IM196136

Working with Inventor View Representations

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Learning Objectives

- Discover the different types of representations - understanding the application for each type
- Learn how to create View Representations
- How to use View Reps to set up presentations and drawing views
- How to efficiently manage View Representations

Description

This class will teach you how to use View Representations in Inventor assemblies. Representations control the visibility and several other characteristics of components within assemblies. Representations let you save specific views of your assembly to prepare for presentations and create drawings. Proper use of Representations will significantly improve your system’s performance when dealing with large, complex assemblies.

Speaker

Mike spent the first 12-years of his career in the Autodesk channel working for an Autodesk reseller as an Application Specialist. During his travels, he was fortunate to help solve many issues utilizing Autodesk software. Mike has been using AutoCAD since r13, cut his solid modeling teeth on Mechanical Desktop, and has been using Inventor since before it was known as Inventor.

Now he is the Technical Services Manager at Prairie Machine a mining equipment manufacturer. Reporting to the general manager, Mike is responsible for overseeing the company's technical operations and the strategic technical growth.

Contents

Representations

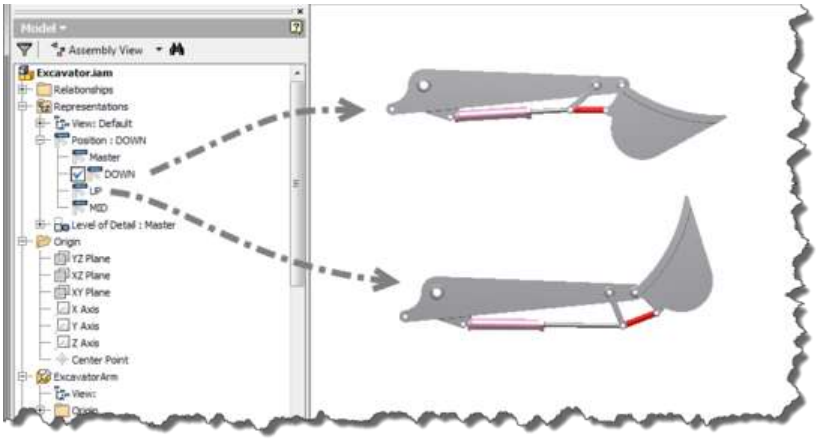
There are three types of representations in Inventor: **View**, **Positional**, and **Level of Detail (LOD)**. Each representation helps you work with assemblies, saving time in many ways as assemblies become larger and more complex.

View Representations

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.

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.



Level of Detail Representations

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.

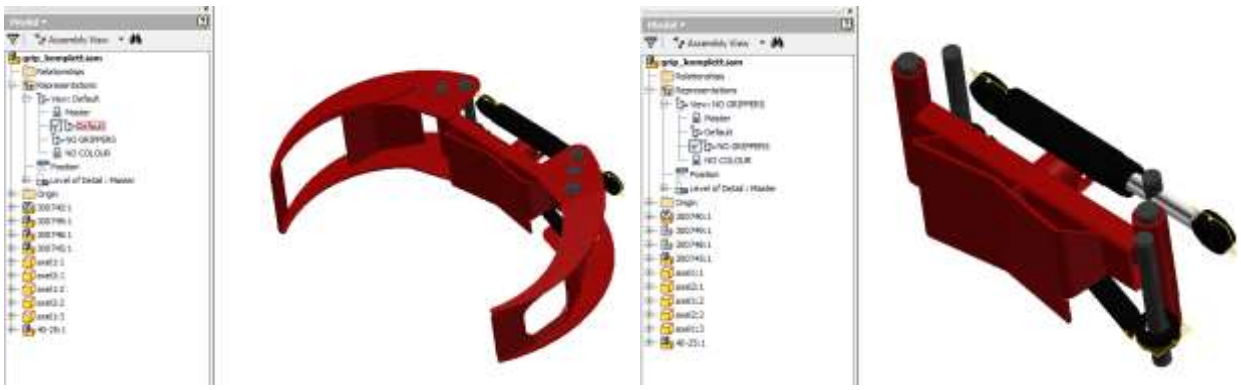
Design View Representations

What are View Reps?

