

# Inventor 2027 What's new

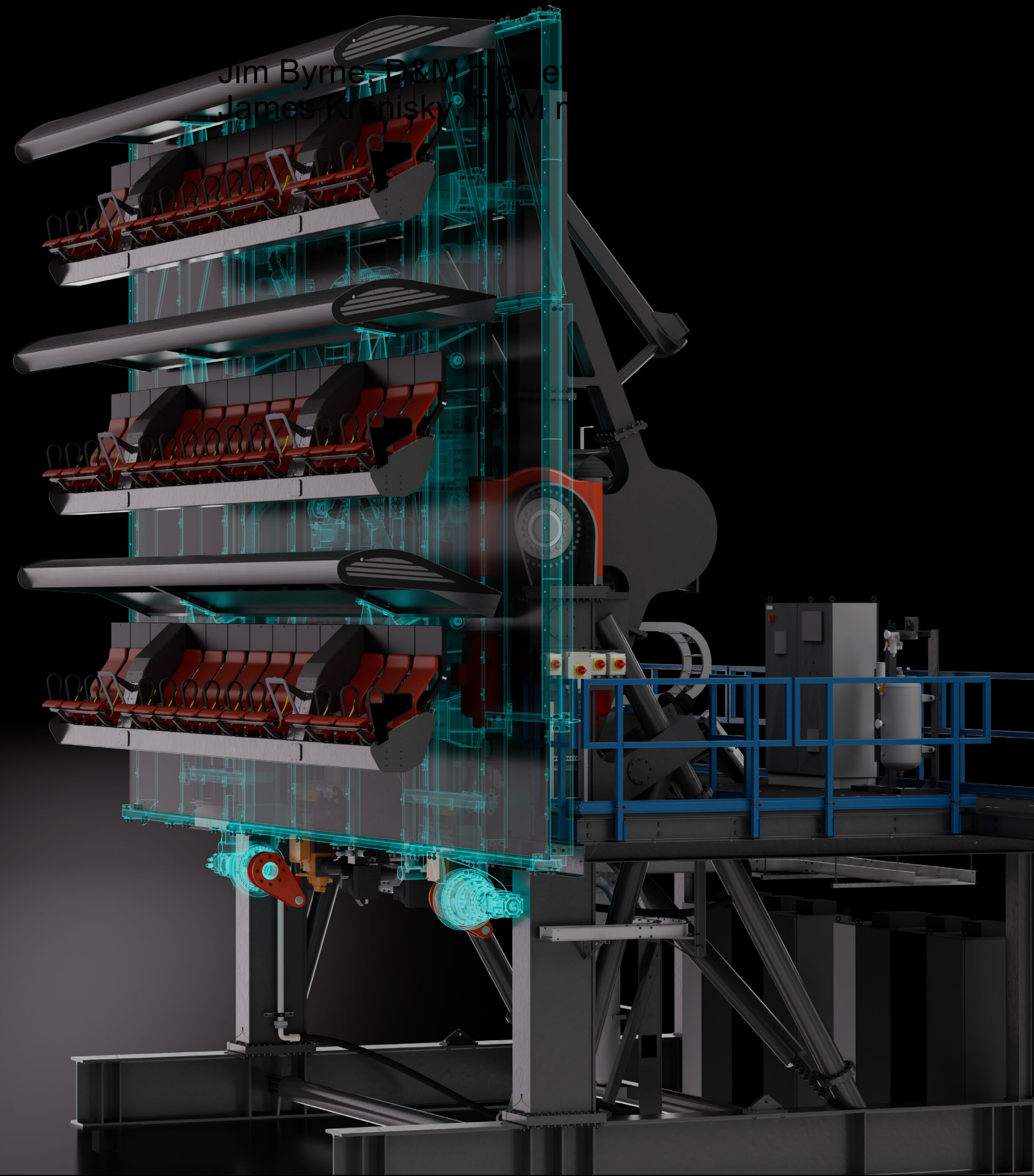


Image courtesy of Dynamic Structures

# Speakers



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Systemkonsulent

Product Owner  
NTI for Inventor



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Systemkonsulent

# Agenda

- Inventor 2027 what's new

- 01 General
- 02 Part modeling
- 03 Assembly modeling
- 04 Drawings
- 05 Model States
- 06 Interoperability

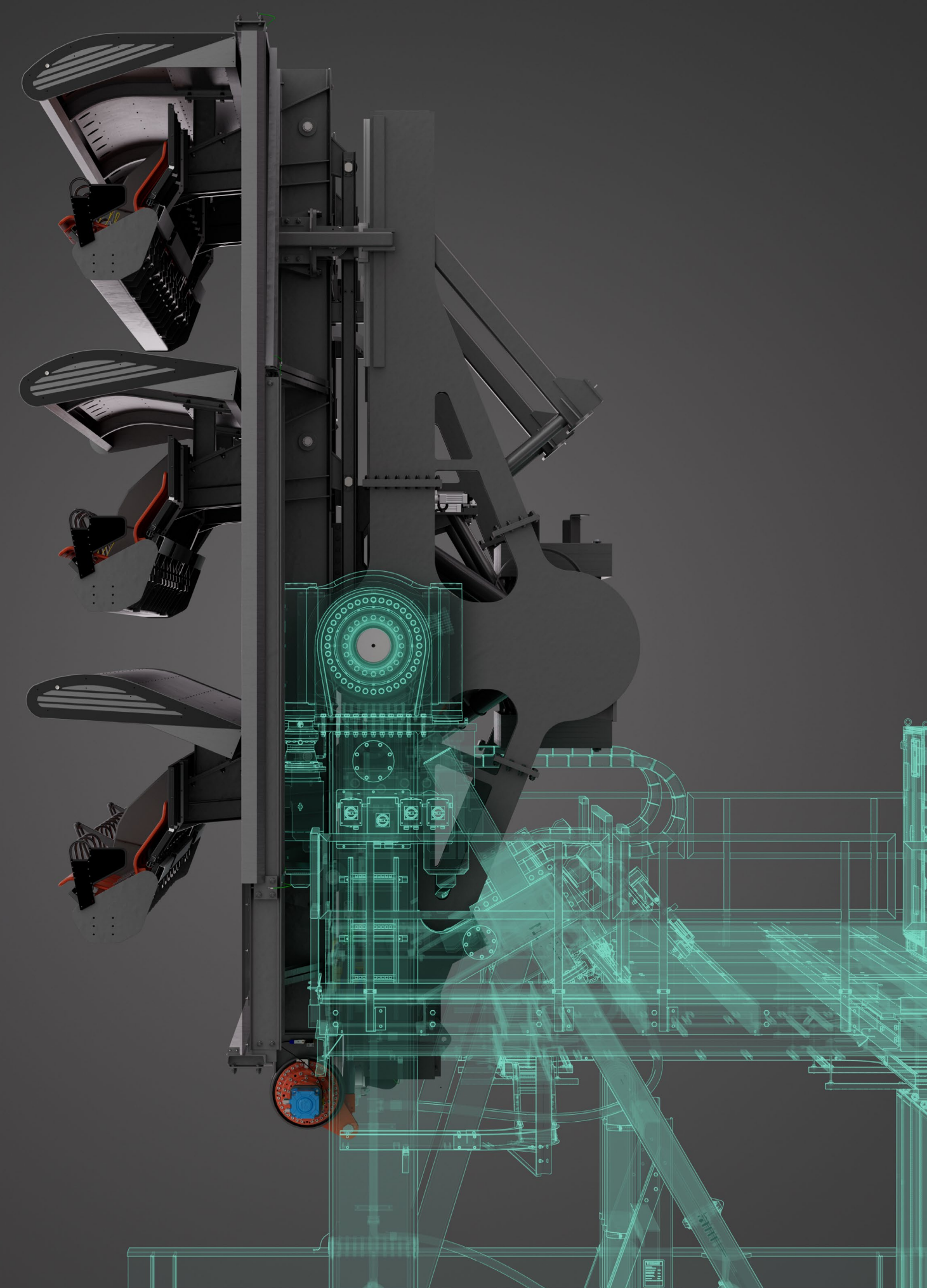
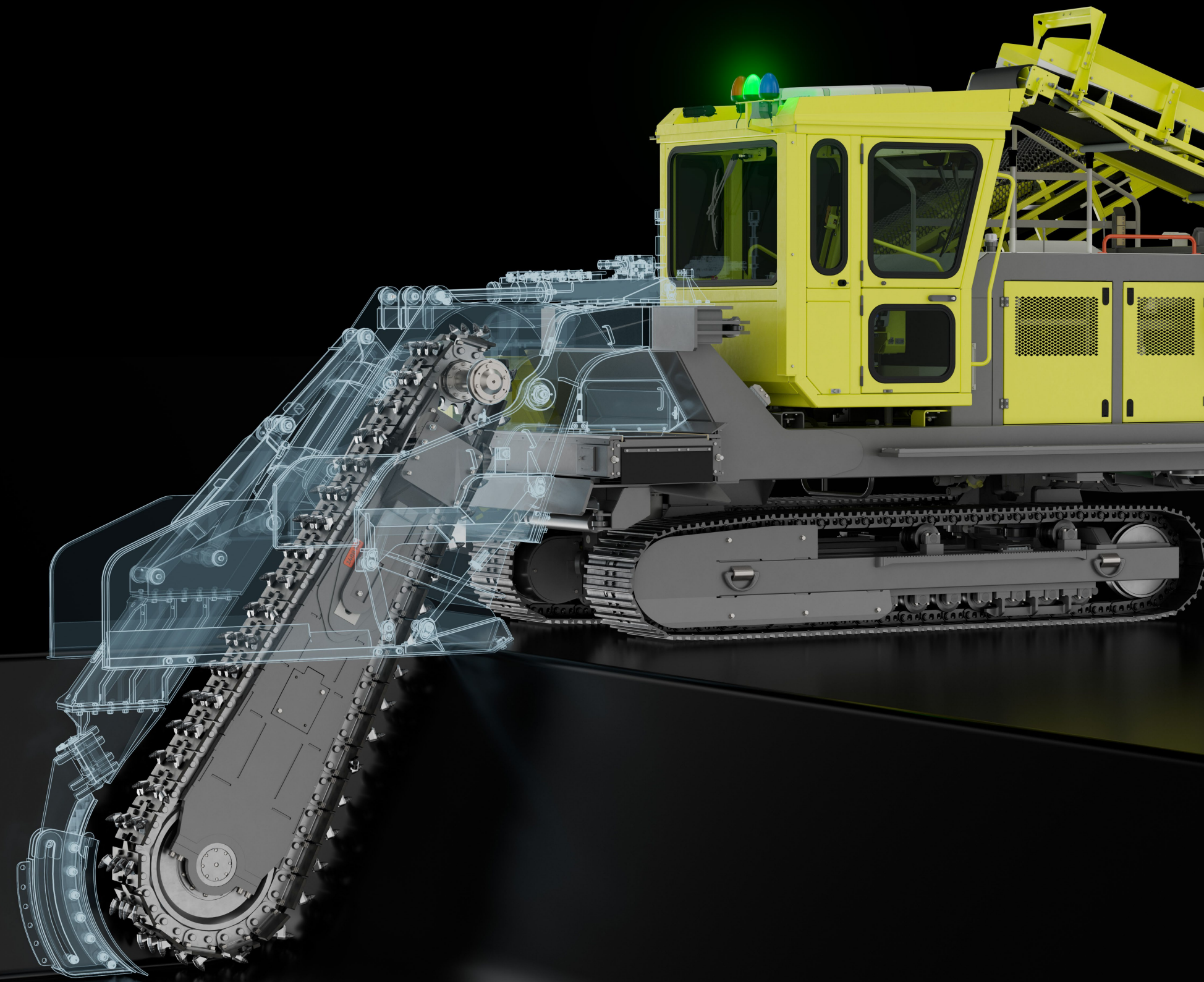


Image courtesy of Dynamic Structures

# General



 **AUTODESK**

Image courtesy of Mastenbroek

# Autodesk Assistant

- Move from question to action without leaving your workflow
- Complete work with more speed and confidence
- Catch manufacturability issues before they reach the shop floor
- Generate renders, exports, and documentation
- Find critical information instantly across models, drawings, and assemblies

Autodesk Assistant x + Tech Preview

Save PNG files of the top level components

I'll export PNG files for each top-level component in your stapler assembly. Let me first identify the top-level components and then export each one individually.

- ✓ Executing get\_component\_states...

Let me get all top-level components including those with default states:

- ✓ Executing get\_component\_states...

Perfect! I can see there are 2 top-level components: Bottom:1 and Upper:1. Now I'll export PNG files for each component. I'll need to open each component individually to export them properly.

- ✓ Executing open\_file...
- ✓ Executing export\_document\_as\_image...

C:\Users\byrnej\OneDrive - Autodesk\Datasets\Samples\Models\Assemblies\Stapler\Bottom\_Component.png

- ✓ Executing close\_document...
- ✓ Executing open\_file...
- ✓ Executing export\_document\_as\_image...

C:\Users\byrnej\OneDrive - Autodesk\Datasets\Samples\Models\Assemblies\Stapler\Upper\_Component.png

- ✓ Executing close\_document...

✓ **PNG Export Complete:**  
2 of 2 top-level components exported successfully

Component	File Name	Status	Location
Bottom:1	Bottom_Component.png	✓ Exported	Stapler folder
Upper:1	Upper_Component.png	✓	Stapler

Ask Autodesk Assistant  ➤

Autodesk Assistant x + Tech Preview

hide all parts that are under 0.07 cubic cm

I'll hide all parts with volumes under 0.07 cubic cm based on the volume data I analyzed earlier.

- ✓ Executing set\_component\_state...

✓ **Parts Under 0.07 cm<sup>3</sup> Hidden Successfully:**  
7 of 7 parts hidden based on volume threshold

Part Name	Location	Volume (cm <sup>3</sup> )	Status
Spring	Bottom:1/	0.003	✗ Hidden (under 0.07)
Channel - Hinge Pin	Upper:1/Channel:1/	0.014	✗ Hidden (under 0.07)
Channel - Pusher	Upper:1/Channel:1/	0.019	✗ Hidden (under 0.07)
Channel - Spring Clip	Upper:1/	0.024	✗ Hidden (under 0.07)
Bottom - Anvil	Bottom:1/	0.032	✗ Hidden (under 0.07)
Staples	Upper:1/Channel:1/	0.042	✗ Hidden (under 0.07)
Channel - Bumper	Upper:1/Channel:1/	0.064	✗ Hidden (under 0.07)

Remaining Visible Parts (Volume > 0.07 cm<sup>3</sup>):

- Bottom - Back: 1.460 cm<sup>3</sup>
- Bottom - Front: 1.336 cm<sup>3</sup>

Ask Autodesk Assistant  ➤

# Autodesk Assistant

## Sample prompts :

### Assembly Analysis & Component Management

"Show me all [material] components and their properties"

### Quality Control & Manufacturing Validation

"Find all fillets under 2mm - our manufacturer can't do this size"

### Configuration Comparison & Management

"What changed between [Model State 1] and [Model State 2]?"

### Documentation & Export Generation

"Export a PNG image for each top -level component"

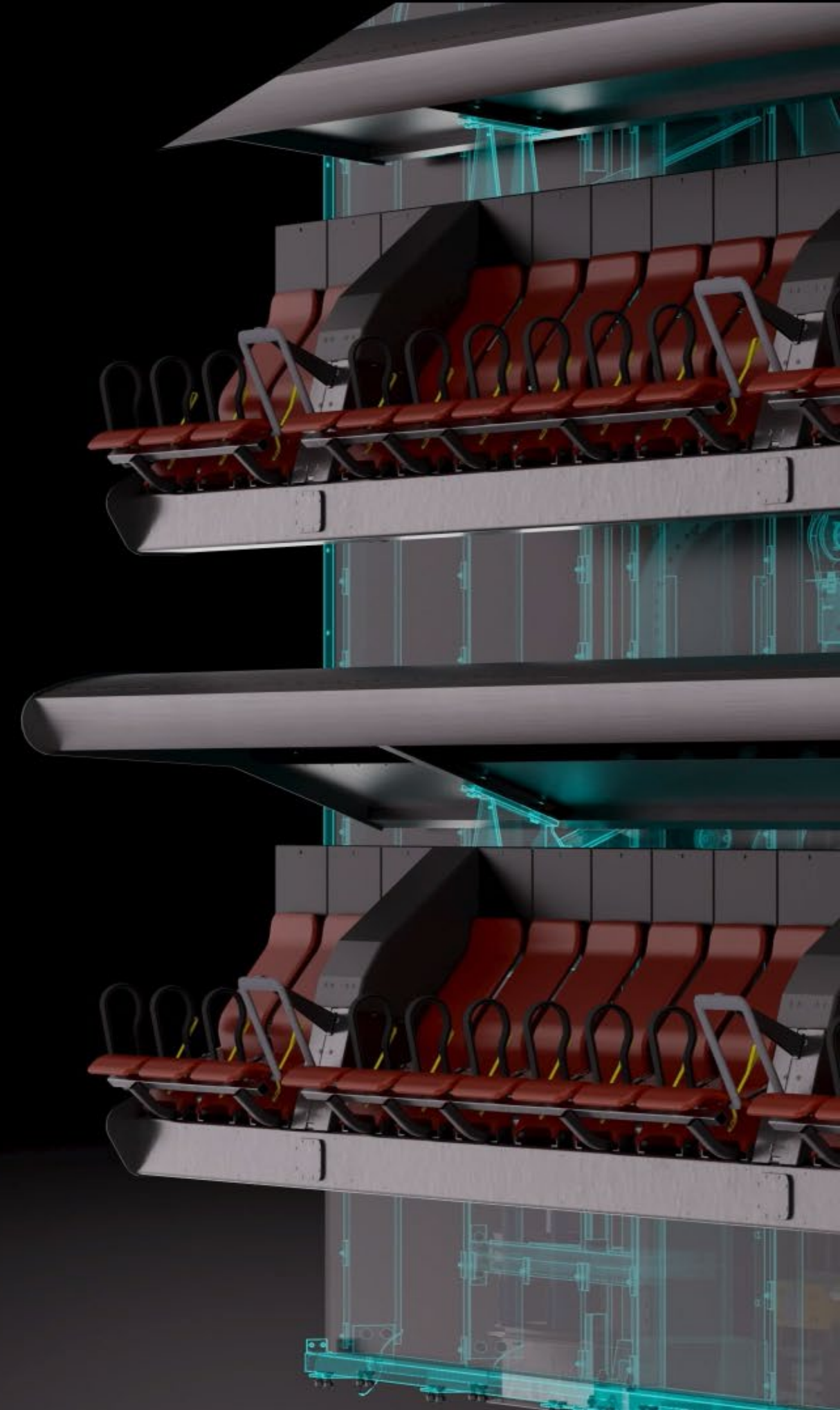
### Property & Parameter Management

"Add a custom property called [property name] to all selected components"



# Inventor Professional

Autodesk Assistant



# iLogic

- Consistent interface w/ Informed Design & Fusion
- Interactive puzzle logic “blocks”
  - Drag & drop / connect
  - Copy / Paste
  - Undo Redo
- Show/Hide code text preview
- Rule checker
- Optional convert to text rule
  - One direction only

The screenshot displays the iLogic software interface. On the left is a sidebar with a category tree including Logic, Loops, Math, Text, Lists, Variables, Functions, Parameters, iProperties, Assembly, Features, Input / Output, Excel, Document, and Workspace Tools. The main workspace shows a rule editor with several logic blocks:

- Height Rule:** An 'if' block with a condition 'Height < 20' leading to 'set Height to 20', and an 'else if' block with a condition 'Height > 96' leading to 'set Height to 96'.
- Width Rule:** An 'if' block with a condition 'Width < 30' leading to 'set Width to 30', and an 'else if' block with a condition 'Width > 60' leading to 'set Width to 60'.
- Width Increment Rule:** A 'set WidthIncrement to 0.25' block followed by a complex calculation: 'set Width to round up (round (Width to 4 decimal places) / WidthIncrement) \* WidthIncrement'.
- Target Rise Rule:** An 'if' block with a condition 'TargetRise < 6' leading to 'set TargetRise to 6', and an 'else if' block with a condition 'TargetRise > 8.5' leading to 'set TargetRise to 8.5'.
- Run Rule:** An 'if' block with a condition 'Run < 8.5' leading to 'set Run to 8.5', and an 'else if' block with a condition 'Run > 16' leading to 'set Run to 16'.

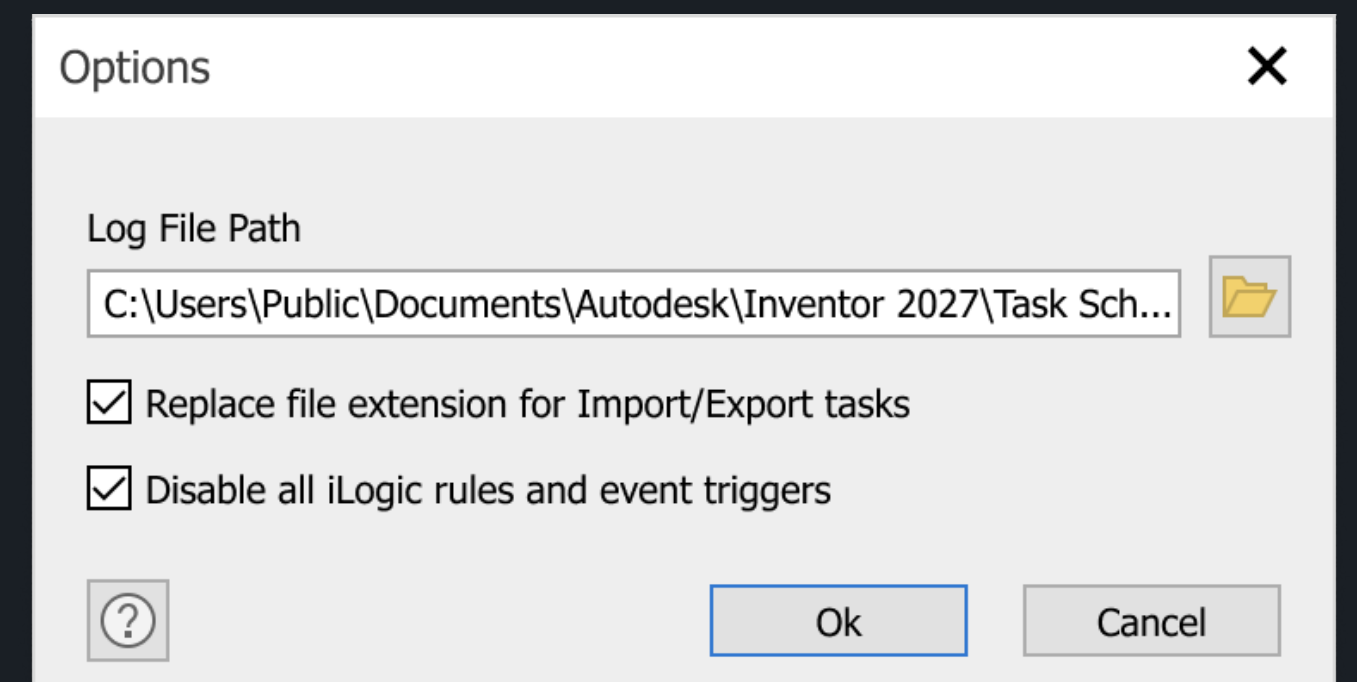
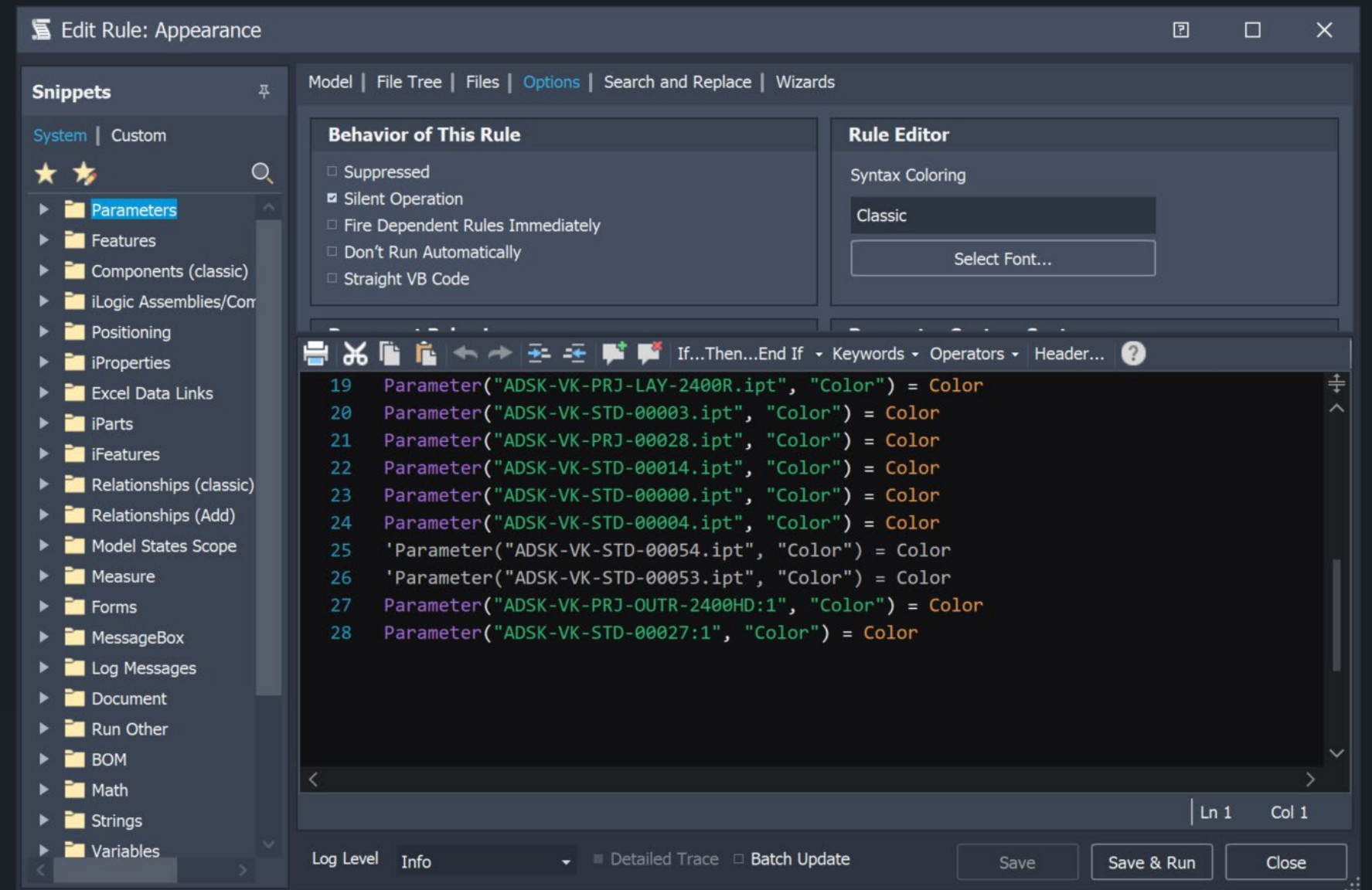
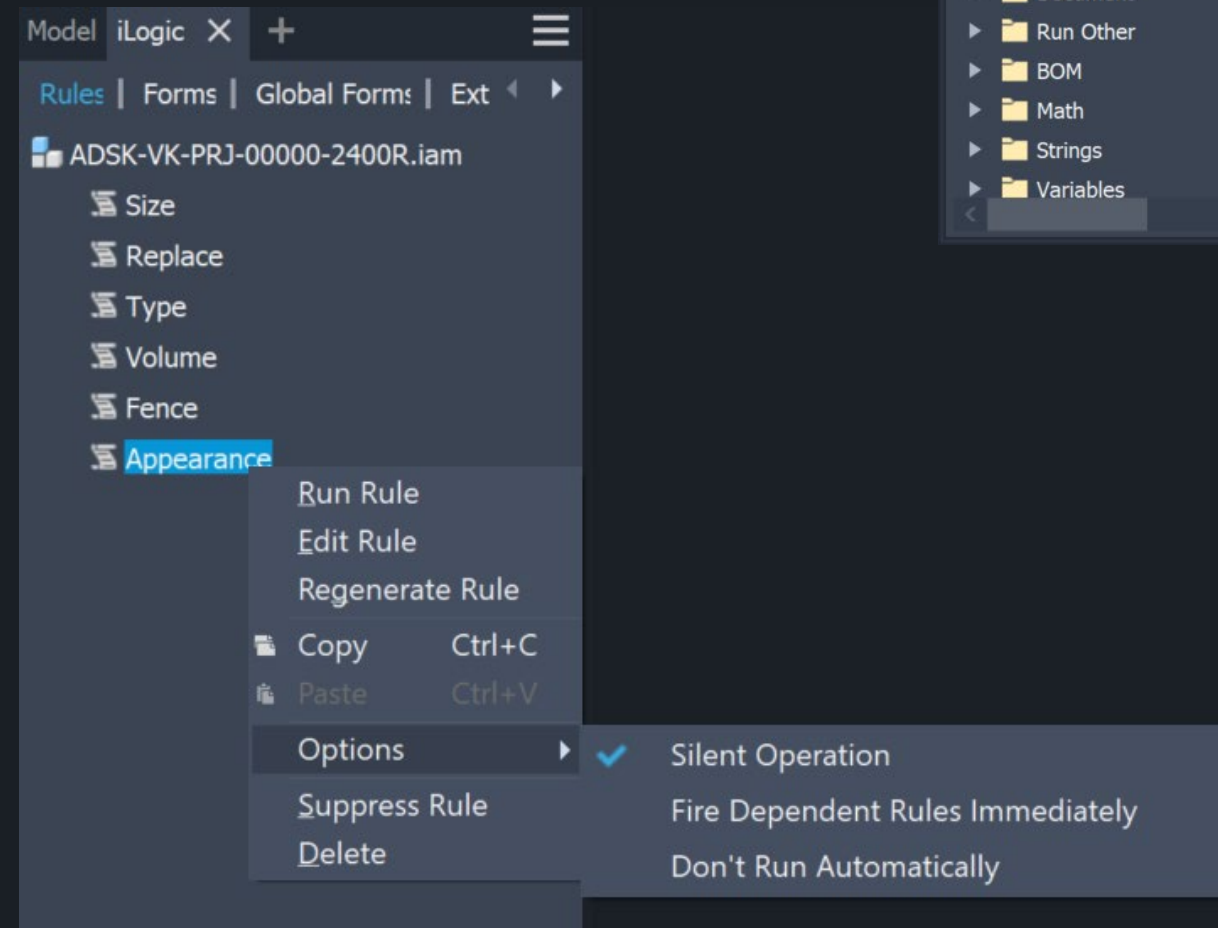
At the bottom, a code preview shows the following text:

```
1 Dim WidthIncrement As Double
2
3
4 RuntimeUtils.AddGetParameterTriggers({Height, Run, TargetRise, Width})
```

Buttons for 'Save', 'Run', and 'Close' are visible at the bottom right of the interface.

