

When modeling any part you are likely to be working to certain parameters which can be used to create construction geometry in your model. In the case of this remote control unit lets assume that the design specification states the part should be no longer than 150. Now lets use that information to define two datum planes. Choose INSERT > MODEL DATUM > PLANE and click on the RIGHT datum in the graphics window. The Offset option is set automatically in the dialog box so type in a value of 150. In the Properties tab type a name of ENDLINE and click OK. Repeat this making a similar datum called MIDLINE at a distance of 75. That has set up the reference geometry for us to use.

We are now going to design the outside shape of the remote. As you can see from the picture this is a complex shape and the simple EXTRUDE and REVOLVE commands would be totally inadequate. We are going to use a command we have already introduced VARIABLE SECTION SWEEP but use it to its full capabilities.

Sketching with Splines

You may remember this command relies on existing curves so we need to draw some curves now. Like many complex shapes lines and arcs aren't suitable for the shapes we want – we will use a free form curve known as a spline.

Choose INSERT > MODEL DATUM > SKETCHED CURVE and choose FRONT as the sketch plane. On entering sketch mode click on the

ENDLINE datum as an additional datum. The  is used to create splines. Choose it now and have a practice – it takes a little getting used to. Each click of the mouse defines a point on the curve and ProEngineer smoothly interpolates between these points. Click the mouse button to

finish drawing a spline. You can then use the selection tool  to edit the curve by dragging any of the control points.

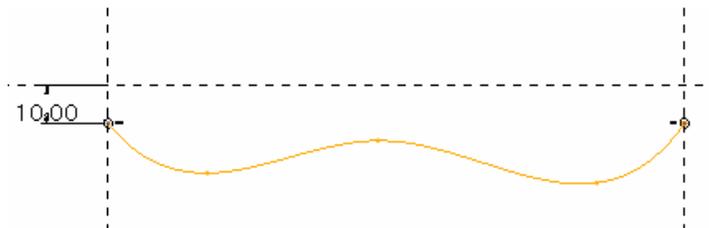


Figure 1 : First control Spline

Once you have got the hang of drawing with splines draw the curve shown in Figure 1. Note it has 5 control points and the first and last points lie on the references and are horizontally inline. Exit sketcher.

Repeat the previous command and draw a second, separate curve. This one is just a simple horizontal line aligned to all references as shown in Figure 2.

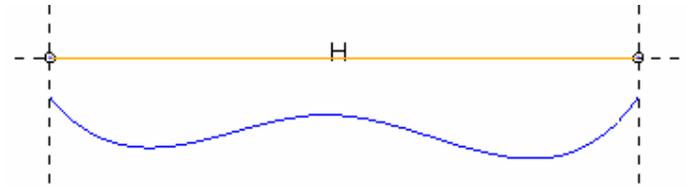


Figure 2 : Second Control Curve - Straight Line

These first two curves define the shape of the remote when viewed from the front. Now we will draw two curves to control the shape when viewed from above. Draw another datum curve using the TOP datum as the sketch plane aligning the ends of the curve as shown in .

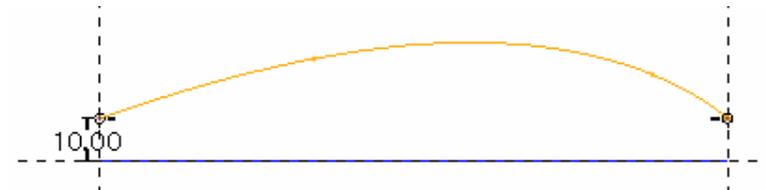


Figure 3 : First Top Spline

The fourth and final curve is identical to the last one so simply click on the last curve in the browser window then choose EDIT > MIRROR and pick the FRONT datum as the mirror plane. You should now have 4 curves and are ready to create the solid.

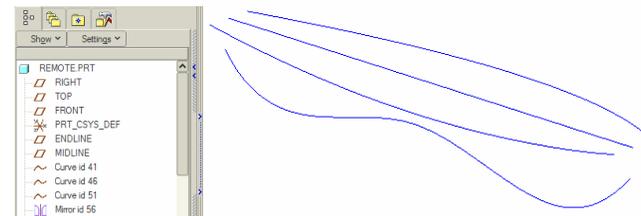


Figure 4 : Four Curves Defined

Sweeping

To make the solid choose INSERT > VARIABLE SECTION SWEEP and click on the straight line curve FIRST (it will be called origin) then the other three curves. Choose the Sweep As Solid icon then enter sketch mode where you will draw the cross-section of the sweep. You should see two references passing through the end of the origin curve and if you look carefully a reference has been added to the end of each of the four curves – shown as a small cross. Draw the section shown in Figure 5 locking on to these references.

