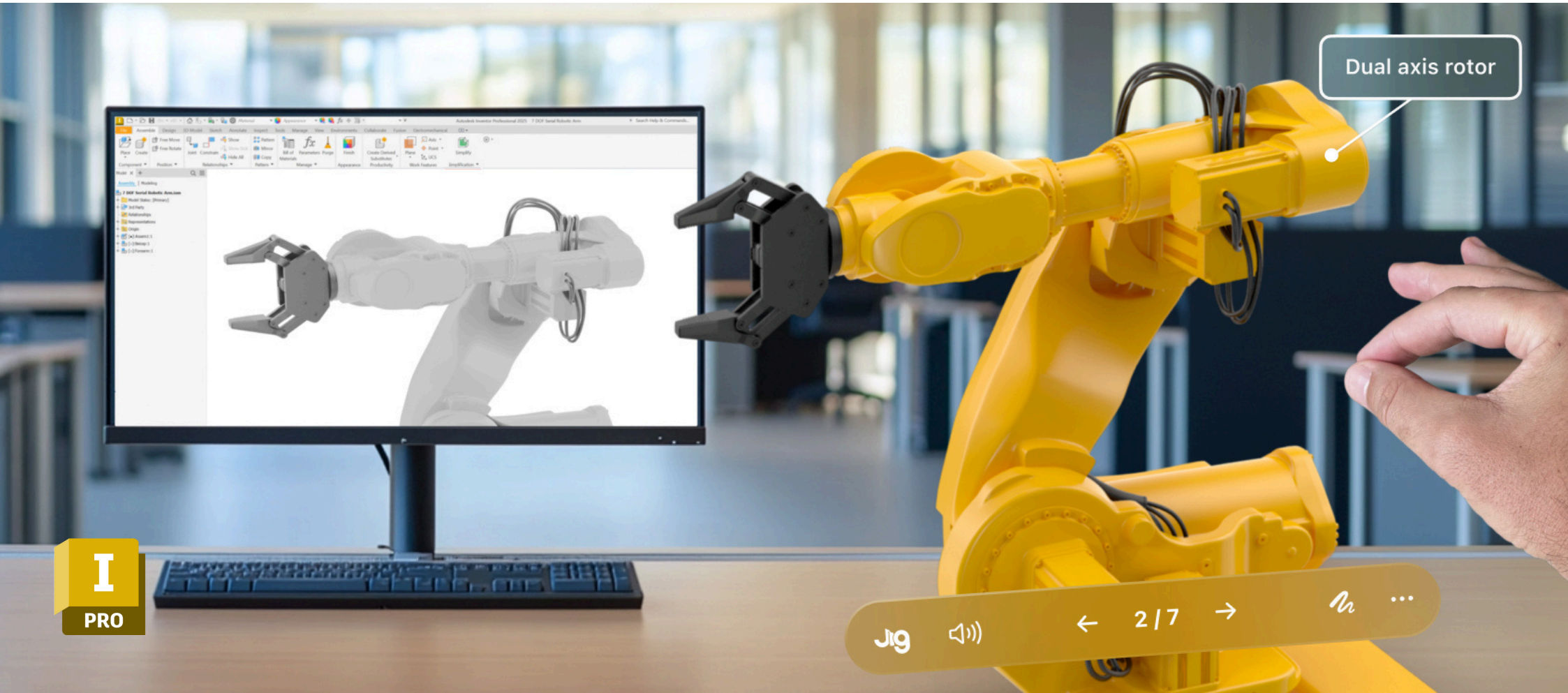
Guide to exporting Inventor files to JigSpace

Current version: 13/12/2024



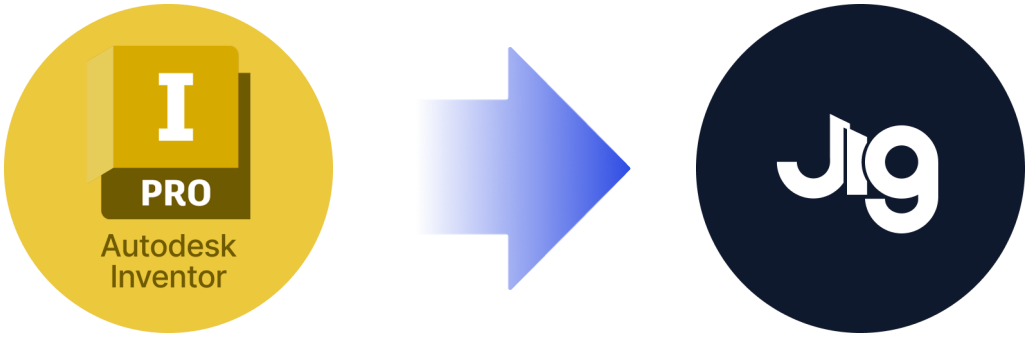
Contents

1 Introduction

Who is it for?

Autodesk Inventor is a powerful CAD software that provides professional-grade tools for 3D modeling and design.

This guide will demonstrate how to organize 3D models into logical assemblies and simplify complex CAD models, reducing file sizes to ensure seamless performance on the JigSpace platform across all devices.



> ***"It's highly valuable in my line of work to be able to quickly and easily visualise - in 1:1 scale - CAD models that I'm working on.***
>
> ***These tools are essential to making that happen."***

