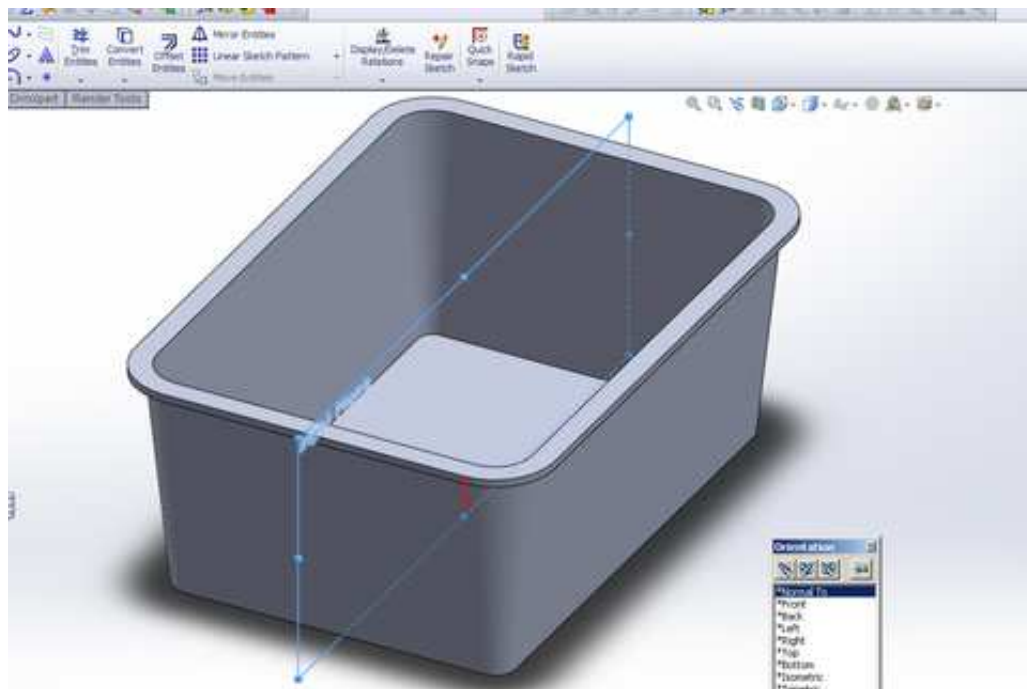
20. Step 20

Now to make the cuts in the side of the basket we will start with opening a sketch on the right plane.



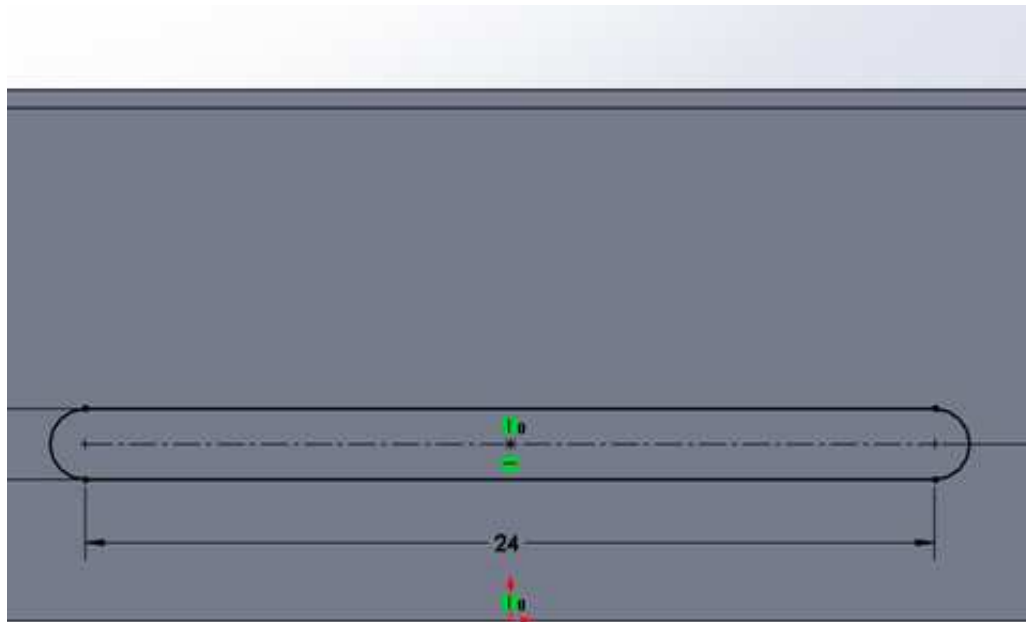
21. Step 21

Hit the space-bar and this will bring up a window to change the view. Lets make it normal to our sketch plane.



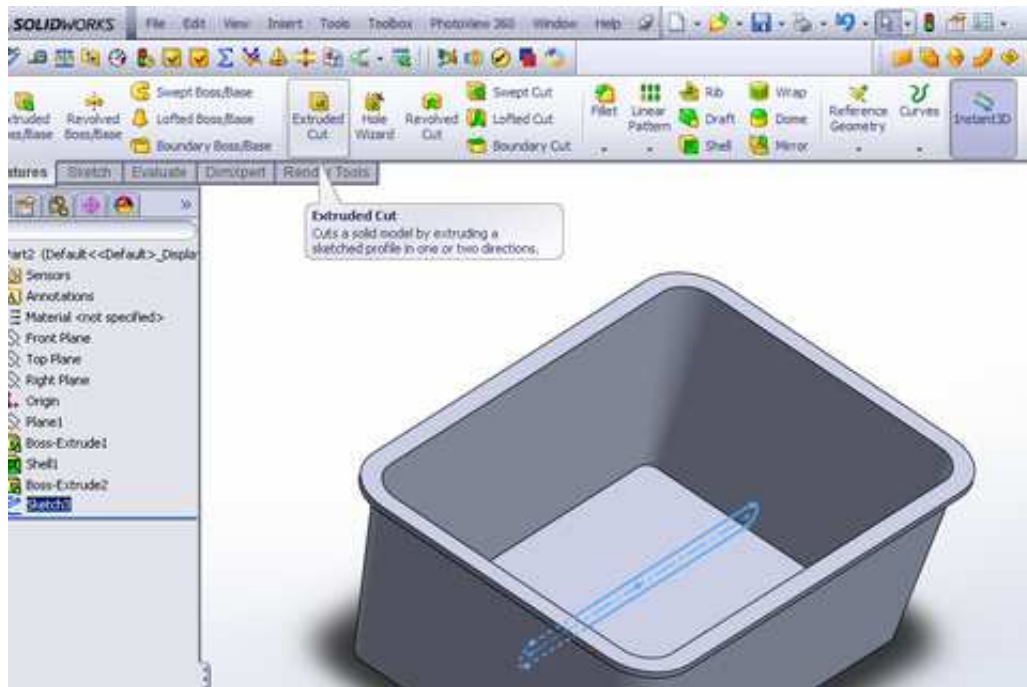
22. Step 22

Draw the sketch shown and then exit out of the sketch.



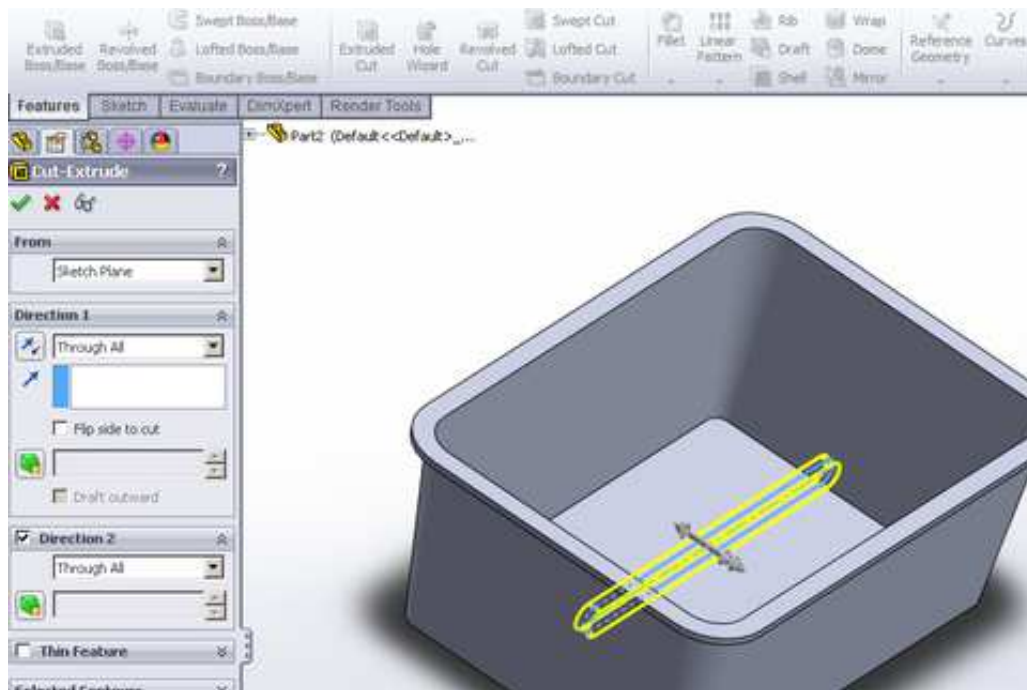
23. Step 23

Now in the features ribbon we will use the extruded cut option.



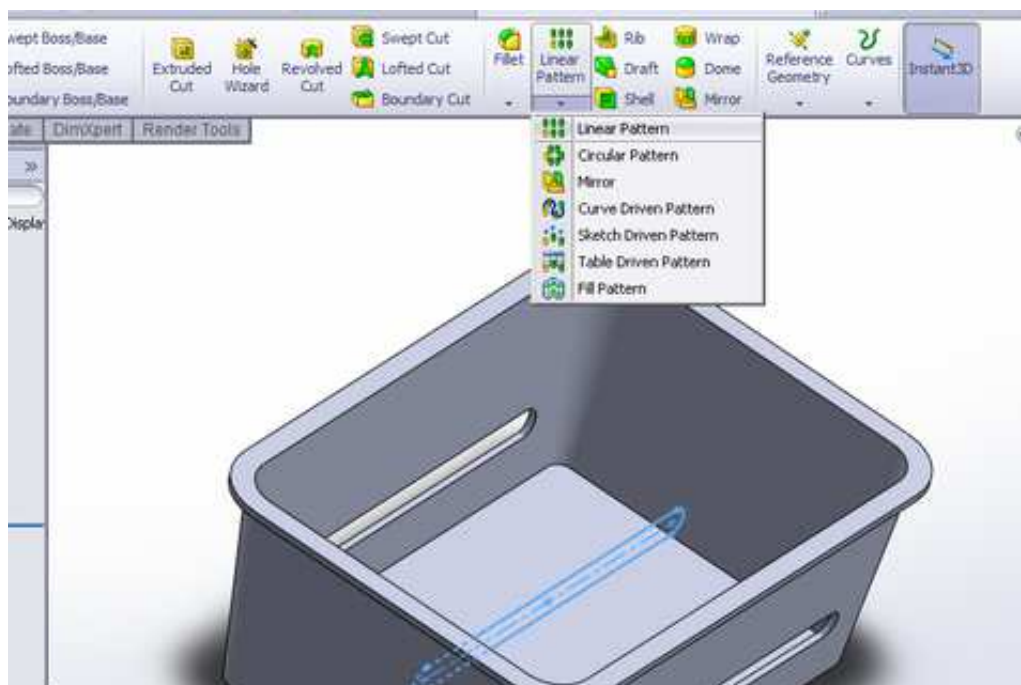
24. Step 24

Set directions 1 and 2 to through all.



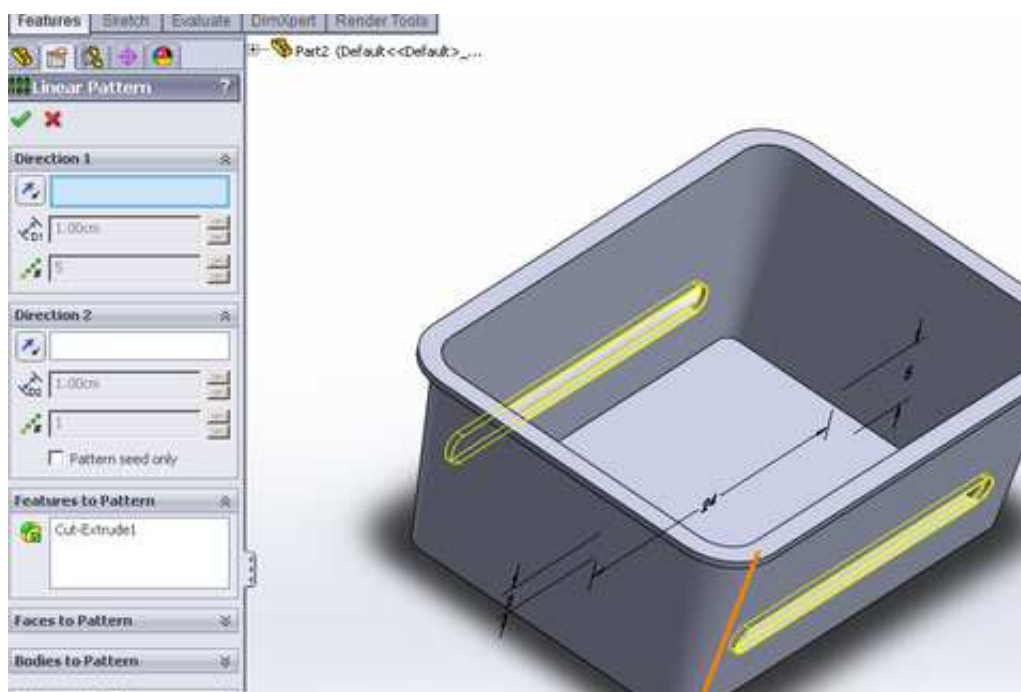
25. Step 25

Now to pattern the cut, use the linear pattern tool.



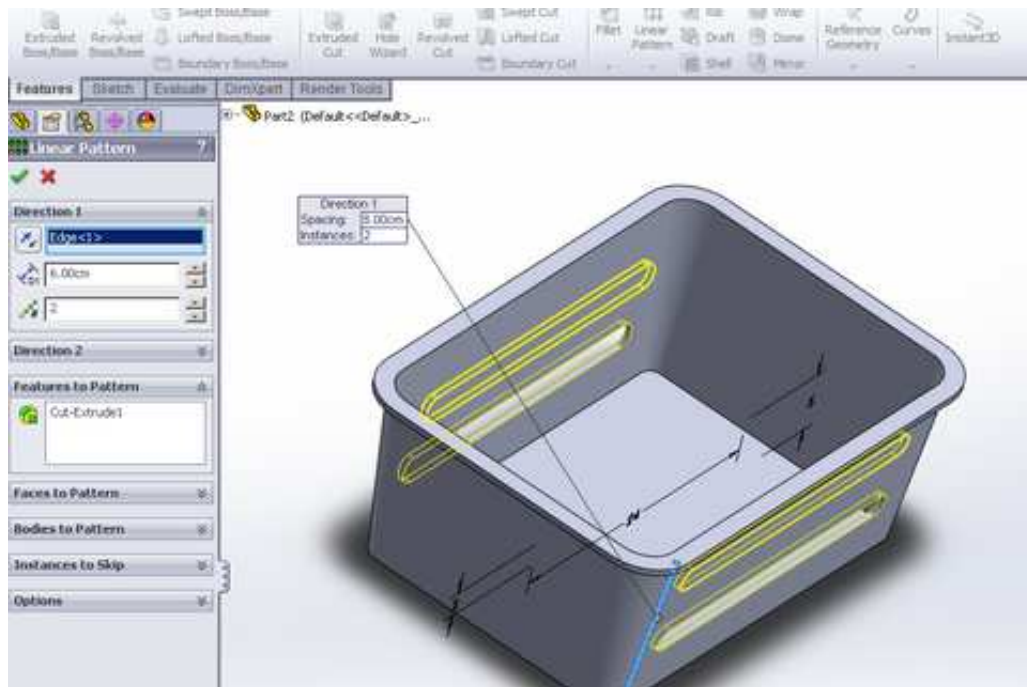
26. Step 26

For Direction 1, choose the vertical edge highlighted in orange.



