



What's New

PowerPack for Inventor 2025.1



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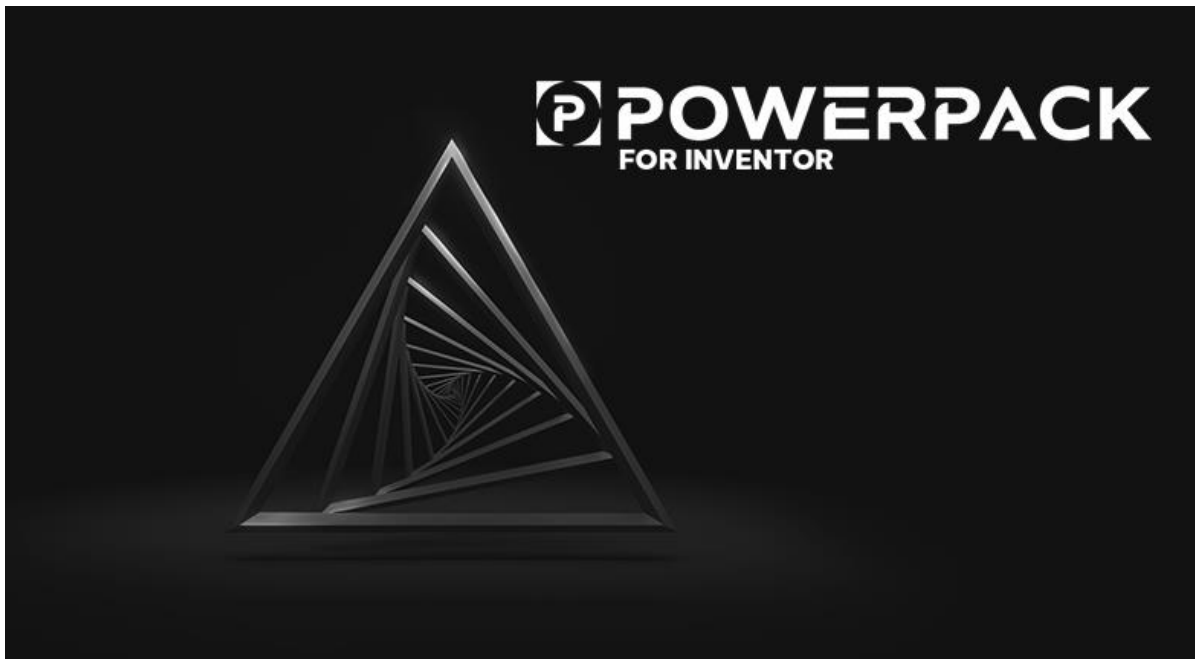
1. GENERAL INFORMATION ABOUT VERSION 2025.1

In 2025.1 version of the **PowerPack for Inventor**, we are introducing 2 new functionalities like **Undercut within a part** and **Grid Annotations**, alongside improvements to the existing features like **BOM to Excel**, **Unresolved References** and **Switch ON/OFF**.

1.1. COMPLIANCY WITH AUTODESK INVENTOR 2025

Due to an upgrade to .NET 8 of Autodesk platforms, all plugins are updated accordingly. This has an impact in the compatibility, that cannot be assured backwards. Therefore, the **PowerPack for Inventor 2025.1** is compliant **only** with **Autodesk Inventor 2025 version** and its updates.

The PowerPack for Inventor 2025.1 can be installed using the Setup PowerPack for Inventor 2025.1. The setup will install version 2025.1 of the PowerPack on Inventor 2025.



2. NEW COMMANDS

In the **2025.1 version of the PowerPack for Inventor**, our focus has been to simplify and speed up various processes. To achieve this goal, we've implemented a command for parts – to create undercuts within parts and another one for drawings, to annotate the grids (work plans and work axis). We have enhanced some existing commands as: Export BOM to Excel, Switch ON/OFF and Unresolved references.

2.1. UNDERCUT

The Inventor undercut (Relief) is only available in Assembly environment if you use Design Accelerator. However, if you need to introduce that feature in a manually modelled part, in part environment, there is no such a feature available.

