ABOUT WELDMENTS FEATURE IN SOLIDWORKS

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Abstract: *In this paperwork are presented, based on example, details on Weldments feature in SOLDWORKS software. This feature allow the designer to model a weldment structure based on a 2D or 3D sketches for defining the basic structure of the part. The weldment structure is created by adding structural members joint in groups of similar members. By following steps presented here the designer can easily model even complex welded structures.*

Keywords: SolidWorks, weldment feature, structural members, weld beads.

1. Introduction

As discussed in [1] every new version of SolidWorks introduces new modules or expand the capabilities of existing ones. So as described in [2] and [3], from SOLIDWORKS 2012 were added various modules which increase the possibilities of a designer to realize complex models, with various features. These modules can lead to optimizing the design and even make a given manufacturing process to become more efficient.

Among these modules/features can be mentioned Sheet Metal, Machining, Casted Parts, Multi-Body, 3D Printed Parts and Weldments.

"The Weldment feature is not a feature in the same sense as a Fillet or an Extrude. Instead, it sets up a design functionality environment. The weldment feature designates the part as a weldment and enables the weldment environment" [4].

For using weldment feature the designer can use a 2D or 3D sketch to define the framework of future structure, and then insert one by one the structural members, joint in groups of similar members.

After that can be added items as gussets and end caps, using dedicated tools on Weldments toolbar.

After creating the first structural member in a part, a weldment feature is created and added to the FeatureManager design tree. SolidWorks also creates two default configurations in the ConfigurationManager: a parent configuration Default<As Machined> and a derived configuration Default<As Welded> [3].

If the profile chosen for structural member has a material assigned, the option Transfer Material from Profile make possible to transfer the material from the profile to the weldment created.

2. Weldment feature – details and capabilities

For beginning working with Weldment feature, the part file is open and then the sketch for future weldment structure is done, click on menu Insert > Weldment (fig.1) or the Weldment icon (on Weldment toolbar) (fig.2).