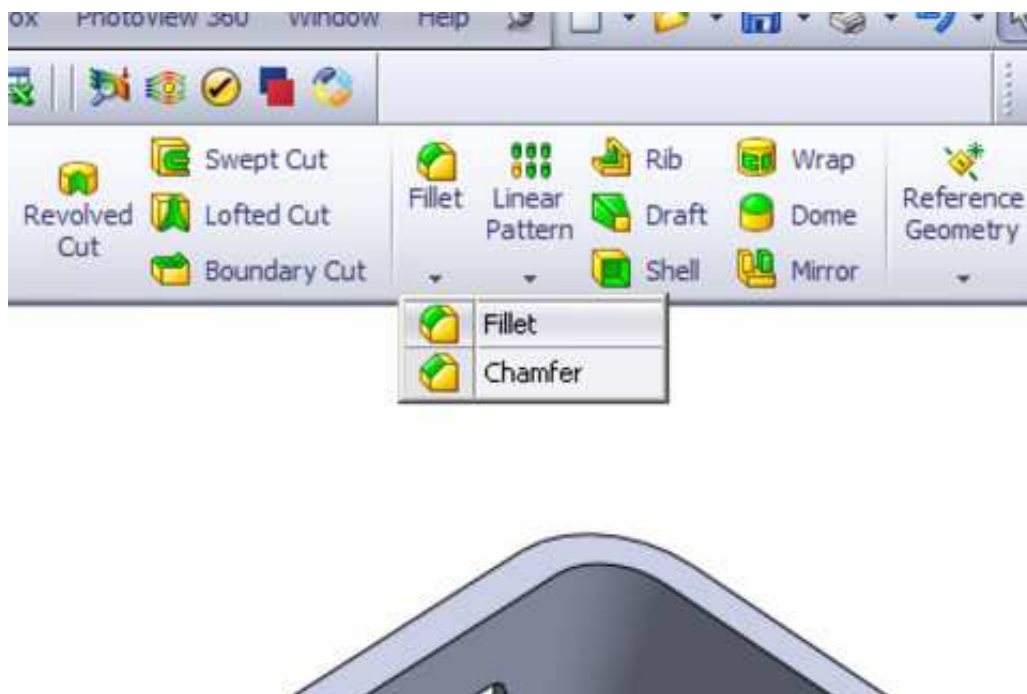
27. Step 27

Set the spacing to 6 cm and the number of instances to 2.
Accept the settings by clicking the green check.



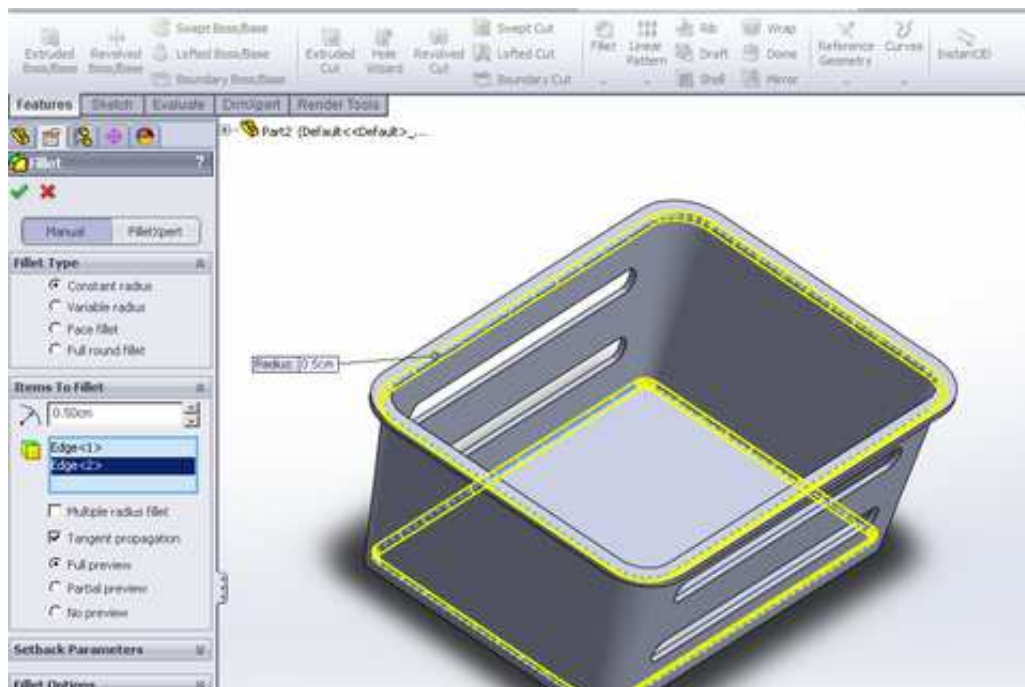
28. Step 28

Now for a few fillets



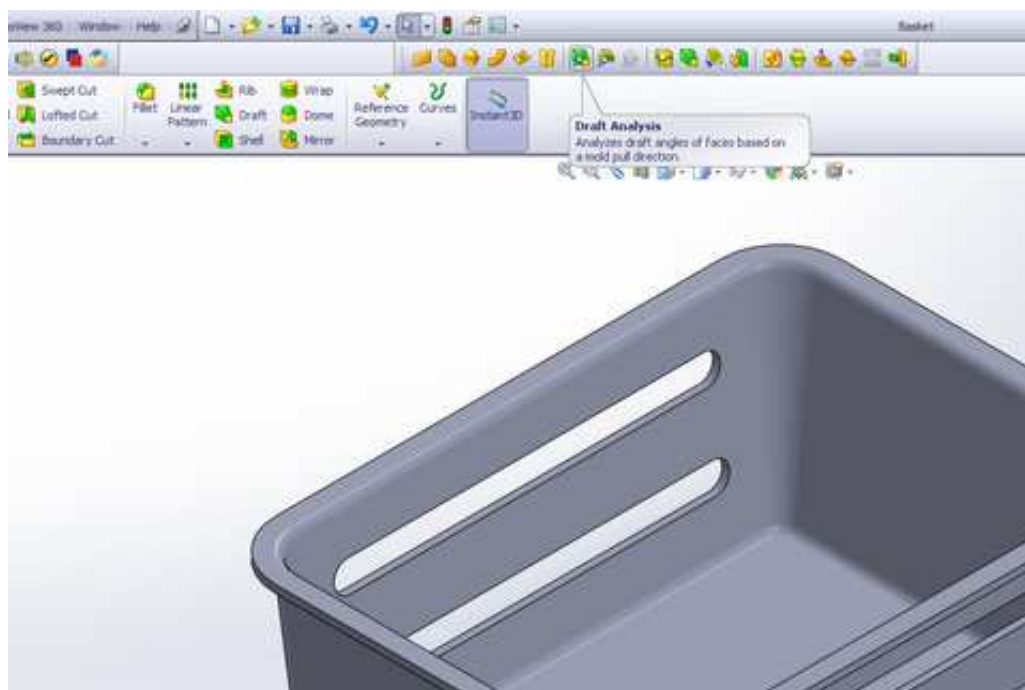
29. Step 29

add .5 cm fillets to the inside top and bottom edges.



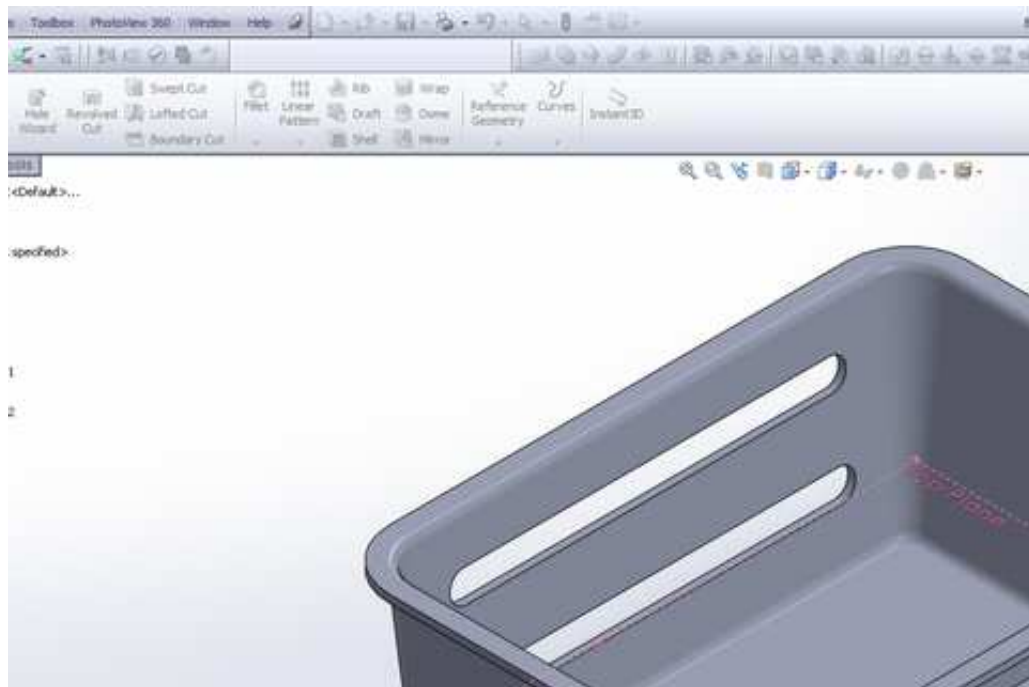
30. Step 30

Now that our part is done, we need to perform Draft Analysis on it before we can start making our mold.



31. Step 31

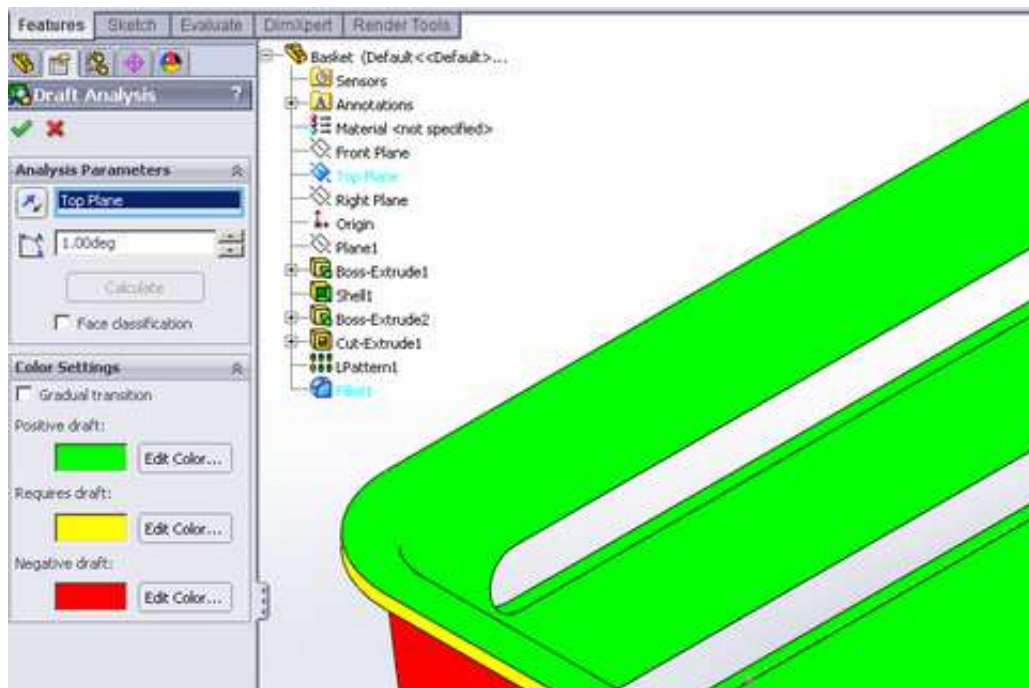
After clicking on the draft analysis icon, we need to select the top plane as the direction of pull and a draft angle of 1 degrees and then press the green check.



32. Step 32

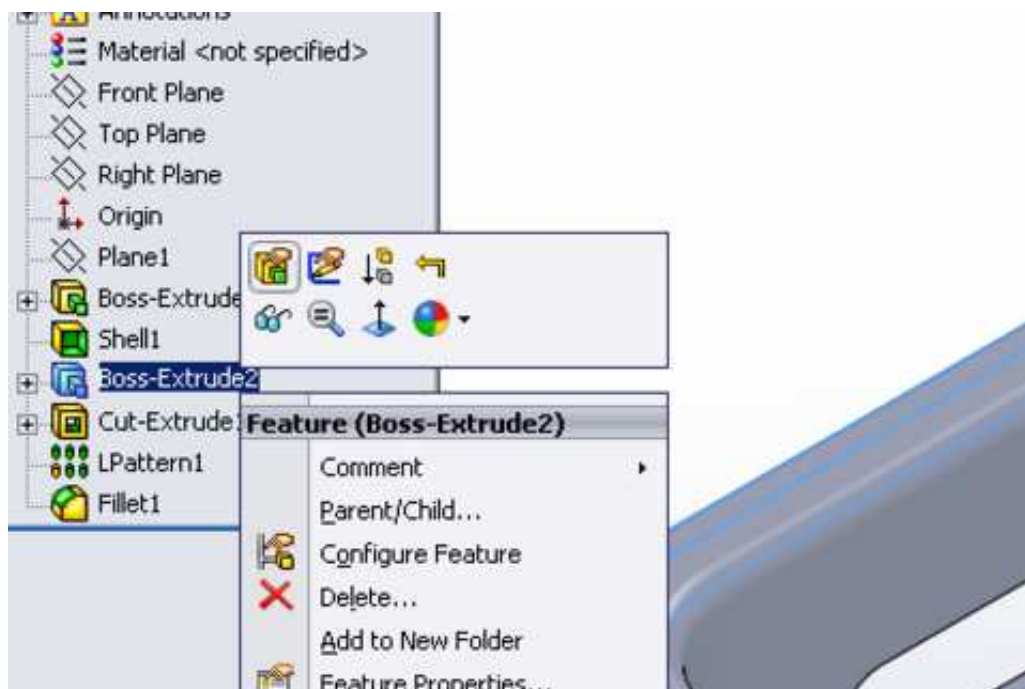
The faces of our part will change color depending on the draft of the face. We can see the edge around the top lip of our basket is yellow and needs draft added to it.

Clicking the green check again will exit out of draft analysis.



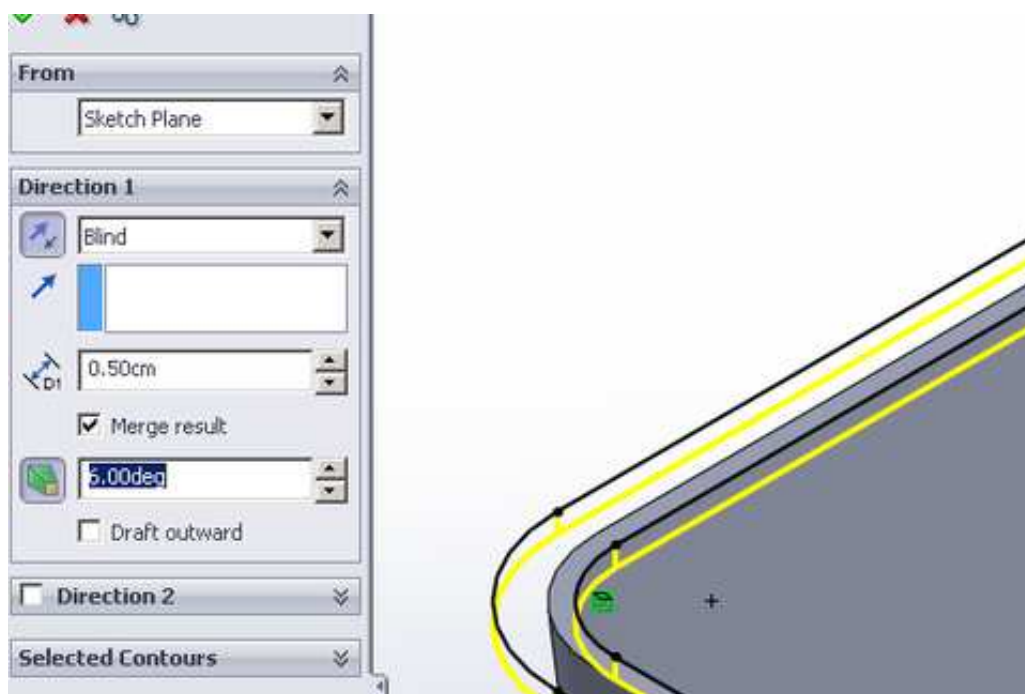
33. Step 33

To add the required draft, we will right click on the feature needing it (boss-extrude2) and click the upper left icon allowing us to edit this feature.