Therefore, we have introduced in the PowerPack the Undercut functionality that allows designers to **create quickly relief grooves** within parts. The implementation for the Undercut command has been done based on the **standard ISO 18388:2016**.

Table 1 — Dimensions of the relief grooves

Dimensions in millimetres

Type	r^a		t_1	t_2	f	g	Corresponding diameter d_1^b for workpieces	
	Series 1	Series 2					subjected to normal loads ^c	subjected to higher alternating loads
E		R0,2	0,1	—	1	—	Over Ø 1,6 up to Ø 3	—
		R0,4	0,2	—	2	—	Over Ø 3 up to Ø 18	—
		R0,6	0,2	—	2	—	Over Ø 10 up to Ø 18	—
		R0,6	0,3	—	2,5	—	Over Ø 18 up to Ø 80	—
		R0,8	0,3	—	2,5	—	Over Ø 18 up to Ø 80	—
		R1	0,2	—	2,5	—	—	Over Ø 18 up to Ø 50
		R1	0,4	—	4	—	Over Ø 80	—
		R1,2	0,2	—	2,5	—	—	Over Ø 18 up to Ø 50
		R1,2	0,4	—	4	—	Over Ø 80	—
		R1,6	0,3	—	4	—	—	Over Ø 50 up to Ø 80
F		R2,5	0,4	—	5	—	—	Over Ø 80 up to Ø 125
		R4	0,5	—	7	—	—	Over Ø 125
		R0,2	0,1	0,1	1	(0,9)	Over Ø 1,6 up to Ø 3	—
		R0,4	0,2	0,1	2	(1,1)	Over Ø 3 up to Ø 18	—
		R0,6	0,2	0,1	2	(1,4)	Over Ø 10 up to Ø 18	—
		R0,6	0,3	0,2	2,5	(2,1)	Over Ø 18 up to Ø 80	—
		R0,8	0,3	0,2	2,5	(2,3)	Over Ø 18 up to Ø 80	—
		R1	0,2	0,1	2,5	(1,8)	—	Over Ø 18 up to Ø 50
		R1	0,4	0,3	4	(3,2)	Over Ø 80	—
		R1,2	0,2	0,1	2,5	(2)	—	Over Ø 18 up to Ø 50
G		R1,2	0,4	0,3	4	(3,4)	Over Ø 80	—
		R1,6	0,3	0,2	4	(3,1)	—	Over Ø 50 up to Ø 80
		R2,5	0,4	0,3	5	(4,8)	—	Over Ø 80 up to Ø 125
		R4	0,5	0,3	7	(6,4)	—	Over Ø 125
		R0,4	0,2	0,2	(0,9)	(1,1)	Over Ø 3 up to Ø 18	—
		R0,8	0,3	0,05	(2,0)	(1,1)	Over Ø 18 up to Ø 80	—
		R1,2	0,3	0,05	(2,4)	(1,5)	—	Over Ø 18 up to Ø 50
H								

^a Relief grooves radii of the series 1 have to be preferred.

^b Does not apply to parts with a short shoulder and thin-walled parts. When a workpiece has different diameters, it may be expedient for manufacturing reasons to use several relief grooves having the same form and size.

^c Type G applies only to workpieces which are not subjected to high fatigue loads.

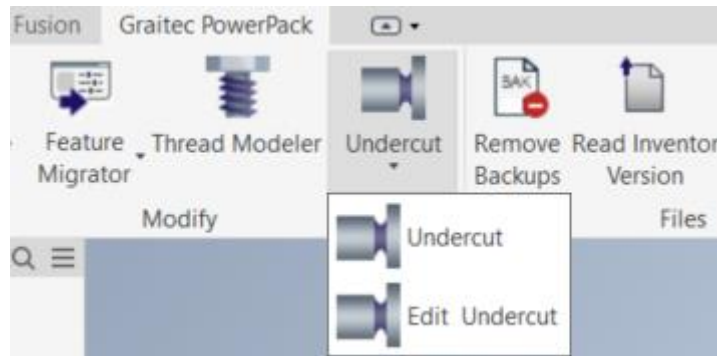
Relief Grooves Dimensions – ISO 18388:2016 (E)

The new functionality is offering the possibility to define relief grooves:

- Based on the loads according to standard ISO 18388:2016.
- By selecting the recommended radius according to the standards.
- In a workpiece defined by the user as a part in Inventor.
- As custom for workpieces that are not in the range covered by the ISO standard.