View Representations *capture* view related information in a named configuration, restorable later. Use **View Representations** 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), **Status** (enabled / disabled), & **Transparency**
- **Colour** (and other style characteristics) of individual components within the assembly
- Sketch & Work Feature **Visibility**
- **Camera State**, **zoom magnification**, and **viewing angle**

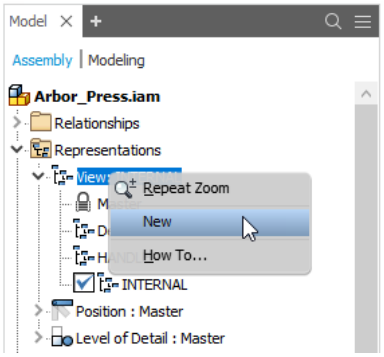


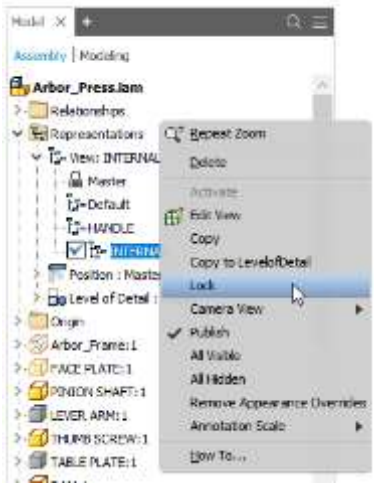
Utilize **View Representations** to:

- Turn off component visibility to simplify tasks
- 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. Use this further when creating Overlay views 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
- Manage component inclusion in drawing views - aids in documenting the assembly and improving the performance view creation
- Create unique views to filter the parts list in a drawing

Working with View Representations

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





Double-click to **activate** the View Rep, *click twice slowly* to **rename,** and drag-and-drop the view to reorder it in the list**.**

Use the *right-click menu* to:

- create **new** View Reps
- **Copy** to create duplicates of existing representations
- **Remove Appearance Overrides** to restore overridden component appearances to their original styles
- Use **All Visible** and **All Hidden** to quickly toggle the visibility of all components (top-level) within the assembly
- Select the desired **Annotation Scale** to manage the size of 3D Annotations
- manage the capturing of camera positions with the **Camera View** options

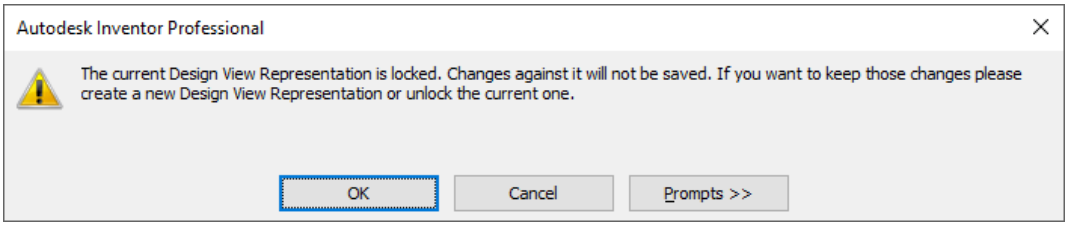
IMPORTANT

Create as many View Representations required, but name them appropriately. The default names Inventor gives new View Reps (View1, View2, etc.) are NOT acceptable!

Changes against it will not be saved?

View Representations are *live* by default. View Reps 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.**

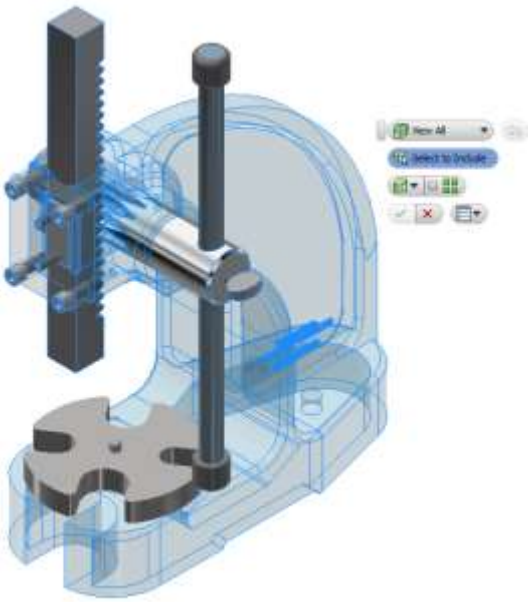
Adjustments are still allowed, even with the view **Locked**. However, these changes are NOT saved to the View Rep. Attempting to save changes to a locked View Rep causes the most panic-inducing message in all of Inventor!



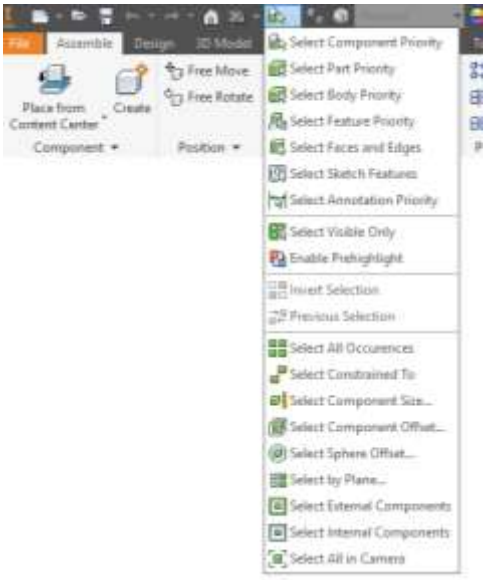
	With unlocked View Representations, **Isolate components** updates the view rep. However, **Undo Isolate** restores the visibility to the state immediately before isolation.

