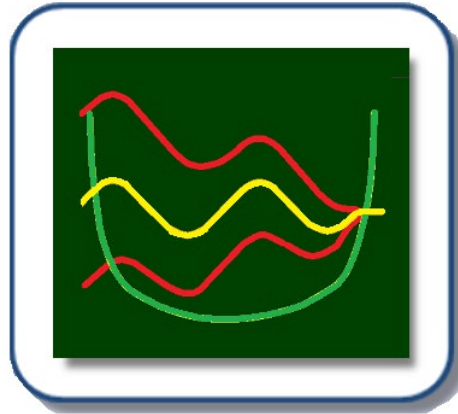
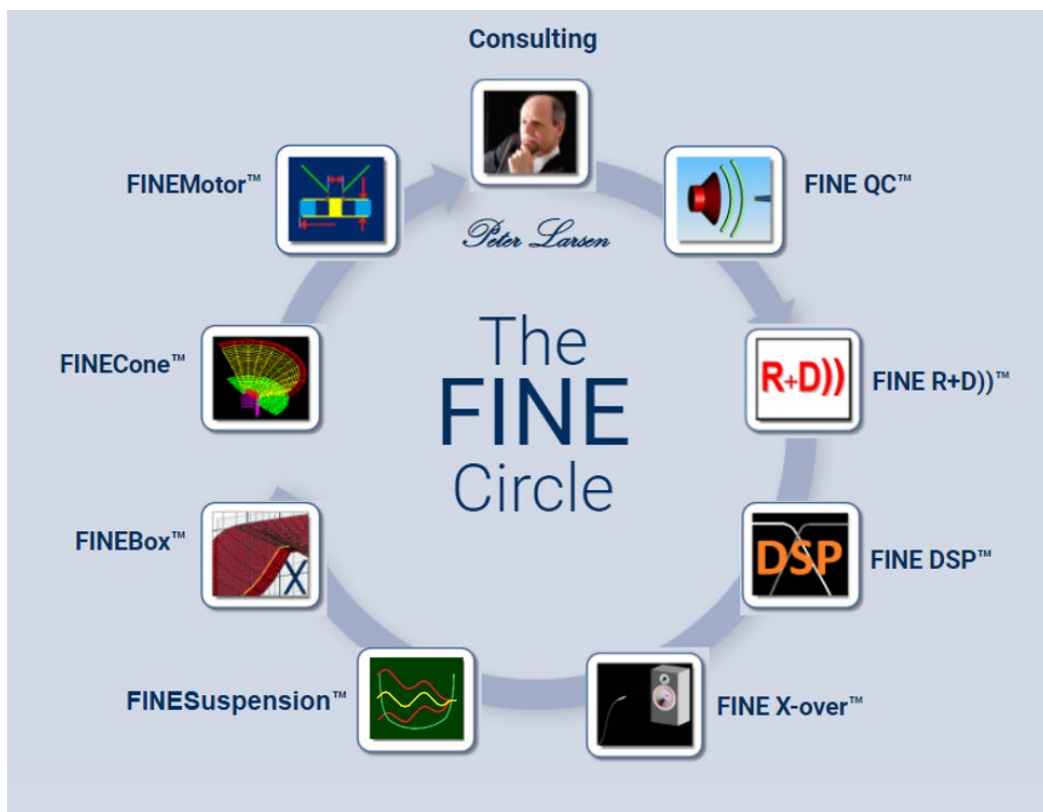


# MANUAL



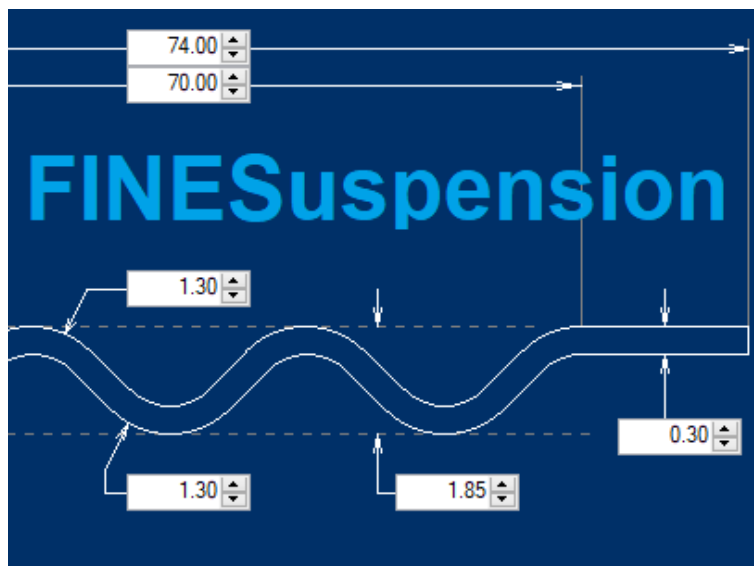
## FINESuspension™

Non-Linear FEA Spider & Surround  
Design Program

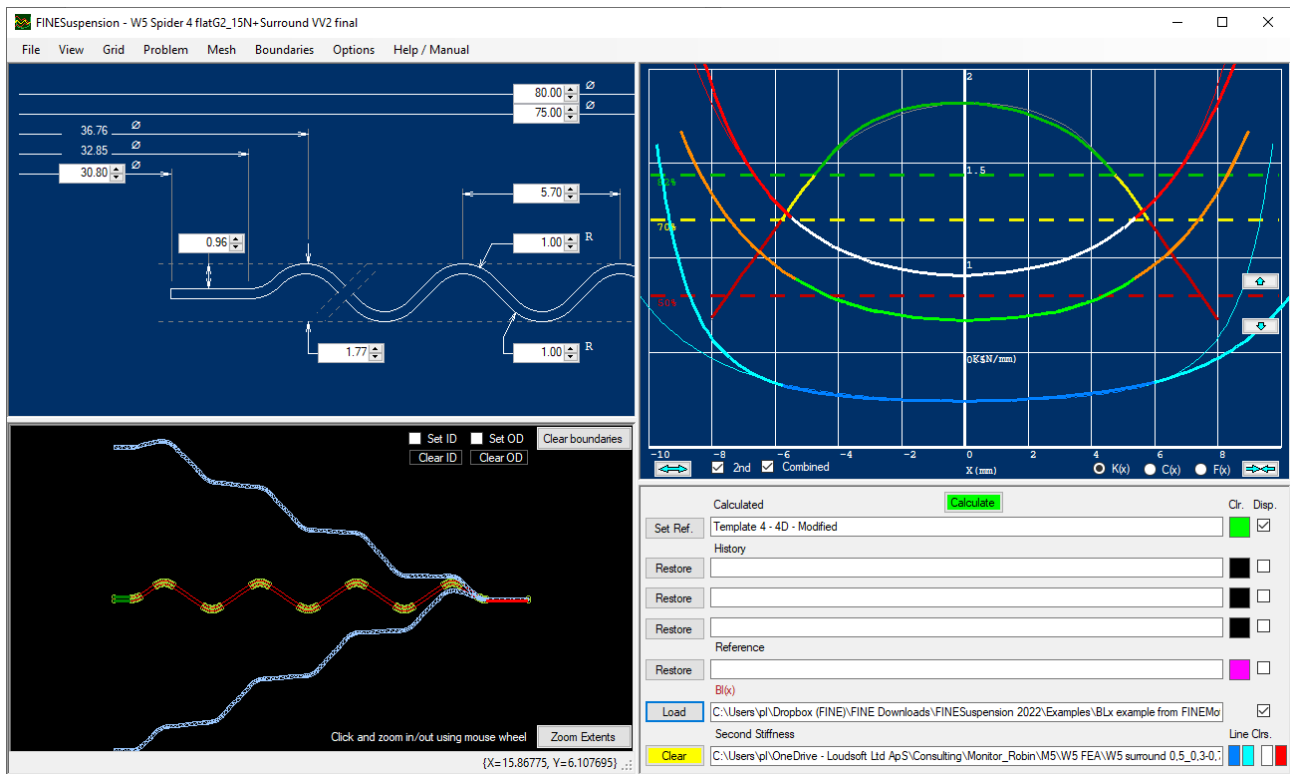


# Contents

FINESuspension™ .....	1
1. Overview.....	3
2. How to install the LOUDSOFT programs .....	4
Design Examples .....	7
2.1 Surround Type1- Design Example .....	7
2.2 Import BL(x) curve for comparison.....	12
2.3 Spider Type 4 – Design Example 2.....	14
2.4 Double Spider Example .....	19
3.1 Combined Spider + Surround .....	20
3.2 Verification with actual spider ZE32/18GZ .....	22
3.3 Geometry Input from CAD (DXF).....	24
3.4 IMPORTANT – DXF hints for avoiding calculation errors .....	25
3.5 Export Geometry in DXF format.....	26
3.6 Compliance Cms(x) .....	27
3.7 Force / Deflection.....	27
3.8 Export K(x) data.....	28
3.9 Display Options .....	29
3.10 Calculation step size.....	29
3.11 Moving Borders .....	30
3.12 Mesh.....	30
3.13 Printing .....	31
3.14 FAQ - Frequently Asked Questions .....	31

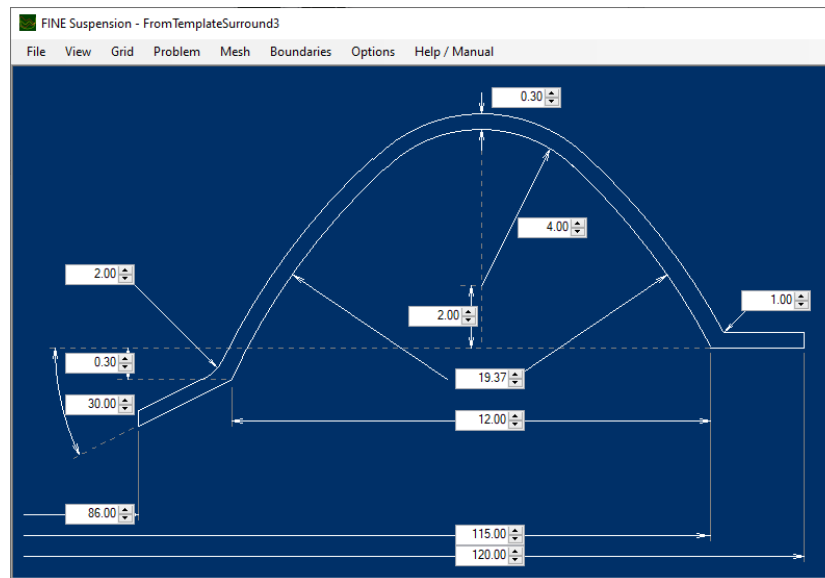


# 1. Overview



FINESuspension is Non-Linear Finite Element software which is ideal for designing the spider and surround for most loudspeaker drivers. The spider and surround curves can also be combined, see section 4.1, page 19.

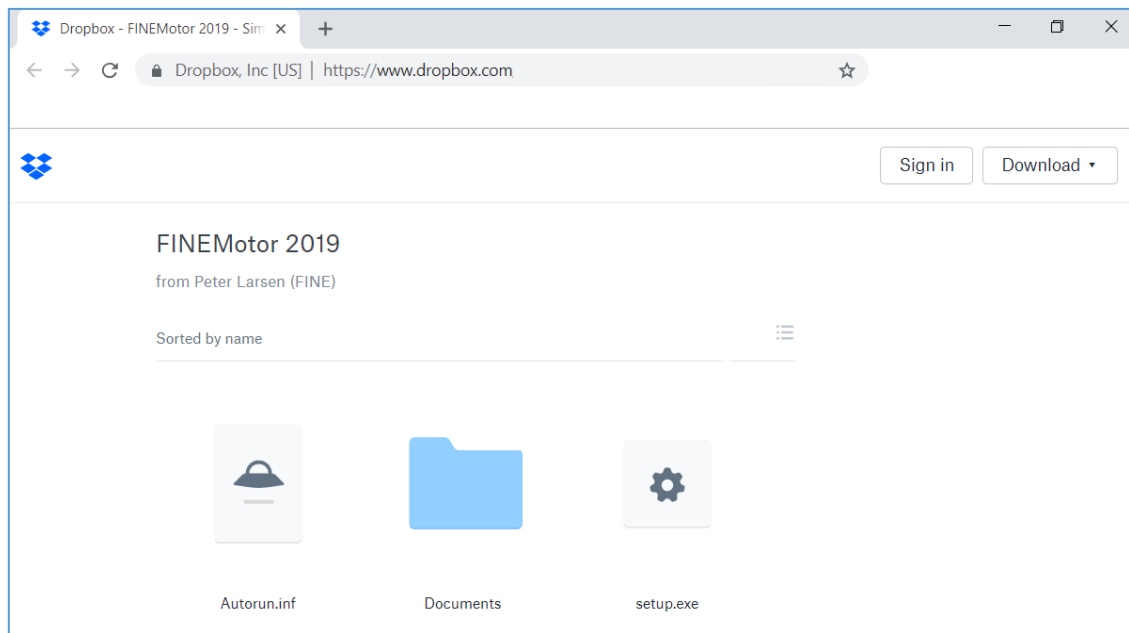
Starting from a template, the user only needs to input dimensions and select a material from a database to get started.