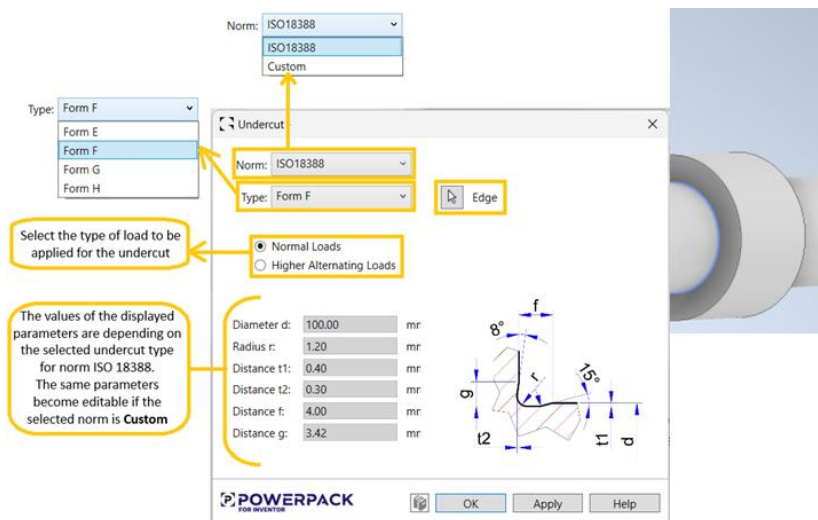
The functionality allows the possibility to edit an already defined relief groove.

The Undercut command is available in the parts ribbon, in the Modify panel, where you have 2 options, one to create a new undercut and the option to edit an existing one.



Undercut Ribbon command

When the command to create a new Undercut is called, a dialog will appear. In this dialog, we can control the norm, the type of undercut, the type of loads applied to the undercut, option to customize the undercut and the edge selector.



Undercut Dialog

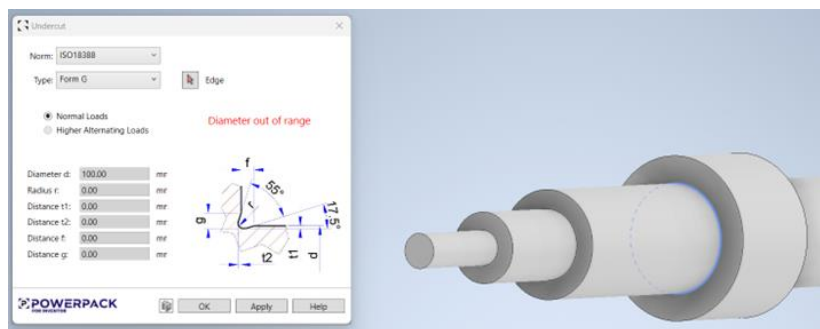
The workflow to create an **undercut with the Powerpack** functionality it's easy and intuitive:

- Call the command while in a part environment.
- In the dialog configure the undercut by setting the desired type, norm and load.
- Using the selector from the dialog, select the edge on which the undercut will be applied.
- Once the edge is selected, the command will measure the diameter of the part and based on that, the rest of the parameters will be updates.
- Click Apply to create the undercut.



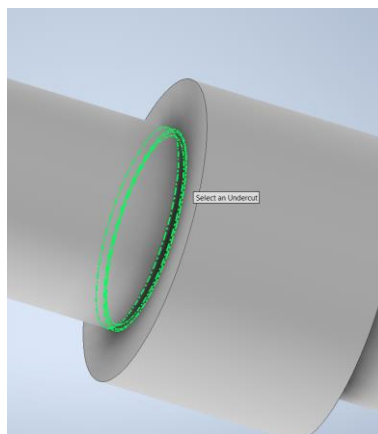
Undercut Iso 18388 form F

Note: The forms apply to a max and a min diameter. If the diameter is out of range for the selected form, a message will be displayed in the dialog and the generation of the undercut will not be possible.



