Advanced Assembly Control with Representations in Inventor

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This class will teach you how to use the 3 types of Representations in Inventor software assemblies. Representations can control the visibility, the suppression, the position, and several other characteristics of components within assemblies. Representations enable you to save specific views of your assembly to prepare for presentations and to create drawings. Proper use of Representations also significantly improves computer performance when dealing with large, complex assemblies.

Learning Objectives

At the end of this class, you will be able to:

- Learn how to create and manipulate the 3 types of Representations

- Learn how to determine which type of Representation is most suitable for a given task

- Learn how to improve computer performance and usability of complex assemblies by using Levels of Detail

- Learn how to improve the quality of drawings and live presentations of complex assemblies

About the Speakers

Chris Benner is the CAD department supervisor at [Powell Fabrication & Manufacturing, Inc](). in St. Louis, Michigan. He has been working as a mechanical designer and drafter for more than 20 years, and he has been using Autodesk, Inc., products for the majority of that time. For the last several years he has been the lead CAD technician and Vault administrator at Powell Fab, integrating it with Inventor software, AutoCAD software, and AutoCAD Electrical software, as well as Microsoft SharePoint 2013 software. He has been implementing changes to the company's document-management procedures using Vault Professional software as the main system for lifecycle and revision control, and he has been engineering change orders to control all drawing changes.

Recently he was promoted to the position of CAD supervisor. Autodesk recognized Chris 2 years ago as an Autodesk Expert Elite for his activity on the Autodesk discussion forums and social media and for his CAD Tips Tricks & Workarounds blog. Design & Motion at [DesignandMotion.net]() has also published his articles, as well as CAD Notes at [CAD Notes.com]()

Mike Thomas graduated with the honor of Most Distinguished Graduate in CAD/CAM engineering technology from the Saskatchewan Institute of Applied Science and Technology (SIAST). He is a specialist in the manufacturing industry with a strong knowledge of the Autodesk, Inc., mechanical products, and he has a solid understanding of document management, hardware, networking, and other Autodesk technologies.

He is now the technical services manager for [Prairie Machine & Parts Mfg. (PM&P)](), and he is responsible for overseeing the Engineering department's technical operations and strategic technical growth. His primary duties include providing ongoing support of critical computer systems and programs, facilitating the interactions between the engineering department and other departments, providing the engineering department with effective systems and technology, and working with PM&P's vice president and engineering managers on the development and implementation of a cohesive strategic plan for the technical growth of the department. Mike is an active contributor to Design & Motion ([www.designandmotion.net]()).

Contents

Representations

There are three types of representations in Inventor: View, Positional, and Level of Detail (LOD). Each representation helps you work with assemblies, saving time in many ways as assemblies become larger and more complex.
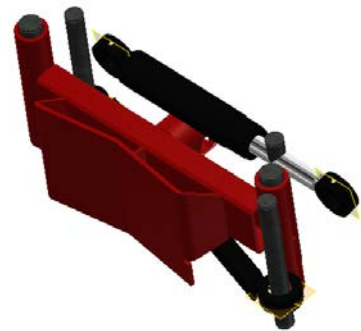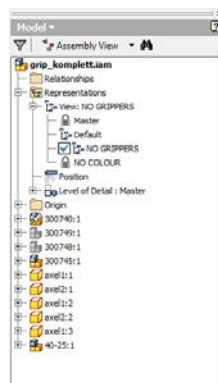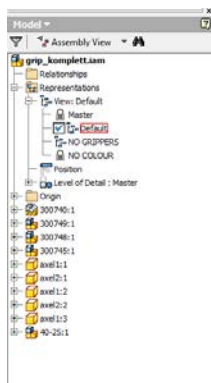
Design View Representations

What are View Reps?

View Representations capture view related information in a named configuration that can be restored at a later date. View Representations are used to control the display state of the assembly to aid during the design process, setup presentation / sales type information, to improve the performance of large assemblies, and to reduce the time it takes Inventor to generate drawings views.

