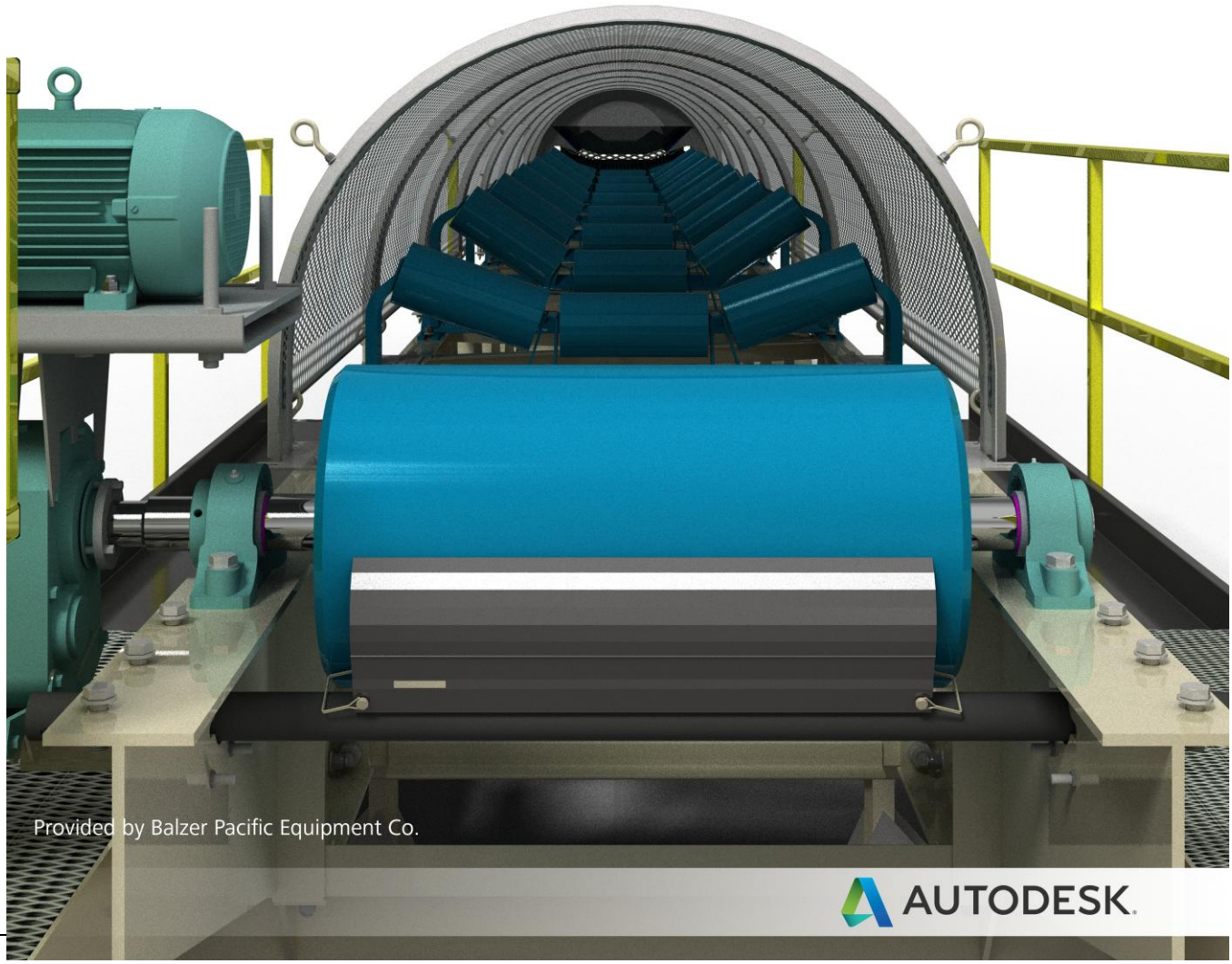
AUTODESK® **INVENTOR®** 2014

Large Assembly Performance



Large Assembly Performance

Content

1 Introduction

This manual is a reference book for working with large assembly in Inventor. You will learn the recommended techniques and workflows for managing and handling large assemblies in Autodesk Inventor. You'll learn how to implement standard processes, tips and tricks, as well as some typical workflows and techniques in Autodesk Inventor 2014 to ensure your desired productivity levels are achieved… or even surpassed.

2 Hardware and System Configuration

2.1 Recommendations

For Inventor 2014 Windows users, we recommend you using below hardware and system to work with large assembly models, with such configurations, you will feel much smoother to run the large datasets.

- 64-bit Windows 7 or Windows 8
- Dual Six core processor (12 cores total)
- 24 GB or more system RAM
- DirectX 11 level graphics card with 2 GB or more video RAM (CAD workstation-class graphics card)
- Two (or more) 1-TB hard disk drives, SATA, SAS, Ultra 160 and Ultra 320 SCSI, SSD, RAID 0 or 1, 7200 RPM or better.

While, if you can't reach above recommendations, please at least configure your system as below, this is the minimum requirement to run large datasets:

- 64-bit Windows 7 or Windows 8
- Quad or Six core Intel Xeon processor
- 12 GB or more system RAM
- DX 11 level graphics card with 2 GB or more video RAM
- At least 1 TB of hard disk space

2.2 Hardware and system detail information

Next, we will talk these hardwares in detail. While you may want to know your system information first, here, we introduce you 3 ways:

- RMB My Computer->click Properties, then view the basic information of the system, and you can click Device Manager to find more information there, eg. Click Display adapters to query the graphics cards and drivers information.
- Query from system's DirectX Diagnostic Tool, which can be launched from Windows Start, typing *dxdiag* in the Search box, and then pressing ENTER, system information and graphics cards information can be browsed from System and Display tabs of the tool.
- Query from Inventor Diagnostics, launch Inventor, open a document, then go to Application Options->Tools->Hardware->Click Diagnostics and Ok button, then you can paste the Inventor Diagnostics information to a text or word file, and watch these system and graphics information there.

Operating System	As we know, running the large assemblies requires more memory, so we recommend you use a 64-bit operating system because it doesn't have any limit of RAM, while for 32-bit operating system, it is restricted to a maximum of 4 GB of RAM.
Graphics Card	This is an important hardware for Inventor such 3D application to run large assemblies. We recommend you use DirectX11 graphics card with more video RAM, because more and more Inventor features are now implemented based on DirectX11 API. Meanwhile, we recommend you use workstation level graphics cards which own better performance compared to consumer level cards, such as NVIDIA Quadro family cards, AMD FirePro familiy cards, etc. Also you can refer to Autodesk certified graphics hardware page to search the recommened cards and drivers: http://usa.autodesk.com/adsk/servlet/cert?siteID=123112&id=16391880 Please always keep your graphics drivers updated, you can use the one from Autodesk certified graphics hardwar page, also it is recommened to download the la-

<table>
<tr><td></td><td>test one from vendor's driver downloads center, such as for NVIDIA driver downloads, you can go to http://www.nvidia.com/Download/index.aspx?lang=en-us , while for AMD Graphics Drivers and Software, you can click this link to find more http://support.amd.com/us/gpudownload/Pages/index.aspx.
If you are using laptop with mobile graphics card, you can get the graphics drivers from the laptop vendor drivers download center.</td></tr>
<tr><td>RAM</td><td>The more RAM you have installed in your machine, the better. Check the motherboard spec to see the maxium amount of RAM that can be installed. And use the fastest RAM available. The following table provides a general guideline for the recommended amount of RAM. Actual requirements vary depending on complexity of part geometry and number of occurrences.
<table>
<tr><th>Number of unique parts</th><th>Recommended RAM in GB</th></tr>
<tr><td>>5000</td><td>6</td></tr>
<tr><td>>10,000</td><td>12</td></tr>
<tr><td>>15,000</td><td>18</td></tr>
<tr><td>>20,000</td><td>24</td></tr>
</table></td></tr>
<tr><td>Hard disk</td><td>Hard drive read/write times affect the time required to open or save an assembly, while Solid-state drives can improve this performance, so if you have enough budget you can use such hard disk. If you don't, please refer to the recommended hard disk.
Inventor uses segment loading when accessing files, which means that only the required parts of the file are loaded in memory. The rest remains on the hard drive. Consequently, when additional segments from the file are required, it is beneficial if they can be read as fast as possible. You can follow our suggestions to improve the hard disk performance, please refer to the help topic "Hard disk drive configuration" in wiki help http://wikihelp.autodesk.com/Inventor/enu/2014/Help/1283-Inventor1283/2454-Assembli2454/3172-Large_As3172/3180-Inventor3180.</td></tr>
<tr><td>Dual processors</td><td>You can gain some benefit from using multiple processors in Inventor. However, a faster, single processor can be more desirable. By design, Inventor is not a multi threaded application. The processing load cannot ordinarily be balanced over multiple processors. However, some specific functionality in Inventor does support multi-core technology.
The wisest choice is to procure the fastest single CPU that your budget allows. If your budget allows, buy the two fastest dual CPUs.</td></tr>
<tr><td>Others</td><td>Such as network impact to Inventor performance, etc. You can get more information from wiki help "Inventor optimalization for large assemblies" section.</td></tr>
</table>

3 Inventor Optimization

Inventor have some settings which can be adjusted to improve performance, let's take a look at which of them are benefical to you.

3.1 Project Settings

Project Files	Inventor->Get Started->Projects	Project files organize Inventor data, and determine the location of the working data, templates, styles, and libraries. Below are some suggestions for you: • Set the Included File path option to point to a single, read-only project file maintained on the network by the CAD administrator. • Workspace should never be on a network location. It is intended to be local on the users' machine. All work should be performed on files held locally and when finished copied back to the network. • The fewer workgroup search paths defined, the better. • An excessive number of mapped network drives will slow down the machines ability to open and save files as Windows attempts to resolve these every time. • Keep relative paths = true. Relative in this instance means relative to the location of the project file. • Keep the Projects directory structure as flat as possible and Windows Explorer Directories structured as needed. Avoid folders containing more than 800 files. Top level assemblies should appear in a folder with the project file (*.ipj) and all the parts\subassemblies it contains should be located in a subfolder. • For portability, define all storage locations as subfolders of a directory, which only contains the project definition file (e.g. project.ipj). For more information, please refer to Tools and Methodologies section of "Inventor optimization for large assemblies".

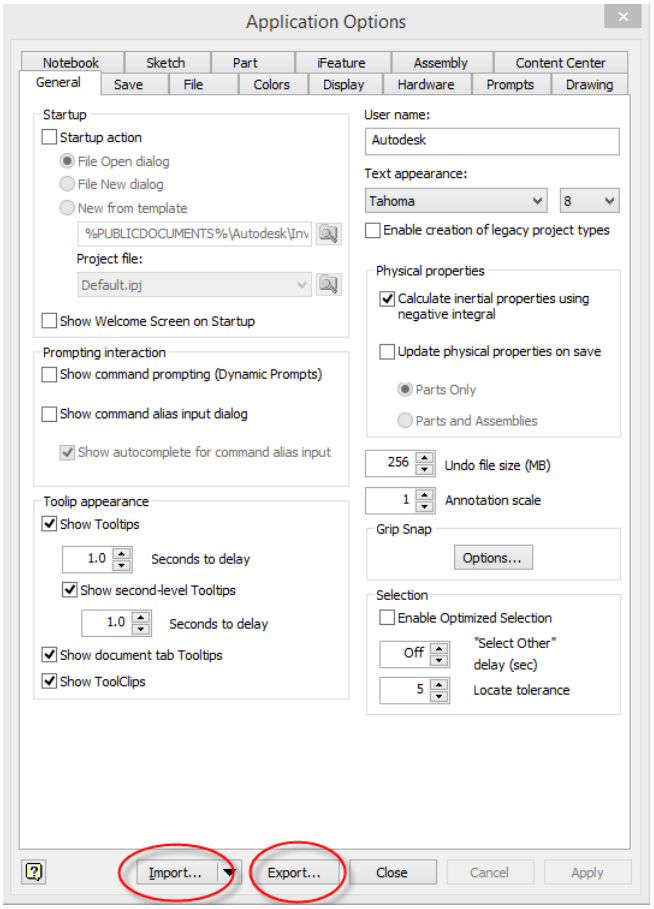
3.2 Application Options

Item	Access from Application Options or Other Place	Recommendation	Reason
Show command prompting	General	Off	When this setting is enabled, it will take additional time to display the command prompts in a tooltip near the cursor.
Enable Optimized Selection	General	On	It can improve graphics performance while pre-highlighting in a large assembly. When selected, the Select Other algorithm initially ranks only the objects closest to the screen. If you click through this first group of objects, it ranks the next group.
Undo File Size (MB)	General	>=1000MB	This setting is to set the size of the temporary file that tracks changes to a model or drawing so that you can undo actions. For large or

			complex models and drawings, increase the size of this file to provide adequate Undo capacity. Enter the size, in megabytes, or click the up or down arrow to select the size.
File Open Options->Assembly-> Design View Rep	File	Last Active & Associative	Opens the File Open Options dialog box. Set the representation that is loaded by default, unless specified by the Options command from the Open dialog box. In Assembly tab, suggest setting Last Active for Design View Rep, and selecting the checkbox of Associative.
Defer update	Assembly	On	This setting is to set preference for updating assemblies when you edit components. Select the box to defer updates of an assembly until you click Update for the assembly file. Clear the check box to update an assembly automatically after you edit a component. In large assembly, the setting is always selected, and you could update the whole assembly when needed manually
Enable Prehighlight	Colors	Off	It shows prehighlighting when the cursor moves over an object. Activated by default. And it takes additional memory to have such option on.
Display Quality	Display	Rough	After changing to Rough setting, the display quality of the datasets is not as good as before, but the performance will improved a little.
Textures On	Display->Settings	Off	If the model has used several appearances or materials with textures, you could turn off this setting to increase the view interactive performance. And the model will show without textures. You can also unselect the Textures On check box from View tab of Ribbon UI to avoid the change to other models with the setting from Application Options.
View transition time	Display	0	This setting controls the time required to transition smoothly between views when using viewing commands (such as Isometric View, Zoom All, Zoom Area, View Face, and so on). Zero transition time causes transition to be abrupt, which can make it difficult to understand changes in position and orientation. But will make you feel better performance for viewing commands.
Minimum frame rate (Hz)	Display	10	With complex models (such as large assemblies with a high component count), specify how fast to update the display during interactive viewing operations (like Rotate, Pan, and Zoom). To maintain a high frame rate, the system sometimes simplifies or discards parts of the view. The view restores when movement ends.

			• **Set 0** Draw everything in the view, regardless of the time required. • **Set 10** Draw at least ten frames per second. • **Set 20** Draw at least twenty frames per second. **Tips:** Increase the frame rate if the model updates too slowly during a viewing operation.
Performance	Hardware	On	This is Inventor's default setting, it favors system performance over graphics quality. it turns off Anti-aliasing for machines running Windows 7 and 8.
Retrieve all model dimensions on view placement	Drawing	Off	Be default it is on, it will take more time and memory to get the dimensions during view placement, so you can clear it if you don't want these dimensions but expect better performance during view creation.
Display line weights	Drawing	Off	This setting enables the display of unique line weights in drawings with more memory consumption. If clear the checkbox, it will show lines without weight differences but a little better performance.
Show preview as	Drawing	Bounding Box	This sets preference for preview images. The default is All Components. Click the arrow to select Partial or Bounding Box. While Bounding Box option takes the least memory consumption. The preview has no effect on the resulting drawing view.
Section View Placement as Uncut	Drawing	On	This setting controls section preview with or without cutting components, and with cutting components takes more time and memory to compute the preview. So recommend use uncut. The preview has no effect on the resulting drawing view.
Enable background updates	Drawing	On	This setting allows you to switch the display of raster drawing views on or off. Raster views increase your productivity when you work with drawings created for large assemblies. You can review a drawing or create drawing annotations before precise calculation of drawing views finishes. Precise drawing views are calculated in the background while you work with raster views.
Use Bitmap	Tools->Document Settings->Drawing->Shaded Views	Always	This setting is only valid for Shaded drawing views, set it to Always, it will use bitmap to generate the shades, which gives better performance compared to Offline only which needs compute the shade time by time during viewing commands.

