ABOUT MODELING LARGE ITEM WITH SHEET METAL LOFT FEATURE IN SOLIDWORKS

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ABSTRACT: *In the present paperwork is presented the method used for modeling a Sheet Metal part in SOLIDWORKS, in order to achieve a faster and more reliable design. There are presented step-by-step the features used for initial modeling, and then, using the features characteristic of Sheet Metal module, obtaining the whole sheet part and also the unfolded part, exported for subsequent sheet metal cutting.*

KEY WORDS: *SOLIDWORKS, Sheet Metal, Lofted Bends, Flatten*

1. Introduction

In a previous study [1], it was presented a modeling strategy for a complex 3D item, an auger gear, by using some of SOLIDWORKS advanced design features, with reference to *Sheet Metal* feature.
In [2] was presented an example of modeling a sheet metal part, a roll support, used for conveyor belts of mining equipment. The support was modeled with common features but in the end it was used features of *Sheet Metal* module, for obtaining the unfolded part.
In this paper will be presented the modeling strategy for a large item, needed in a large installation. The item, made of S355J0 structural steel, or ST 53-3U, according to SR EN 10025 [3] (former OL52-3k, according to Romanian old standard), is shown in figure 1.
It can be seen that the item length is over 8000 mm.

2. Modeling the part

The item is made of sheet metal of 5 mm, with different curves on the two ends, so it is obvious the use of *Sheet Metal* feature of SOLIDWORKS.

The modeling of mentioned item starts with creation of two planes situated at right distance (figure 2), that will be used for creation of end section of item. The next steps use this two planes, but for further precision can be inserted some intermediate planes, each with corresponded section.

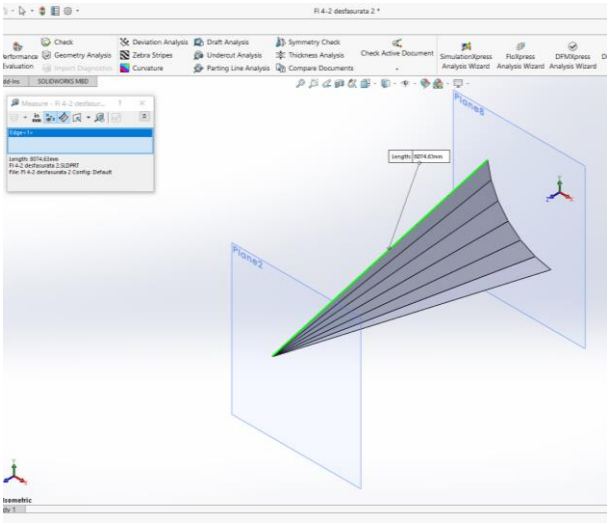

Figure 1. The item modeled with *Sheet Metal*

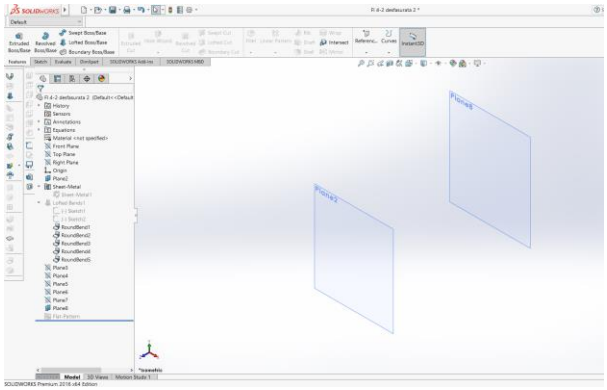

Figure 2. The two initial planes

The next step is a creation of the two end section, dimensioned according to documentation, in the two planes created previously.
In figure 3 and 4 are shown these two end-sections.