Assembly View Representations capture the following:

- Component Visibility (as in on or off)
- Colour (and other style characteristics) of individual components within the assembly
- Sketch & Work Feature Visibility
- Zoom magnification and viewing angle

View Representations can be used to:

- Turn off component visibility (as needed) to simplify the current task
- Capture viewing angle and zoom factor to return to at a future time. Convenient for presentations
- Assign a unique colour to a component instance (say opaque – see through one instance)
- Control sketch and work feature visibility to reduce the clutter on the screen
- Simplify positional representations by turning off all unnecessary components. This can be further used when creating Overlay views in the drawing to only show the components effected by the positional representation
- Help manage multi-user projects in that each user can have their own design view focusing on specific areas of the assembly
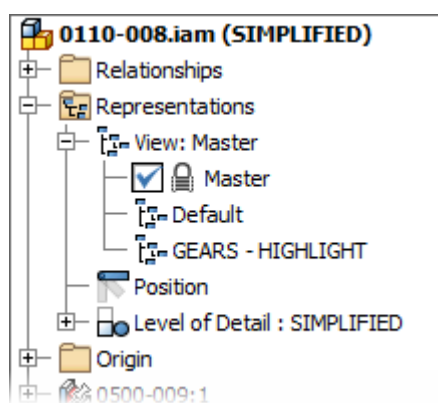
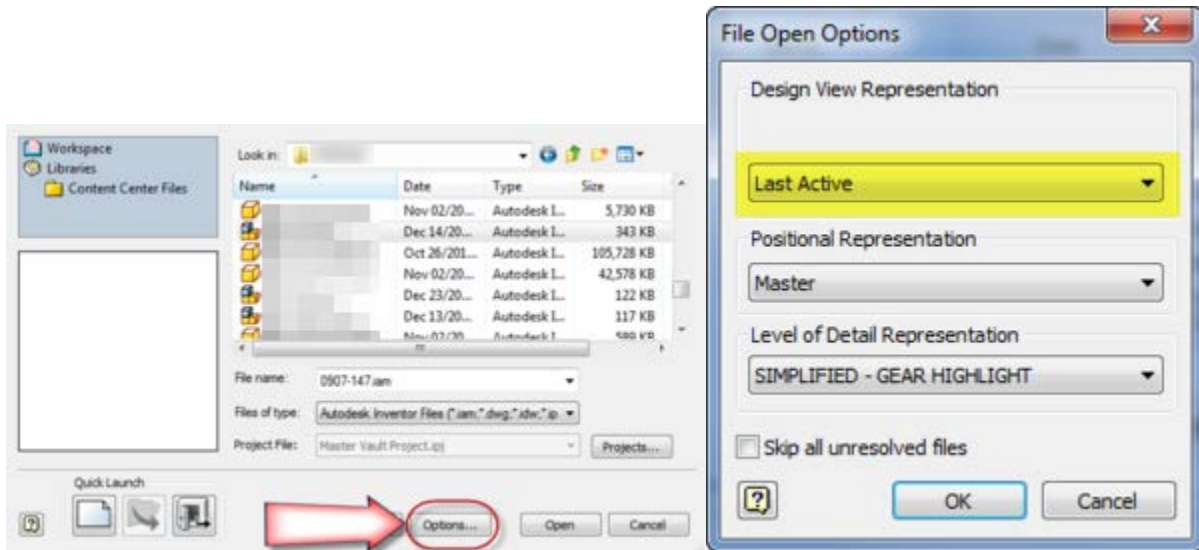| IMPORTANT! |
| --- |
| Creating drawing views within Inventor can be quite time consuming especially for larger assemblies. Improve performance by reducing the amount of components within the view that Inventor has to work with. |
| Although View Representations can control component visibility, which can improve the performance of Inventor while generating drawing views, it is **HIGHLY RECOMMENDED** to use **Level of Detail Representations** to manage the visibility of the components. Level of Details provide the greatest amount of performance gains for drawing views and when dealing with the Assembly in general. |

Working with View Representations

Use the Inventor browser to create and manage View Representations. All options are available via the right-click menu


- To create a new View Rep, Right click the Master View Rep and select New.
- Double-click to activate the View Rep, click twice slowly to rename
- View Representations are "*live*" in that they update automatically as the viewing information of the assembly is adjusted
- Lock View Reps to prevent any changes. To lock a view representation. right-click on it in the browser and select **Lock**

When opening an assembly you can specify the view rep that will be active as the assembly is opened. By default, the last active View Representation is the one used.