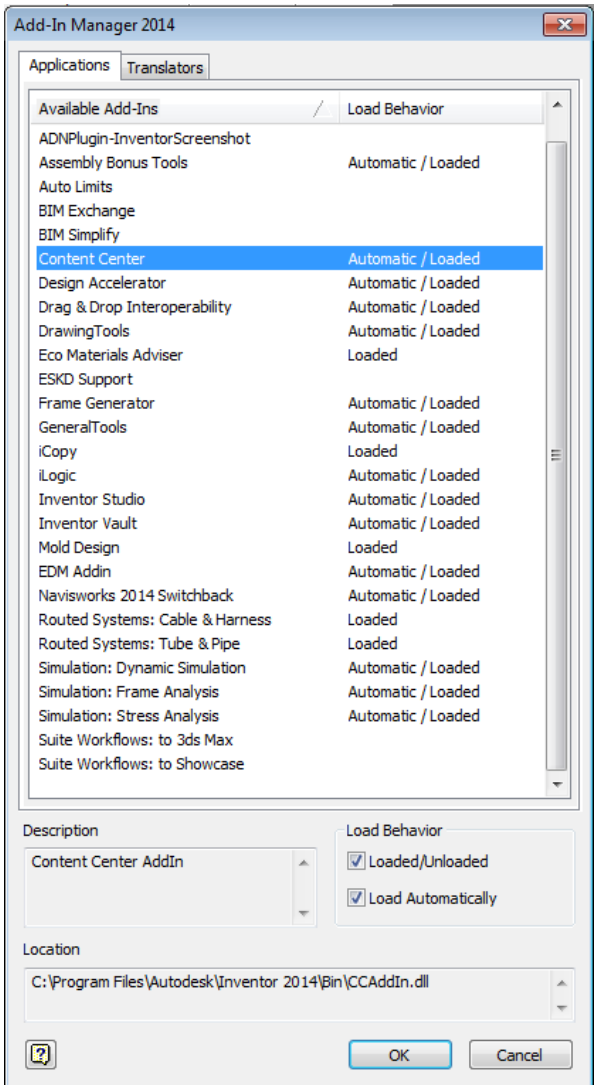
Tips: To be convient, we recommend you export these Application Options settings into an xml file, and save it in local, next time, you can directly import this xml file into Inventor to have the customized Application Options instead specify one by one. This can be done by the Export and Import button under Application Options dialog.



3.3 Others

3.3.1 Unload unneccessary add-in to speed Inventor

Use the Add-In Manager to specify which add-ins to load at startup. You can also unload certain add-ins while in session. The available add-ins and their load behavior is listed in the dialog box. Add-ins not loaded at startup is listed without a load behavior. When an add-in is unloaded, its functionality is not available. However, all translator add-ins (such as DXF, DWF, and DWG) automatically load when that format is selected when doing a Save Copy As. By default, the translators are not loaded at startup. You cannot unload or load an Autodesk Inventor Professional add-in in session. Restart the application for your selection to take effect.



You may want to consider unloading to:

1. Reduce the time it takes Autodesk Inventor or Autodesk Inventor Professional to load initially.
2. Reduce the memory footprint for the application and increase the capacity. An increased capacity enables you to work with larger assemblies. If you experience capacity issues while using Autodesk Inventor Professional, unload some of the larger add-ins (such as Tube & Pipe, Cable & Harness) if they are not used in your design. In most cases, it is recommended that you do not unload Autodesk add-ins.
3. Correct problems with the application. If you experience problems with Autodesk Inventor, start to unload non-Autodesk add-ins systematically to see if unloading corrects the problems. It is possible for some non-Autodesk Inventor add-ins to be invalid for a current release, and unloading them may correct problems.

3.3.2 Organize Design Files in a Folder Structure

Avoid storing all assembly files in one folder. When Autodesk Inventor must search through hundreds or thousands of files in a single folder to find the correct file, it can take longer to open files. As part of the file name, Autodesk Inventor stores the path to the folder where a file is located. When there are fewer files in a folder, the file is located and opened more quickly.

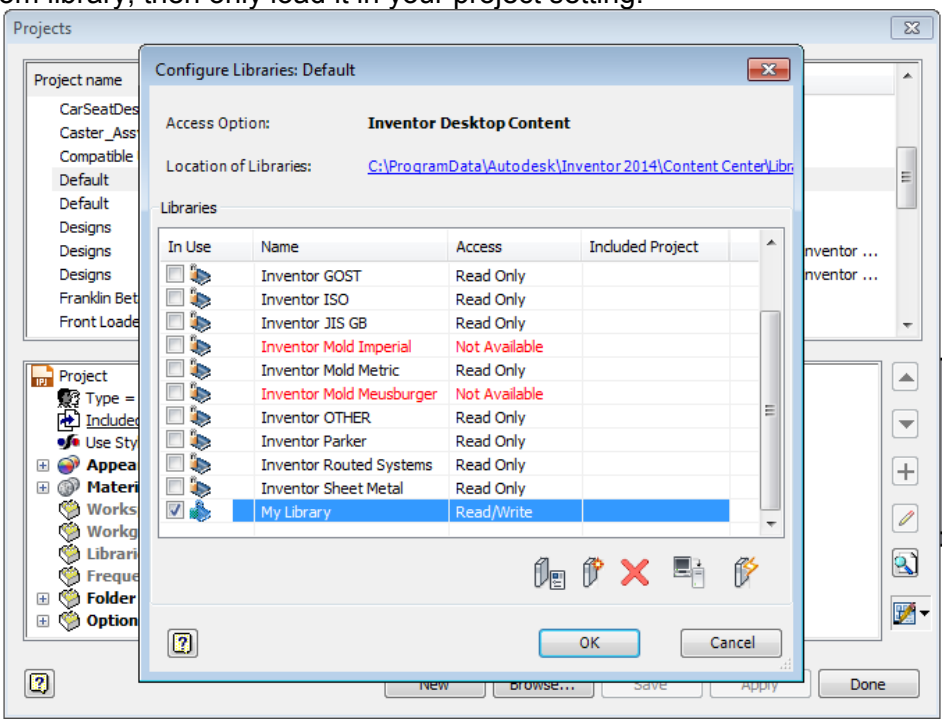
3.3.3 Plan for Assembly Hierarchy Efficiency

Plan the top-level assembly and subassembly structure before you create parts. Create logical subassemblies and combine them into larger assemblies. Keep all components used in a subassembly in the same directory. Create a shared network directory for components that will be shared by many designers on many projects. Assign the Summary and Project properties for individual components. Create a unique template and use it to create components for a specific project or subassembly. Predefine common properties in the template so all components created from that template inherit the properties. Search for attributes both inside and outside Autodesk Inventor to find needed component files. Save and name attribute searches that you are likely to use again.

3.3.4 Create custom Content Center Library

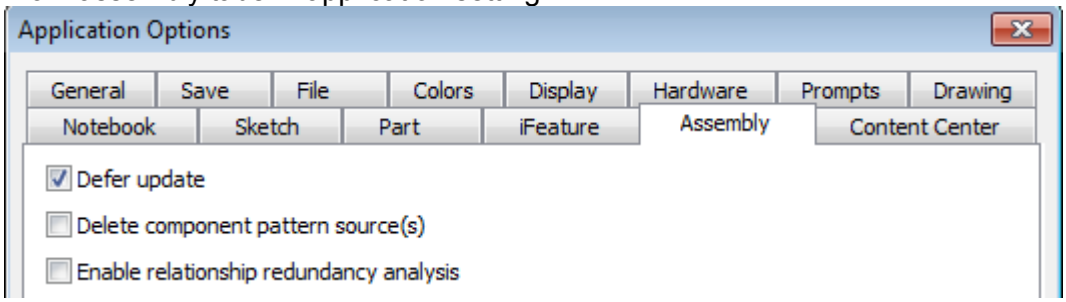
Inventor Content Center database contains over 750,000 parts and covers 18 international standards. Considering your design requirement, only load neccessary library will boost your speed to load Content Center and place from it.

If you only use small portion of content of the library, you can also consider to copy your required families to the custom library, then only load it in your project setting.



3.3.5 Defer update

By default, assemblies automatically update when changes are made. As assemblies become larger, this update requires more computational time. If it is not important to monitor updates simulataneously, it is recommended that you defer automatica updates to speed up the edits. you can select the checkbox to defer update from assembly tabe in application setting.



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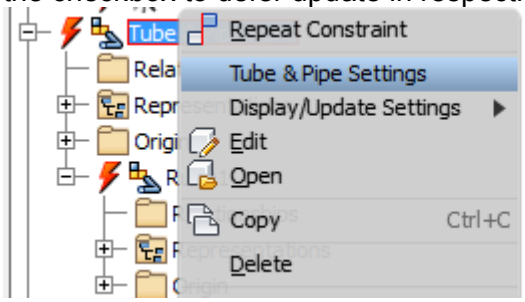
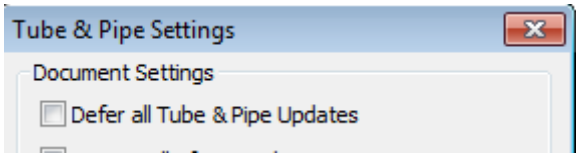
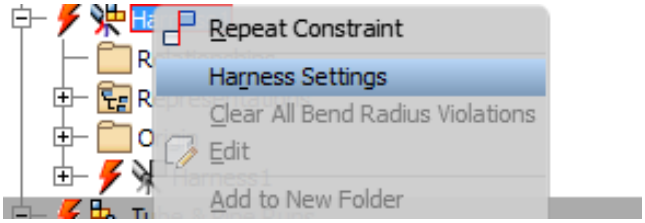
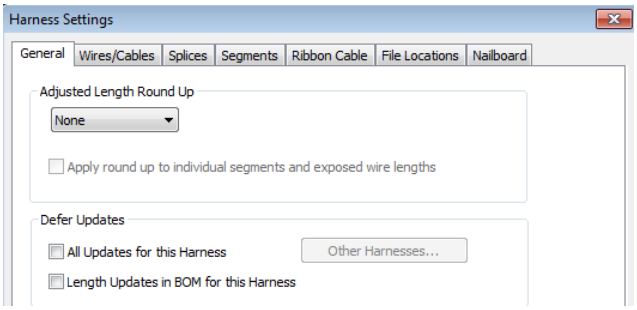
If there is any update, the yellow update flashlight icon is on. You can update it later in idle time by task scheduler or to click the icon.

For tube and pipe, cable and harness, they are more expensive in computer memory, and you can select the checkbox to defer update in respective setting dialog by RMB click on the top node.









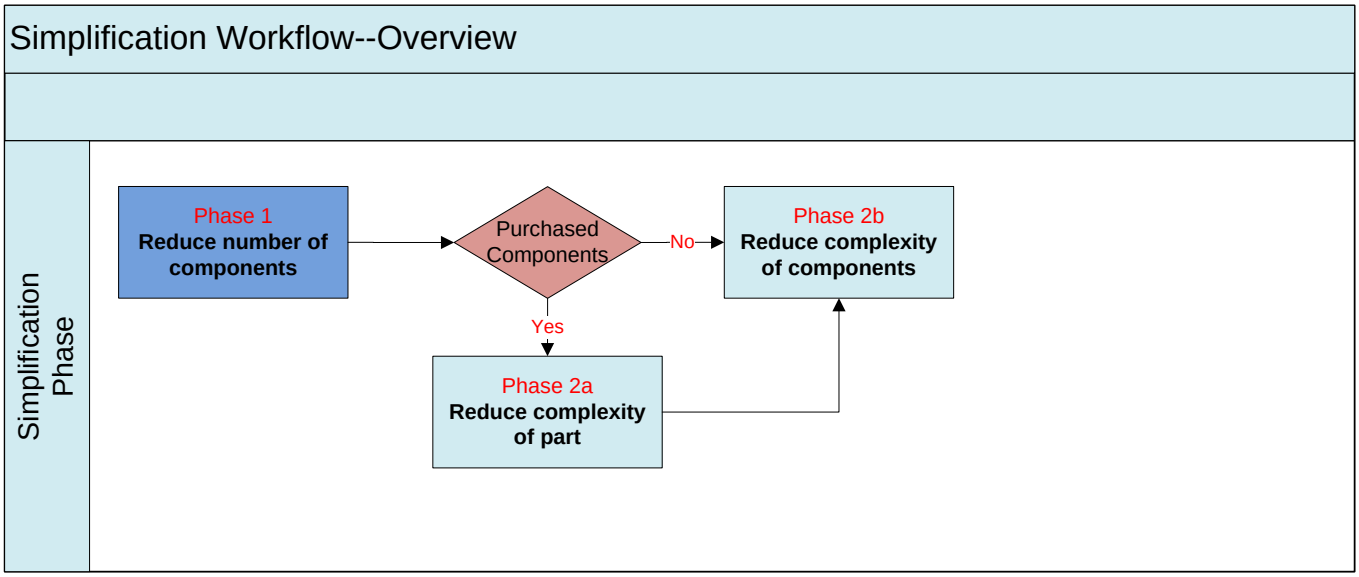
Not like generic assembly, Deferred items of T&P and C&H are indicated with the Update Deferred icon in the browser. If items in a harness part have updates deferred, the items are not shown in the graphics window. Noticed These items will not be updated until you uncheck the defer setting.

A word of caution: use this approach only as a last option, as one can forget to reset it to the normal 'un-checked' condition.

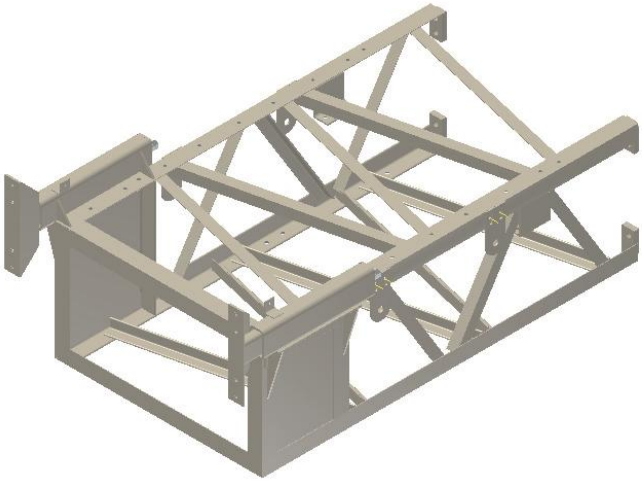
4 Large Assembly Methodology

4.1 Simplification Workflow

Simplification is a key point to reduce the number and complexity of parts in large assemblies. Especially purchased components and company standard components should be modeled as simple as possible and as detailed as necessary. Because these types of components are normally reused often but there is no need to edit them – so keep them simple!
This simplification work pays off every time you insert such a component. Before you start simplifying components we recommend making a copy and using this copy to generate the simplified part.