# iLogic

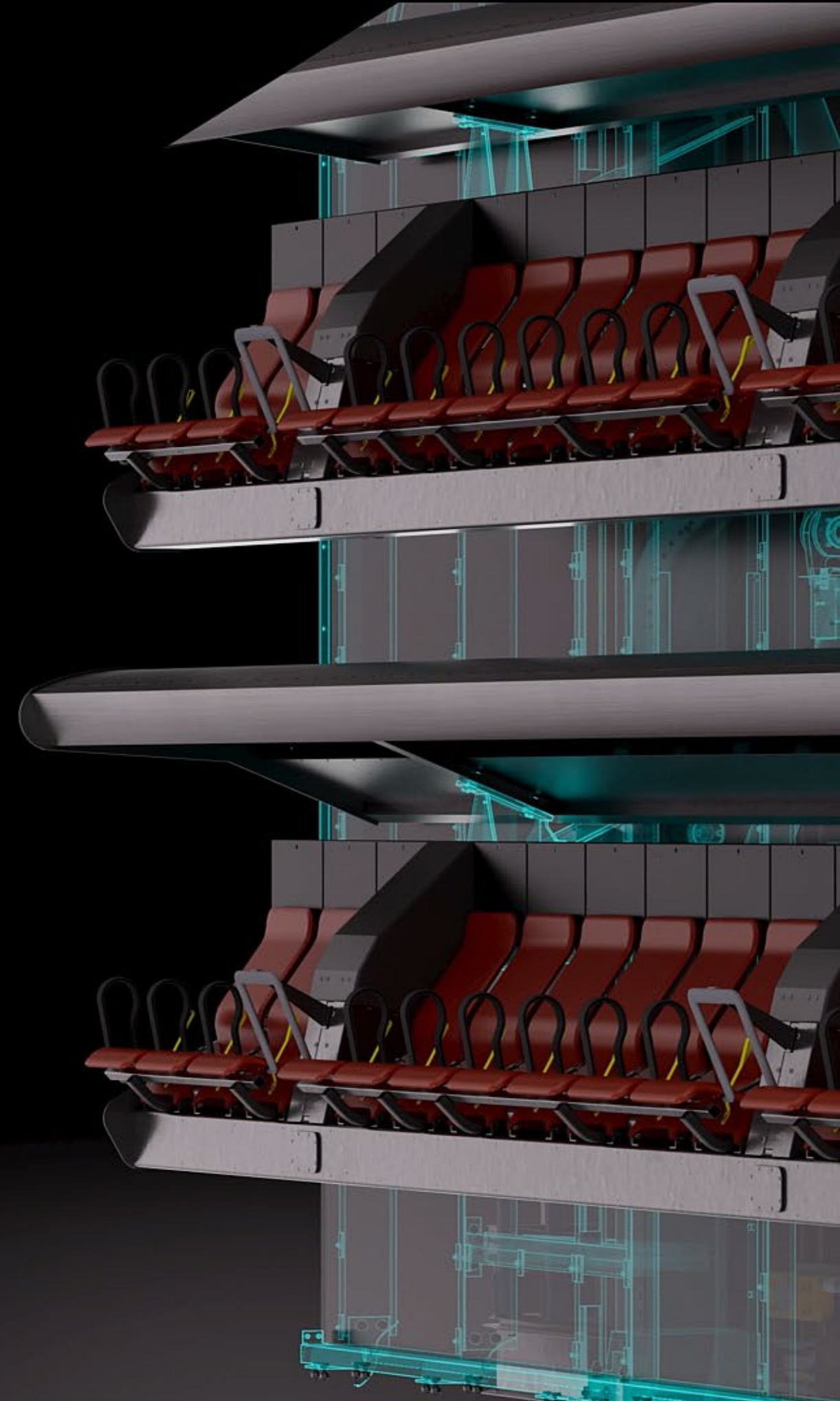
- Run options
- Task Scheduler – Disable iLogic rules and event triggers





**Inventor Professional**

iLogic codeblocks



# Parameters

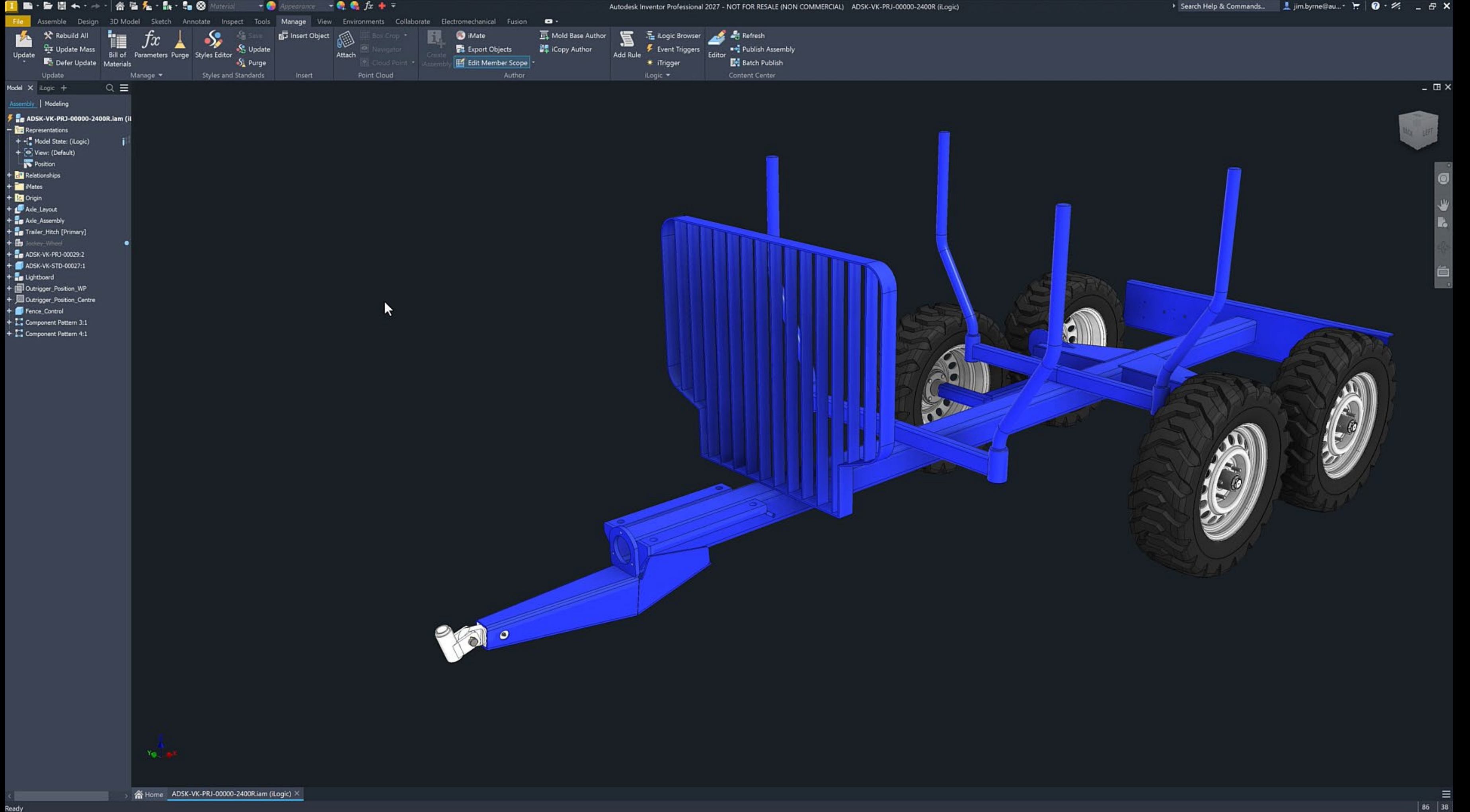
Keep parameters dialog open while working

Autodesk Inventor Professional 2027 - NOT FOR RESALE (NON COMMERCIAL) ADSK-VK-STD-00027

Parameters: ADSK-VK-STD-00027

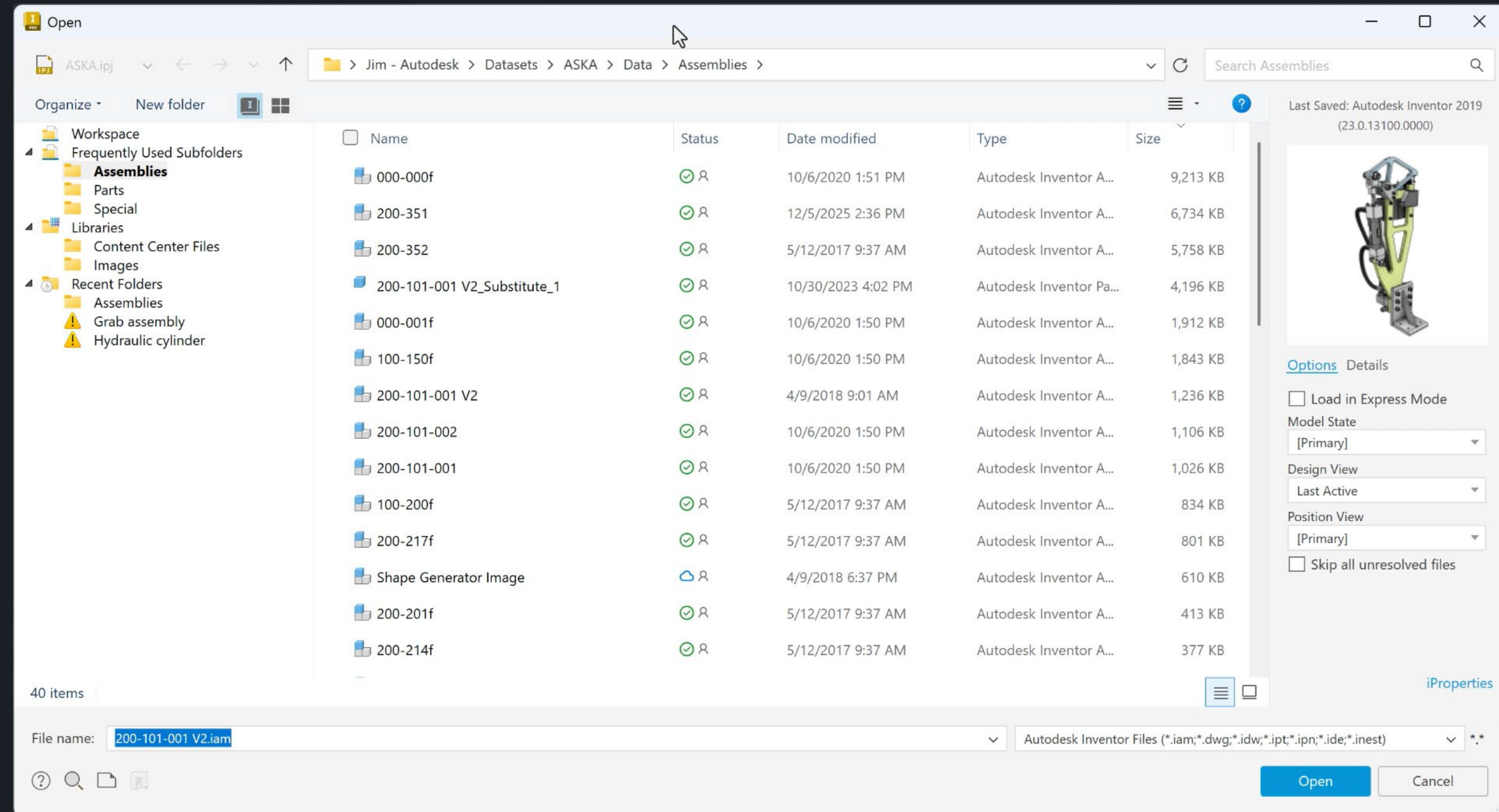
Parameter Name	Consumed	Unit/Typ	Equation	Nominal	Driving Rule	Tolera	Model Va	Key	Comment
<b>Model Parameters</b>									
d0	Sketch1	mm	200 mm	200.0...		☉ <	200.0...	<input type="checkbox"/>	
d1	Sketch1	mm	40 mm	40.00...		☉ <	40.00...	<input type="checkbox"/>	
d2	Extrusio...	mm	Length	1150.0...		☉ <	1150.0...	<input type="checkbox"/>	
d3	Extrusio...	deg	0.0 deg	0.000...		☉ <	0.000...	<input type="checkbox"/>	
d4	Fillet1	mm	10 mm	10.00...		☉ <	10.00...	<input type="checkbox"/>	
d5	Shell1	mm	5 mm	5.000...		☉ <	5.000...	<input type="checkbox"/>	
p2	d13, W...	mm	Length / 2 ul - 100 mm	475.0...		☉ <	475.0...	<input type="checkbox"/>	
p1	d10, W...	mm	Length / 2 ul - 250 mm	325.0...		☉ <	325.0...	<input type="checkbox"/>	
d8	Rectan...	ul	2 ul	2.000...		☉ <	2.000...	<input type="checkbox"/>	
d10	Rectan...	mm	p1 * 2 ul	650.0...		☉ <	650.0...	<input type="checkbox"/>	
d11	Rectan...	ul	2 ul	2.000...		☉ <	2.000...	<input type="checkbox"/>	
d13	Rectan...	mm	p2 * 2 ul	950.0...		☉ <	950.0...	<input type="checkbox"/>	
d15	Extrusio...	deg	0.0 deg	0.000...		☉ <	0.000...	<input type="checkbox"/>	
d17	Extrusio...	deg	0.0 deg	0.000...		☉ <	0.000...	<input type="checkbox"/>	
d18	Sketch3	mm	6 mm	6.000...		☉ <	6.000...	<input type="checkbox"/>	
d19	Sketch3	mm	3.000 mm	3.000...		☉ <	3.000...	<input type="checkbox"/>	
d20	Sketch3	mm	70.000 mm	70.00...		☉ <	70.00...	<input type="checkbox"/>	
d21	d22, Sk...	mm	40.000 mm	40.00...		☉ <	40.00...	<input type="checkbox"/>	
d22	Sketch3	mm	d21 / 2 ul	20.00...		☉ <	20.00...	<input type="checkbox"/>	
d23	Sketch3	mm	25 mm	25.00...		☉ <	25.00...	<input type="checkbox"/>	
Cutout_distance	Sketch8	mm	350 mm	350.0...		☉ <	350.0...	<input type="checkbox"/>	
<b>User Parameters</b>									
Length	p1, p2, ...	mm	1150 mm	1150.0...		☉	1150.0...	<input checked="" type="checkbox"/>	
Color		Text	Blue					<input type="checkbox"/>	

350 mm



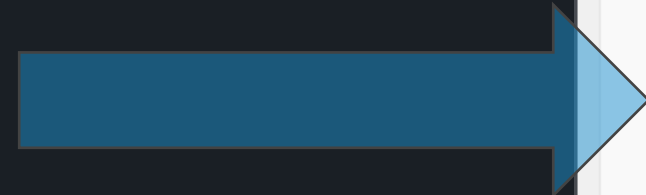
# Open and save

- File search
- Tree view options



# Hardware support

## DirectX 12 support



Application Options

Sketch Part iFeature Assembly Content Center  
General Save File Colors Display Hardware Prompts Drawing Notebook

Graphics Setting  
Note: Changing the graphics setting does not take effect until Inventor is restarted.

Graphics Driver: DirectX 12  
DirectX 11  
DirectX 12

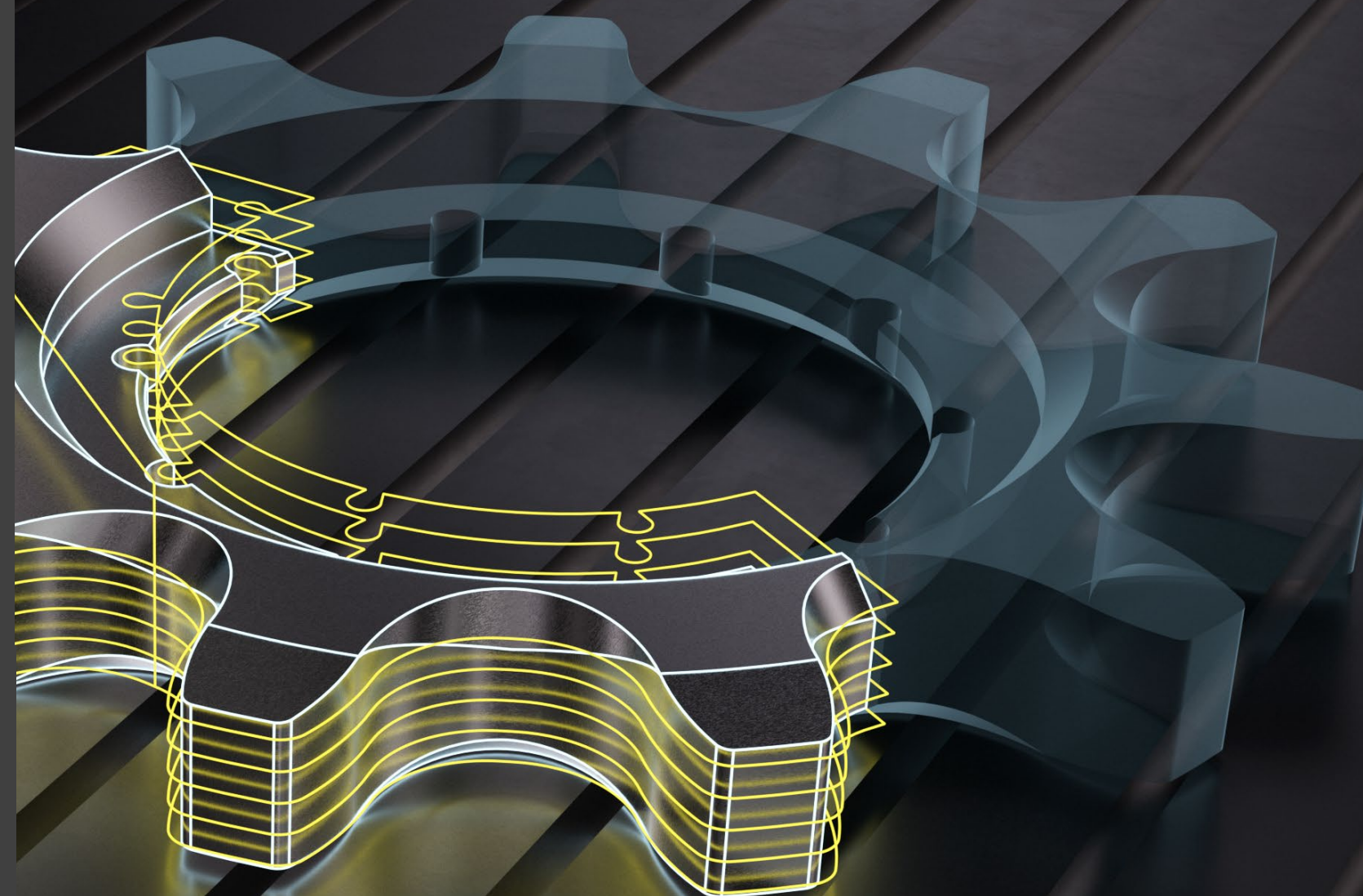
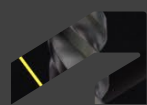
Quality  
Use this setting for the highest quality realistic visualization.

Performance  
Use this setting when performance is desired over realistic visualization such as in modeling.

Conservative  
Use this setting for conservative graphics hardware use with Inventor.

Software graphics  
Use only for systems with unrecognized graphics hardware or when graphics hardware does not support desired functionality.