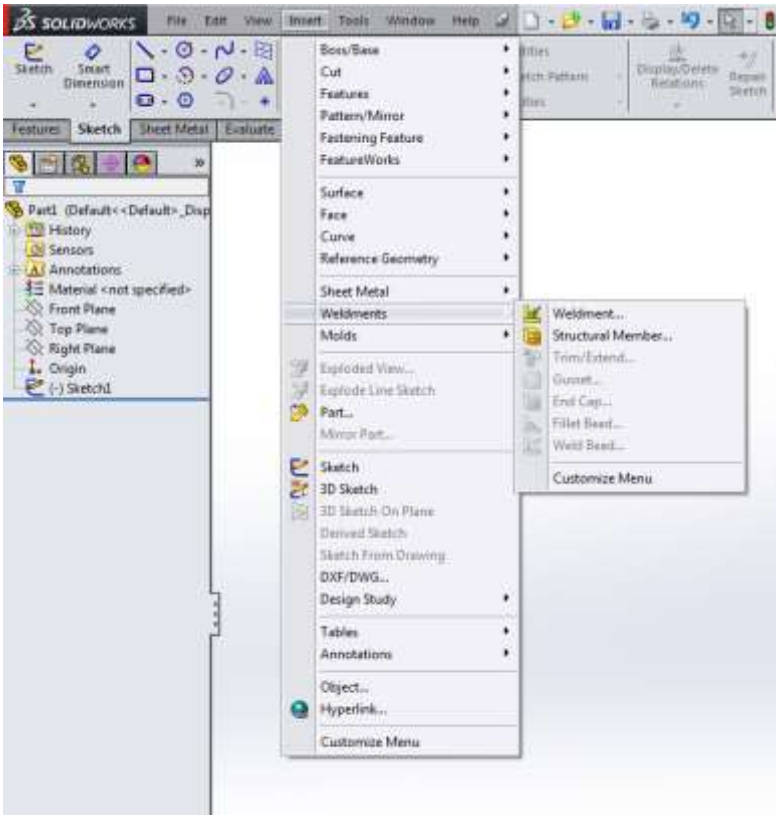

Figure 1. The menu Insert>Weldments>Weldment

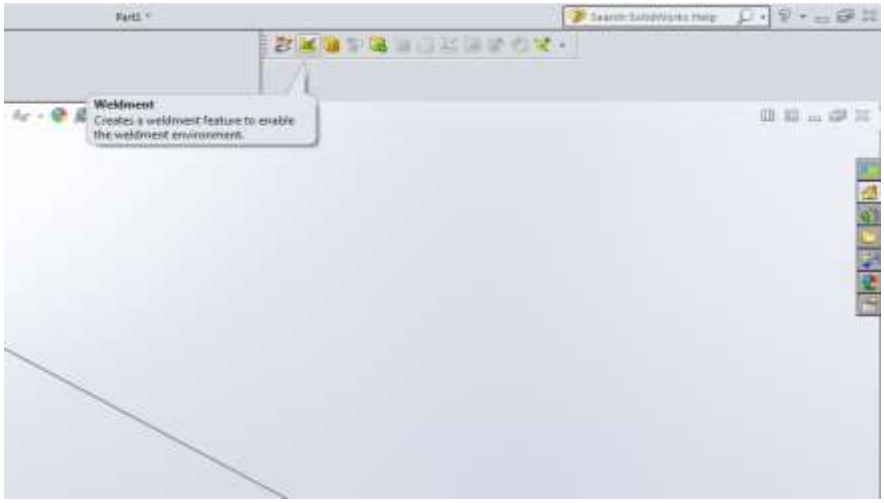

Figure 2. The Weldments toolbar

At this moment, the *Task Manager tree* appears in the *left* with Weldment feature shown, waiting for structural members to be inserted [4].

Using the proper icon or Insert>Weldments>Structural members, one by one the items needed to structure are added, selecting the profile and sketch segments to define the path for the structural member. Some additional parameters must be specified for correct insertion of structural member (fig. 3).

Groups are a collection of related segments of structural members. Types of groups are *Contiguous* (joined segments) or *Parallel* (parallel segments).