4.1.1 Phase 1 – Reduce number of components

1) In these steps, we mainly consider selection tools to reduce the number of components. Open assembly " 57581.iam " 2) Activate or create Level Of Detail (LOD) "Simplified"	

3) Reduce number of components:

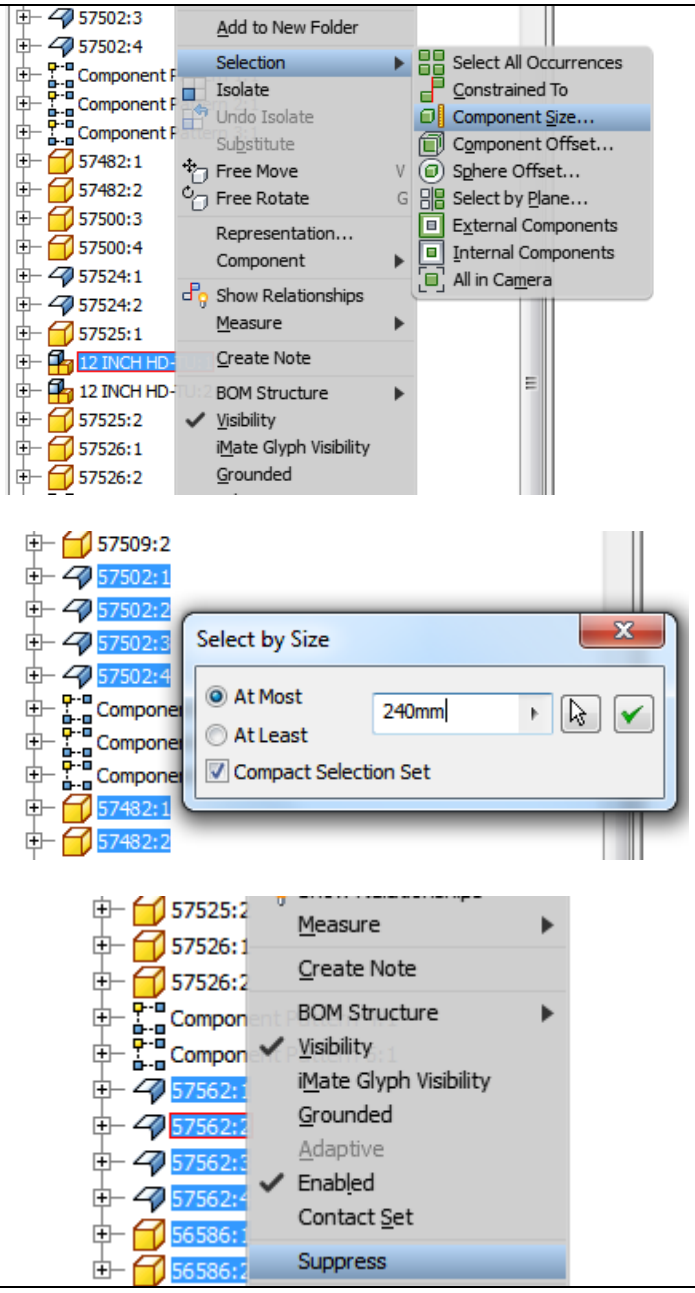
Now select all the parts that you don't need to show in the simplified representation and suppress them.

Especially fasteners and hidden details.

It's very helpful, to use the Component selection tools to do that.

e.g. Internal components, Components smaller than …

This step reduces the number of components, that show up in the derived component dialog later.



4.1.2 Phase 2a – Reduce complexity for part

The only difference is that there is a phase 2a to handle manufactured part or purchased components. Since we don't need the detail of these components, we can ignore these part features that have no effects on others, and only care its boundary, so we can suppress these unnecessary features and reduce complexity. There are two cases for introduction –

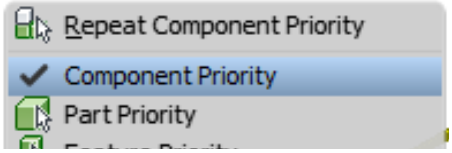
4.1.2.1 Manually suppressing unneccessary features

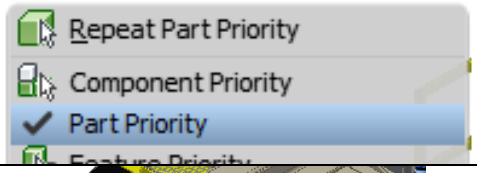
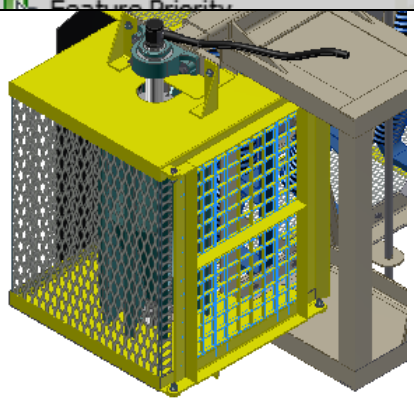
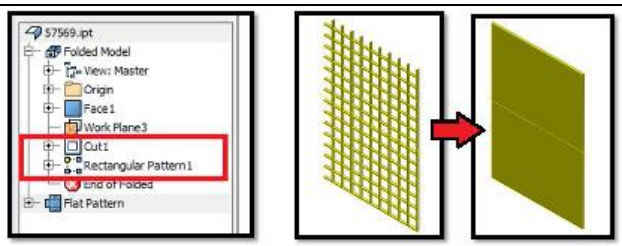
1) Launch Autodesk Inventor 2014 if it is not already open.

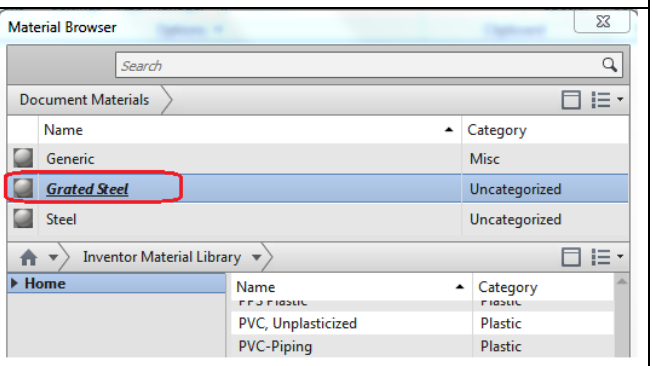
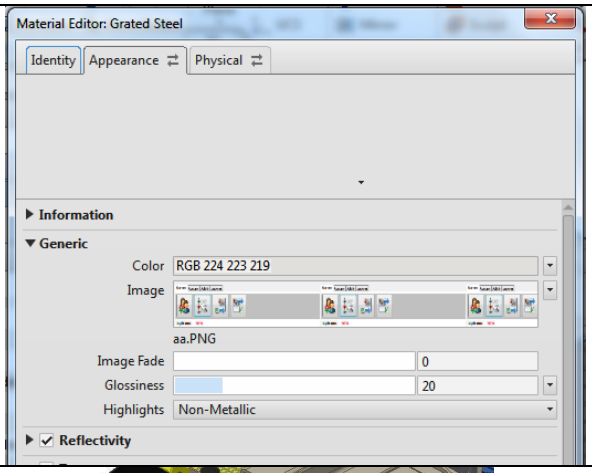
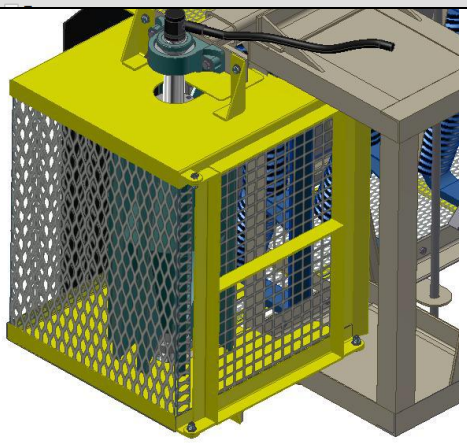
2) Ensure that the project is set to "19525100" by clicking on the "Project" icon in the Ribbon.

3) Click "Done" if needed.

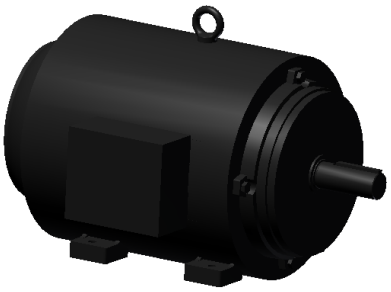
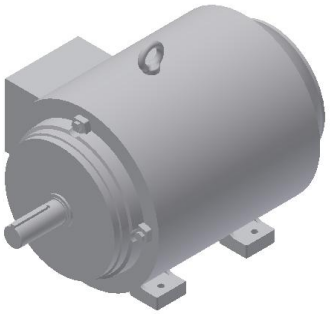
4) Click "Open" and browse for "19525100.iam" then click "Open".



5) In the modeling area, hold “Shift” and right mouse click to bring up the “Selection Filters” and change from “Component Priority” to “Part Priority”	
6) Rotate the model so you are looking at the bottom side. Use both “Shift” and middle mouse wheel to rotate or select “Bottom” on the view cube. 7) At the tail end of the conveyor is a pulley guard (see image below). On the bottom of the pulley guard is grated steel (part 57569.ipt). Open this part in a new window.	
(HINT) You can use the “Find” icon and search for the specific things in an assembly. They will be highlighted in the model tree and the graphics window. This can save time when dealing with large assemblies. The user modeled this with two features and a pattern. Even though the pattern might reflect how the part will look, this creates more work for Inventor. To put a value to this, the part as is requires 786KB of disk space. With some modification to the part you will bring this file size down to 105KB. *The intent is to keep the flat pattern. Select the features shown below in the red box and delete them.* *This will reduce the file size to 96KB. At this point, the file is now a solid. For accurate information, especially center of gravity, this is not acceptable. What you need to do is change the physical properties of the part to visually look the same, and accurately reflect the weight. You will need to come up with a density that will be used to calculate a true weight. Here is how I did it.*	
From the original part iProperties I found the weight to be 16.95 lbmass and the volume to be 59.68 in^3. After deleting the features the volume is now 352.58 in^3. Since I want the weight to remain the same, take the old weight and divide it by the new volume to get a corrected value to use as the density. $Corrected\ Density = {}^{16.95\ lbmass}/_{352.58\ in^3} = 0.048\ {}^{lbmass}/_{in^3}$ *Now you need to modify the style of the part to reflect the corrected density and visually look the same.* *(HINT) You could use the “Find” icon and search for the specific values in an assembly like part number, description etc. After searching, they will be highlighted in the model tree and the graphics*	

window. This can save time when dealing with large assemblies.	
8) In the Ribbon, select the “Material” under the “Tools” tab. Since a form of Steel is assigned to the part, you can use this as the base. Right-click “Steel” to duplicate new material. Name this new material “Grated Steel”. Assign the new material “Grated Steel” to the part.	
9) Now you will need to change the part to look like the grated steel. Edit the material “Grated Steel”, change to appearance tab, add the image below to Generic section. 	
10) Now you should be looking at the assembly and the Grated Steel should be visually and physically acceptable. An alternative method is to just override the weight.	
(HINT) Now that you have gone through the trouble of changing the part, you could use this as a Template for creating grated steel models in the future. Create a folder in your Templates Directory named “Grated Steel” and add a copy of this part to the folder. When you hit the “New Document” icon you will now see a folder with your grated steel template. If your templates are in a network location then everyone in the company will be able to use this. *(HINT) Now that the new part is solid, this means that drawing views will not show the objects behind the plate in non-hidden line views. Use Break Out to view files that are hidden behind solid objects.*	

4.1.2.2 BIM simplification for the 3rd party part

1) Open part TEFC NEMA, 215T.ipt, which is a purchased component with ~9M file size.	
2) Go to BIM environment, and use Remove Details, Fill Voids and Define Envelopes comands to simplify complexity.	
3) Export it to SAT file, and import it to Inventor, then save it, and the filesize is ~1.3M	

4.1.3 Phase 2b – Reduce complexity of components using VR and LOD

For large assembly, at the upper level, there are some components that are not so important or so small that can be neglect on a print.

View Representation and Level of Detail are two main tools to be used to reduce number of components.

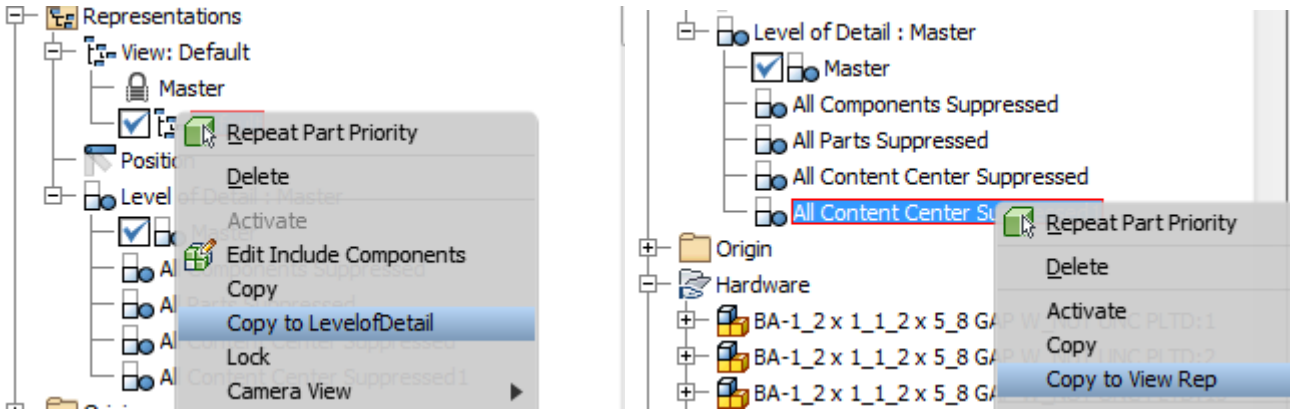
When considering LOD and VR for simplification, the LOD like “Simplify” could be defined in assembly template.

- **View Representation**

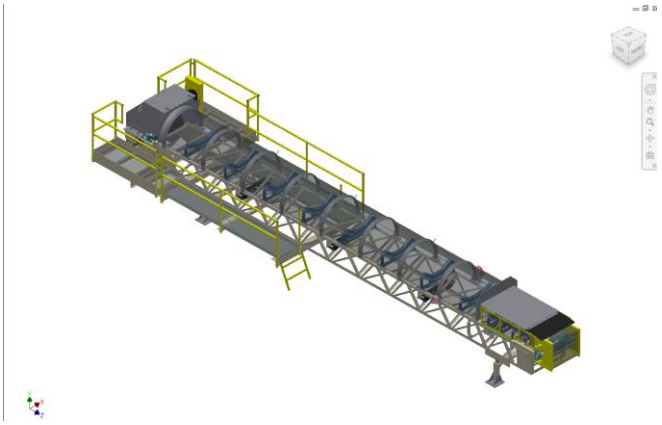
View Representations (View Reps), formerly known as Design Views, are a good place to start controlling the view of the displayed objects. View Representations improve performance by reducing the load on the Graphical Processing Unit (GPU) while data is held in memory.

- **Level of Detail (LOD)**

The LOD’s an assembly tool to organize simplification levels. The information if a component is suppressed (unloaded) or unsuppressed (loaded) is stored in the LOD’s. Comparing to View Reps, suppressed component will be unloaded, it will reduce memory consumption in the assembly and drawings, and improve performance much better than View Reps.
It’s quite easy to copy View Reps to LOD and vice versa.

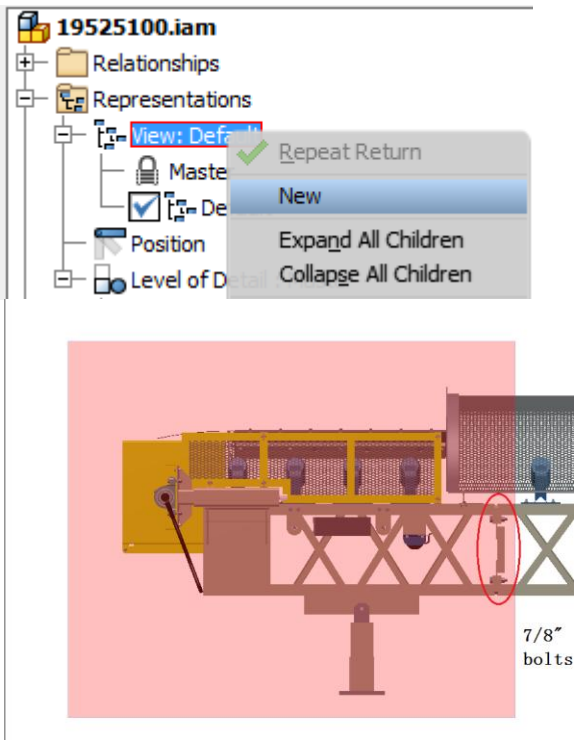


In below case, we will use a sample data to demostrate the recommended workflow. First we load the assembly 19525100.iam.