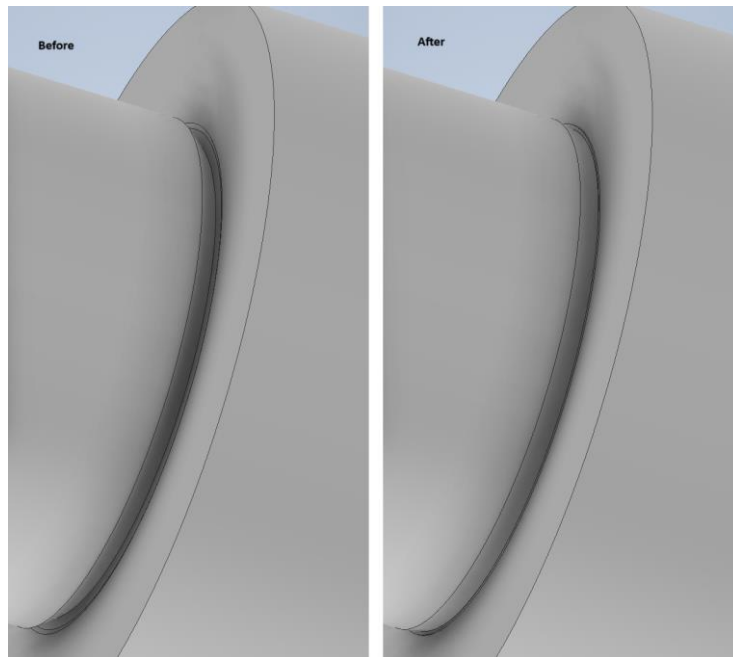
Diameter out of range

To edit a relief groove, run the **Edit Undercut command** and select the Undercut.

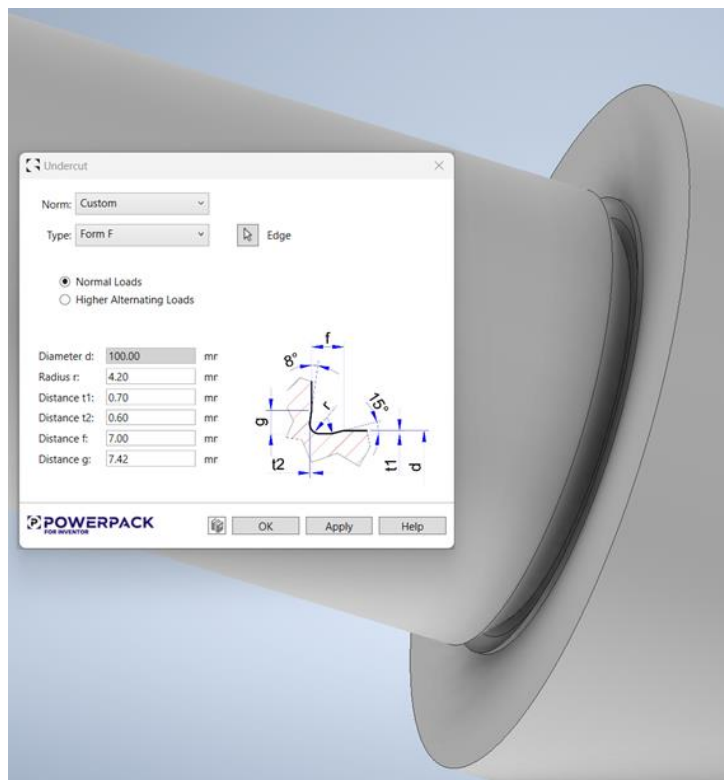


Undercut selection

The dialog will list the type of undercut previously created. This can be changed by redefining the form, norm and its parameters.



The **custom mode** allows the modification of all parameters except the Diameter, which is detected automatically based on the selected workpiece. To enable it, just change the norm from Iso to custom.



