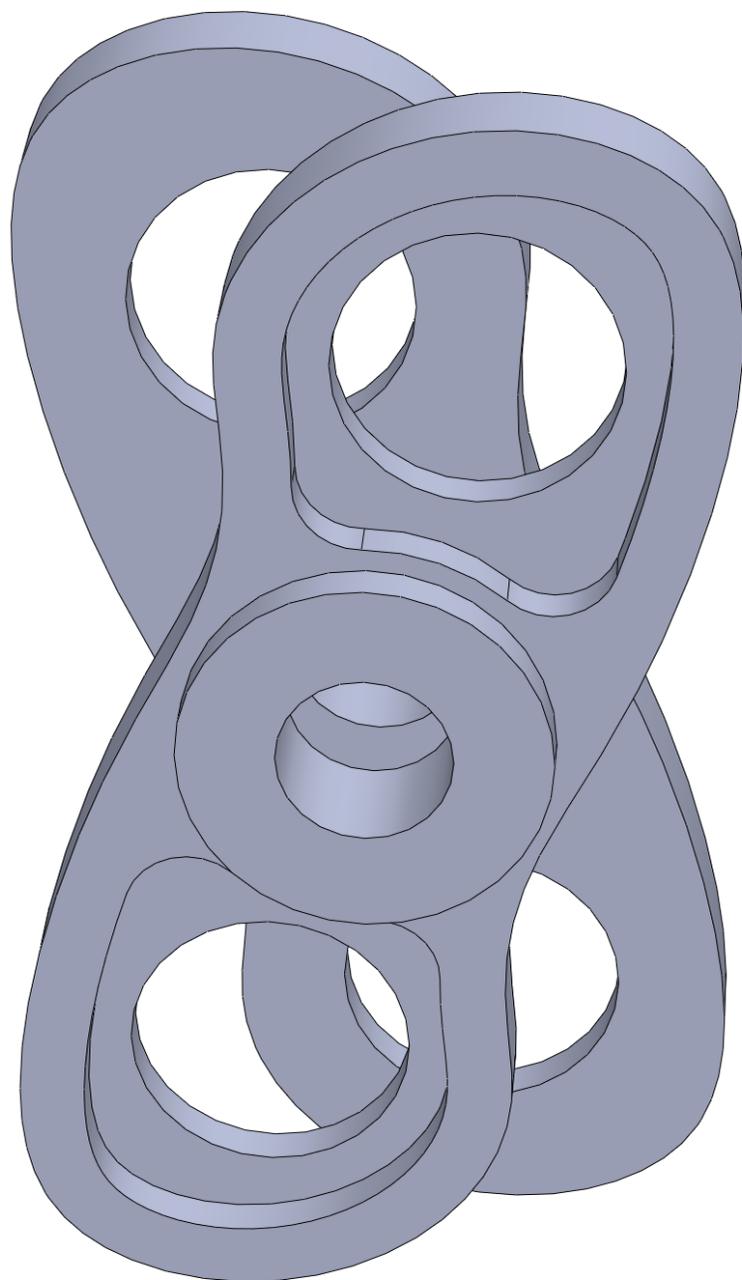


Instructions for modelling the Cams in Solidworks



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Step by step instructions

The cams for the Marchetti cam action engine are drawn with the help of a spline or curve through a set of X-Y-Z coordinates. These coordinates are calculated with the help of a mathematical formula which is explained in a separate document on <http://marchetti-engine.com>. The dimensions used in this demonstration are an approximation of the historical engine geometry. The true historical dimensions are unfortunately unknown.

Drawing the cam curve

To get started, create a new CAD model part and save it in a project folder. This PDF document has a text file embedded inside it, you can access it by clicking or double-clicking on [this link](#) (also available on the website). Now you can save the text file to your project folder. In Solidworks, use *Insert* → *Curve* → *Curve Through XYZ Points*:

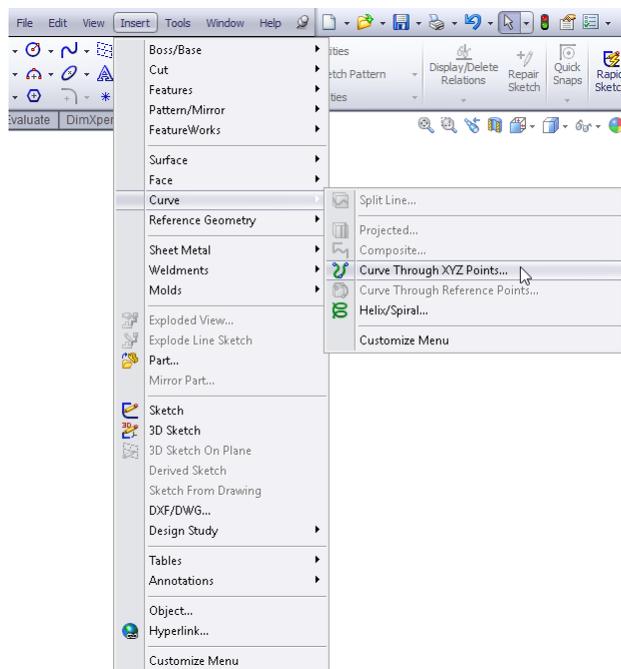


Figure 1: Solidworks user interface: the insert curve option

In the menu 'Curve File' that pops up, select the file type 'Text Files (*.txt)' and open the text file from the previous step. The confirmation should look like the following:

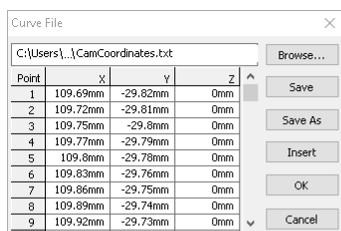


Figure 2: Solidworks user interface: confirming the curve

The next step is to open a sketch on the front (X-Y) plane. After opening the sketch, press the



button and select the curve. This convert the curve to a sketch line that can be used as part of the sketch. Now you can draw a circle at exactly the origin for the main engine axis. Your sketch should now look as shown in Figure 3 below:

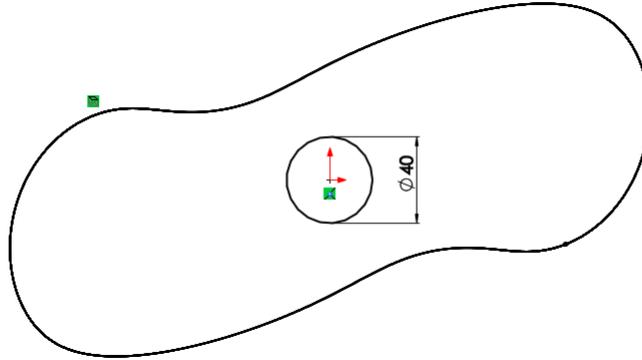


Figure 3: Solidworks user interface: using the curve in a sketch

The sketch can now be extruded, for example 20mm . The original curve can be hidden because it is not needed any more.

Creating a simple Marchetti cam assembly

Before creating the assembly, the rocker arms and wheels have to be created with the dimensions shown in Figures 4 below. The same thickness of 20mm can be used.

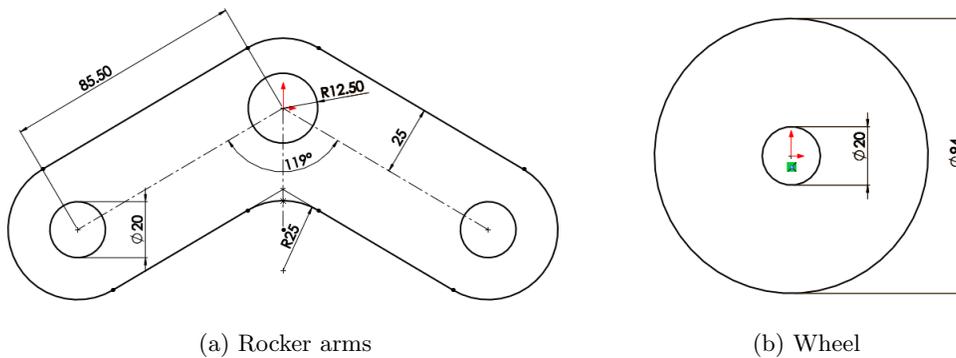


Figure 4: Solidworks geometry of the rocker arms and wheel

While creating a new Solidworks assembly, do not insert the parts yet. A sketch is first made on the front plane to define the pivoting points of the rocker arms:

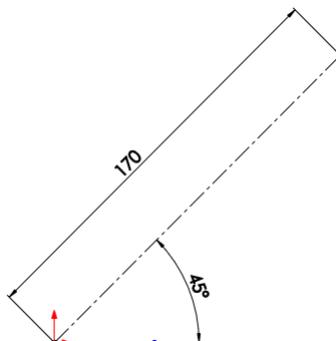


Figure 5: Solidworks user interface: the first sketch in the assembly

Insert the first cam and don't forget to unfreeze it by clicking right mouse and selecting 'float'. Create the first mate that sets the front face of the cam coincident to the front face of the assembly. Next, create a concentric mate between the origin and the hole inside the cam. The assembly with just the cam should look as follows:

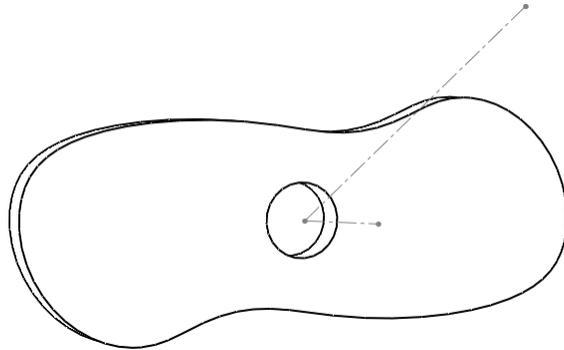


Figure 6: Solidworks user interface: inserting the first cam

Next, insert the second cam and apply the following two mates:

- Concentric mate between the holes in both cams.
- Distance mate between the cams, equal to the thickness of the rocker arms (20mm)
- Coincident mate between the top plane of cam 1 to the top plane of cam 2

The assembly should now look as follows:

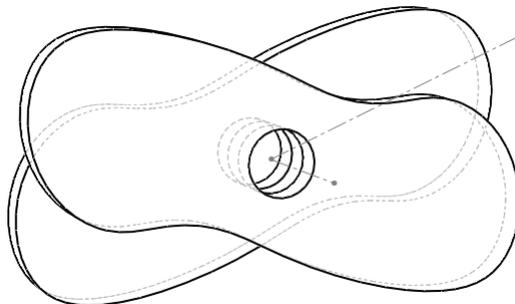


Figure 7: Solidworks user interface: assembly of the cams

The next step is to insert rocker arms and two of the wheels, place the parts in the correct position with the following mates:

- Parallel or coincident mate between the front faces of the wheels and rocker arms with respect to the front face of the assembly.
- Concentric mate between the endpoint of the sketch line drawn in Figure 6 and the pivoting point of the rocker arms.

The final step is to create a cam mate between the cam and the wheels (cam follower). Press the  and scroll down to Cam mate. First select the curved face of cam 1 and the circular face of the wheel that rolls over this face. Before confirming the mate, your selection should look as follows:

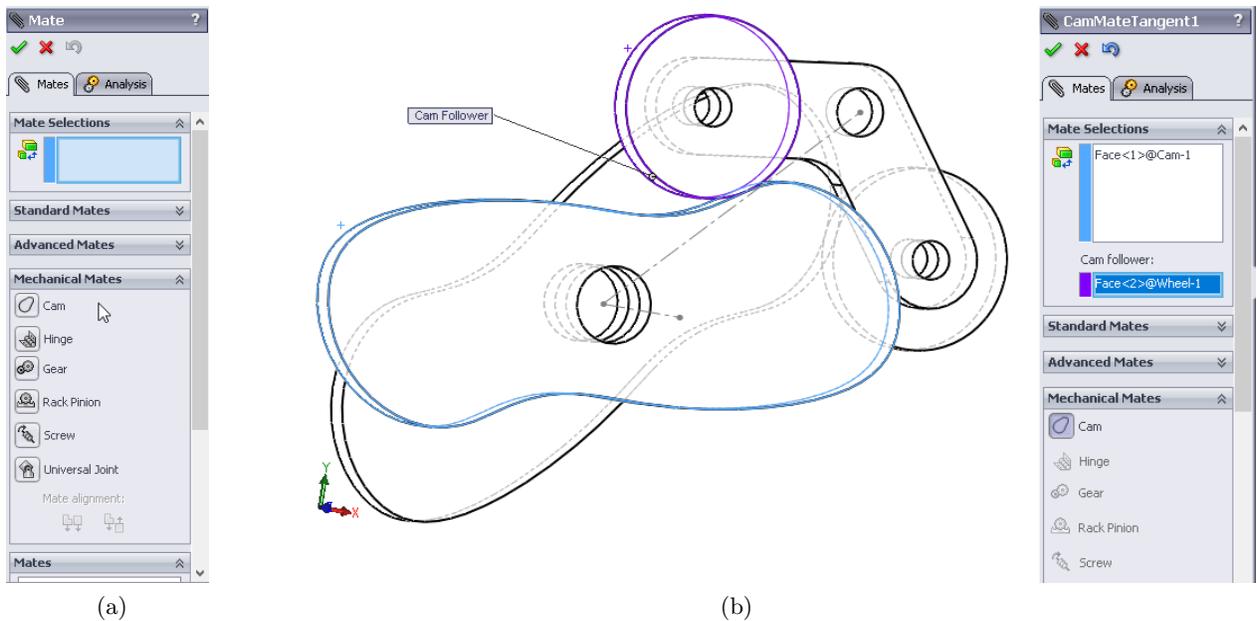


Figure 8: Solidworks user interface: Selecting and applying the cam mate

After confirming the mate, try to move the cam assembly and verify that the wheel properly follows it. A problem that often occurs is that the wheel suddenly starts following the inside of the curved cam face instead of rolling on the outside. It may be difficult to prevent the cam follower mate from snapping to wrong side, but sometimes it helps to press the  button after that mate snapped to the wrong side. If your assembly has become more complicated and it happens again, then the easiest solution is pressing the undo button after the cam follower mate snaps. The model should look as follows:

Figure 9: Solidworks user interface: Simple cam demonstration

Please note that the pivoting point distance ($170mm$), the arm length ($85.5mm$) and the wheel diameter ($94mm$) can not be altered. This would require a new set of X-Y-Z coordinates to be calculated from the new parameters and then re-imported. Instructions for calculating and exporting the X-Y-Z coordinates will be available on <http://marchetti-engine.com>.

Good luck!