4.1.3.1 View Representation

1) Change the selection filter back to “Component Priority” by holding “Shift” and right clicking in a blank space in the model window.
2) Select “Right” on the view cube.
3) To create a View Representation, right click on View and select “New”.
4) Name the new view “Tail Section – Purchased Components”. To change the name, perform two slow clicks on the new View.
5) Left click and hold in the upper left area above the conveyor.
6) While holding the left mouse button, drag to the lower right stopping just past the 7/8” bolts that connect the two lattice sections together.
7) Once the selection is made, right click and select “Isolate”. Hit “Save” and save all if prompted.
8) Now select the guards (in yellow) and the hopper (in grey) and turn the visibility of those off. Don’t forget the Black flap on the top.
9) Now select the support leg and pins. Turn the visibility off also.
10) Create a folder to place all those purchased



components, here we create a folder named "Hardware"

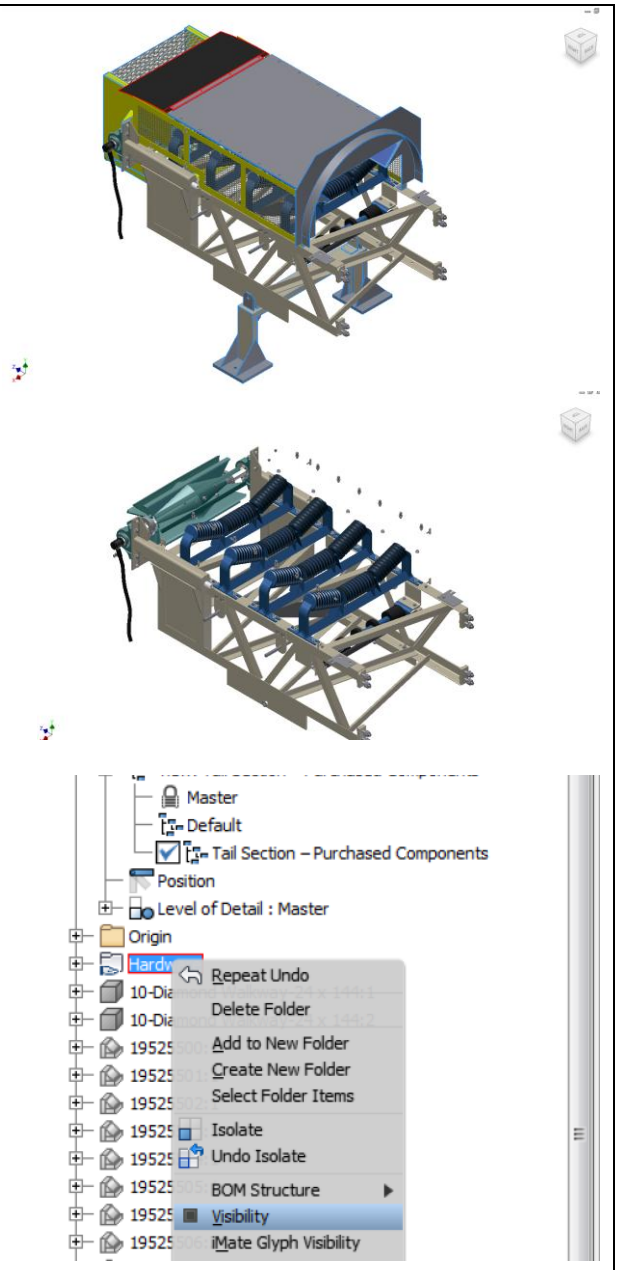
11) Save the assembly and keep open.
12) Right click on the "Hardware" folder and turn the visibility of all hardware off.

13) Once the information shown on the screen is correct, hit the F6 key, right click on View Representation "Tail Section – Purchased Components" and select "Lock". This will keep new additions to the assembly from showing up in this View Representation.

14) Save the assembly and keep open.

Tips: *It is very convenient to create Representation when components are grouped with folders.*

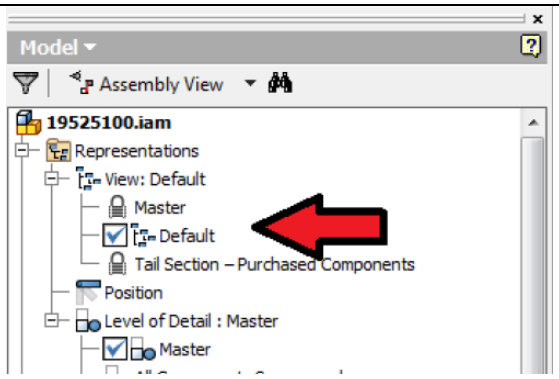
- *(HINT) To put components into a folder, either select them in the model tree and choose "Add to new folder" from the right click options or right click in a white spot on the model tree and choose "Create new folder" then drag and drop components into this new folder.*
- *(NOTE) Creating component patterns with items in a folder will remove them from the folder.*
- *(NOTE) You will not be able to take advantage of "Alpha Sort" once you create a folder.*



4.1.3.2 Level of Detail

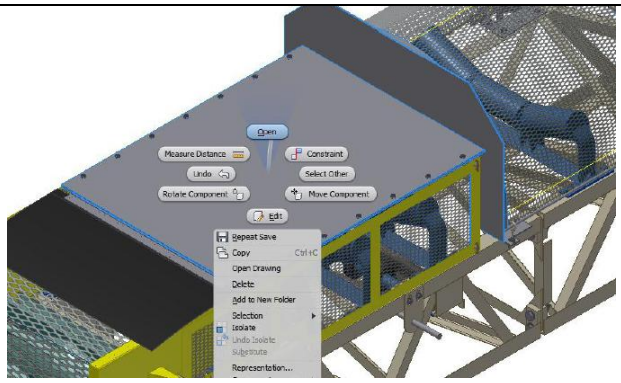
1) Make sure the View Representation is set to Default or Master.
2) Right click on "Level of Detail:Master" and choose "New Level of Detail"
3) Name the new LOD "Large Assembly" and save the model.
4) At the tail end of the conveyor, right click on the hopper and select "Open"
5) Switch to the "View" tab above the Ribbon, select "Object Visibility" and remove the check next to "All Work features".

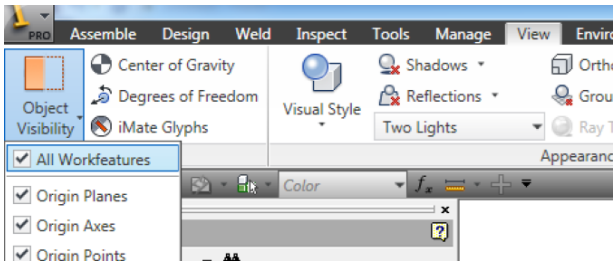
Tips: *(HINT) Work features take up resources. I find them to be very important when designing and constraining but are not needed when the*

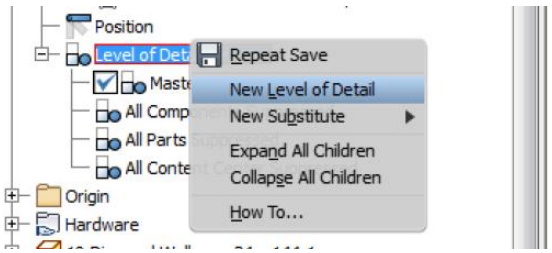


model is complete. If you do not need to have them visible, turn them off to save system resources.

(HINT) When dealing with large assemblies it is important to remove components from memory that are not necessary in the overall layout.

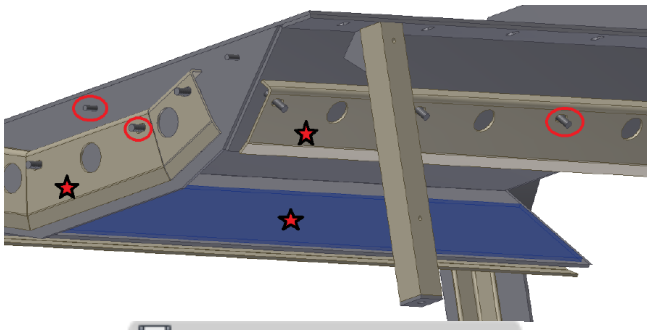




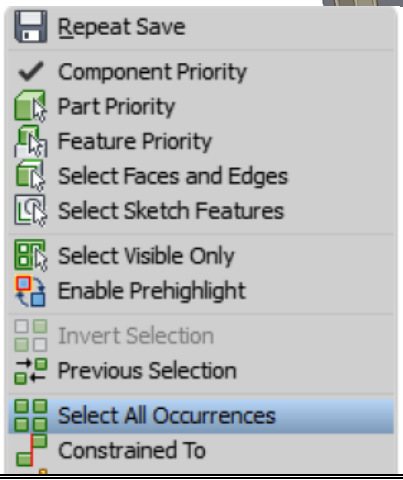


6) Now that 57584.iam is open in a new window, right click on “Level of Detail:Master” and choose “New Level of Detail” and name it “Large Assembly”. To change the name, perform two slow clicks on the new LOD.

7) Suppress all components that are not important at the upper level when modeling. Usually, we try to keep enough of the outside so it is recognized on a print and leave critical parts that defin an inlet, discharge or mounting points. Here, all items marked can be suppressed.



Tips: Selection Tool
(HINT) Using the selection filters is a good way to select multiple parts that are the same, not near each other and do not belong in a component pattern. In the window, hold “Shift” and right click and choose “Select All Occurrences”.



8) Save and close the sub assembly to return to main assembly.
9) Now let’s look at the tail end again. Right click on the tail section lattice frame (57581.iam) and open in a new window.
10) Create a new LOD called “Large Assembly”.

Feel free to suppress components in the assembly that you believe will have little effect on the way the upper level assembly will look on a drawing. I will have some suggestions shown below.

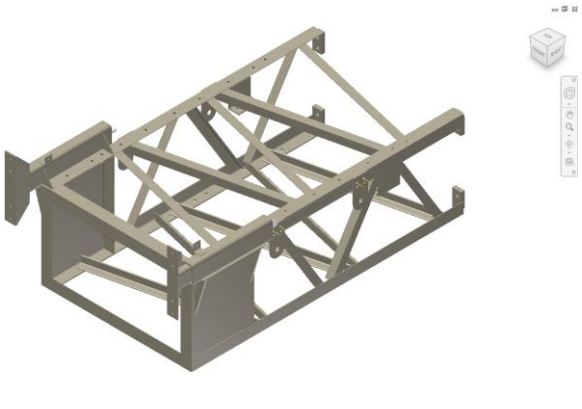
11) Use your selection filter and set to “Part Priority”.

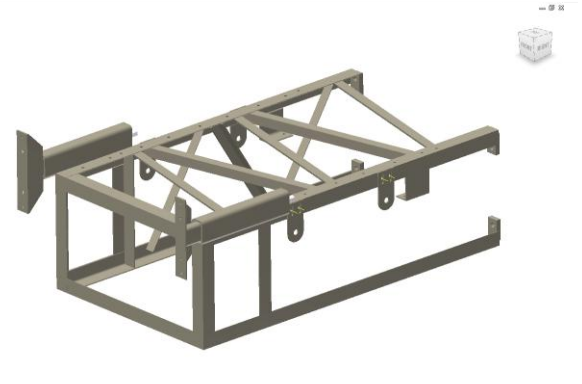
Tips:
The goal here is to keep enough information for you to work at connection points, inlet and discharge areas. Most important though, you want this to look good on a drawing or be able to display your intent of the design with as few lines as possible. Just like parts consume resources and effect performance, lines on drawings do the same thing.

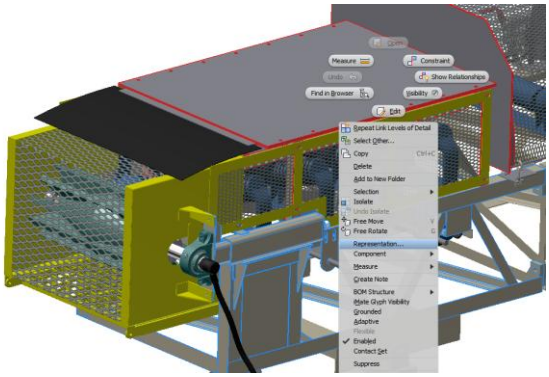
12) In this case, we removed gussets, the two side plates and the cross angles on one side and the bottom. This will still give the drawing the same appearance and reduce the amount of detail the system needs to manage.
13) Save and close to return to the main assembly.
14) If you switched to Part Priority, switch back to Component Priority.
15) With the main assembly open, hold control then select the hopper and lattice frame that you created the Large Assembly LOD.
16) Right click and select the “Level of Detail Representation” dropdown. In there you will find both of the LOD’s created for the two sub-assemblies. Select “Large Assembly” and hit OK and then save the model.

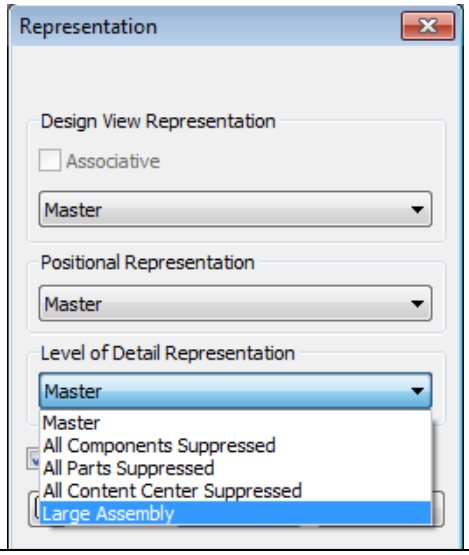
- *(Hint)it is more efficient to create Representations right before taking a large assembly to the drawing stage. Last minute changes to the design can cost you time to go through the assemblies and update the LOD’s.*

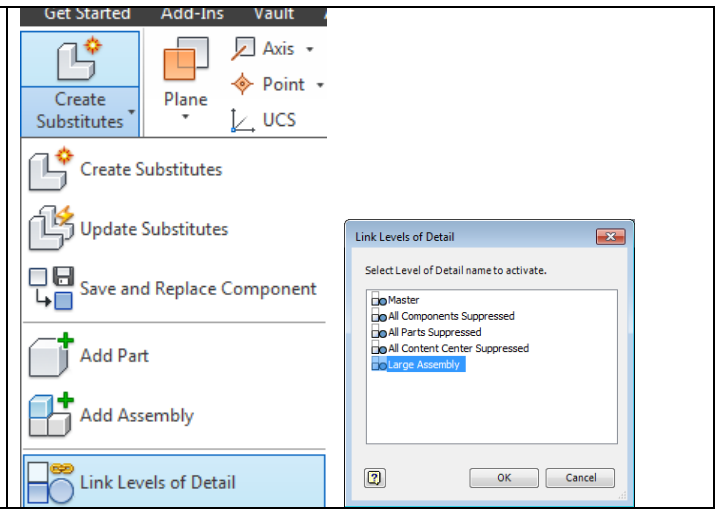
- **Tips***: If there are many sub assemblies have defined LOD with same name (i.e, Large Assembly in this case). An alternative solution is to Link Level of details from productivity tool in Assemble Tab.*









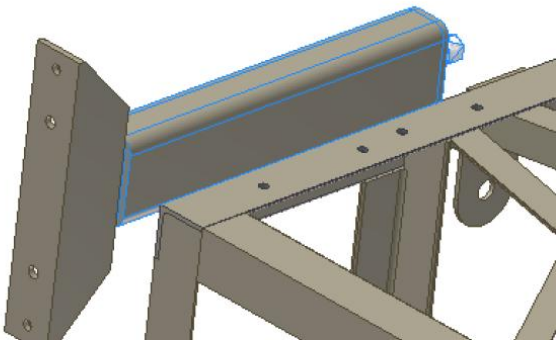


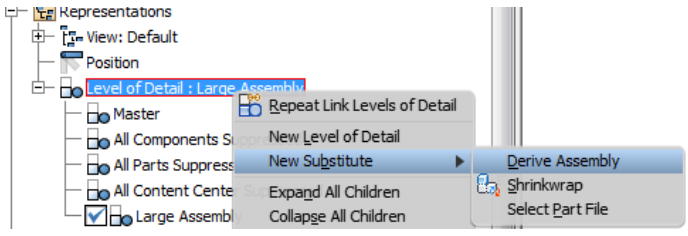
In Level of Detail Representation, beside the way to suppress unneed components and define a representation to manage it, another way is substittute. Substitute LOD's provide a comprehensive set of tools to create a lightweight or simplified part. Use them to protect intellectual property, significantly reduce memory consumption on upstream assemblies, or to exchange a part file "assembly skin" with others.
All substitute representations preserve the original mass properties and parts list.