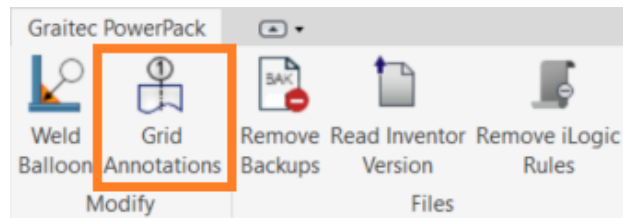
Custom mode

2.2. GRID ANNOTATIONS

The annotation process of working planes and axes in Inventor is an extremely time-consuming one, especially when working on multi-sheet drawings. This is because balloons are non-BOM items, and therefore a sketch symbol with prompted entry must be used and then manually edit the text to match the work plane/axis name or other requirements.

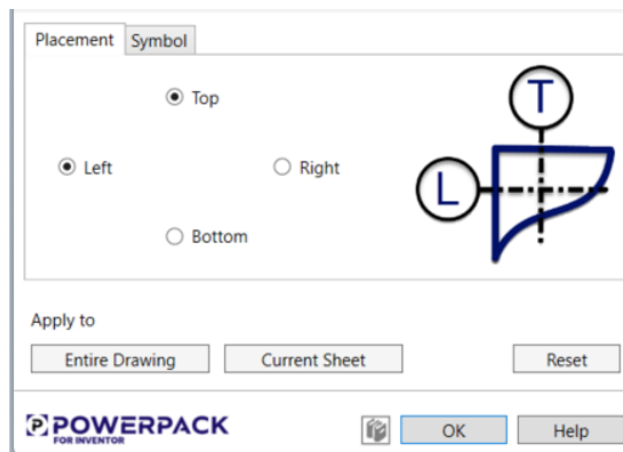
The new **Grid Annotations** command has been created to avoid losing time on such a repetitive task. The command **automates the process of adding Balloon annotations to Work planes and axes** that are included in the drawing views. The applicability is for residential building modelling, structural steel, or any construction-related projects.

The command is in the PowerPack Drawings ribbon, the Modify panel:



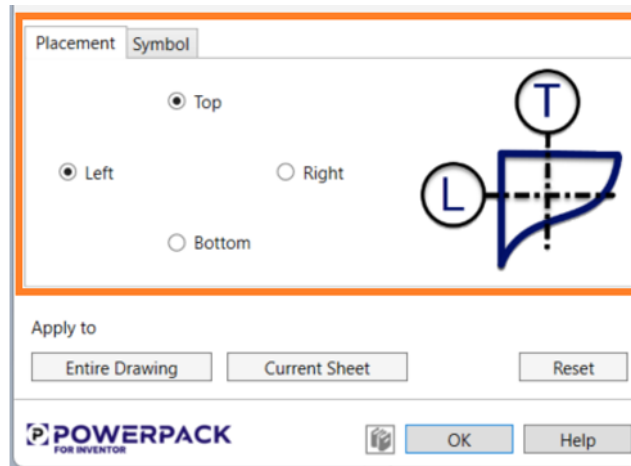
Grid Annotations ribbon

Once activated, a dialog with all options will be displayed.



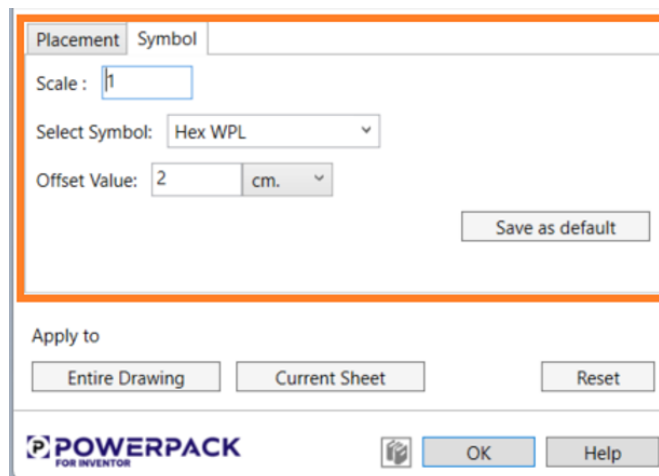
Grid Annotations dialog

The dialog has 2 tabs. The Placement tab is helping to position the annotations: Top – Left; Top – Right; Bottom – Left and Bottom – Right.