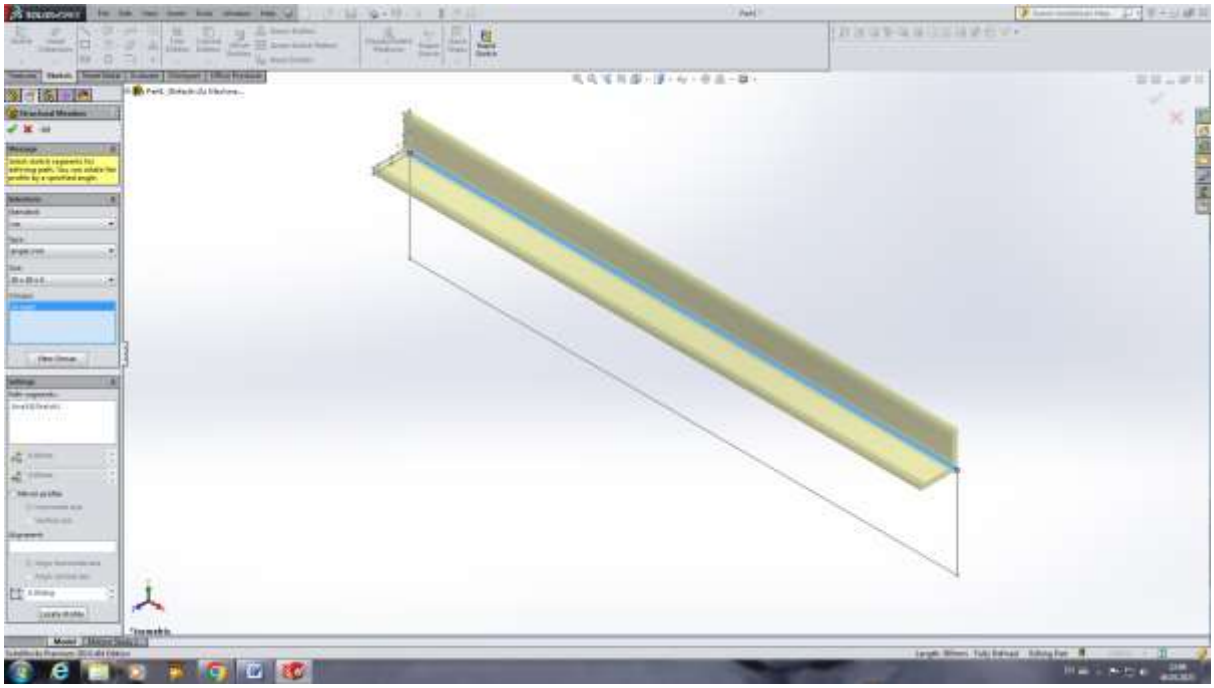

Figure 3. Inserting structural members

Structural member – add or edit a structural member in a weldment part. The designer must specify the profile of the structural member by selecting a standard, type, and size. Some of the profile types are: angle iron, rectangular tube, pipe, c channel.

Pierce point - defines the location of the profile, relative to the sketch segment used to create the structural member. For changing the pierce point (insertion point) edit structural member and clicking *Locate profile* and select the desired point on sketch for alignment.

Trim/Extend – let designer specify how the ends of structural members will be trimmed/ extended for correct position in welded part. The options are: End trim, End miter, End butt 1, End butt 2, with icons that show how the trim will look like.

Gussets – are items that reinforce the area between two intersecting structural members with planar faces. There are two types of gussets: triangular and polygonal (fig.4)

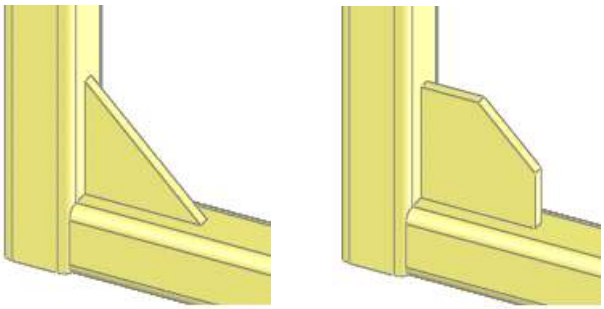

Figure 4. Types of gussets [4]

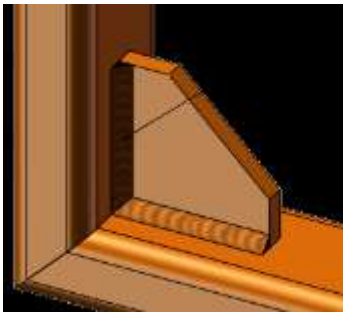

Figure 5. Adding fillet beads [4]

End caps – To close off open structural members, it can be added end caps, including internal end caps. It can be added also *fillets* or *chamfers* to all end caps.

Weld bead – permits to add weld beads to weldment parts and assemblies and automatically create weld symbols.

Fillet bead – permits to add full length or intermittent fillet weld beads between any intersecting weldment entities such as structural members, plate weldments, or gussets (fig.5).

Pattern and Mirror support – The designer can pattern and mirror structure system members using the Linear Pattern, Circular Pattern, or Mirror tools [5].

3. Weldment feature – example of welded structure

In this section it will be presented an example of a welded frame needed for mining equipment, as a subassembly. The frame is made of USt-52-3U, DIN 17100, and the equivalent chosen in SOLIDWORKS is S355 J0 steel.

To begin the modeling the part file is open and then is done the 3D sketch for future weldment structure (fig.6). The 3D sketch is fully dimensioned, for future verification of structural members inserted.