Now we will create LOD's using the Substitute "Derive Assembly" and "Shrinkwrap". The example you will work on will expose you to their differences. Both methods have their place but it is up to you to decide which method works the best on what information is needed for the shop to fabricate and assemble your designs.

4.1.3.3 Create Level of Detail—Derive Assembly

1) Continue to use assembly 19525100.iam.
2) First you will look at the "Derive Assembly" Substitute. A derived part is a new single part with a reference to an assembly or part. This method will copy bodies, sketches, work features and parameters. By default they will be associative.
3) Open the Tail End Lattice Section (57581.iam). Open the Take-Up body (12 INCH HD-TU.iam).
4) The dialog gives you the option to change the file name, choose a template and where to save the file. For this, hit the dropdown for Templates and choose "Standard-no sketch.ipt" and hit OK.
5) The next dialog will control the amount of detail you want to show in the model. Make your window look like mine. My preference of Derive style is to choose the second option for "Solid body keep seems between planar faces". This gives a better representation of the model and will not merge faces together but will not give the best performance since more lines are being drawn on the model. This



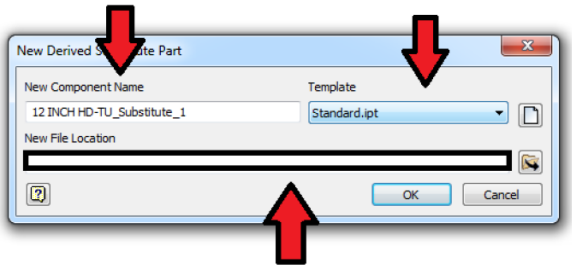


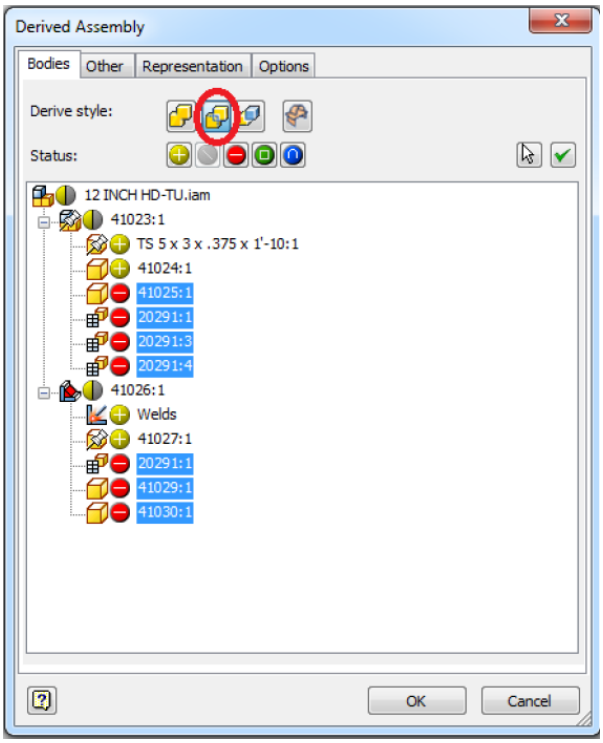
option also provides better control on what you constrain to the faces and displays better in the drawing. Once you have highlighted the parts below, choose the Red button to not have them show up in the derived part.

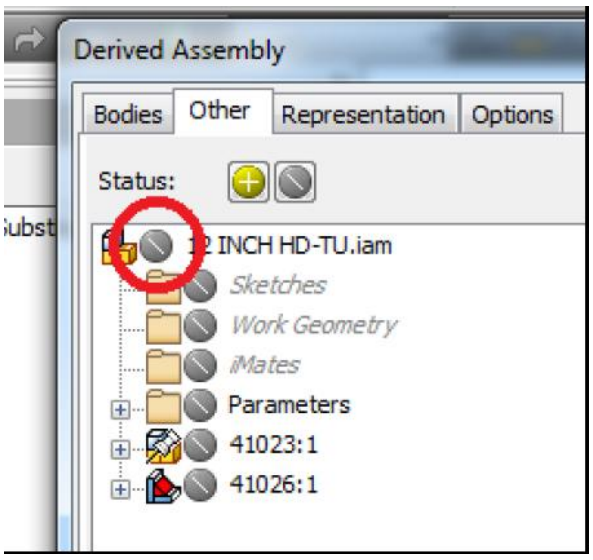
(HINT) using the "Single composite feature" will produce a derived part with the smallest file size.

6) Select "Other" tab to remove iMates, sketches, parameters and work features. Click the Circle next to the assembly and change the status to "Exclude the Selected Objects" for all components.
7) The Representation tab will allow you to derive the assembly based on an existing Design View, Positional or Level of Detail Representation.
8) get into the Options tab later. For now hit OK with the default settings.
9) Save and close the Take-Up assembly.
10) Select both Take-Up assemblies right click then choose "Representation".
11) Select the Substitute LOD you just created then hit OK, save the model and close the tail end lattice section returning to the main assembly.
12) The coolest part of this is that LOD's can be used at any level in the assembly structure and carry to the upper level assemblies to simplify the model and reduce the demand on the system. This is where I have seen people (including myself) believe that a derived substitute without removing parts or features is better. Creating a derived assembly will be better on visual performance (i.e. pan\zoom\rotate) but will create a file that requires the same amount of RAM to have open. For systems with RAM limitations this will be something to pay attention to. The key is to remove features, especially curved features as these tend to demand more from Inventor's shape modeler. Using the Derive Assembly gives you the most flexibility on managing your data.

(HINT) Use this method to create simplified models for FEA and Dynamic Simulation.







4.1.3.4 Create Level of Detail – Shrinkwrap

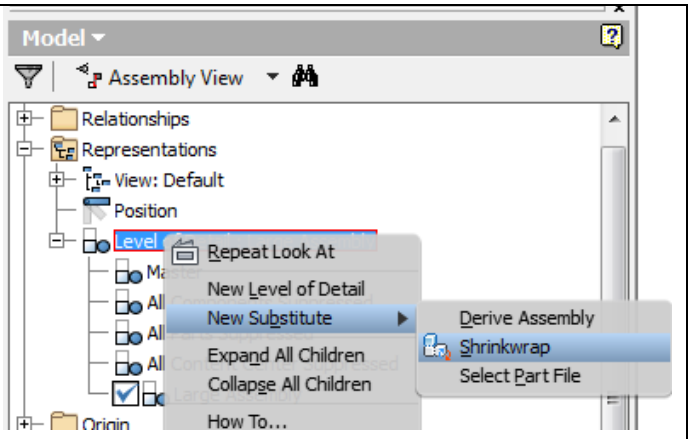
In the next example I will have you create a Shrinkwrap version of the middle lattice section. Shrinkwrap will strip the model of more information than Derive Assembly will. This does a great job of simplifying the model with little input.

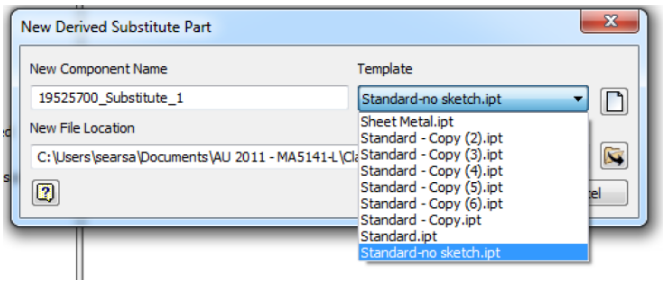
1) Open the middle lattice section “19525700.iam” in a new window.
2) Right click on “Level of Detail:Master”, “New Substitute” and choose “Shrinkwrap”.
3) Change the Template to “Standard-no sketch” and leave the default setting for the rest and hit OK.

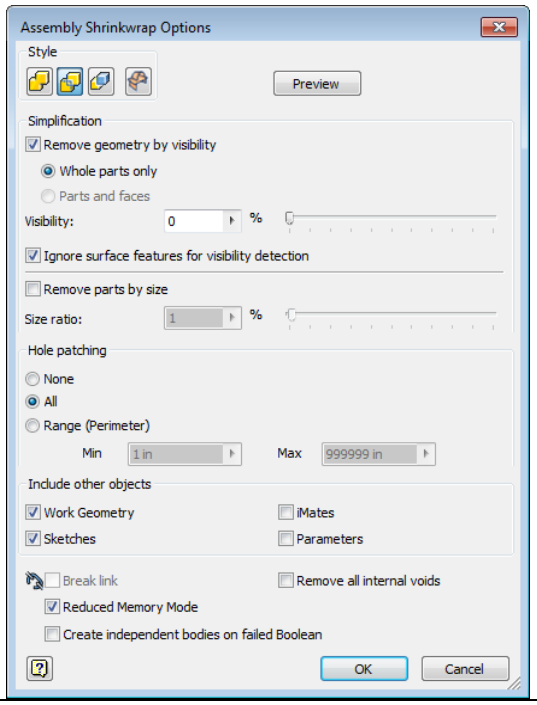
Now we have the Shrinkwrap options window. This is the same as the Derived Assembly, “Options” tab except that you can choose the “Style” of part to create. Using the default settings will produce the best performance results while visually showing enough to show the design intent.

4) Remove the check mark next to “Work Geometry” under the “Include other objects” section.
5) Use the “Preview” button to see what the part will look like before committing.

(HINT) If you wanted to take this deeper you can remove parts by a size ratio and work features to improve system performance. The size ratio is an approximation for selecting parts in an assembly to remove. To calculate the size ratio for removing components that you decide are note needed, take the longest measurement found on the geometry extents and divide it by the longest measurement of the part you want to remove from the model. Bumping this up 2-3% will ensure you remove the parts you want while maintaining the design intent.





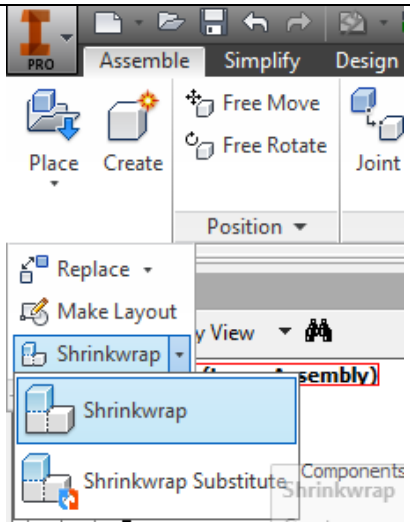


As to shrinkwrap, Inventor also provides another three methods to create it.

1) From Ribbon Command

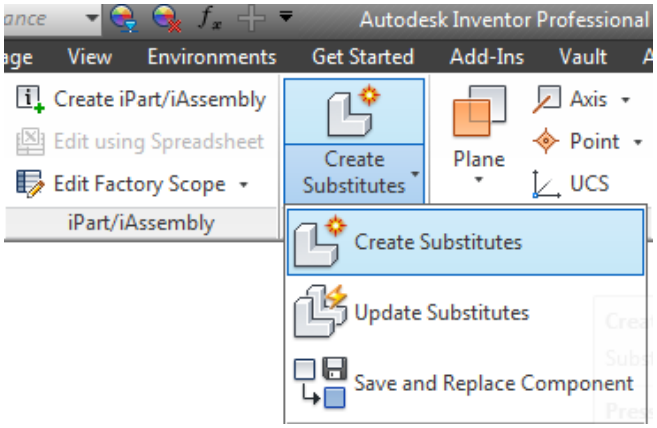
In the Ribbon, Assemble tab, under component is an expandable icon called “Shrinkwrap”. Selecting Shrinkwrap from the drop down will produce a simplified version of the assembly in a new window. Use the Shrinkwrap icon from the dropdown to create simplified files that you will send to customers. Using Shrinkwrap Substitute produces a new assembly LOD.

(NOTE) If the active assembly state is an LOD, then using these options will create Shrinkwrap versions of the active assembly LOD. This is a great way to take advantage of an already simplified assembly and minimize the part count in the open document.



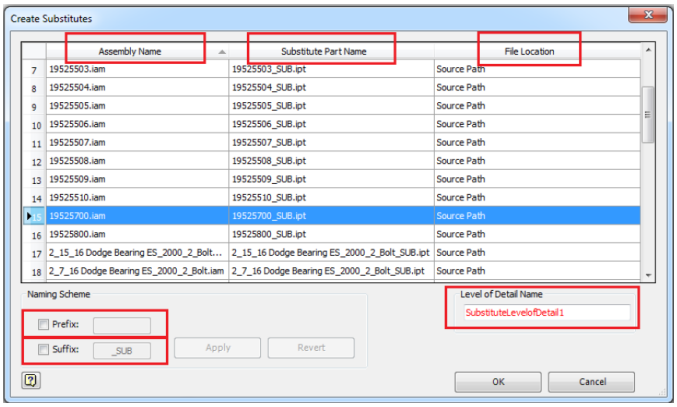
2) From Command in Productivity tab

Another choice is to select “Create Substitutes” under the “Productivity” tab. This will create Substitutes for all sub-assemblies. Let’s try this one.



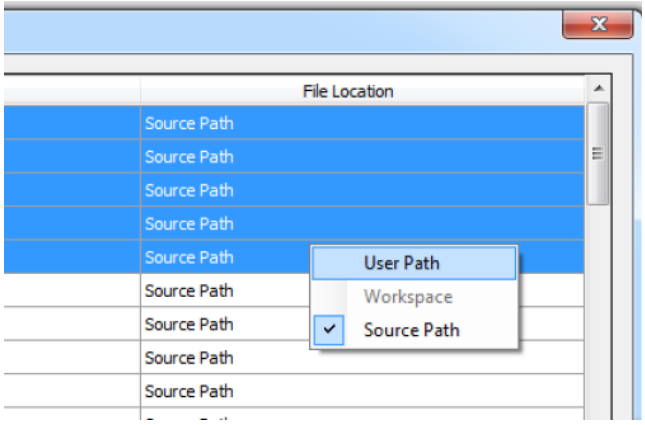
Change the Level of Detail to “Master”. If the assembly is in a LOD state where two assemblies with the same file name are set to different Representations, the program can’t perform two edits on a single document.

expand the Productivity panel and choose “Create Substitutes”



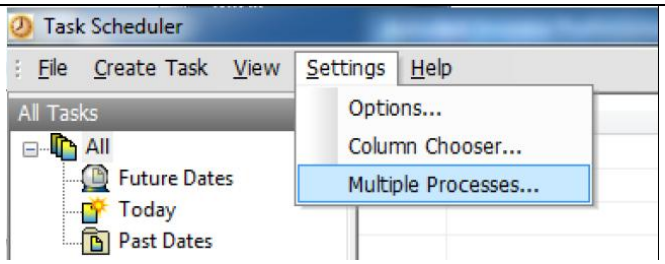
(NOTE) Make sure to hit “Apply” when changing the “Naming Scheme”. If you forget to do this Inventor will create Substitute file names the way it is displayed in the “Substitute Part Name” column. This will create a lot of work to remove what was done once created.

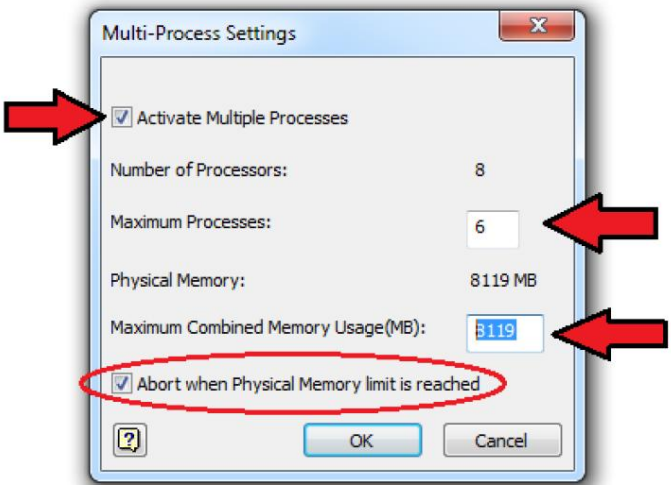
You can change the file save location by highlighting several rows, right clicking on “Source Path” and choose “User Path”. This will allow you to change the location of the Substitute.