34. Step 34

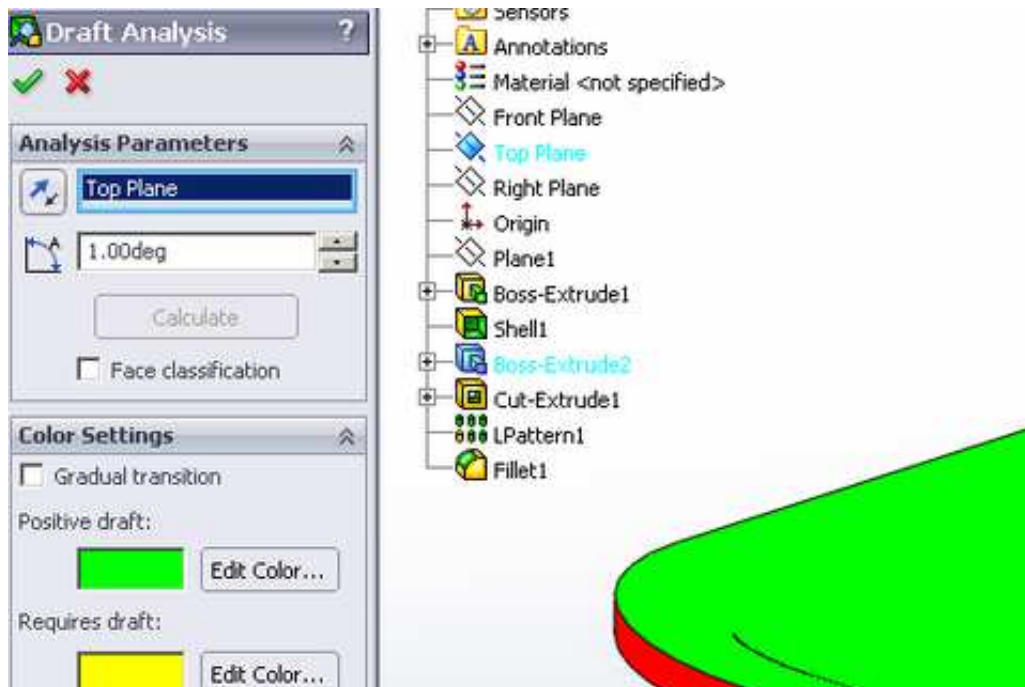
click the draft icon and this will draft the feature.



35. Step 35

Now when we perform our draft analysis we can see that face is no longer yellow and is now red.

Our part is almost ready to make a mold now.



36. Step 36

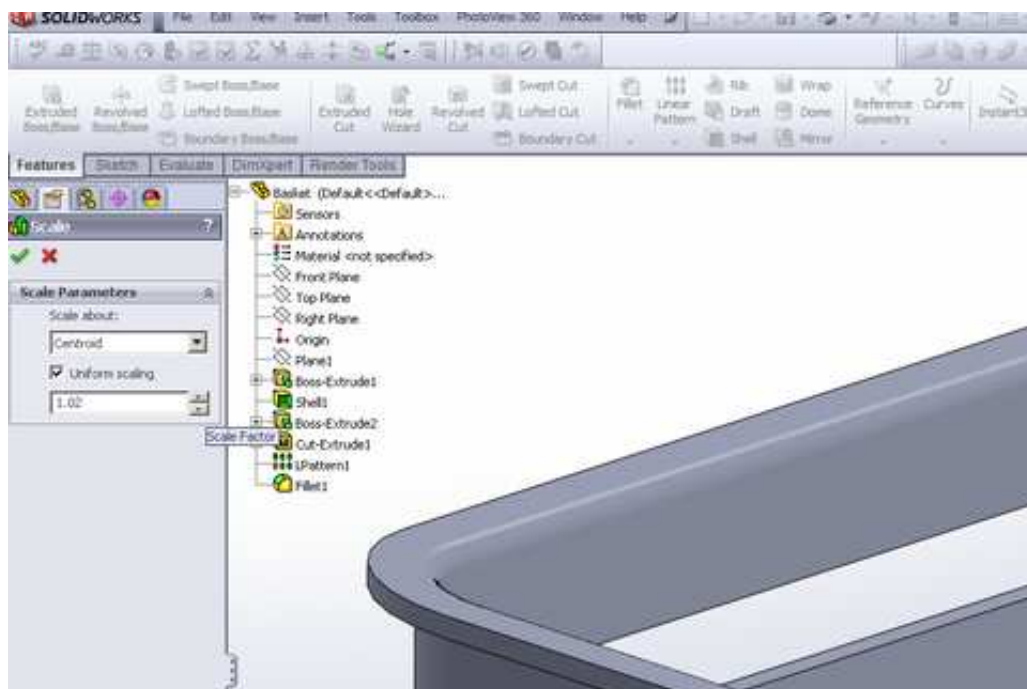
Depending on the material, the part will shrink when cooled after molding. Therefore our mold needs to be a little larger than we want our final part to be. So we will scale up our part.

Click the scale button in mold tools menu.



37. Step 37

For this model we will scale it uniformly about the centroid with a scale factor of 1.02 but you can play around with the options to see what will happen.

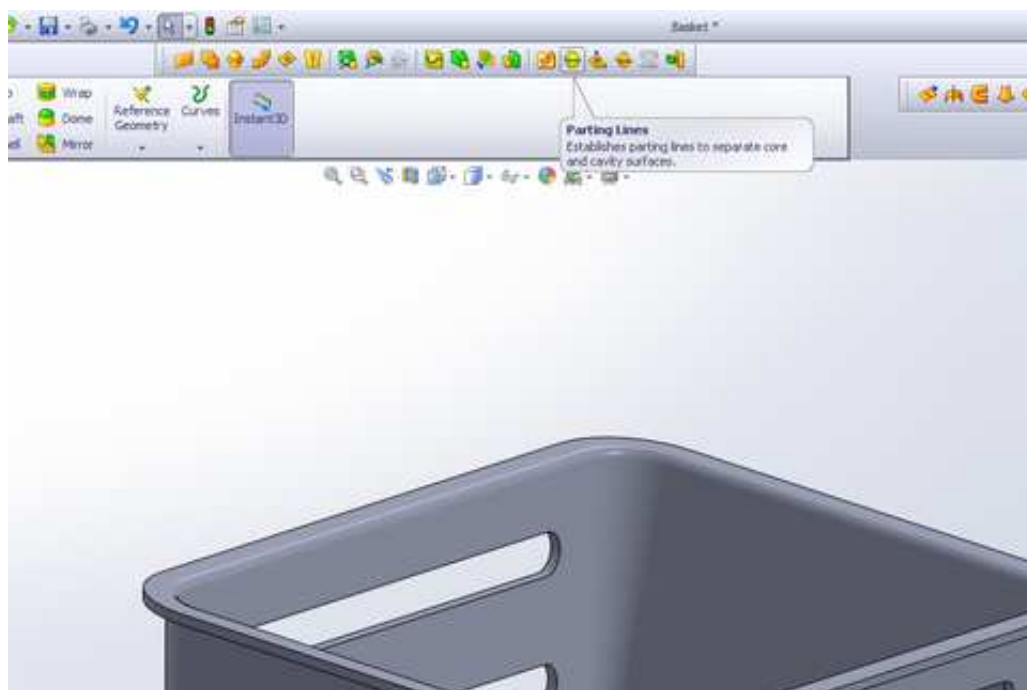


38. Step 38

Now to make a parting line.

The parting line will separate the positive and negatively drafted faces (the red from the green when we did the draft analysis) and this is also where the two halves of our mold will meet.

Click the Parting Line icon.

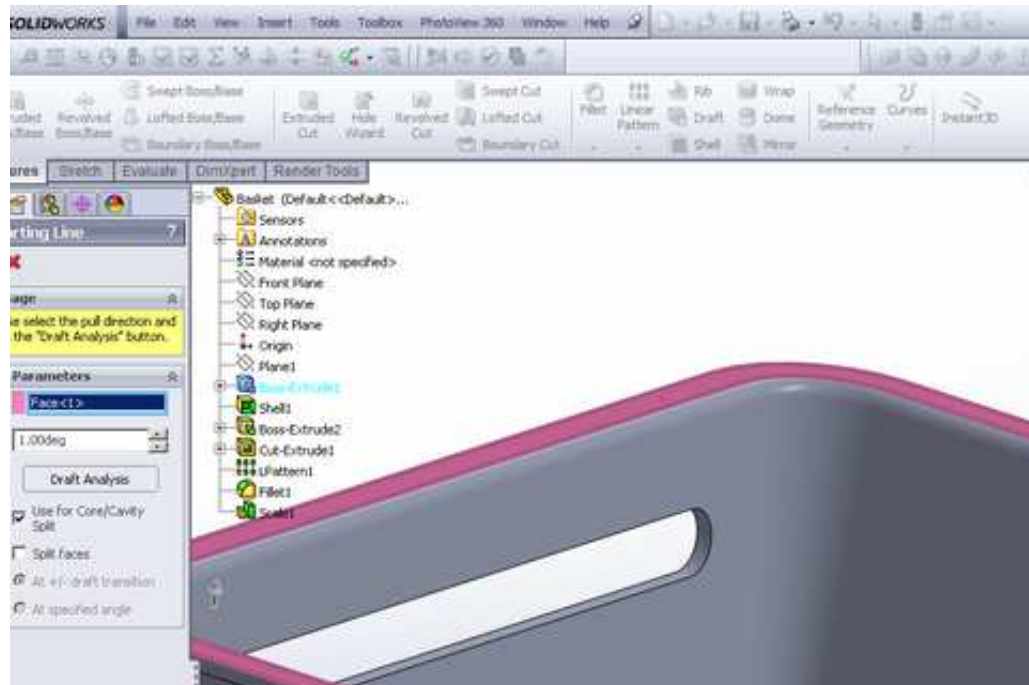


39. Step 39

Select the top face of the basket to define the direction of pull for our mold and set the draft angle to 1 degree.

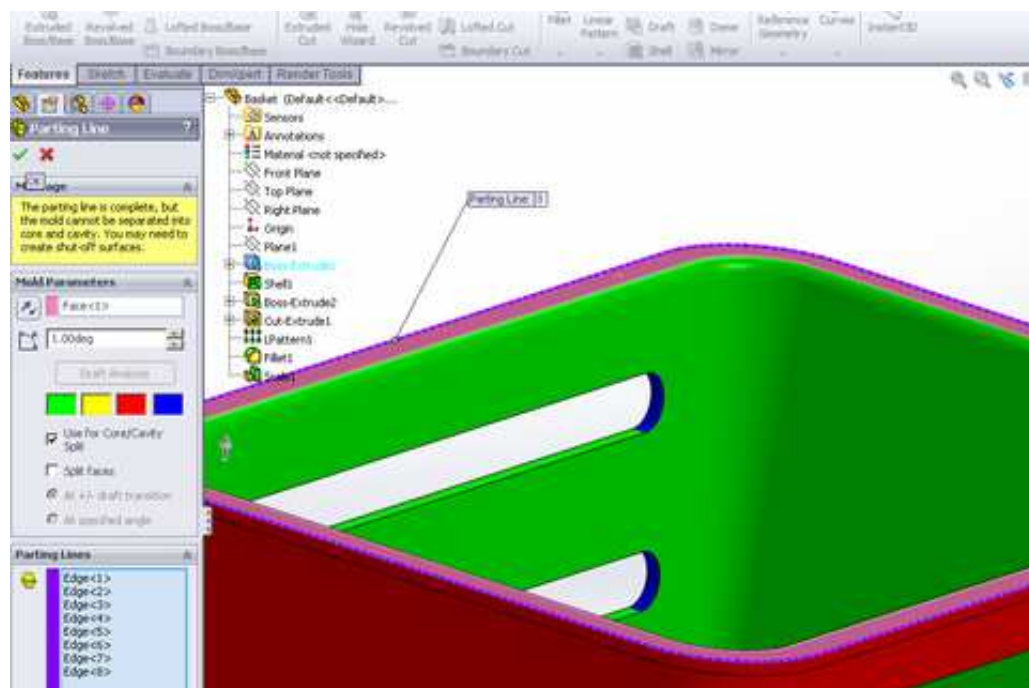
Also be sure that the box for "use for core/cavity" is checked.

Then click the draft analysis button.



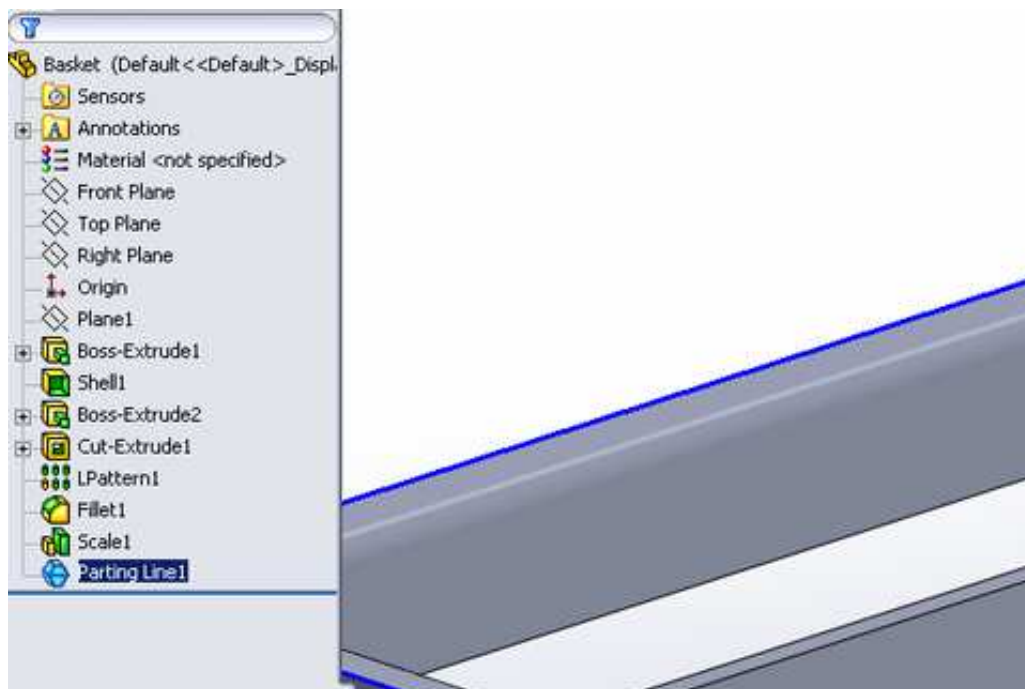
40. Step 40

the parting line is created at the intersection of the positive and negatively drafted faces (red and green).



41. Step 41

click the green check and our parting line is created.



42. Step 42

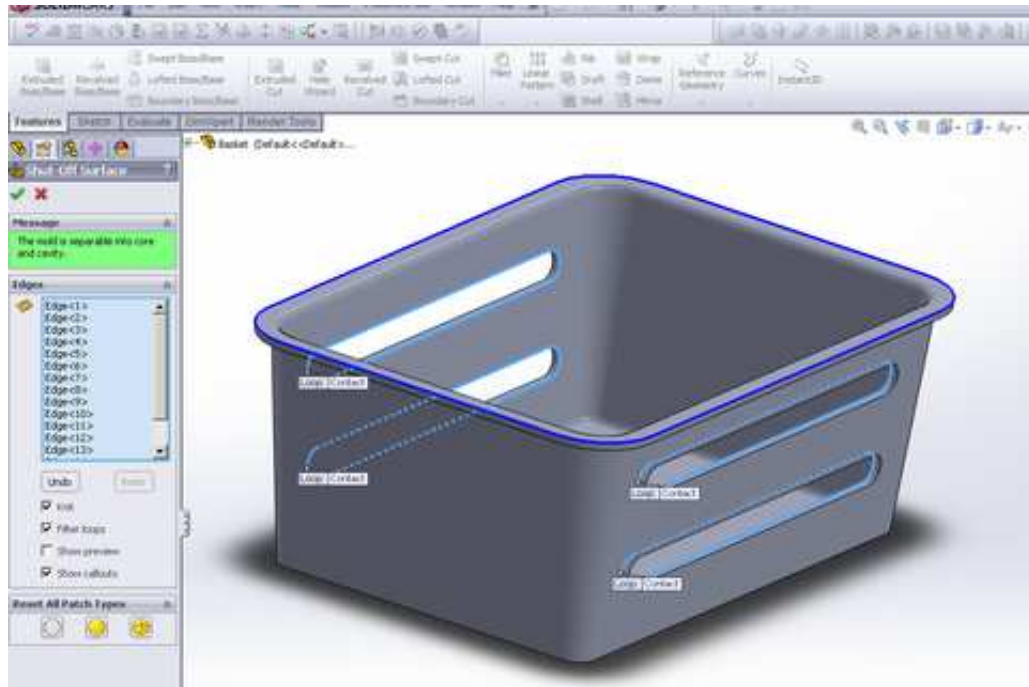
Now to create shut-off surfaces. We need these because of the cutout holes in the side of the basket. These surfaces will denote where the core and cavity halves of the mold will meet.

Click on the shut-off surfaces icon.



