ABOUT SOLIDWORKS MODELING ADVANCED FEATURES

Prof. Cătălin Iancu, *Constantin Brâncuşi University of Târgu-Jiu, ROMANIA*

ABSTRACT: *In this paperwork are presented some SOLIDWORKS advanced features for modeling special items as auger gear and the steps to be taken in order to use these features for better design. There are presented the features that are used for modeling, and then the steps that must be taken in order to obtain the 3D model of a coil and the unfolded coil for subsequent sheet metal cutting.*

KEY WORDS: SOLIDWORKS, advanced features, Sheet Metal, Helix/Spiral, Loft/Lofted Bend.

1. INTRODUCTION

In [1], it has been presented some of SOLIDWORKS advanced design features, such as SOLIDWORKS Configuration facility, which allows create multiple variations of a part or assembly model within a single document. So Configurations facility is providing the tools for developing and managing families of models with different dimensions, components, or other parameters.
Besides these useful features, in the main module of SOLIDWORKS are also other built-in features that permit modeling of special items as *auger gear* or so called *feed screw*.

In the following will be presented the method for modeling a special *auger gear*, used in an industrial dusting filter for cement industry. The modeling is needed for obtaining the unfolded coil, made of S235 JR structural steel, according to [2] - SR EN 10025.
The unfolded coil is needed for sheet metal cutting, then rolling and welding for obtaining the whole auger gear.
For modeling will be used features as *Helix/Spiral* and *Loft* (Lofted bend) of *Sheet Metal* module.
In figure 1 is presented the auger gear that must be modeled.

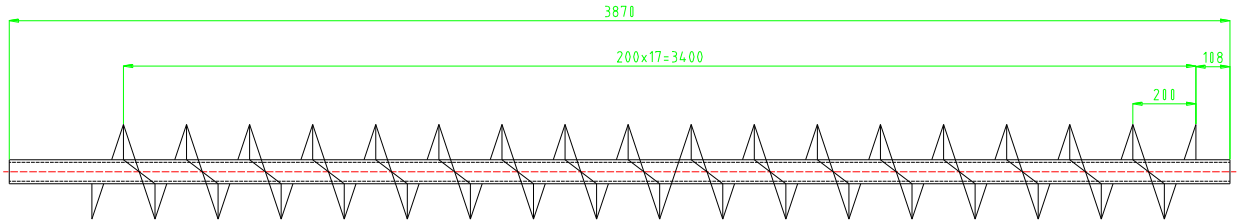

Fig.1. Auger gear (feed screw)

2. FEATURES USED [3]

Helix/Spiral – is a 3D curve obtained by specifying a circular sketch, pitch, number of revolutions, and height.

For creating a helix or spiral curve in a part, one must following steps:
- In a part, do one of the following:
  - Open a sketch and sketch a circle.
  - Select a sketch that contains a circle.

The diameter of the circle controls the starting diameter of the helix or spiral.

- Click Helix and Spiral (Curves toolbar) or Insert > Curve > Helix/Spiral.
- Set values in the Helix/Spiral Property Manager.
- Click .

Loft - creates a feature by making transitions between profiles. A loft can be a base, boss, cut, or surface.
One can create a loft using 2 or more profiles. Only the first, last, or first and last profiles can be points. All sketch entities, including guide curves and profiles, can be contained in a single 3D sketch.

Sheet Metal feature tab – Sheet metal parts are generally used as enclosures for components or to provide support to other components. A sheet metal part can exist on its own without any references to the parts it will enclose, or one can design the sheet metal part in the context of an assembly that contains the enclosed components.

Lofted Bends - Lofted bends in sheet metal parts use two open-profile sketches that are connected by a loft.
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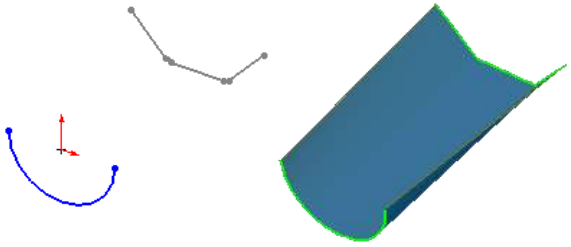

Fig. 2. Lofted bends

Characteristics of lofted bends are:
-Cannot be mirrored;
- Require two sketches that include:
- Open profiles without sharp edges;
- Aligned profile openings to ensure flat pattern accuracy;
-Profile segments in each sketch must be the same type.