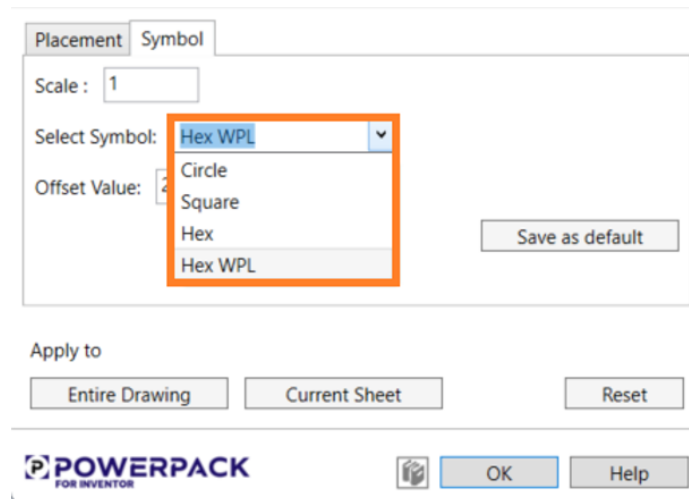
Placement tab

The second tab, Symbol, allows the selection of the symbol used for the annotations. Here a scale and an offset of the symbol can be set as well.



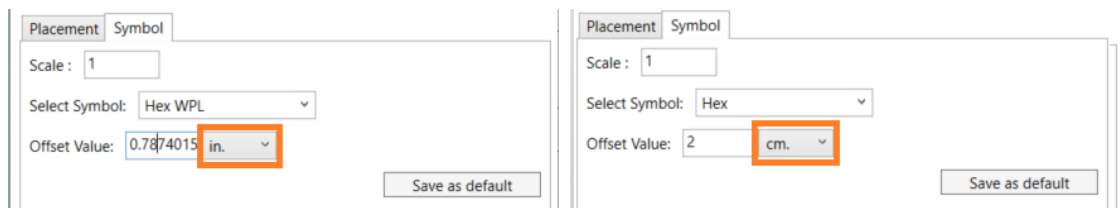
Symbol tab

The Symbol selector is populated with symbols available in your drawing template. If the drawing has no symbols defined, the selector will be empty.



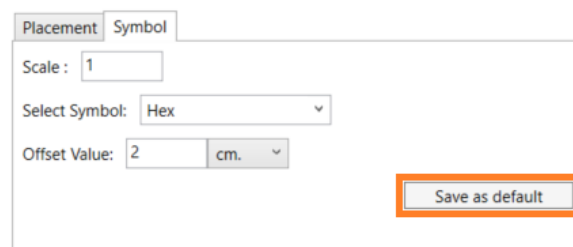
Symbol selector

The Offset Value units is defaulted automatically based on the settings of the drawing. If the template is set in cm, the offset will be by default in cm. If the template is set in inch, then the offset will be set in inch.



Offset Value units

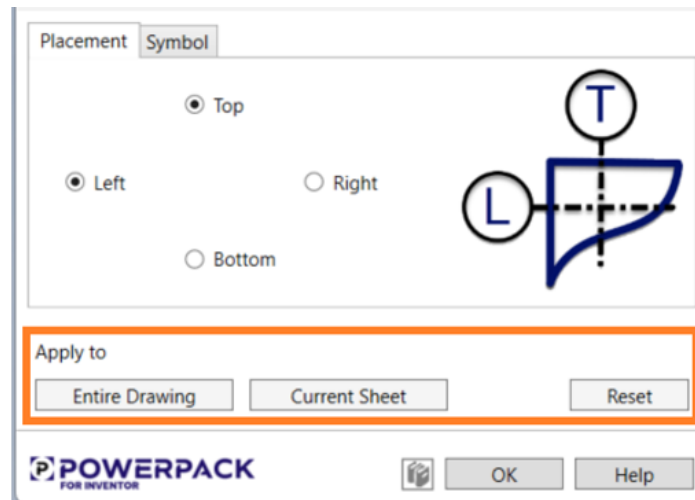
The “Save as default” option is saving the values set for Scale, Symbol and Offset, so next time the command is run, the values don’t need to be set again.