43. Step 43

The openings in the part are automatically detected and the inner edge of each opening is automatically selected.
Be sure the knit option is selected.



44. Step 44

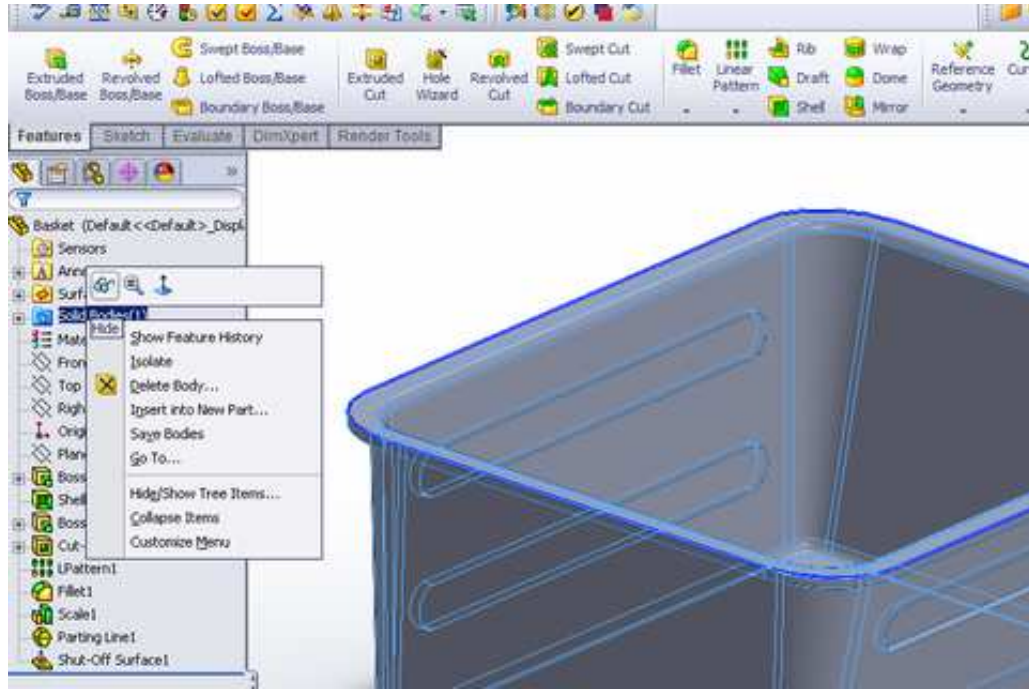
In the reset all patch types menu, click the all tangent button (the right one with a cross in the ball).



45. Step 45

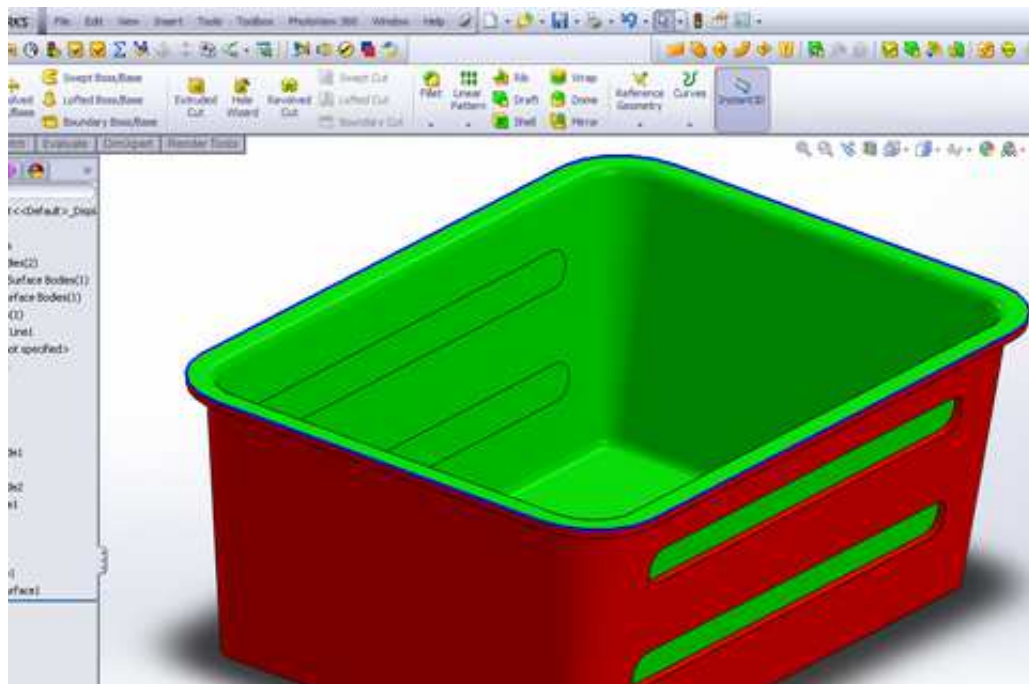
after clicking the green check, you will see in the feature tree that you now have both solid bodies and surface bodies.

You can hide the solid body and expand the surface bodies menu and hide any of the surface bodies to see what you have.



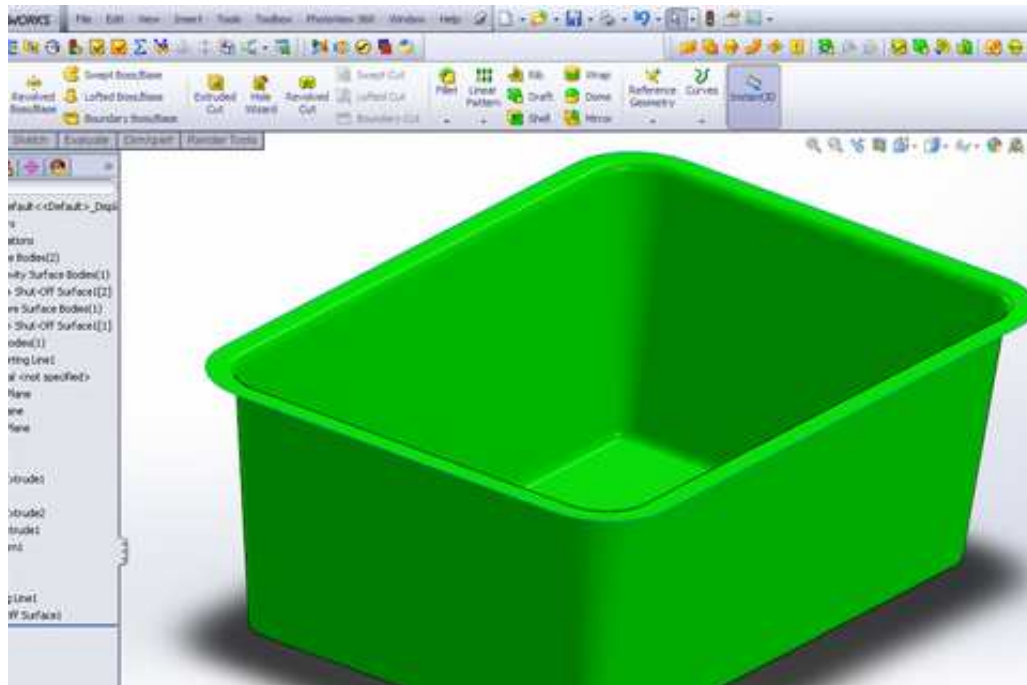
46. Step 46

Here are the cavity and core surface bodies with the solid body of the part hidden...



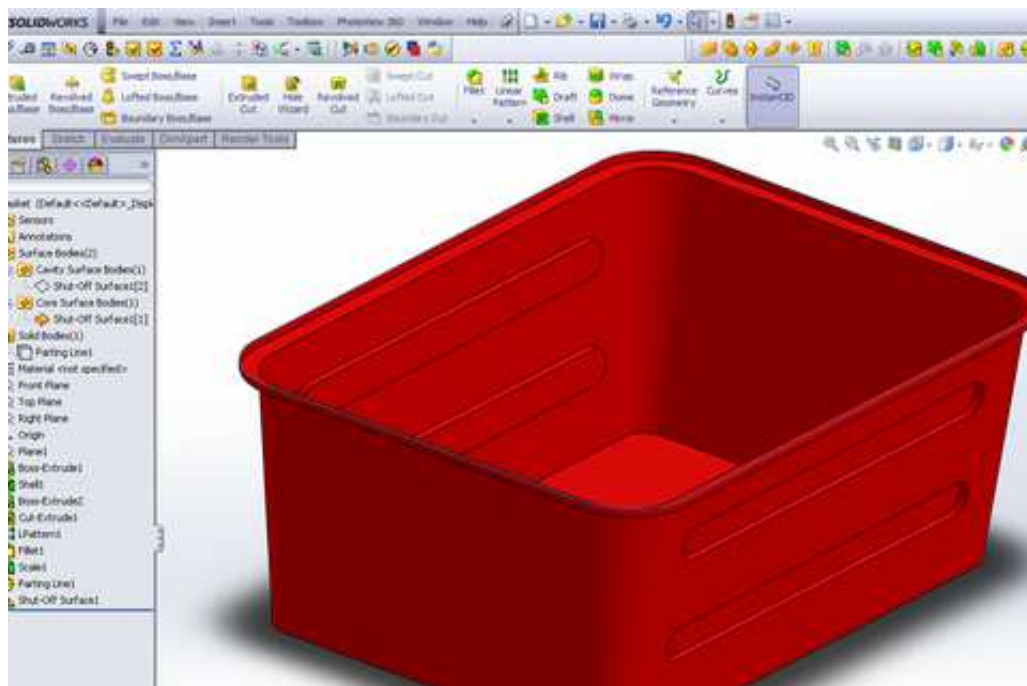
47. Step 47

just the cavity surface...



48. Step 48

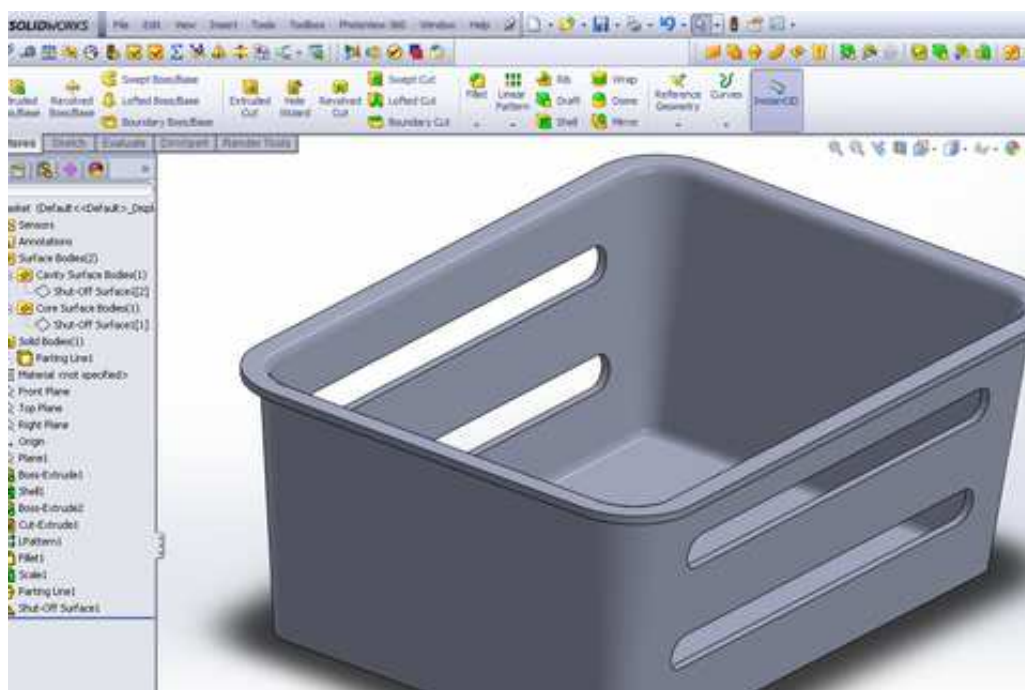
just the core surface...



49. Step 49

and just the solid body of the part.

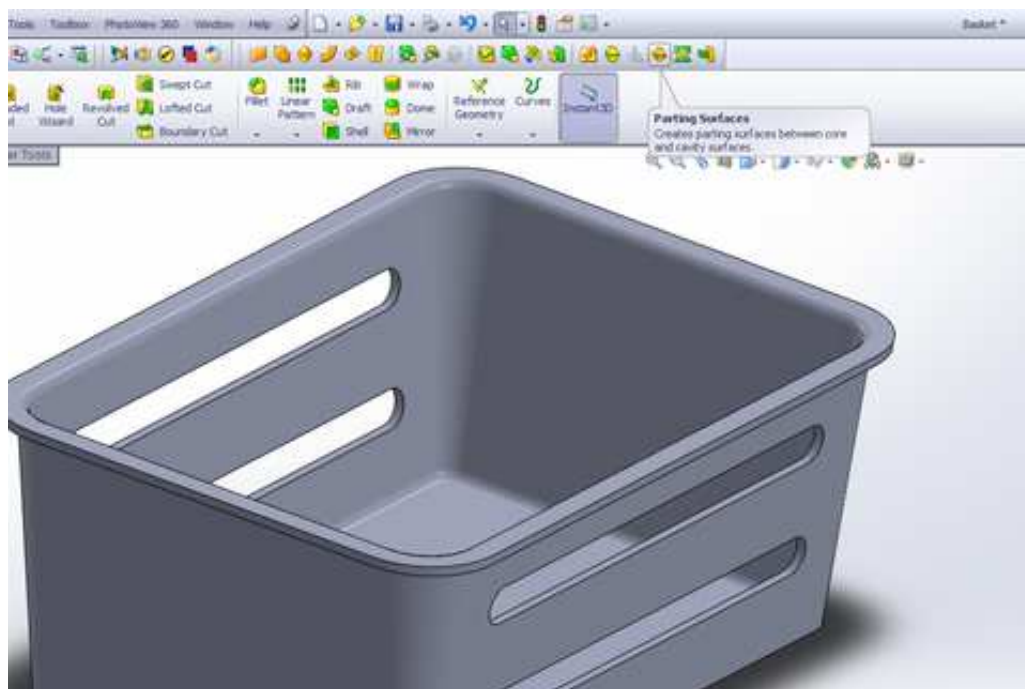
(notice that part is encased between the two surfaces)



50. Step 50

Now we need to create a parting surface. This surface extends away from the part and is the surface that separates the two halves of the mold from each other.

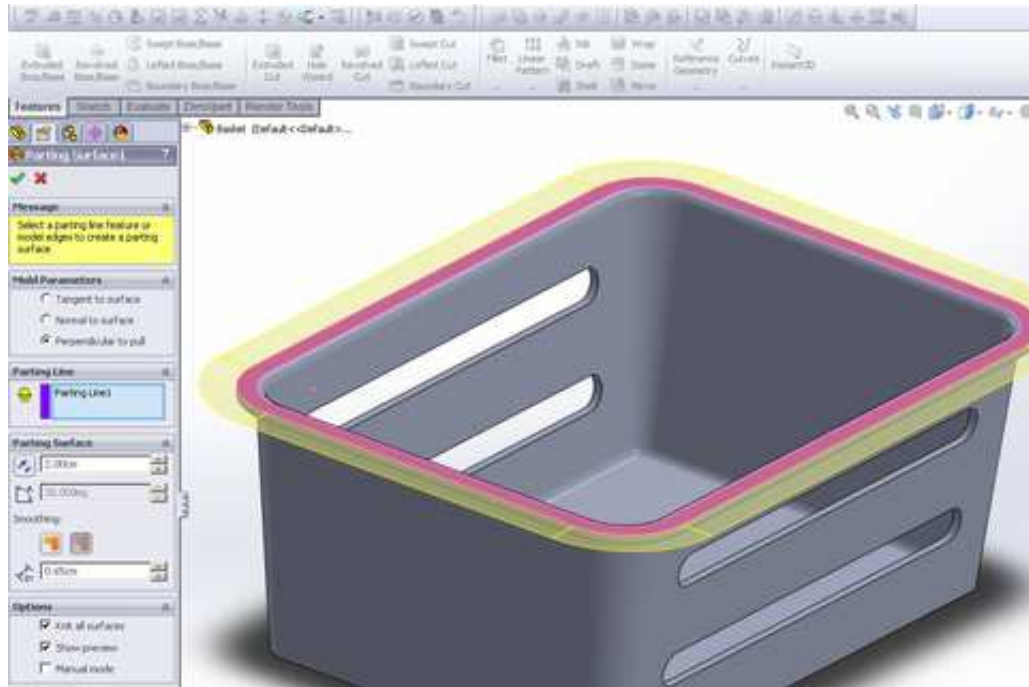
Click on the Parting Surfaces icon.