Using View Representations, you can setup the assembly specifying view orientation, visibility, color, and other attributes of the components. When creating drawing views you set the View Representations to apply to the view.
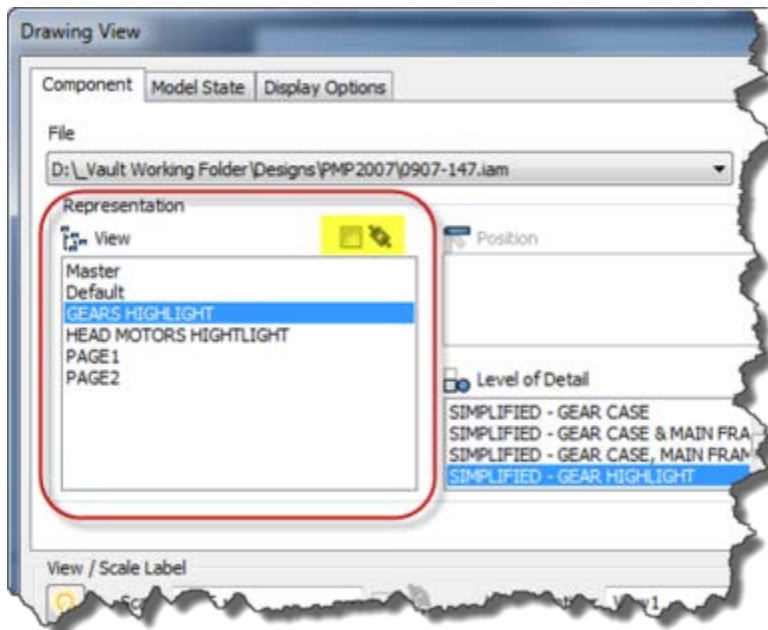
If the view rep is set to associative:

- The drawing view will update as the assembly view rep is adjusted.
- You cannot adjust component visibility within the drawing view from within the drawing's browser.

IMPORTANT
Create as many View Representations required for your Assembly but rename them appropriately. The default names Inventor gives new View Reps (View1, View2, etc) is NOT acceptable!

Design View Associativity

When you place a component into an assembly, the component remains associated to its View Representations. This means that changes to the component (as in changing workfeature visibility or turning parts on/off) automatically updates the instances of the component in the assembly.
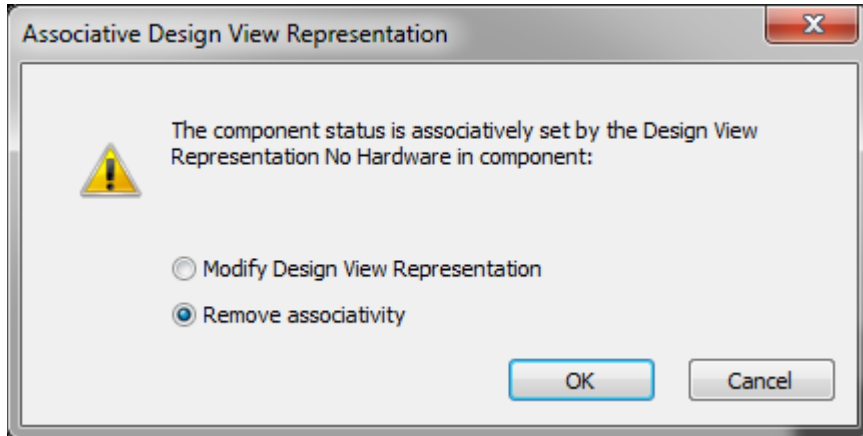

When you make a change to this component in context of the assembly, you are really asking it to go against the view rep it is associated with. Because Inventor is not sure what to do, it prompts you.

If you select *Remove associativity*, the component will no longer be associated to its own view representations. You will see your changes in the assembly but any changes to the components view reps do not apply to the instance in the assembly.

If you select *Modify Design View Representation* then you are updating the view rep not just for that instance but also in the component itself. This means that anywhere that component is used will see the changes to the view rep.

Take for example where the Brace Assembly is placed into the assembly so that the View Rep IN ASSEMBLY is active and associative.
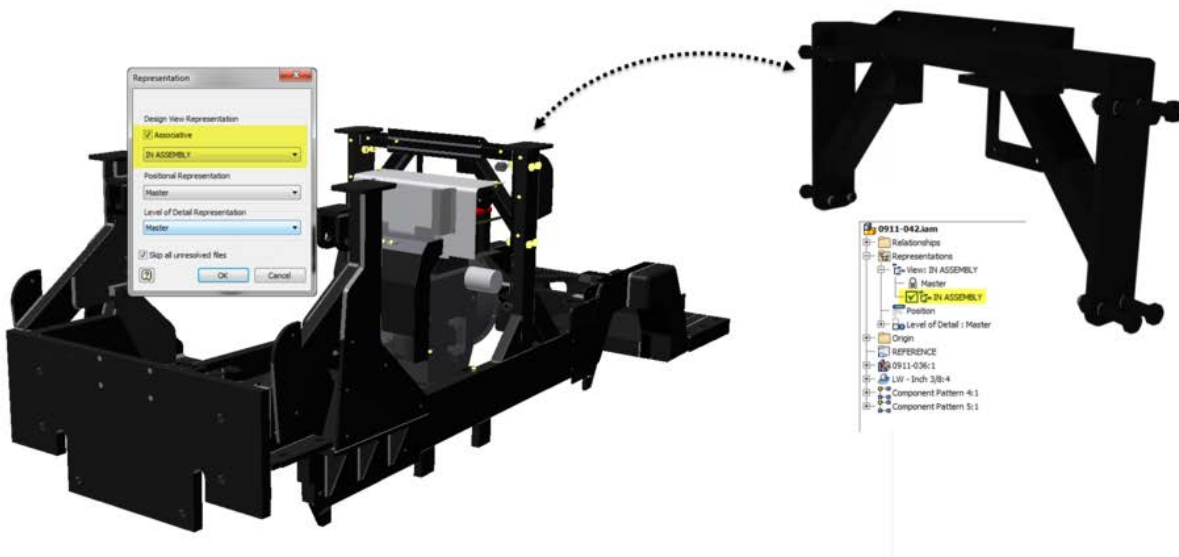
Any View Rep changes to the Brace are automatically reflected in Tractor Assembly.