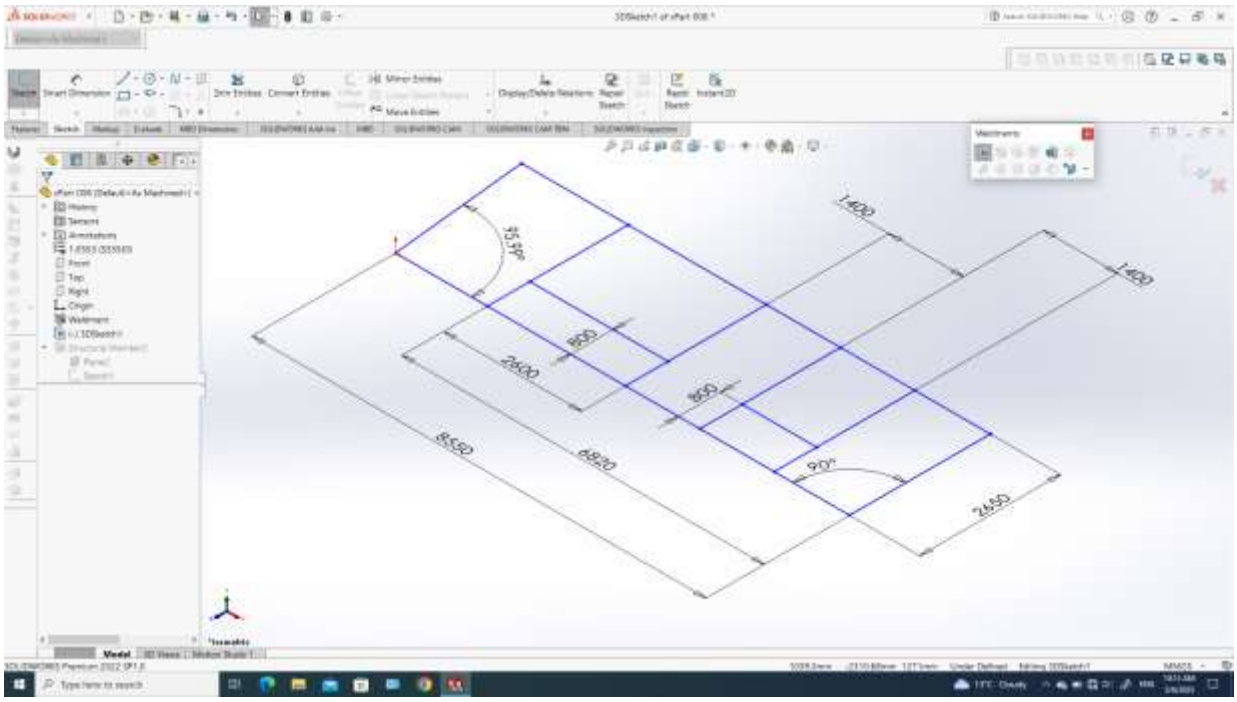

Figure 6. 3D sketch of structure

Next, one by one, the *structural members* are added, according to technical documentation of mining equipment modeled. It will be used as type *sb beam*, selected from the library provided in SolidWorks, after selecting the right standard (ISO, ANSI) (fig. 7). Additional weldment profiles are available on the Design Library tab, under SOLIDWORKS Content, where can be downloaded as .zip files