Kaitlyn Lee
Industrial Designer at DTCo

2 Export formats

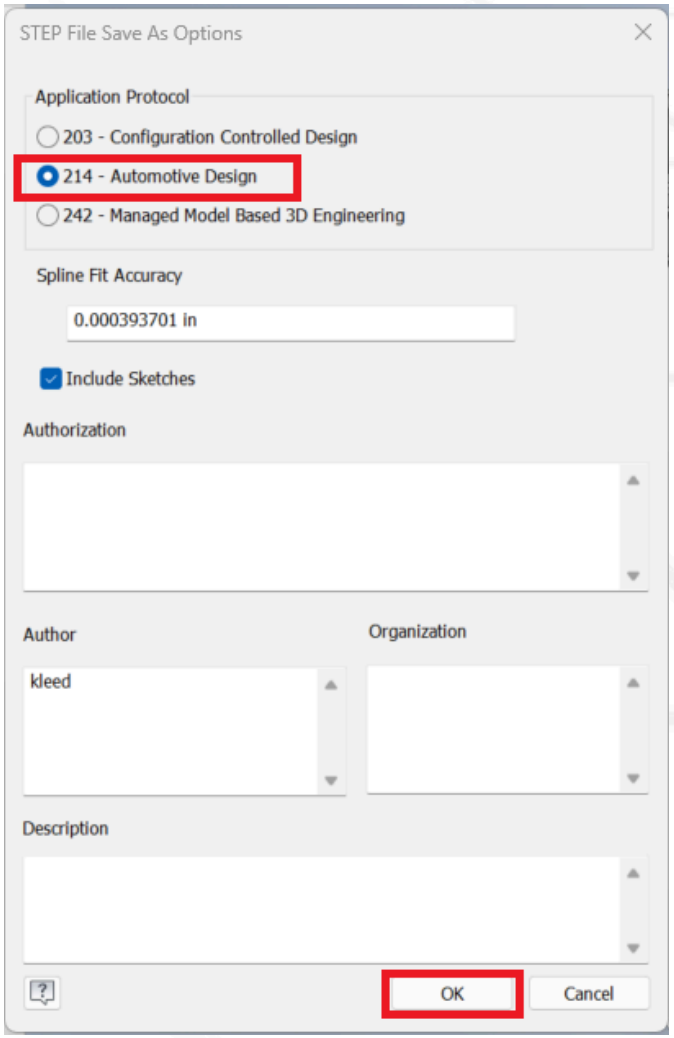
STEP files

For best results, we recommend you export all Autodesk Inventor assemblies as a STEP file for importing into Jig. These typically have the extension .step or .stp

STEP files have a 3 main configurations. We recommend exporting to type **STEP AP214**

STEP AP203 generally creates the smallest number of objects in your Jig (and best performance) however it excludes color properties contained in STEP AP214 format. If these are not required, then you should use AP203.

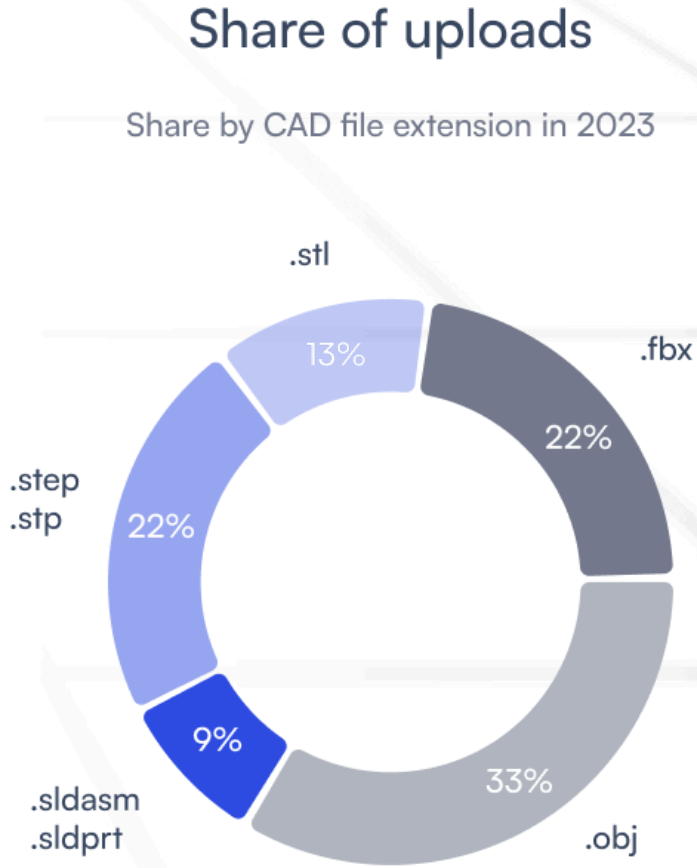
We do not recommend exporting to STEP AP242 which contains unnecessary content and data that will not be used in JigSpace will reduce the presentation performance.