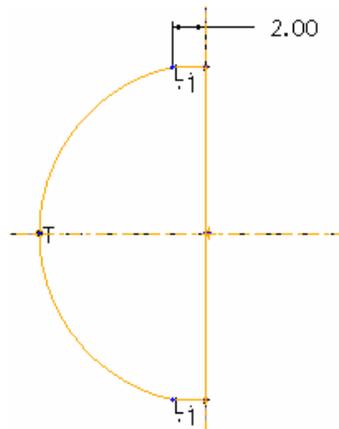


Figure 5 : Sweep Cross Section

After leaving the sketcher you should already see a prediction of the final shape in the graphics window – if you don't you have done something wrong. Check you have selected the curves in the correct order and drawn the correct section.

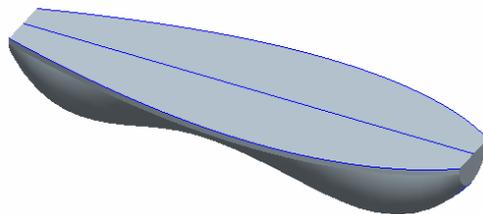


Figure 6 : The Sweep

To make the flat ends of the sweep more interesting we will use an extrusion to cut them. You will need to create two separate cuts using the TOP datum as the sketch plane. The sketches for these are shown in Figure 7. **They must be drawn as two separate cuts.**

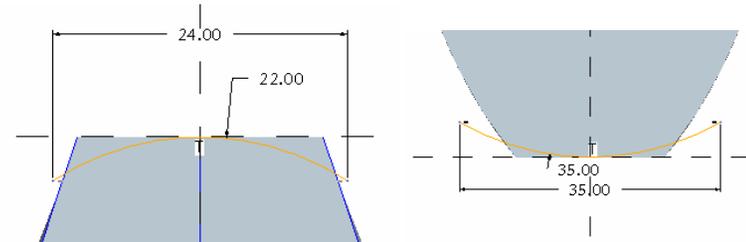


Figure 7 : Separate End Cuts

Blending

Don't try this now but this is not the only way of creating such a shape. An alternative which might be more appropriate in some circumstances is blending. With blending you draw (or select) several cross section curves then create (using INSERT > BLEND) a solid which 'morphs' between these.

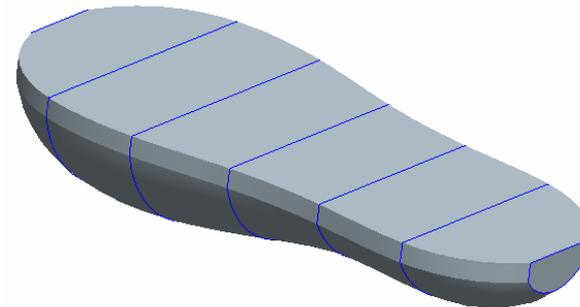


Figure 8 : A Blend

Try this in your own time in a different part file.

Cut Reversal

Next step is to add a battery compartment. Although this is a simple shape we will use it to illustrate a useful technique. Start the extrusion like all others selecting FRONT as the sketch plane and drawing the simple shape in Figure 9. Notice the extra reference that has been added to the bottom edge of the sweep. Exit sketcher.

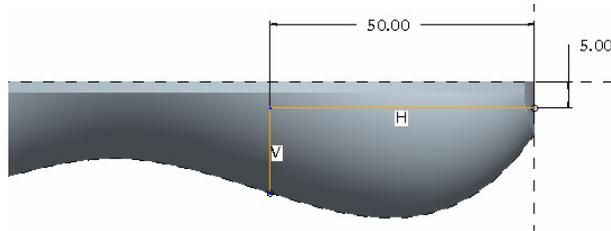


Figure 9 : Battery Compartment

This cut needs to go right through the sweep in both directions. The correct way to achieve this is to click on the Options menu in the dashboard and choose Through All in both the Side 1 and Side 2 fields. Now click on