Save as default

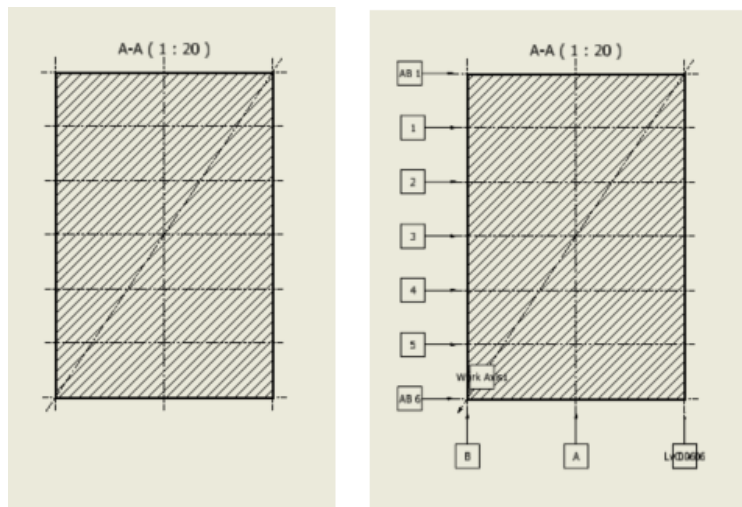
Note: If the command is started in a different drawing with different temple, the symbol set previous will not be available, as it's not available in the new drawing.

In the bottom area of the dialog, we have the options to apply the symbols to the “**Entire Drawing**” and “**Current Sheet**”, while the “**Reset**” option is allowing to remove the symbols from the drawing.



Apply and Reset options

Note: If the dialog is closed and reopen, the Reset option will not remove the symbols previously generated.



Without and with symbols

3. IMPROVEMENTS

In the 2025.1 version, arrives with several updates for important functionalities such as: Unresolved References, Export BOM to Excel and Switch ON/OFF.

3.1. UNRESOLVED REFERENCES

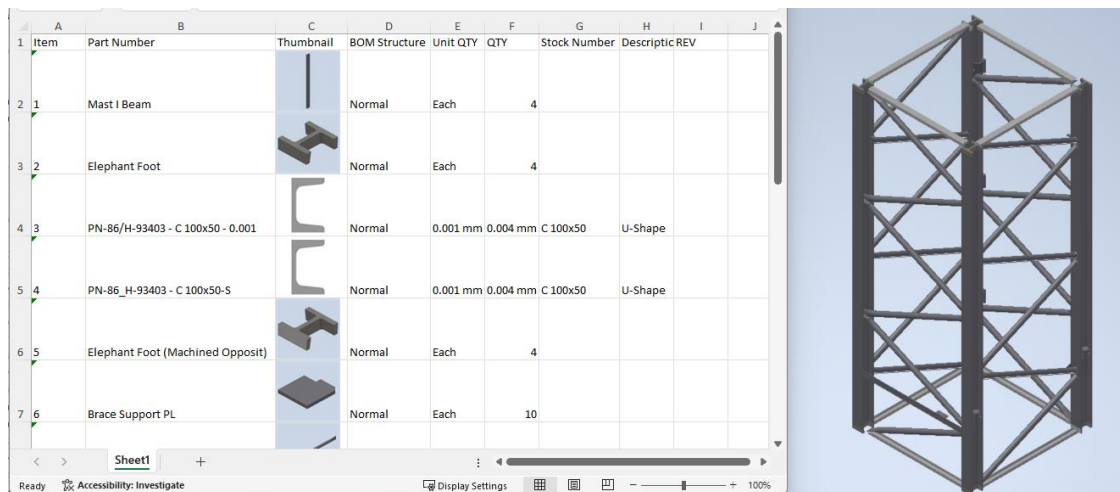
The Unresolved References has been launched in version 2025 and the functionality was searching for missing references at the assembly level which was open without going in the subassemblies loaded in the main assembly. This workflow was forcing the user to open each subassembly and apply the same methodology to identify missing references and fix them.