2 CAD data imported

File formats accepted by JigSpace

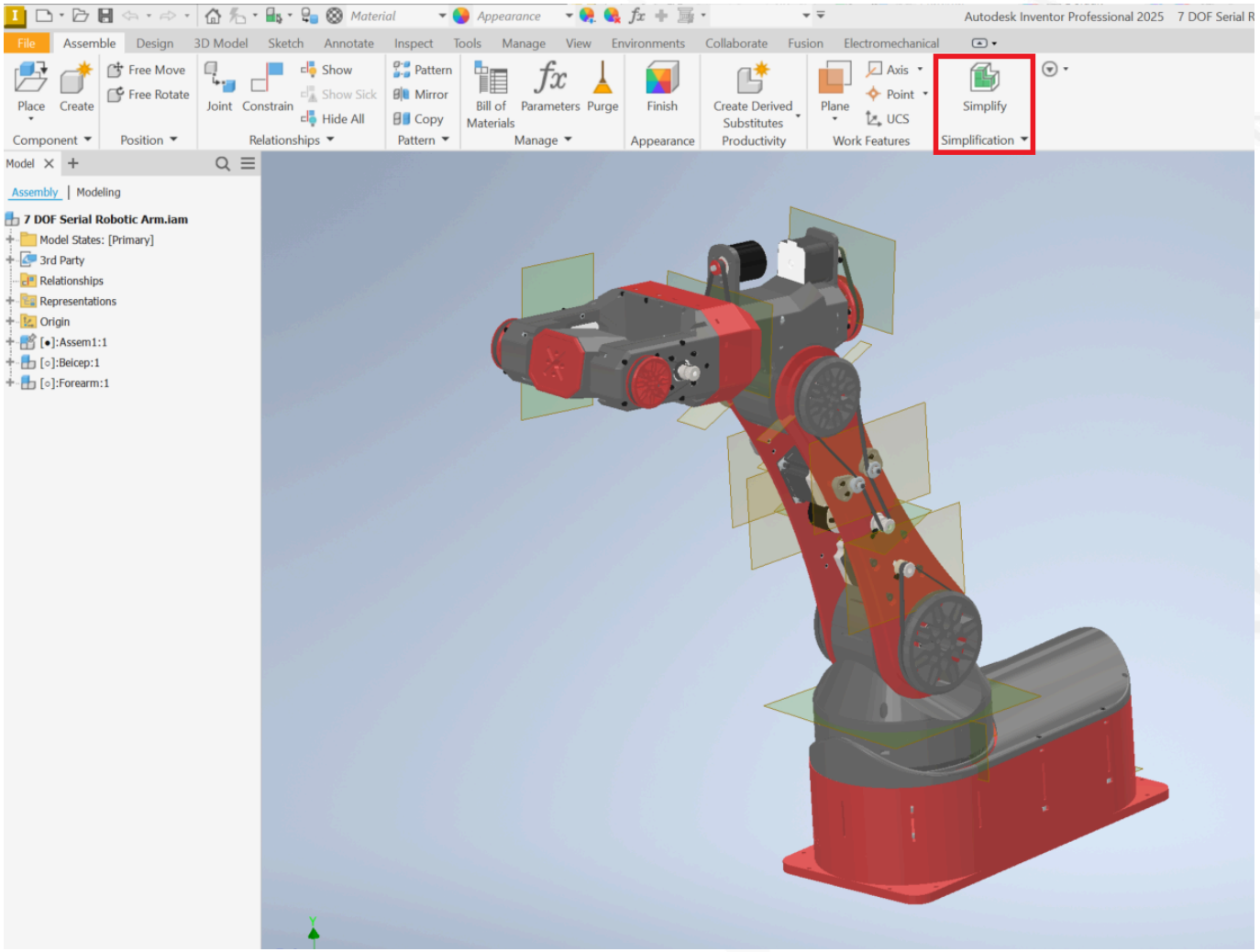
CAD file	3D mesh	Scale	Color	Labels
.fbx	✓	✓	✓	✓
.gltf / .glb	✓	✓	✓	✓
.obj	✓	✓		✓
Recommended for Inventor and JigSpace				
.step / .stp	✓	✓	✓	✓
.stl	✓	✓		
.sldasm / .sldprt	✓	✓	✓	✓
.usdz	✓	✓	✓	✓



3 Export assemblies

This tool can only be used with assemblies, please see “Export Single parts” page 15.

1. In Inventor, navigate to the Assemble tab > Simplify tool

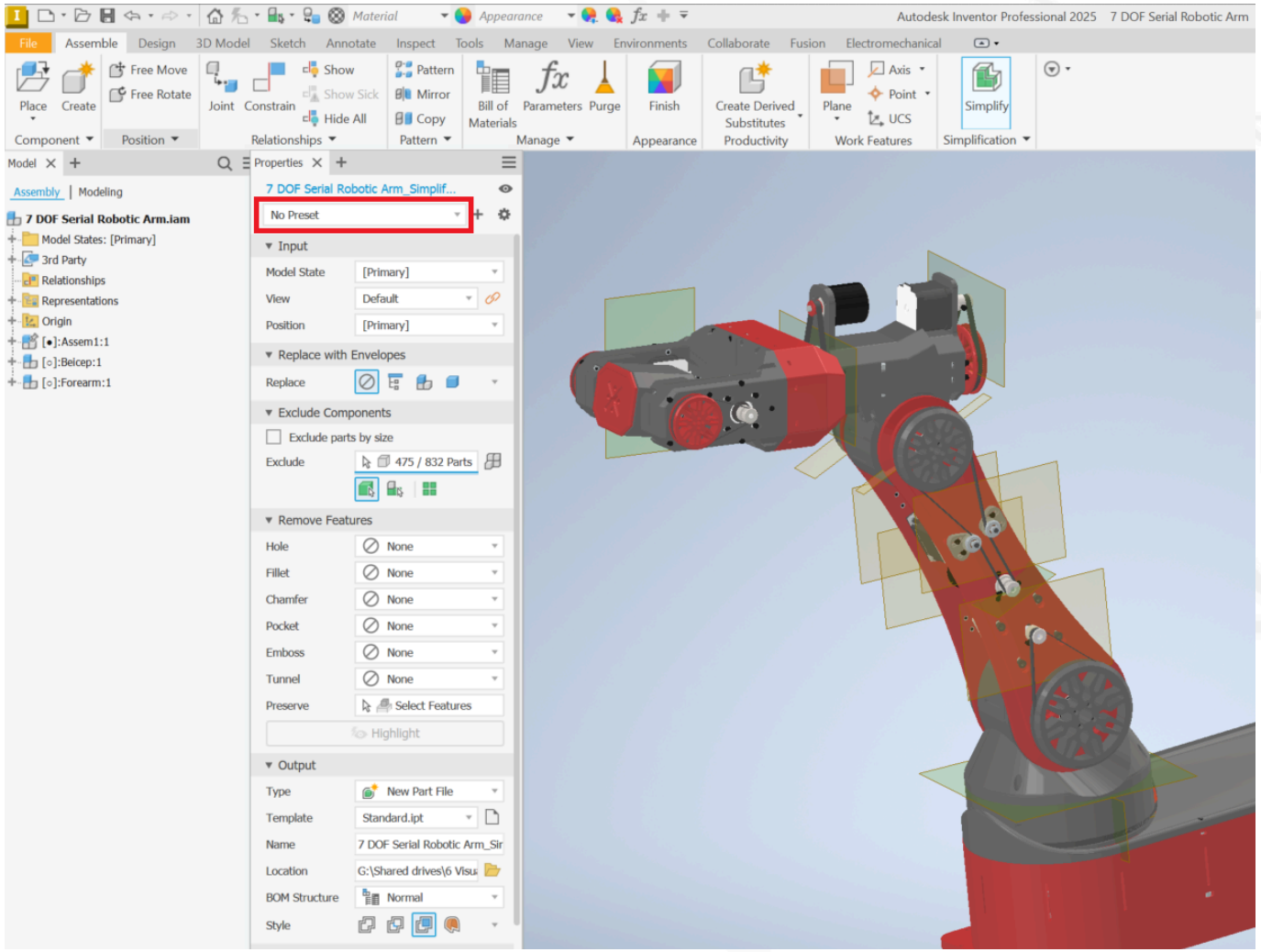


3 Export assemblies

This tool can only be used with assemblies, please see Process #2 for single parts.