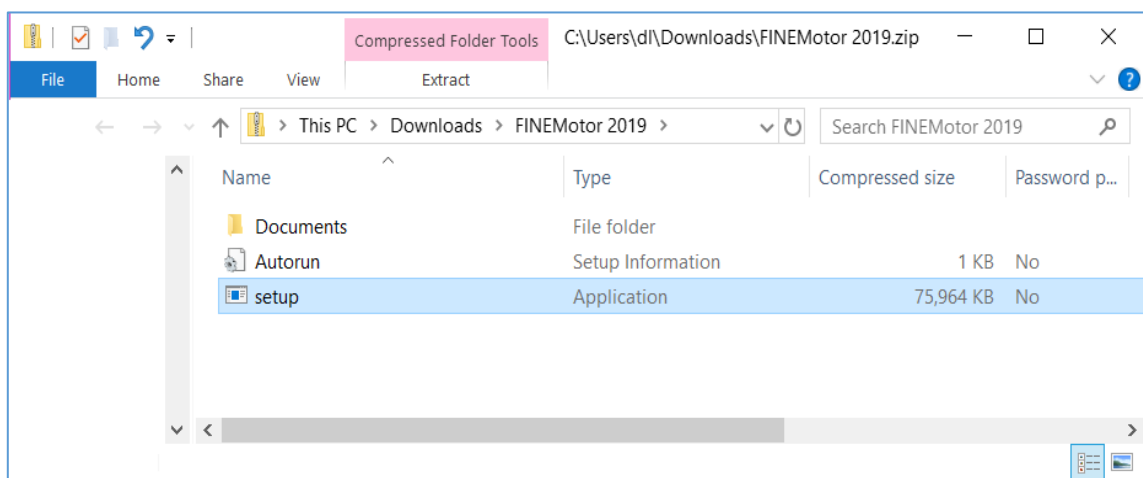
## 2. How to install the LOUDSOFT programs

**The computer must be connected to internet during installation.**

- When you click on the license link, you must download the program. Click on "Download" at the top right corner of your screen. **Make sure to save all the files from this link. Preferable on a memory stick, you might need it later, if your computer crashes. We change the links from time to time for security. If you don't have the files, you will have to pay for an upgrade in order to get a new one.**



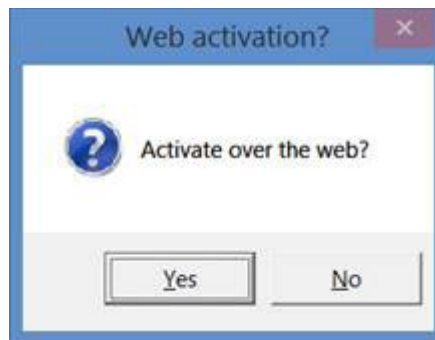
- Now find the downloaded file at the left bottom of your screen. Click on it and you will see the picture below. Click on “Setup” and follow the instructions to install the program.



- Click OK on the first dialog



- Click YES on the second dialog



- Fill in your license ID and password in next dialog. Your License ID is XXXXXXXXX, and your password is Pppppppp

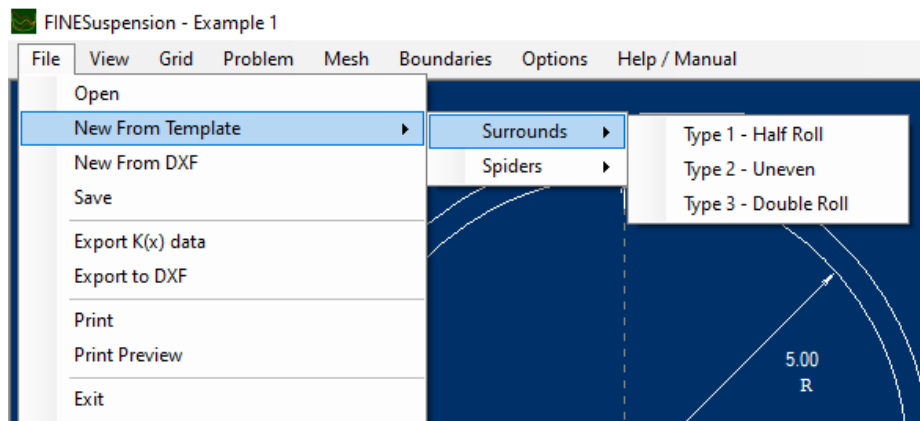


- Click OK. It is all done!

# Design Examples

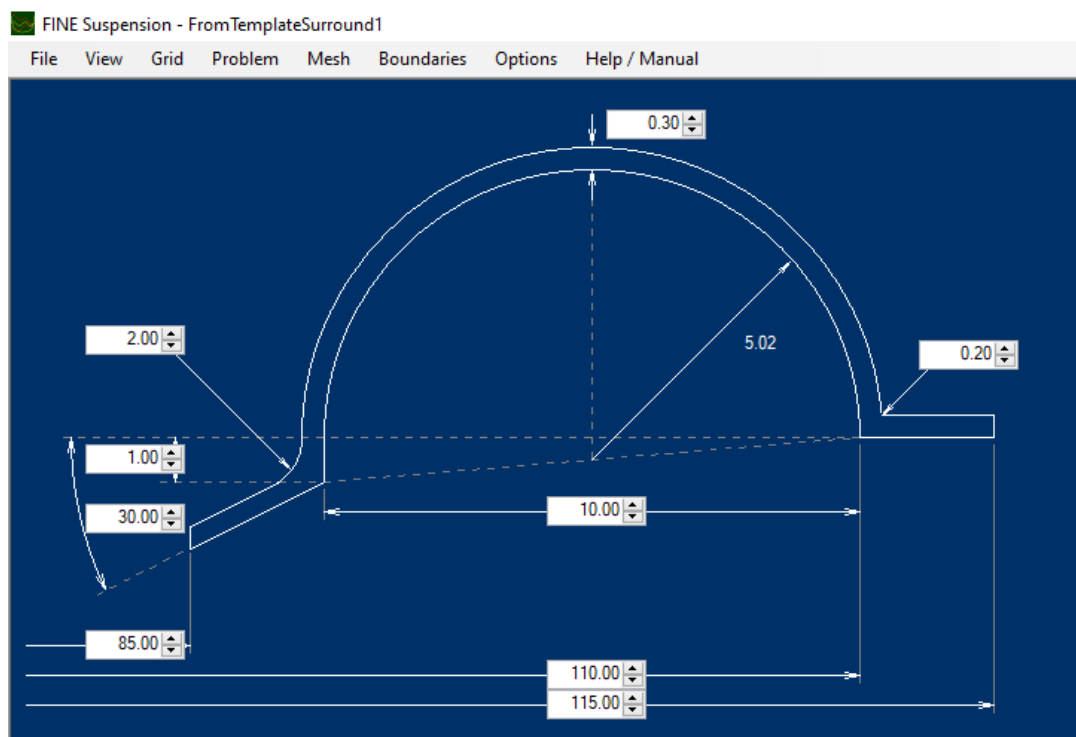
## 2.1 Surround Type1- Design Example

In this example we want to design a half-roll surround in rubber. From File we choose Type 1 template: Half Roll surround.

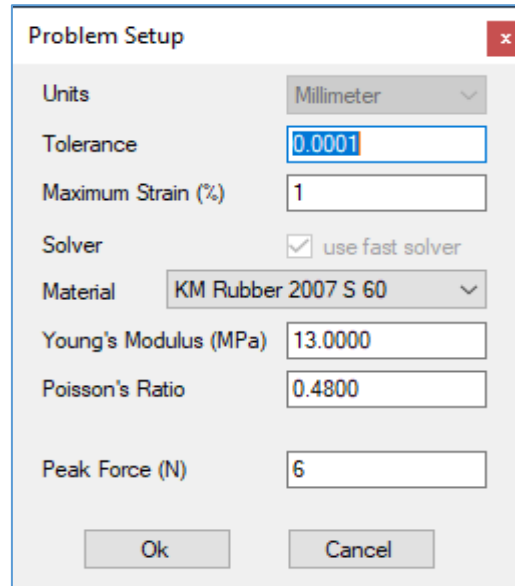


The Type 1 template is shown below. However, for our application we need to change three dimensions from the given (default) dimensions:

Our frame has an inner surround edge ID=110mm, so we need to change 100 to 110, and the OD to 115mm. Since the surround inside diameter is 10, the cone OD will be  $110 - (2 \times 10) = 90\text{mm}$ . Likewise the ID may be 85mm ensuring a short flange.



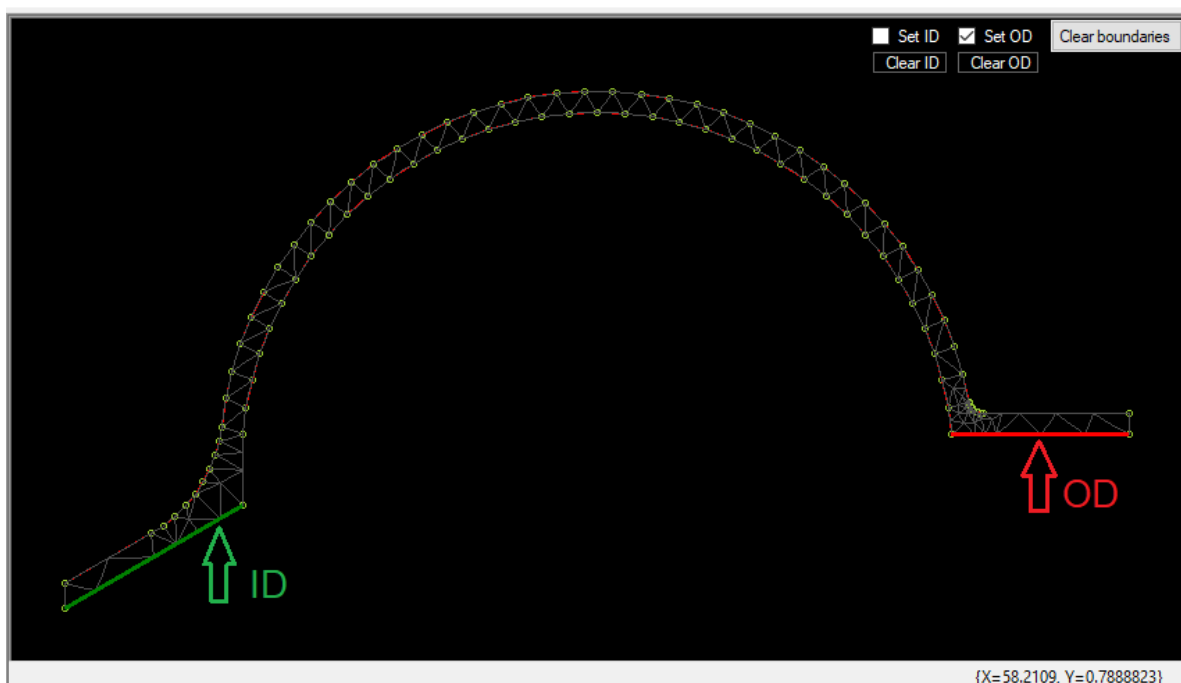
After accepting these dimensions, we only need to choose the material. Having chosen KM rubber 2007 S60 (Shore 60), we need to specify the Peak Force (N). This is defined as  $F = BL \cdot i$  ( $i$ = current (Amps)). Assuming a max current of 1.2A, then  $F = 5 \cdot 1.2 = 6$  N.



The 'Problem Setup' dialog box contains the following fields and options:

- Units: Millimeter
- Tolerance: 0.0001
- Maximum Strain (%): 1
- Solver: ☒ use fast solver
- Material: KM Rubber 2007 S 60
- Young's Modulus (MPa): 13.0000
- Poisson's Ratio: 0.4800
- Peak Force (N): 6
- Buttons: Ok, Cancel

The geometry is Auto meshed and shown in the lower left window.

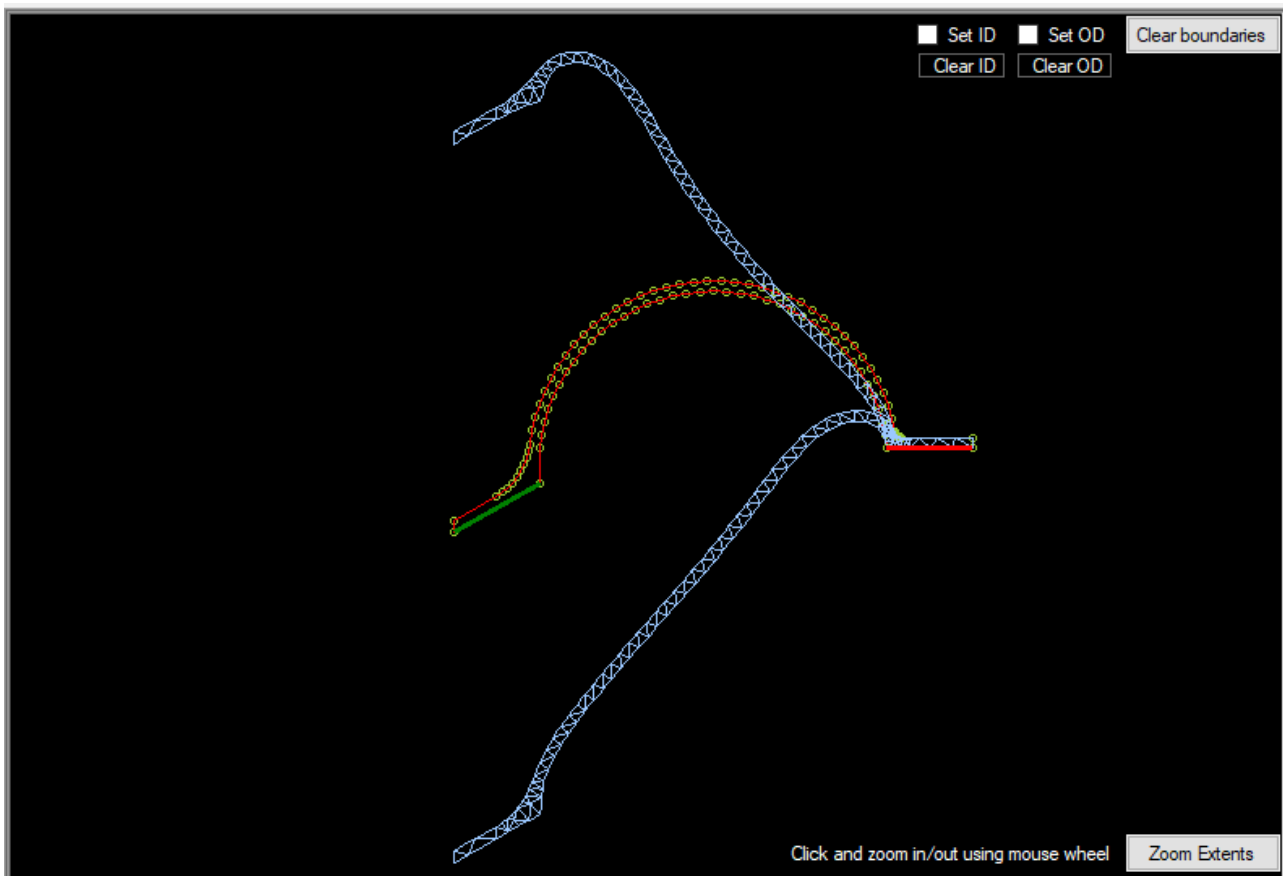


Before we can calculate we must specify which part will be glued to the frame, and which part will be glued to the cone. First click (V) in [Set ID] and point to the inner flange, which should be glued to the cone, and it will be highlighted in green.