Edit View

Utilize Edit View to work with component visibility dynamically within the modeling window opposed to within the browser.

	Use the **View Options** to toggle the display between **all components, excluded** (invisible), and **included** (visible) components. Toggle the selection priority between **Component**, **Part**, & **Parent** priority Enable **All Occurrences** to select all instances when selecting a single component.

Utilize the Selection Tools



Utilize Inventor's selection tools to aid in setting component view configuration.

Examples:

- **All in Camera** to select only components that you can see (in the view plane of the active camera)
- **Internal Components** to select the components you cannot see, the ones blocked by external components.
- **All Occurrences** to quickly select all instances of a component within the assembly

Setting the Active View Representation

Assemblies – On Open & Placing Components

When opening an assembly or when placing a component, specify the desired active view rep using the **Options**. By default, the **last active** representation is used.

When opening an assembly, Inventor only loads the visible components into memory (good for performance).

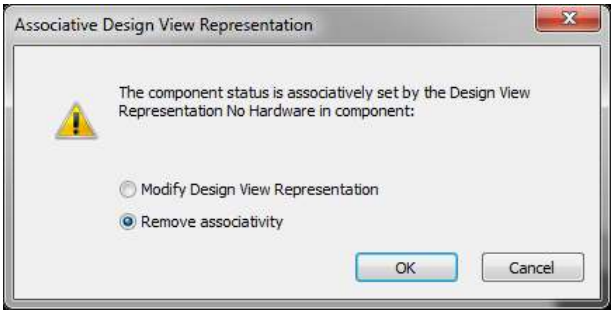
Especially with larger assemblies, it is advantageous to use the Nothing Visible option to open the assembly with all components turned off (invisible) and then manually turn on only the components you need to work with.

Whenever a component is made visible, it is loaded into memory. Thus, if you have several view representations and activate each of them, all components made visible across the representations are loaded into memory.

Without using Level of Details, the only method to unload unnecessary components is to close the document and reopen it with a specific view representation.

Design View Associativity

When you place a subassembly into an assembly, you have the option to have the component remains associated to its View Representations. Therefore, as you make changes within the subassembly (as in changing work feature visibility or turning parts on/off) Inventor automatically updates all instances of this subassembly within the assembly.



When you make a change to this subassembly within the assembly, you are really asking it to go against its associated View Rep. Because Inventor is unsure of what to do, it prompts you.

Select **Remove associativity** to break the link to its own representations. The changes are applied within the assembly, but these changes do not propagate into the subassembly.

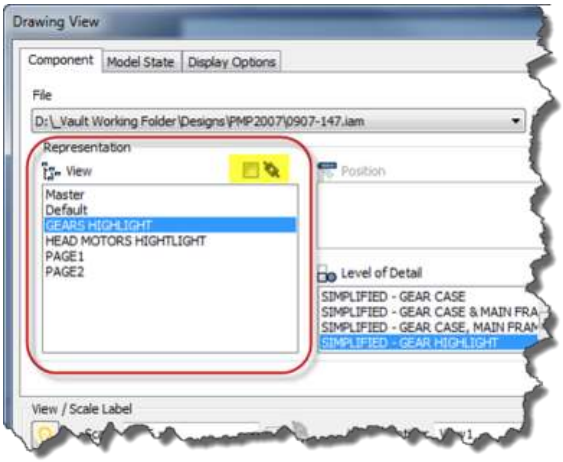
If you select **Modify Design View Representation**, Inventor updates the view rep not just for that instance within the assembly, but also in the subassembly itself. This means that anywhere that subassembly is used, you will see the changes.

Drawing Views

When creating drawing views, use View Representations to manage the visibility and transparency of the components displayed in the view.

To make most of the performance benefits and memory savings of utilizing View Representations with drawing views first close the assembly. This ensures its graphics are not loaded into memory.

Then, within the drawing, create the desired view selecting a view representation showing only the components that you want to see. Invisible components in the view representation are not loaded into memory.