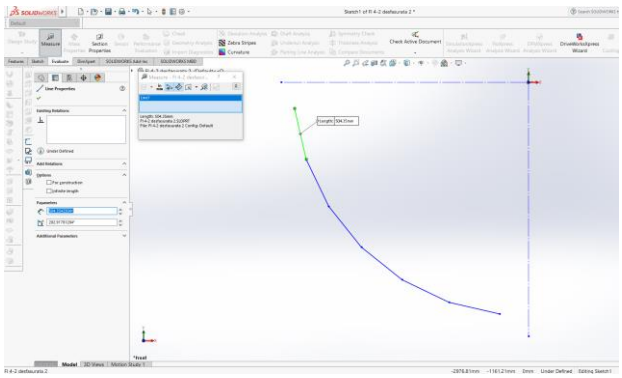

Figure 3. Sketch1 – the first end-section

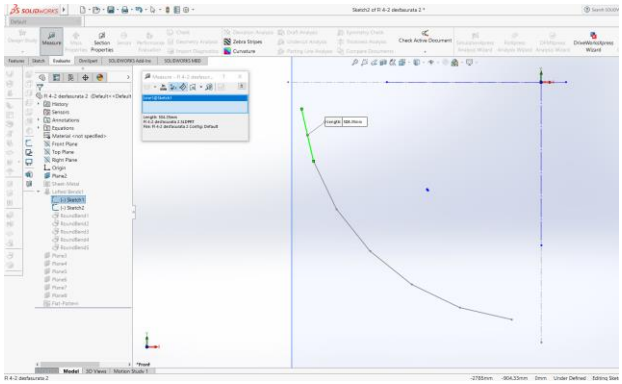

Figure 4. Sketch2 – the second end-section

Next steps will use the *Sheet Metal* feature of SOLIDWORKS.
This can be done by accessing **Sheet Metal** feature tab, or by using Insert menu, as shown in figure 5.

A sheet metal part is generally used as a single independent item or as enclosures for other components or to providing support to other components in an assembly.
As stated in [4], "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" [4].

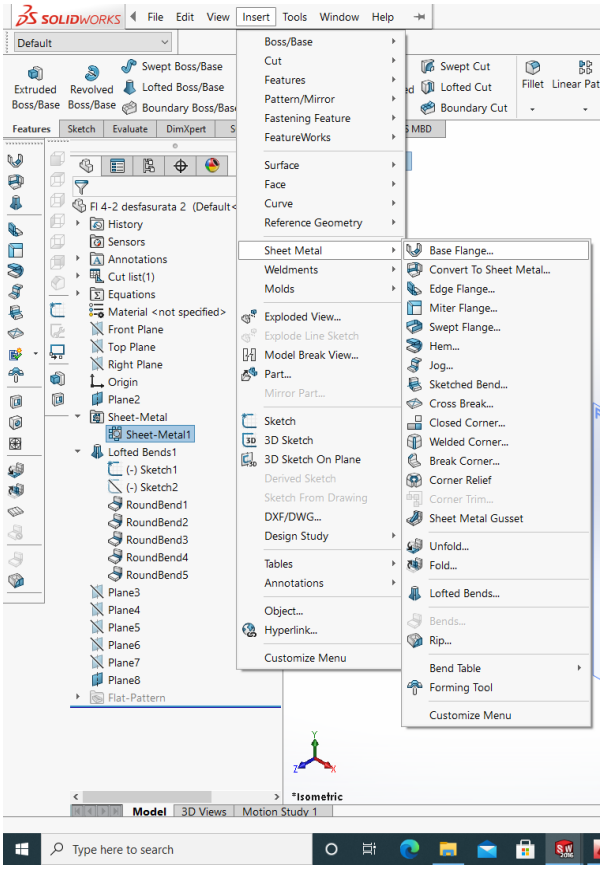

Figure 5. Insert/*Sheet Metal* menu

In [5] there is a well documented presentation of Sheet Metal feature, with good examples. Based on that, it will be used the *Lofted bends* feature, as the end-section of item are different.
Accessing *Lofted Bends* can be done by using **Sheet Metal** feature tab, or by using Insert menu, as seen in figure 6.
The result of using Sheet Metal/Lofted bends feature is a dialog cassette, presented in figure 7. In this dialog one can specify:

- the two sketches between will be generated the lofted bends;
- number of bends;

- the facet value;
- sheet metal parameters as thickness of sheet and minimum bending radius
-