3. MODELING ONE COIL

The modeling of the auger gear starts with a simple sketch of two circles that will control the starting diameter of the inner and outer helix or spiral.

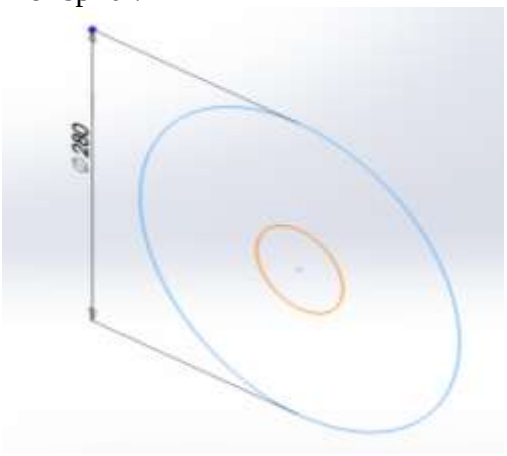

Fig. 3. Starting sketch

The next step is the creation of outer helix (one coil) – figure 4, and the inner helix – figure 5.

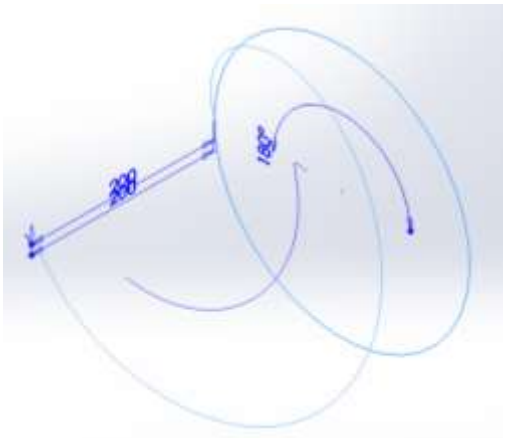

Fig. 4. Outer helix

Fig. 5. Inner helix

The 2 spirals have been defined by pitch and revolution, as seen in figure 6.

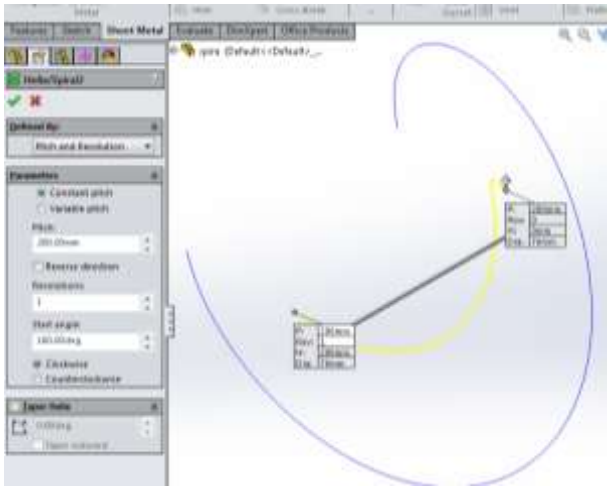

Fig. 6. Helix/Spiral definition

Fig. 7. 3DSketches from Helixes (3D Curves)

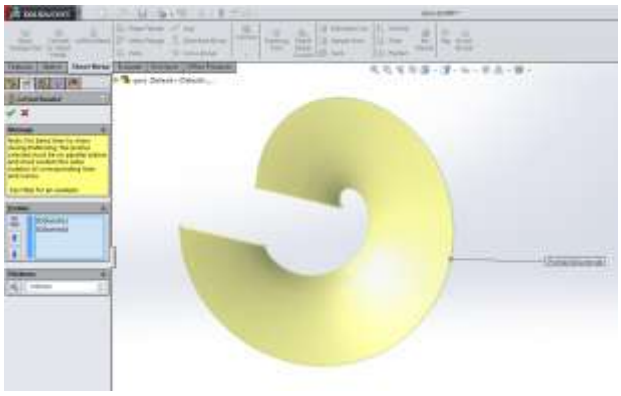

Fig. 8. Lofted Bends for specified thickness

- Next step is to use Flatten feature on **Sheet Metal** tab to obtain the unfolded coil of auger gear, as seen in figure 9.