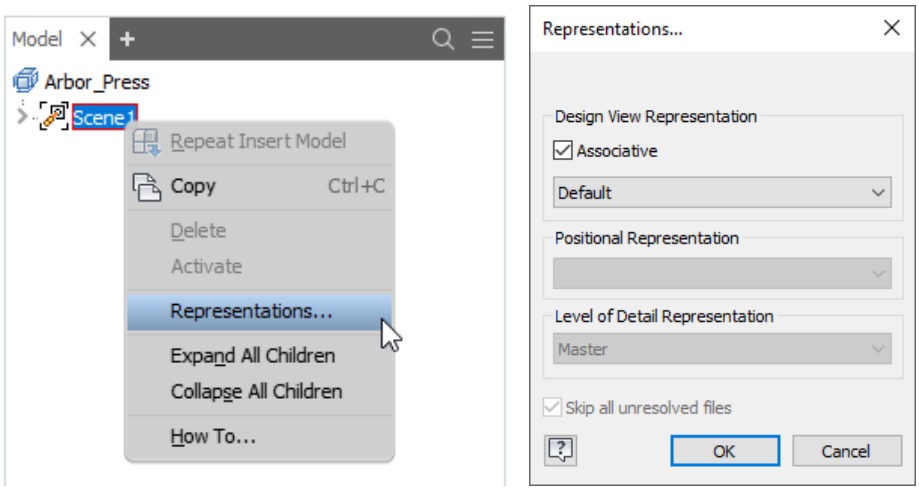


When the view rep is set to **associative** the drawing view updates as the assembly view rep is adjusted.

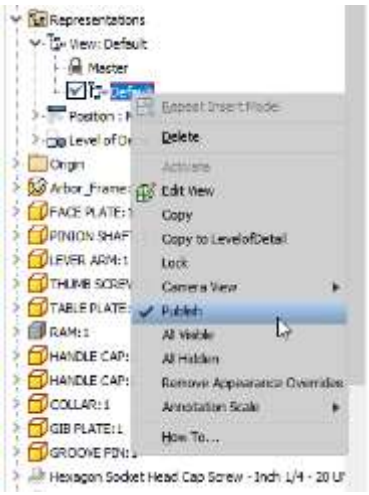
Note, that with this enabled you cannot adjust component visibility within the drawing view from within the drawing's browser.

With Presentations

Use **View Representations** to manage component visibility. Like with drawing views, use the **Associative** option so that changes to the assembly automatically reflect within the Presentation.



3D PDF Export

	To include access to the View Rep within a 3D PDF, *right-click* the **View Representation** and select **Publish**.

View Representations vs. Level of Details (LOD)

Level of Details

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 it only reads the required data from the disk.

One method of optimizing performance is Inventor loads a component into memory when it is made visible. Thus, if you have several view representations and activate each of them, all components made visible are loaded. However, there is no method to unload these components (even if made invisible) other than closing Inventor and reopening the assembly.

As 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 and reducing the number of components.

Just as with View Representations, Level of Detail Representations are created and managed within the Inventor Browser. All options are available via the right-click menu.

Moving from One to Another Type

Copying to a Level of Detail

Within the browser right-click the View Representation and select **Copy to Level of Detail** – Inventor creates a LOD matching the View Rep by suppressing (removed from memory) all invisible components.

Copying to a View Representation

With existing LODs, quickly generate a matching View Representation. Within the browser, right-click the LOD and select **Copy to View Representation** – Inventor creates a View Rep, un-suppressing suppressed components but making them invisible.

What Should I use?

Both **View** and **LOD representations** control the number of components visible, thus improving both the assembly's performance and capacity. Changing between **View Representations** does not unload or reload components. **Levels of Detail Representations** provide tools to help manage the RAM usage of Inventor.

	The selection of non-components, like *Work Features,* can limit access to suppressing components. In these situations, create a View Rep first to utilize ALL of Inventor Selection features.

The easiest way to remember the difference between the two is **View Representations** capture the *look* whereas **Level of Details** capture the components status within memory (assembly performance.

When Do I Use a Level of Detail?

- When you need to reduced memory (aka large assemblies)
- When placing a reduced complex component into a sub-assembly
- When removing IP (intellectual property)
- With Assembly Configurations

When Do I NOT User a Level of Detail?

- Switching frequently between different sets of visible components
- With Drawing Views

IMPORTANT!
 Creating drawing views within Inventor can be quite time consuming especially for larger assemblies. Improve performance by reducing the number of components within the view that Inventor must work with. It is **HIGHLY RECOMMENDED** to use **View Representations** to manage the visibility of the components within a drawing view. If multiple LODs are used in a drawing, performance can suffer.

When creating a drawing of a top-level assembly, Level of Details suppress unneeded components or replaces the collection of parts with a single part representation. As a result, Inventor does not include them when computing the drawing view.

However, only use one LOD per drawing to avoid introducing performance impact. For each drawing view using a different LOD, a copy of the assembly is loaded into memory. This increases memory usage (drastically) thus decreasing performance.

The *Autodesk Best Practice* is using **View Representations** instead of Level of Details for drawing view creation. Use the Master LOD for all drawing views.