1. In Inventor, navigate to the Assemble tab > Simplify tool

2. By default, the last preset is set. Change this to ‘no preset’ to begin with



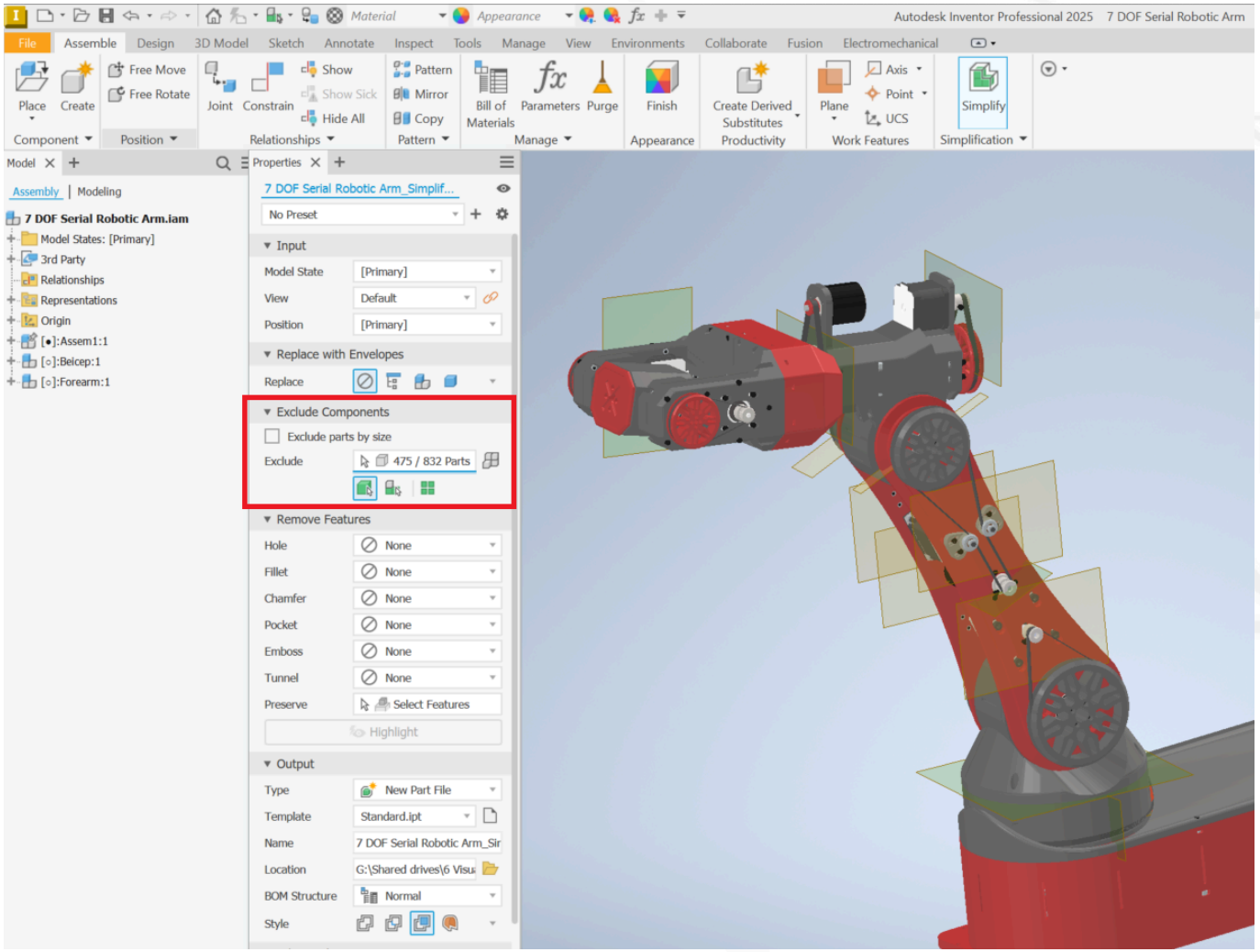
3 Export assemblies

This tool can only be used with assemblies, please see Process #2 for single parts.

1. In Inventor, navigate to the Assemble tab > Simplify tool

2. By default, the last preset is set. Change this to ‘no preset’ to begin with

3. Now click the tickbox on for ‘exclude parts by size’



3 Export assemblies

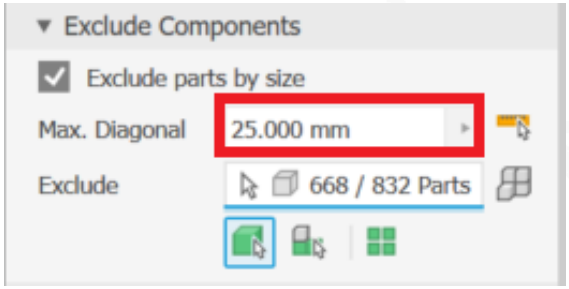
This tool can only be used with assemblies, please see Process #2 for single parts.

1. In Inventor, navigate to the Assemble tab > Simplify tool

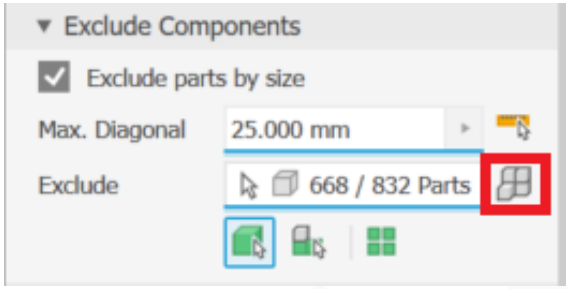
2. By default, the last preset is set. Change this to ‘no preset’ to begin with

3. Now click the tickbox on for ‘exclude parts by size’

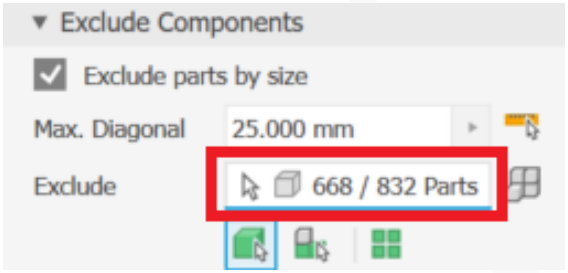
4. Adjust the max. diagonal size. Depending on your model the size to exclude will vary. A preview of how many parts will be excluded is displayed below.