Then click (V) in [Set OD] and point to the underside of the outer flange, which will be glued to the frame, now highlighted in red. (if you fail to point correctly, just press Clear ID or OD)

Now we are ready to calculate by pressing the green Calculate button (lower right).

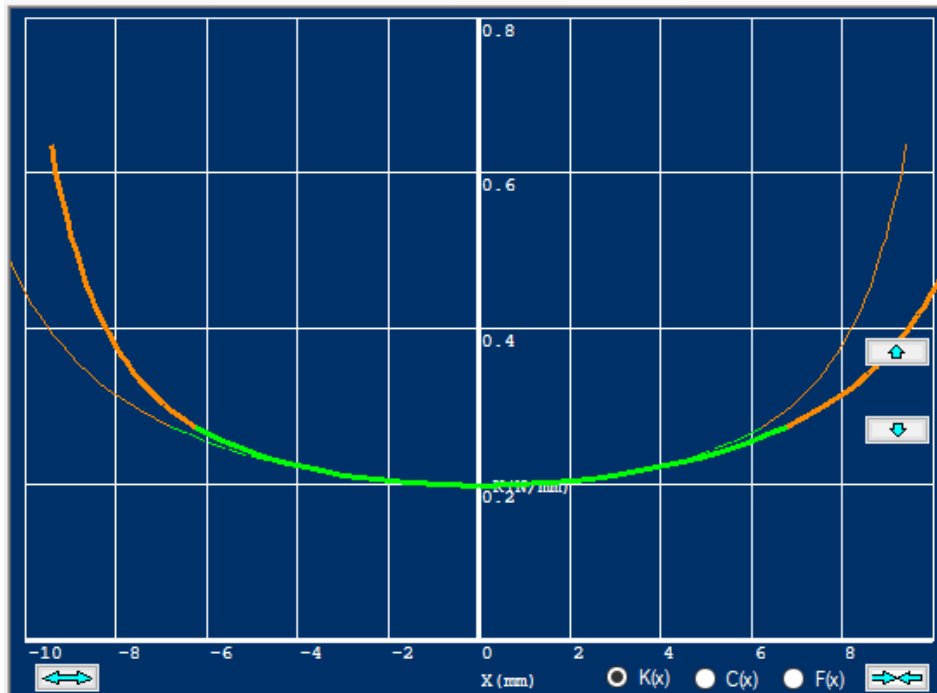


The deformed surround is displayed in steps up and down.

The calculated stiffness versus displacement  $K(x)$  curve is now shown in the upper right window (shown on next page). When the surround is stretched out the stiffness increases, which is displayed as higher  $K(X)$  at maximum excursion up (+) and down (-). This curve shape is preferable as the increased stiffness help controlling the loudspeaker cone at extreme excursions up and down.

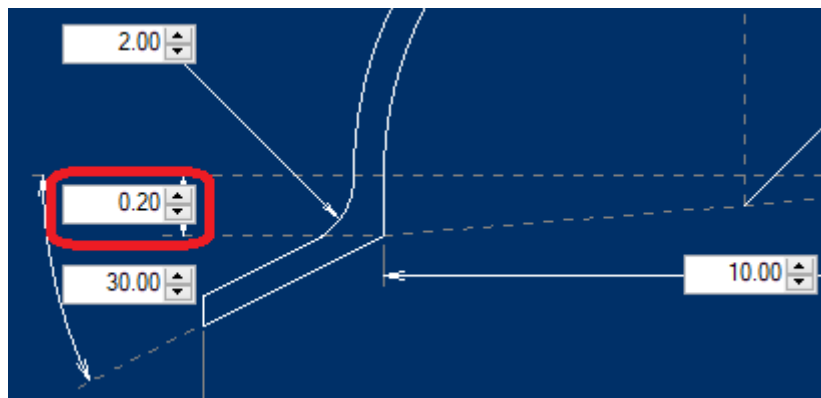
Ideally the U-shaped  $K_{ms}$  curve should be symmetrical along the Y-axis. A non-symmetrical  $K_{ms}$  curve will produce a DC offset (X) of the VC position, which would cause intermodulation distortion.

Therefore, the mirrored  $K_{ms}$  curve is shown as the dashed curve, making it easy to spot any non-symmetries.

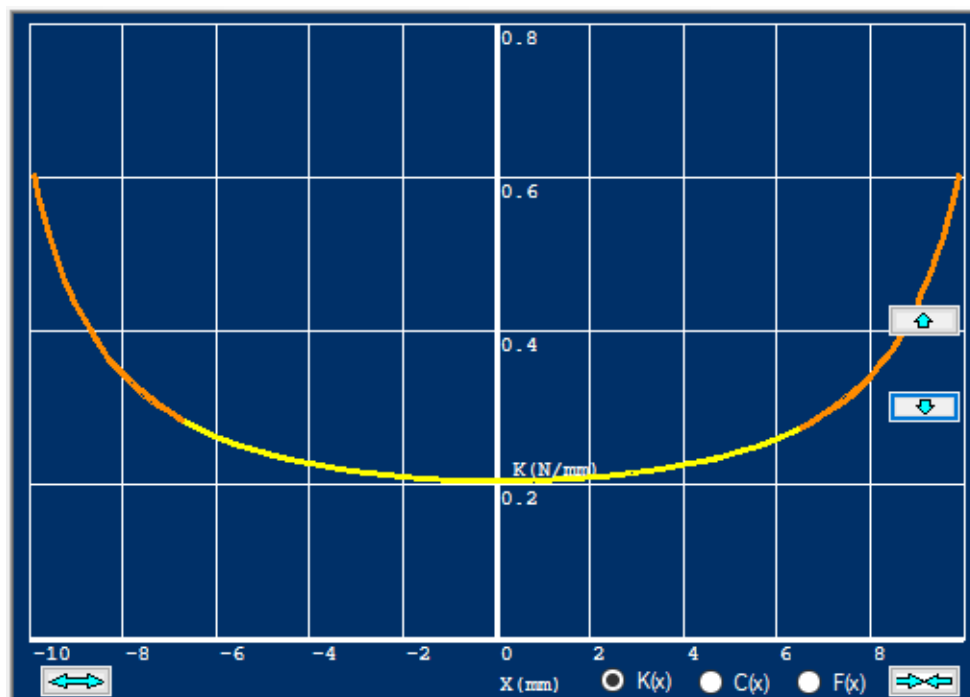
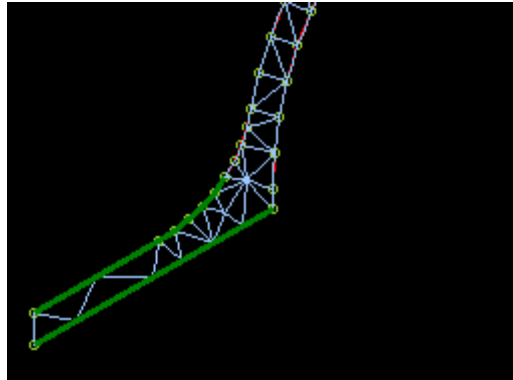


The green part of the curve indicates where the compliance ( $C_{ms}$  or  $C(x)$ ) is still 75% of the max. See page 11. This is specified in IEC 62458, and ensures that the distortion, including intermodulation IMD is below 10%, which is considered good.

In this case the two curves do not overlap, indicating non-symmetry and a high DC offset. Fortunately, we can remedy this in an uncomplicated way by editing the geometry dimensions. After decreasing the 1mm overhang in a couple of steps, we find that 0.2mm is good.

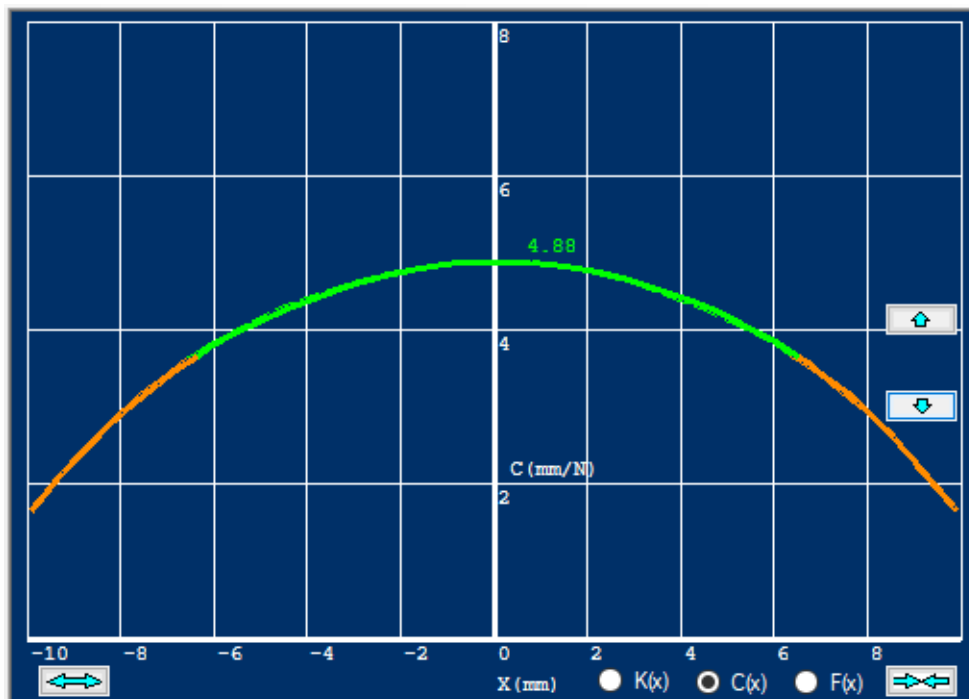


Inspecting the surround flange from the picture on page 8, we notice that the inner flange is slightly distorted at the maximum positions. Assuming the cone is much stiffer than the surround, we may additionally constrain the upper side of the surround flange as shown in the small picture below. This was done by clicking and zooming up using the mouse wheel.



The resulting  $K(x)$  curve is fully symmetrical as the two curves overlap completely:

Instead, we can select  $C(x)$  and get the compliance  $Cms(x)$  shown.



The green part of the curve indicates the compliance is still 75% of the max. This is specified in IEC 62458, and ensures that the distortion, including intermodulation IMD is below 10%, which is considered good.

The compliance for  $x=0$  is also given as 4.88 (mm/N). This can be used in other software calculations, for example using FINECone and FINEMotor.

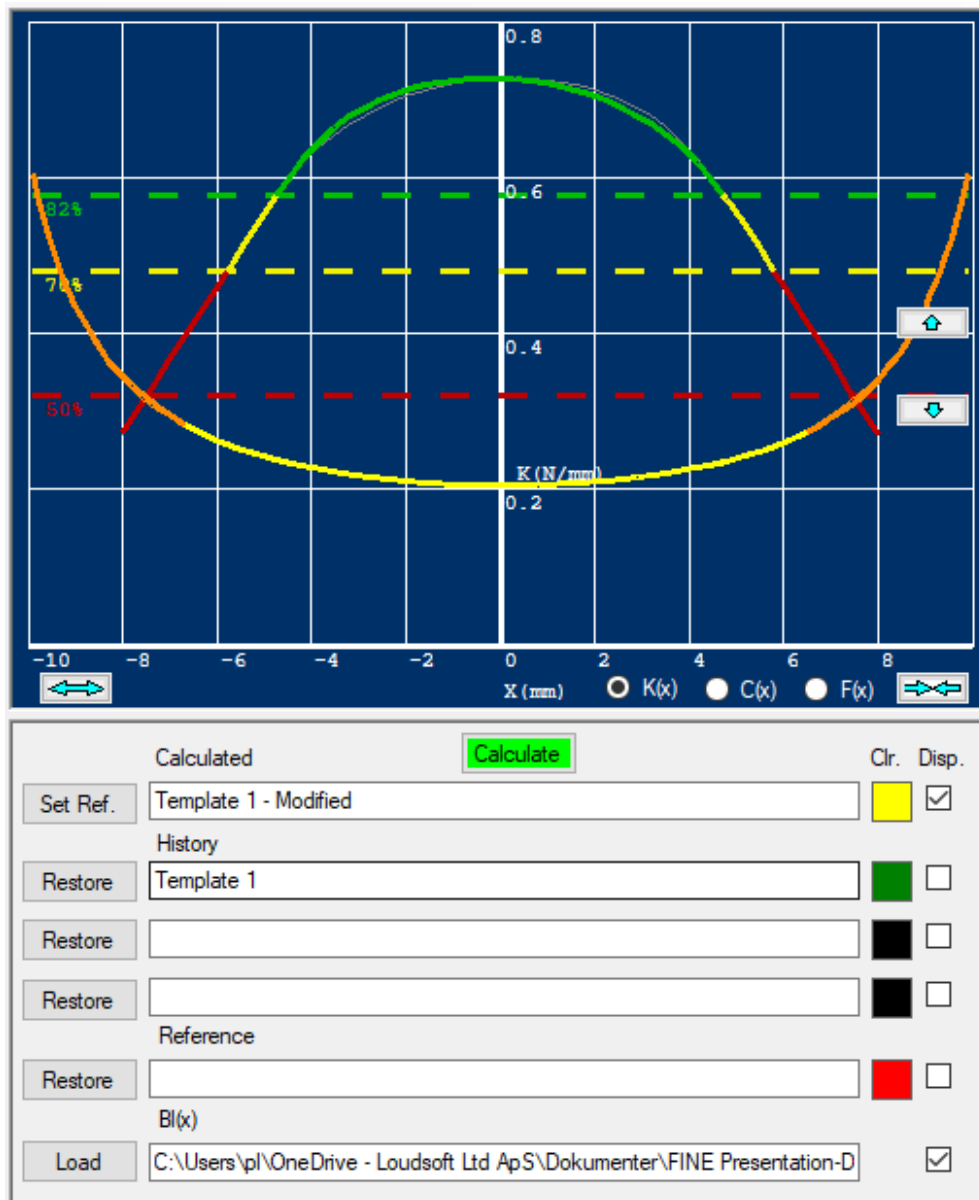
## 2.2 Import BL(x) curve for comparison

In addition, you can load a BL(x) curve for comparison in the second last line. The BL(x) curve can be exported as a text file from FINEMotor. For x-values where BL is below 82% of max. (red part of BLx), a higher stiffness is required to control the moving cone and Voice Coil.