51. Step 51

for this mold we will use the mold parameter of perpendicular to pull and the parting line will automatically be selected.

Make sure the rest of the settings are as shown in the picture and then click the green check to complete.



52. Step 52

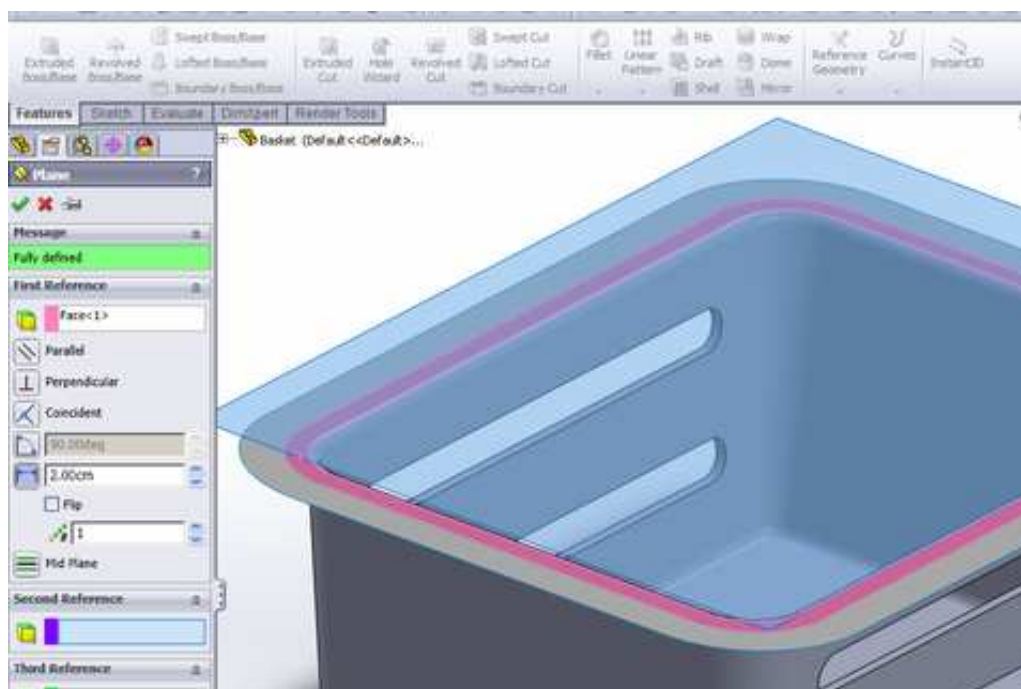
Now that we have these three surfaces it is time to create the mold.

First we need to create a plane that will be where our two halves of the mold meet.



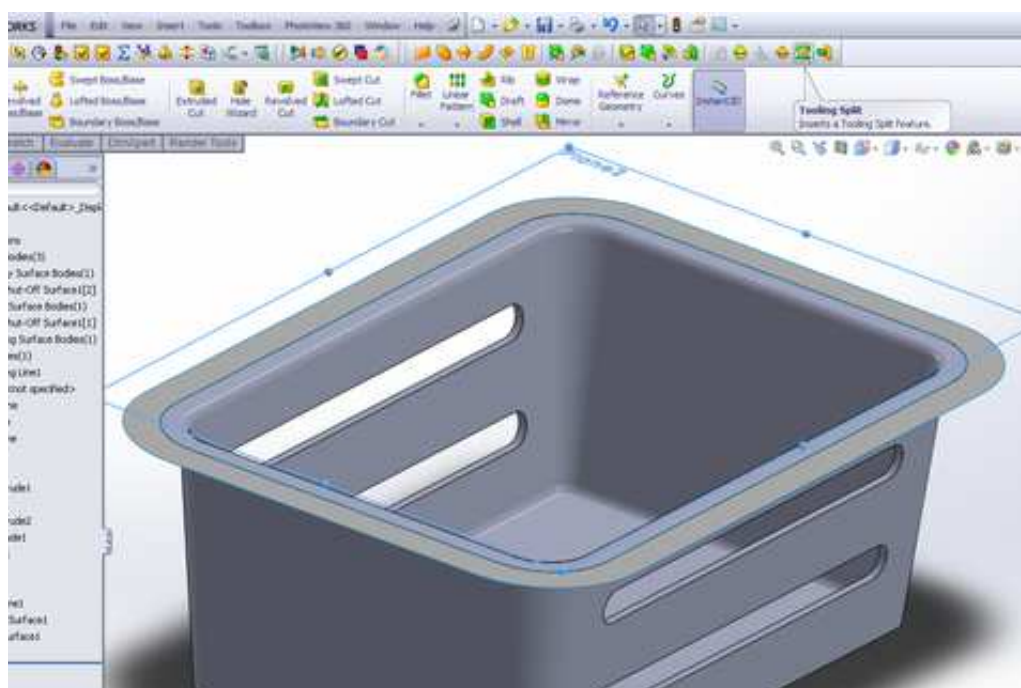
53. Step 53

create a plane that is offset 2 cm above the top of our part.



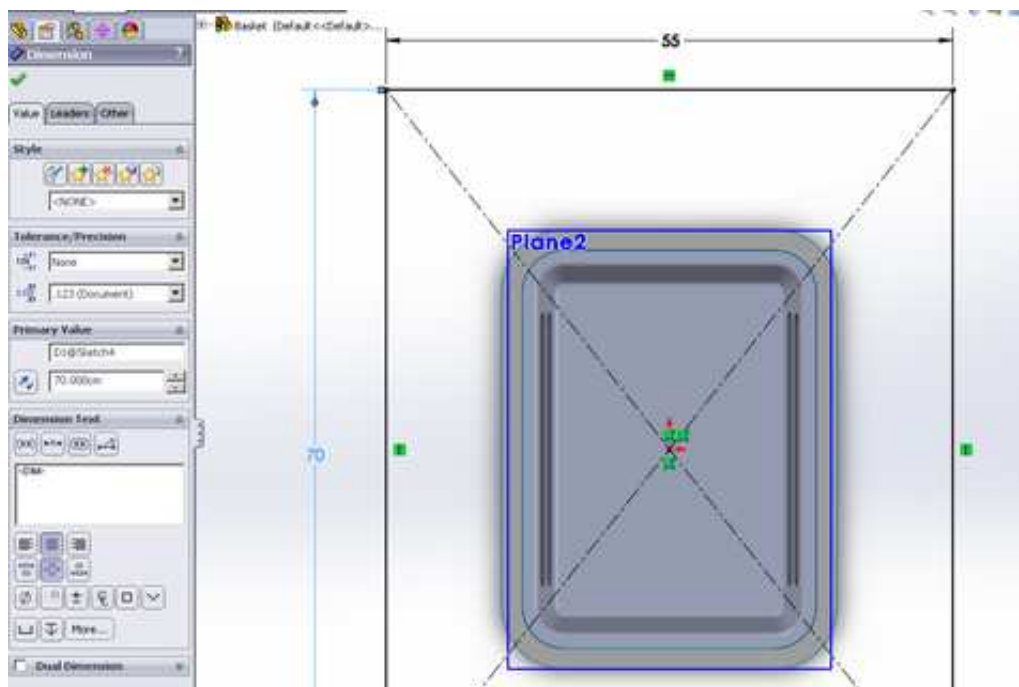
54. Step 54

with this new plane still selected, click the tooling split icon.



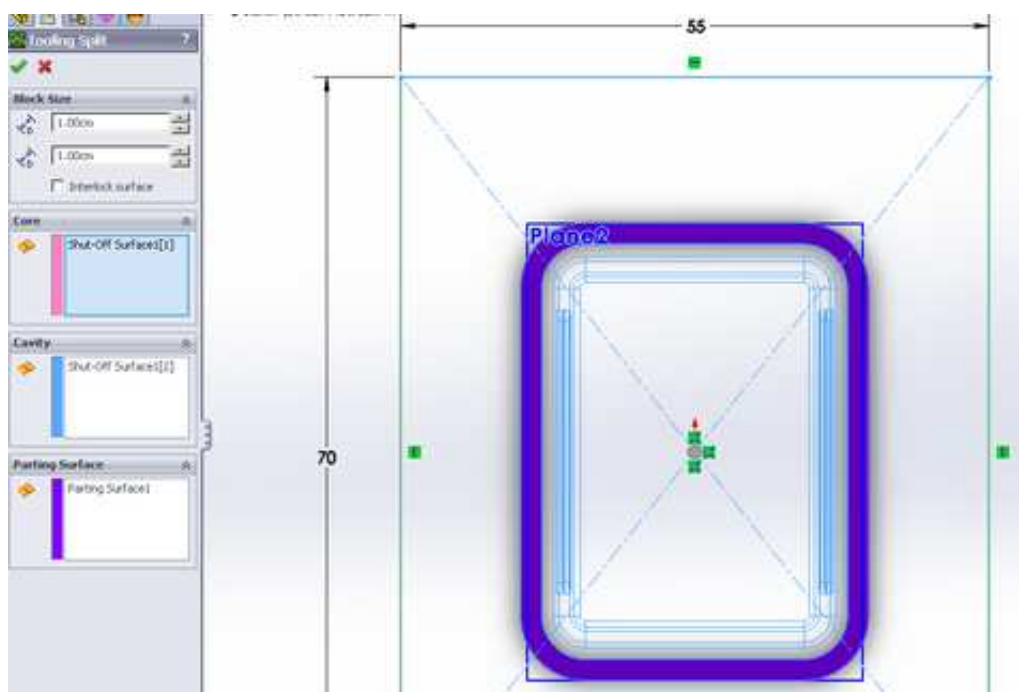
55. Step 55

This will open a new sketch on that plane. Draw the sketch shown in the picture.



56. Step 56

When you exit the sketch, it will bring you to the tooling split Property Manager.

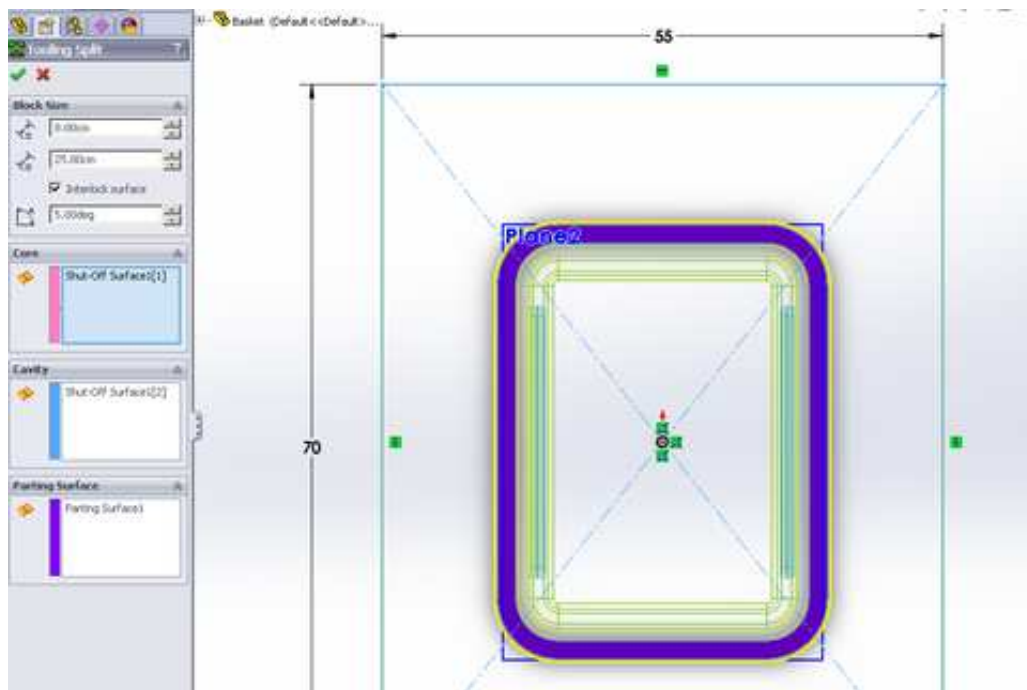


57. Step 57

select the interlock surface box. this allows the two halves of the mold to interlock with each other allowing for them to stay properly aligned.

Set the other dimensions as shown in the picture.

It already knows what are surfaces are and has them properly selected.

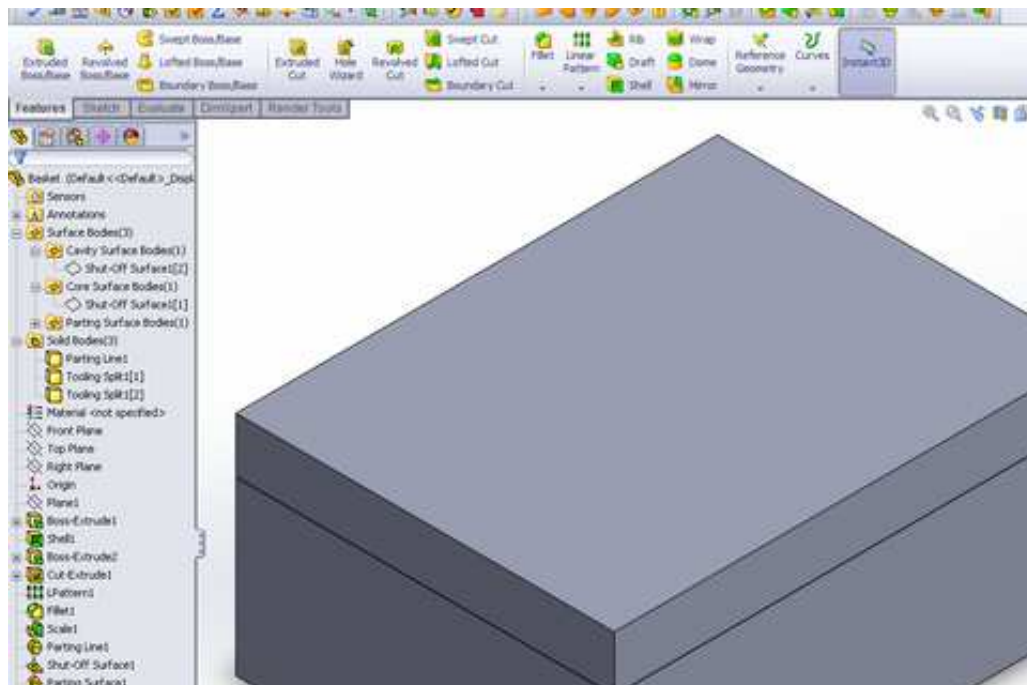


58. Step 58

click the green check and our mold is created.

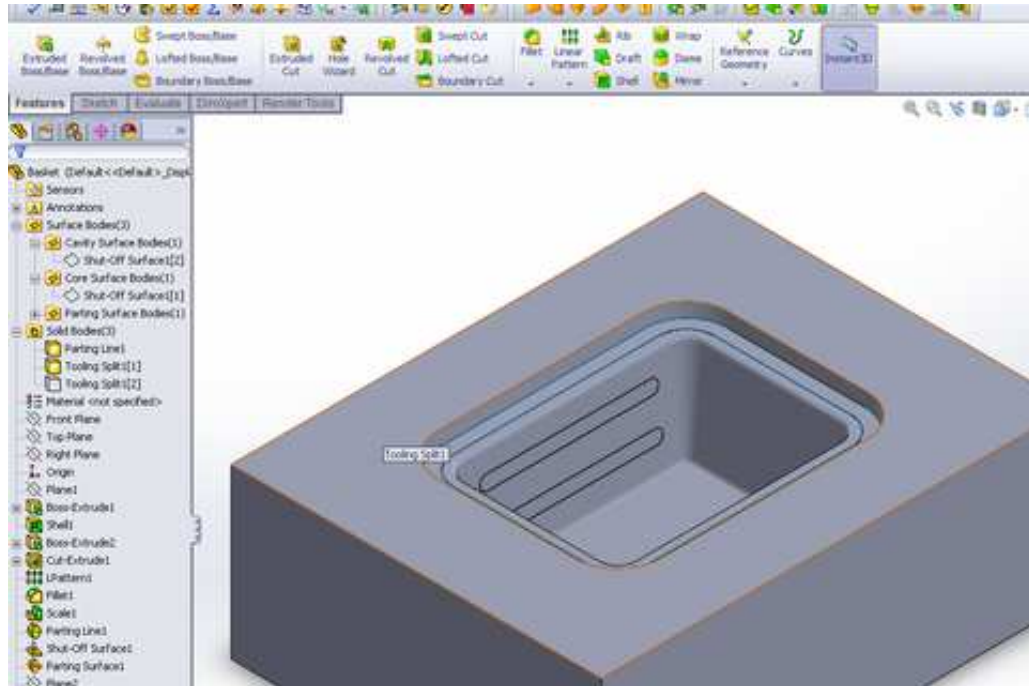
if you look at the feature tree, you can see that now we have three surface bodies and three solid bodies.

take some time to hide these to see what we have.