At this point of modeling we must have a little discussion. In order to obtain one unfolded coil, needed for sheet metal cutting, it will be used the **Sheet Metal** module.
By using **Sheet Metal** module, in the end we'll be able to use **Flatten** feature to obtain

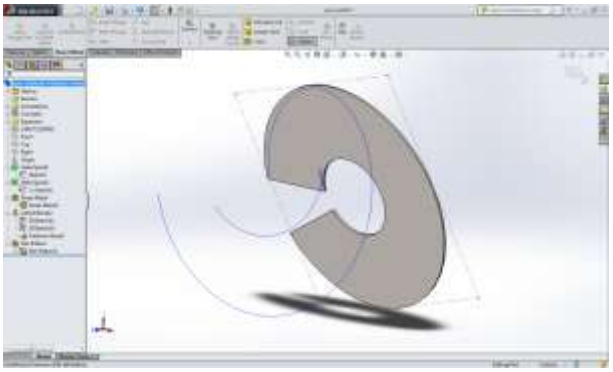

Fig. 9. Unfolded coil – Flatten feature

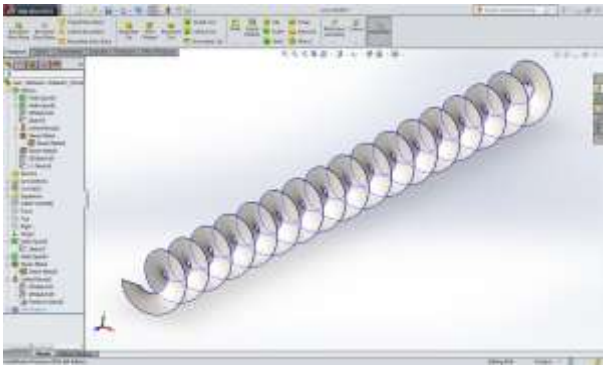

Fig. 10. Model of auger gear

This unfolded coil is used for the drawing needed for cutting pieces out of a sheet metal in order to fabricate the auger gear.

5. CONCLUSIONS

As it was presented, SOLIDWORKS presents a series of advanced design features intended to optimizing design, such as SOLIDWORKS Configuration facility, which allows create multiple variations of a part or assembly model within a single document. Besides these useful features, in the main module of SOLIDWORKS are also other built-in features that permit modeling of special geometry items as *auger gear*.

For this purpose were used features as *Helix/Spiral* and *Loft* (Lofted bend) of *Sheet Metal* module. Following the steps presented, one can obtain the geometric model, used for sheet metal cutting, then rolling and welding for obtaining the whole auger gear.

In the same time the geometric model of the auger gear can be used for various analyses, like stress analysis (SimulationXpress), Costing, Sustainability, or any other Geometry analysis, so the possibilities for optimizing design are very broad, so one can save time and money for a quite consuming activity – CAD/CAE/CAx.

REFERENCES

[1]. Iancu C., *About SOLIDWORKS Configurations for design*, CONFERENG 2015, International Conference of Enginering Faculty of "Constantin Brâncuşi" University, Târgu-Jiu, University Annals no.3/2015, Engineering series, ISSN 1842-4856, pp.79-85

[2]. SR EN 10025+A1:1994 "Hot rolled products of non-alloy steels. Technical conditions of delivery"

[3]. Lombard, M., Solid Works Bible, Wiley, USA, ISBN 978-1-118-50840-4, 2013

[4]. SolidWorks Advanced Modules, Dassault Systèmes SolidWorks Corporation, Waltham, MA, USA, 2014.