Therefore, it is recommended that the spider  $Kms$  should increase significantly from minimum to provide good mechanical control when BL is  $<82\%$  of maximum.

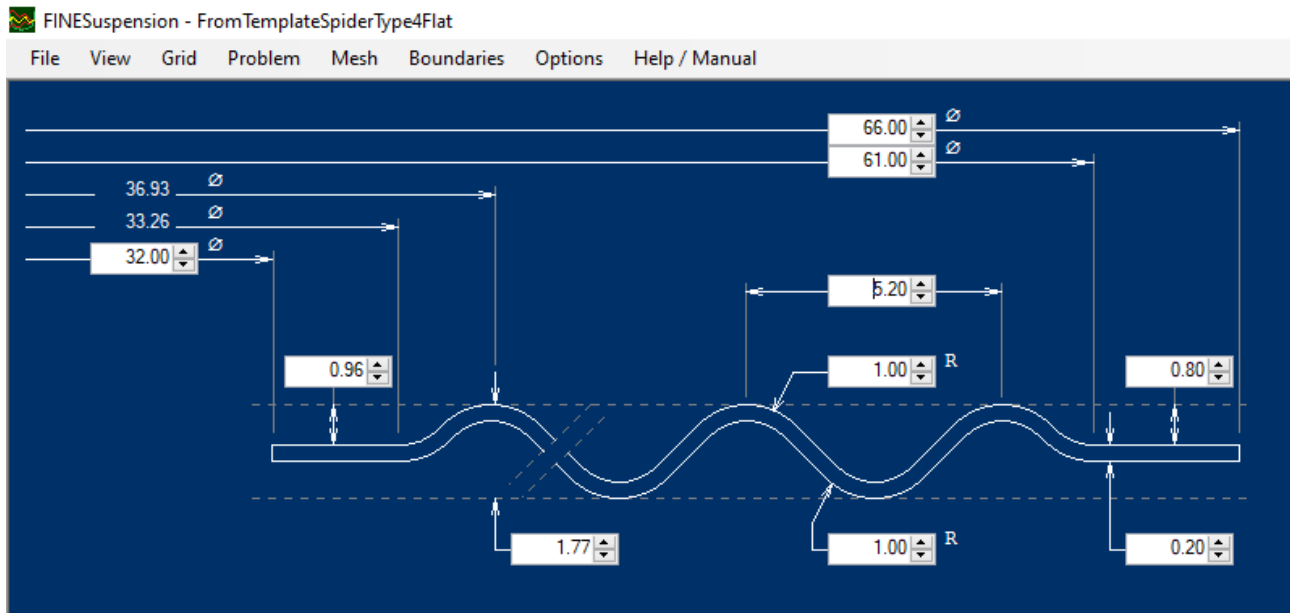
The results are displayed in a stack of curves, so that the latest curve is on top, and the previous shift down, shown in a new color.

In this example we have made a couple of steps, which are all displayed below, see picture. For clarity you can deselect any of the curves by removing the [v].



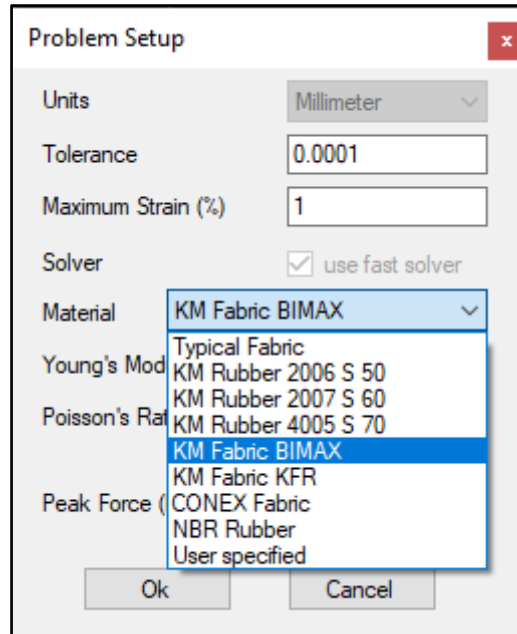
## 2.3 Spider Type 4 – Design Example 2

In this example we want to design a spider. Select the flat spider Template 4 from the File menu.



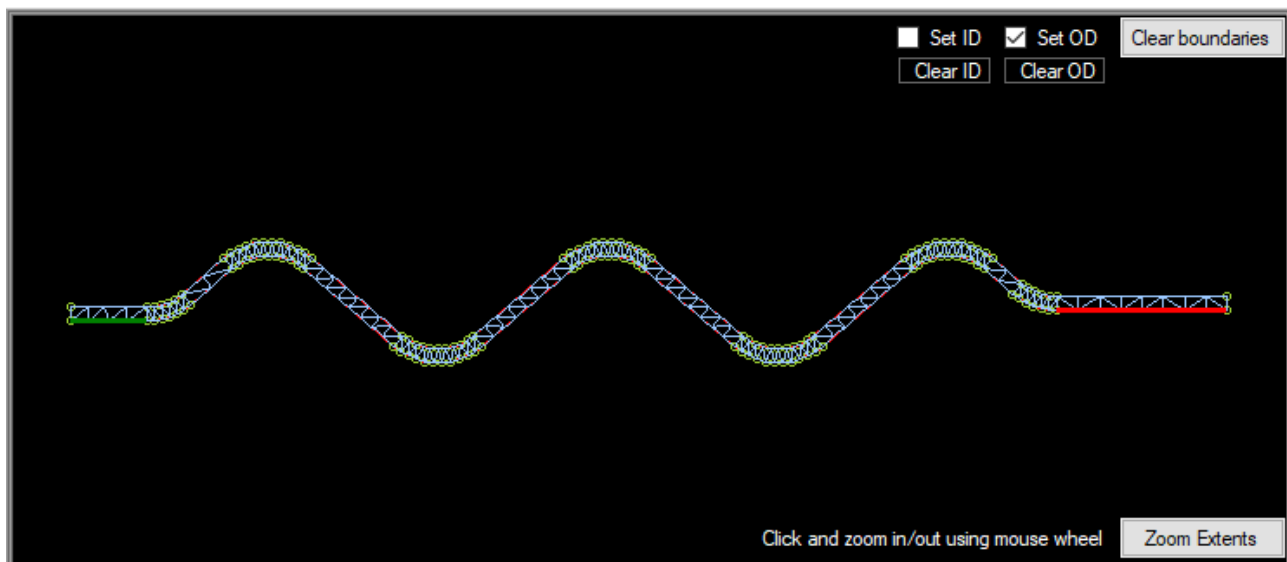
Our frame has a spider landing area (flange) which is close to 61mm, so we can use the default 61 and 66mm OD. Choosing 5mm (instead of 5.2) between waves gives a nice spider profile. (see next picture).

*Note that the number of waves will auto-update when dimensions are changed!*



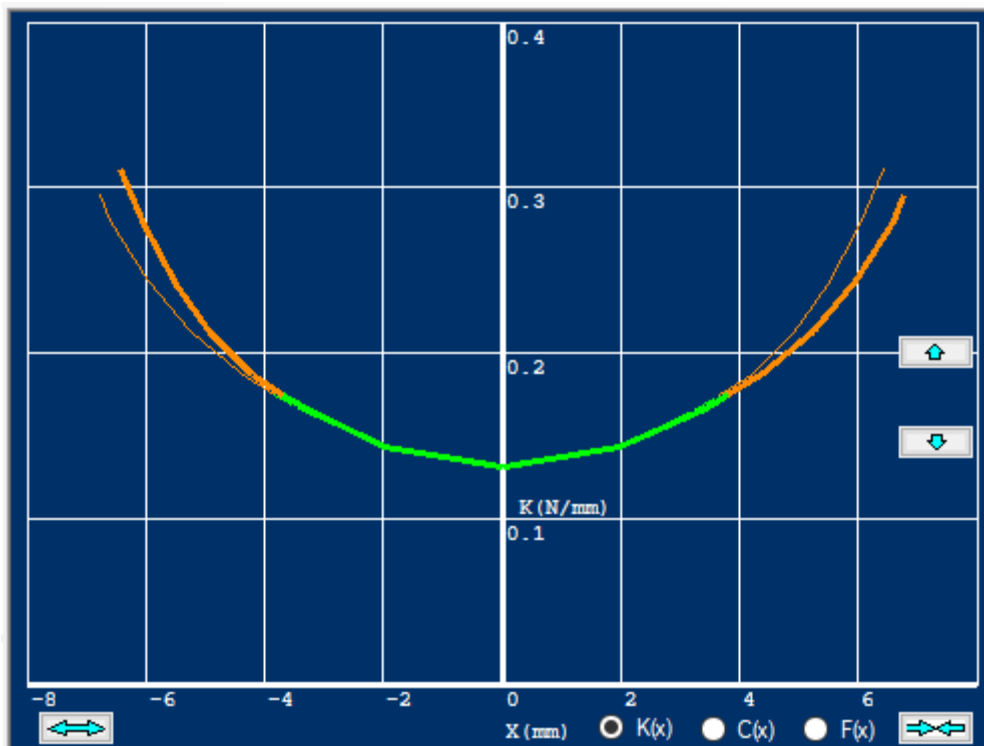
We will choose the strong BIMAX 120 material from KM (Kurt Muller) for this spider. This material is typically 80 MPA (Stiffness/Young's Modulus).

Let us set the Peak Force =2 N. This is defined as  $F = BL \cdot i$ .

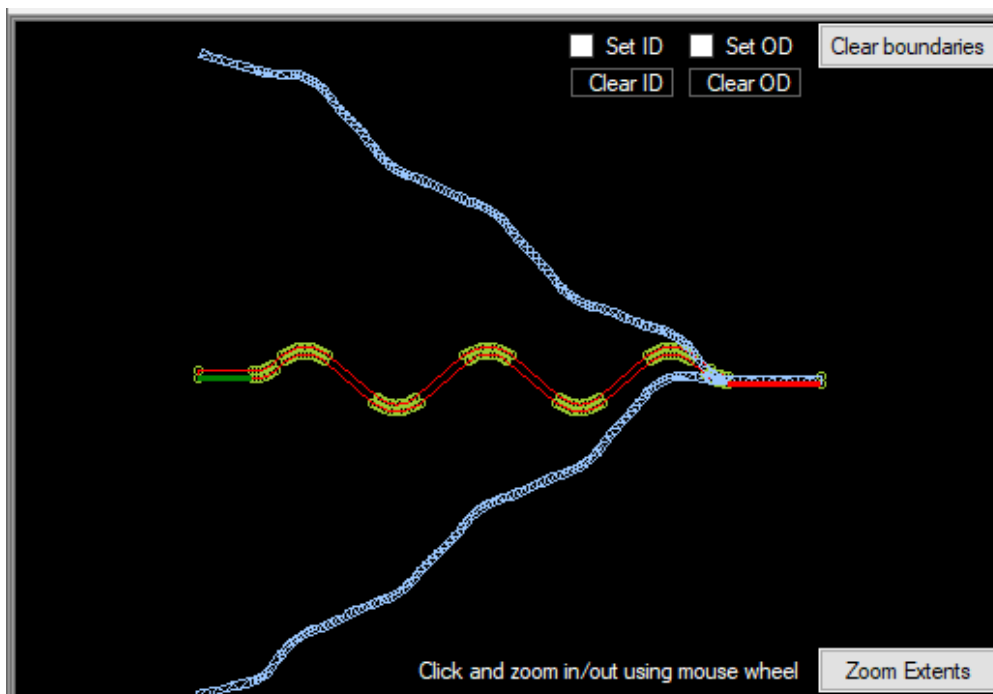


The inner flange is set as ID (green). This means that it should be fully covered by stiff glue. (You can also point at additional small segments if you want them included, covered by glue (=stiff). The inner flange can include a bend fitting the Voice Coil. But if it is covered by stiff glue, you will see no difference in these calculations.

The outer flange will be glued to the frame and was selected in red (Set OD).



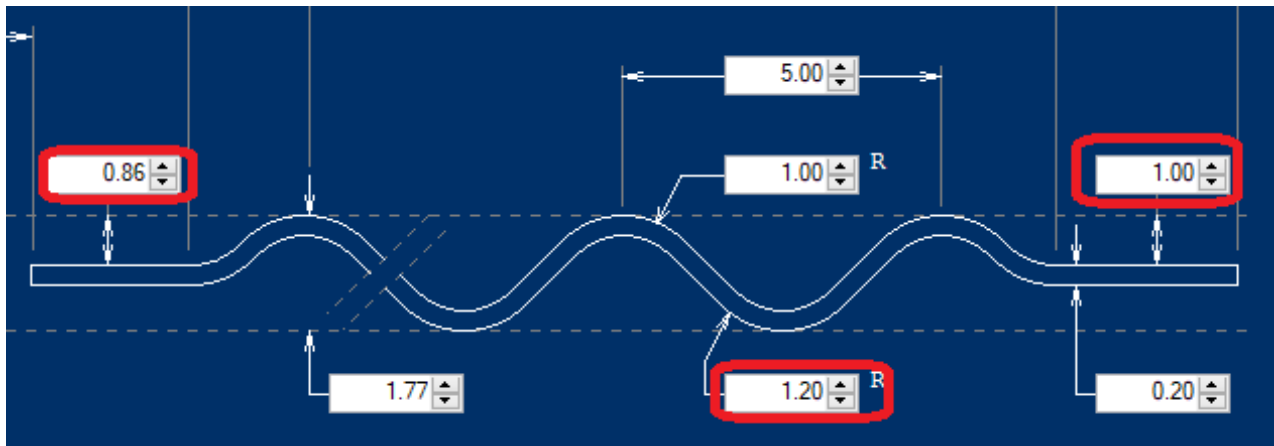
The calculated Stiffness curve  $K(x)$  is nicely U-shaped, but not fully symmetrical (i.e. the mirror curve - - - does not overlap the main curve).