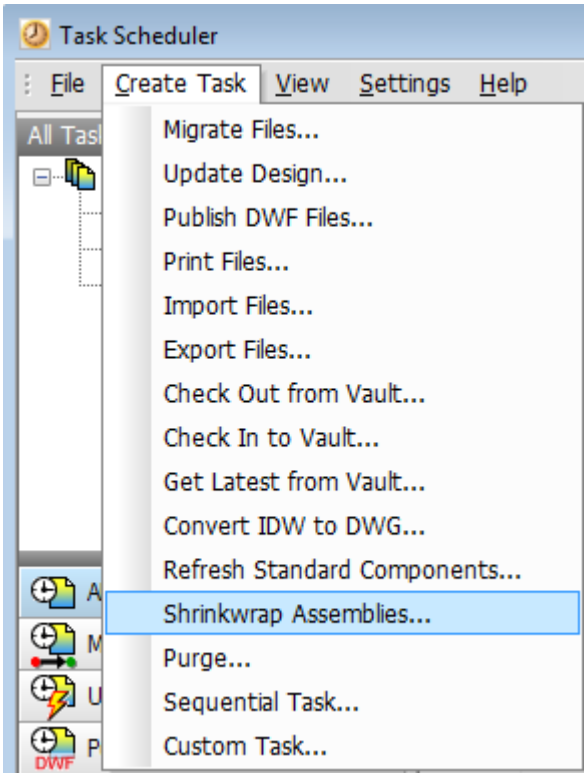


If you are not satisfied with the default results from the automated process, the Substitutes can modified.

Click the OK button and accept the message about creating the Substitutes and read thru the Task Scheduler information below while Inventor creates the substitutes







3) Through Task Scheduler

Task Scheduler is another resource available for you to create simplified models. This tool is extensive and is designed to run tasks using all of your system resources. It is important to understand the output from running tasks so perform testing with files that are not going to be used for production.

Task Scheduler was designed to perform tasks with minimal input from the user to automate more of your time consuming duties like creating Substitutes of very large assemblies. You might need to make some adjustments to the output if the results do not match your company standards. Editing the Derived component is done the same way as described above.

The benefit to using Task Scheduler is that multiple cores can be leveraged. In other words, each task you define can be performed simultaneously with other tasks. Sequential tasks are treated as one task, grouped together, so only one core is dedicated to processing it. You can use Inventor and Task Scheduler at the same time.

1) Open Task Scheduler from Start\All Programs\Autodesk\Autodesk Inventor 2012\Tools\Task Scheduler.
2) Once Task Scheduler is opened, click on the "Settings" button at the top and select "Multiple Processes"

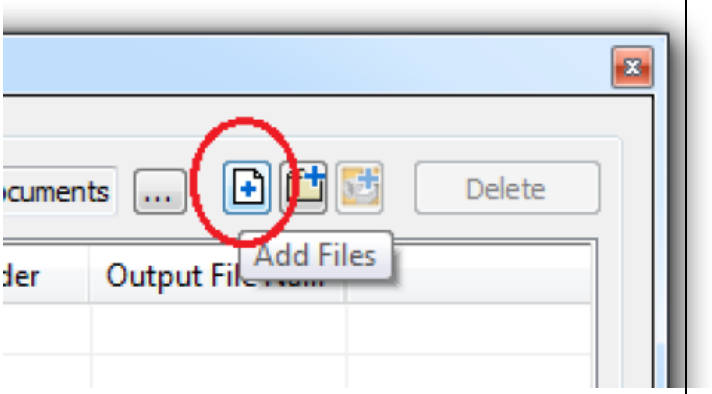
Large Assembly Performace

I like to leave 2 processes free and keep the "Abort" option checked. Running multiple processes of Inventor can consume all available RAM. Once that happens Inventor will start paging. Once paging happens, your efficiency is significantly reduced.

3) Select "Create Task" then "Shrinkwrap Assemblies"

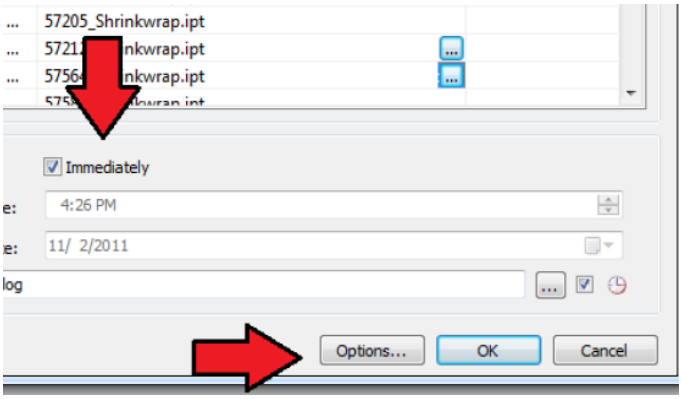
The Project path should point to the data you are working on. If it does not, browse for the 19525100.ipj file.

4) Now you need to tell the program what data to perform the task on. Hit the first icon to "Add Files".



You should be looking at your data.

5) Select a group of assemblies. Try 5-6 of the first assemblies.
6) If you want to run the task now, check the box next to "Immediately" and click the options button.



(HINT) Sometimes it is better to create the individual tasks all at once without running immediately. This gives you some flexibility to change to tasks.

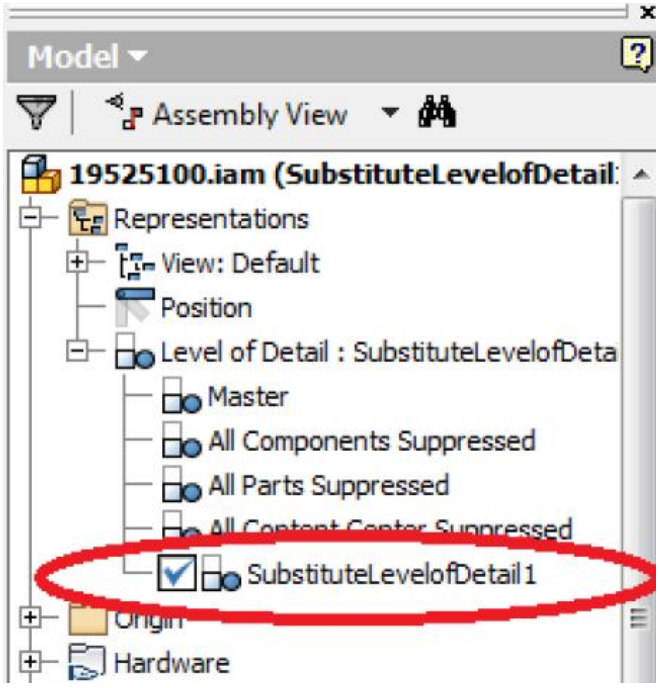
TIPS:

Set a time after you leave for the day to perform these tasks. If your data is on a network, the reduced traffic will allow your tasks to run faster so you can be more efficient.

Select the options that you want for the Substitutes and hit OK to exit the options and OK again to start the task.

The Shrinkwrap substitutes are added to a new LOD for each assembly in the main assembly. They will not be created for sub-assemblies of sub-assemblies of the main assembly.

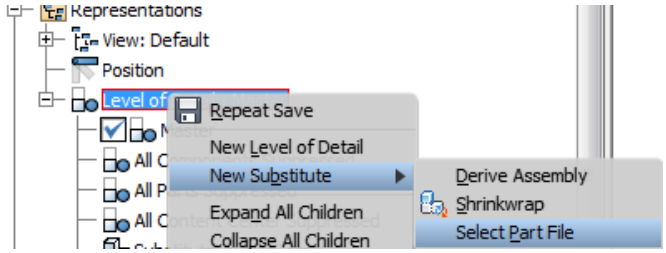
(HINT) While this task is running you can create other tasks by going back to step 6 above and selecting a different group of assemblies.



By now Inventor should be done creating the Substitutes.

7) Update and Save the model.
8) Keep this assembly open and in the new LOD you just created.

4.1.3.5 Create Level of Detail –Select Part File

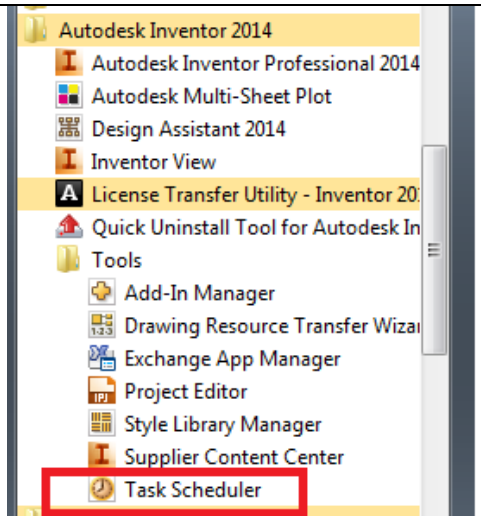
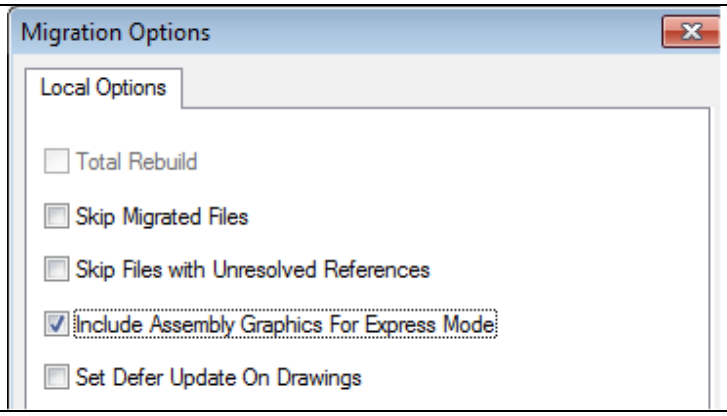
From RMB Context menu of Level of detail, there is the third way to create substitute, that is, select Part file. With this option we can add in a dummy part to be a place holder for a much more complex part. You can just make a cylinder or box to hold the location while you work around it. Considering previous case to simplify the motor with detailed features, we can select the simplfied part file to represent the complex one.	

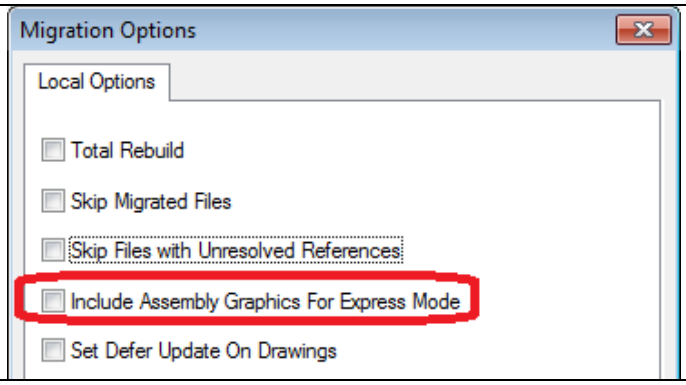
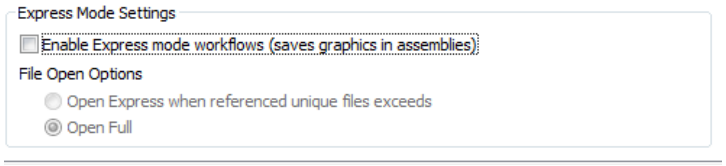
4.2 Large Assembly Express Mode

4.2.1 Open and review large assembly in Express Mode

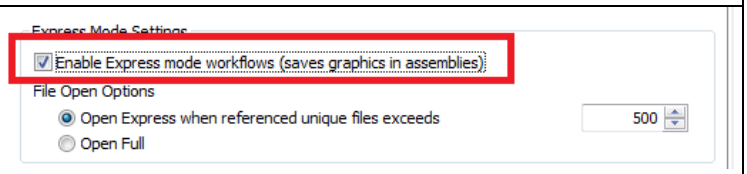
In Inventor 2014, we introduce new technology to reduce the time for opening a large assembly. In this case, user can open and review the assembly in short time.
In order to implement this, we should firstly, migrate the dataset into Inventor 2014 version or save the dataset into Inventor 2014 version with necessary settings and then make sure the settings are as expected. After all the preparation, the large assembly will be open faster since only necessary components will be loaded instead of loading all components as previous release.

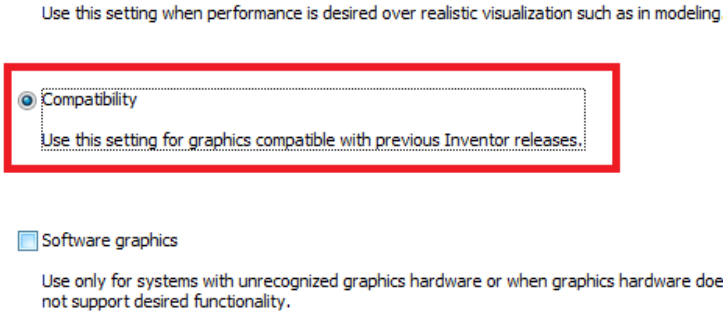
4.2.1.1 Migrate Dataset to Inventor 2014

1) Launch Inventor 2014 Task Scheduler from “Start”->”All Programs”->”Autodesk”->”Inventor 2014”->”Tools”->”Task Scheduler” 2）In the Task Scheduler, invoke “Migrate Files” dialog from the Tab “Create Task”->”Migrate Files…”. 3）Add the folder of dataset wants to be migrated in the Migrate Files	
4）Click the “Options…” button in the “Migrate Files” dialog, and make sure the checkbox “Include Assembly Graphics For Express Mode” is Checked in the “Migration Options” dialog.	

5 ） Combine other options you want to check as usual. Click OK button to close “Migration Options” dialog. And click OK button in “Migrate Files” dialog to start the task. **Warning**: 1. Make sure to back up your dataset before migrating if you need to open the dataset in lower Inventor version in future. Because migrated dataset will NOT be open in previous Inventor version! The size of the assembly will enlarge after migration. This is as expected.	
Tips : How to remove saved cache graphics: **Solution1:** Migrate the dataset with Task Scheduler. And make sure the checkbox “Include Assembly Graphics For Express Mode” is Unchecked in the “Migration Options” dialog.	
Solution2: Launch Inventor 2014 and invoke “Application Options” dialog from Ribbon “Tools”->”Options”->”Application Options”. Make sure the checkbox under “Assembly” tab for “Enable Express mode workflows (saves graphics in assemblies” is unchecked. Open the assembly in Inventor and resave it.	

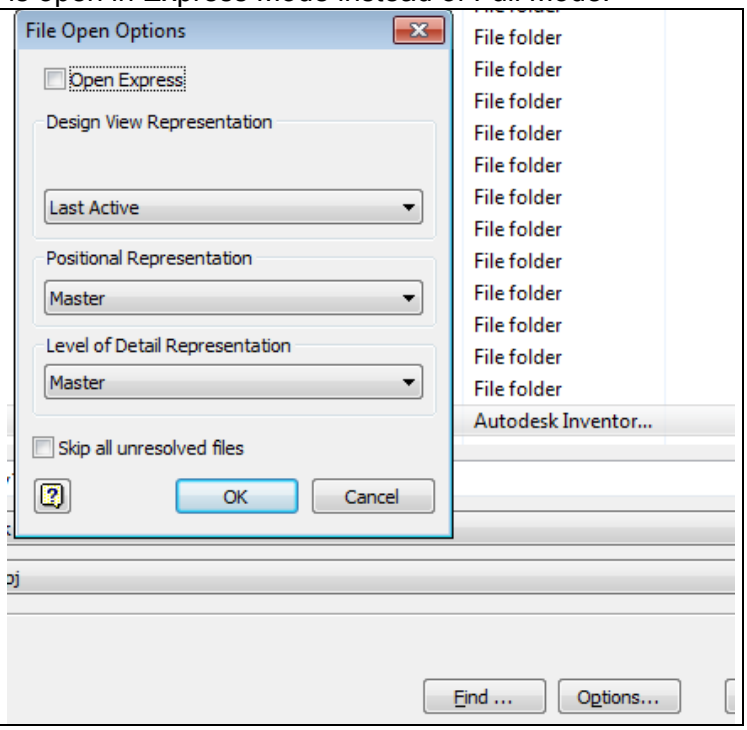
4.2.1.2 Necessary settings

1) Launch Inventor 2014 and invoke “Application Options” dialog from Ribbon “Tools”->”Options”->”Application Options”.	
2) Make sure the checkbox under “Assembly” tab for “Enable Express mode workflows (saves graphics in assemblies” is checked. You can also check Radio “Open Express when referenced unique files exceeds” and choose the number when unique files exceeds that number, the assembly will be opened in Express Mode by default.	