In 2025.1 version, the workflow has been improved by applying the search of missing references in the subassembly as well, from the assembly level.

3.2. EXPORT BOM TO EXCEL

The Export BOM to Excel tool is very useful. Previously, thumbnail images were included in the Excel file but not within the cells themselves. This required users to embed the images into the cells manually, which was time-consuming.

Starting from version 2025.1, the Export BOM to Excel command will generate an Excel file where all thumbnail images are placed directly within the cells. This ensures that when the cells are resized, the images remain correctly positioned.



	A	B	C	D	E	F	G	H	I	J
	Item	Part Number	Thumbnail	BOM Structure	Unit QTY	QTY	Stock Number	Descriptive	REV	
2	1	Mast I Beam		Normal	Each	4				
3	2	Elephant Foot		Normal	Each	4				
4	3	PN-86/H-93403 - C 100x50 - 0.001		Normal	0.001 mm	0.004 mm	C 100x50	U-Shape		
5	4	PN-86_H-93403 - C 100x50-S		Normal	0.001 mm	0.004 mm	C 100x50	U-Shape		
6	5	Elephant Foot (Machined Opposit)		Normal	Each	4				
7	6	Brace Support PL		Normal	Each	10				

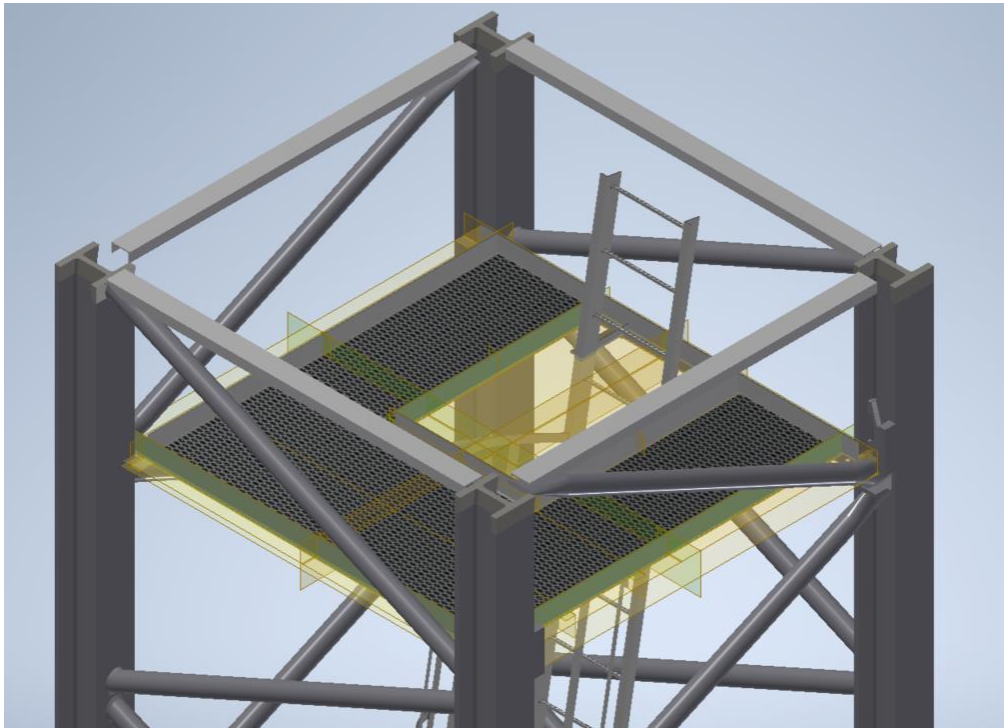
Embedded Thumbnails within cells

3.3. SWITCH ON/OFF – ELEMENTS BY SELECTIONS

Currently, all Switch ON/OFF commands affect the entire model, even if only one part is selected. To provide users with a more detailed view, the Switch ON/OFF commands will now also work on selected parts.

Starting with version 2025.1, if a part is selected before running the command, the working objects will be turned on or off only on the selected element.

If no part is selected, the working objects will be turned on or off on the entire model.



Switch ON/OFF on selection