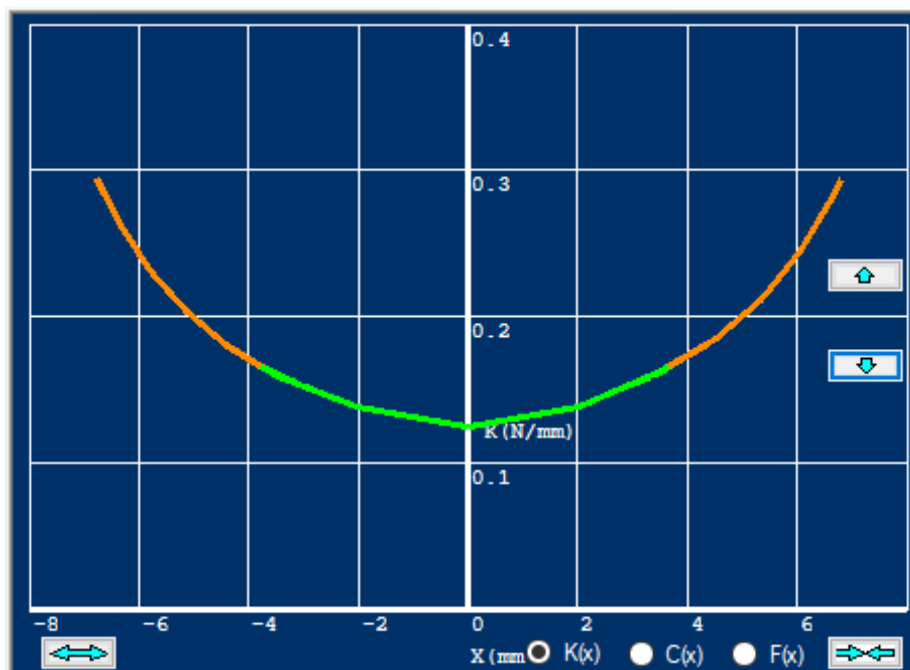
The excursion in the negative direction (down) is less than the positive. The max up and down movements are shown in the picture below.



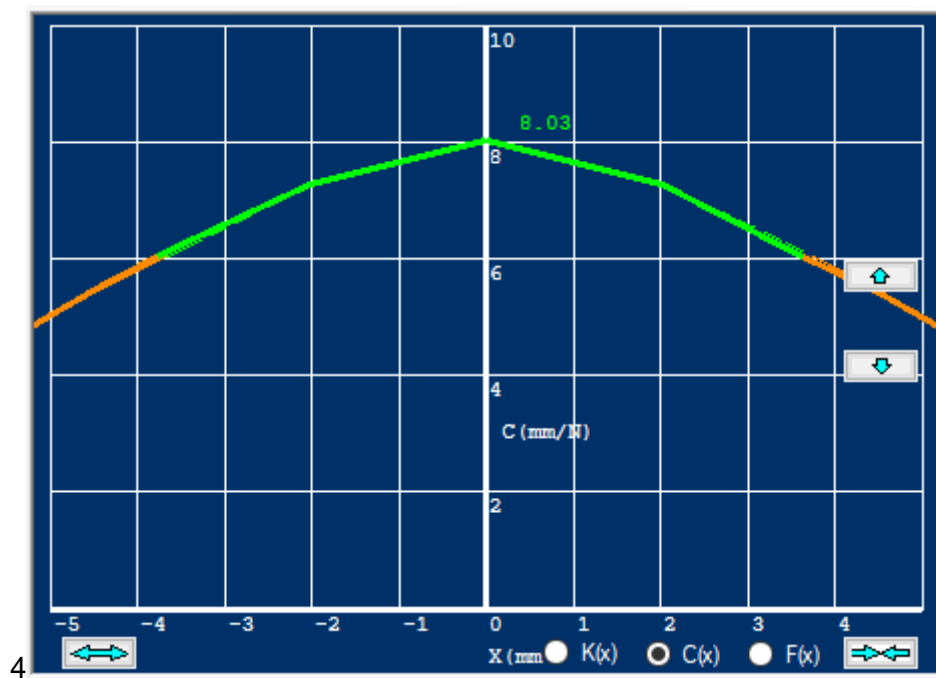
If we move the inner flange down and outside flange up, we can increase the downward excursion.



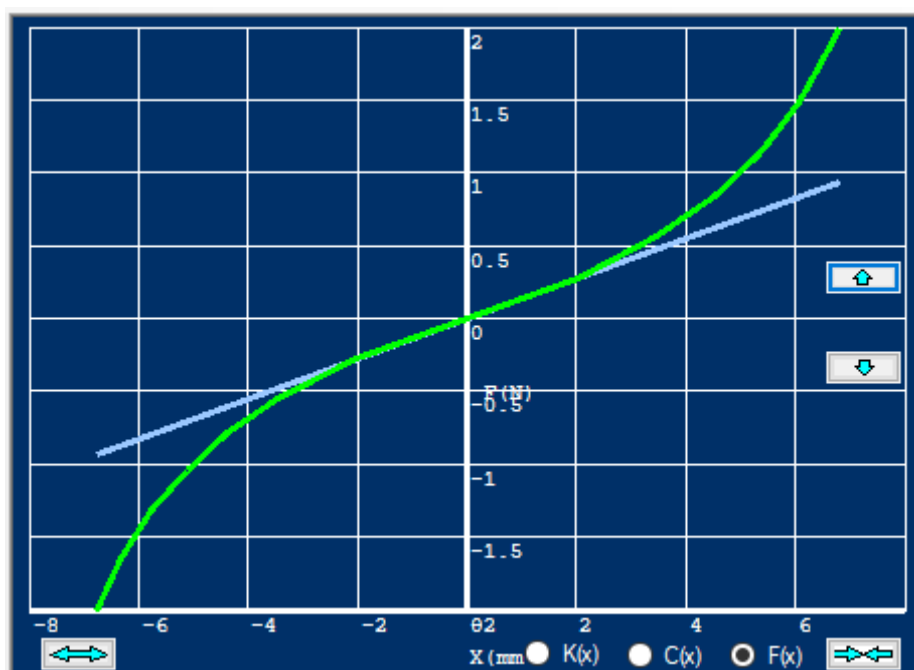
The distance from the flange up to the top wave is 0.96mm. But adjusting this to 0.86mm and outside offset to 1mm, plus increasing the bottom roll radius to 1.20, we get the next calculated  $K(x)$  curve:



The new  $K(x)$  curve is fully symmetric, which ensures no DC-offset and low (intermodulation-) distortion. This curve means that this spider will stiffen up after appr.  $\pm 4$  mm excursion.

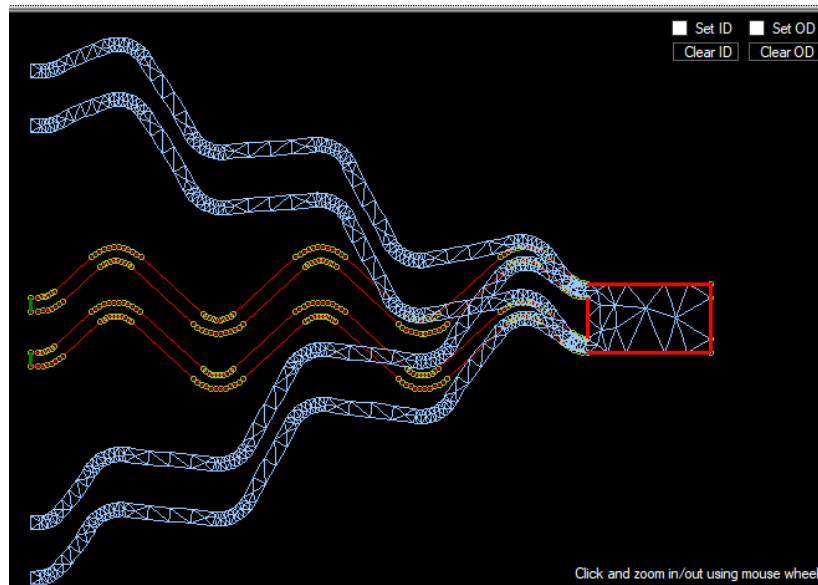


Instead of  $K(x)$  the calculated result can be shown as  $C_m(x)$ :  $C_m$  for  $X=0$  is 8.03 mm/N



The same data can be displayed as Force/Deflection ( $F/X$ ), which is linear for small deflections, indicated by the light blue line. For larger deflections  $X$ , the non-linearity shows both for positive (+ $X$ ) and negative (- $X$ ) deflections.

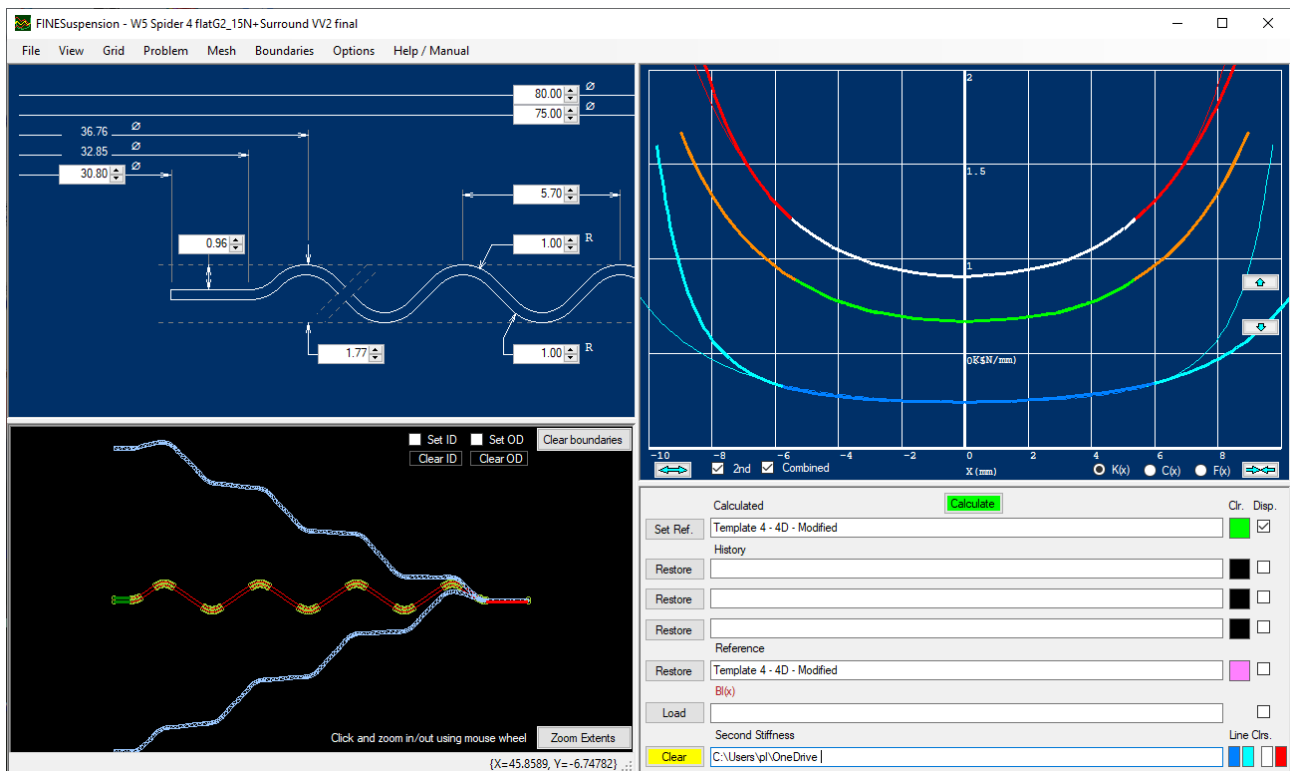
## 2.4 Double Spider Example



You can calculate a double spider as in the above example, by drawing the DXF file with two spiders connected to a common base. The base should then be constrained by using Set OD, as indicated in red.

See also the next section.

### 3.1 Combined Spider + Surround



From this version you can combine a spider + surround. This way you can calculate the total stiffness and compliance of a real speaker.

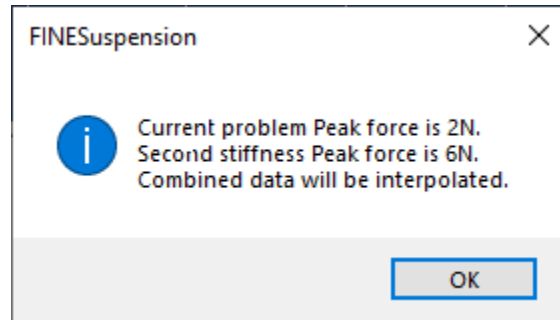
When you for example has calculated a spider (green), you can add another element like a surround (blue). Go to the last line (lower right) and press Add. Then you may add the corresponding surround, which was previously calculated in FINESuspension.

In the above example, the lower blue curve is the surround stiffness ( $K_{ms}(x)$ ), which is much lower than the spider.