Notice: Please make sure the Graphics Settings is NOT using “Compatibility”, Otherwise, the assembly will be open in Full mode no matter if it has been migrated or the Express Mode Settings correctly.	

4.2.1.3 Open the Large Assembly

There’re two ways to make sure the assembly is open in Express Mode instead of Full Mode:

1) If you know the referenced unique files number in the assembly, you can set the “Open Express when referenced unique files exceeds” less than the number and open the assembly directly. 2) If you have no idea about the referenced unique files number in the assembly, when open the assembly, you still have chance to choose open the assembly in Express mode: 2.1) Invoke Open dialog and select the assembly you are going to open. Do not open it directly but click “Options…” button to invoke the “File Open Options” dialog. 2.2) Check the checkbox “Open Express” in the “File Open Options” dialog.	

4.2.1.4 Limitation

1. Currently only referenced unique files number is huge, the open time will be reduced significantly but for an assembly with lots of patterned components, the improvement for opening is very limited.
2. Only the Last Active Design View and Master Positional and LOD Representation will be open if you want to open the large assembly in Express Mode.
3. Section View is not supported in Express mode.
4. Realistic and Monochrome visual styles don’t work in Express Mode.

4.2.2 Place Large Assembly in Express mode

We always encounter the issue that placing component costs a lot of time especially when the placed component is a large assembly. This issue will make the user upset if they are using Bottom-Update design style.

In Inventor 2014, we start supporting to place assembly in Express mode which means not all components of placed assembly will be loaded but only the necessary ones.

In order to implement this, the placed assembly and top assembly should also be migrated to Inventor 2014 version with specific settings. Refer to the section “Migrate Dataset” in opening LA in Express Mode for detailed information.

After the preparation, open the top assembly in Express mode and place the large assembly. In the “Place Component” dialog, click “Options…” button and Invoke the “File Open Options” dialog similar

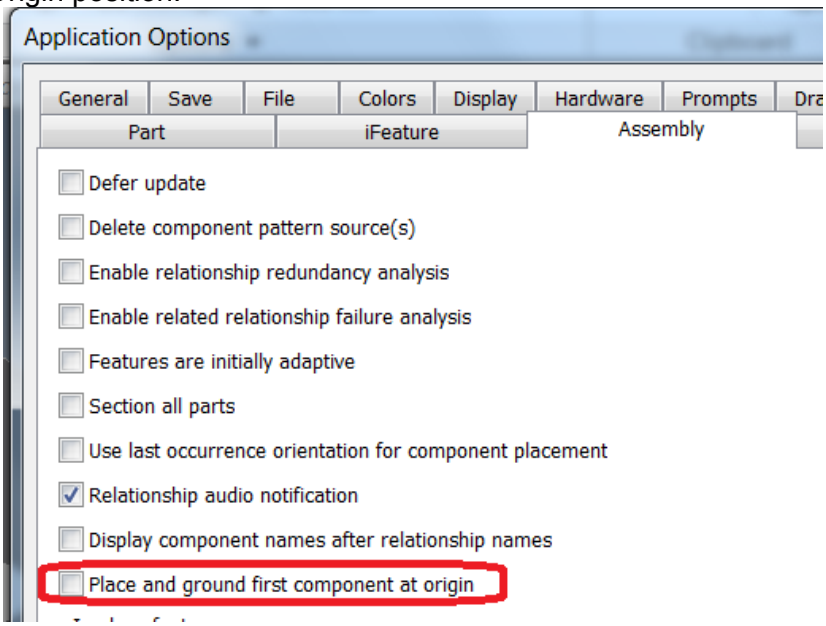
with Open Large Assembly, and make sure “Open Express” checkbox is checked. Then you can place the component in short time.

Note: This implementation is only workable with Inventor 2014 SP1 and higher service packs.

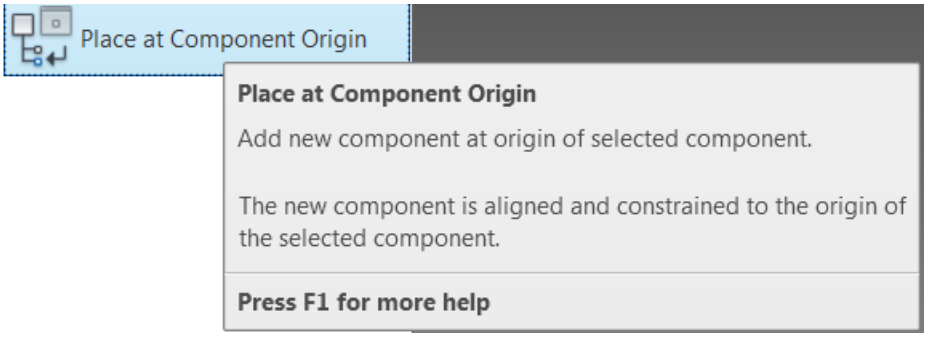
As to place component command, there are some tips for your reference:

- As we know, the first component placed in a top assembly will be placed at the Origin of the top assembly. From R2014, there is an option in Application option to control the placement behavior of first component:
  Application Options->Assembly tab->Place and ground first component at origin.

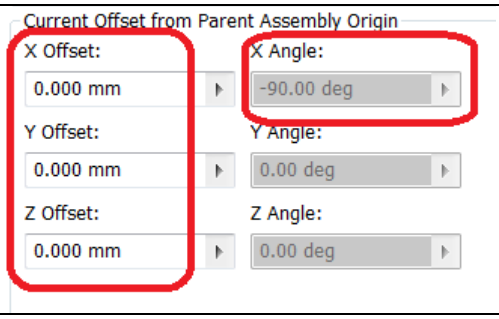
  If the checkbox “Place and ground first component at origin” isn’t checked, we could decide whether the component is placed at Origin or other place. If it’s checked, the first component will be still placed in Origin position.



- There has been some complains that only the first component could be placed at Origin, and user also want to place non-first components at Origin.
  In such condition, before R2014, user has to use the “Place at Component Origin“ command from Assemble tab->Productivity panel to place the following components at the Origin of the first component.



- From R2014, when place component, we could rotate the orientation of the component during placemnt. The rotation is along the rotate axis in the graphic window by following right hand rule.
  And we could rotate the component then place it at Origin, the Orientation will be kept during placement. See the detail information below. Here is an example for rotate X 90 degress then place grounded at Origin.

Workflows	iProperties->Occurrence
Rotate X 90 + Place grounded at Origin	

4.3 Matrix from Derive assembly, Shinkwrap and Express mode

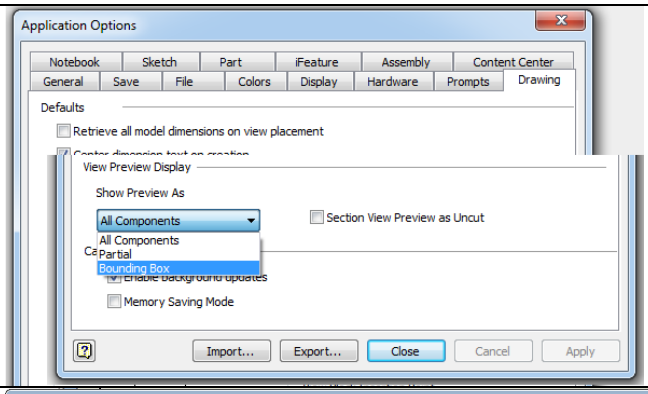
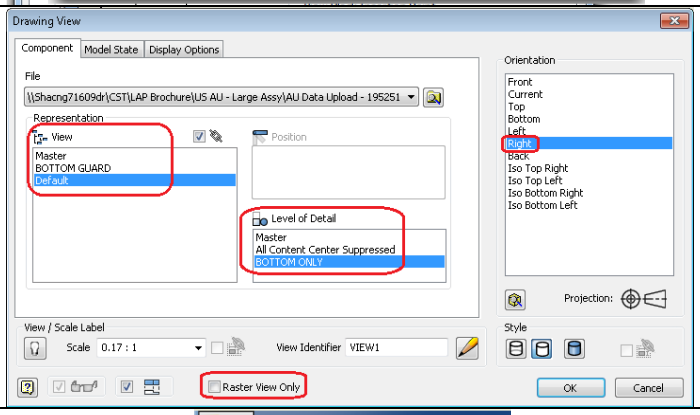
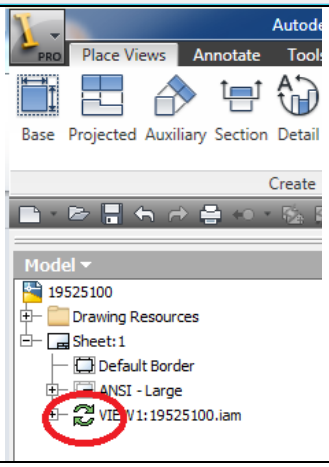
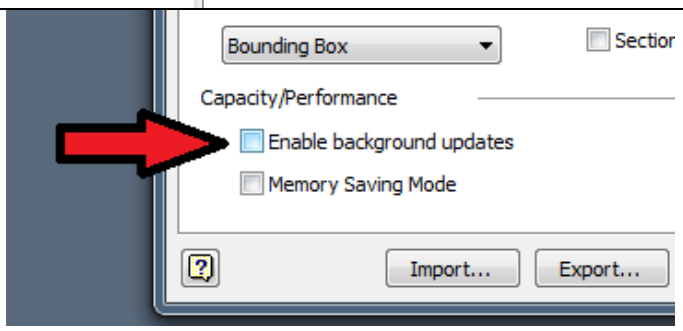
Functionality	Case
Derive Assembly	• Create lightweight structure • The derived part can be scaled larger or smaller than the original or mirrored using any of the origin work planes of the base assembly. The location and orientation of the derived geometry is the same as the base. • By selecting different combinations of parts and subassemblies, multiple derived parts can be created from a single assembly. • The derived part can contain components as multiple solid bodies. • The derived part can be used as a Substitute Level of Detail in the assembly it was derived from. This reduces memory consumption in upper-level assemblies. • Weld beads could be selected as a solid.
Shrinkwrap	• Keep mass of the original assembly, • hidden some important details • Patch some holes which doesn't relate with downstream workflow • The Shrinkwraped part can be used as a Substitute Level of Detail in the assembly it was derived from. This reduces memory consumption in upper-level assemblies.
Express mode	• Reduce the time for openning large assembly. • Improve the performance of reviewing large assembly like zoom in/out, pan and rotate. • Express mode can make placing large assembly in short time.

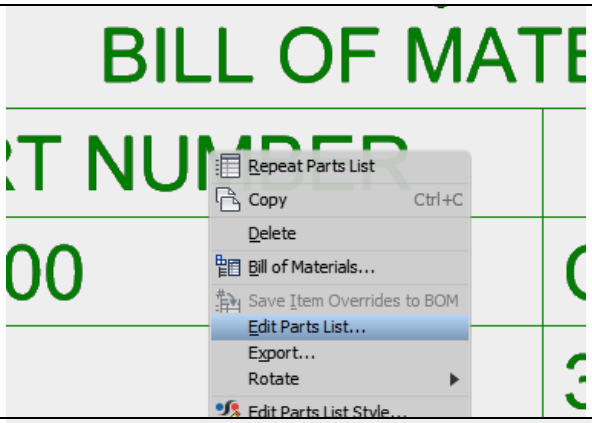
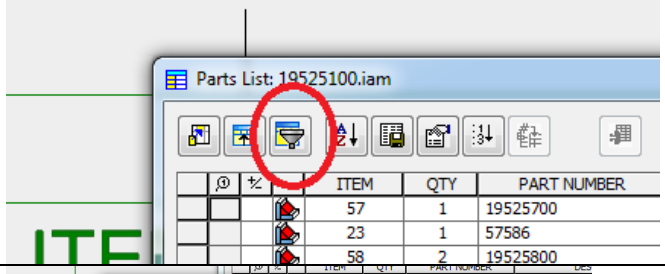
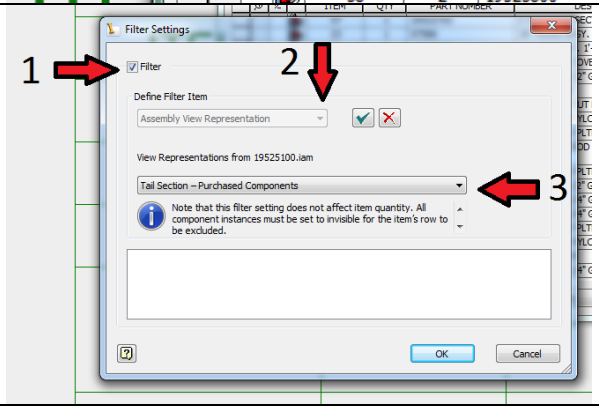
4.4 Drawing

4.4.1 Level of details & view representation for drawing

Now let's look at using LOD's and view representation in a drawing. LOD's are more valuable if you are having large assembly performance limitations.

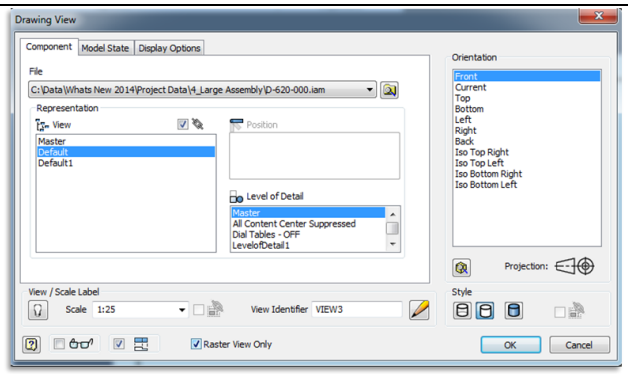
Large Assembly Performace

1) When creating drawing views of large assemblies change the setting in the application options\drawing tab to show the preview as a bounding box. 2) You should still have the assembly open and in the LOD state you just created. *(HINT) When I create drawing views I will create the "Base View" first and then create the projected views only when needed. Inventor needs to calculate all lines for all views.*	
3) Open a new drawing and select "Base View". 4) In the "Drawing View" dialog, select one design view, and your LOD should be highlighted. 5) Select the "Right" orientation and hit OK. *(HINT) One option "Raster View Only" is in "Drawing View" dialog, and the default is off. When it is checked, it only generate the raster view. For more details about raster view, please see the next section.*	
6) Once the view is placed you should see an icon in the model tree that shows the view is being generated.	
7) Inventor 2014 will use all cores to process views now. In the Application Options\Drawing Tab you have the option to enable background updates on view creation. What this will do is allow you to work on the drawing while the lines are being drawn and each available processor core will be used to generate a view. A snapshot, called a raster image, of the desired view is shown while the lines are being drawn. This snapshot has some intelligence in that you can add dimensions and balloons while the view is being generated.	
8) To see this, open Windows Task Manager to view all cores. 9) In the drawing, place several projected views bounding boxes, right click and choose "Create" then roast a marshmallow over	

the cores!	
10) When the drawing views are finished, part list is added to document. View Representations can be used as a filter for a BOM. 11) If the BOM is filtered to use a specific View Representation, only the parts that are visible will show up. The only downfall to this is that the quantity shown in a filtered BOM shows the total quantity and not the actual amount displayed in the view. To access the BOM filters, right click on the BOM and select “Edit Parts List”.	
12) In the Parts List, select the “Filter Settings” icon.	
13) In the Filter Settings dialog, follow the settings below. 14) *(HINT) You can use more than one filter for more control over what information is shown in the BOM against what is shown in the drawing view. Try setting the filter for “Purchased” and then add your View Representation. To remove a filter either uncheck it or use the delete key. Filters are applied to the BOM in the order listed.*	

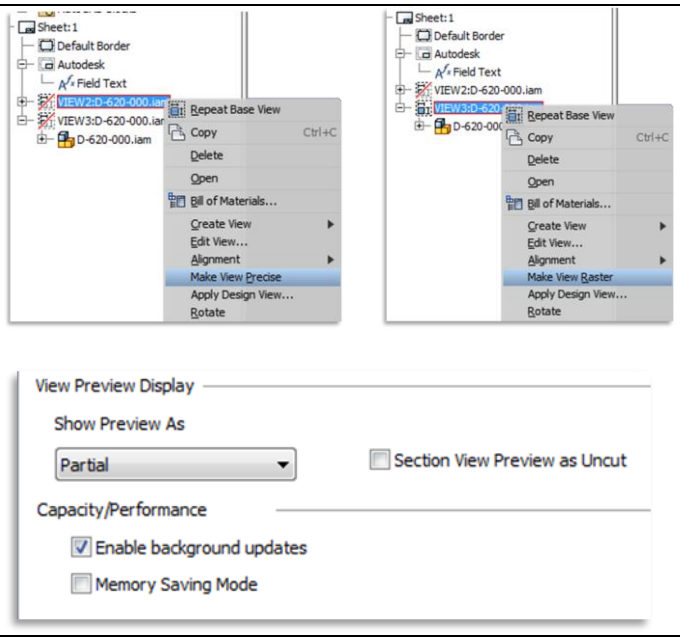
4.4.2 Raster view

Inventor 2014 supports to only create raster view, which will help users quick generate the drawing view without precise calculation, although the assembly is large. For raster view, most of annotations can be added, but the quality of raster views is not good for printing or zooming in/out. So the raster views are required to be converted to precise view when it is used for production.

