BOMs Away! Working with and Autodesk® Inventor® Bills of Materials

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In many cases, the bill of materials is the most important part of your drawings when it comes to communicating within and outside your corporation. This class will examine the power of the bills of material (BOM) produced by Autodesk Inventor software.

Learning Objectives

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

- Use the BOM interface and examine how the BOM is built and maintained
- Explain the two BOM types as well as how the BOM structure impacts what you see in both the assembly BOM and the drawing parts list
- Use quantity and part number merging to build cut lists and other types of parts lists.
- Define how the bill of materials can be exported to a neutral format and used in other systems and see how the Inventor bill of materials compares to an item master created using Autodesk® Vault Professional software.

About the Speaker

A specialist in the manufacturing industry Mike provides training, support, and implementation on manufacturing solutions. His strong Autodesk mechanical knowledge is supplemented with a strong understanding of document management, hardware, networking, and other Autodesk technologies. With many certifications both present (Inventor Certification) and past (MICE, DMICE) and his past experience in the Autodesk Reseller channel he is very highly regarded within the industry

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Inventor’s Bill of Materials

What is a Bill of Materials?

Concepts in Enterprise Resource Planning defines the bill of materials as “A bill of materials (or BOM) is a list of the raw materials, sub-assemblies, intermediate assemblies, sub-components, components, parts and the quantities of each needed to manufacture an end product. No physical dimension is described in a BOM.”

The Autodesk Wiki defines it as “A bill of materials is a table that contains information about the parts within an assembly. The table includes quantities, names, costs, vendors, or any other information someone building the assembly needs. Quantities on the bill of materials are updated when a part is added or removed from the assembly.”

What is Inventor’s Bill of Materials?

A Bill of Materials (BOM) is a listing of the components contained within the assembly. It is important as it defines what makes up the assembly and what needs to be ordered. Inventor stores BOM information within that assembly including the component’s quantities, iProperties and Structure information. [iProperties being the meta data that describes each component (like description, title, part number, mass, etc, etc, etc.).]

Figure 1: iProperties "feeding" the Bill of Materials

Not only can the iProperty information be viewed from the BOM but can also be edited. Using the BOM can be a very efficient method of modifying iProperty information quickly and efficiently on one component but multiple components as well.
Working with the Bill of Materials

The bill of materials in Inventor is a listing of all the components in the assembly. It is accessible from the Assemble Tab, Manage Panel in the Inventor Assembly ribbon.

The Inventor BOM dialog always shows 3-tabs: Model Data, Structured, and Parts Only.

- The "Model Data" tab will always show all of the parts and sub-assemblies regardless of the settings applied to the data
- "Parts Only" displays all the parts, no subassemblies or assembly structure. All parts nested within a subassembly are “promoted” to the top level creating a “flat” view of the components
- "Structured" is a view of the assembly showing the assembly structure hierarchy, including subassemblies and their components. Subassemblies can be expanded to display their children.

Once the view is enabled (via right-click on the tab heading) you can use the View Properties to set tab specific options.

On the Structured tab you can set the available levels and the delimiter that is used to distinguish the sub-component levels.

On the Parts Only tab you can set whether the components are numerically item numbered or alpha.

Sub-Assemblies can be expanded to show the components contained within. This is done with the + shown in the first column of the BOM editor.
Rows, Columns & Sorting the Data

The columns with the BOM dialog can be dragged and dropped to locate in the desired location. Any iProperty can be added as a column using the Column Chooser option.

![Figure 2: Adding iProperty Columns](image)

The components in the BOM will display in the order they were created or placed into the assembly.

- Rows can be dragged and dropped to reorder
- The data can be sorted by clicking on the column header (Reverse by clicking it again)
- This data can be sorted based on any iProperty, including custom iProperties. The item numbers can then renumbered to restore the sequential ordering.

![Figure 3: Sorting BOM data](image)
The sort feature allows you to sort by multiple properties [NOTE: Data can only be sorted from the Structured or Parts Only tabs]

When everything is sorted to your liking use the Renumber feature to reset the item numbers in sequential order
When in Doubt Right-Click

The right click menu within the Bill of Material provides many options to work with the data:

- **Find & Replace** feature to quickly located properties or make changes
- **CAPITALIZE** option to change selected cells text to all uppercase
- **Copy & Paste** information from within the BOM or from other sources. Paste into multiple lines
- **Open** components directly from the BOM

Virtual Components

A Virtual Component is a special type of component that has no geometry but is a placeholder for iProperty information. As there is no geometry no external file is generated nor is required. Virtual Components appear and behave in the BOM like a regular component and contain the full set of iProperties

Figure 5: Creating Virtual Components
Exporting Information

The BOM data can also be exported to an external file. Once exported the external data is NOT linked to the Inventor BOM, meaning changes within Inventor will need a re-export of the BOM to update the external data.

The data can be exported from the Structured or Parts Only tabs and can be exported to a wide variety of formats including Microsoft Excel (.xls), Microsoft Access (.mdb), Dbase (.dbf), Tab-delimited (.txt), and Comma-delimited (.csv).

All though the data is static once exported the BOM is wide “open” to creating custom macros or Inventor add-ins to automate the process.

Figure 6: Exporting the BOM
Bill of Materials of Structure

The Bill of Material *Structure* is used to create a more realistic BOM and filter out components within the assembly that are used in its construction but are not actually part of the assembly.