# Part modeling



 AUTODESK

# Slot feature

- Property panel
- Fastener clearance
- In - canvas manipulators
- Slot style

Properties X +

Slot1 > Sketch6

No Preset + ⚙



▼ Input Geometry



Positions  $\leftarrow$  4 Positions  $\rightarrow$  +





Direction  $\leftarrow$  Direction  $\rightarrow$   $\leftarrow$   $\rightarrow$

Rotation 0.00 deg  $\leftarrow$   $\rightarrow$

▼ Type

Fillets  

Hole  

Seat    

▼ Fastener





Standard Ansi Unified Screw Thre... ▼

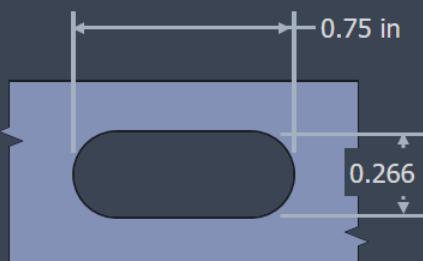
Fastener Type Hex Head Bolt ▼




Size 1/4 ▼



Fit Normal ▼

▼ Behavior

Center Marks    

 0.75 in  
0.266 in

Termination    ▼

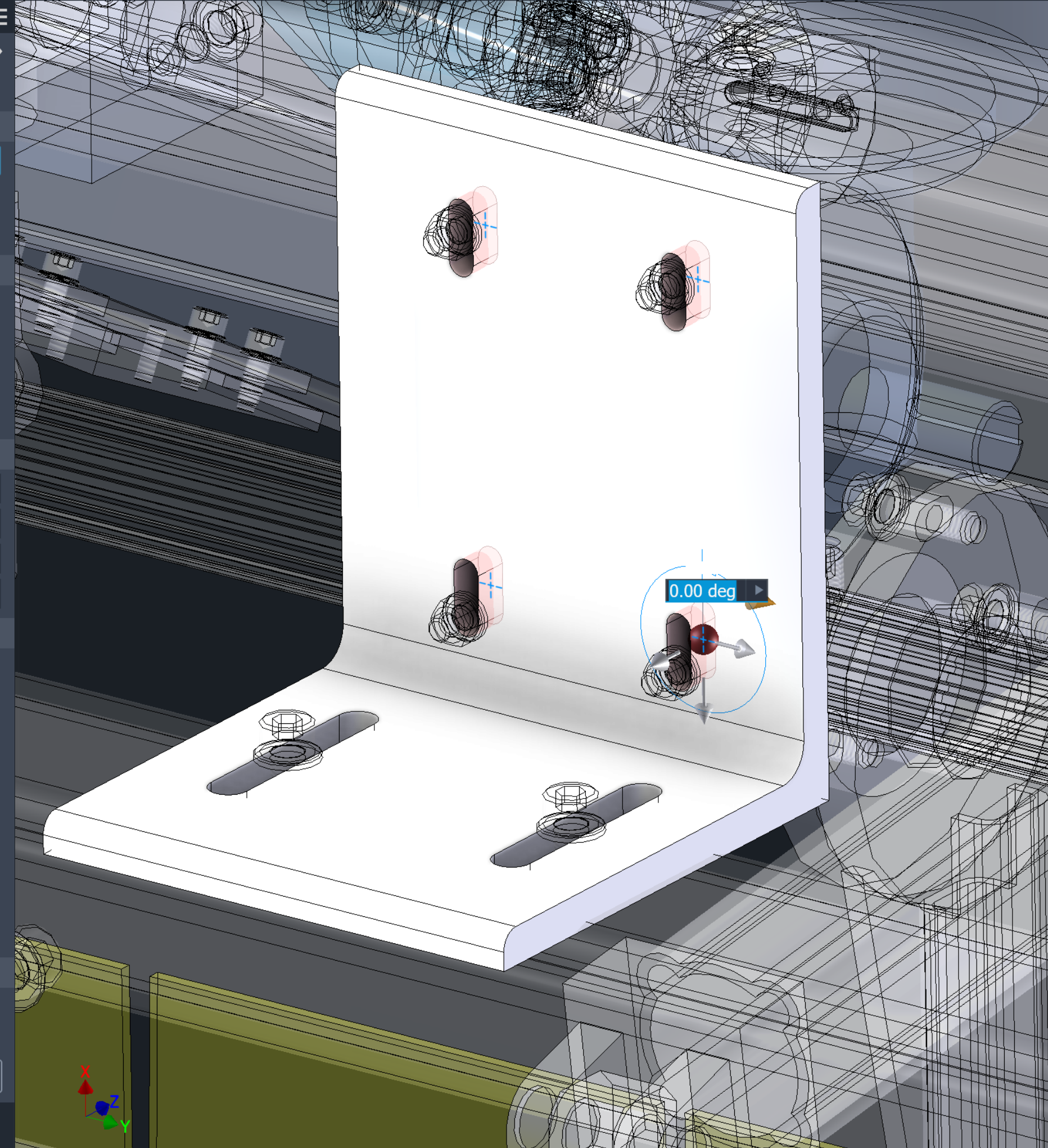
Direction   ▼

▼ Advanced Properties

iMate

Extend Start

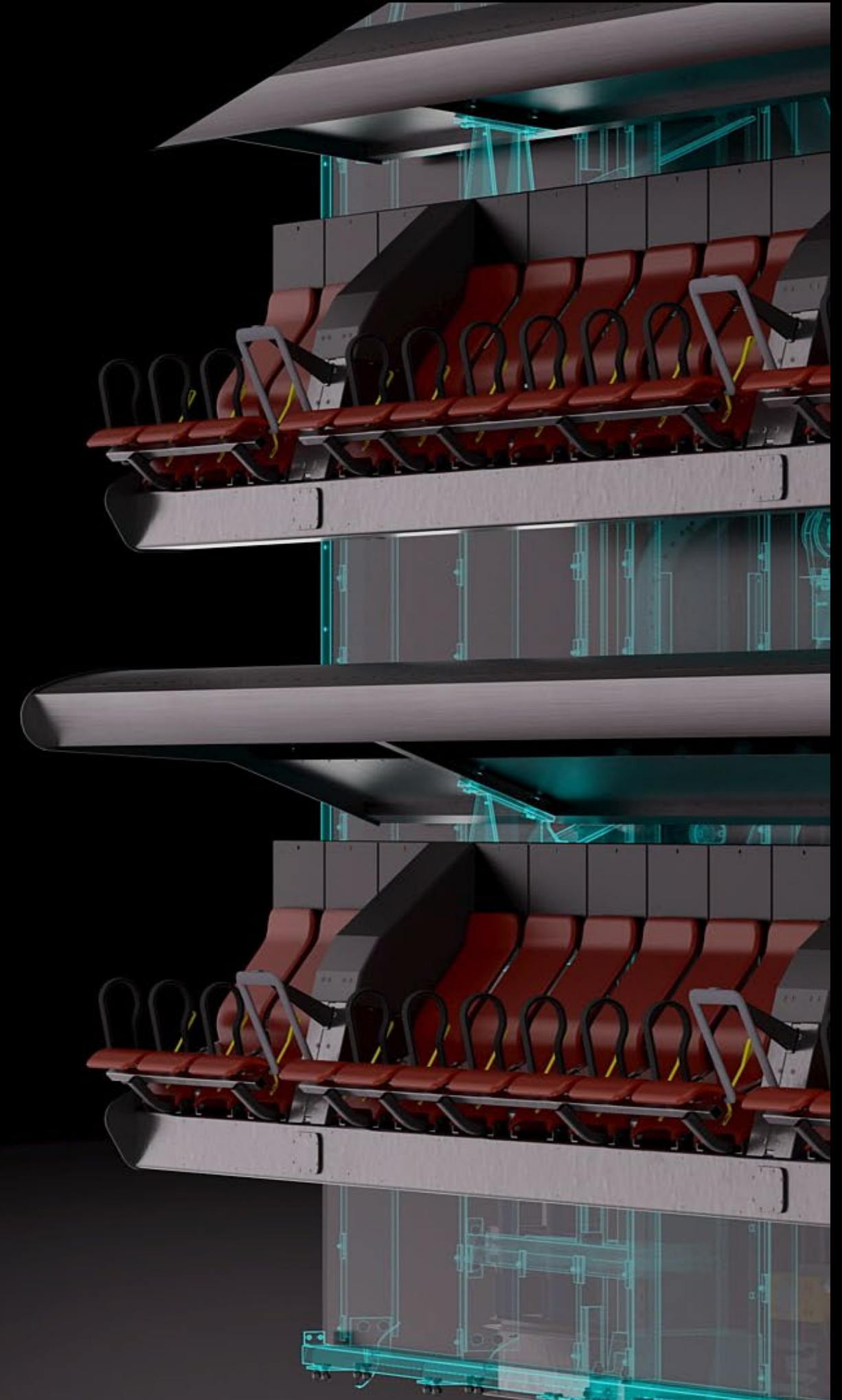
OK Cancel +





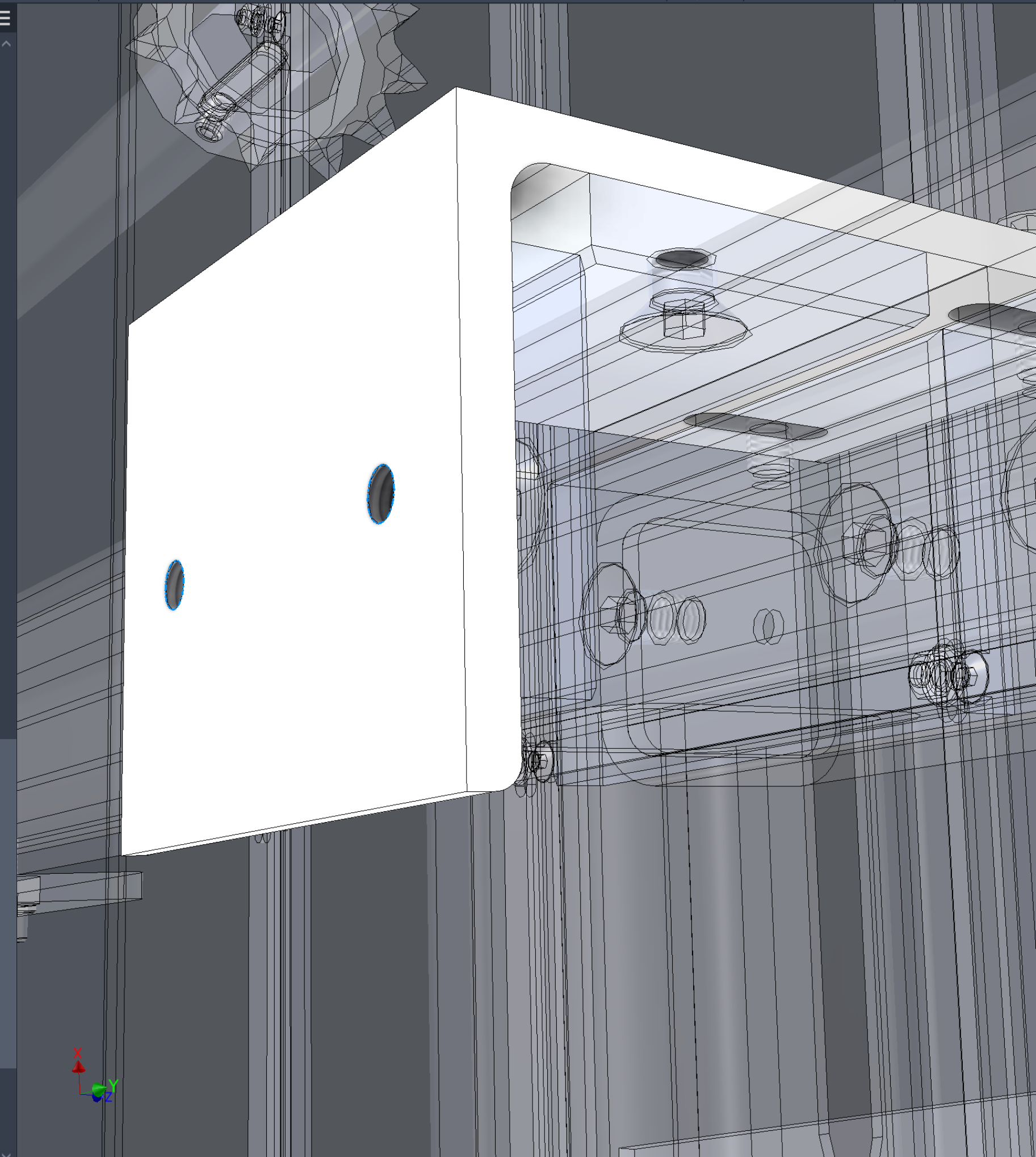
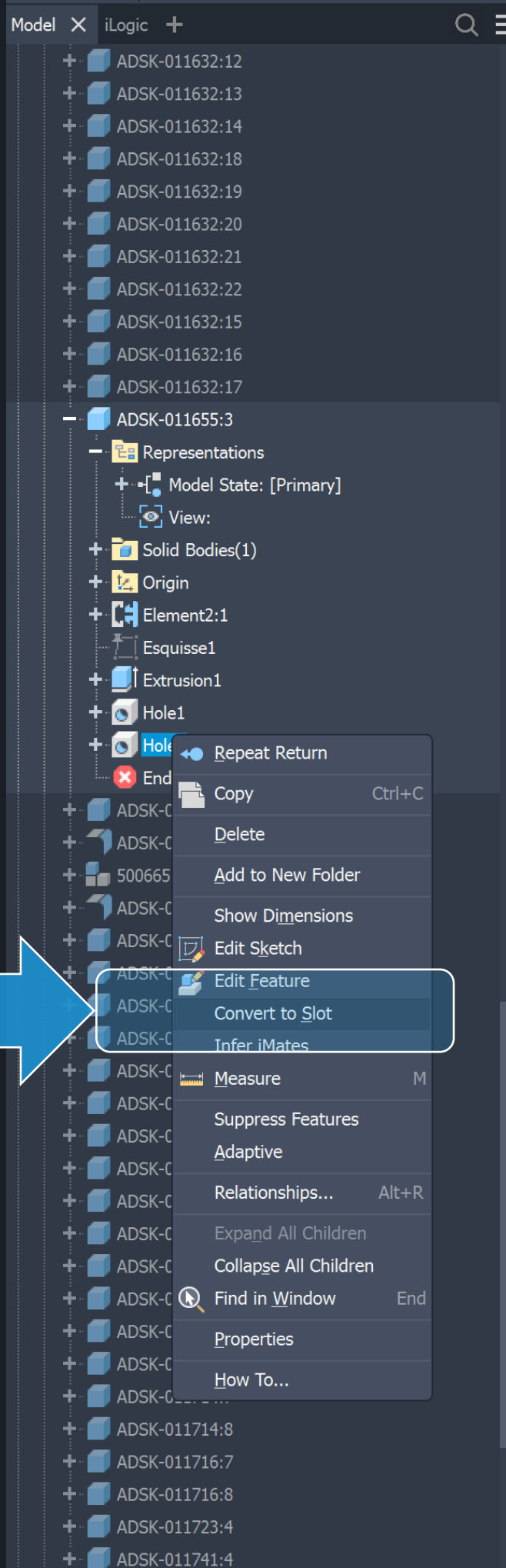
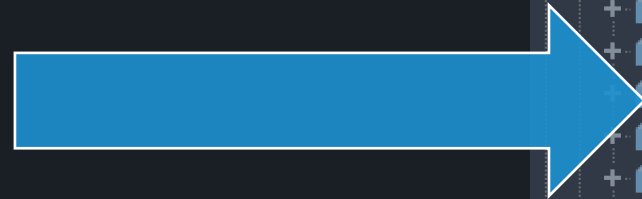
Inventor Professional

Slot feature



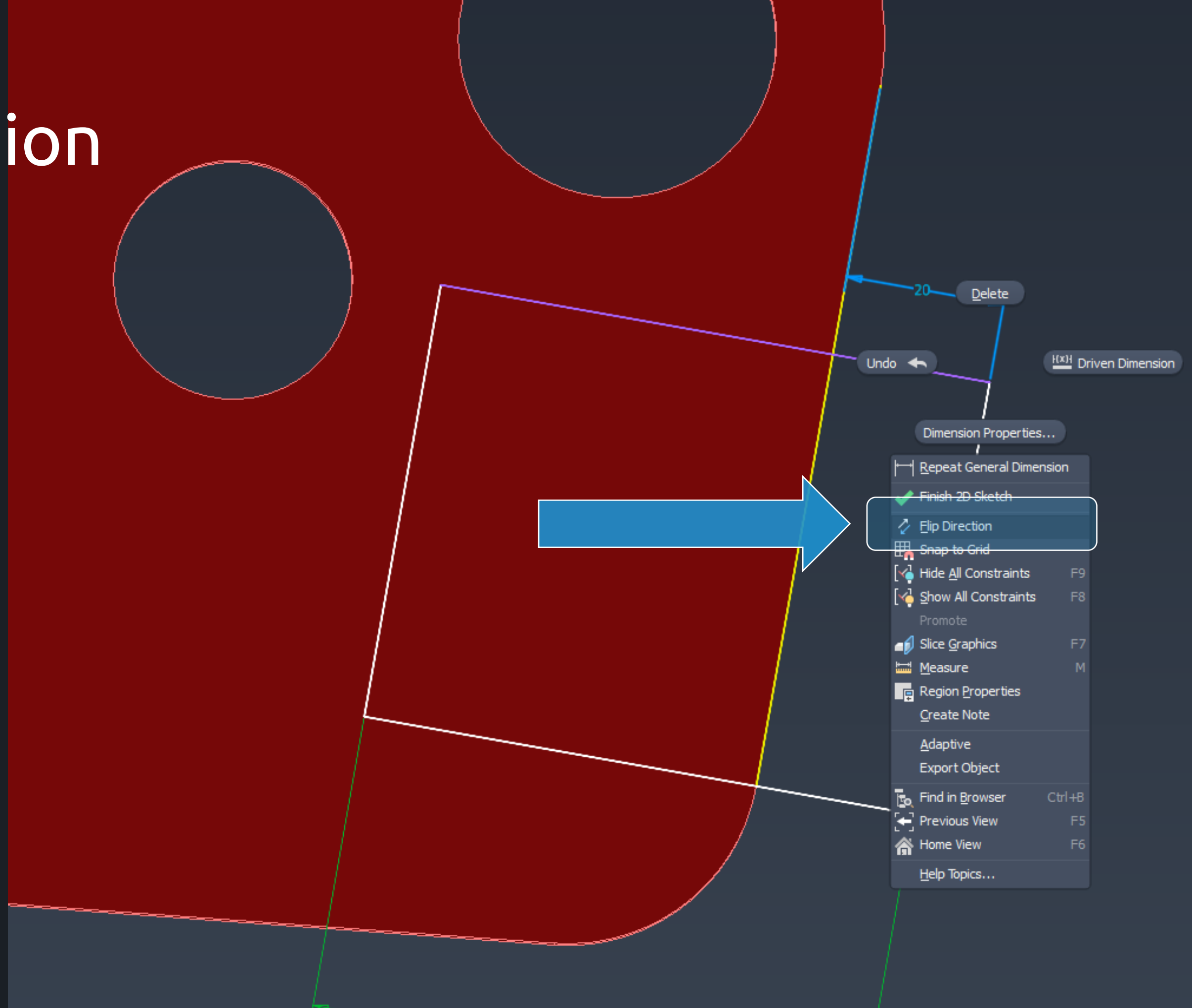
# Slot feature

Convert hole features to slot features



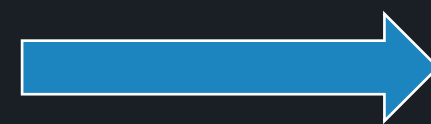
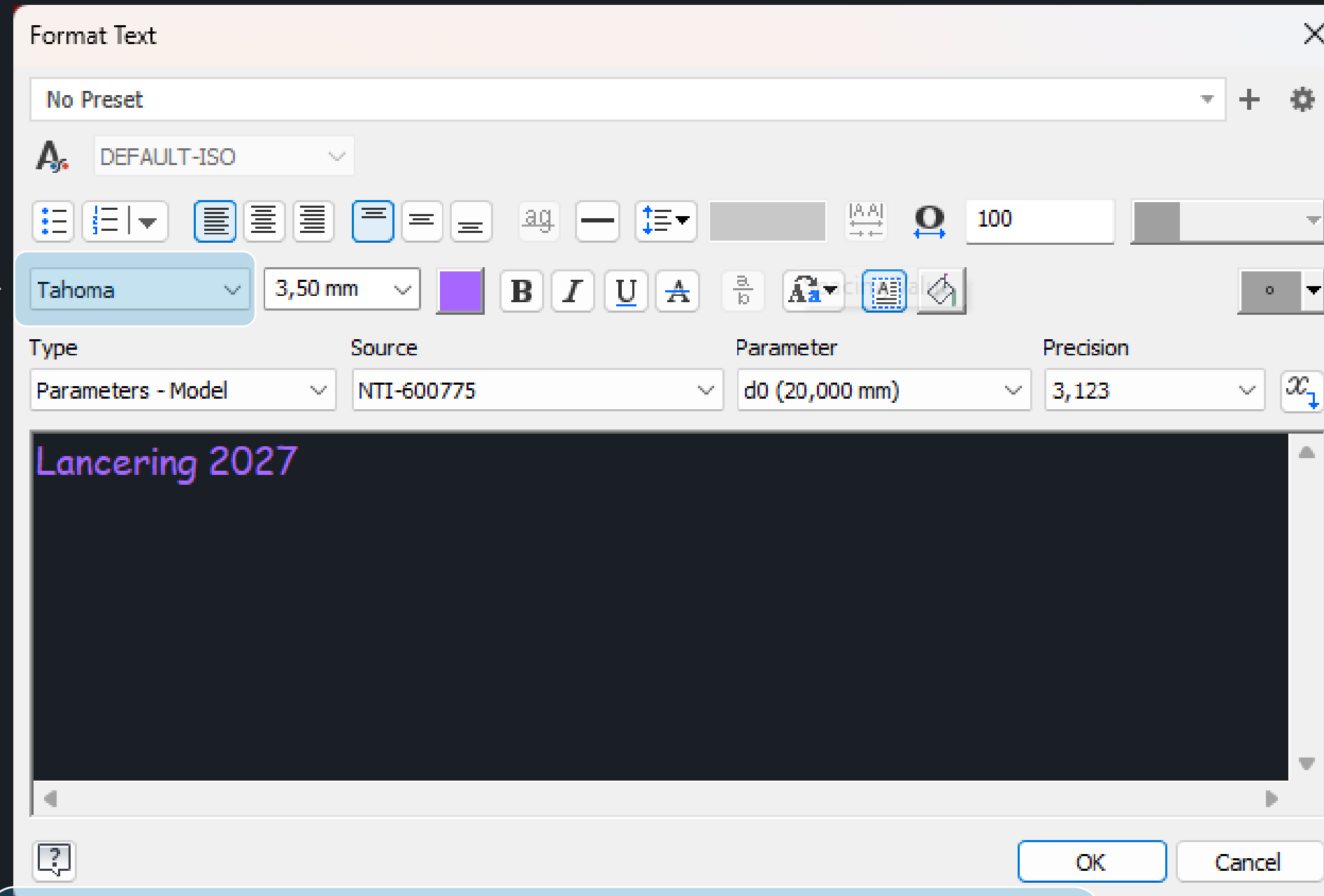
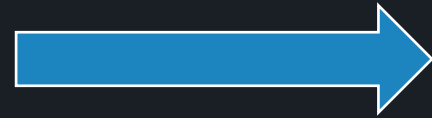
# Flip sketch dimension

Ability to reverse or flip a sketch dimension to opposite side.  
2D sketch only.



# Preview for sketch text

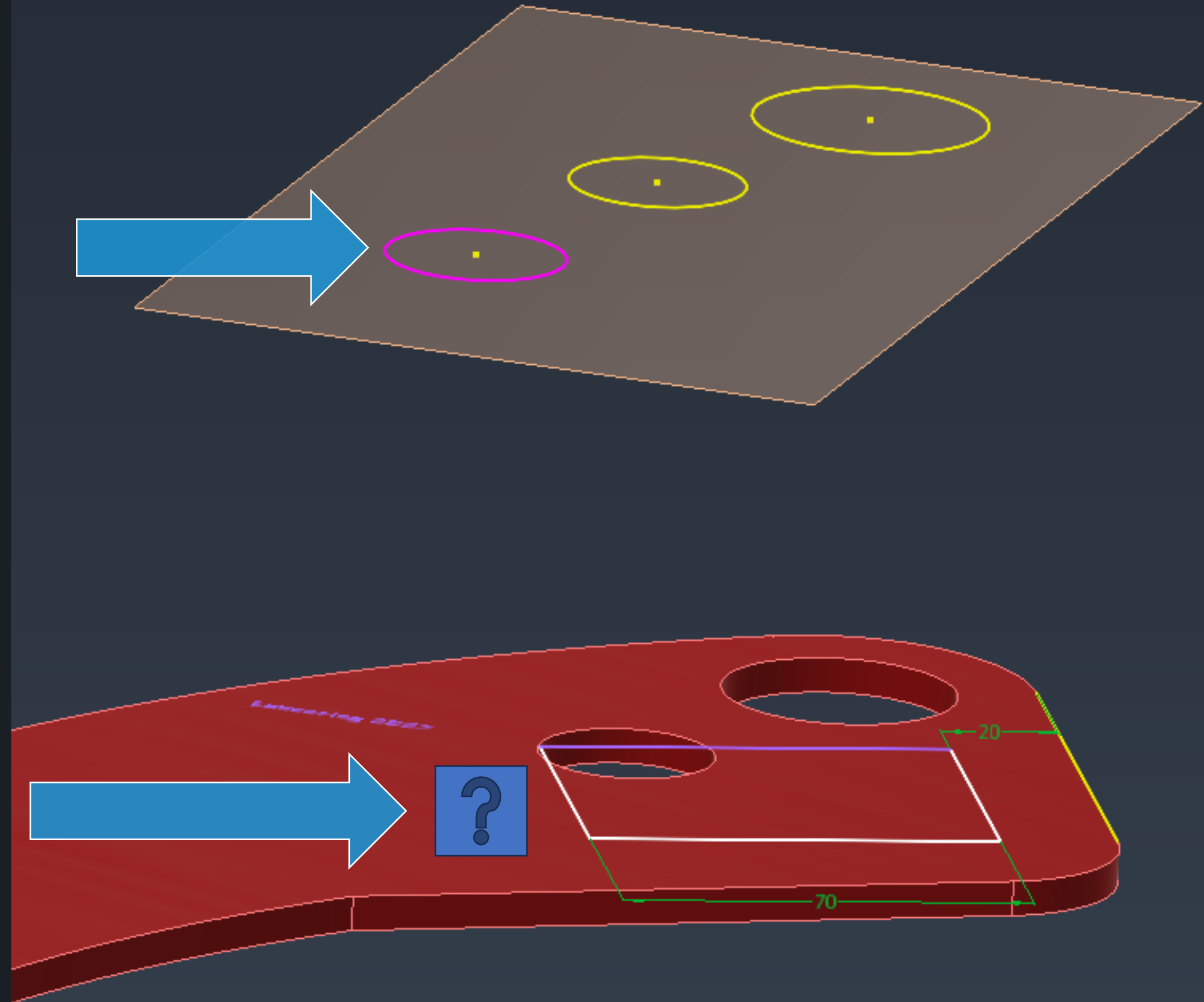
- We have improved the usability of the sketch text command and added a preview so that the user can intuitively see the result.
- Supported environment:
  - Part environment
  - Assembly environment
  - Drawing environment
  - Plat Pattern/Sheet Metal environment



Lancing 2027

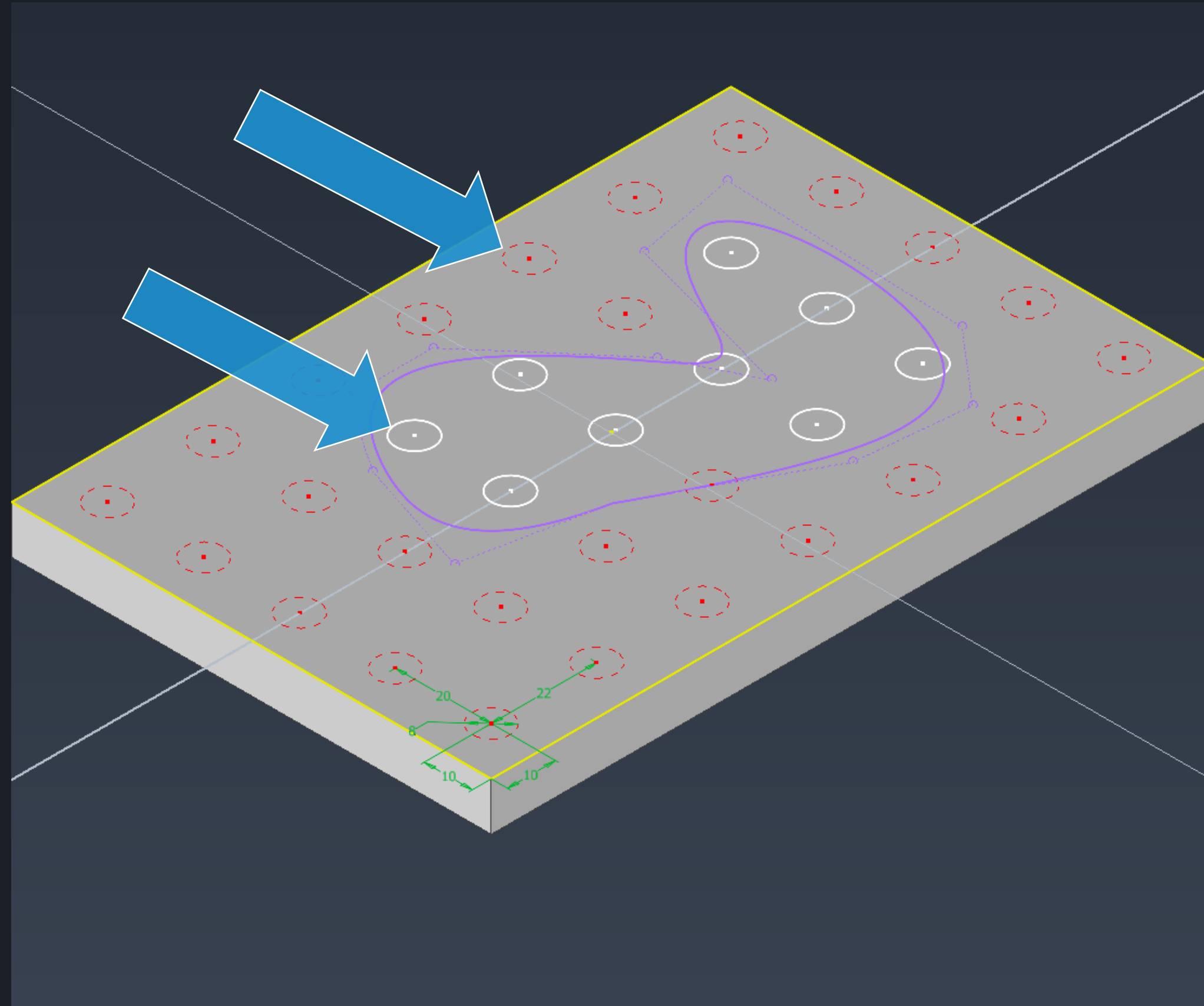
# Support for replacing sketch references

- Previously, when a reference (projected geometry) was lost, the user had to delete it and select a new one to constrain the geometry.
- Now, Inventor supports 'Replace Reference' for projected sketch geometry.
- For a single projected geometry in a sketch, the right-click context menu option "Replace Reference" is available.
- Geometry types can only be replaced with the same type of geometry —line for line and circle for circle.



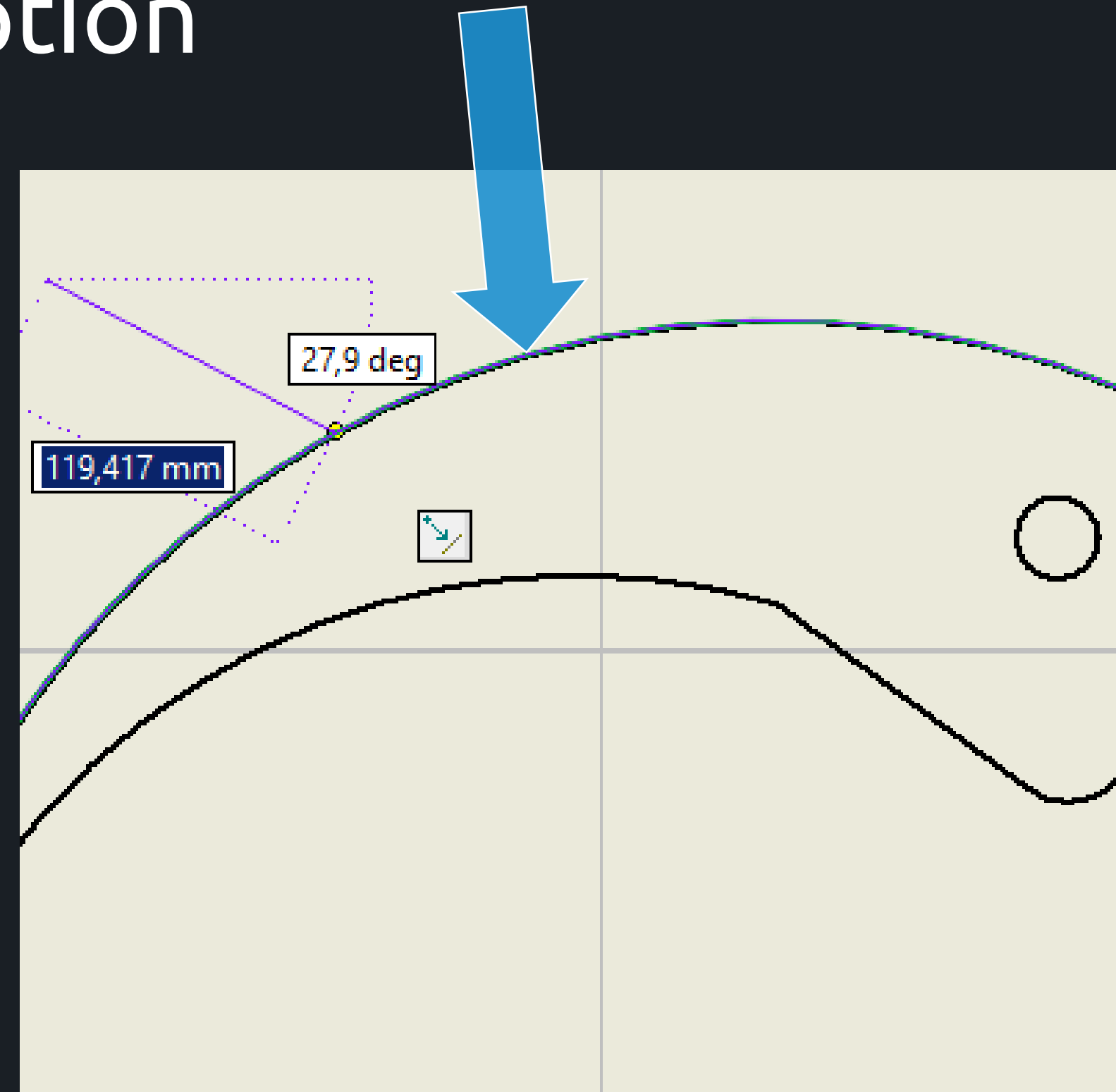
# Display sketch patterns

When a sketch entity is patterned with a boundary, entities will be displayed in different colors if they are outside the boundary.



