

## Instructions for Import of Curves from Ath Coordinate files into Solidworks with an Excel Macro

A copy of Microsoft Excel is required to run the macro and manipulate Solidworks.

Add the following lines into an Ath Configuration file in the output section.

This will output the coordinate files at the correct scale for Solidworks set to MMGS (Millimetre gram Seconds). If your base template is set to MKS change scale to 0.001. If you are using IPS change the units to MMGS or MKS before starting or trial and error to work out the correct scale.

Output.Coords = 1

Output.Coords.Scale = 1.0

Output.Coords.NumProfiles = 4

Output.Coords.Delimiter = ,

Output.Coords.SeparateFiles = yes

Output.Coords.FileExt = txt

The profile and slice files are the ones needed.

At this stage Ath cannot properly use a space as a separator, this is planned to be fixed so the following instructions may not be necessary in the future. At that point, the Delimiter will need to be adjusted from a comma.

For now, open these files in a Text Editor to change the , separator to a space.

Notepad++ (Free) can open all the files and process them at the same time which is much quicker than opening and processing them one by one.

Open the files, for Notepad++ go the Search Menu, choose Replace. In Find What type a comma(,) and in Replace with type hit the spacebar. Then choose Replace in all Opened Documents.

Check to make sure the columns have been separated by a space if it has, choose save all.

The files are now prepared.

The macro file can be downloaded from this link:

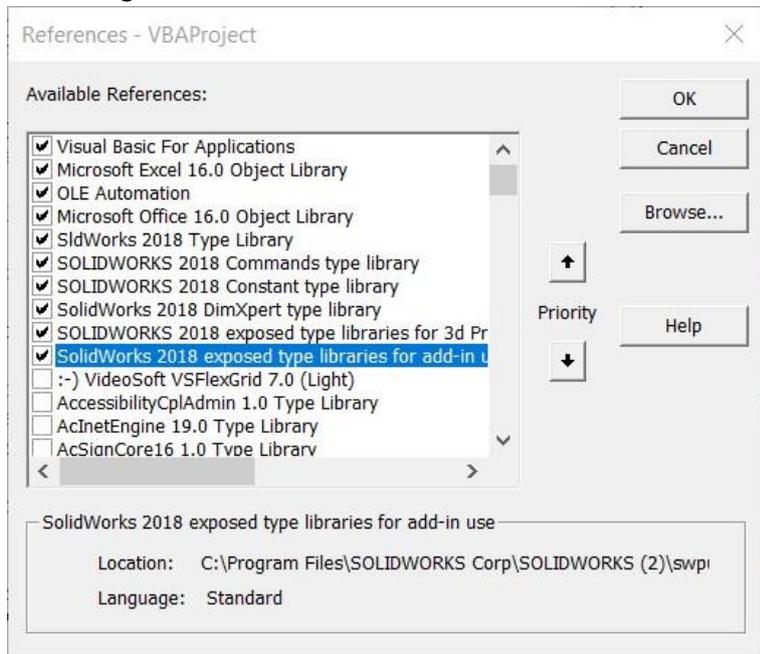
<https://www.diyaudio.com/forums/multi-way/338806-acoustic-horn-design-easy-ath4-post6358858.html>

Open the macro in Excel, be sure to enable the content when prompted or the macro will not work. Check to see if the Developer tab is shown, if it is hidden follow the instructions here to display it

<https://support.microsoft.com/en-us/office/show-the-developer-tab-e1192344-5e56-4d45-931b-e5fd9bea2d45>

In the Developer tab click the Visual Basic Button which will open the editor. Then click Tools>Reference

Check if the Solidworks references have been checked. Without them this code will fail. Should look something like this



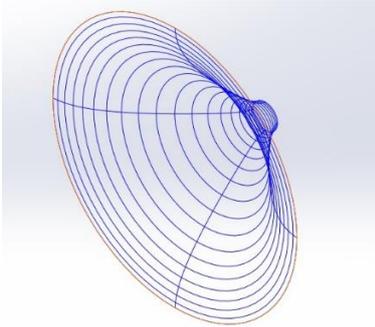
The code in the macro expects a new blank part to be open in solidworks. It can be made to open solidworks and make the part by uncommenting the three lines shown here. Just remove the ' from the front of the line. There are three modules of code and the same will need to be done to all of them.

```
Dim swApp As SolidWorks.SolidWorks
Sub main ()
    'Opens SolidWorks(R) and creates a new part:
    Set swApp = CreateObject("SldWorks.Application")
    'swApp.Visible = True
    'swApp.NewPart
    'swApp.FrameState = 1
```

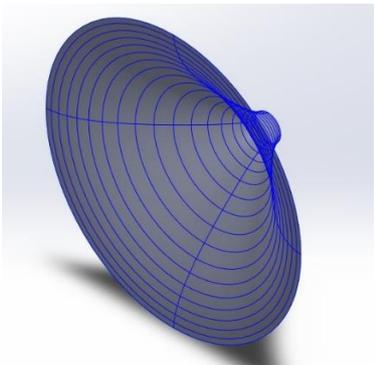
There are three buttons on Sheet 1

	A	B
1		
2		
3		
4		
5	Curves Only	
6		
7		
8		
9		
10		
11	Curves and Surface Loft	
12		
13		
14		
15		
16		
17		
18	Curves and Solid Loft	
19		
20		
21		

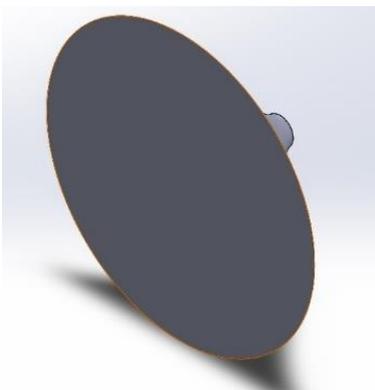
The names are self-explanatory. Click curves only to just import the slices and profiles as curves.



Curves and Surface Loft imports the curves and then lofts them to a surface.



Curves and Solid Loft imports the curves and then lofts them to a solid (Boss Base)



When a button is clicked a file dialog will open where you can navigate to where you saved the processed slice and profile files (do not select the config file). Select them all and click open. A few seconds later the part should appear in solidworks.