However, if I make changes to the visibility of components in the Brace from within the Tractor I am prompted by Inventor to either modify the associativity or remove it.

In this case, I decide to remove the associativity. This means the Canopy Brace is updated but the changes are not sent back to sub-assembly.

At any point, I can reset the View Rep used and make is associative making the instance in the assembly appear as in the sub-assembly. Any changes I made to sub-component visibility within the assembly will be lost as the components view rep will set the visibility details.

Positional Representations

Positional representations capture "*snapshots*" of assemblies to allow for motion studies and evaluation of the assembly in various positions. Use these snapshots to review motion and evaluate the position of assembly components in both the modeling and drawing environments.
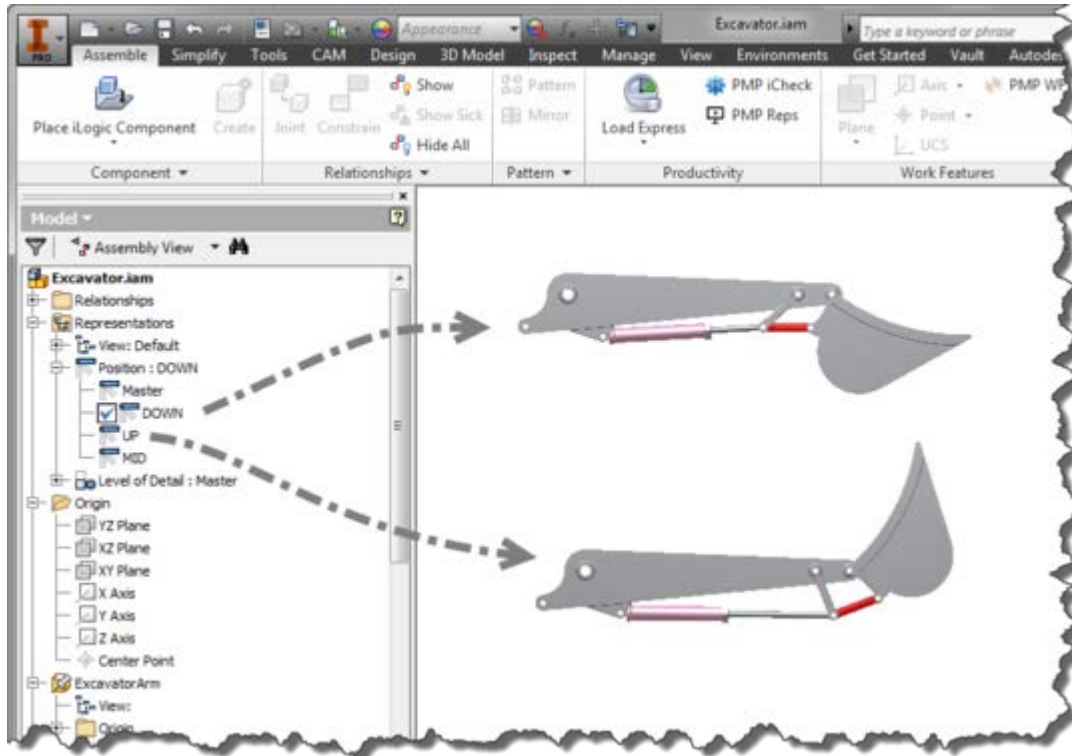
Positional Representations allow us to display different positions of the assembly components. The *Master* Positional Representation (automatically created) represents the default state of the assembly, where your modeling operations take place. You cannot edit the *Master* Positional Representation,

Creating a new **Positional Representations** allows us to make changes to the component positions and constraint settings without affecting the original component positions in the Master. For example,a Hydraulic Cylinder can have a retracted and extended position

The following steps are used to create a Positional Representation:

1. Create a Positional Representation.
2. Modify (Override) values for the Positional Representation.
3. Save the changes in the assembly.