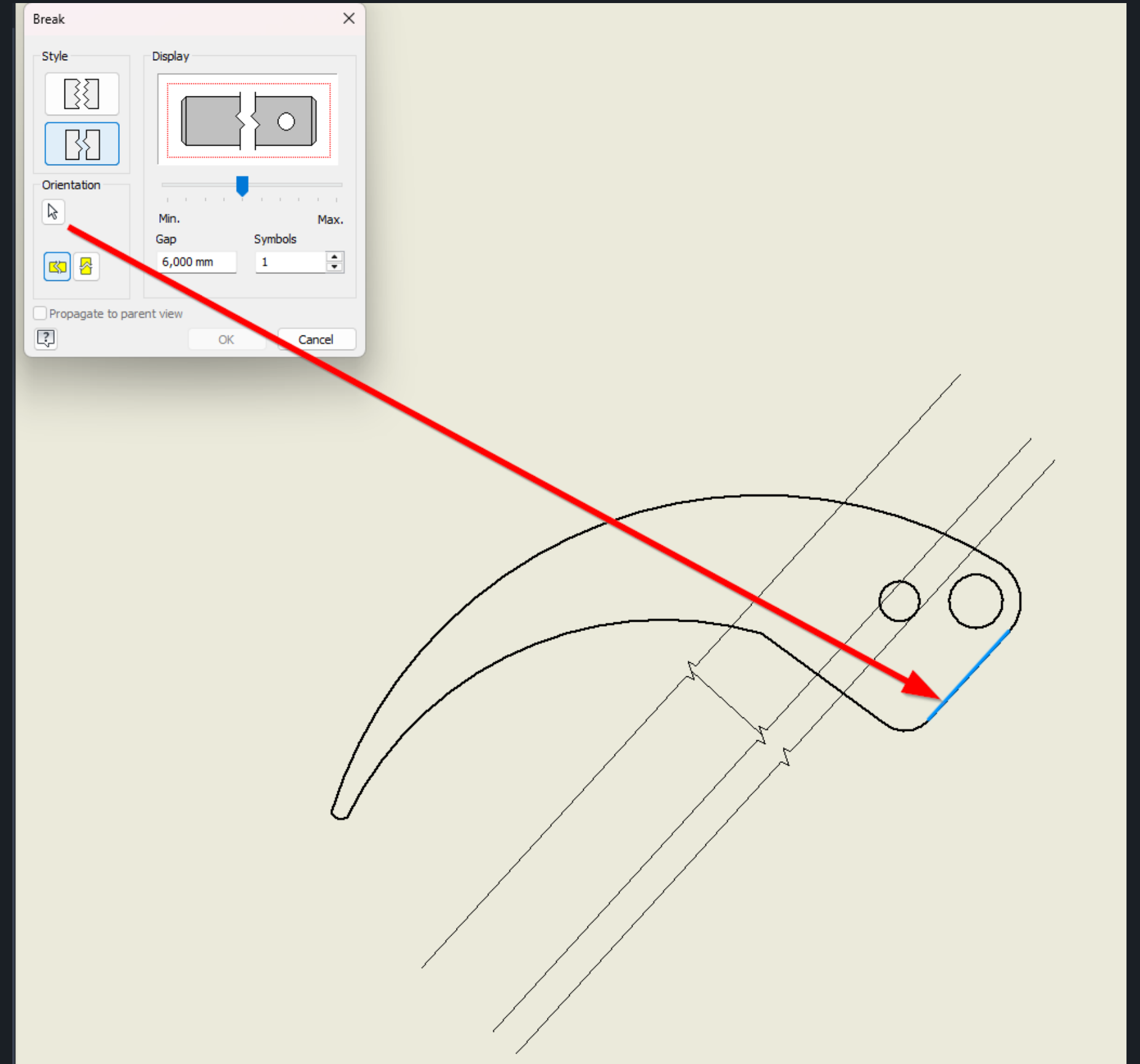
# Enable "Autoproject" context option

- Just like Part sketch environment, now Drawing view sketch also has a context menu option to 'Autoproject' when creating a geometry.
- The "Autoproject" context option should be linked to and toggle the Application option "Autoproject edges during curve creation"



# Aligned Broken Views

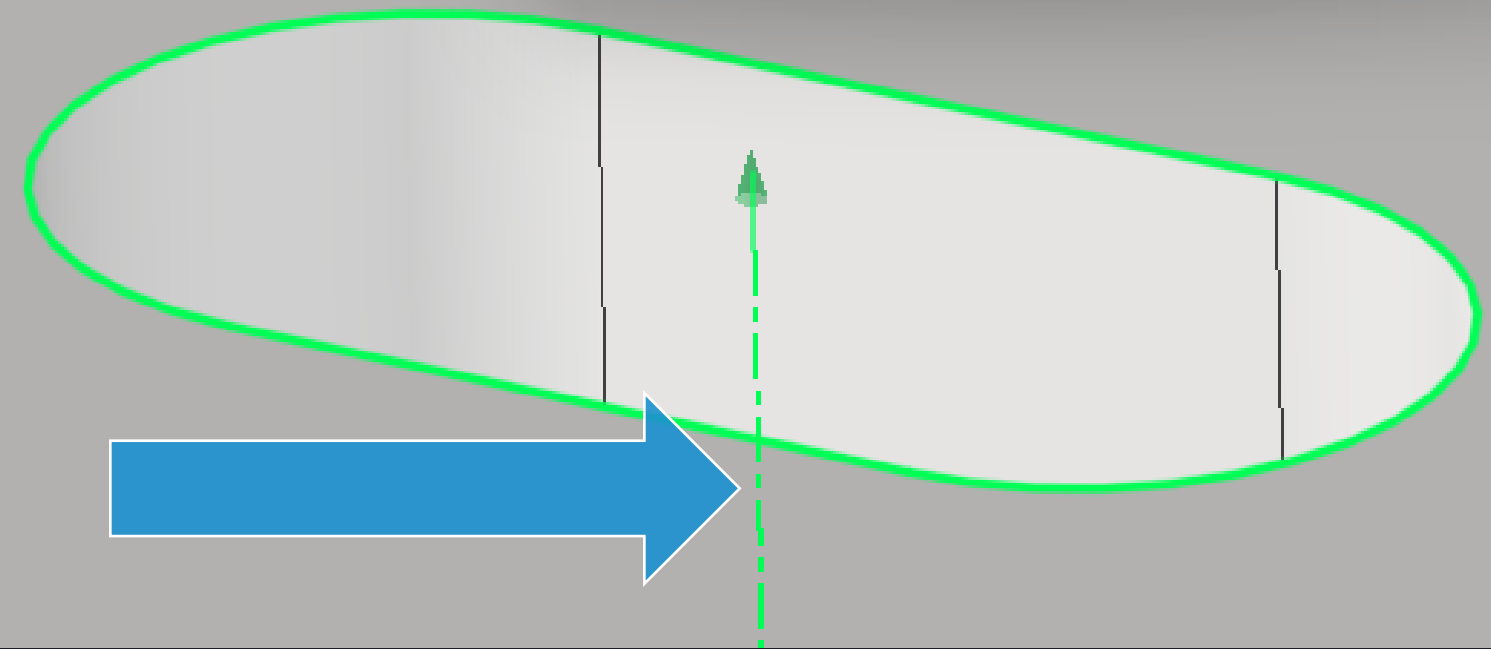
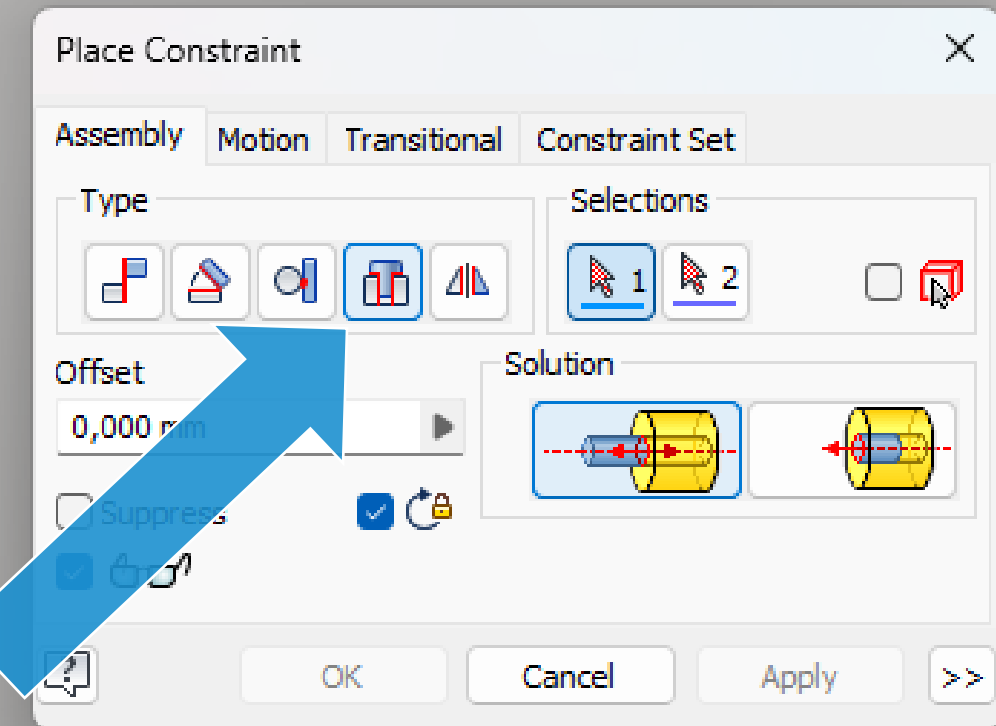
- When placing Broken Views, Inventor now allows you to align breaks to the view geometry.
- A new edge selector button has been added to the Orientation section of the Break dialog, allowing you to select the view geometry to which it should be.



# Constraint & iMate for Slot

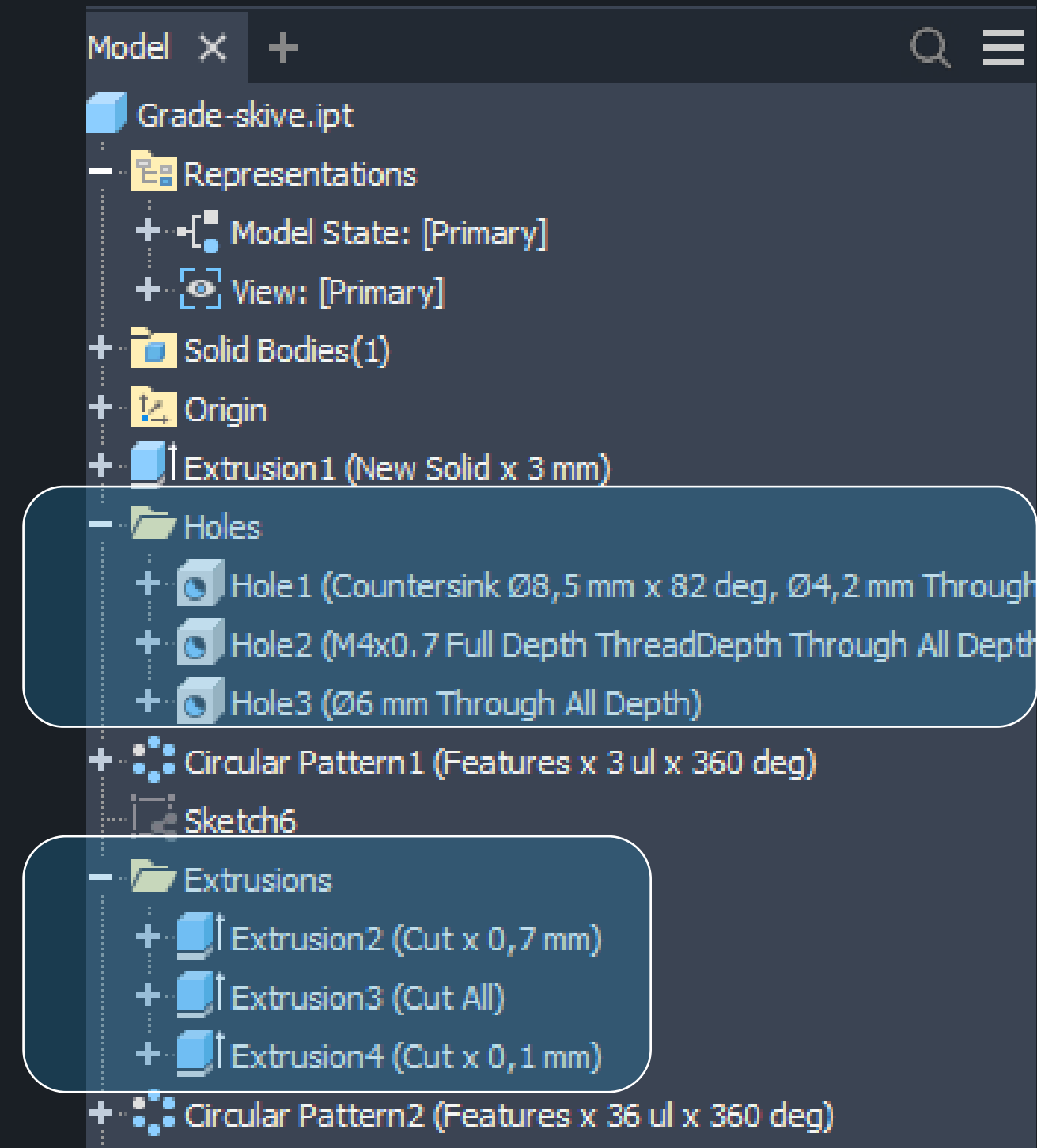
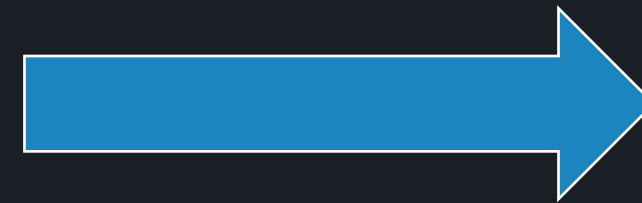
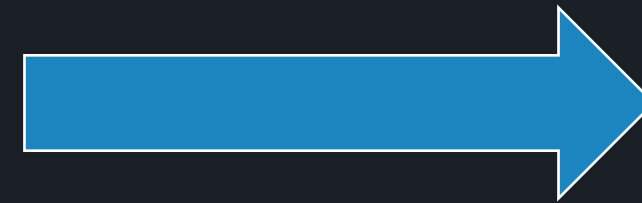
- Creating Insert constraint base on the Slot feature
  - We have supported creating the insert constraint base on the slot feature.
  - If we create the slot via slot command, Inventor will recognize the slot feature when creating the insert constraint.
  - Works for both part and assembly slot features.
- Support creating iMate for slot in the following ways:
  - Manage Tab -> iMate -> switch insert type -> select the slot feature
  - Check the iMate option on Slot command property panel while creating the slot.
  - Infer iMates

**NOTE!**  
Only with Slot Feature



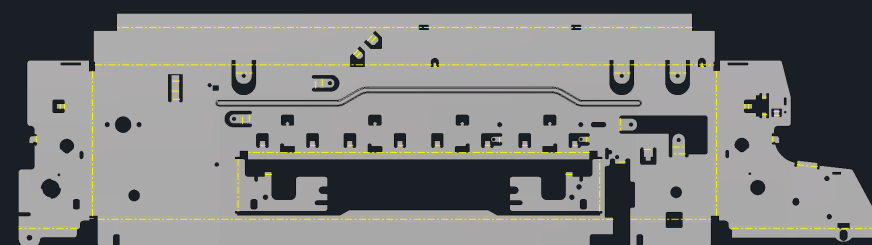
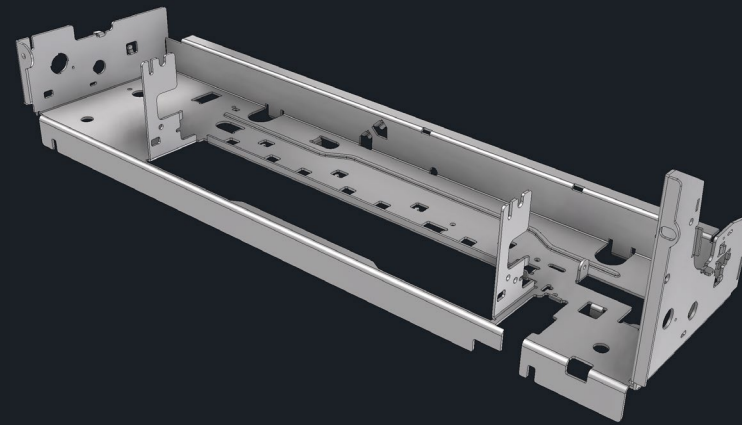
# Provided the ability to create a browser folder in part

- Create New Folder
- Add to New Folder
- Drag and Drop Objects
- Delete Folder
- Select Folder
- Expand Folders
  
- API Support



# Sheet metal

## Parameter synchronization



Parameters: Frame, Cartridge Body (Finished)

Type Keyword...

Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tolerance	Model Value	Key	Export	Comment
d879	Sketch11	mm	25.8 mm	25.800000	<Default	25.800000	<input type="checkbox"/>	<input type="checkbox"/>	
d880	Sketch11	mm	6.151 mm	6.150920	<Default	6.150920	<input type="checkbox"/>	<input type="checkbox"/>	
d881	Sketch11	mm	5.273 mm	5.272691	<Default	5.272691	<input type="checkbox"/>	<input type="checkbox"/>	
d882	Sketch11	mm	1.400 mm	1.400000	<Default	1.400000	<input type="checkbox"/>	<input type="checkbox"/>	
d884	Sketch14	mm	0.1 mm	0.100000	<Default	0.100000	<input type="checkbox"/>	<input type="checkbox"/>	
d885	Sketch14	mm	0.1 mm	0.100000	<Default	0.100000	<input type="checkbox"/>	<input type="checkbox"/>	
d886	Sketch14	mm	0.1 mm	0.100000	<Default	0.100000	<input type="checkbox"/>	<input type="checkbox"/>	
d887	Sketch14	mm	0.1 mm	0.100000	<Default	0.100000	<input type="checkbox"/>	<input type="checkbox"/>	
d889	Cut29	mm	Thickness	1.270000	<Default	1.270000	<input type="checkbox"/>	<input type="checkbox"/>	
d890	Cut29	deg	0 deg	0.000000	<Default	0.000000	<input type="checkbox"/>	<input type="checkbox"/>	
Reference Parameters									
d209		mm	2.000 mm	2.000000	<Default	2.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d306		mm	9.081 mm	9.080729	<Default	9.080729	<input type="checkbox"/>	<input type="checkbox"/>	
d815		mm	18.200 mm	18.200000	<Default	18.200000	<input type="checkbox"/>	<input type="checkbox"/>	
d816		mm	7.500 mm	7.500000	<Default	7.500000	<input type="checkbox"/>	<input type="checkbox"/>	
d888	Face13, Fac...	mm	Thickness	1.270000	<Default	1.270000	<input type="checkbox"/>	<input type="checkbox"/>	
User Parameters									
Notes		Text	Sheet_metal_example				<input type="checkbox"/>	<input type="checkbox"/>	

Buttons: Add Numeric, Update, Purge Unused, Import from XML, Link, Immediate Update, Export to XML, Reset Tolerance, Done

Parameters: Frame, Cartridge Body (Finished)

Type Keyword...

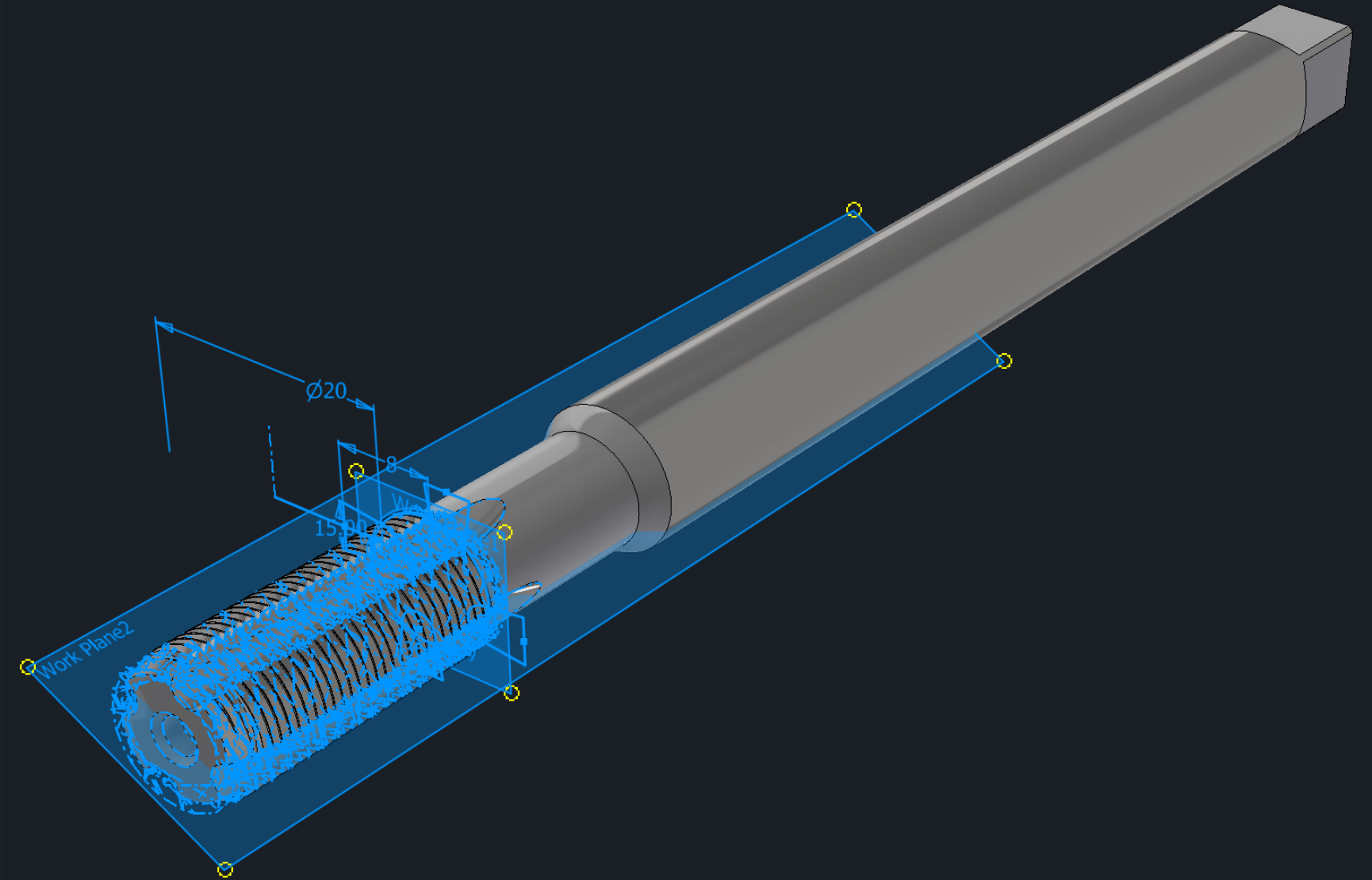
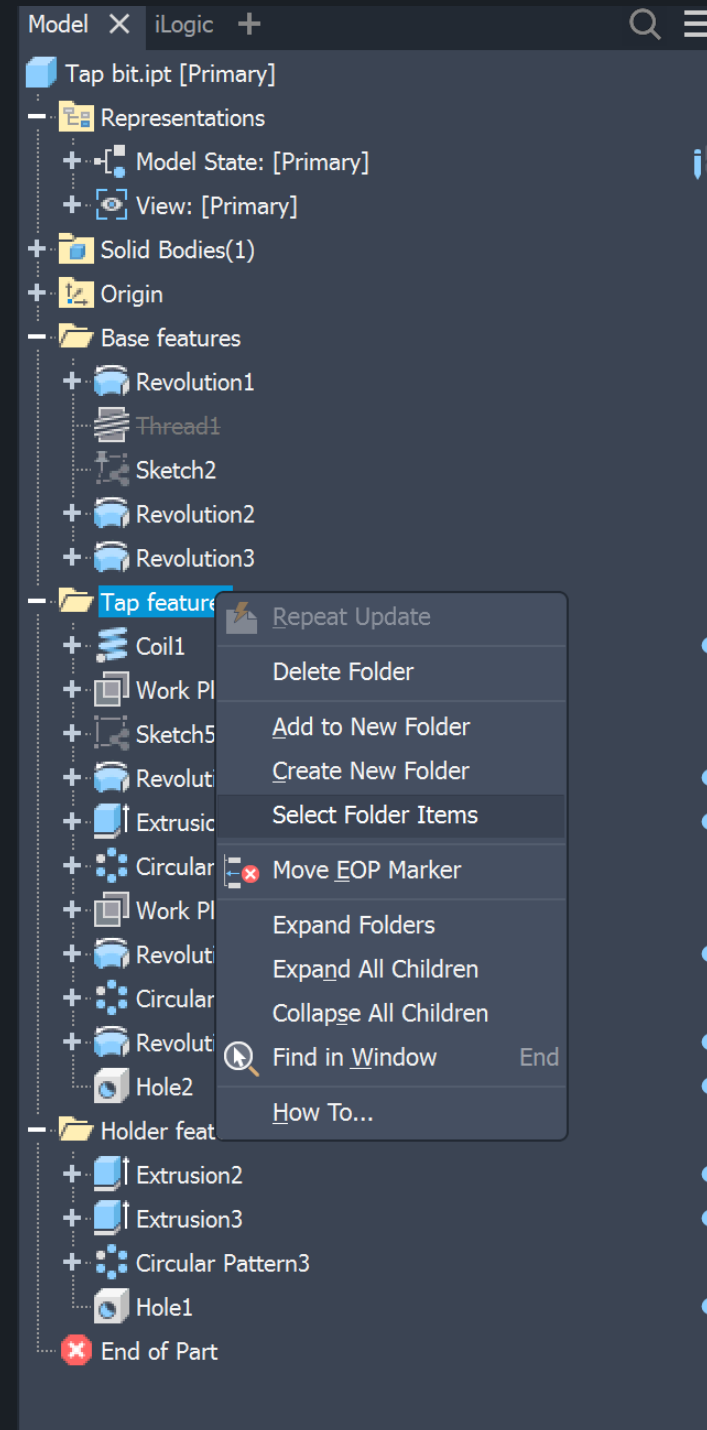
Parameter Name	Consumed by	Unit/Type	Equation	Nominal Value	Tolerance	Model Value	Key	Export	Comment
Model Parameters									
Reference Parameters									
BendRadius		mm	0.635 mm	0.635000		0.635000	<input type="checkbox"/>	<input type="checkbox"/>	
BendReliefDepth		mm	0.635 mm	0.635000		0.635000	<input type="checkbox"/>	<input type="checkbox"/>	
BendReliefWidth		mm	1.270 mm	1.270000		1.270000	<input type="checkbox"/>	<input type="checkbox"/>	
CornerReliefSize		mm	5.080 mm	5.080000		5.080000	<input type="checkbox"/>	<input type="checkbox"/>	
GapSize		mm	1.270 mm	1.270000		1.270000	<input type="checkbox"/>	<input type="checkbox"/>	
JacobiRadiusSize		mm	0.635 mm	0.635000		0.635000	<input type="checkbox"/>	<input type="checkbox"/>	
MinimumRemnant		mm	2.540 mm	2.540000		2.540000	<input type="checkbox"/>	<input type="checkbox"/>	
Stiffner_Length		mm	65.000 mm	65.000000		65.000000	<input type="checkbox"/>	<input type="checkbox"/>	
Stiffner_Width		mm	3.000 mm	3.000000		3.000000	<input type="checkbox"/>	<input type="checkbox"/>	
Thickness		mm	1.270 mm	1.270000		1.270000	<input type="checkbox"/>	<input type="checkbox"/>	
TransitionRadius		mm	0.635 mm	0.635000		0.635000	<input type="checkbox"/>	<input type="checkbox"/>	
Notes		Text	Sheet_metal_example				<input type="checkbox"/>	<input type="checkbox"/>	
User Parameters									

Buttons: Add Numeric, Update, Purge Unused, Import from XML, Link, Immediate Update, Export to XML, Reset Tolerance, Done

# Part browser

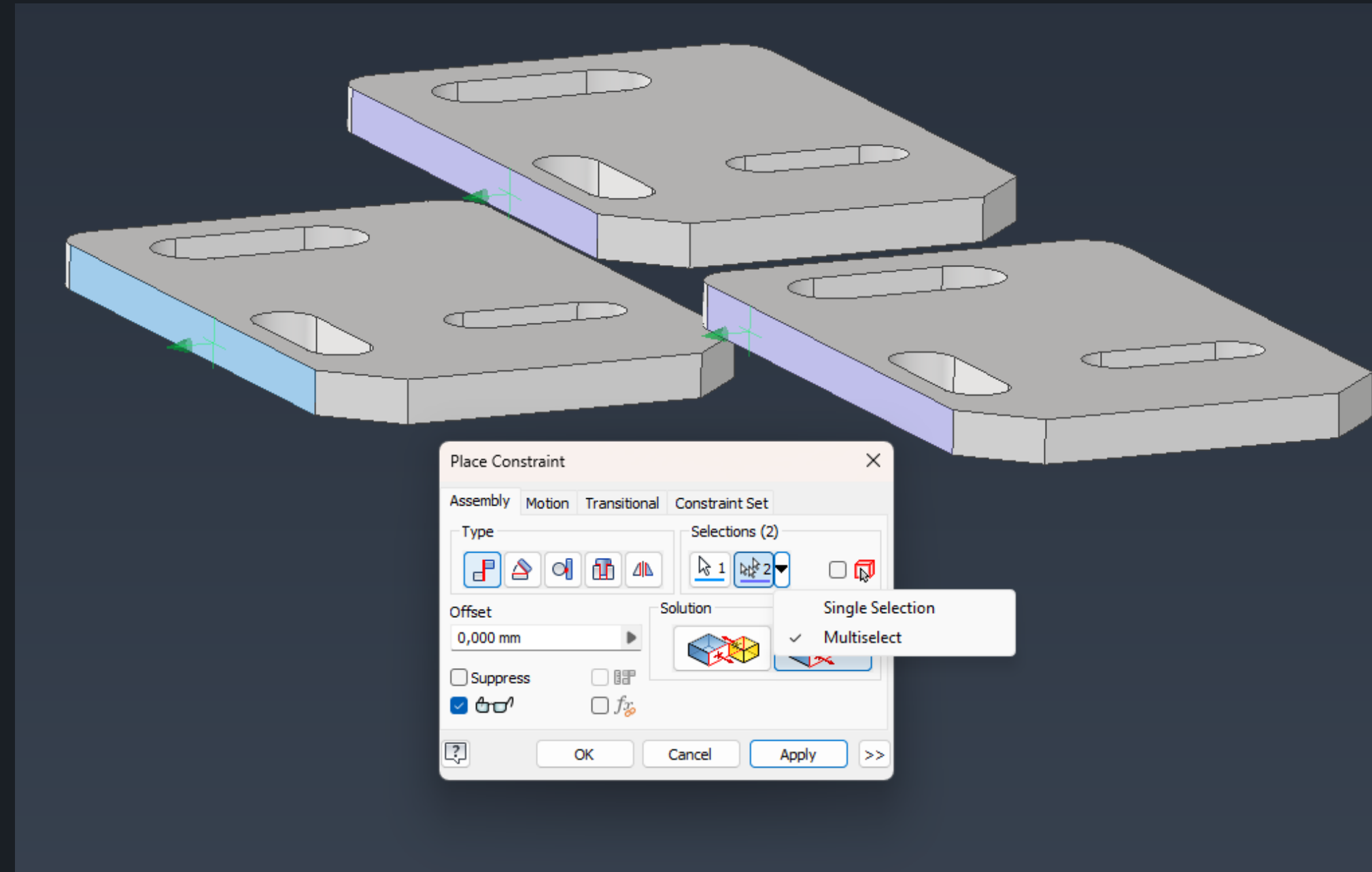
## Folder management

- Create new folders and subfolders
- Add to new folder
- Select folder items
- Expand folders