the preview button  in the dashboard. You should see one of the

shapes in Figure 10. Click on  again then click on the second

 button in the dashboard to reverse the material to be removed by the cut. Preview and you should see the other shape in Figure 10. One of these shapes is the start of the remote control and the other is the start of the battery cover which will exactly match the remote. So finish the extrusion ensuring you have the correct side to make the main body of the remote. If you now choose FILE > SAVE A COPY and type the name BatteryCover in the New Name field you will have a copy of the current model saved. Later we can go back to this second model and EDIT DEFINITION on the last feature (the cut) and reverse its direction to start to define the battery cover in the sure knowledge that they will exactly match each other. Two models for the price of one!



Figure 10 : Reversing a Cut

Now we will make two screw holes at the opposite end to the battery compartment to join the parts of the remote together. First create a new datum plane Offset from the RIGHT datum by 30 and call it HOLES. Make a revolve feature then draw the sketch in Figure 11 on this datum. Exit sketcher and choose the Remove material button to make the first hole. The second hole is identical so choose EDIT > FEATURE OPERATIONS > COPY > MIRROR | DEPENDENT | DONE > (select the cut feature) > DONE > (pick the FRONT plane) > DONE to make a copy on the opposite side of FRONT datum.

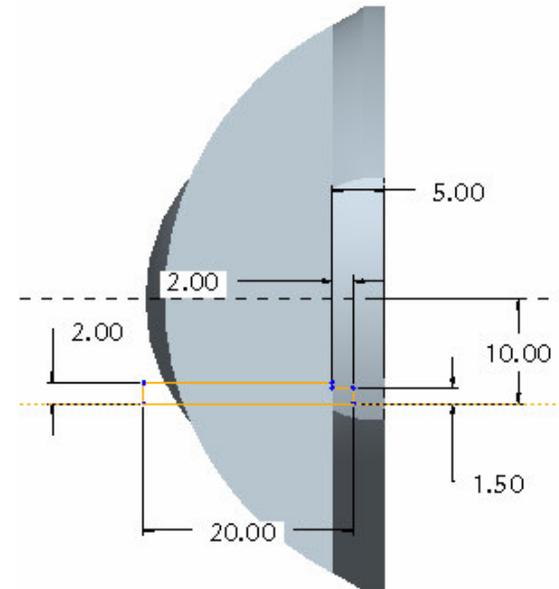


Figure 11 : Screw Holes Revolved Sketch

Now it is time to hollow out the remote control using the INSERT > SHELL function. Choose a thickness of 1. Which surfaces should be removed from the shell? Obviously the large flat surface on the top of the remote but the holes also need to be open. Select the circular surfaces at the bottom of both holes too (hold the CTRL key to select several surfaces).

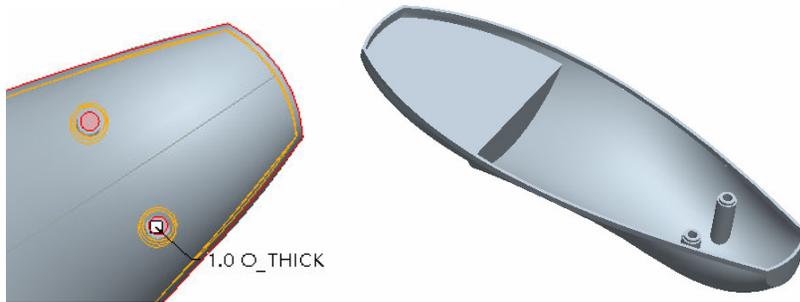


Figure 12 : Shell Creation

The surfaces of the holes look a little fragile – they need some supports to ensure they don't get broken off. We will add a thin web of material between the hole surfaces and the outside wall of the shell. You might think this is a simple extruded protrusion but it is easy to make an invalid model if you do that. The correct term is a non-manifold model because the extrusion just touches the hole surface tangentially – it does not mate with the surface correctly – and there is a gap.