Creating Positional Representations

To create a new Positional Representation, expand the Representations folder in the Model browser. Right click and select New, expand the Position branch to view the Master and the new Positional Representations.

Modify (Override) values for the Positional Representation

Make the desired Positional Representation active before assigning overrides. Double-click the positional rep within the browser to make it active. Locate the constraint that needs to be modified in the Model browser, right-click and select Modify (Override). When the Override Object dialog box appears, adjust the values.

What can I or can't I do with positional representations?

You cannot perform the following operations when a positional representation is active:

- Add or delete components.
- Restructure the assembly.
- Adjust a pattern.
- Create (or delete) work features.
- Create, modify, or delete assembly features.
- Toggle adaptivity.

IMPORTANT

Be careful with your Level of Details when working with Positional Reps as you do NOT want to suppress components constrained by the constraints you are overriding as this will cause you nothing but grief and headaches

Save the changes in the assembly.

When a Positional Representation is active, you are not able to save any changes to an assembly.

To save the changes in the assembly, activate the Master in the top-level assembly.

Level of Detail Representations

To deal with large assemblies Inventor uses an Adaptive Data Engine built on a segmented database. This database organizes data for quick retrieval and segments the data so only the required data is read from the disk. When opening an assembly, Inventor only loads what is absolutely necessary. However as Inventor assemblies get larger (as in the number of components) and the components within that assembly get more complex Inventor requires more and more memory (both Video and RAM) to deal with the assembly. **Levels of Detail Representations** provide tools to help manage the RAM usage of Inventor by reducing the complexity of the components in the assembly and reducing the number of components in the active assembly.

The easiest way to remember the difference between View Representations and LOD's is that View Reps capture the "look" of a component whereas Level of Details are for assembly performance, capturing whether the component should be loaded into memory.

Working with the Standard Level of Details
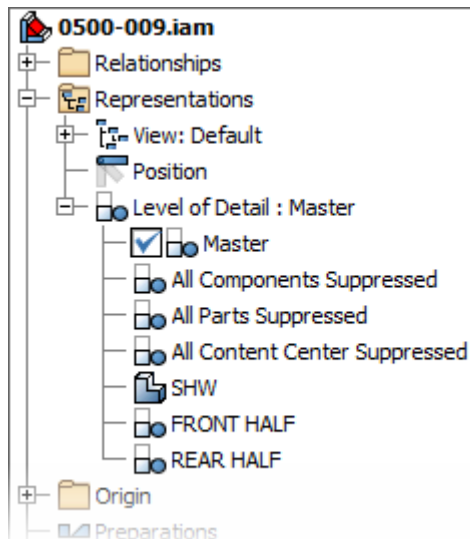
Level of Detail Representations are created and managed within the Inventor Browser. All options are available via the right-click menu.

IMPORTANT!

Parts and Subassemblies CANNOT be suppressed without a Level of Detail. If you do not create a LOD prior to suppressing components a LOD will be created for you. These auto-created LODs will be prefaced with a ~

Each assembly has four built in LOD's: Master, All Components Suppressed, All Parts Suppressed, and All Content Center Suppressed:

1. Master is the default LOD, This LOD will always show each assembly and part as is, no components suppressed
2. All Content Center Suppressed will suppress any component originating from the Content Center
3. All Parts Suppressed suppresses all parts, but leaves the sub-assemblies unsuppressed
4. All Components Suppressed suppresses everything… all parts, all sub-assemblies

IMPORTANT

Level of Details can be nested in that an assembly LOD can call a sub-assembly LOD

The advantage of using All Components Suppressed is that any assembly regardless of the number of parts can be opened almost instantly. Once open additional LODs can be created to unsuppress the components you wish to work on / with

IMPORTANT

The Bill of Materials (BOM) will ALWAYS show the master LOD. You may notice that will accessing the BOM that Inventor loads in all of the components, this means that it may take longer than normal to access the BOM as its loading components into memory