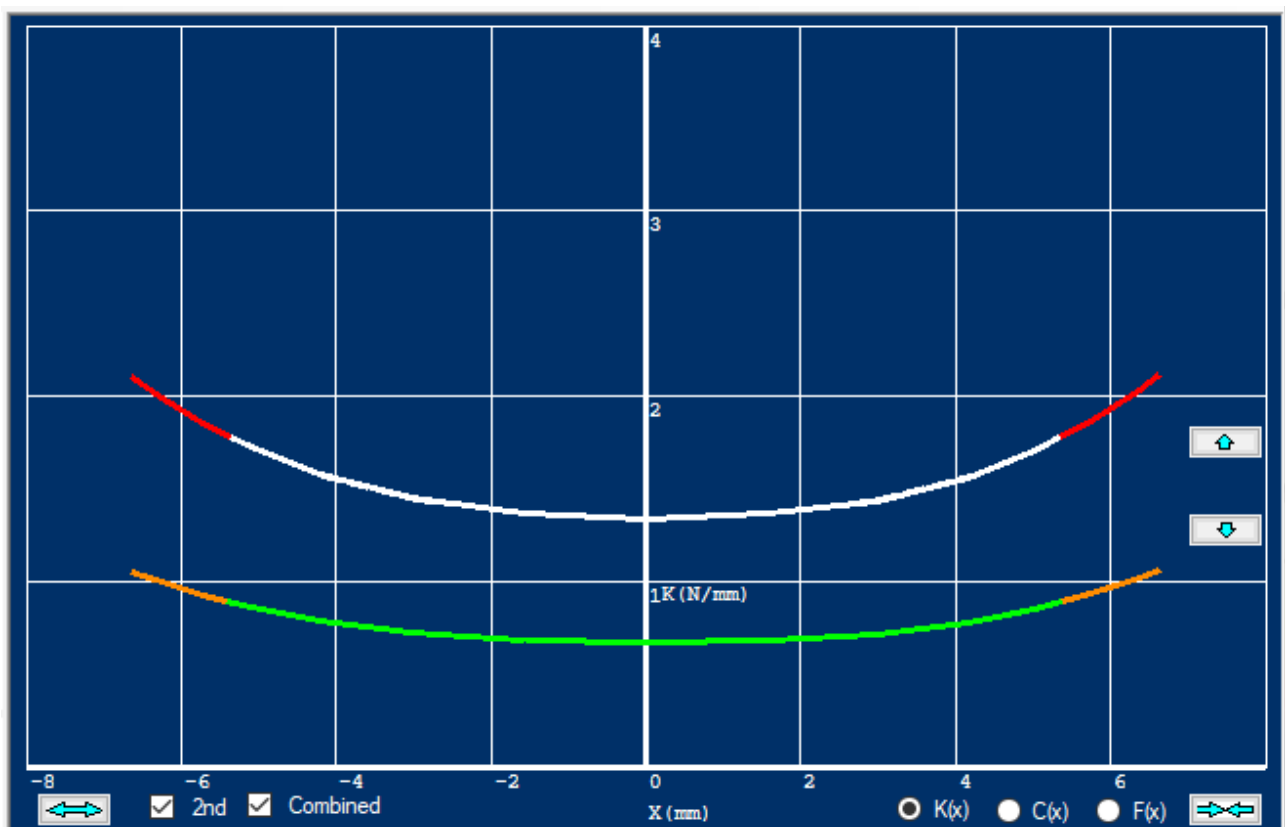
The combined stiffness  $K_{ms}(x)$  is shown as the upper white curve. Actually, it is not much higher than the spider, because the surround is much softer than the spider.

Both  $K_{ms}(x)$ ,  $C_{ms}(x)$  and  $F_{ms}(x)$  ( $F(x)$ ,  $M(x)$  and  $F(x)$ ) curves will be added and can be displayed as selected.

If the  $x$  range is too limited; you may need to recalculate one of them with a larger Force ( $F$ ). The curves will be interpolated in any case, see next:

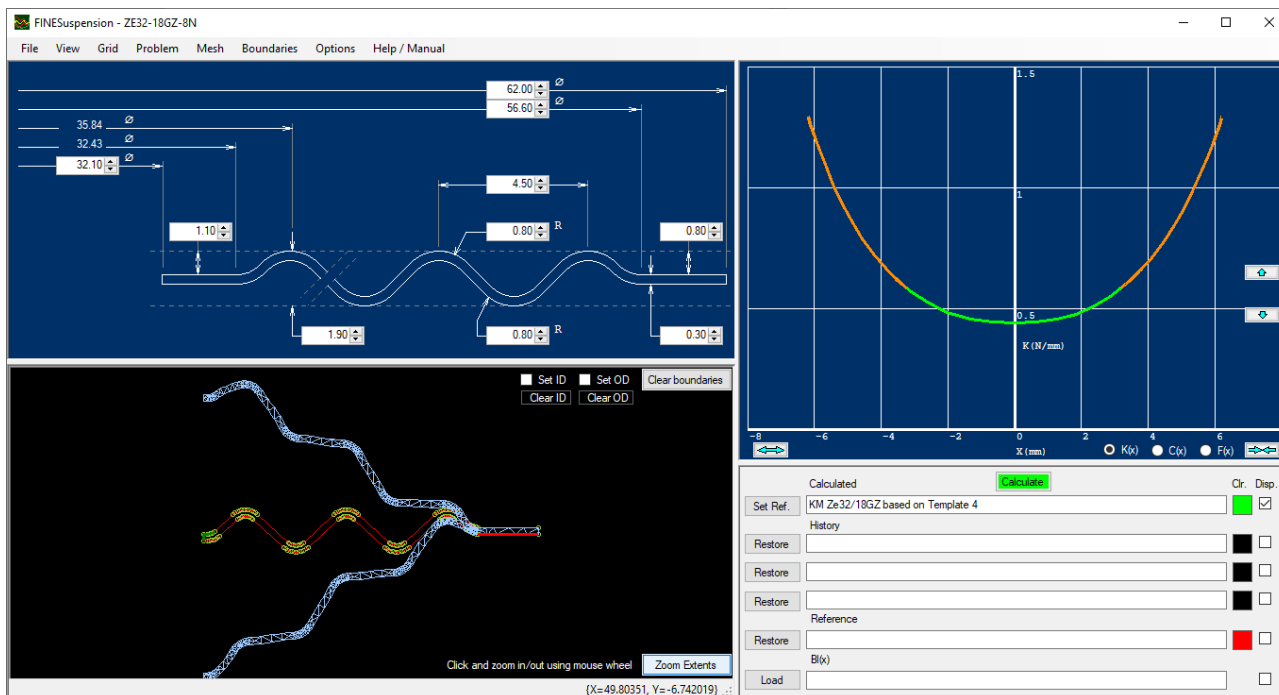
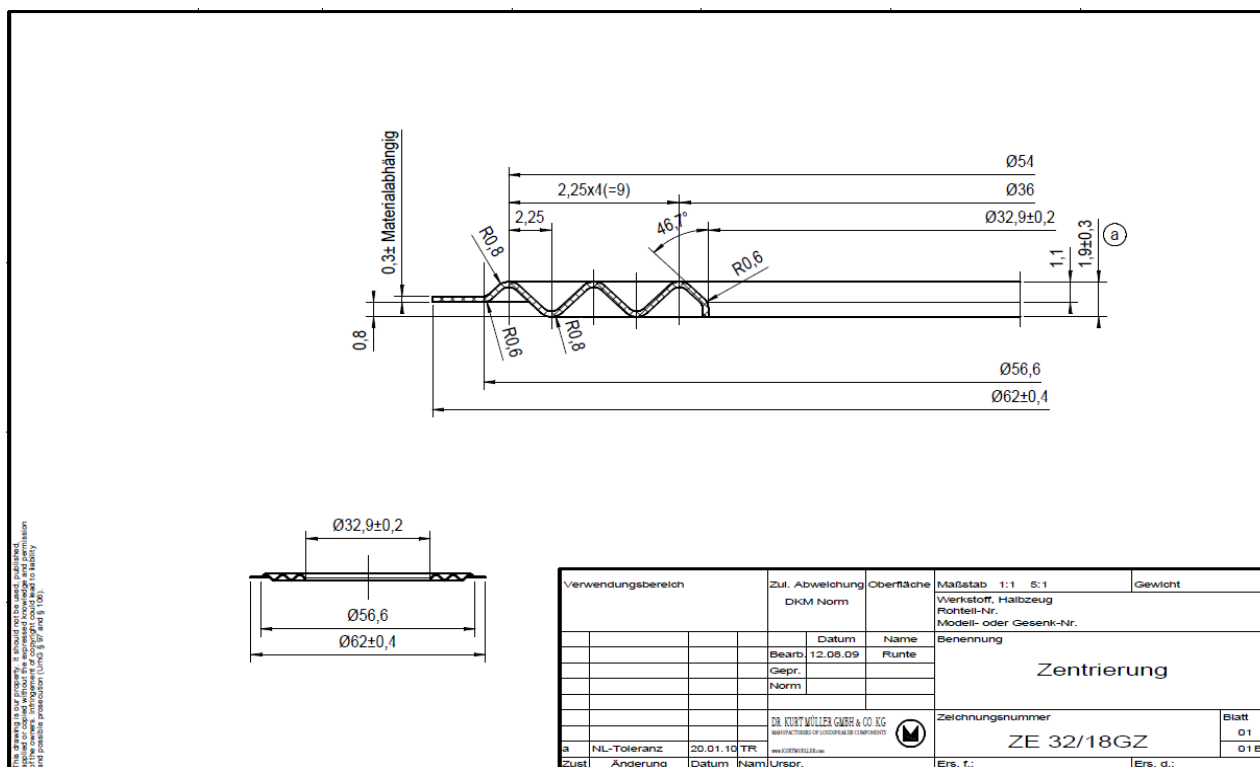


However you can also calculate for example two of the same spiders, see the example below. The lower (green) curve is the stiffness  $K_{ms}(x)$  of each spider. The white curve is two spiders combined, which is then doubled in stiffness  $K_{ms}(x)$ .

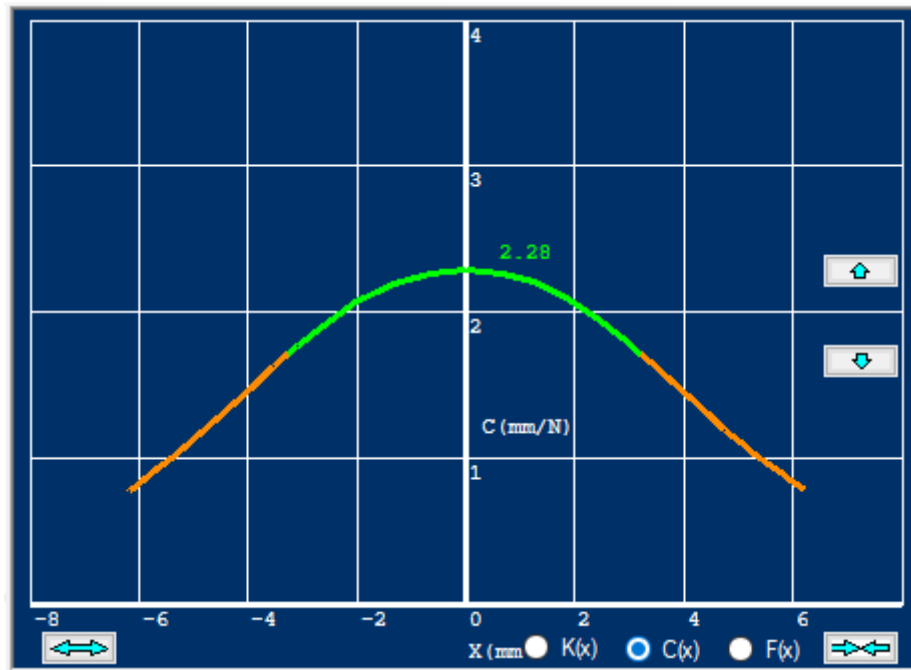


*Note. You can add a second stiffness item in the **last line (lower right)**. Then it may be cleared again if you need another.*

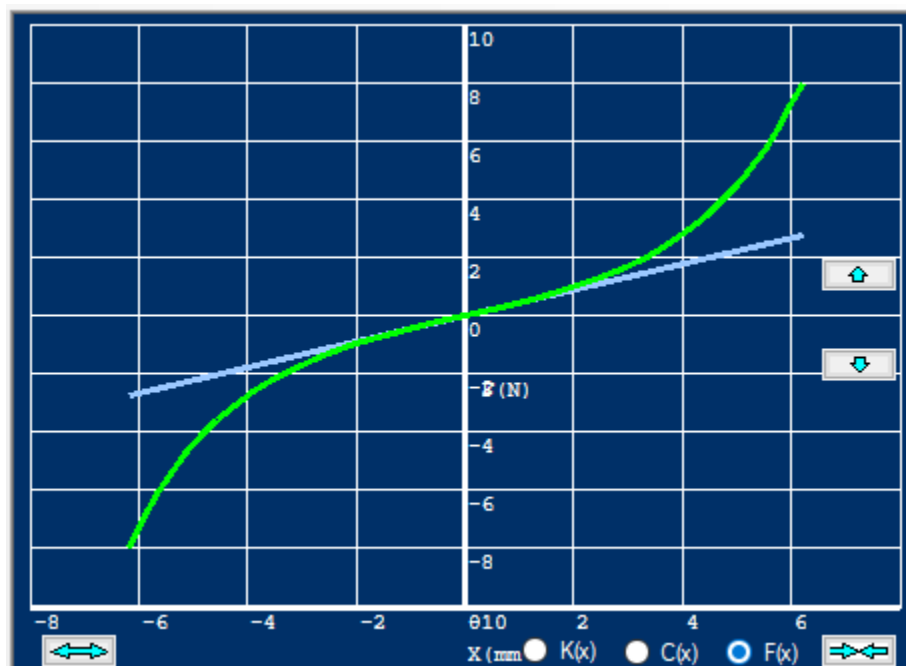
### 3.2 Verification with actual spider ZE32/18GZ



The KM spider ZE32/18GZ spider in BIMAX120 has been verified above. The precise clamping/constraint at the VC neck is slightly different, but the compliance  $C_{ms}$  is quite close.