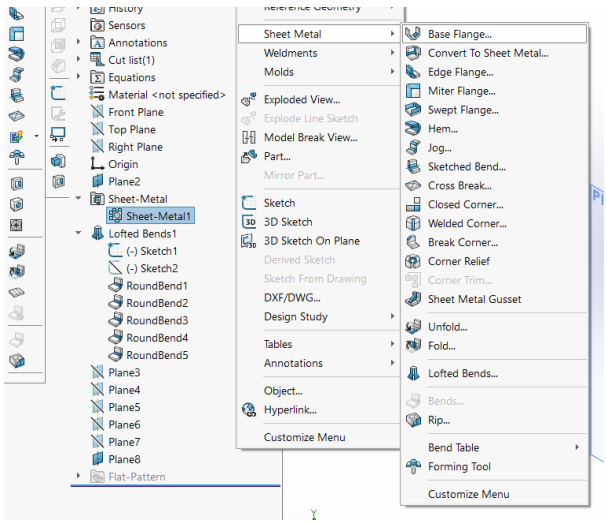

Figure 6. *Sheet Metal/Lofted bends* menu

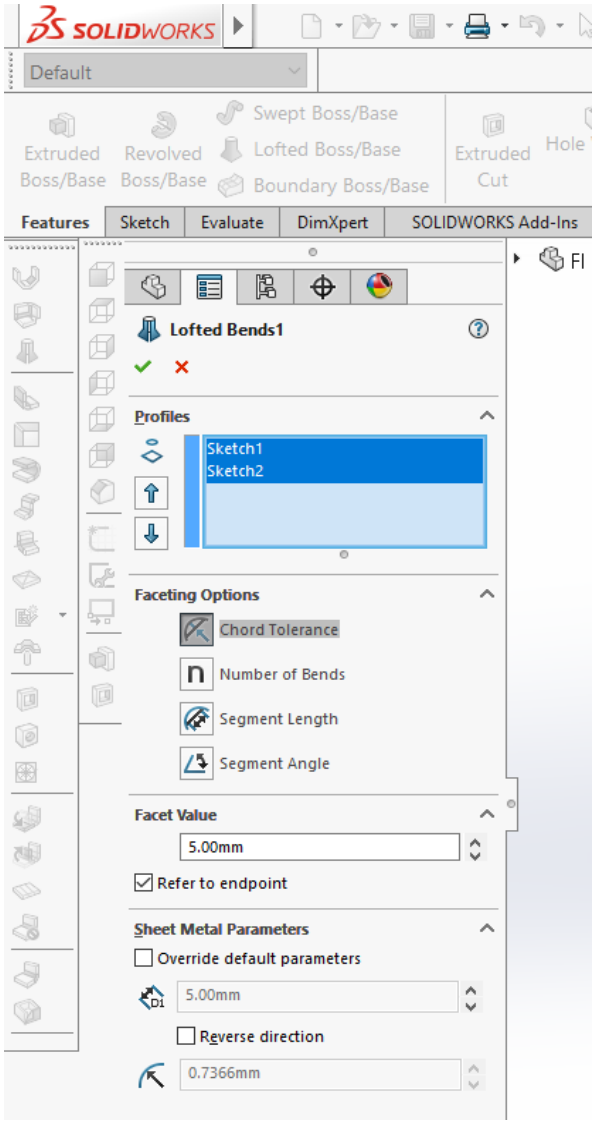

Figure 7. *Lofted bends* options

The result of *Lofted Bends* feature, meaning obtaining the 3D shape of the item modeled is presented in figure 8.

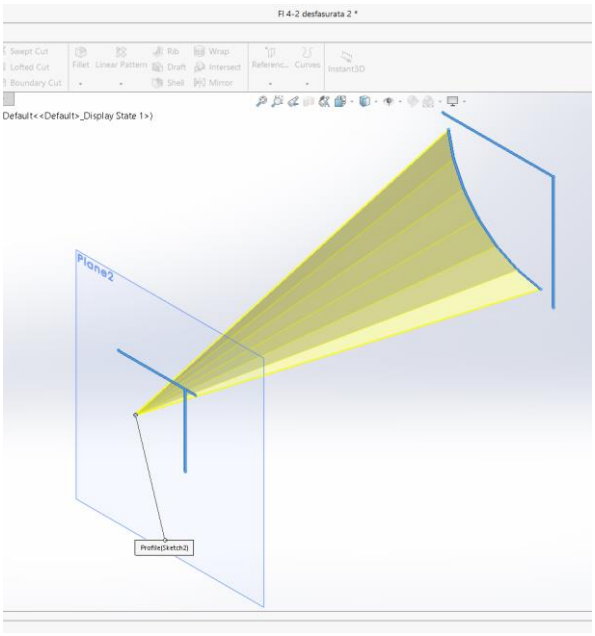

Figure 8. *Lofted bends* result

In figure 9 is presented a detail of the small end of the item, where can be observed the bends and the thickness of sheet.

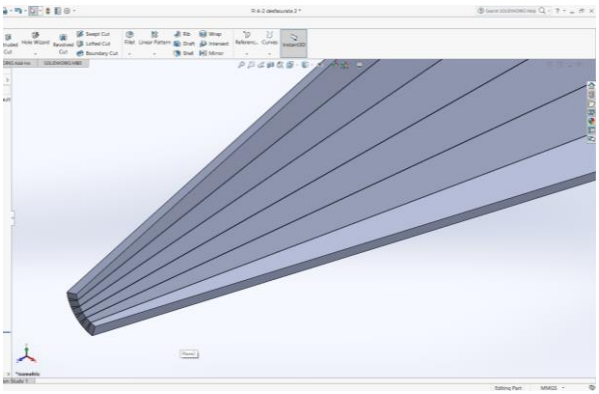

Figure 9. Detail of bended item

The last step is to use *Flatten* feature on **Sheet Metal** tab to obtain the unfolded item of sheet metal, as shown in figure 10.

The shape and dimensions of unfolded piece of sheet metal can be exported as .dwg or .dxf files, used for cutting pieces out of a sheet metal in order to cut the piece of a large sheet, or to compose the item from smaller piece of sheet metal, by a cutting machine.