Setting the LOD on open

The active level of detail can be set as you are opening the assembly. This is a great way to decrease the time it takes to open an assembly. For example, use the All Parts Suppressed LOD to open the assembly and then unsuppress the components requiring changes.
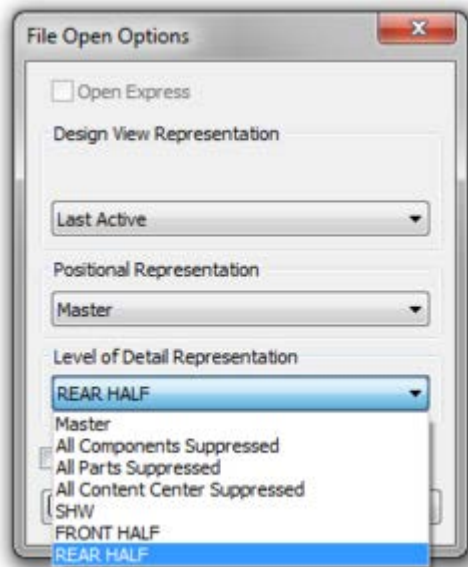
LOD & Mass

When working with LOD's you may be prompted on how you want to deal with the Mass properties. You have the option to either calculate the mass properties for the active LOD, meaning any suppressed component will be ignored. Alternatively, you can calculate the mass properties for the master level of detail even though you have a different LOD active.

Linking LOD
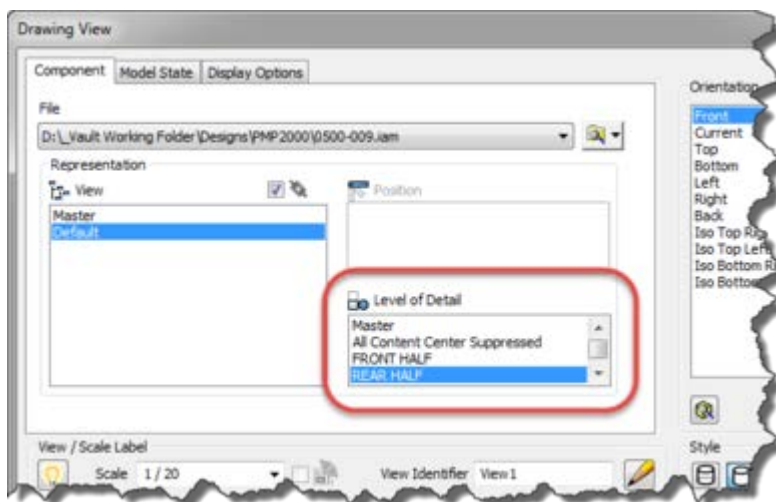
Link Levels of Detail

Within the Productivity Tools, area of an Inventor assembly is the Link Level of Details functionality. It "provides the means to select a named Level of Detail representation from the active assembly, and activates a Level of Detail representation with the same name within subassemblies containing a Level of Detail representation with that name."

| IMPORTANT! |
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| The names of the LOD must be spelled EXACTLY the same and MUST BE IN THE SAME CASE, as in all uppercase or lowercase lettering |

Creating Level of Details

LOD's have 3-main types

- Component Suppression
- Substitute - Assembly Simplification
- Component Substitution

Component Suppression

The simplest option, this type of LOD captures component suppression. With the LOD active suppress components by selecting on the screen or in the browser, right-clicking and select suppress.
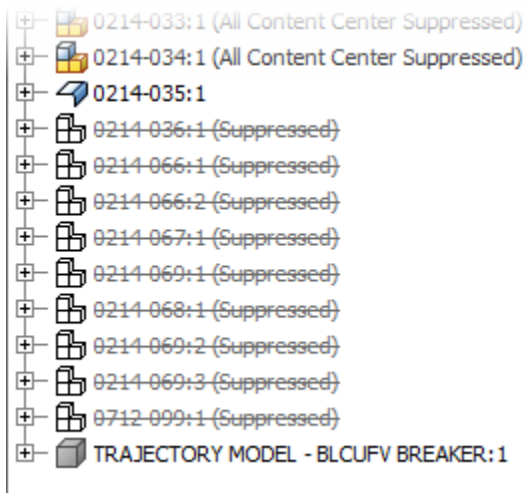

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⚠	Suppressed components cannot be selected on the screen and are represented in the browser greyed and with a line thru them.

Substitutes

By creating a new Substitute LOD you are creating a new part that will be used in place of the assembly component. On top of reducing the number of components, the creation of this substitute provides options to simplify the part.