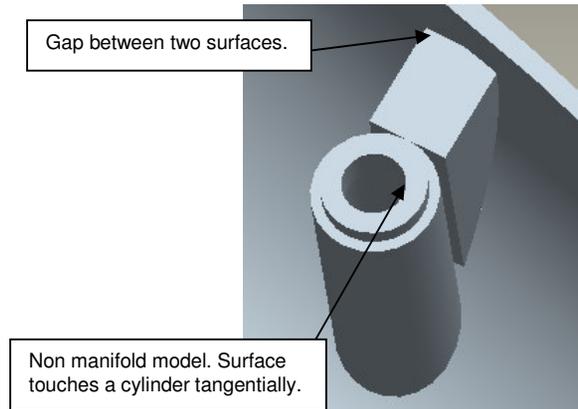


Figure 13 : Invalid Extrusion

ProEngineer has a special function to avoid this problem. It is like an intelligent extrusion command that automatically mates to adjoining surfaces correctly – it's called a rib.

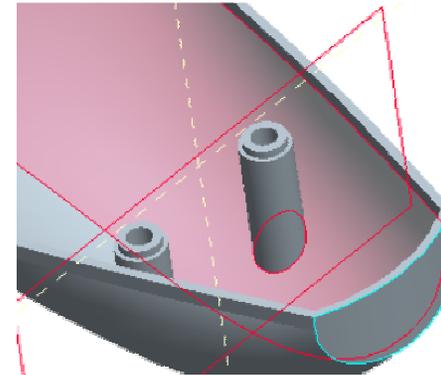


Figure 14 : Surfaces for Intersection

Before making the rib we need to prepare some geometry. The rib command requires you to draw a shape to enclose the material to be added. So we need a line which touches the outside of the hole surfaces and also touches the inside of the shell. The hole surfaces have a 'true' silhouette so you can easily create a reference for that and draw to that reference. But the 'problem' is the inside of the shell – since that is a freeform surface it does not have a silhouette – we need to make one. The line we need to reference is a curve along the intersection between the HOLES datum and the inside of the shell. To create this curve select on internal surface of shell shown in Figure 14. The first time you pick this surface you actually select the whole shell feature – we only want one surface of the shell. Pick again in the same place and Pro Engineer will 'look inside' the shell and find the surface (depending on how you drew the original section curve for the body - Figure 5 - you will either select the whole internal surface or just half of it). Next with the CTRL key held pick the HOLES datum plane. The geometry is selected so now choose EDIT > INTERSECTION. You should see the intersection curve created.

Now we are ready to create the rib feature. The command is INSERT > RIB – try it now.



Figure 15 : The Rib Dashboard

Go into sketch mode picking the HOLES datum you created earlier as the sketching plane. The curve you just created can be picked as a reference curve along with external surface of the holes. Draw a line between these two curves. Because the ends of this line are locked onto the references which themselves are locked onto the underlying surfaces the rib will correctly join to these surfaces.

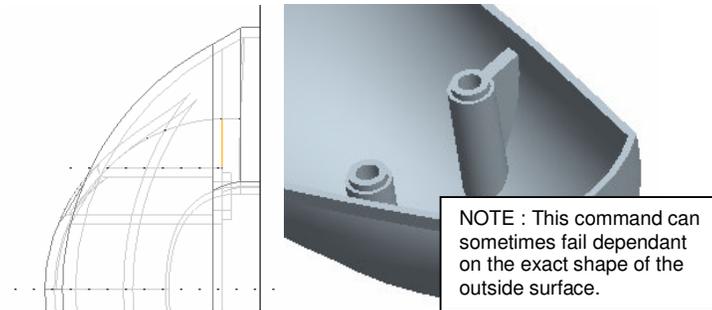


Figure 16 : The Sketch Curve and Rib

Close the sketch. Check that the arrow drawn on the curve points towards the material which you want added – if it doesn't use the  icon in the references menu to change it. Type a thickness of 2 and end the dashboard with the green tick. Create a second rib on the opposite side using the same technique.