Tip: Try out different values to determine which will suit your needs more.



Tip: click this button to visually see which parts are being excluded

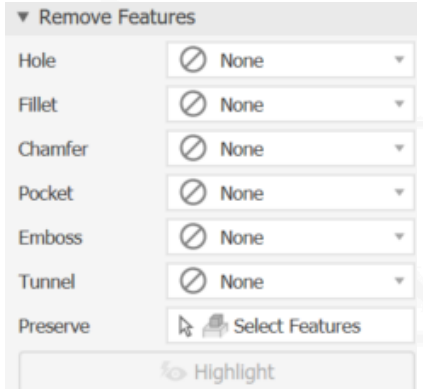


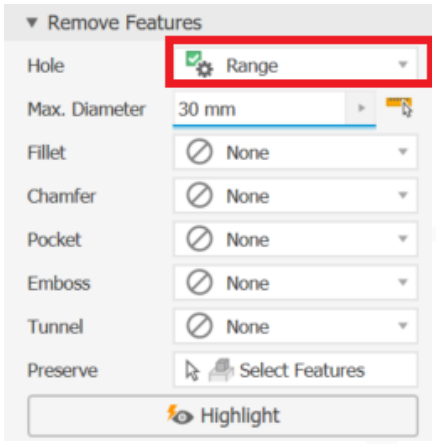
Tip: you can also see how many parts are being excluded here (this is updated when you change the ‘max. diagonal’ value):

3 Export assemblies

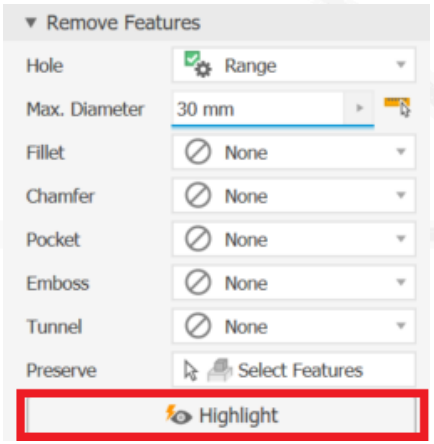
This tool can only be used with assemblies, please see Process #2 for single parts.

5. (Optional) if you would like to remove features as well as parts, you can do so by using the ‘remove features’ section





Select the feature to remove > in the dropdown menu > select range > specify the range



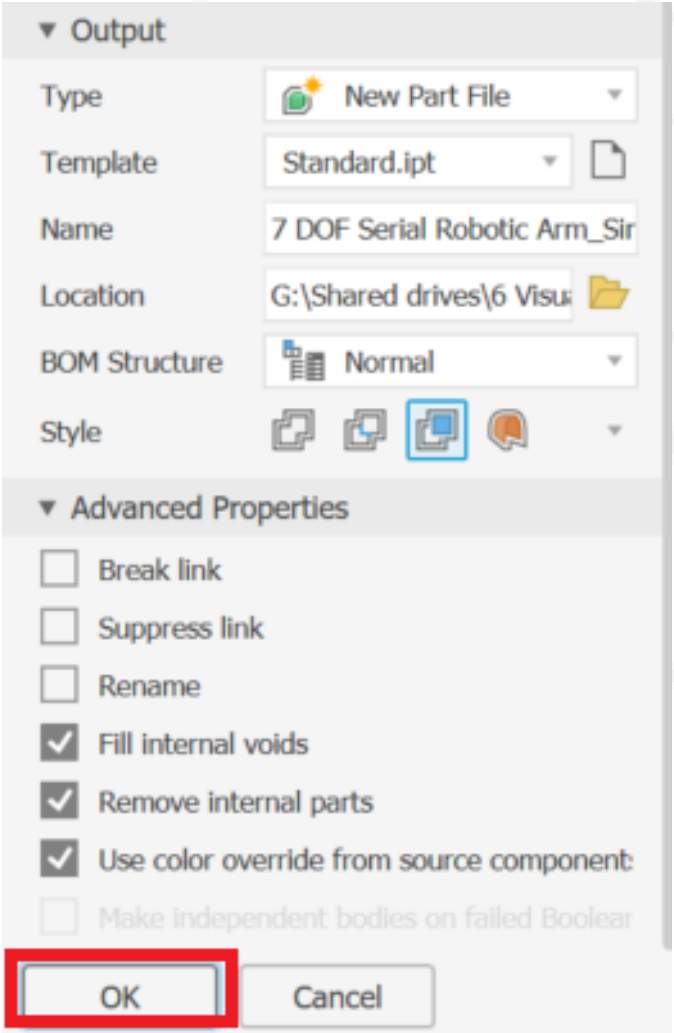
Tip: press the highlight button to view all the features that are being removed

3 Export assemblies

This tool can only be used with assemblies, please see Process #2 for single parts.

5. (Optional) if you would like to remove features as well as parts, you can do so by using the 'remove features' section

6. Once satisfied with all of the parameters, click ok and Inventor will remove everything specified. By default it will output it as a separate part file, and it is recommended to keep it this way