When generating a derived assembly you control which parts are included, how they are included, and whether a solid or a surface model is generated.
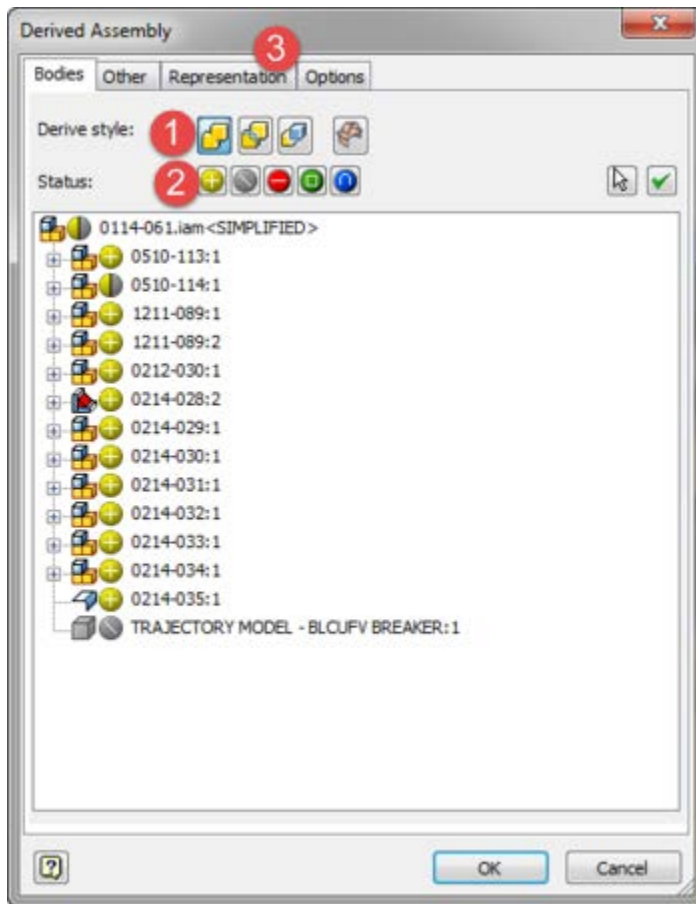
1. **Derive Style:** Solid – No Edges/Merge Seams, Solid – With Edges/Maintaining Seams, Multi-body Solid, or a Surface model
2. **Derived Type**: To include or exclude the component or to perform a Boolean operation (as in use the component to subtract / union / intersect). Select the icon to set the operation for the component. The default is the yellow "+" which means the component is included and added to the substitute
3. Use the **Representation** tab to select pre-existing representations (View, Positional, or LOD) to base the new substitute off.

The shrink-wrap option can be thought of taking shrink wrap and covering the outside of your assembly. This shrink-wrapped representation is used as a substitute within the assembly as a level of detail.
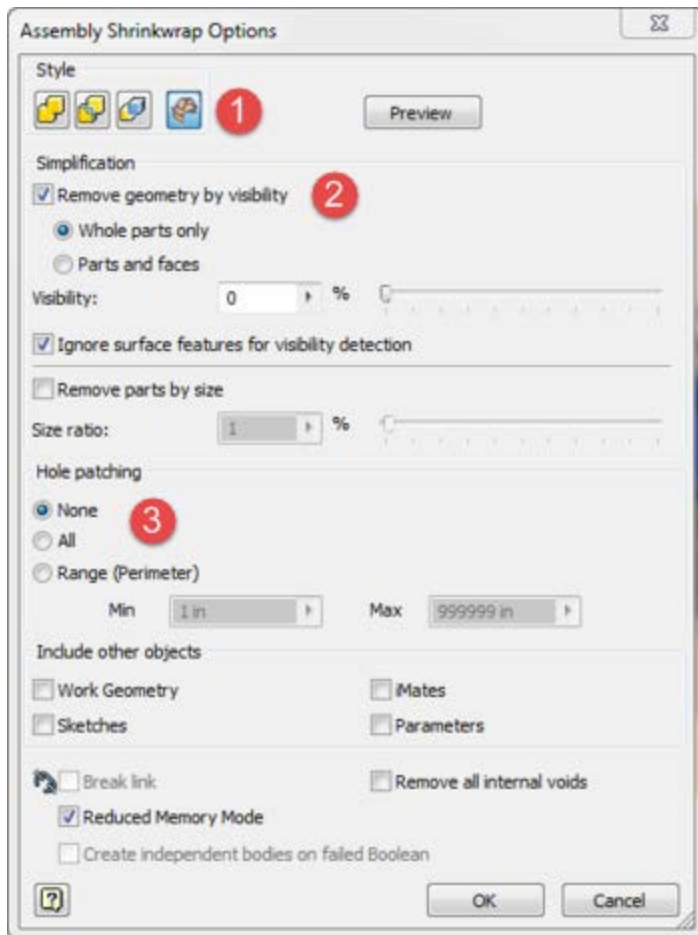
1. **Output** (either solid or surface)
2. **Simplification** - Components can be removed based on percentage visible.
3. **Hole Patching** is used to patch any cutout in the model (it doesn't have to be circular). This is useful to simplify the part, but is optional.