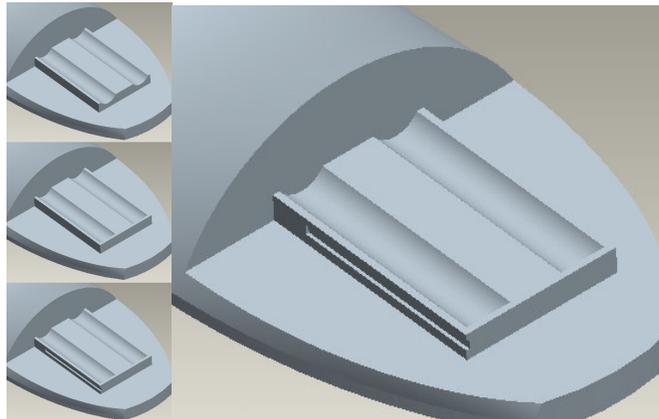


Figure 17 : Battery Holder

The next step is the battery holder. This is not complicated it is made up of two extrusions and a cut. The cut is sketched onto the side of battery holder. Rather than making a second cut on the other side you can use another command to make a copied mirror. EDIT > FEATURE OPERATIONS > COPY > MIRROR | DEPENDENT | DONE > (select the cut feature) > DONE > (pick the FRONT plane) > DONE.

Full Round

Here is a chance to demonstrate a new type of round. Up till now all rounds have been edge rounds – rounds applied to a existing edge. There are other options for rounds in ProEngineer for example the FULL ROUND. We can use this to add a round to the end of slots (Note : this could have been added by drawing the correct shape for the initial cut but then we wouldn't have had an excuse to demonstrate full rounds!). Choose INSERT > ROUND as before and select the two edges shown in Figure 18. By default you will get edge rounds on these selected edges. Click on the SETS tab in the dashboard and you will see a button called Full Round – this button is only active if you have exactly two edges selected. Click on this to change the type of round and you should see the round created.

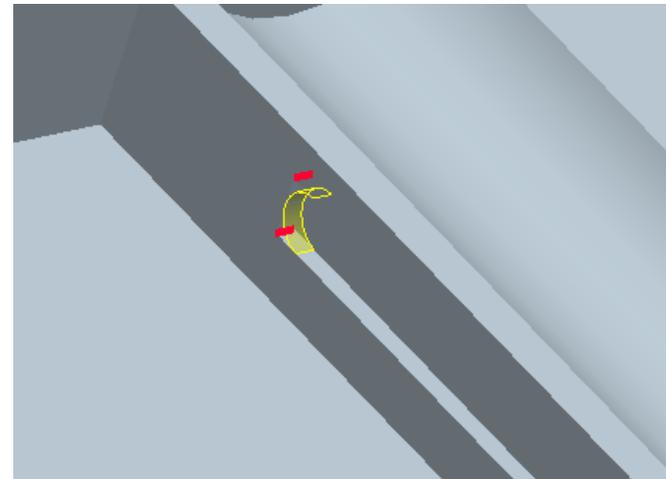


Figure 18 : Full Round

Now we want a round on the other slot to. Since for a full round you can only have two edges selected we can't select any more edges. You could

close the dashboard and repeat the procedure above but there is an alternative that allows you to group similar rounds together. In the Sets menu you should see the name Set1 – right click on this and choose Add. Set2 will be created and you can now select the two edges on the other slot creating two rounds in one command.

Using Projection Curves

Now we will add a simple logo to the remote. This is a letter S surrounded by a circle. If the surface was flat this would be a simple matter of drawing a circle and two arcs for the S then using the SWEEP command to cut away material. But the surface isn't flat so how do we draw a curve onto a non flat surface? The answer is we cant! But we can project curves onto a surface. Choose INSERT > MODEL DATUM > SKETCHED CURVE and pick the TOP datum as the sketching plane. Draw a circle and two arcs to make the S. Exit sketcher.

Now click on the curve in the browser panel and choose EDIT > PROJECT and continue at the warning message. At the project dashboard pick the External surface of the remote (depending on how you drew the original section curve for the body - Figure 5 - you will may need to select twice using the CTRL key to get the whole surface). Close the dashboard. A copy of the curve will now be sitting on the surface.

Now you have the curves you can use the INSERT > VARIABLE SECTION SWEEP command using these curves and a circular cross-section to cut the grooves in the surface. You will have to do a separate sweep for each of the two curves. If you need reminding how to do these simple sweeps refer to the section Sweep Features in the Intermediate Modelling Tutorial.