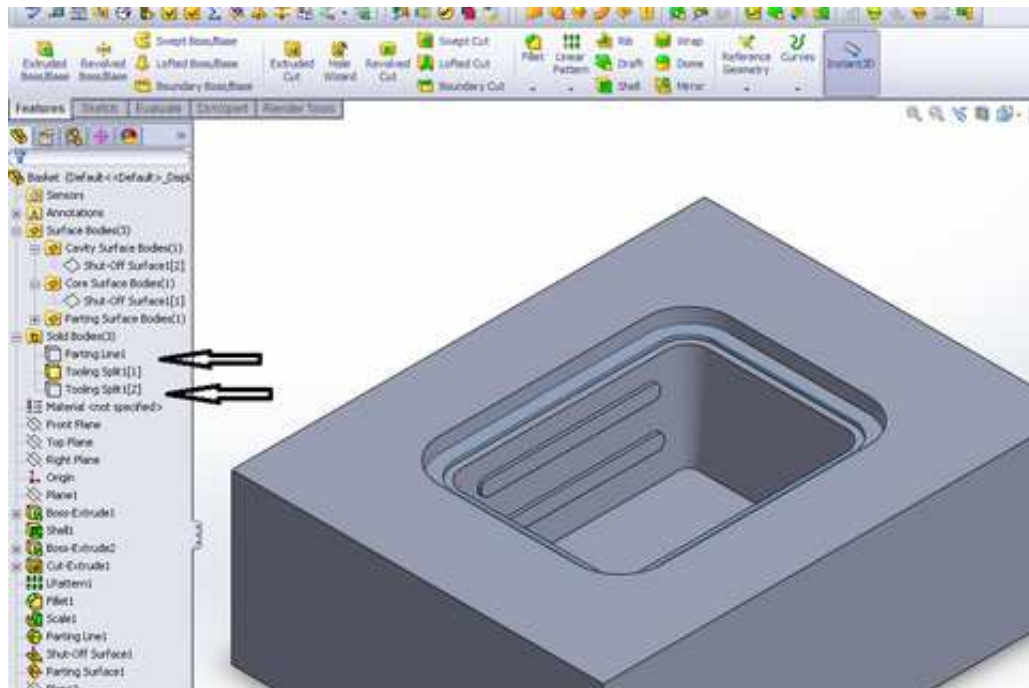
59. Step 59

Here is the cavity and the part. As you can see, the part will be stuck in the cavity because of the cutouts in the basket.



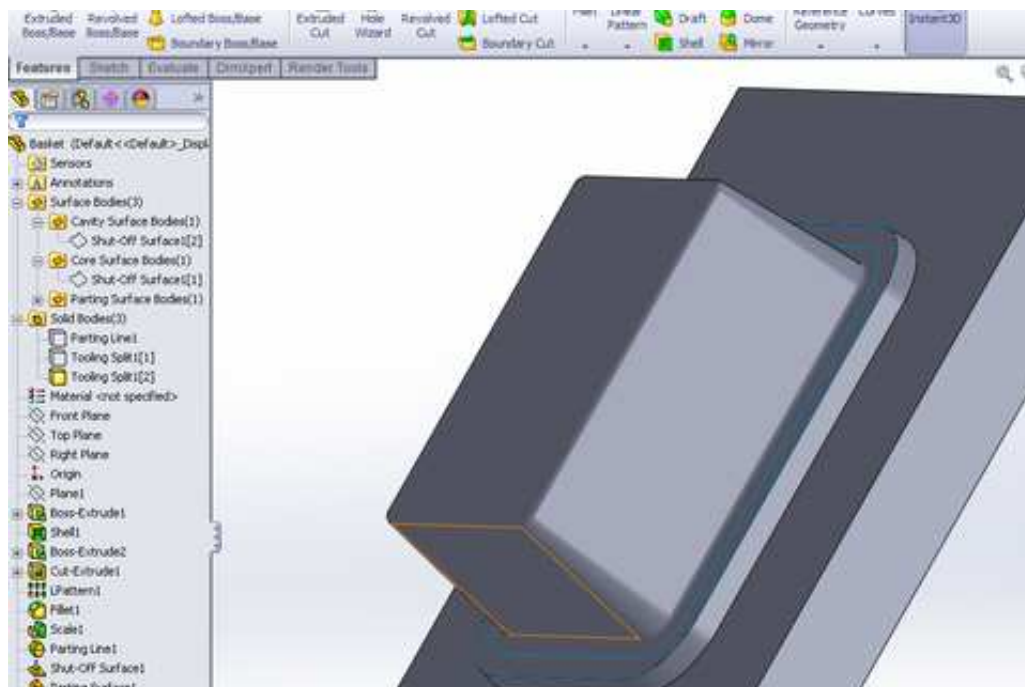
60. Step 60

just the cavity...



61. Step 61

and the core...



62. Step 62

and our basket...

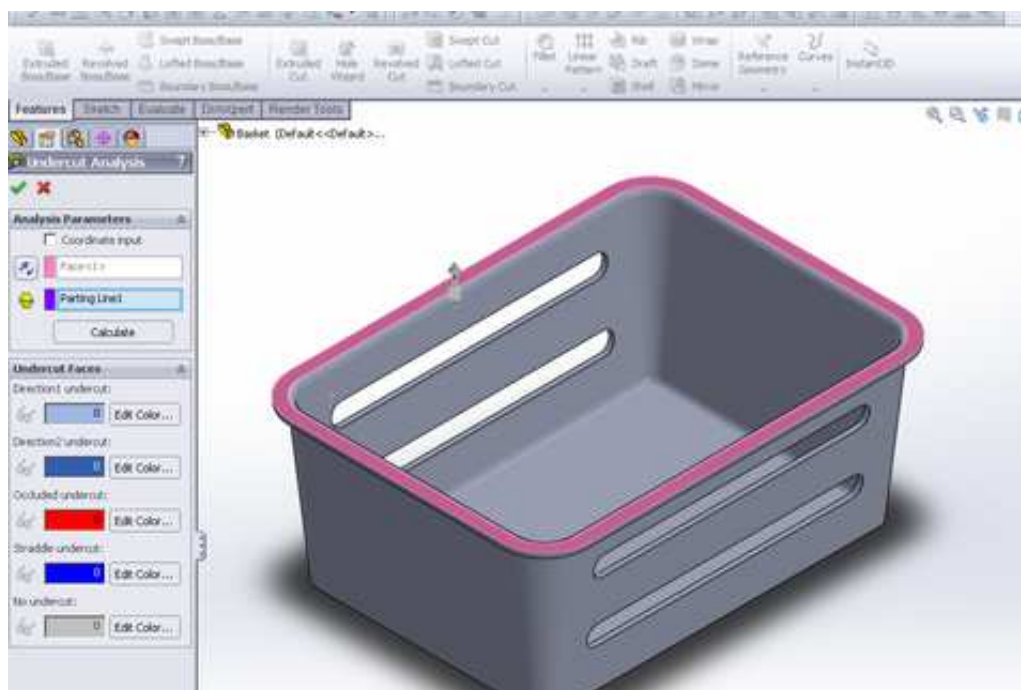
Now lets make sure everything is hidden except for our basket so we can perform undercut analysis on it.

Click the undercut analysis icon.



63. Step 63

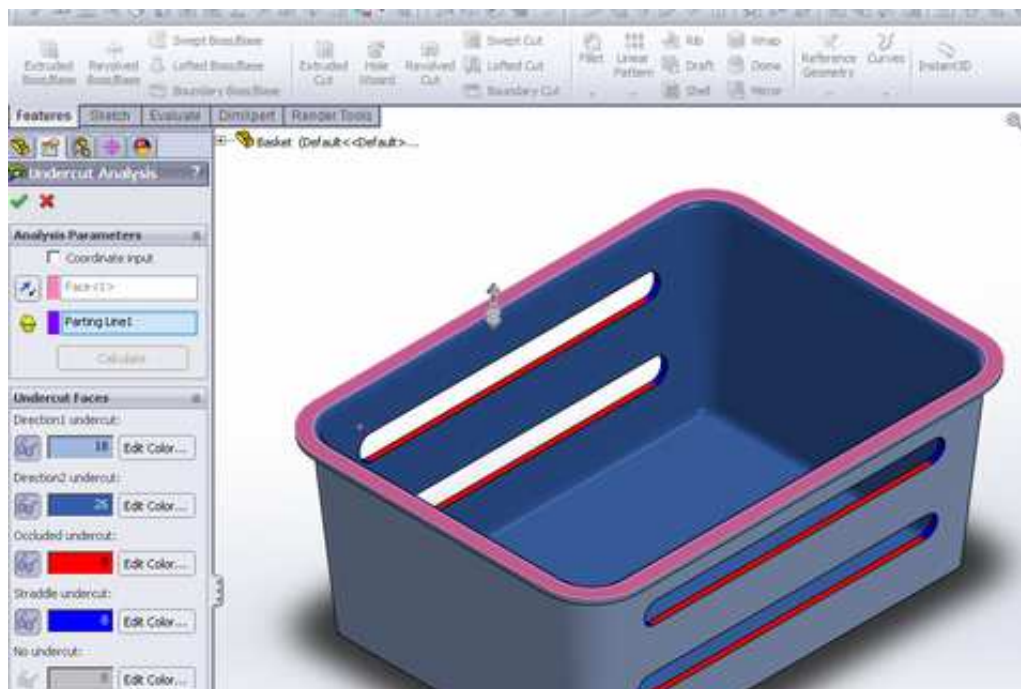
The direction of pull is already decided and we just have to click the green check.



64. Step 64

like in the draft analysis, different faces become color coded. The red faces are the ones we need to create side cores to mold.

Click the check or x to exit the undercut analysis colors and exit.



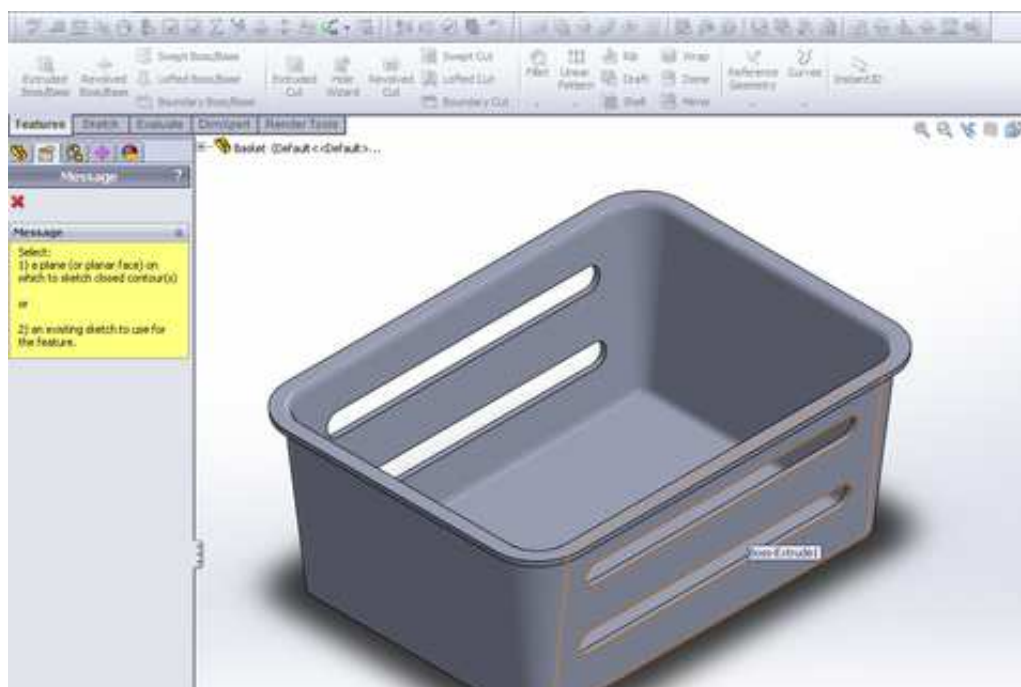
65. Step 65

To create the cores for the basket cutouts, click the core icon.



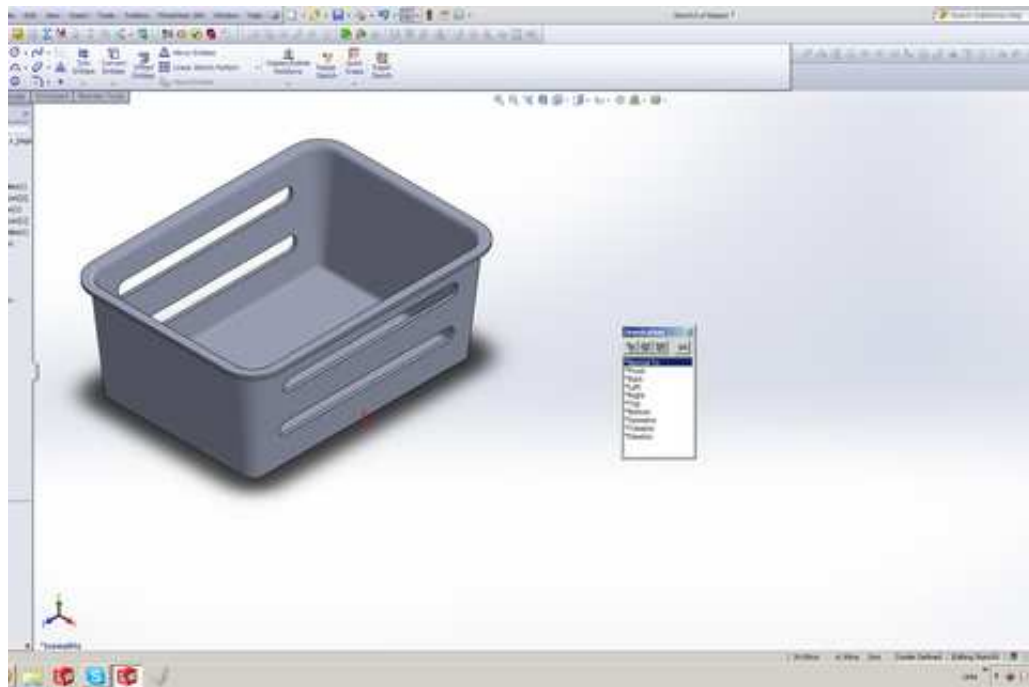
66. Step 66

Select the outer face of the basket with the cuts in it.



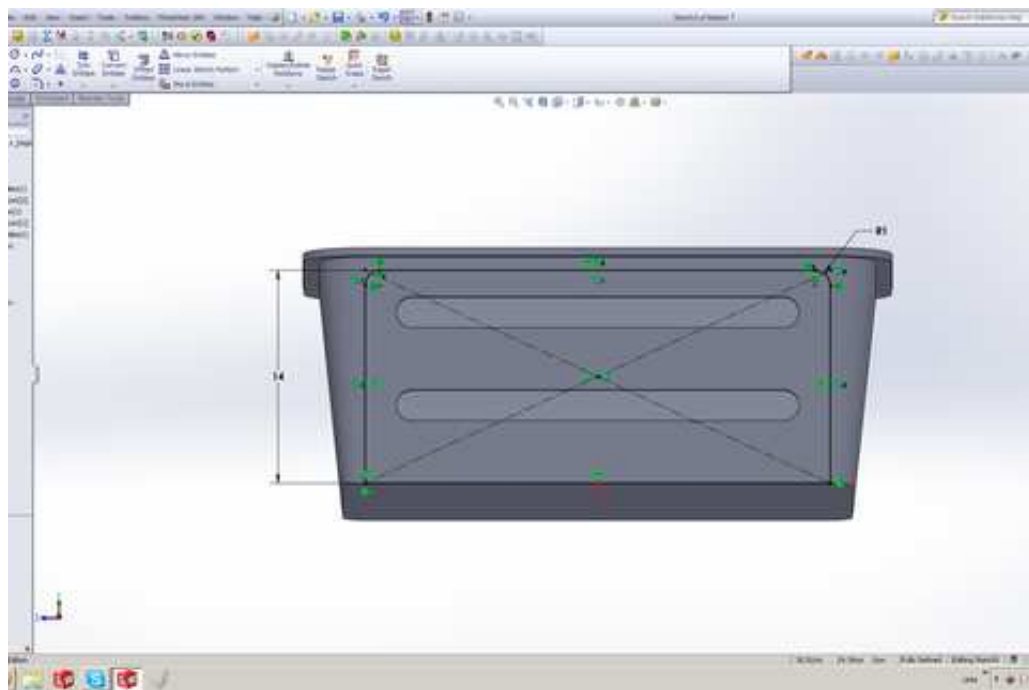
67. Step 67

This will open a sketch. I find it easier to sketch with normal to the sketch plane.