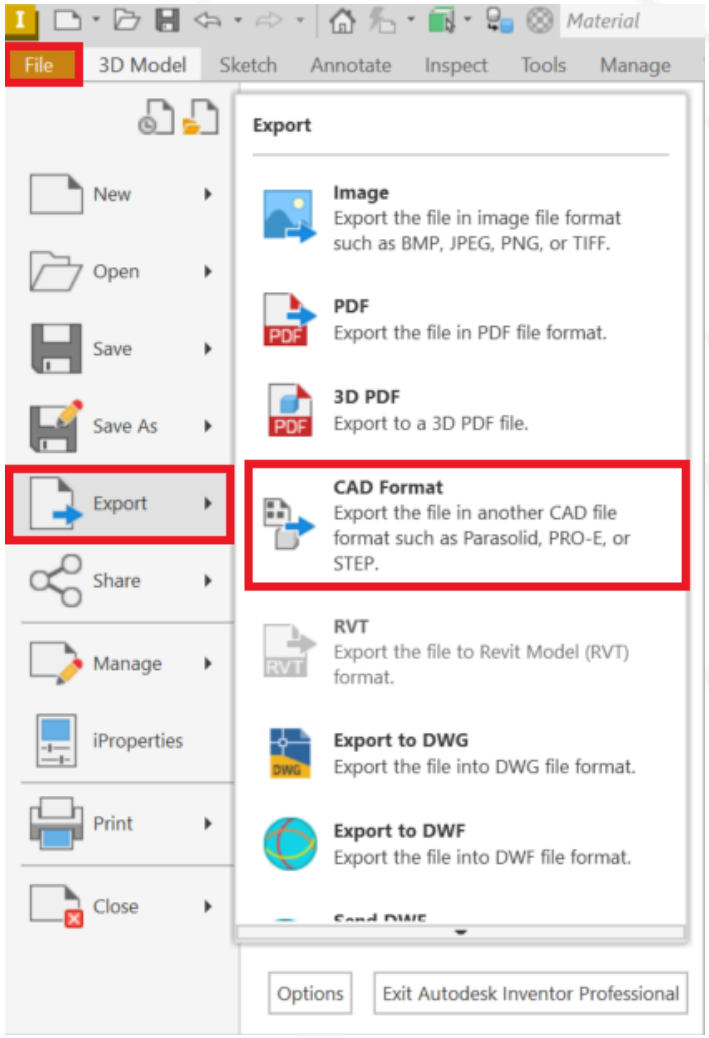
3 Export assemblies

This tool can only be used with assemblies, please see Process #2 for single parts.

5. (Optional) if you would like to remove features as well as parts, you can do so by using the ‘remove features’ section

6. Once satisfied with all of the parameters, click ok and Inventor will remove everything specified. By default it will output it as a separate part file, and it is recommended to keep it this way

7. From there nothing further needs to be done in Inventor as the model will be exported as a .step file. Navigate to file > export > CAD formats



3 Export assemblies

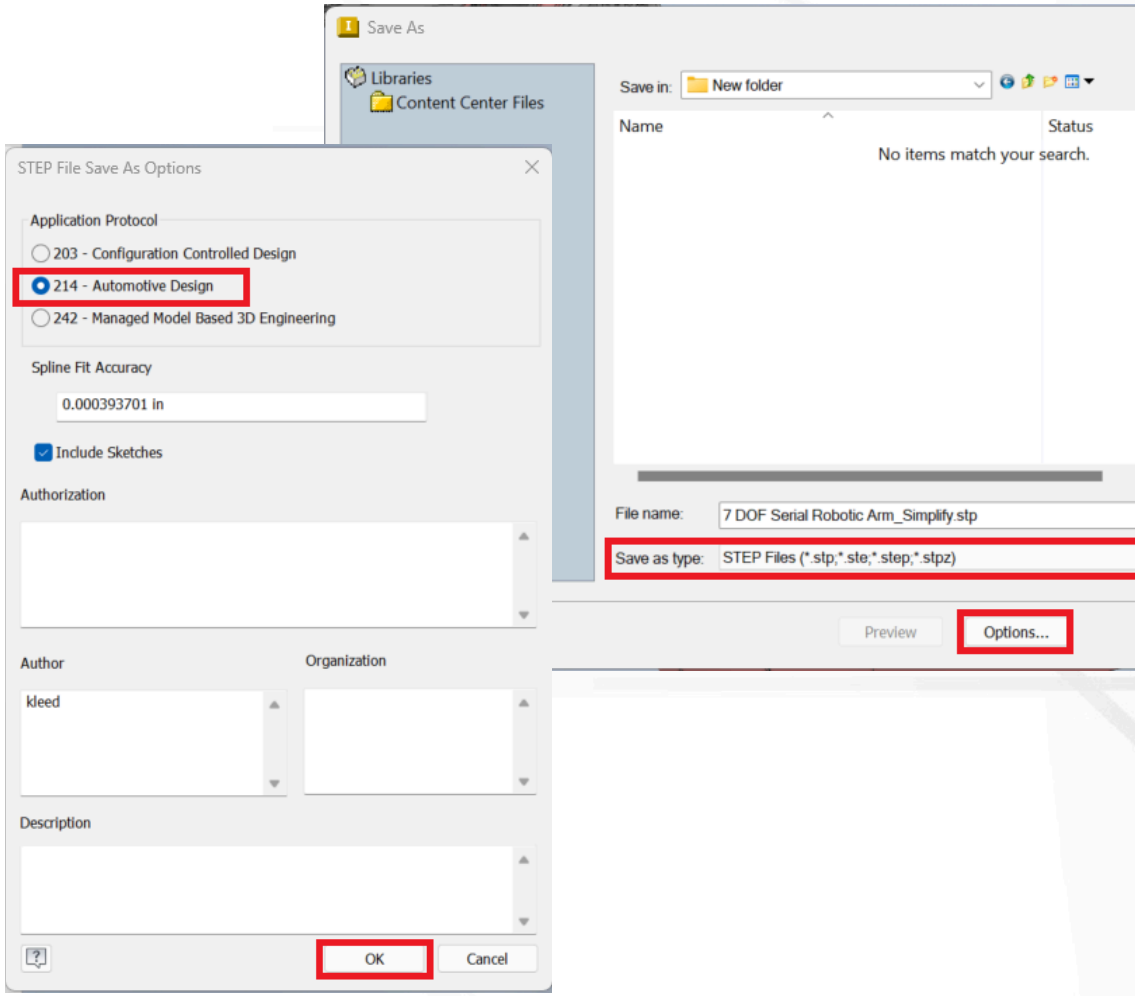
This tool can only be used with assemblies, please see Process #2 for single parts.

5. (Optional) if you would like to remove features as well as parts, you can do so by using the 'remove features' section

6. Once satisfied with all of the parameters, click ok and Inventor will remove everything specified. By default it will output it as a separate part file, and it is recommended to keep it this way

7. From there nothing further needs to be done in Inventor as the model will be exported as a .step file. Navigate to file > export > CAD formats

8. Select .step file and ensure that it is a step214 format (in the options)



3 Export assemblies

This tool can only be used with assemblies, please see Process #2 for single parts.