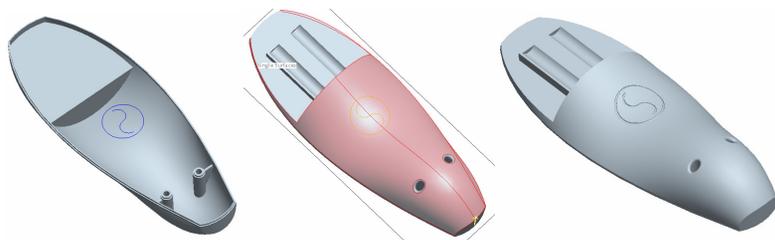


Figure 19 : Using Projection Curves

Using Offset Curves

To finish this part we will add a cut to the top edge to make a dust seal when this part is assembled with the keypad. As always there are several ways of approaching this – we will use a simple extrusion.

Choose INSERT > EXTRUDE and pick the TOP datum as the sketching plane. We will use a command to make the curve we need which was introduced in the Introduction To Modelling Tutorial. The edges of the seal will follow the outside edges of the remote. Chose the command SKETCH > EDGE > OFFSET  and in the Type dialog choose CHAIN. Now pick on an outside edge of the remote – one edge highlights. Now pick on an adjacent edge – the whole loop around the remote highlights and you choose ACCEPT in the side menu. Enter an offset distance of 0.5 – a negative value may be needed to go the opposite way to the direction arrow. A series of lines is created offset from the edge of the surface. Exit sketcher.

Choose the options to remove 1 material into the remote and that's it the model is finished. Remember, you can use the second  icon to change the material side to be removed.



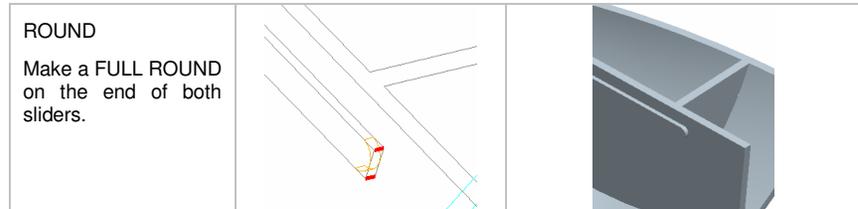
Figure 20 : Completed Remote with Dust Seal

The Battery Cover

Remember that we saved the model earlier to the name BatteryCover. Open this model now and you will see the remote at a much earlier stage of its development. We saved this so that we could easily make the cover for the battery. The last feature in the browser should be a cut. Right click

on this and choose EDIT DEFINITION. This takes you back to the dashboard with all the options set. Reverse the side of the cut to remove material by pressing the second icon. Close the dashboard and you should have the battery cover. Here are some pictures to help you finish the model. Your dimensions may vary a little from those stated – feel free to use a bit of creativity.

| | | |
|--|--|--|
| <p>REVOLVE</p> <p>Remove material for a finger grip.</p> <p>Sketch on FRONT and choose 360 degree option.</p> | | |
| <p>ROUND</p> <p>Add 4 round</p> | | |
| <p>SHELL</p> <p>Remove 2 faces and choose 1 thickness.</p> | | |
| <p>EXTRUSION</p> <p>Sketch on TOP. Mirror to make second side. Use Extrude To Next option.</p> | | |
| <p>EXTRUSION</p> <p>Make a new datum 8 away from front face of cover. Sketch on this datum. Mirror to make second side. Use Extrude To Next option.</p> | | |



That's the both halves of the model completed. In a later tutorial you will learn how to assemble these two pieces together.

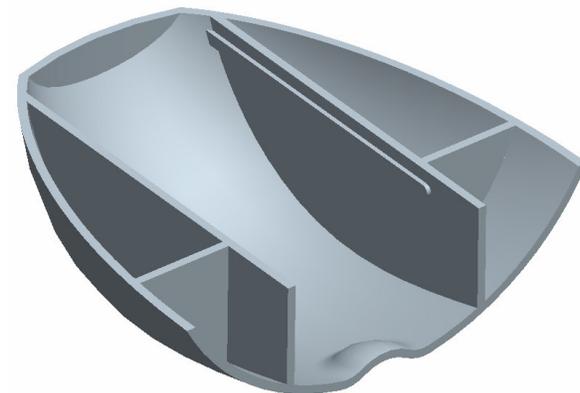


Figure 21 : The Completed Battery Cover

Review

So what should you have learnt?

- How to create complex surfaces.
- How to make thin walls with shell.
- How to create ribs.
- How to create full rounds.
- How to use projection curves.

Any problems with these? Then you should go back through the tutorial – perhaps several times – until you can complete it without any help.