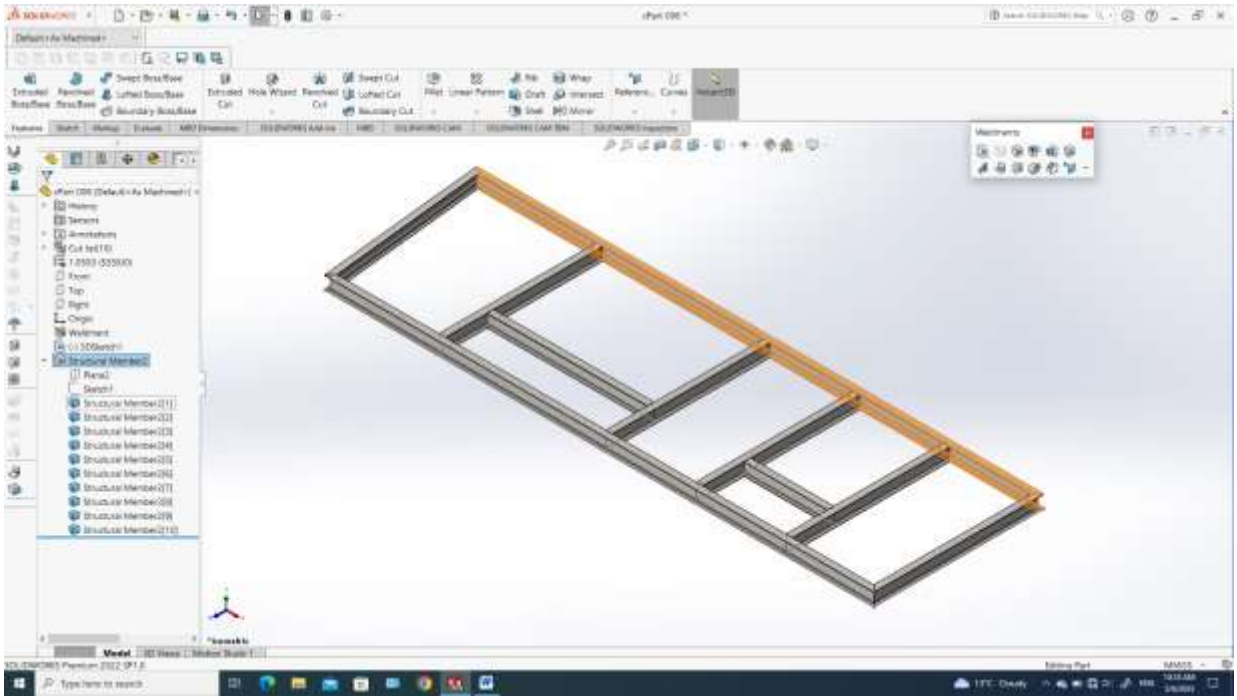

Figure 7. Adding structural member

After adding all structural members needed, and creating the correspondent groups, can be made adjustments by Trim/Extend feature and Weld bead/fillet bead to the joint areas (similar to fig.5), (figure 8).