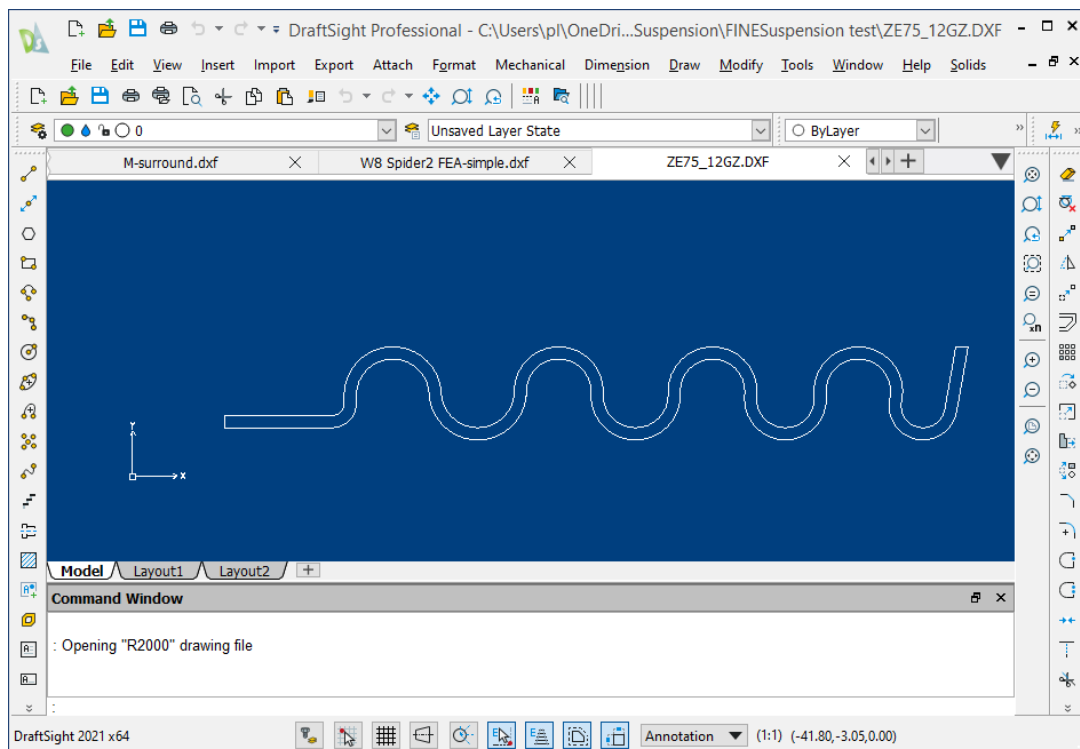
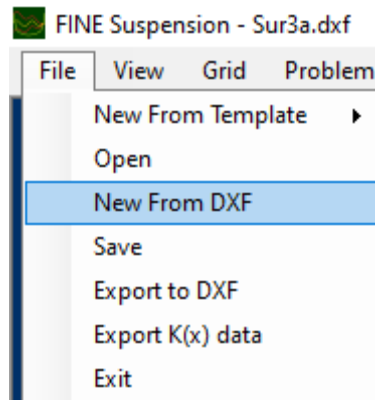
Cms of ZE32/18GZ spider in BIMAX120



The  $F(X)$  (Hysteresis) of ZE32/18GZ spider in BIMAX120 is very similar to the measured curve, but with slightly lower displacement. Often actual spiders show a little less excursion as some materials stiffen up at high forces. Typical rubber surrounds behave more as simulated.

### 3.3 Geometry Input from CAD (DXF)

In case you cannot find a suitable geometry using the templates, you can always make your special geometry as a CAD file, in DXF format. This format was created by AutoCAD and can be saved by nearly all CAD programs. You may **save in the original R12 ASCII format** for best compatibility.



This is an example of a special spider geometry, which cannot be created using the templates. We recommend DraftSight and QCad as good programs for designing DXF files, but most CAD programs can be used. Note that there must be NO overlaps or gaps. Use high accuracy

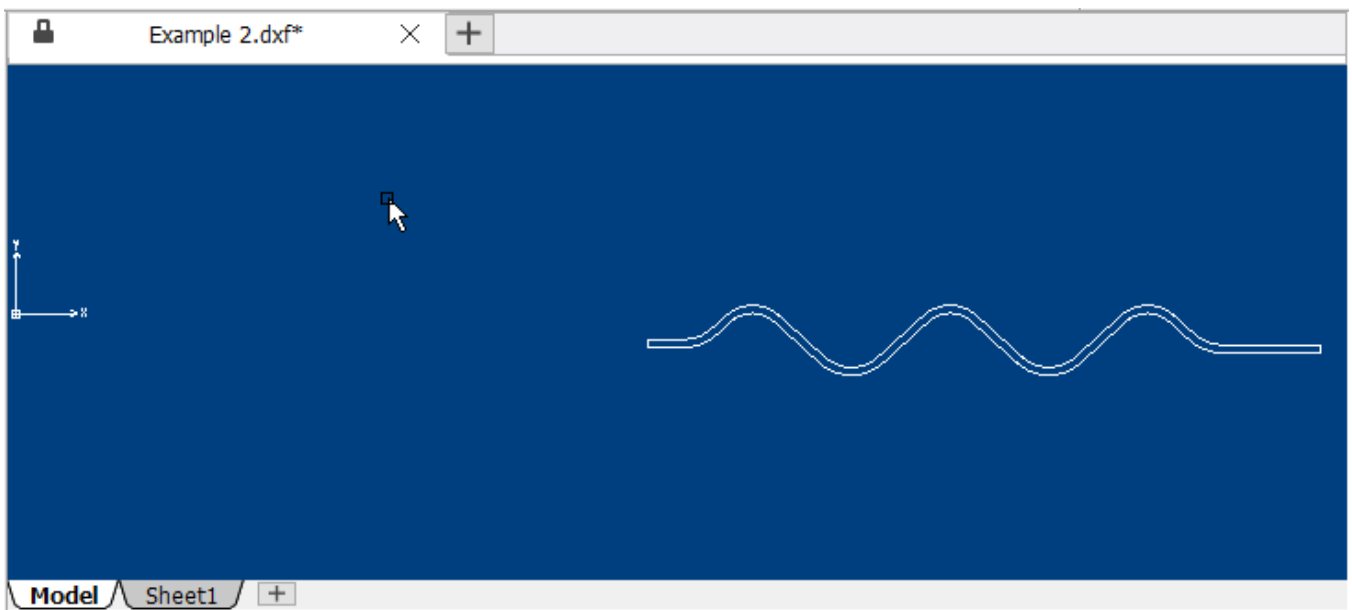


(minimum 6 decimals after comma) and draw all in ONE layer (usually 0). DO NOT SHOW DIMENSIONS.

If you only need small changes from a template you can export the current geometry to a DXF file, and then edit it as required, and import it again as a DXF file. This is a convenient method for several changes.

### 3.4 IMPORTANT – DXF hints for avoiding calculation errors

**Making DXF files for importing the geometry into FINESuspension is simple, but a few necessary requirements are given below:**

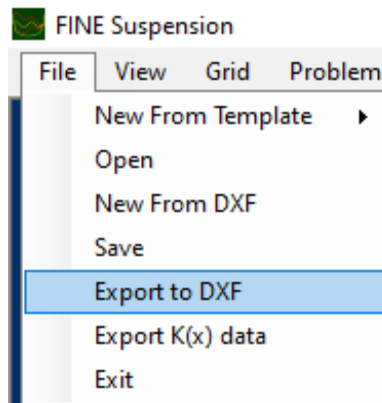


## Requirements

- NO DIMENSIONS or Text is allowed
- No centreline(s) or points
- No gaps between lines and arcs
- Use High precision setting in the CAD system
- No other items like points or symbols in DXF drawing
- Save in the original Autocad12 DXF (ASCII) format if possible

See also FAQ section 3.14

### 3.5 Export Geometry in DXF format



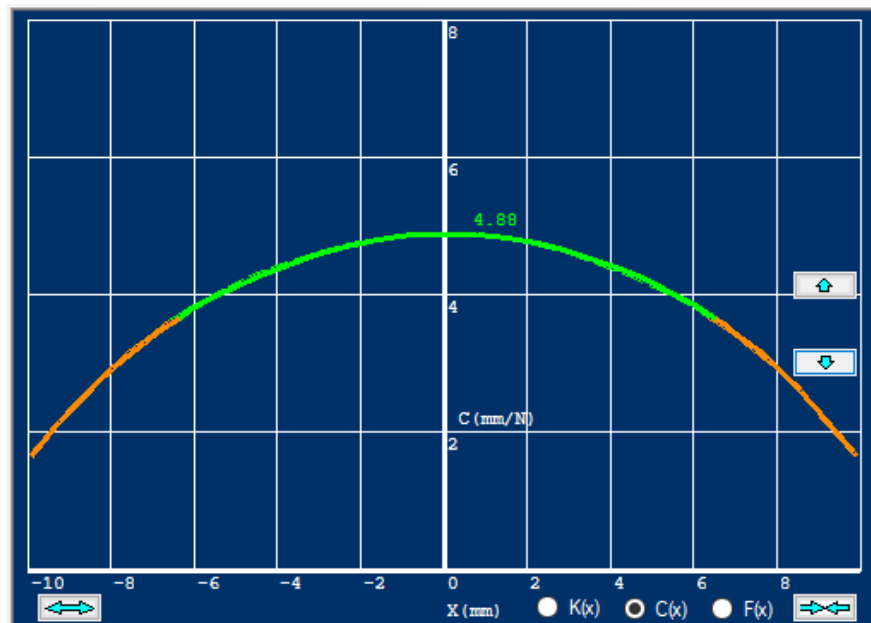
Select this option when you want to get a simple CAD drawing of the simulated model. However, you can also make changes using a CAD system like DraftSight, and then import the changed DXF drawing using the New from DXF option. (Note the DXF may be **saved in the original R12 ASCII format** for best compatibility)

If you for example need a vertical neck on you spider, you can export the used geometry and change the neck and continue using the New from DXF option.

*Please note that the spider neck and other elements, which are selected as ID (in green color), are constrained, meaning that these elements are not allowed to bend. Therefore, it may be adequate simulating a flat neck, even if you plan to use an almost vertical neck.*

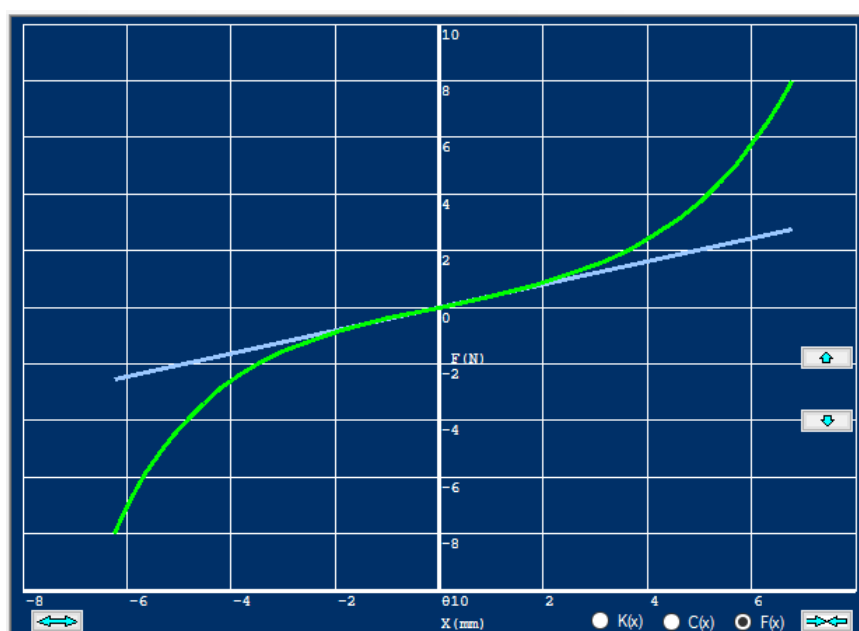
### 3.6 Compliance Cms(x)

The default  $K(x)$  curves are very useful, but you can also display the same data as compliance  $Cms(x)$ .

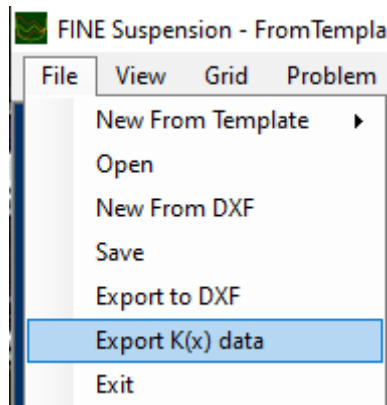


### 3.7 Force / Deflection

Another way to display the non-linearity is the Force/Deflection curve  $F(x)$ . The initial ~linear part is shown as the asymptote (Light blue).



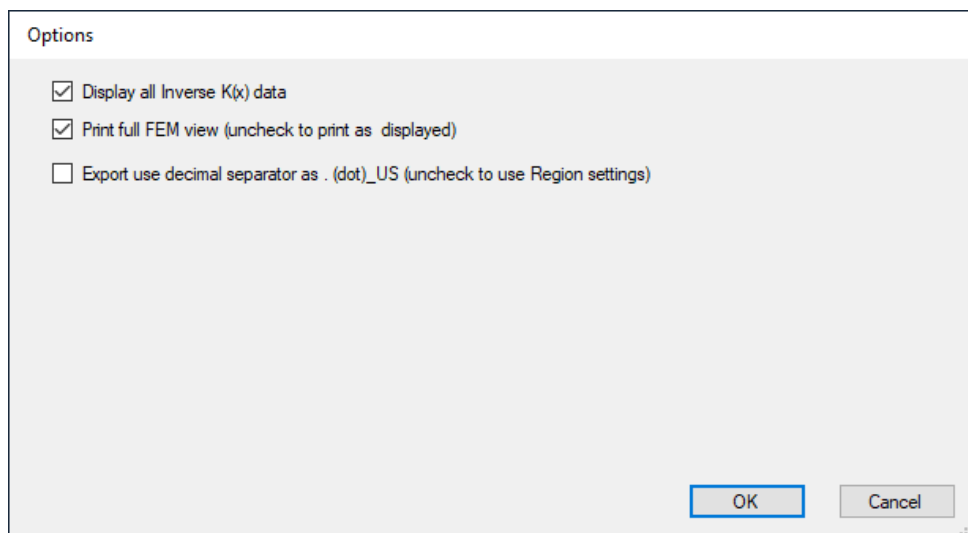
### 3.8 Export K(x) data



You can export all the curves (including K(x)) as data to Excel in \*.CSV format. This will allow you to include the data in reports or other calculations.