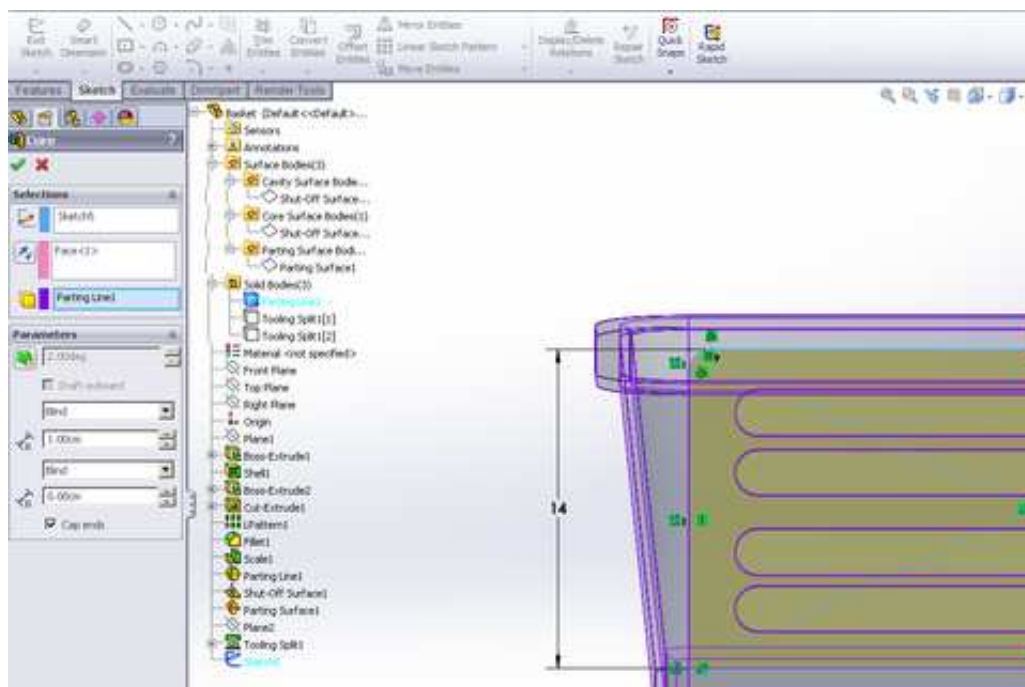
68. Step 68

create this sketch.



69. Step 69

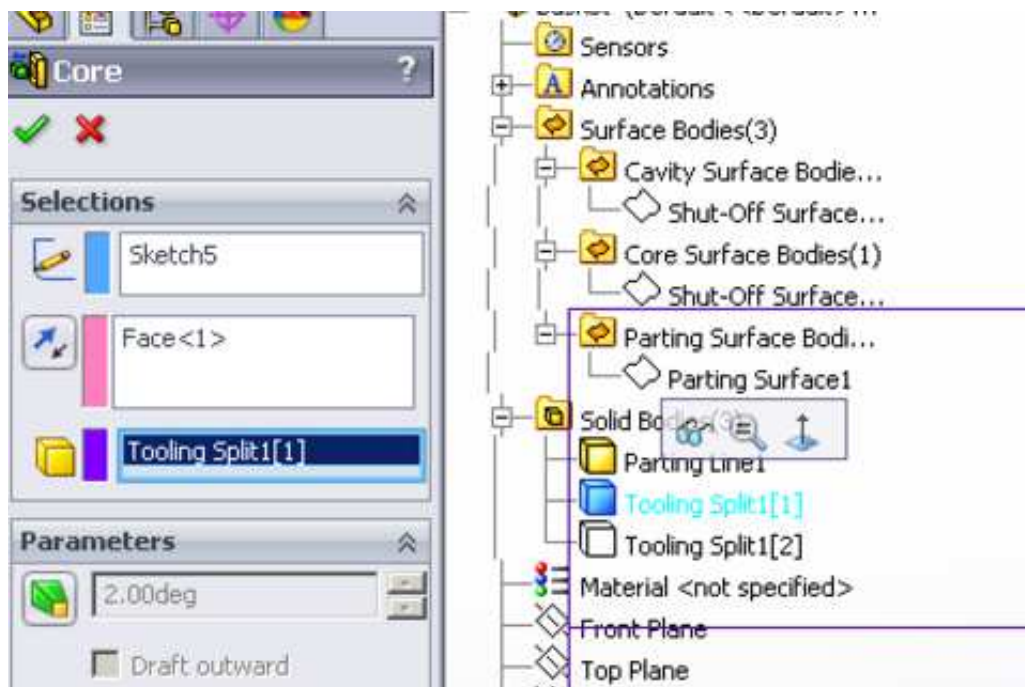
When you exit the sketch, the core property manager will appear.



70. Step 70

change the core/cavity body to tooling split one

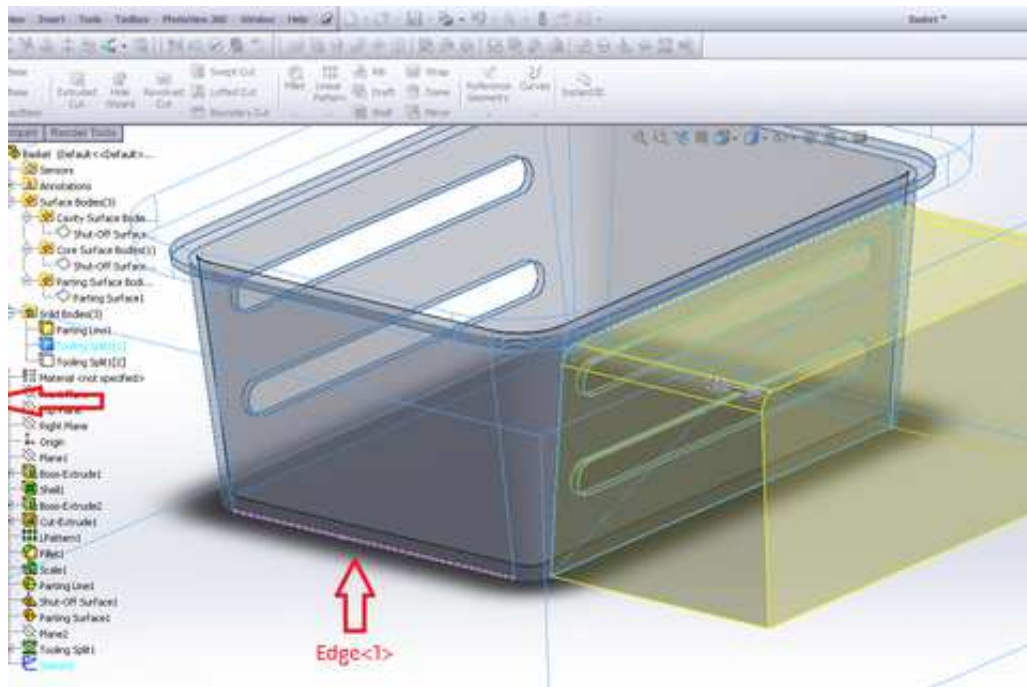
See picture



71. Step 71

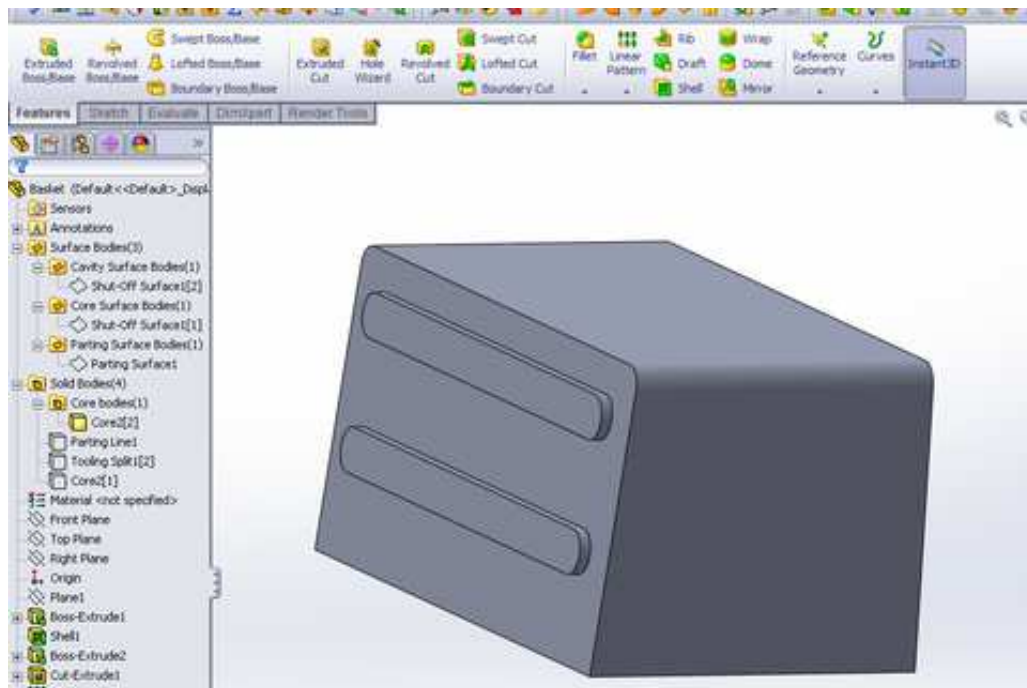
in the direction field select a line in direction of pull of the core.

select though all for direction 1 and un-check the cap ends box.



72. Step 72

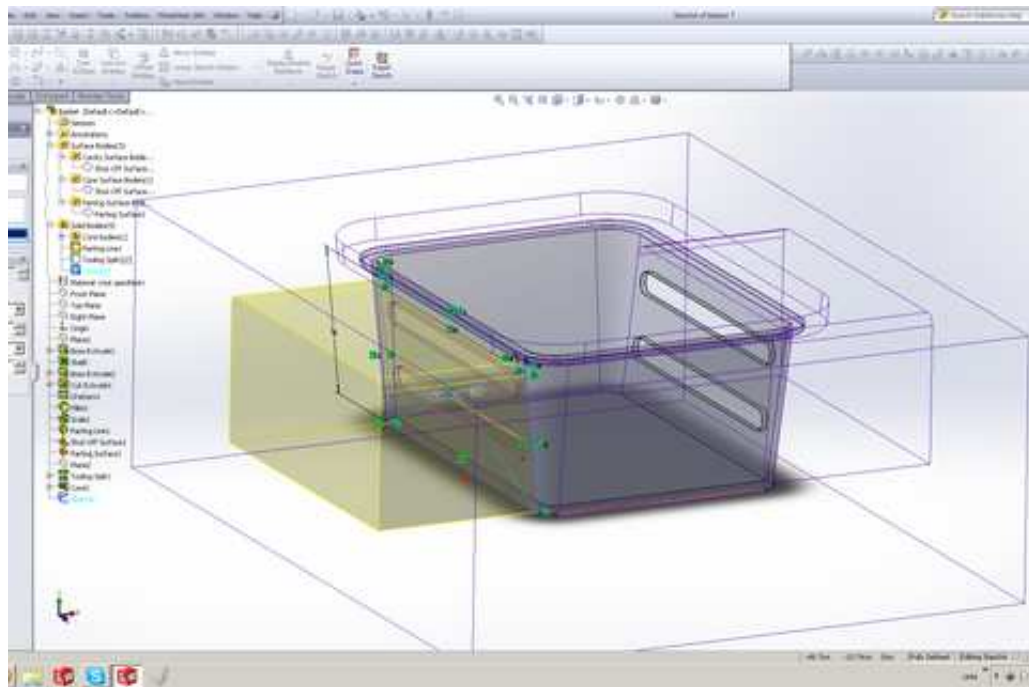
when you click the green check your core will be created.



73. Step 73

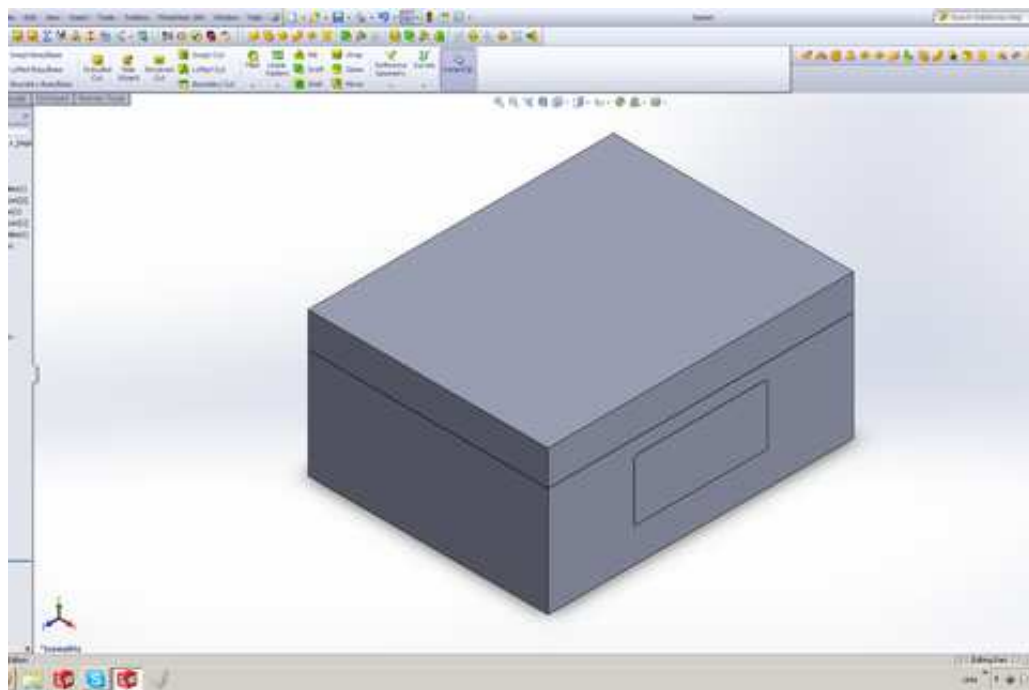
Now repeat steps 65 to 72 and reference this picture to create the core on the other side.

(some of the direction will be goofy if you try to do it exactly as done in 65 - 72 so look at the settings below.)



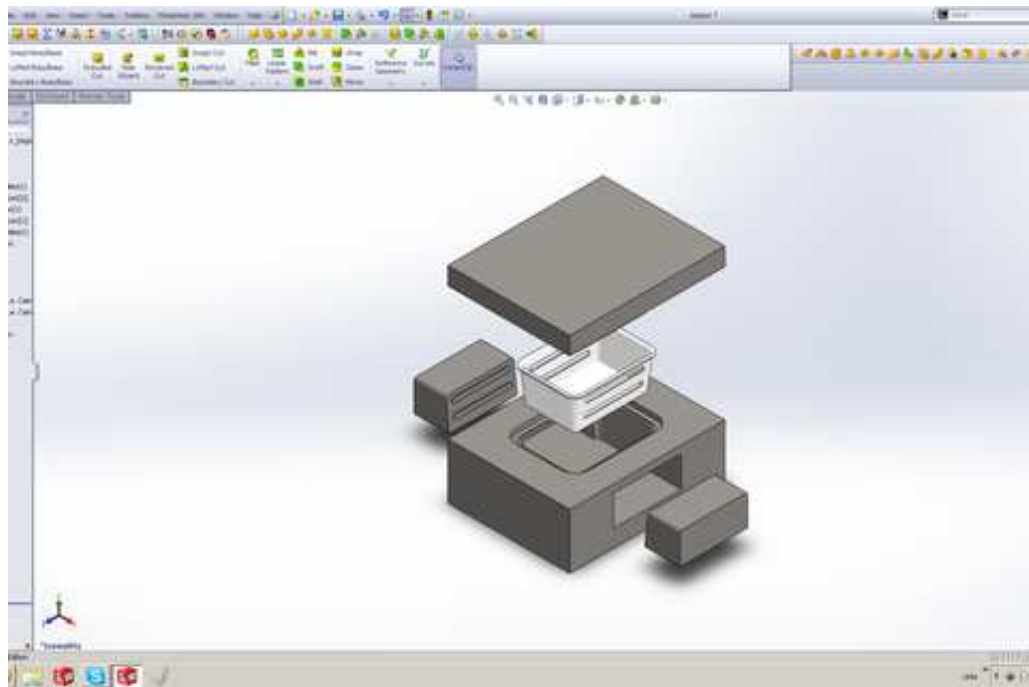
74. Step 74

Here is the completed mold parts.