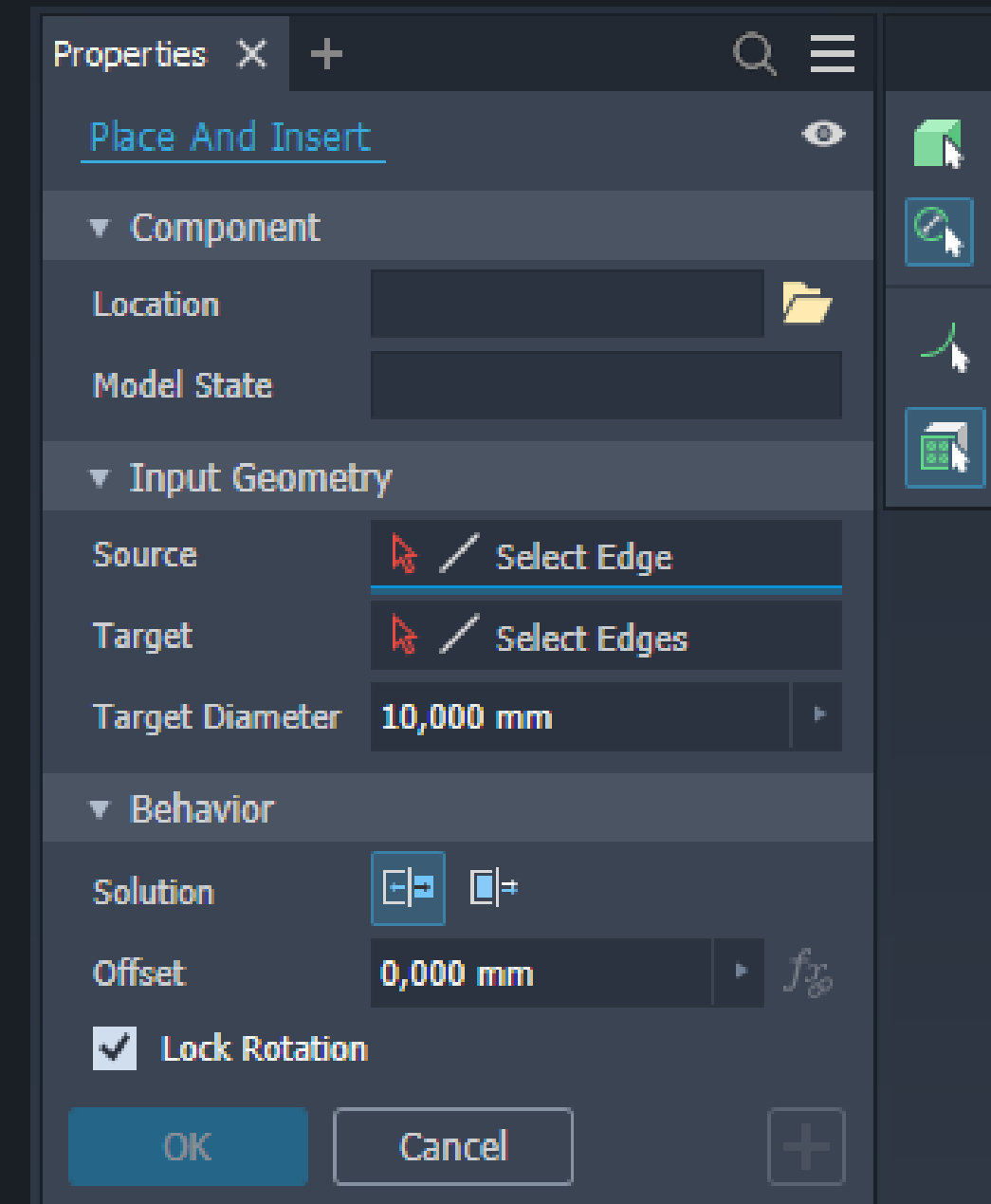
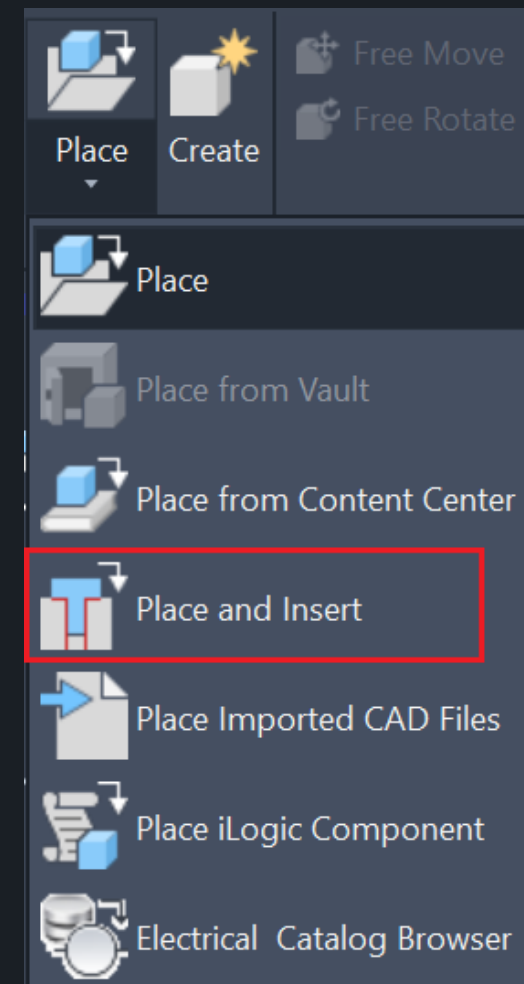
# Mate/Fulsh constraint supports multi selections

- We have supported multiselect constraints for Mate constraint. If you switch to another constraint type, the multi selection mode will be hidden.
- Support Pick part first .
- Switch to multi selection mode from the flyout menu.
- Provide the “Link Parameters” option to link all the offset parameters to the first constraint made in the operation
- Provide the number of selections for the second selector
- For editing workflow, multi selection mode will be disabled.

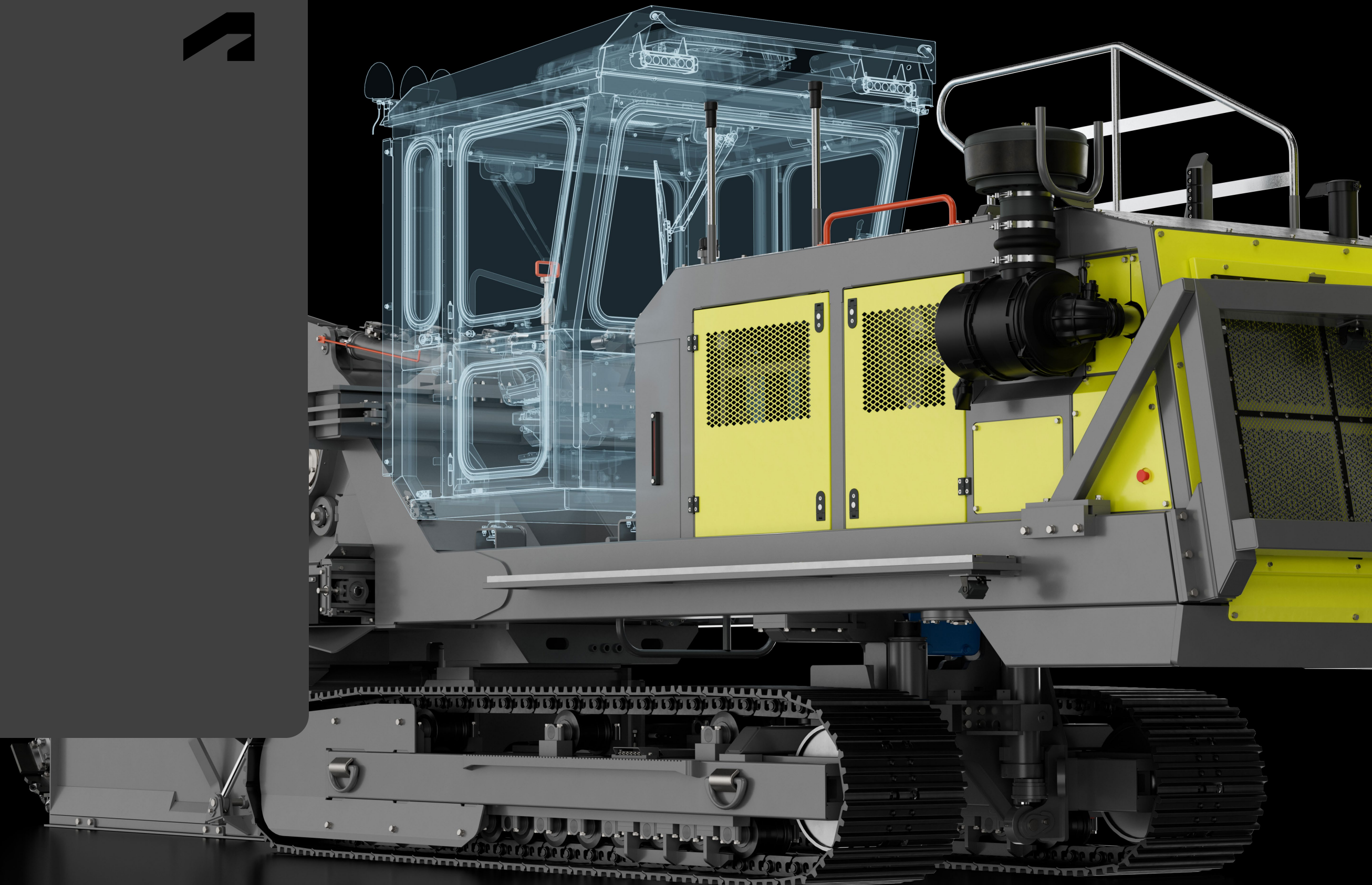


# Place and Insert Command for multiple insert constraints

- Place and Insert Command
  - We have added the Place and Insert Command to the dropdown menu for place in assembly environment.
- Browse the component which you want to place via the Browse button.
  - The selected model state will be shown on the panel.
- Source selector and Target selector:
  - Only supports selection of circular edges.
  - Clear the selector by clicking the cross button.
  - Press [Ctrl] or [Shift] + click to deselect the edge.
  - The Target selector supports multiple selections.
  - The offset value can be linked to the first value if check the Link button.
  - Supports click apply to create constraints and invoke the new panel.
- Size Filter
  - We have added the size filter to filter out the target edges that are outside the specified size.
- Pick Component First
  - We have added the Pick Component First button. When it is active, select a component before each target geometry selection so the immediate geometry selection is restricted only on the picked component.
- Can be used in combination with the size filter



# Assembly modeling



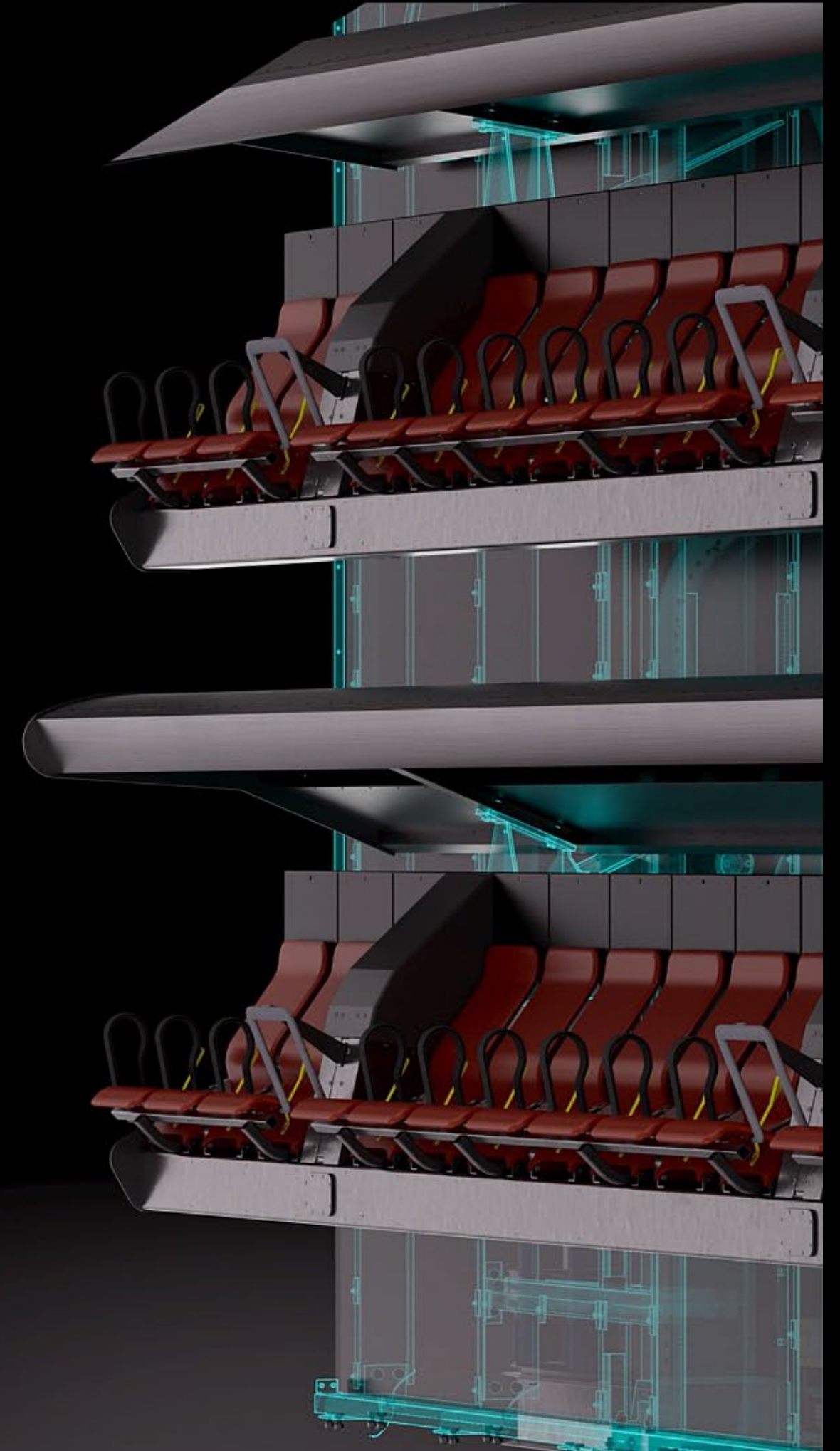
 AUTODESK

Image courtesy of Mastenbroek

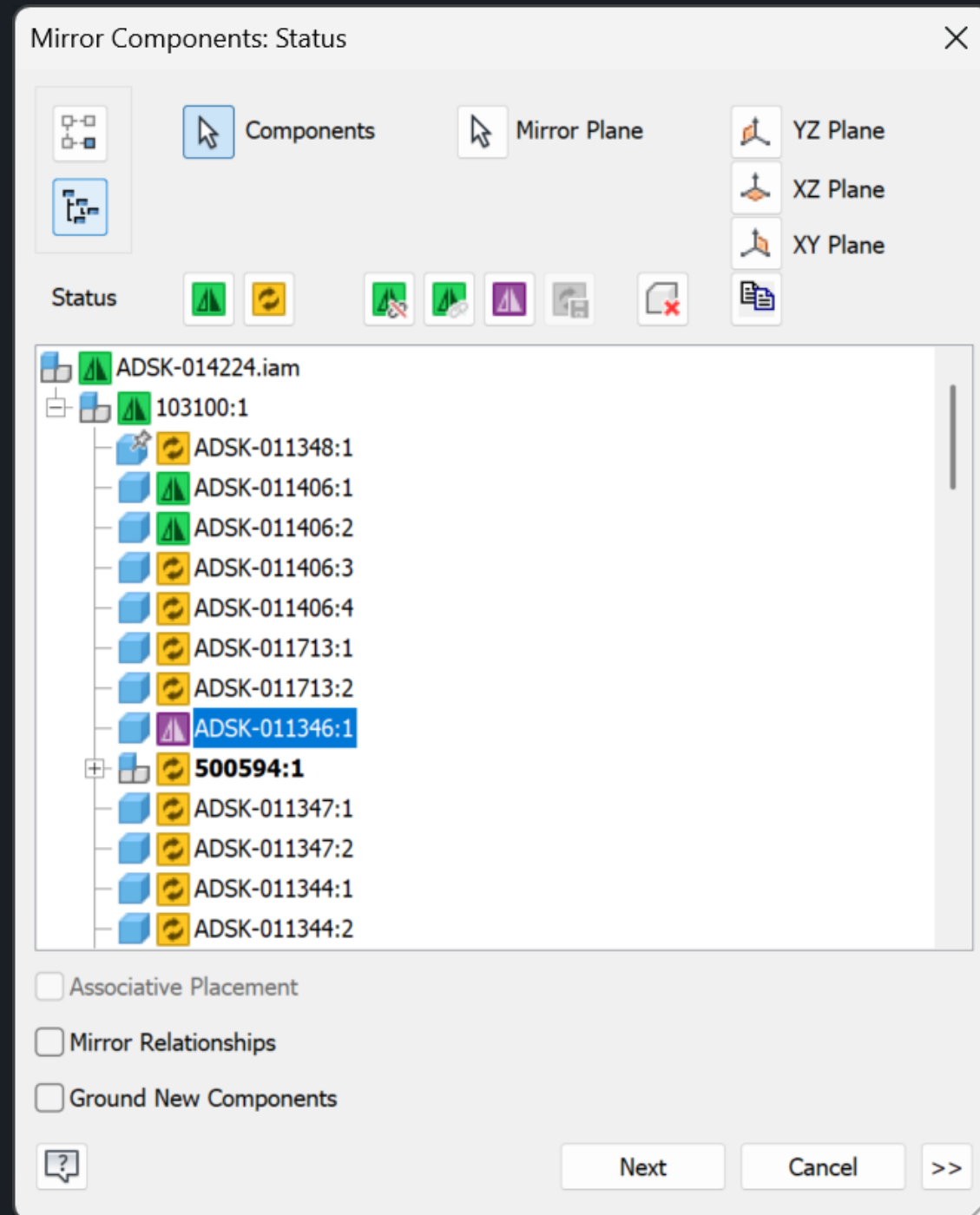


## Inventor Professional

Assembly mirror  
enhancements



# Assembly mirror



Legacy options  
Updated icons



Reuse



Mirror, derived body



Exclude

New options



Break associativity



Suppress associativity



Copy, add mirror feature

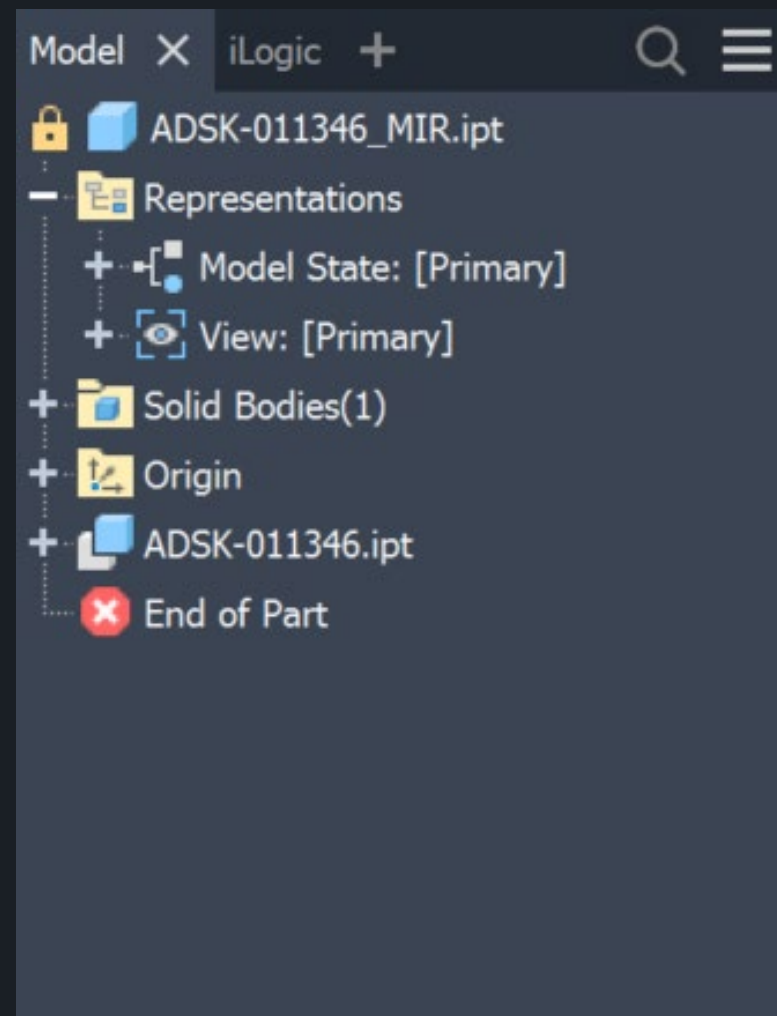


Reuse, but also save a copy

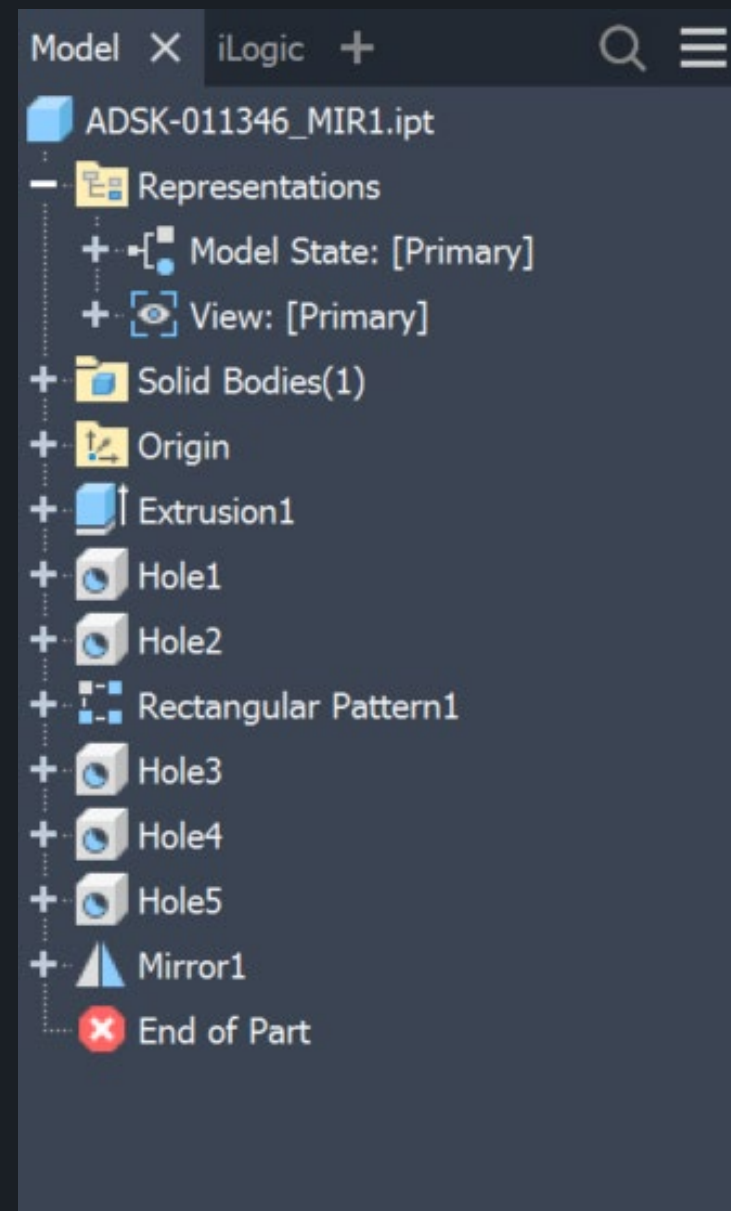
# Assembly mirror



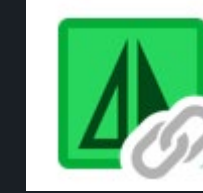
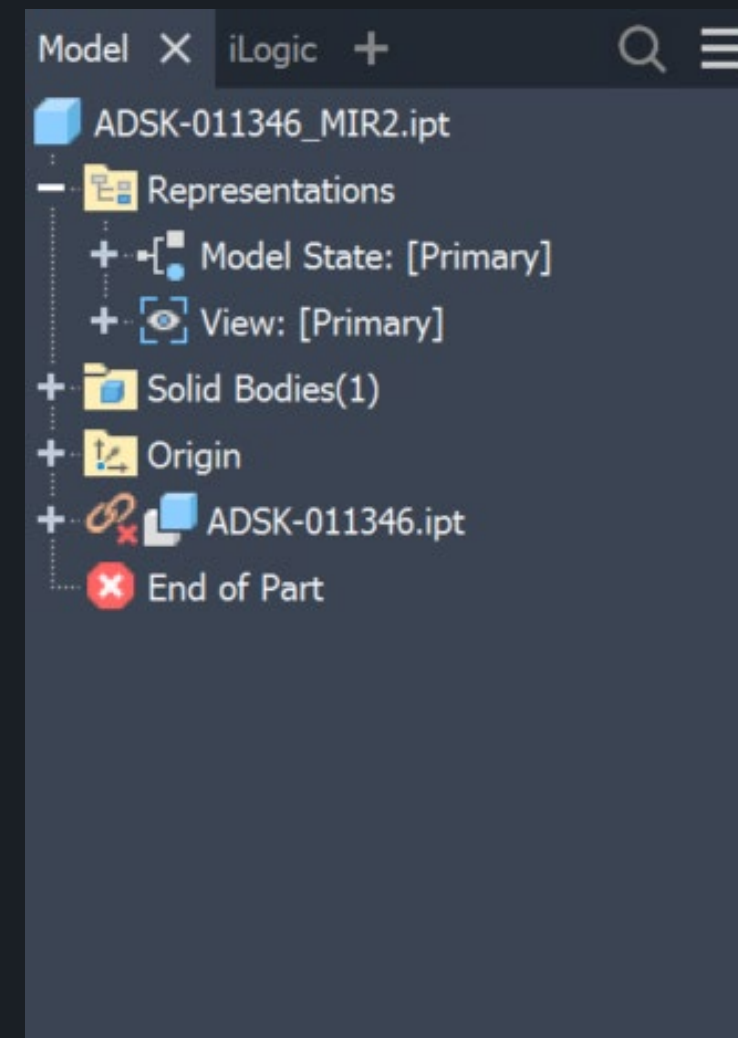
Create a mirrored / derived component



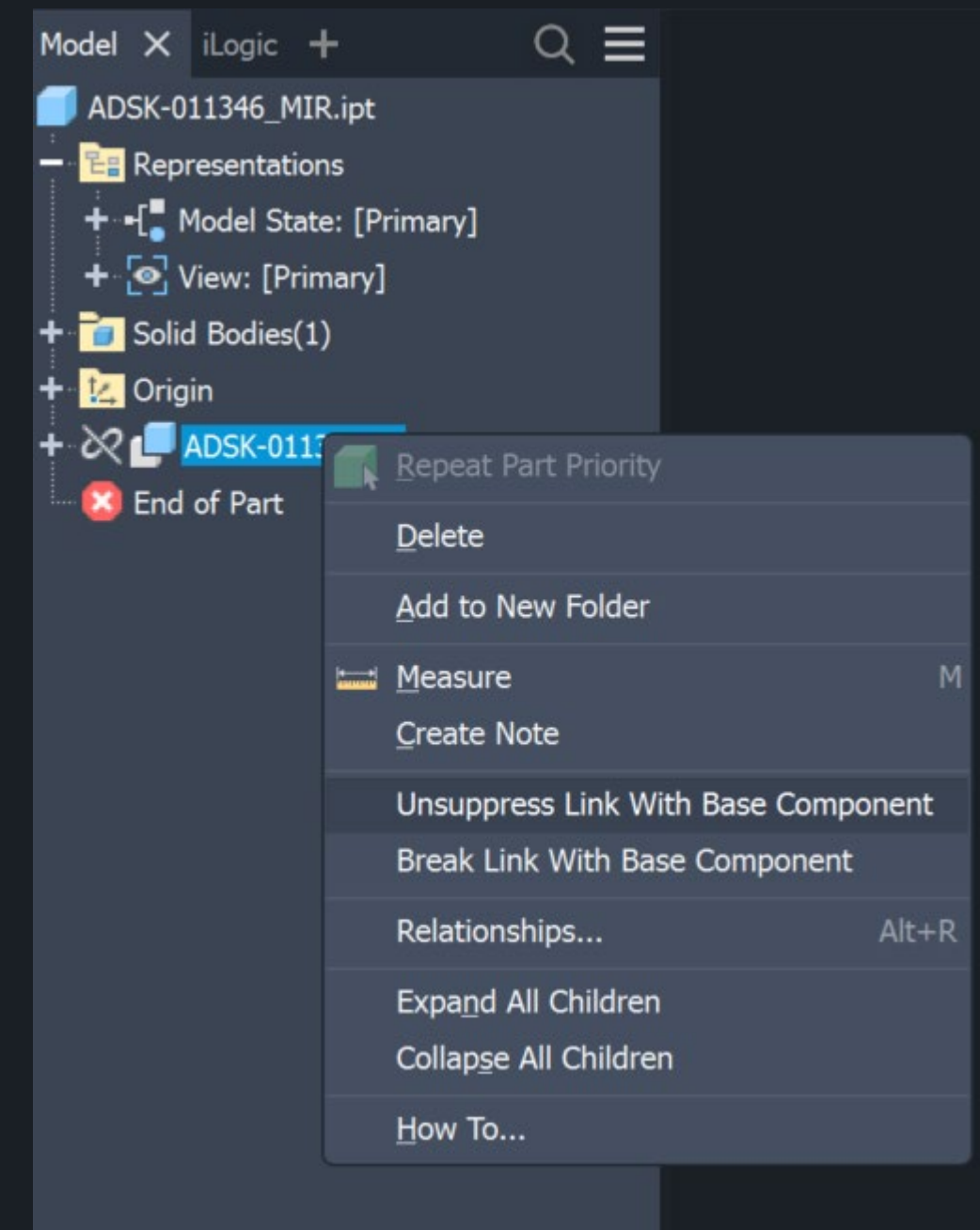
Create a mirrored component with a mirrored feature