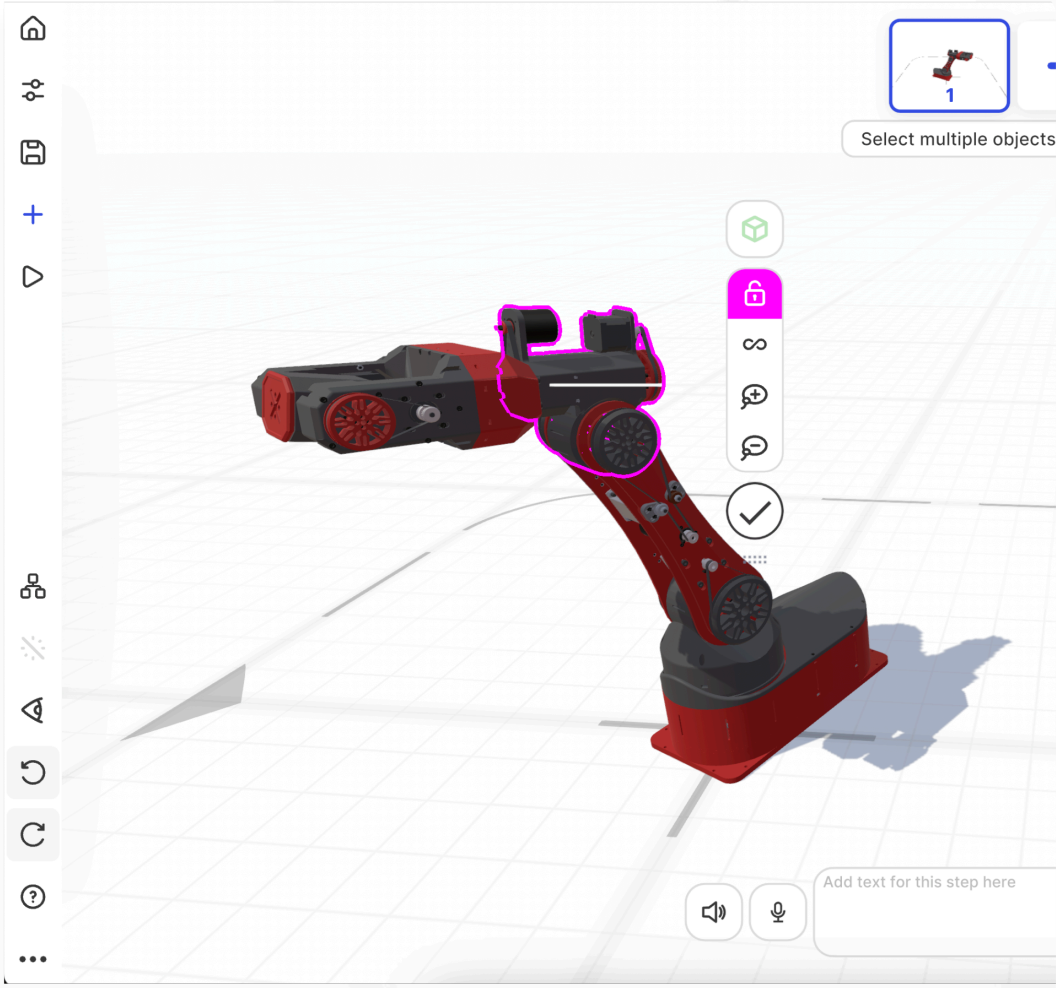
5. (Optional) if you would like to remove features as well as parts, you can do so by using the ‘remove features’ section

6. Once satisfied with all of the parameters, click ok and Inventor will remove everything specified. By default it will output it as a separate part file, and it is recommended to keep it this way

7. From there nothing further needs to be done in Inventor as the model will be exported as a .step file. Navigate to file > export > CAD formats

8. Select .step file and ensure that it is a step214 format (in the options)

9. Import the file into JigSpace, and regroup subassemblies using JigSpace’s grouping tools



4 Export single parts

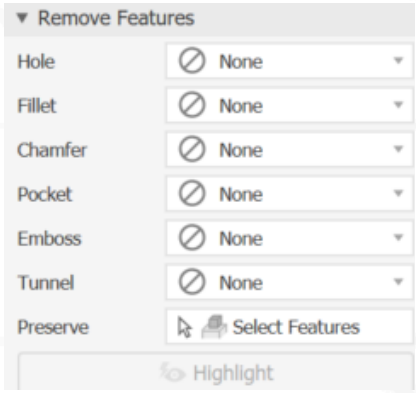
The Simplify tool in Autodesk Inventor can only be used on assemblies, so if you have a complex single part that you'd like to simplify in Inventor, simply follow this process:

1. Create a new assembly file

2. Import the part you wish to simplify

3. Follow the simplify process in the previous section (pages 6 - 14)

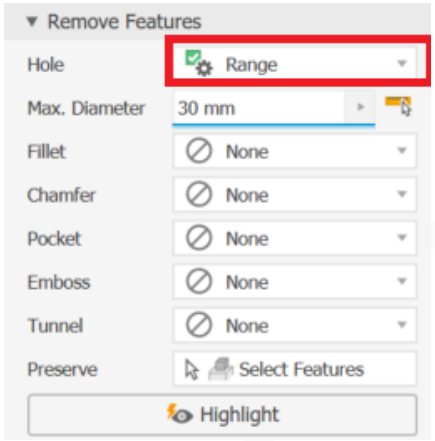
4. Export simplified assembly as a STEP file.



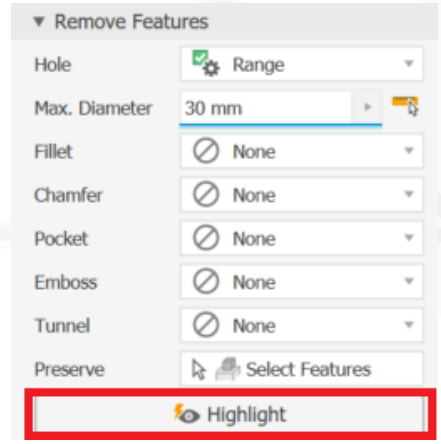
Remove features

You will need to specify the 'remove features' portion (see page 8), as there is only a single part.