Part Substitution (Select Part File)

The last option is to model a simplified version of the assembly as a stand-alone part file. This file is then selected when creating the Level of Detail representation.

Inventor Simplify Tools

"*When you do not need to display all the details provided in a part (.ipt) file for an assembly, create a simple representation of the assembly with only the critical information, such as mounting points. Use the Simplify tools to reduce model complexity for easier and faster referencing throughout your design process.*"
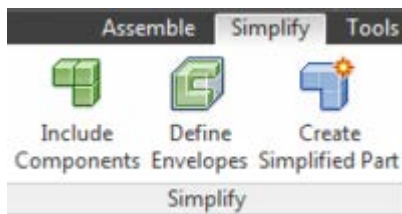
Why would you simplify the model?

- Remove intellectual property so the model can be shared with others without the possibility of reverse engineering
- Improve performance

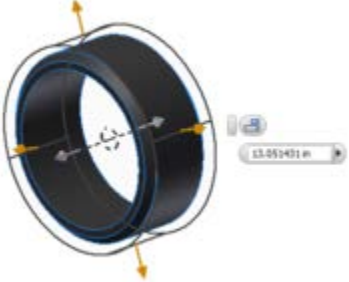| IMPORTANT! |
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| In some instances, these tools are referred to as BIM Simplify tools, BIM standing for Building Information Model. Although we will not be exchanging information with Autodesk Revit there are still opportunities to use these tools to simplify the model |

Include Components

The Include Components feature creates a new View Representation (labeled Simple View 1, Simple View 2, etc) and provides tools to select just the components you want to include in the new View Rep.

	Changes the View to show all components, only included components, or only excluded components. Continue Picking as needed regardless of the view set
	Specifies the Selection Priority: Component (sub-assemblies) or Part (selects parts regardless if in a sub-assembly)
	Use the Select All Occurrences option to select all component occurrences (instances) of the selected component
	Undoes the previous selection
	NOTE: Shift + selection deselects the component from inclusion

Define Envelope

Generates an Envelope object, a rectangular or cylindrical bounding box representation of the selected component. Use this to remove complexity from the exterior of included parts

	1. Defines the Simple Shape; Bounding Box or Bounding Cylinder 2. Specifies the Selection Priority: Component (sub-assemblies) or Part (selects parts regardless if in a sub-assembly) 3. Use the Select All Occurrences option to select all component occurrences (instances) of the selected component 4. Toggle to show or hide the original components 5. Pick a face to get the material applied to that face, useful to insure envelope parts match the colour and appearance of the components around it 6. Changes the material that is applied to the selected (simplified) geometry.
	Once selected you can use the arrows on the screen to adjust the bounding object; single click to specify the offset distance or click-and-drag to dynamically adjust the size of the object. An align option will also appear after picking an arrow allowing you to pick an existing face to snap the envelope face to.

The simplified envelopes will appear in the browser. If you want to redefine an envelope, delete it and define a new envelope.

To view the original assembly without the envelopes, in the browser, drag and drop the End of Feature node above the Envelope node, and select Show Original of the envelope from the context menu.

Create Simplified Part

Create Simplified Part derives all included components (and envelopes) into a single part. This can be used to create a new Level of Detail in the assembly
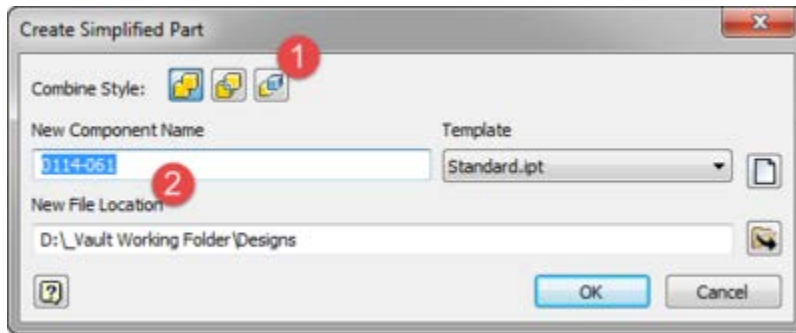
1. Select the Combine Style: single solid body with seams merged, a single solid body with seams maintained, or each solid as a separate body.
2. Specify the filename and location. The filename of the simplified component should always start with the assembly number, as in 0211-001-SIMPLIFIED.ipt