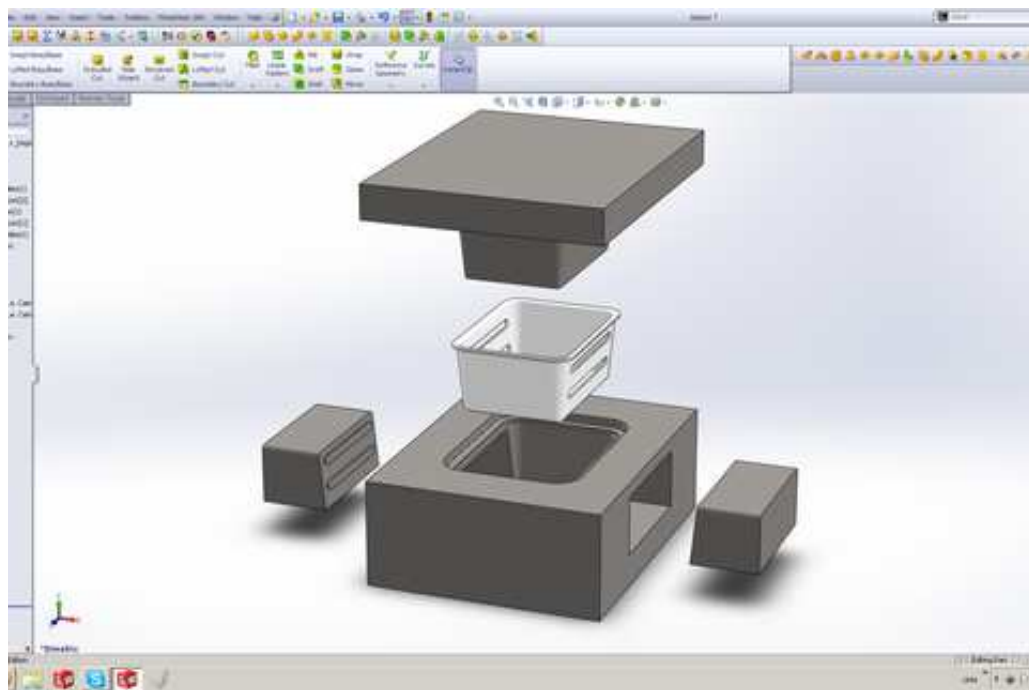
75. Step 75

I moved the individual solid bodies apart like an exploded view so you can see the different parts.



76. Step 76

A different angle



77. Step 77

Here is some of the great FREE stuff sent to me from www.protolabs.com



[It worked \(27\)](#)

[Not helpful \(3\)](#)

[Comment \(3\)](#)

1.



[barbad](#) [Report spam](#) over 1 year ago

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2.



[Abdul Haseeb Malik](#) [Report spam](#) 9 months ago

thanks

3.



[Hunt GE](#) [Report spam](#) 3 months ago

good

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[anhquoc38](#) over 1 year ago

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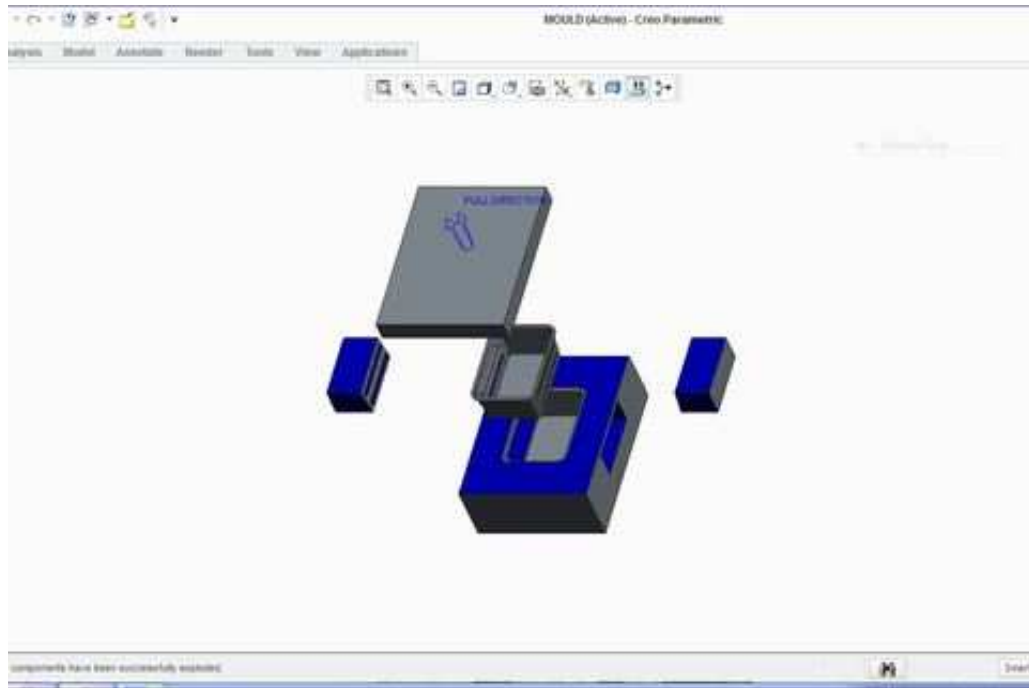
It worked ^^

I made it in Creo Parametric 2.0

Same to this tutorial.

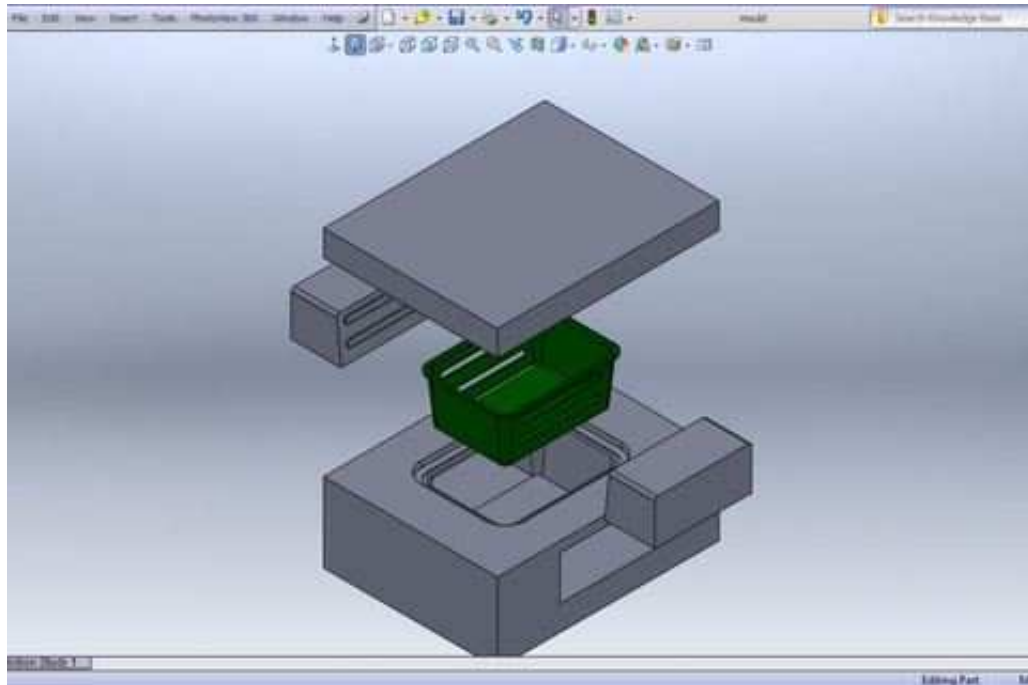
1. Step 1

I made it in Creo Parametric 2.0



2. Step 2

I made it in Solidworks 2012



[It worked \(5\)](#)

[Not helpful \(2\)](#)

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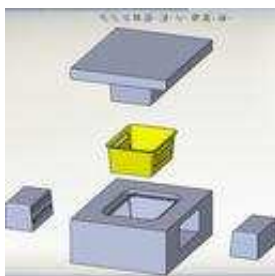


[Dima Brahim](#) 10 months ago

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it works thx

Added images



[It worked \(2\)](#)

[Not helpful \(0\)](#)

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[William](#) over 1 year ago

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Great tut Stephen even if it is for SW LoL'ss
but very helpful and useful for many others to learn more about SW.
:))))

[It worked \(1\)](#)

[Not helpful \(0\)](#)

[Comment \(2\)](#)



1.

[Stephen Nyberg](#) [Report spam](#) over 1 year ago

Ha ha Thanks i guess ;) Do you ride motorcycles? Because if you did i bet you would be a Harley guy



2.

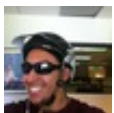
[William](#) [Report spam](#) over 1 year ago

LoL'ss Stephen, no I do not ride a motorcycle as if I did I would be dead as I would need to push it to its limit all the time when driving it.

:(

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[Alan](#) 3 months ago

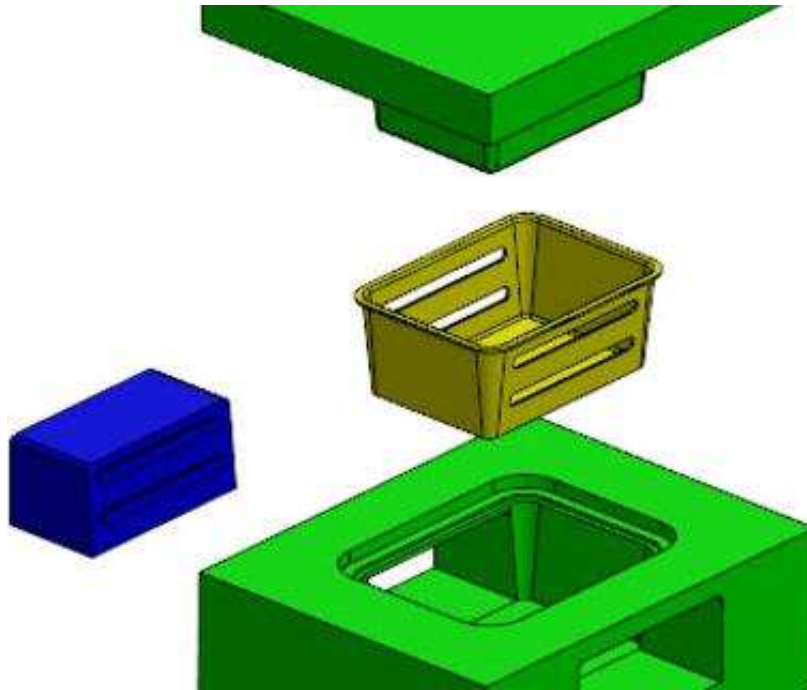
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Created in SW2014 SP4.0

Great tutorial!!

My question is how or where will the mold be injected?

1. Step 1



[It worked \(1\)](#)

[Not helpful \(0\)](#)

[Comment \(1\)](#)



1.

[Stephen Nyberg](#) [Report spam](#) 3 months ago

Nice glad you liked it. This was to help learn how to use the tools in SW, not make an actual mold.

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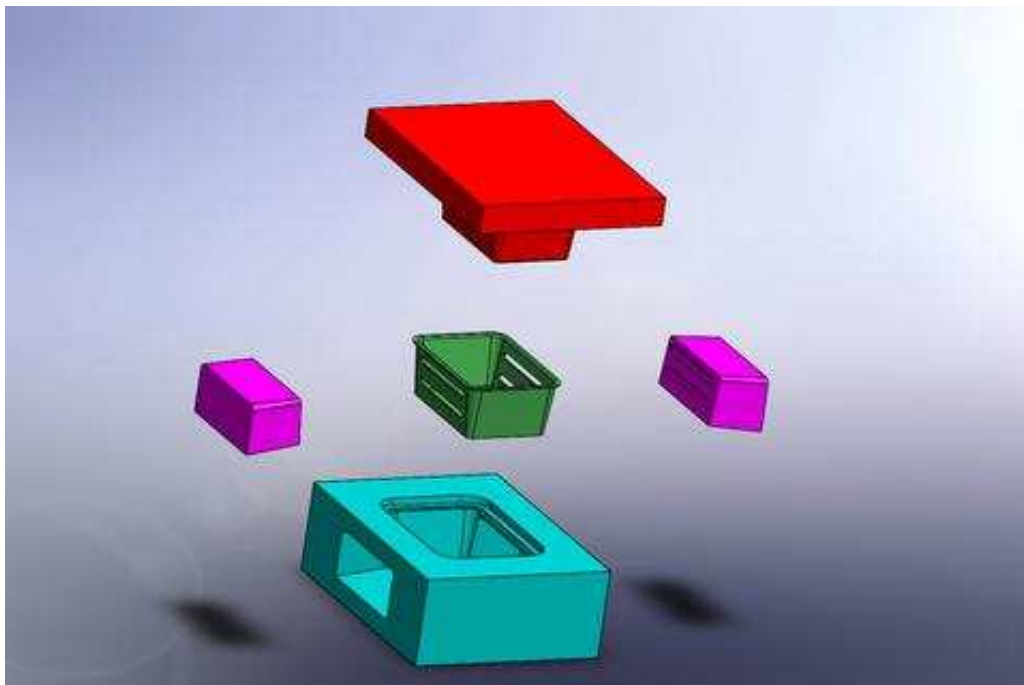


[fouad alsham](#) about 1 month ago

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i making thes

1. Step 1



[It worked \(1\)](#)

[Not helpful \(0\)](#)

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[Damjan](#) over 1 year ago

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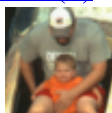
Hello,

is there a free ebook for Injection molding for dummies we can get? I am from Europe so sadly i cannot order the book from the site you mention.

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Comment \(2\)](#)



1.

[Stephen Nyberg](#) [Report spam](#) over 1 year ago

That book is by ProtoLabs and they sent it to me for free. I am not sure if they have anything out of the US. You would have to contact them. Sorry.

Add a comment

[Post comment](#)

• [Steen Winther](#) over 1 year ago

- [Report spam](#)

Nice one Stephen. As much as I love SolidWorks, I have to admit that the mold tools are very basic to the point of being useless. Matt Lombard writes, in one of his brilliant books, that they are great for Frisbees and dinner plates but useless for more complex objects, and this is exactly right according to my experience.

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Add comment](#)

[Post comment](#)

• [eng tohamy](#) over 1 year ago

- [Report spam](#)

thanks

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Add comment](#)

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• [Daniel Flores](#) over 1 year ago

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i may have done it slightly differently.

make the sides 7-8 degrees, and use sliding shutoff between the mold halves to form the holes in the sides.

this would eliminate using cams in the mold to form the slots...thus reducing the cost of manufacturing.

will post a tutorial in the next few days

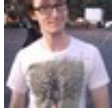
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[Not helpful \(0\)](#)

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[Noel Stalker](#) over 1 year ago

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sorry but I'm a total Solidworks noob. Where is the options button in step 3?

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Comment \(1\)](#)



1.

[Stephen Nyberg](#) [Report spam](#) over 1 year ago

If you click on the pictur you will see the button circled in red and marked with a one
You can also get to the same menu i think under "tools" and then near the bottom is
"options" Hope this helps

On the tutorial above you may need to click the immagine to see it all

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[محمد سعيد](#) over 1 year ago

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good job

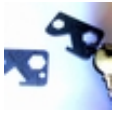
[It worked \(0\)](#)

[Not helpful \(0\)](#)

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• [Simonr](#) over 1 year ago

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Thanks Stephen - this was one of the biggest holes in my Solidworks knowledge! I've made a few molds - but done essentially the same thing manually. This is way easier!

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Comment \(1\)](#)



1.

[Stephen Nyberg](#) [Report spam](#) over 1 year ago

Glad i could help out I also liked your comment in the challenge about the "gold star" award



• [reza](#) over 1 year ago

- [Report spam](#)

tanks

[It worked \(0\)](#)

[Not helpful \(0\)](#)

[Add comment](#)



• [syf](#) over 1 year ago

- [Report spam](#)

GOODMAN

[It worked \(0\)](#)

[Not helpful \(0\)](#)

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[Zaini](#) 10 months ago

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nice job

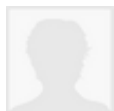
[It worked \(0\)](#)

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[saad bouta](#) 10 months ago

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nice job, thnak s

[It worked \(0\)](#)

[Not helpful \(0\)](#)

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[hasri](#) 5 months ago

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coll thanks mate's....

[It worked \(0\)](#)

[Not helpful \(0\)](#)

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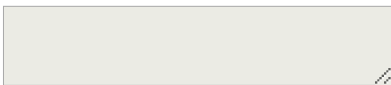
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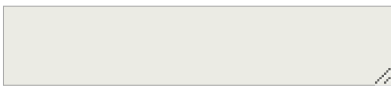


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2. Step



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