Since this is heavily dependent on the model, you will need to try different values to find out what works best for you



Select the feature to remove > in the dropdown menu > select range > specify the range

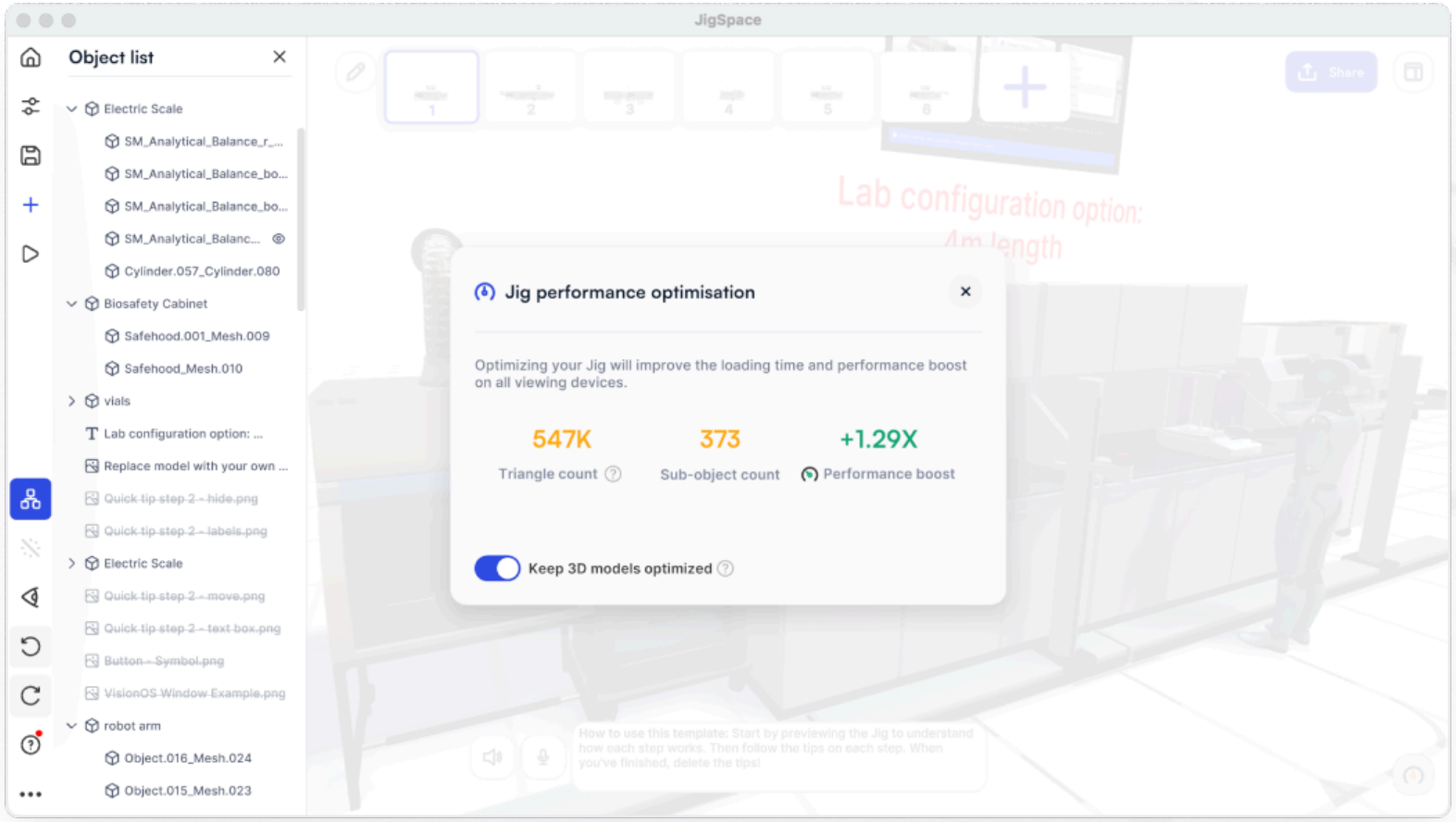


Tip: press the highlight button to view all the features that are being removed

5 Using the performance indicator

When uploading 3D models to your Jigs, the performance indicator displays a read out of your current triangle count and number of meshes (or sub-objects). This helps you determine when to optimize or simplify the 3D models in your Jig.

For best results we recommend less than 200,000 triangles (or polygons) and less than 50 sub-objects. Jigs also have an option to automatically keep your 3D models optimized, which we recommend keeping toggled on.



6 Jig Certification Program

Take your 3D optimization to the next level with our training Certification Program.

How to earn certification

1. Contact us: Contact us about upgrading to an Enterprise license
2. Schedule Training: Complete the required training modules with your customer success manager
3. Submit a Jig: Submit a Jig for evaluation (required for editor and creator certifications).

Issuing your certificate

After successfully completing your certification, you will be issued with:

- a certification badge to add to your LinkedIn profile or CV to show to your peers and current employer
- a certificate of achievement is available upon request, confirming your skills have passed assessment.

If you are being certified as part of a team-wide program, we will also notify your team manager and admins.

About DTCo

This guide was prepared in collaboration with Design Technology Company (DTCo).

DTCo provides a low risk, easy to engage, scalability option for mechanical design teams. Our literacy with design technologies makes us an essential part of any engineering project.

With over 20 mechanical designers have deep Computer Aided Design experience that is enhanced by a team with a broad understanding of the currently available design technologies. These include laser scanning, photogrammetry, cloud based collaboration for CAD, virtual reality (VR) and augmented reality (AR).

Learn more: www.designtechco.com.au

About JigSpace

JigSpace enables teams with big ideas to create and share stunning 3D presentations and augmented reality experiences - we call them “Jigs” - in seconds They can be viewed on any device, from mobile to web browsers to the Apple Vision Pro.

Create a free account to try for your self.

Showcase what makes your product unique, go deep with step-by-step instructions, and make it engaging training simulations in augmented reality.

Learn more: www.jig.com