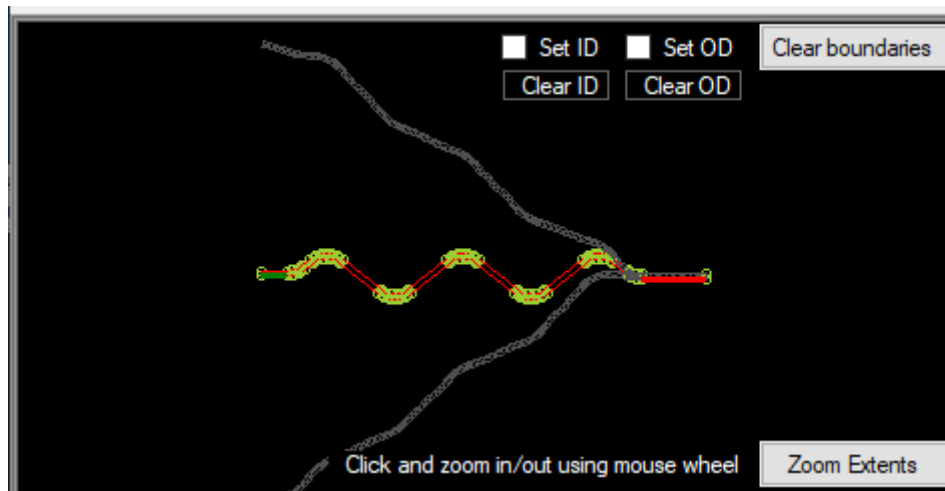
The FEA solver in FINESuspension uses the point (.) as decimal comma. That is the default for English speaking countries or regions. However, in other regions where the comma (,) is used as decimal comma, FINESuspension will automatically convert the used comma.

In other words, you should be able to use your normal decimal comma in FINESuspension and use the exported K(x) files in your Excel as normal.



In any case this behavior is set in Options, where you can change the exported \*.CSV to use either the (.) or (,) as decimal comma as needed.

### 3.9 Display Options



Clicking Zoom Extents will display the geometry with minimum and maximum excursion.

*Note that you can click on one point and zoom up/down using the mouse wheel from exactly that point.*

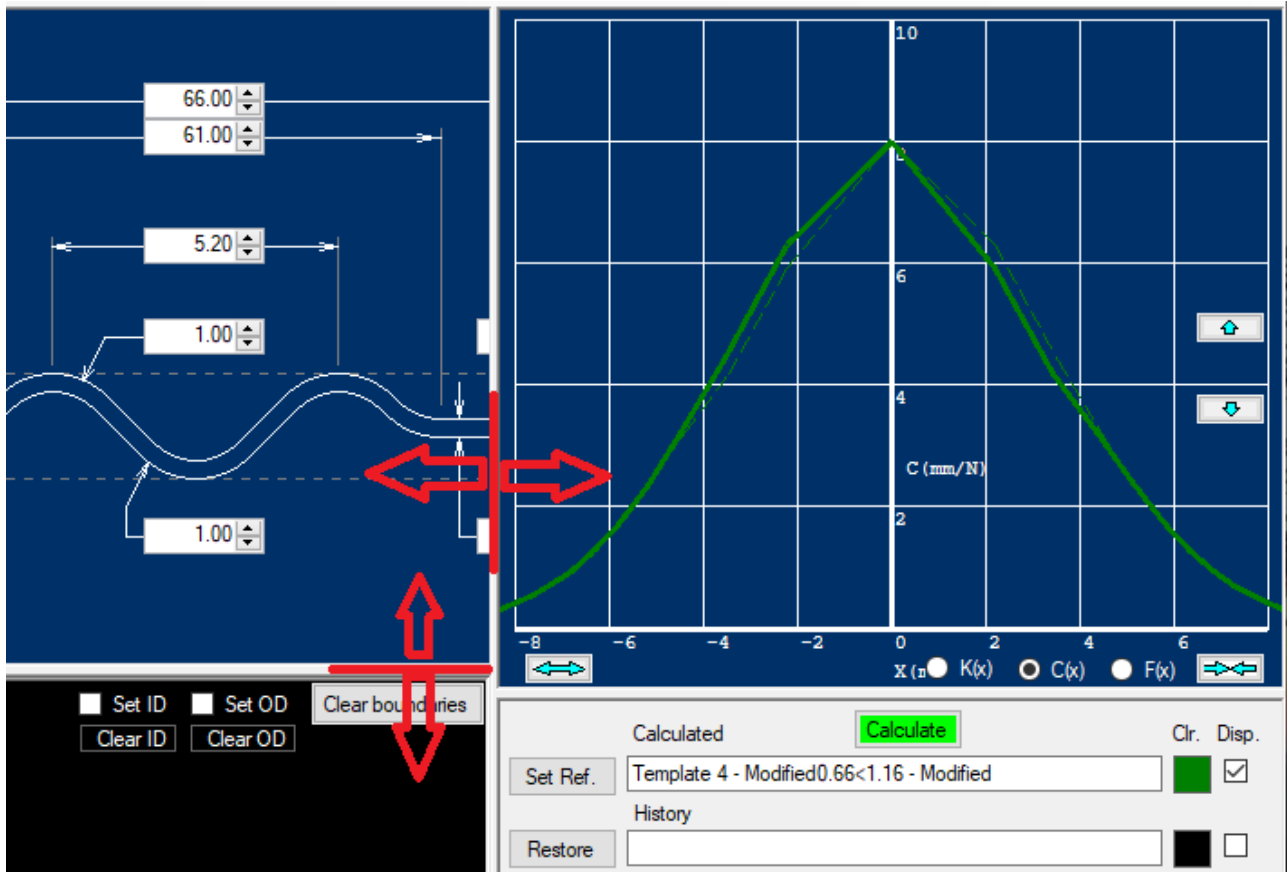
### 3.10 Calculation step size

A screenshot of a 'Problem Setup' dialog box. It contains several input fields and checkboxes. The 'Units' dropdown is set to 'Millimeter'. The 'Tolerance' field is set to '0.0001'. The 'Maximum Strain (%)' field is set to '0.5'. The 'Solver' section has a checked box for 'use fast solver'. The 'Material' dropdown is set to 'KM Fabric BIMAX'. The 'Young's Modulus (MPa)' field is set to '80.0000'. The 'Poisson's Ratio' field is set to '0.3300'. The 'Peak Force (N)' field is set to '2'. At the bottom are 'Ok' and 'Cancel' buttons.

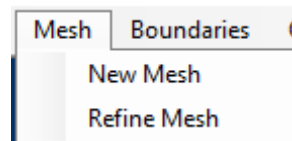
The Maximum Strain determines the step size for the displacement. It is by default 1%. Decreasing it to 0.5% or lower will produce a smoother curve, which will take longer to calculate.

### 3.11 Moving Borders

It is possible to change the size of each window by dragging the border lines.



### 3.12 Mesh

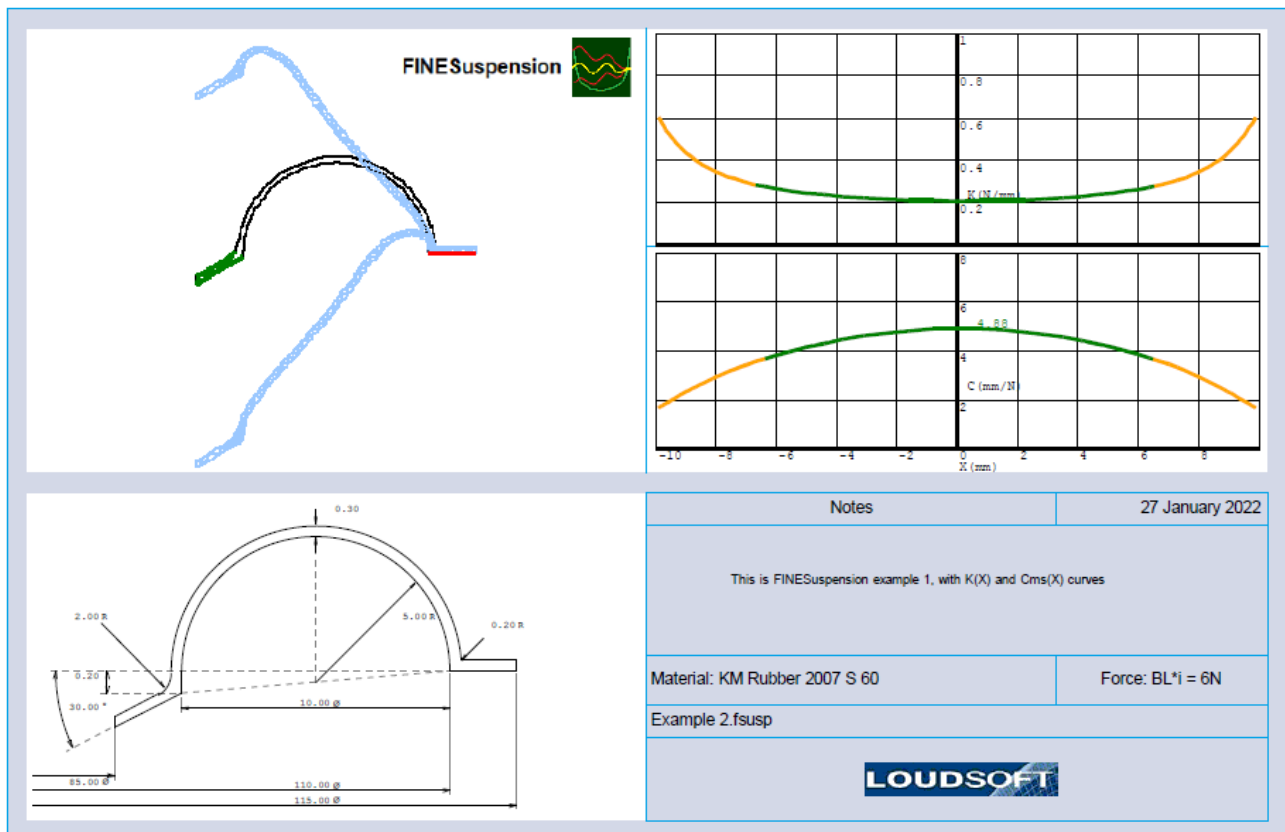


The meshing is automatic, and normally provides very high accuracy. However, you can refine the mesh. Please note that will cause the FEA calculations taking much longer time.

### 3.13 Printing

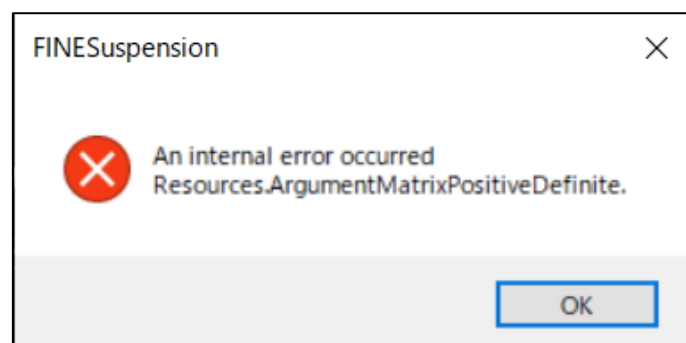
Below is an example of printing from FINESuspension. Both the dimension sketch and max and min deformations are shown. Also, the resulting  $K(X)$  and  $Cms(x)$  curves with green (75%) range are displayed.

From the Problem menu you can select Notes, where you can input notes to be shown on the print.



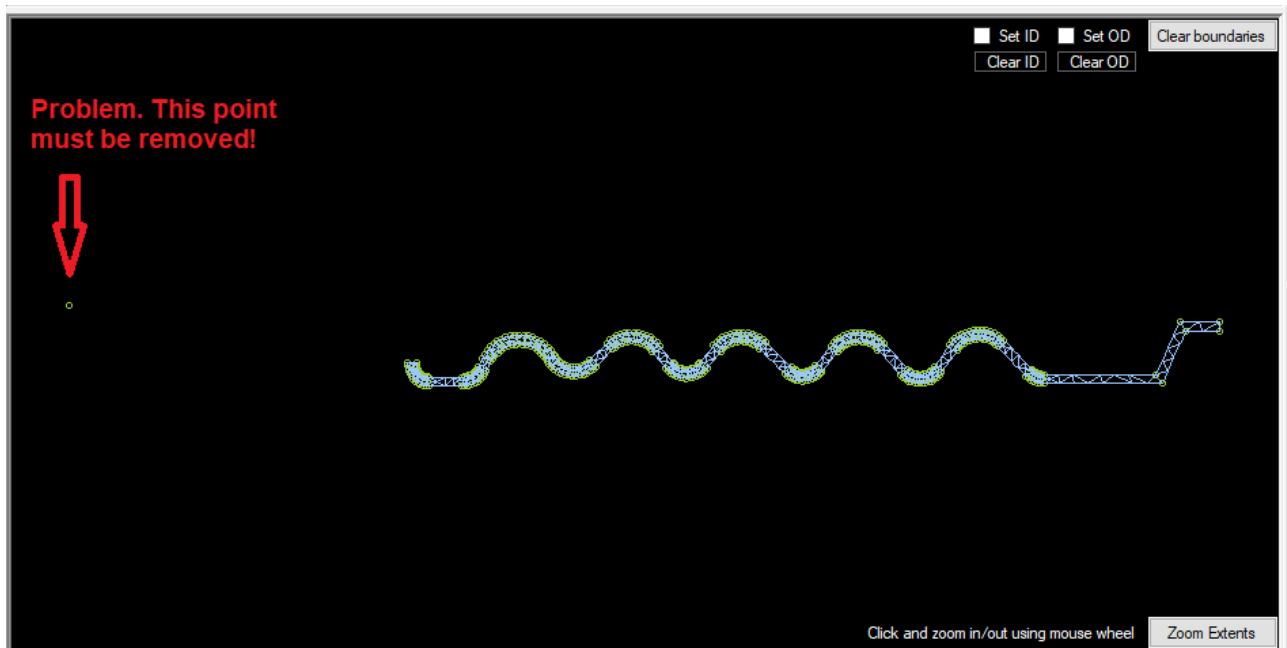
### 3.14 FAQ - Frequently Asked Questions

Cannot calculate:



If you see this error message then the FINESuspension program is not failing. But the structure /DXF has an improper structure, which cannot be handled by the FEA engine.

*You should therefore check your DXF file for errors. See the important DXF Hints section 4.4 for avoiding calculation errors*



The DXF file failed in the above picture. That is because one point was left outside the spider itself. The FEA engine cannot calculate this because it must be one continuous structure.



[www.loudsoft.com](http://www.loudsoft.com)