Create a mirrored / derived and break the associativity

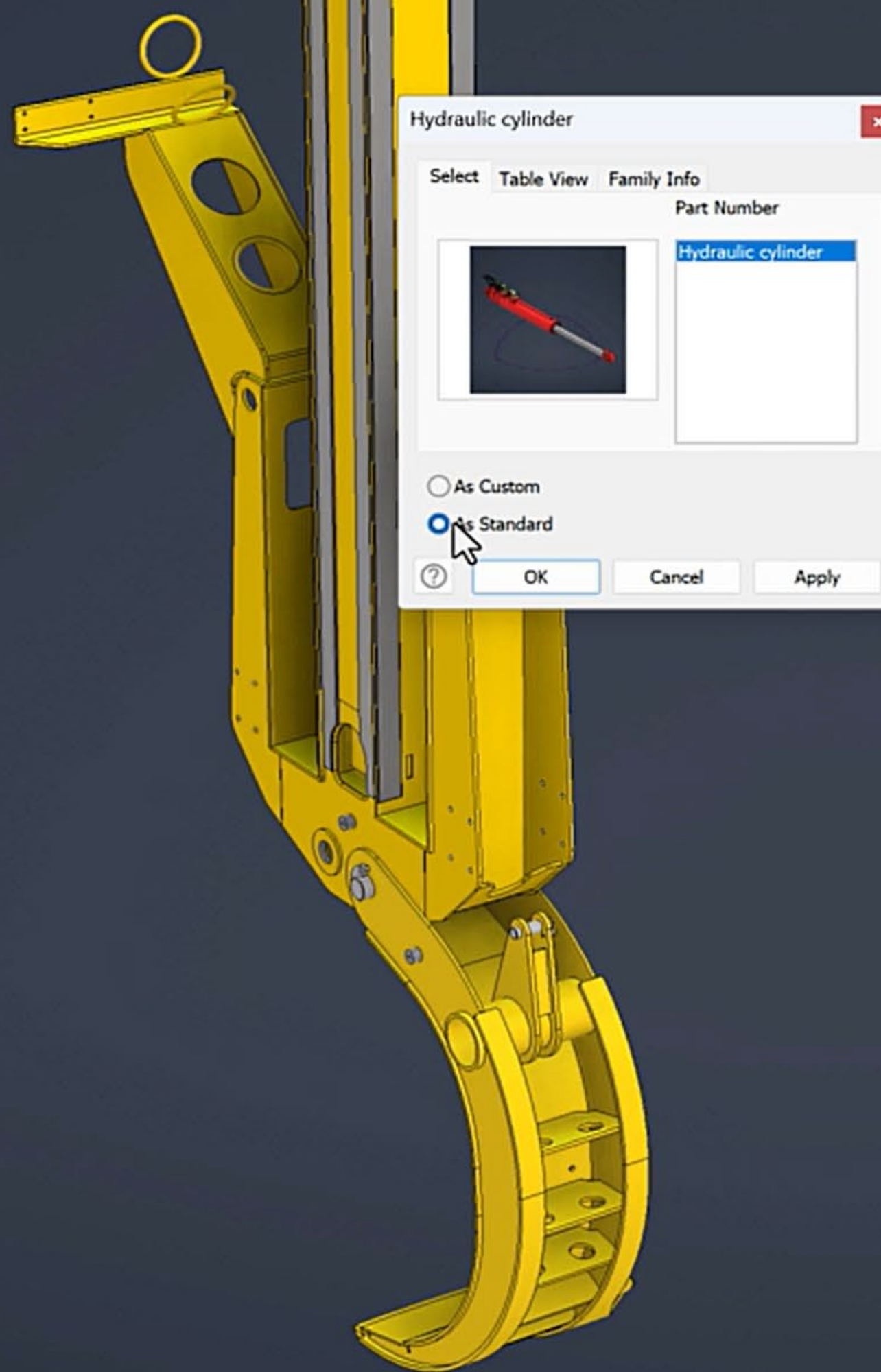


Create a mirrored / derived and suppress the associativity



# Content Center

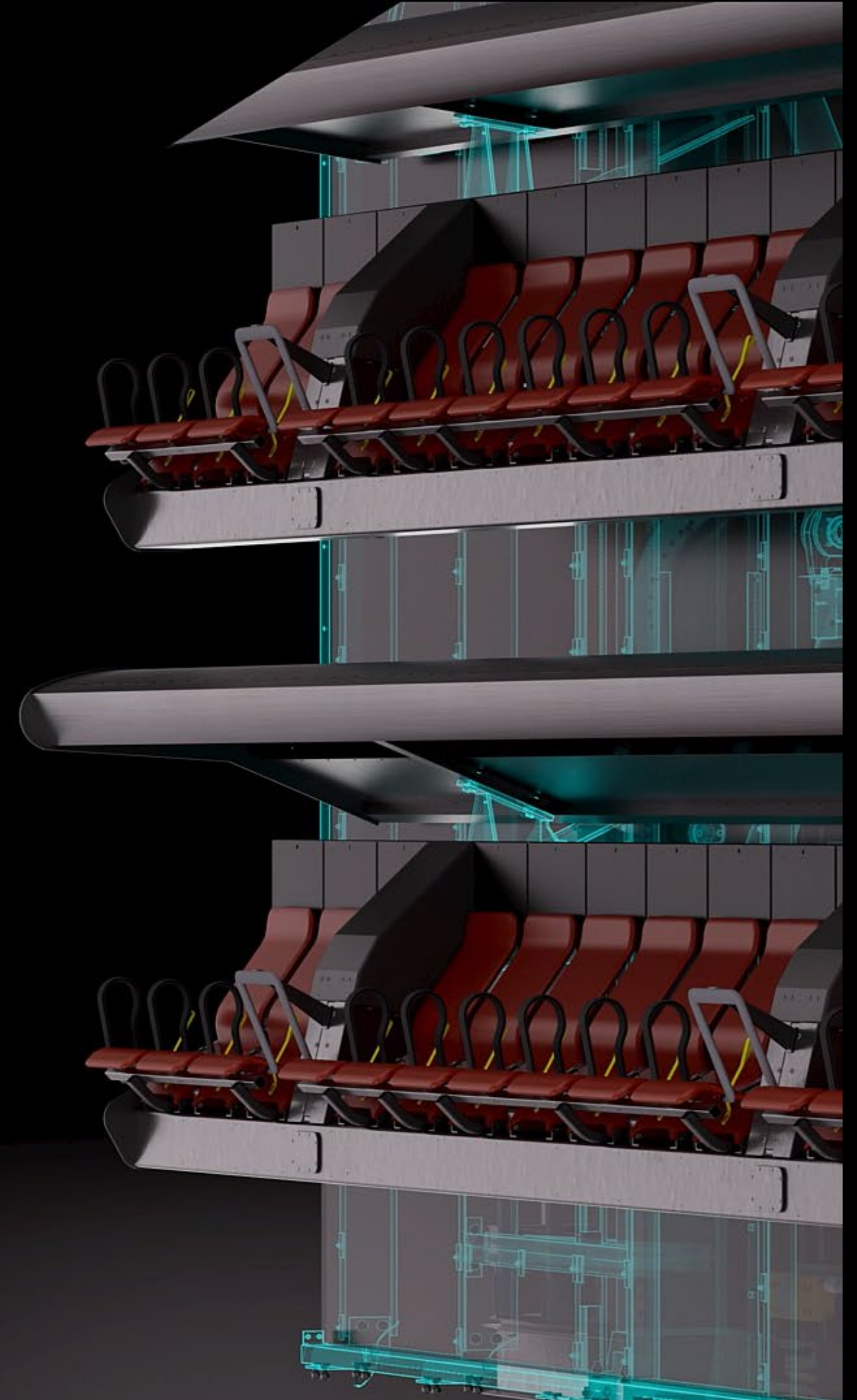
- Publish assemblies
- Replace templates for assembly families





# Inventor Professional

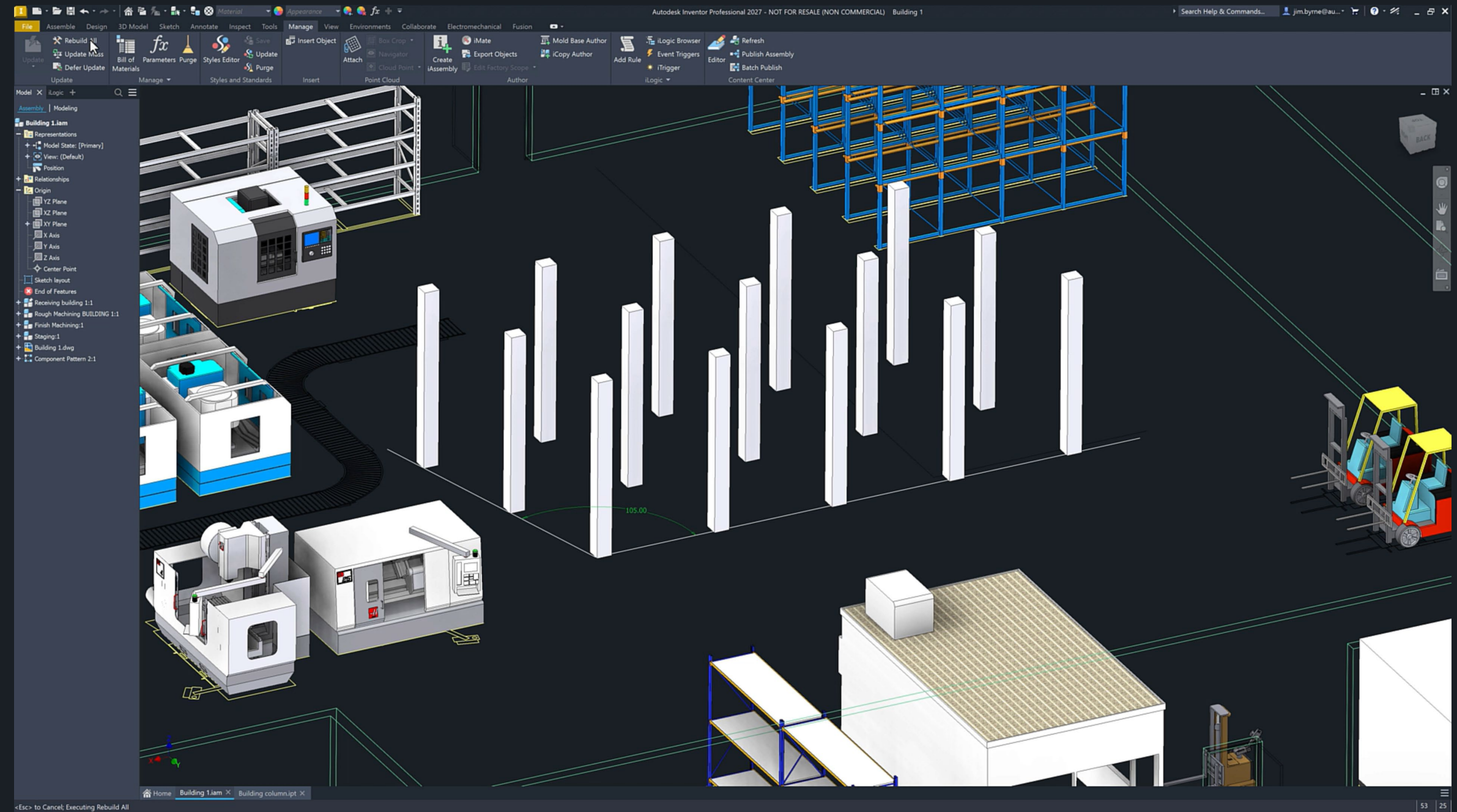
Content Center  
Publish assemblies

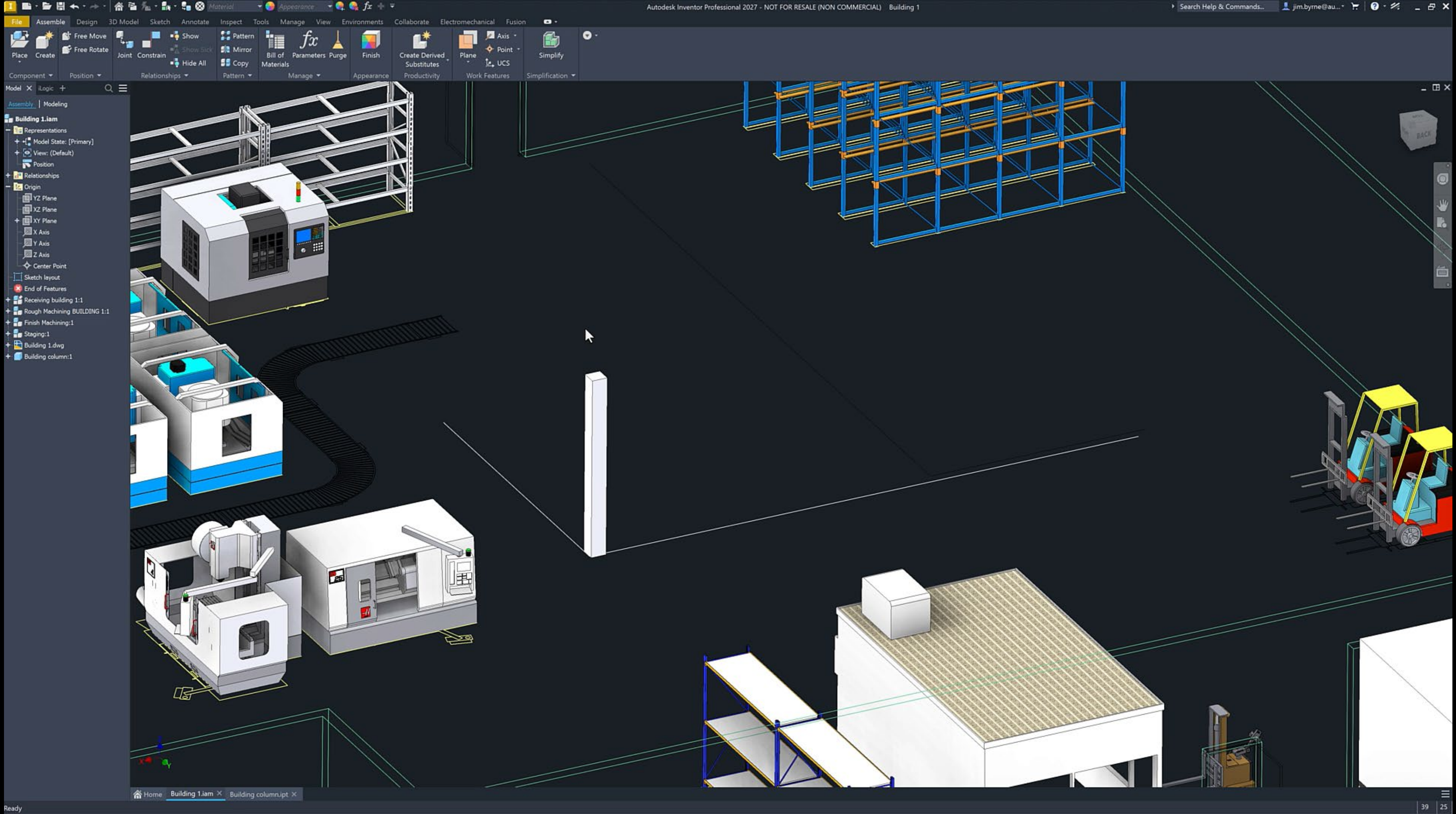


# Component pattern

Sketch lines for  
pattern direction

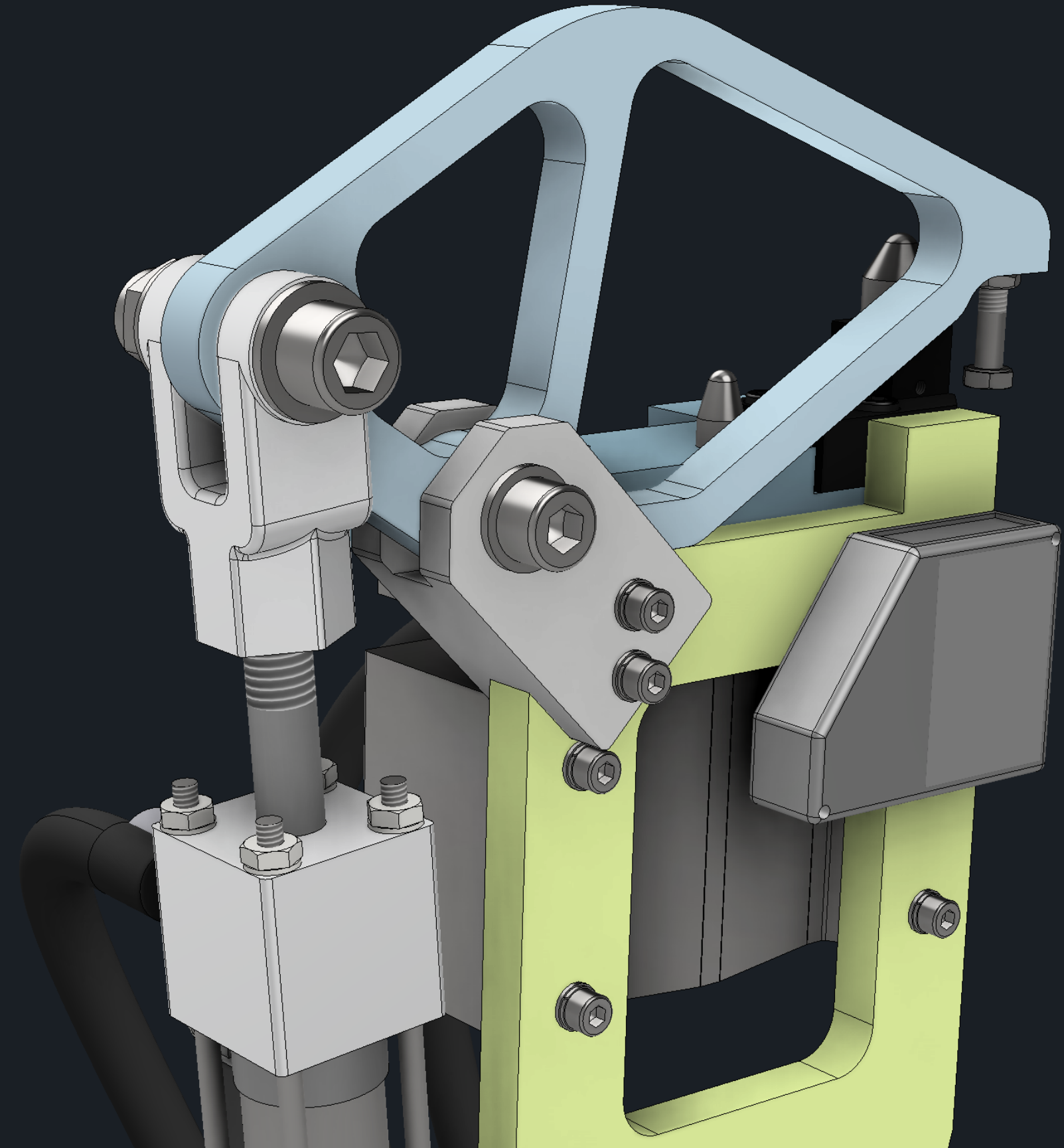
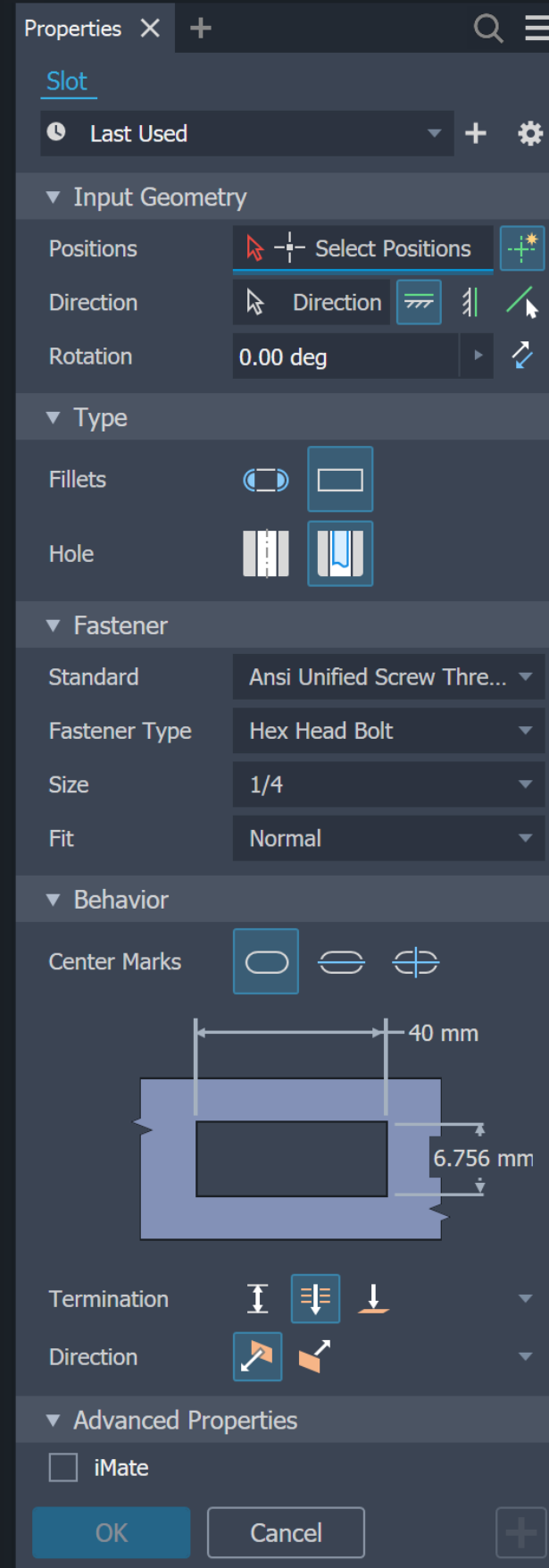
- Rectangular
- Circular





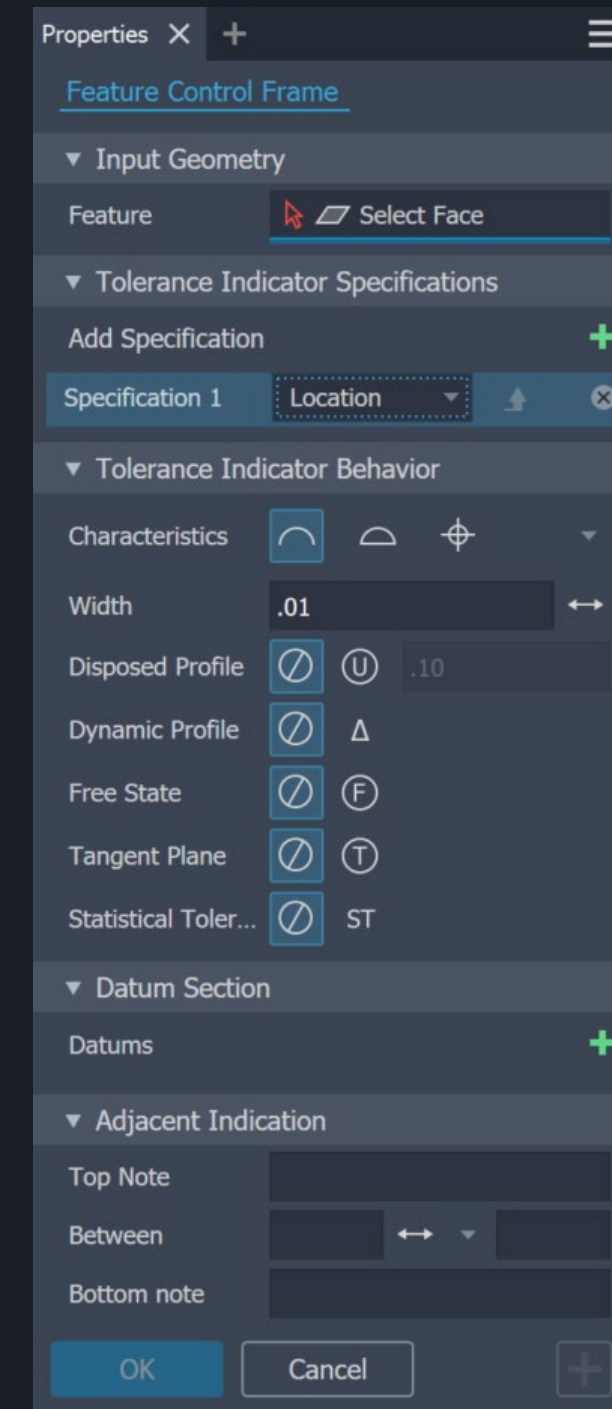
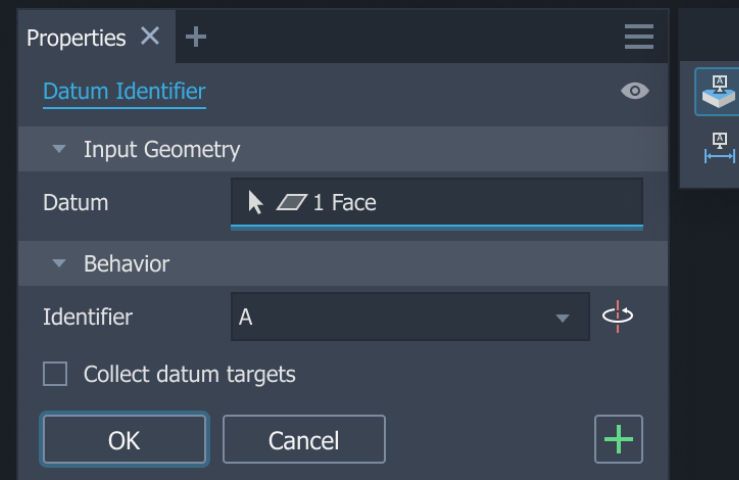
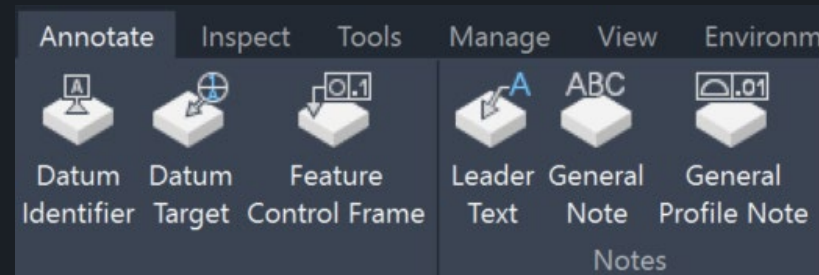
# Assembly modeling

## Assembly slot feature

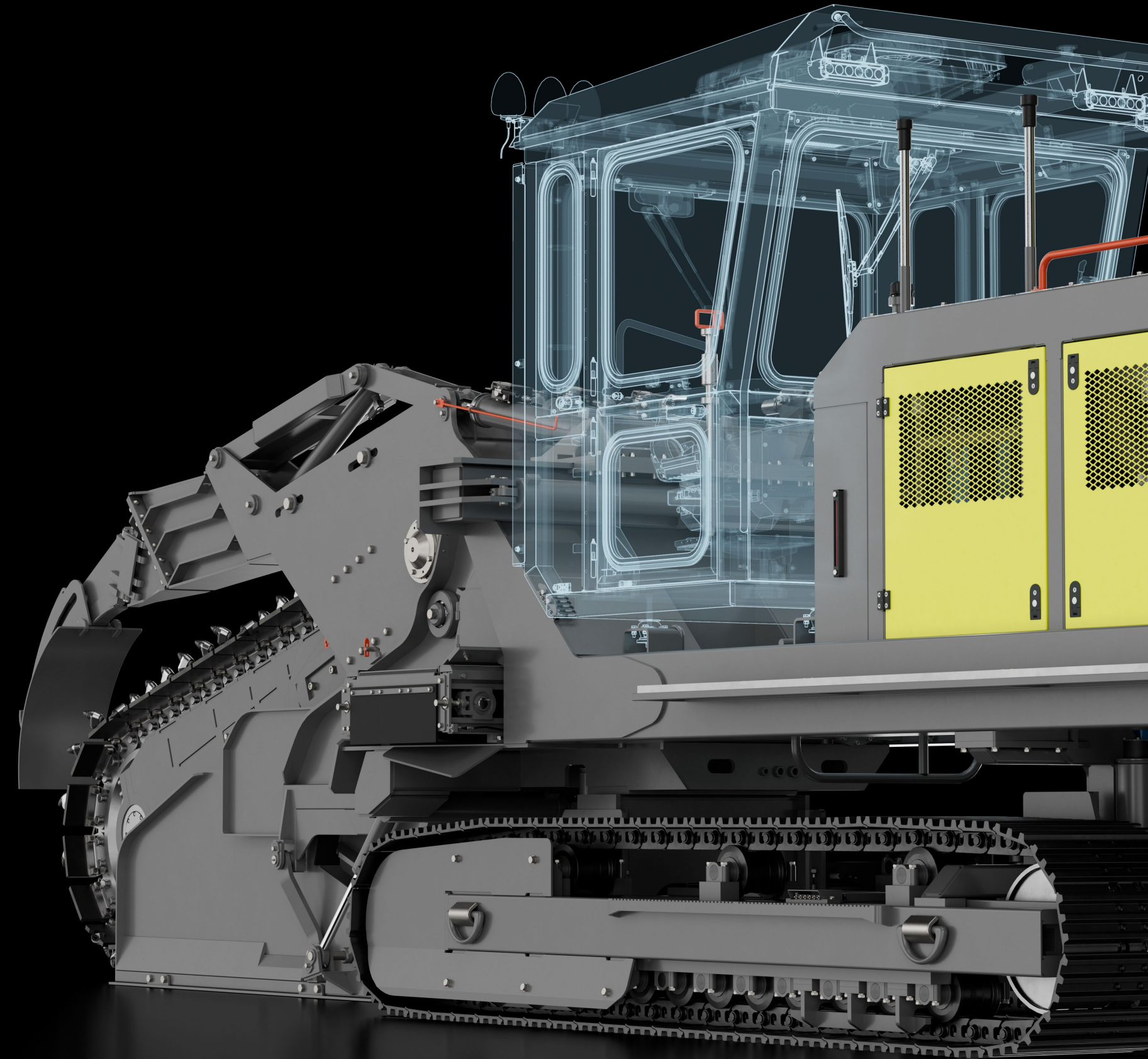


# Assembly 3DA / MBD support

- Fully Annotate Weldments & Assemblies
- More flexible than Tolerance Features
- Modern UI