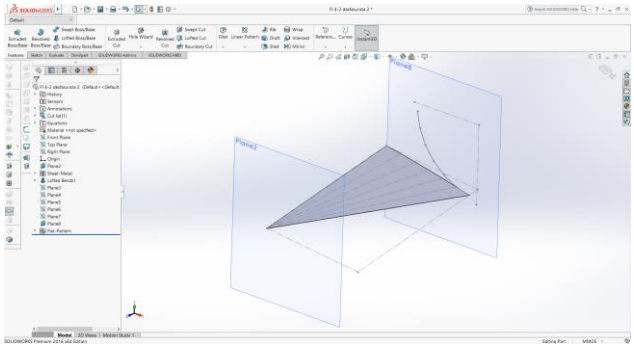

Figure 10. Unfolded idem, using *Flatten* feature

3. CONCLUSIONS

As it can be seen, SOLIDWORKS software has a series of advanced design features in the core of the software, intended to help faster modeling and optimizing design.

For modeling this item, made of sheet metal, the following steps were taken:

- *Plane* creation for the two end planes of the item;
- *Sketch*, to create the 2 sketches needed for subsequent modeling;
- *Sheet Metal* module/tab and *Lofted Bends* feature for obtaining the item made of sheet metal;
- *Sheet Metal* module/tab and *Flatten* feature to obtain the unfolded support.

Following these steps, and using the feature presented, especially *Sheet Metal* module, it can be obtained the correct geometric model of modeled bended item, shown in figure 11. In the end is obtained the unfolded shape, used for subsequent sheet metal cutting, then mechanically bended and welded in its assembly, saving time and potential errors in calculating the correct unfolded part.

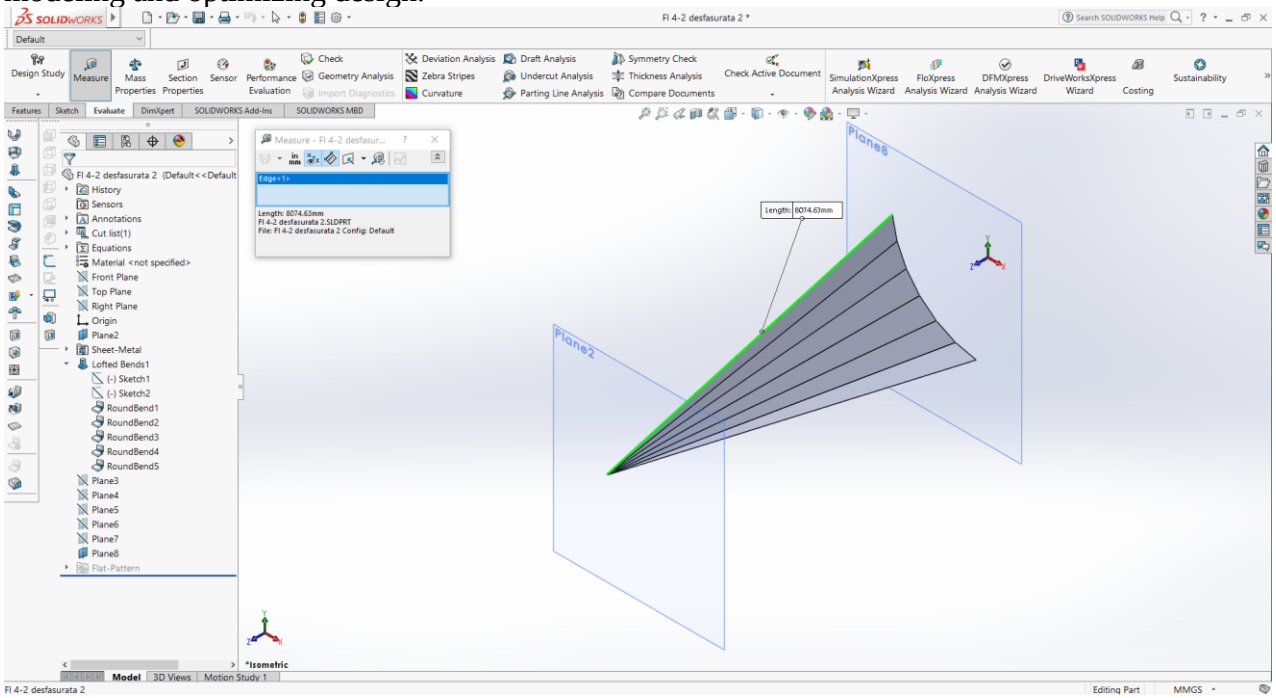

Figure 11. The large item modeled with *Sheet Metal* feature

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