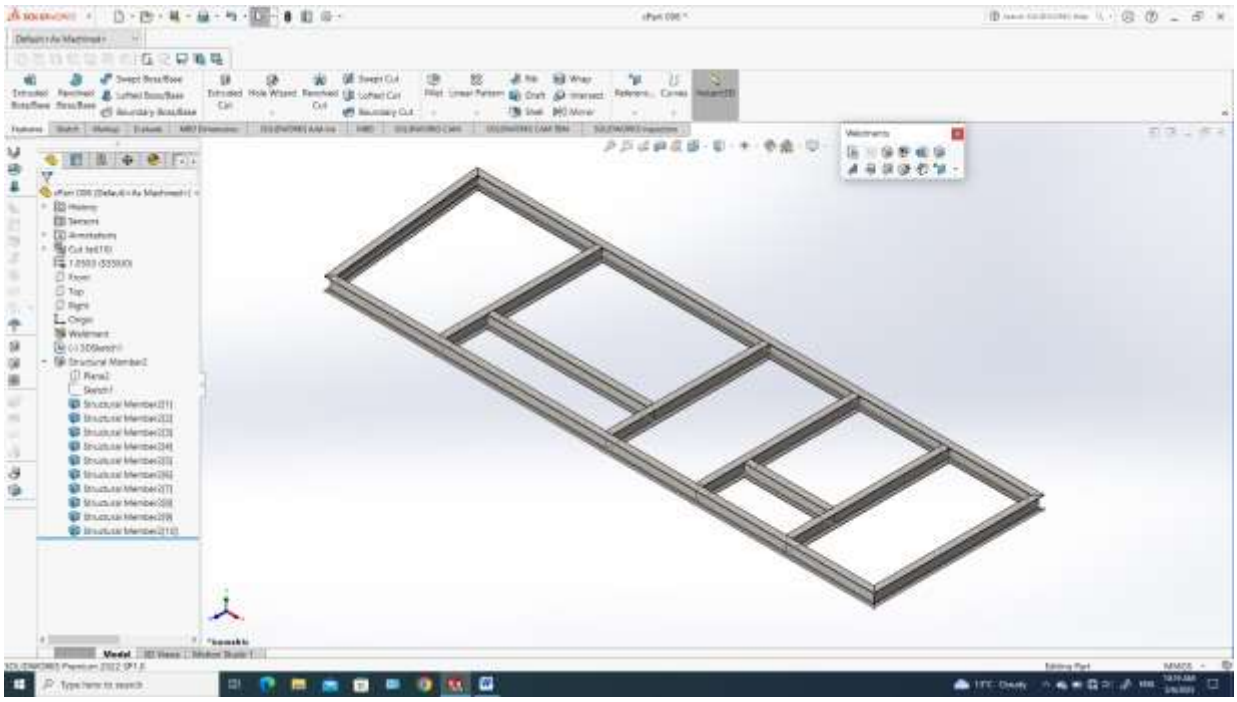

Figure 8. All structural members inserted

In the end the designer can use the weldment *cut list table* to add a BOM-like (Bill Of Materials) table for cut, welded structural shapes. This *list table* (fig.9) can be inserted in a future welded structure drawing.

ITEM NO.	QTY.	DESCRIPTION	LENGTH
1	2	TUBE, SQUARE 30 X 30 X 2.60	334.8
2	2	TUBE, SQUARE 30 X 30 X 2.60	274.8
3	1	C CHANNEL 80 X 8	285.2
4	2	TUBE, SQUARE 30 X 30 X 2.60	884.4
5	4	END CAP, 28.7 SQUARE	

Figure 9. Example of weldment list table

4. Conclusions

As presented here step-by-step, SOLIDWORKS *Weldment feature* can be used by any designer to model a simple or complex welded structure. First it is used a 2D or 3D sketch to define the framework of future structure, and then insert one by one the *structural members*, joint in groups of similar members.

The structural members can be trimmed/extended as needed to obtain the right structure. After that can be added items as *gussets* and *end caps*, using dedicated tools on Weldments toolbar. Also can be added *Weld bead/Fillet beads* to weldment parts and assemblies and automatically create weld symbols.

As a conclusion, by following the steps and the settings described here, and making all the necessary changes, as presented in [6] and [7], the designer can model a future welded 2D/3D structure, saving time and money by using SOLIDWORKS *Weldment feature.* This can be useful to estimate profiles needed and costs before effectively start the real production, and thus make decisions that can make a specific industrial process more economic and productive.

References

[1]. **Iancu C**., *About SolidWorks modeling advanced features,* CONFERENG 2016, Târgu-Jiu, University Annals no.4/2016, Engineering series, ISSN 1842-4856, pp.162-167

[2]. **Iancu C**., *About SimulationExpress module features*, Fiability & Durability Revue, ISSN 1844-640X, 1/2015

[3]. **Iancu C.,** *About SolidWorks sustainability module capability*, Fiability & Durability Revue, ISSN 1844-640X, pp.229-235,1/ 2014

[4]. *SolidWorks user help*, Dassault Systèmes SolidWorks Corporation, Waltham, MA, USA, 2014-2022.

[5]. *SolidWorks Advanced Modules*, Dassault Systèmes SolidWorks Corporation, Waltham, MA, USA, 2012 - 2016

[6]. **Lombard, M.,** *SolidWorks Bible*, Wiley, USA, ISBN 978-1-118-50840-4, 2013

[7]. Paul Tran, *SOLIDWORKS 2020 Advanced Techniques*, SDC Publications, ISBN 978-1630573164, 2020