# Drawings

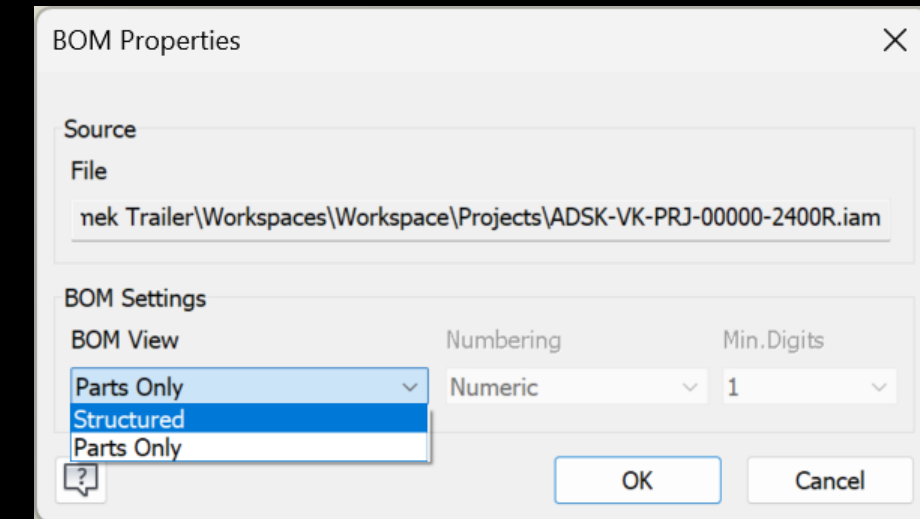
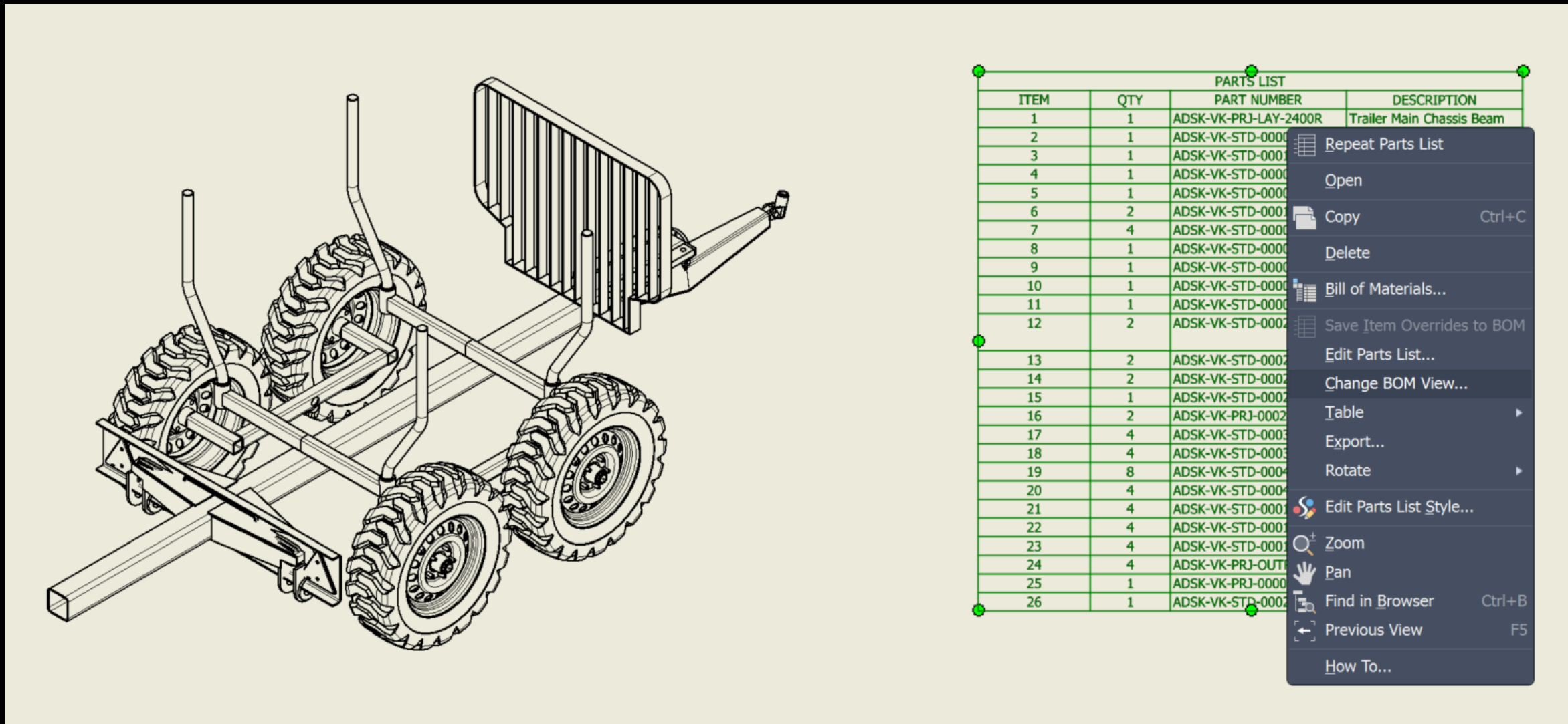


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Image courtesy of Mastenbroek

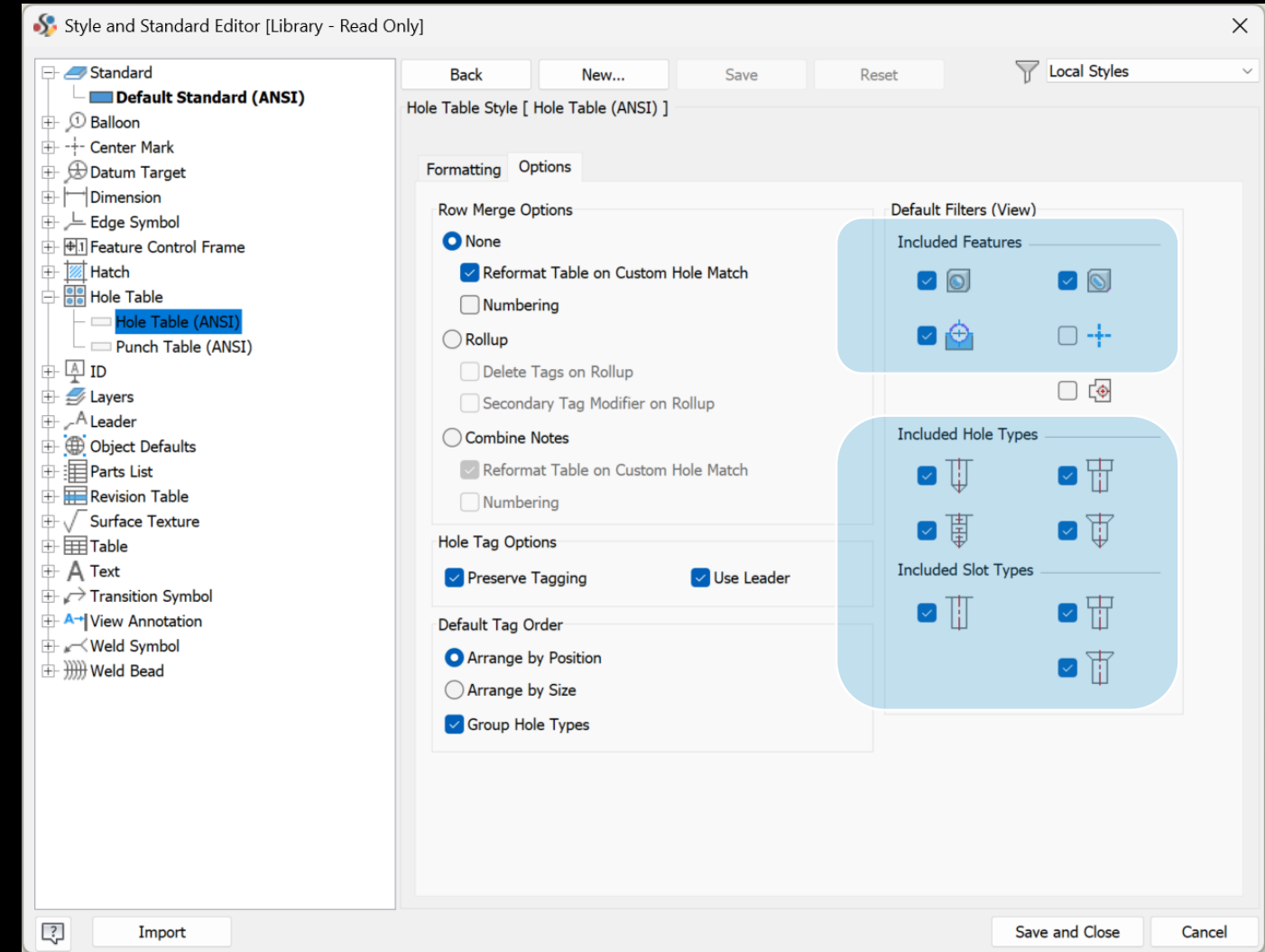
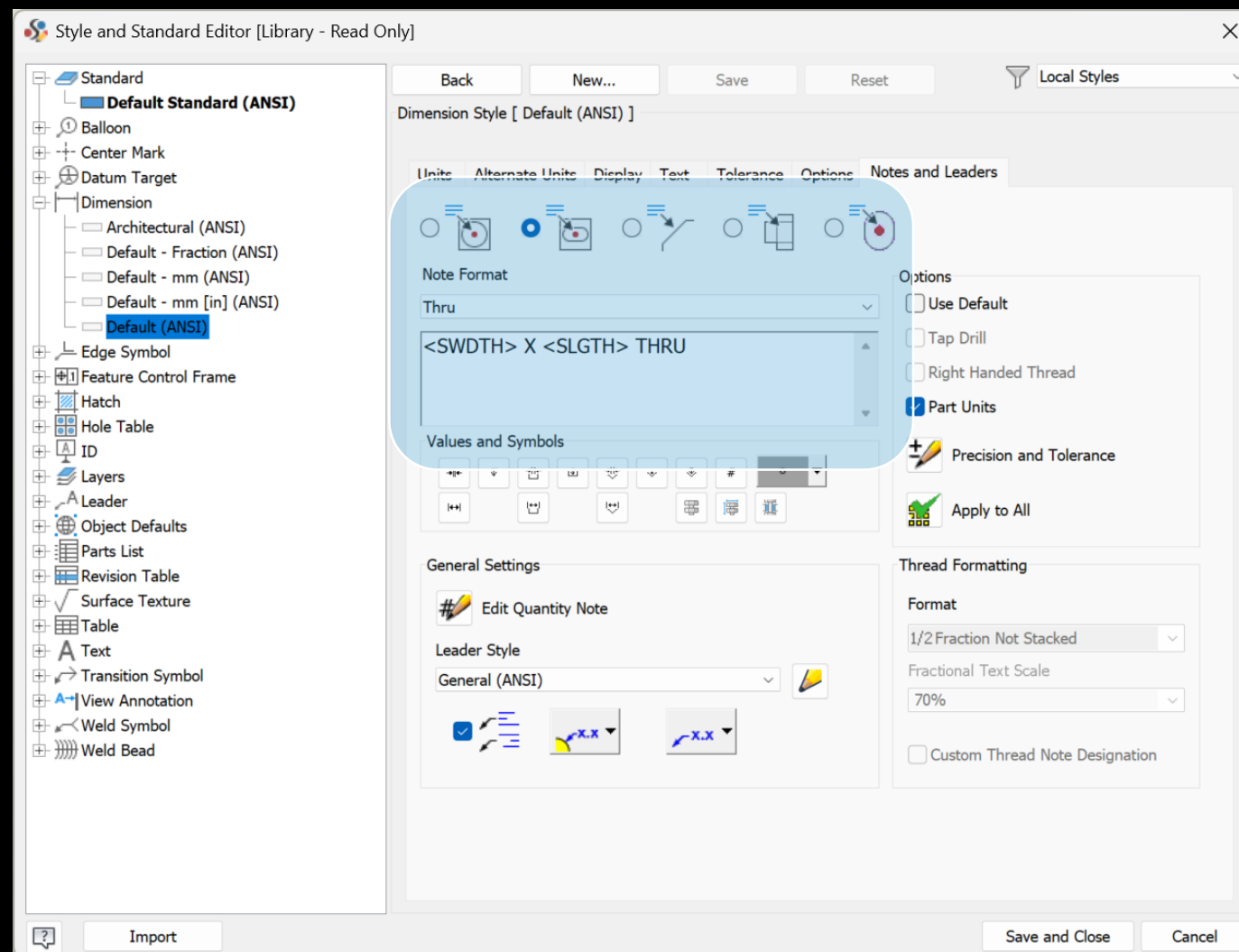
# Bill of material

## Change BOM view



# Drawing style

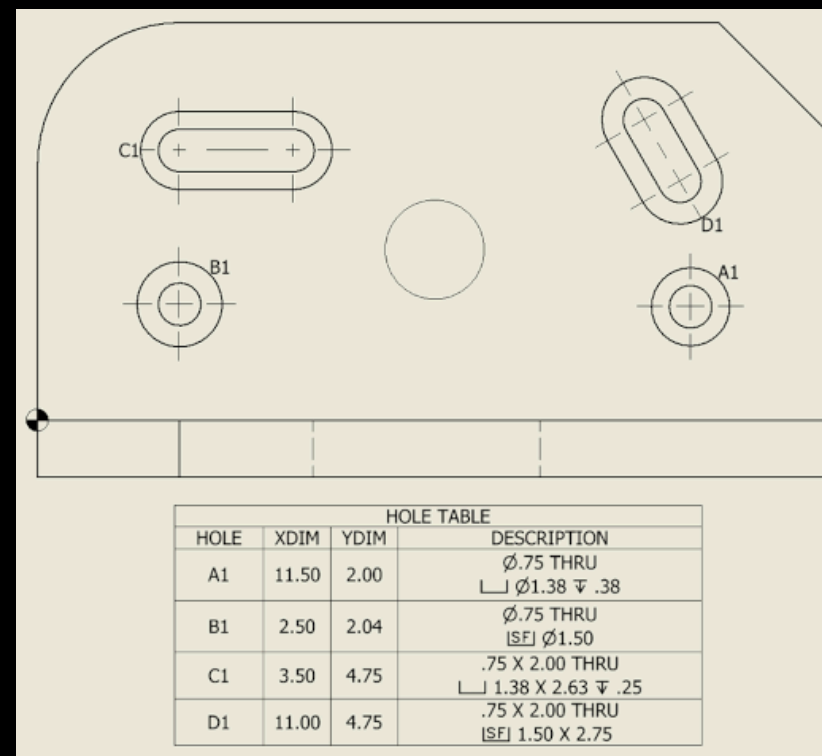
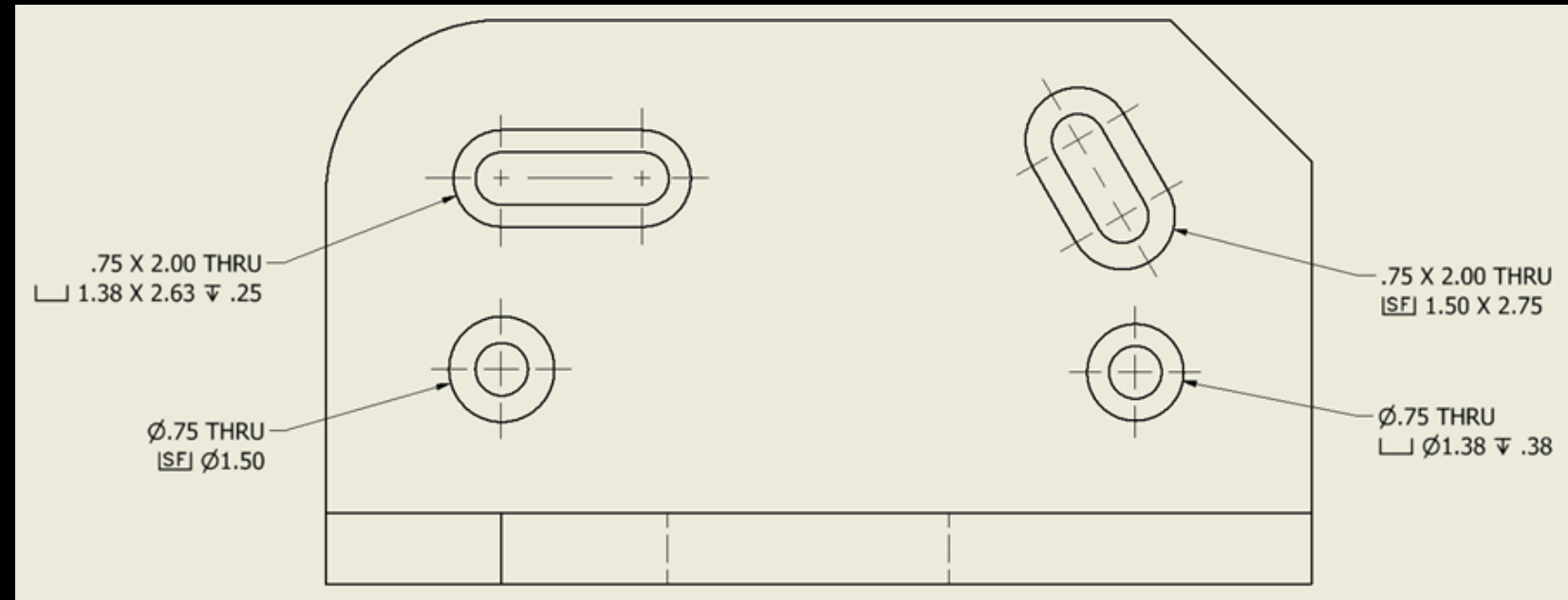
- Slot note dimension style
- Hole table style
- Center mark style



# Drawing annotations

## Annotations for slots

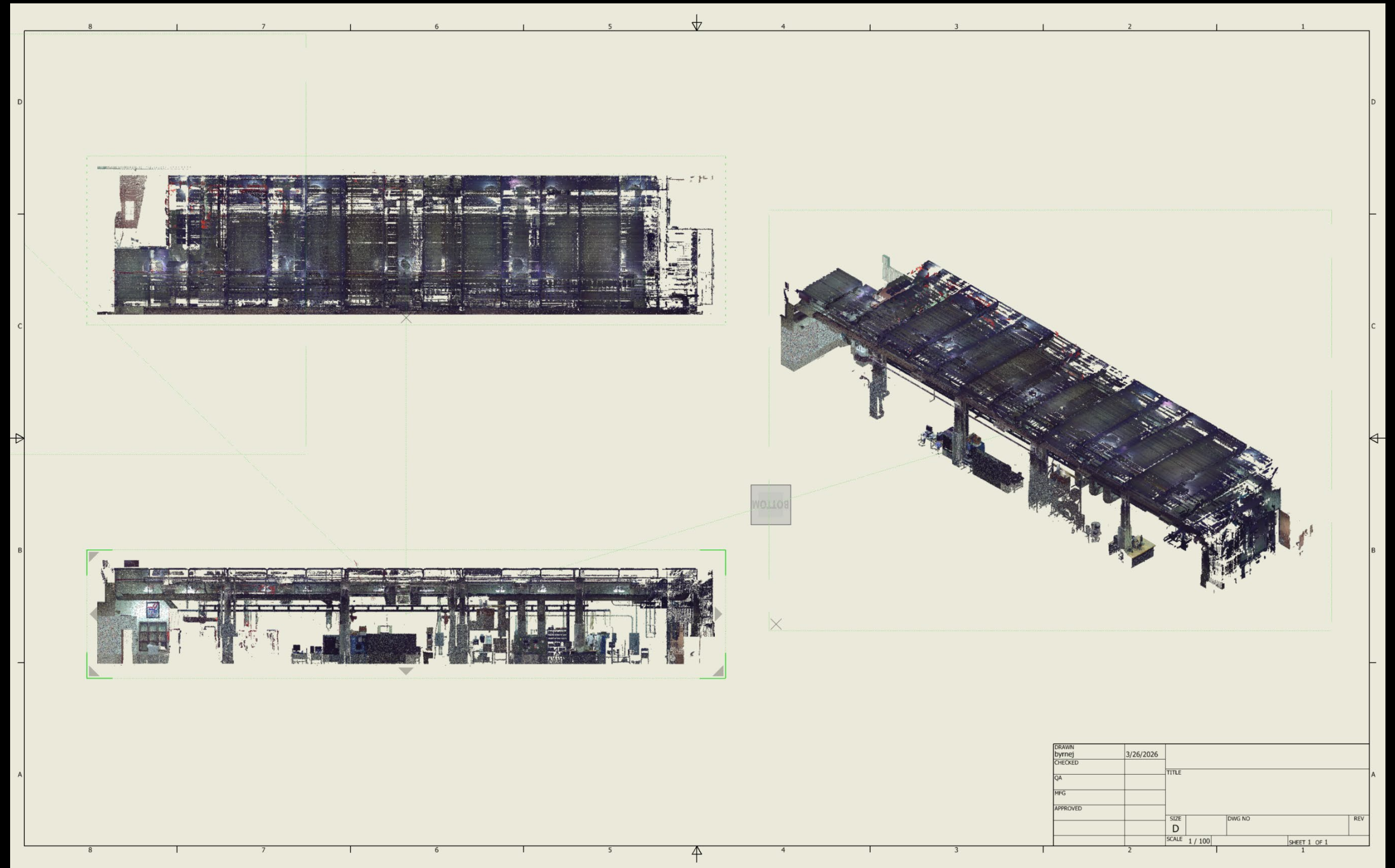
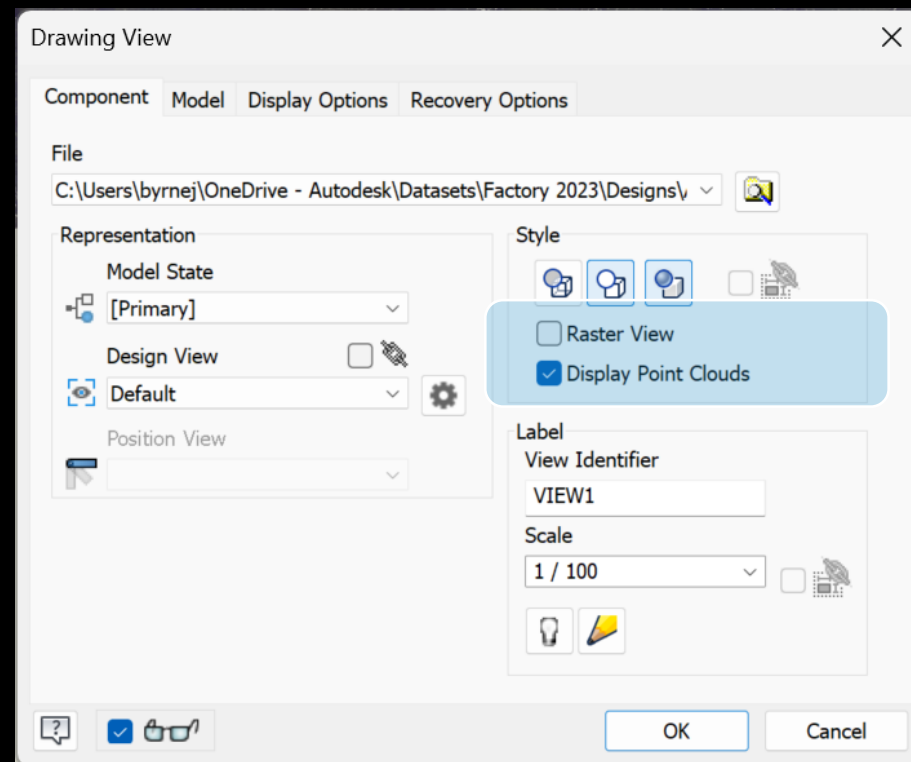
- Slot note
- Slot in hole table
- Center marks and centerlines
- Automated center lines



HOLE TABLE			
HOLE	XDIM	YDIM	DESCRIPTION
A1	11.50	2.00	$\phi .75$ THRU $\lfloor \phi 1.38 \nabla .38$
B1	2.50	2.04	$\phi .75$ THRU $\lfloor \phi 1.50$
C1	3.50	4.75	$.75 \times 2.00$ THRU $\lfloor 1.38 \times 2.63 \nabla .25$
D1	11.00	4.75	$.75 \times 2.00$ THRU $\lfloor 1.50 \times 2.75$

# Drawing views

- Point clouds
  - Shaded views



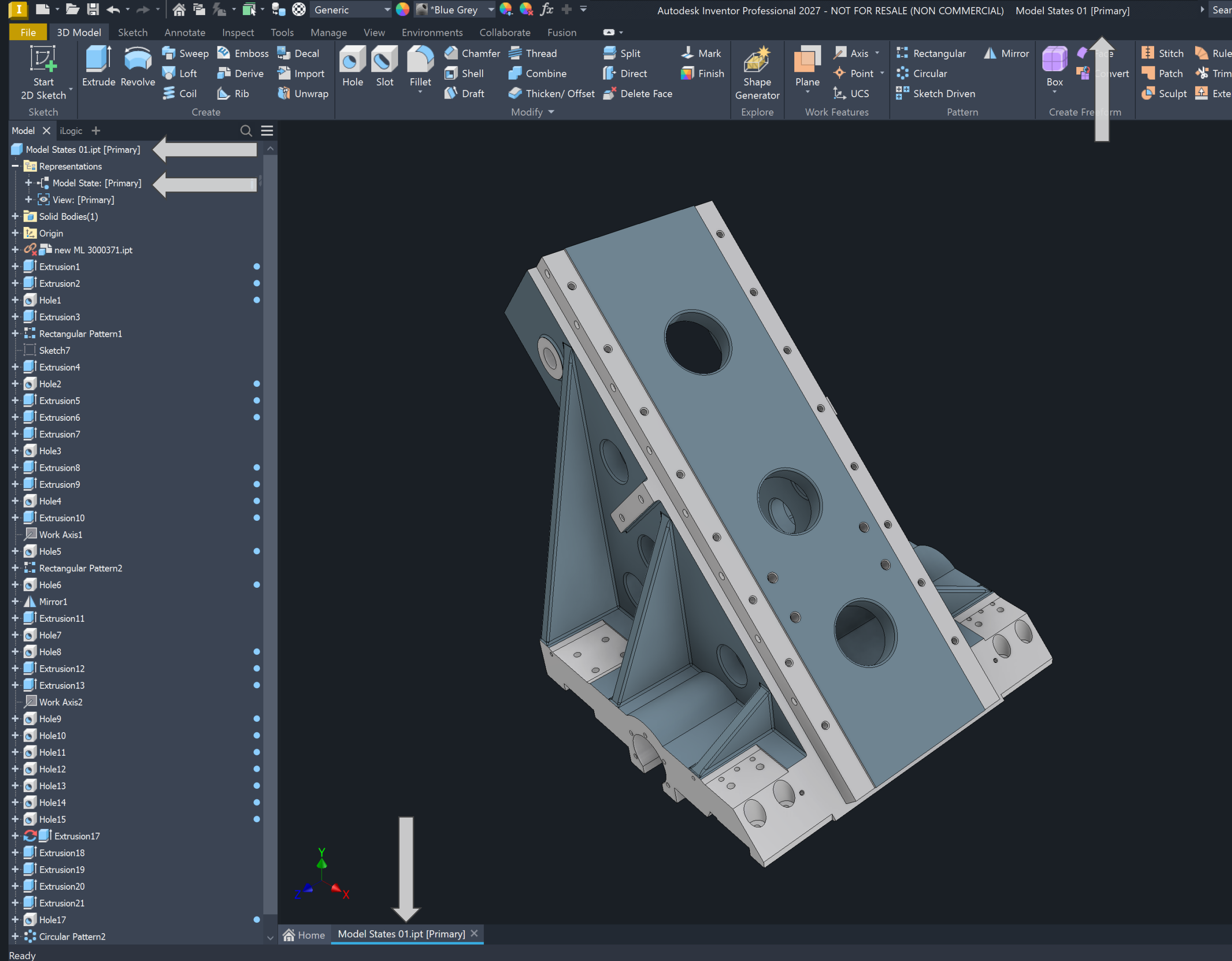
# Model States



 AUTODESK

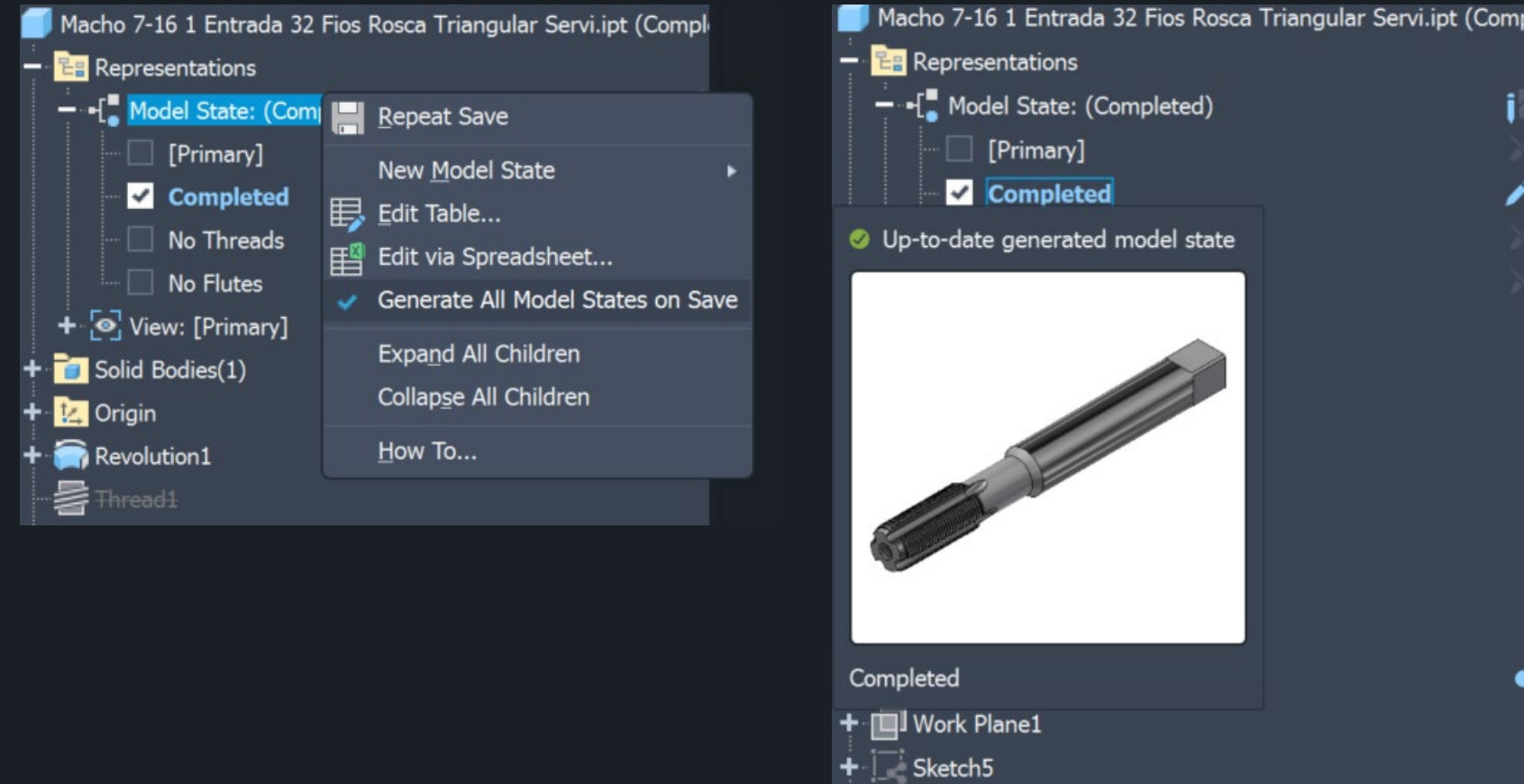
# Model States

- Inventor 2026
- Application options
  - Active Model State



# Model States

- Generate all Model States on save
- Preview image for each Model State
  - BOM
  - Browser
- Table compare
  - Inventor 2026.1