1) For the new design, raster view can be quickly created, and add the annotations. When done, we convert it to precise view as the final result. Maybe it will cost a long time for conversion, but we could make it done when the user is not busy.	
2) For the legacy design reused, firstly we need to convert the precise views, then update the 3D design, finally convert the raster view to	

precise view when the calculations of raster view updating are done.

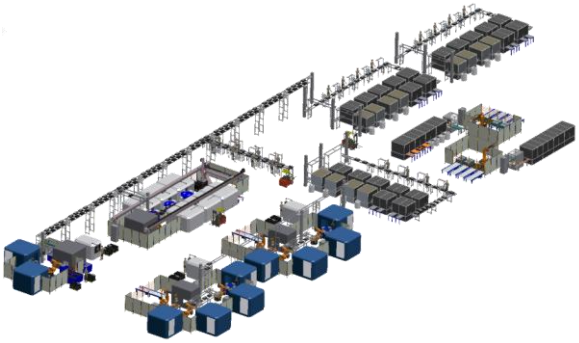
(HINT) In the Application Options\Drawing Tab you have the option to enable background updates on view creation.
What this will do is allow you to work on the drawing while the lines are being drawn and each available processor core will be used to generate a view.



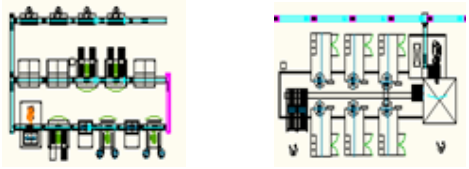
4.4.3 Drawing views composed for extremely large assembly

For some customers, they use Inventor to design manufacture process line, and want to generate the drawing view based on the 3D model. But the top assembly is extremely large.

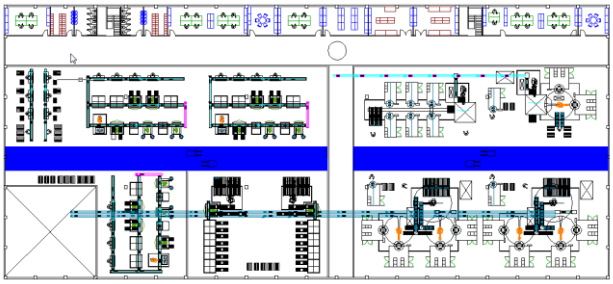
1) As the top assembly is composed by several components, and these components are on one layout. If we use the top assembly to generate the drawing view directly, it will take a very long time to generate the drawing view, sometime maybe the drawing view can't be generated as the top assembly is too large.



2) We need to consider generating the drawing view for every component.



3) Compose these drawing views in one layout for the top assembly.



5 Best Practice: Typical LAM Workflow in Inventor 2014

5.1 Bottom Up vs. Top Down Design

Mechanical design is a highly creative and complex activity. Powerful computers form a basis for competitive solutions, reduce product development time and provide means of communication within the design team.

When we talked CAD design methodology, there are two typical design method will be mentioned, that is, Bottom Up and Top Down Design.

Bottom Up is the traditional way of building assemblies. You first define the different parts. Then you put them into sub assemblies using assembly constraints. The sub assemblies are then placed into higher lever assemblies up to the top level assembly and in this way, you are working your way from the bottom up. This assembly method will create assemblies with a number lot of relationships between parts and assemblies. This approach can lead to two things:

1. It can consume system resources and slows down performance.
2. It can make models hard to change due to a lot of cross references and/or referenced geometry that gets deleted.

A simplified example of this approach is the building of a house of cards. You start from the bottom leaning the cards towards each other for support and build your way upwards where every new card placed are dependent on the previously placed cards. If you then need to perform a design change like replacing one of the cards in the first layer with a new one you'll run into stability problems since later placed cards are dependent on the card you remove.

Skeletal, top down design is a method of working with large assemblies. Basically, you put all known factors into a skeleton file which is then used as a base for most of the sub assemblies and parts that make up the assembly.

Top Down is a method where you start defining the end result and build in all of the known design criteria. This becomes the base for underlying sub assemblies and parts. In this way you will have a single conceptual file containing the overall information of the design with a single place for incorporating design changes.

Working this way provides a much more stable Inventor model, faster updates, more available resources for handling larger data sets, an easier way of working in a collaborative environment and a better way of doing design work in general.

Skeletal Design Methods
There are four different methods that can be used for skeletal design:

1. Skeleton File

   A skeleton file can contain any type of elements in any combination. Typical information created in a skeleton file may be:

   a. Sketches: used to define the concept of the final product or may be used as a layout of different areas of the assembly.

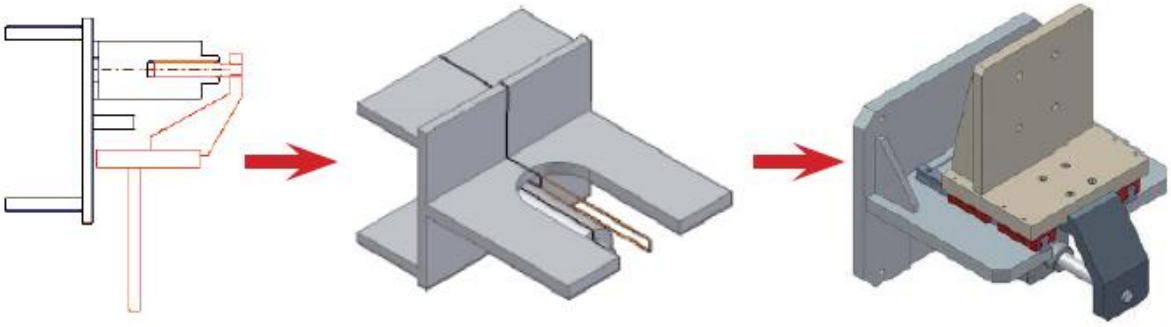
b. Work geometry: used to define important connection points, axial directions, and work planes that defin assembly levels.

c. Parameters: define important values for part size, angles for placements, and other known values for the design.

d. Solids: used to define volumes for final assembly as a single part or for defining sub-areas of the design.

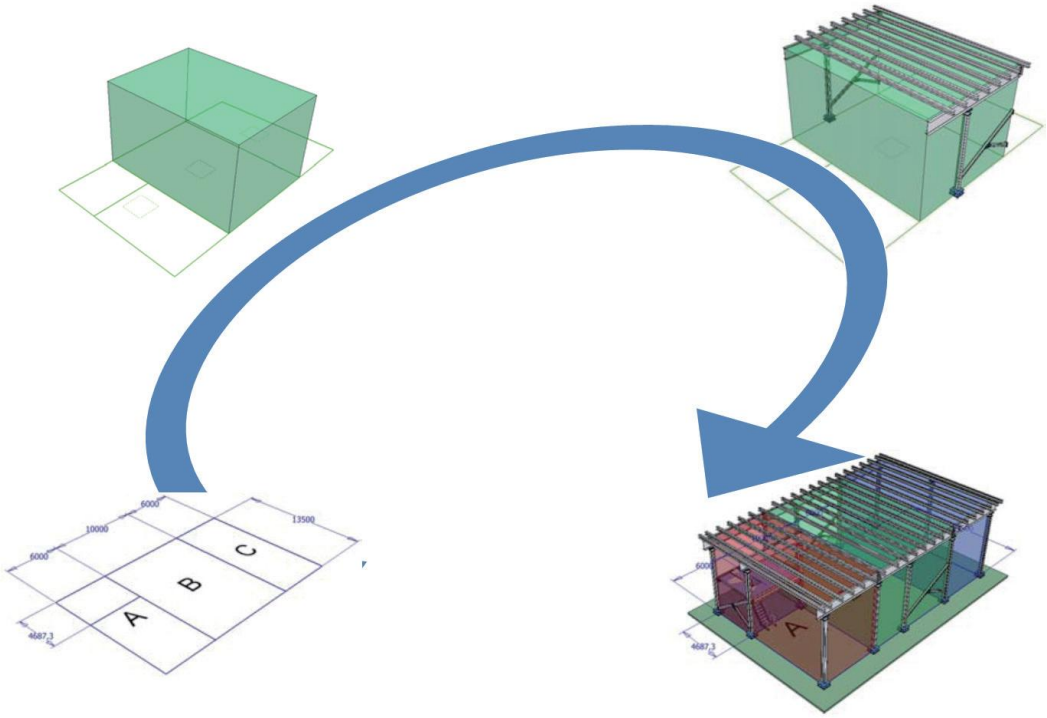
2. Master Sketch

A master sketch can be used to control the form, fit, and function of a design.



3. Absolute Origin

This is a method where all sub assemblies are placed at the same origin. Every sub-assembly is based on an origin part. This origin part is usually a sketch or a solid defining the volume. All sub-assemblies are positioned against the origin parts 0,0,0 location and then grounded.



4. Workspace Envelope Method

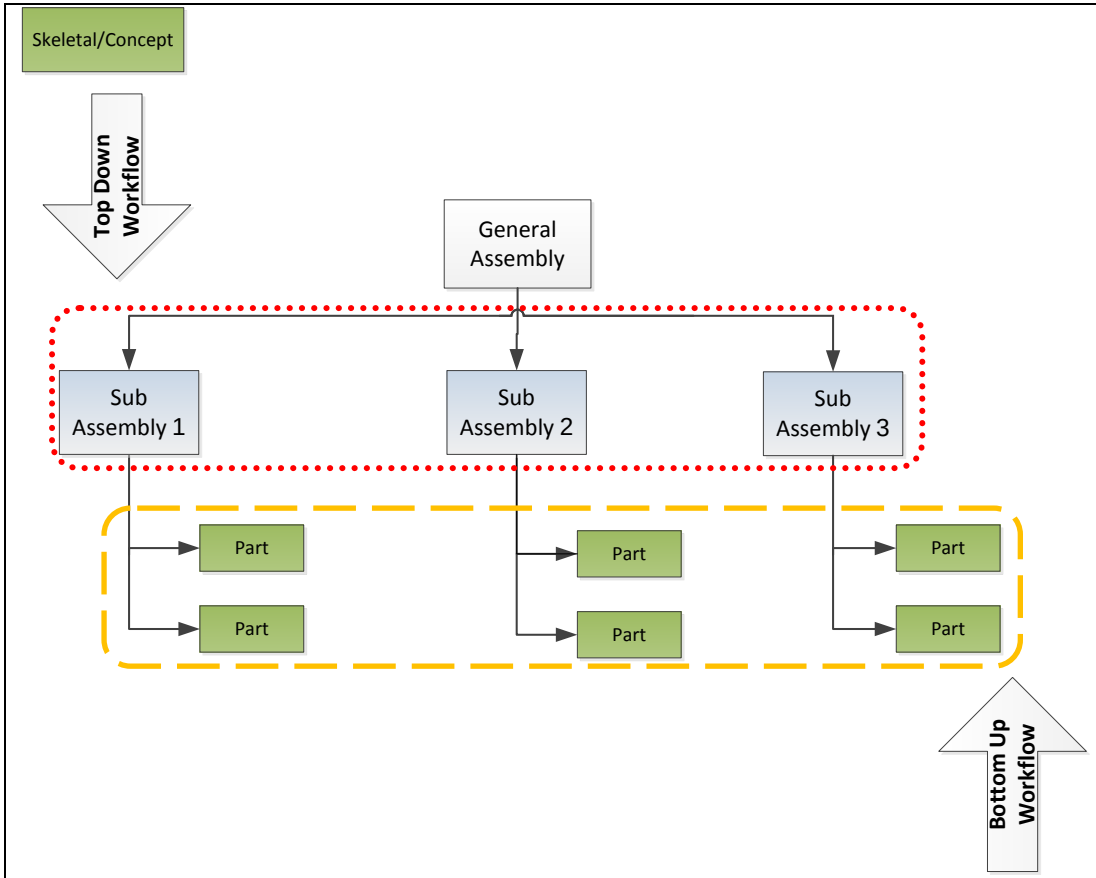
This method uses defined workspace for individual assembly components and mutual contacts of the components. Boundary faces or placeholder solids define the workspace.Internal structure of individual components (subassemblies) is created inside the workspace. Again, the procedure is tightly bound to component simplification method, as the placeholder solids may be used for parent assembly composition. Practical use of this method corresponds to the master sketch method.

There is neither a universal procedure nor a general method. The essential aim of the design includes an easy-to-manufacture product, cost-effective production, and useful drawing documentation. Top Down and Bottom Up all have its applicability on specific design cases. In reality, it's very common to and necessary to use combination of multiple methods to produce complex assemblies.

5.2 LAM workflow process

Autodesk Inventor provides rich functionality to support different design method here, and we will not describe more details of design method. From below image, we would like to show you a whole picture of LAM in your design process.

Large Assembly Performance



Skeletal Design Method (Top-Down)

- Skeleton file
- Master Sketch
- Absolute Origin
- Workspace Envelope
- Please refer section 5.1 for these methods

Assembly Level

- Reduce number of components, please refer section 4.1.1
- LOD, Shrinkwrap & Derive, please refer section 4.1.3
- Express mode, please refer section 4.2

Part level

- Suppress unnecessary features, please refer section 4.1.2.1
- BIM Simplification, please refer section 4.1.2.2

Bottom-Up

- Project settings, please refer section 3.1
- Plan for Assembly Hierarchy Efficiency, please refer section 3.3.3

Drawing

- LOD & View Representation, please refer section 4.4.1
- Raster view, please refer section 4.4.2
- Drawing views composed, please refer section 4.4.3

6 Special Thanks

This manual is a gathering of experience and knowledge on large assembly, and the main goal is to help our customer better use Autodesk Inventor product in their design and work. Here special thanks Andy Sears to provide the support and material for reference, and Inventor QA and development team for the offline support.

7 Useful Links

- Dataset used in this document: http://a360.co/16JostY
- Help Document: http://help.autodesk.com/view/INVNTOR/2014/ENU/?contextId=HELP_SYSTEM
- News Group Forum: http://forums.autodesk.com/t5/Inventor/ct-p/70
- Inventor IdeaStation: http://forums.autodesk.com/t5/Inventor-IdeaStation/idb-p/v1232