Model State Table

Parameters Properties Suppression iFeatures iMates Threads

Macho 7-16 1 Entrada 32 Fios Rosca Triangular Servi.ipt

Revolution1

- x= d0 [1 in]
- x= d2 [150 deg]
- x= d3 [d2]
- x= d4 [7/16"]
- x= d5 [12 mm]
- x= d6 [3/8"]
- x= d7 [150 mm]
- x= d9 [15 mm]

Thread1

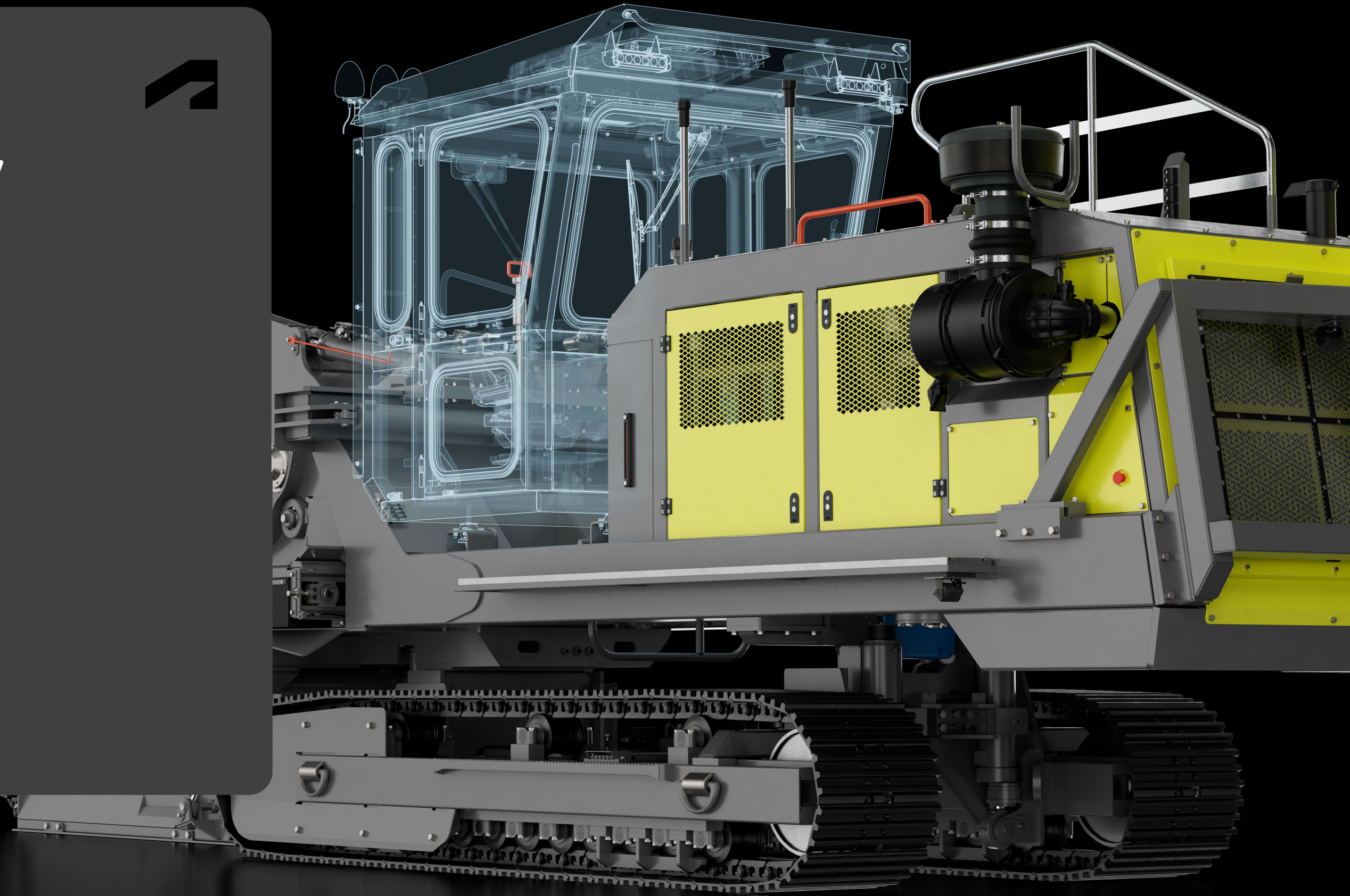
Revolution2

	Member	Coil1	Revolution4	Extrusion1	Revolution5	Extrusion2	Extrusion3	Hole1	Hole2	Revolution6	d7
1	[Primary]	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	150 mm
2	Long complete	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	150
3	Short complete	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	100 mm
4	No threads	Suppress	Compute	Compute	Compute	Compute	Compute	Compute	Compute	Compute	100 mm
5	No grooves	Suppress	Suppress	Suppress	Suppress	Compute	Compute	Compute	Compute	Compute	100 mm
6	No holes	Suppress	Suppress	Suppress	Suppress	Suppress	Suppress	Suppress	Suppress	Compute	100 mm

# Interoperability



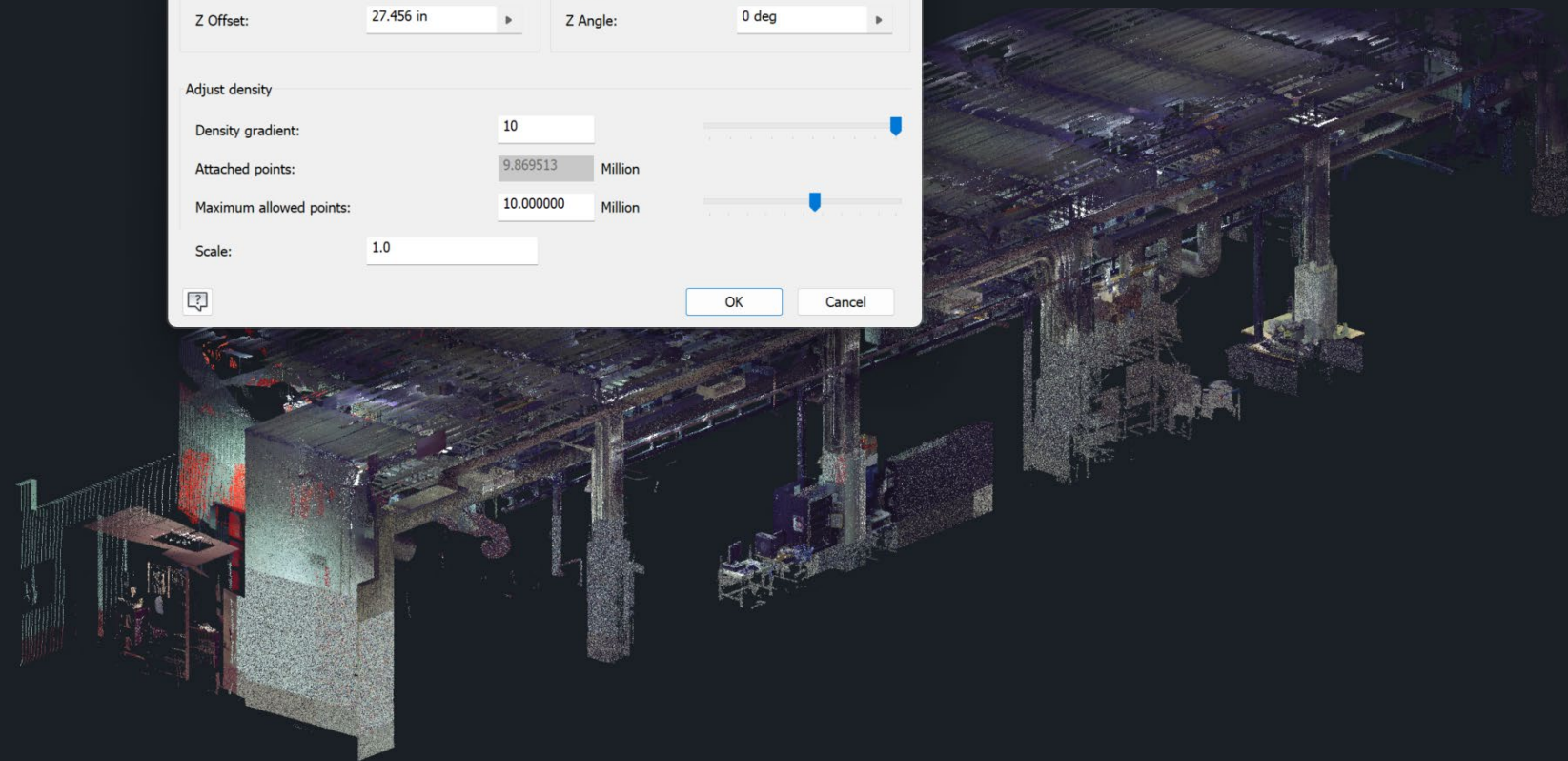
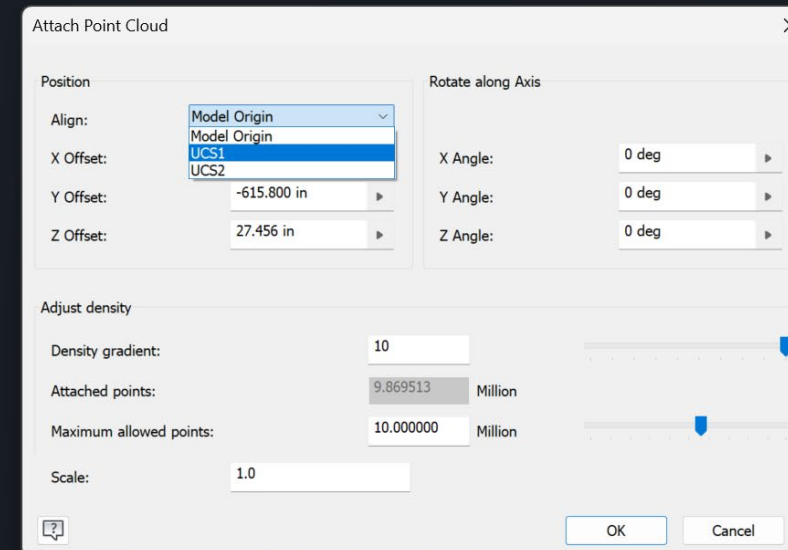
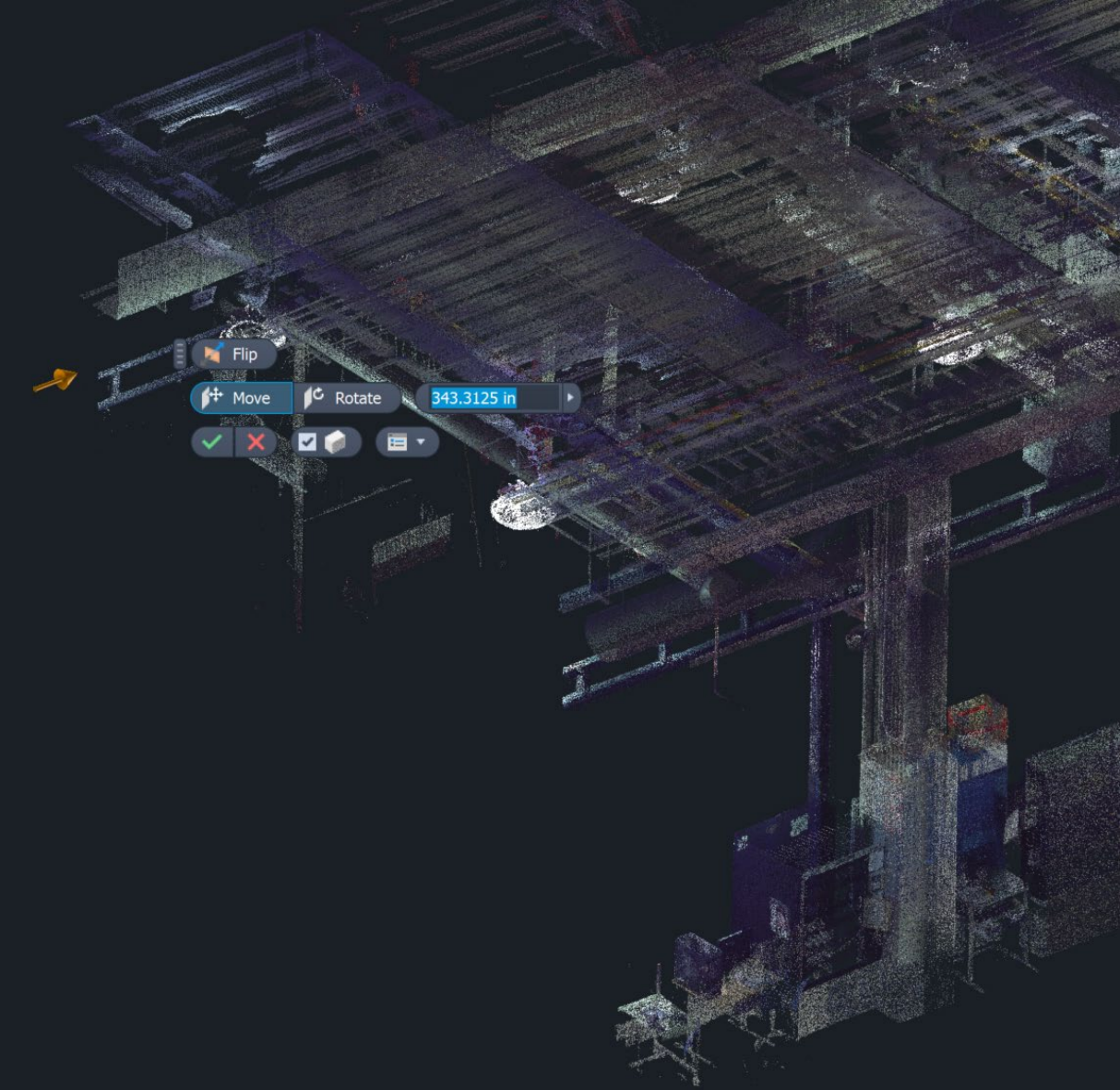
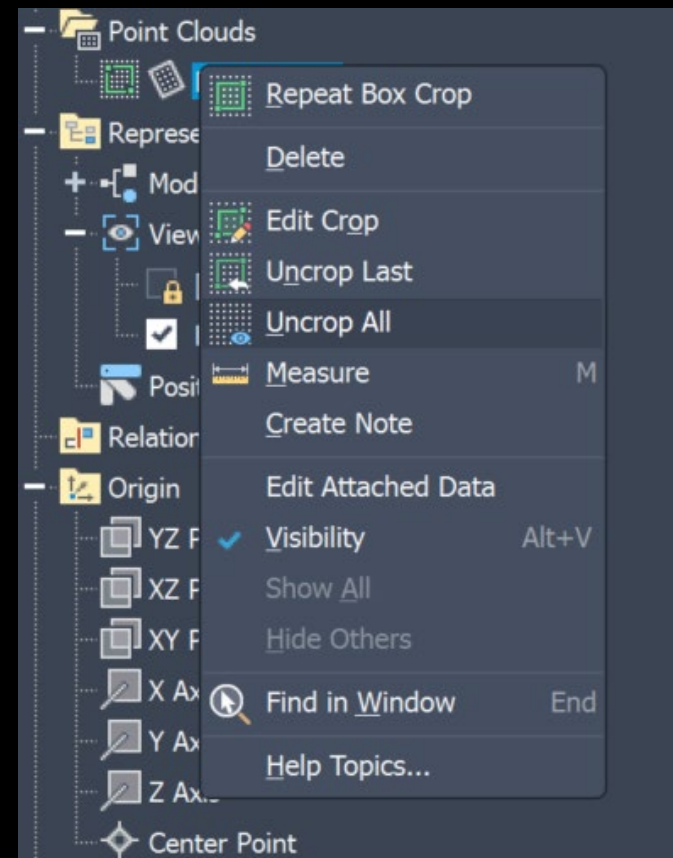
 AUTODESK



# Point cloud

Click to add subtitle

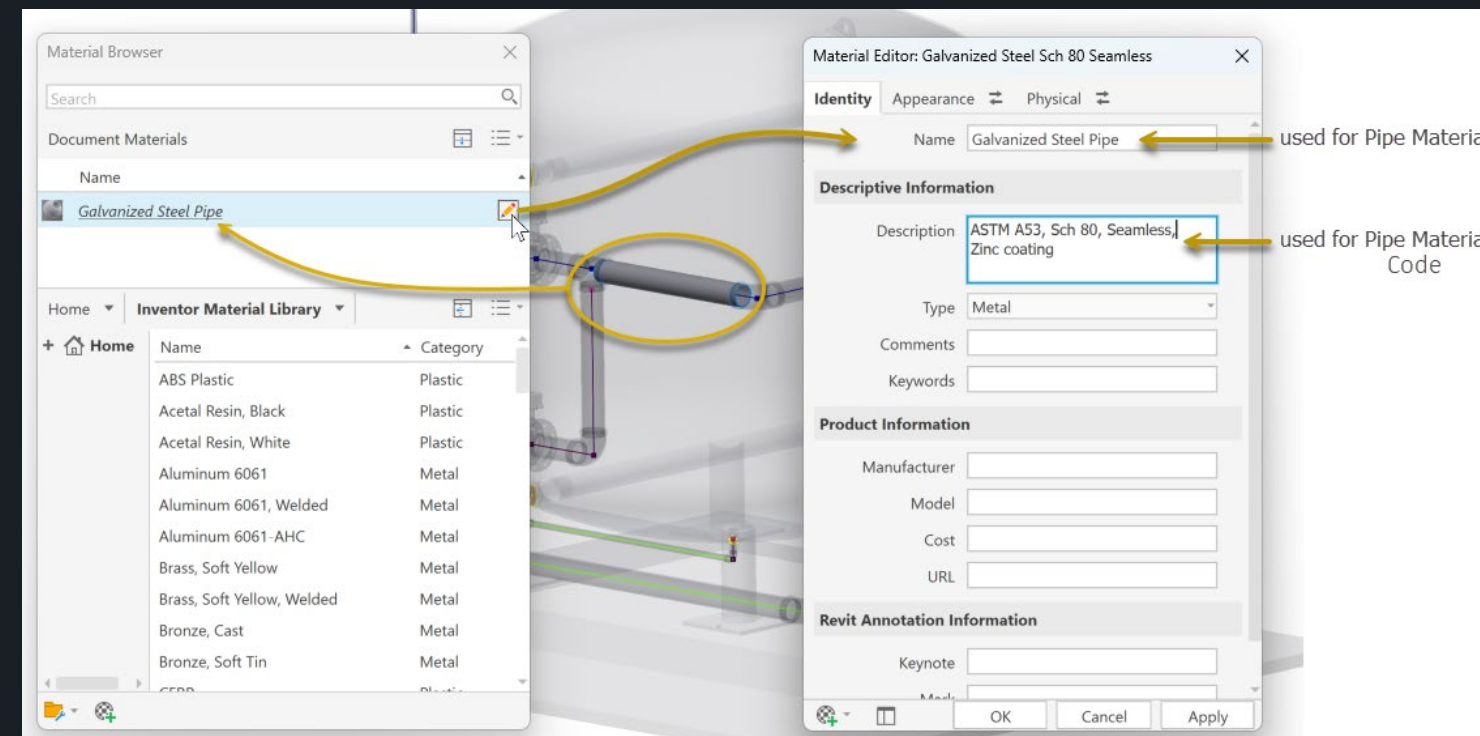
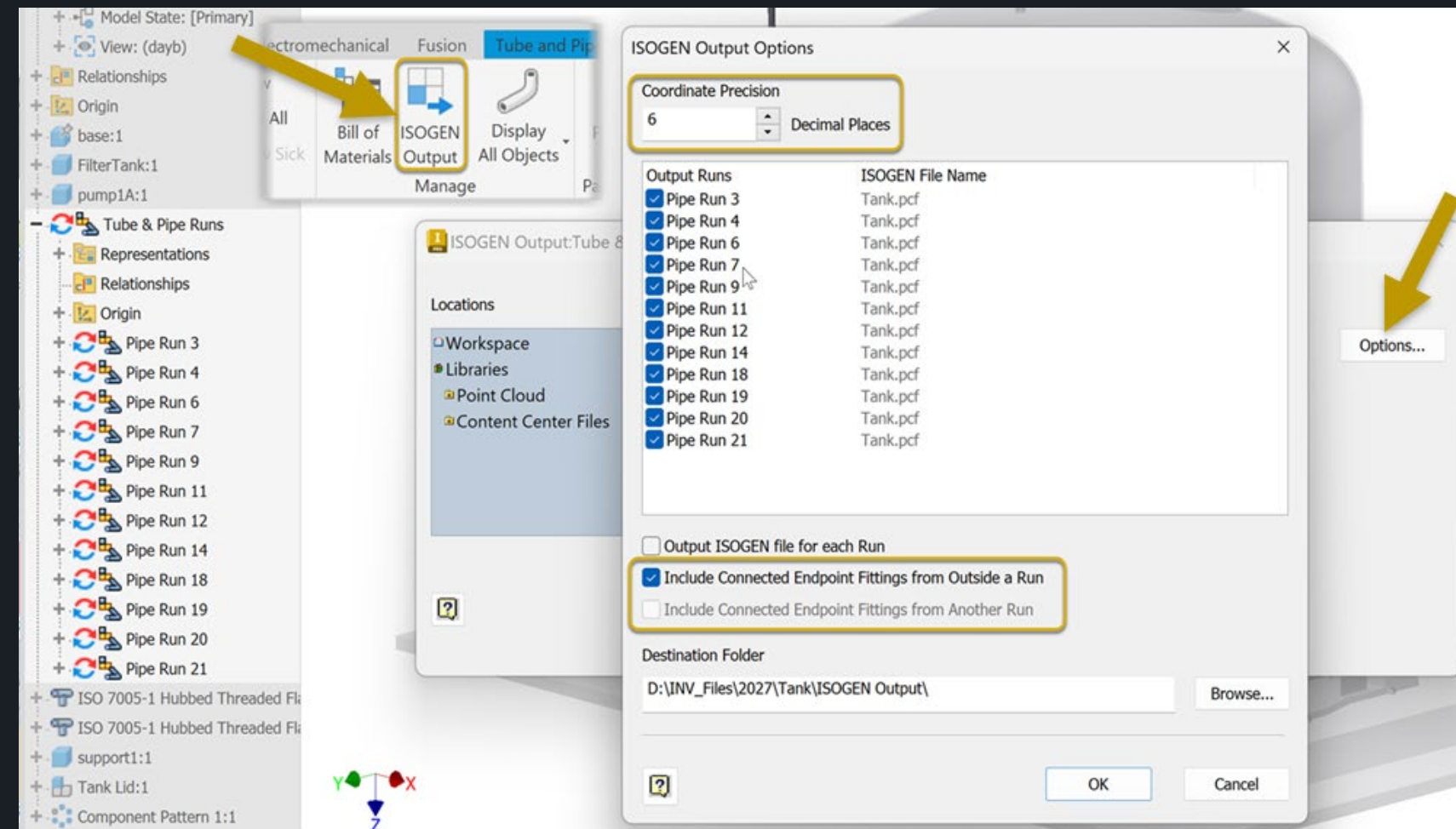
- Section views
- Improved cropping
- UCS-based positioning
- Browser visibility
- Drawing view support



# Tube and Pipe

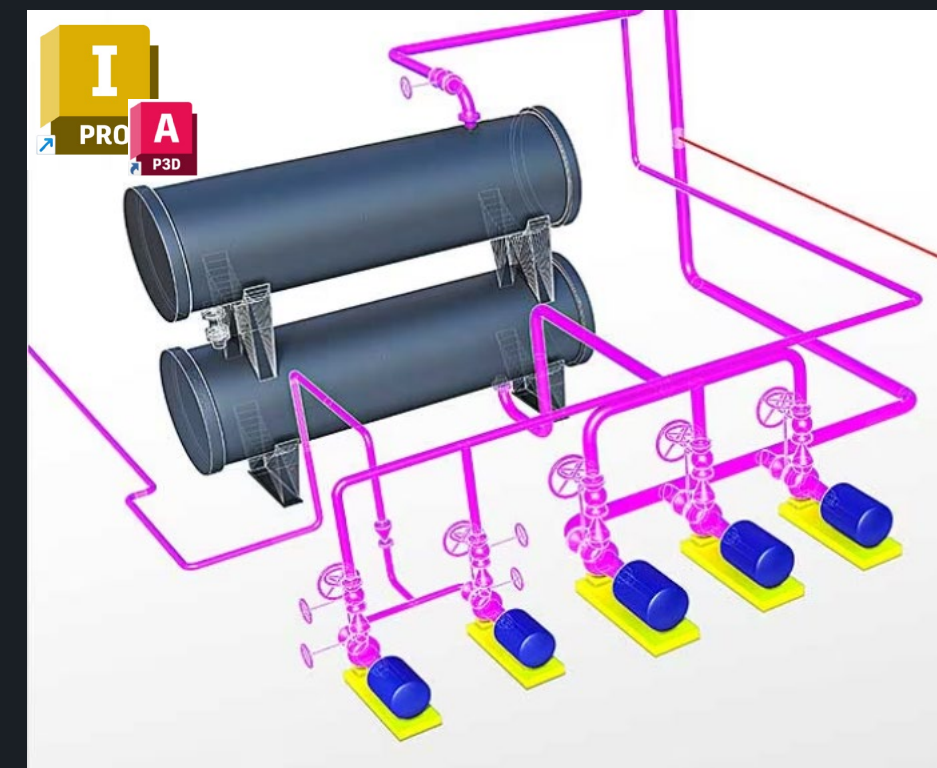
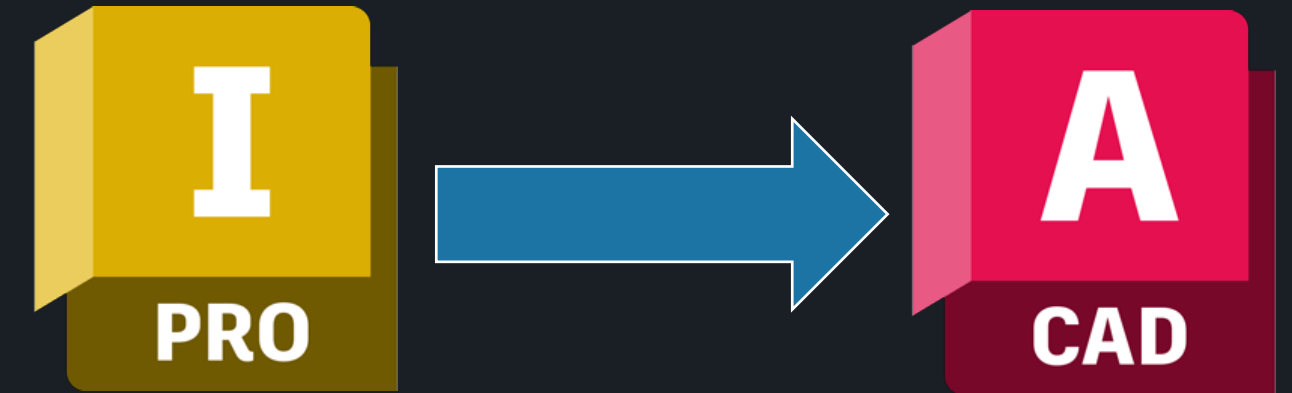
## PCF export

- Control coordinate precision up to six decimal places
- Material property
- Enhanced endpoint identification



# AutoCAD Plant3D

- Tube and Pipe -> ISOGEN Drawing
- Export missing material data from Inventor Material
  - Inventor Material Name -> PCF Material Name
  - Description -> PCF Material Code
- Fixed export issue dealing with multiple connected runs, including connected fittings in other runs
- Include pipe run end connection name in export
- Export end codes for fitting based on the T&P fitting authoring setting
  - Center-to-center dimensioning type (BS, FL)
- Enhanced decimal place control for coordinate precision





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