- **Normal** is the default setting, your parts and subassemblies appear as they are in both the Structured and Parts Only list.
- **Phantom** is used for subassemblies built to group components but the subassembly itself is not required to be shown in the BOM. The “children” components are promoted into the Structured BOM view.
- **Purchased** is for components bought and not manufactured. They will be listed as a single line in the Parts Only BOM view even if its an assembly.
- **Reference** is for components placed into the assembly for the sole purpose of construction. These components (both parts & assemblies) do NOT appear in the BOM (Parts only or structured). Reference components are not included in mass calculations. They can be displayed differently in the drawing environment.
- **Inseparable** is intended for subassemblies like weldments that are treated like a single part with completed. They are treated as a single line item in the Parts Only BOM view UNLESS the child component is purchased, which will be promoted to be shown.

The BOM Structure can be defined from the *Occurrence* tab of the Properties dialog box, from the BOM dialog, or during component creation. The structure of the component can also be set in the Document Settings so that any time the component is used the structure is already set.

Reference can be set on the instance after it is placed into the assembly via a right-click.

![Figure 7: Setting Structure using Document Settings](image1)
![Figure 8: Setting Structure using the BOM](image2)
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Figure 9: Adjusting Component BOM Structure while creating a new component

Figure 10: Adjusting Component BOM Structure via Occurrences

**Normal** - the default status for a component and the components are placed in the BOM based on their parent assembly.

**Phantom** – Used for sub-assemblies “phantom” components are not included in the BOM. Sub-components of the phantom assembly are “promoted” to be shown at the same level as their phantom parent. This is used when an assembly is created simply for grouping components together and not desired to be listed in the Bill of Material / Parts List.

Figure 11: “Normal” BOM Structured View

Figure 12: “Normal” BOM Parts-Only View

Figure 13: Phantom Structure
**Reference** – used when the component is only used in the construction of the assembly but is not required for the actual assembling. Neither the component nor its sub-components are shown in the BOM and by default the drawing views will show reference components as dashed lines.

![Structure changed to “Reference”](image)

**Figure 14: “Reference” BOM Structure**

![Setting how Reference Data should be displayed](image)

**Figure 15: Reference Data Options**

The reference option is useful when components or assemblies have been placed into the assembly for use as a reference as in to build and position the other components contained within the assembly. Although important for the construction of the assembly these components are not required to be displayed within the BOM and Parts List.
**Purchased** – In the Structured tab “purchased” components behave as normal components. In the parts-only view “purchased” assemblies will show as 1-line item and can not be expanded to show its sub-components. Used when the item is purchased and will be delivered as a single item. There is no requirement to know the items that comprise that item.

This is used when the entire assembly is ordered and delivered as a single item and not as individual components.

![Structure changed to "Purchased"
assembly still expandable in Structured tab](image)

**Inseparable** – similar to purchased components in that within the parts-only view the item will be listed as 1-line item. The difference is if a sub-component of an inseparable assembly is tagged as “purchased” the purchased sub-item will be “promoted” to be displayed with the inseparable component.

This used with assemblies like weldments where the components are always grouped together but it is important that anything that is purchased from outside is identify to make sure it is ordered.

![Structure changed to Inseparable
Inseparable assembly and "purchased"
sub-component are listed in parts-only view](image)
Quantity

Quantity in the BOM can be shown in many different ways, including: item quantity, unit quantity, & total quantity. These can be selected as the other columns with the Choose Column button.

Every part has a Base Quantity and Base Unit which defines how the quantity of the component is shown in the BOM and drawing Parts List. This is set within the Document Settings of the part.

Within the Document Settings for a component you can change the **Base Quantity**. Base Quantity by default is “Each” meaning the BOM will list the number of instances of the component in the assembly.

**Base quantity** can be changed to show any parameter of the component (for example, the length of the component). The value of the parameter will be shown in the BOM and the BOM will always show the updated value of parameter.

Figure 16: Adjusting Item Quantity

The default is typically “Each” meaning the BOM will show the number of instances of the component within the assembly. Any parameter can be used as the quantity meaning the quantity can represent length or other dimensions of the component. Any length unit (i.e. in, mm, m, etc.) can be set

**ITEM QTY** always shows the number of instances of the component within the assembly regardless of its quantity value

**BASE QTY** shows the base quantity (from the document settings of the component) of one instance of the component in the unit of the parameter used.

**UNIT QTY** shows the base quantity of one instance of the component but is expressed in the base unit set in the document settings of the component **Unit quantity** uses the Base Unit set in the components.

Figure 17: Showing different types of quantity

**QTY** (or TOTAL QTY) shows the sum of all the instances in the assembly (Unit quantity X item quantity)

**Total quantity** or QTY is the **Base quantity** multiplied by the **Item quantity**
Component Equivalence

If components in an assembly have the same part number they will be automatically merged into a single line item. If this is not desired it can be disabled in the Part Number Row Merge Settings dialog.

![Part Number Row Merge Settings](image1.png)

Figure 18: Part Number merging

The part number property can be modified directly in the Part Number column within the BOM editor or using the iProperties dialog for the component. If the other iProperties of the combined items do not match, the BOM reports the property as "varies".

![Initial View, part numbers differ](image2.png)

Figure 19: Initial View, part numbers differ

![Part Numbers are changed to "123" initiating part number merging.](image3.png)

Figure 20: Part Numbers are changed to “123” initiating part number merging.
The Drawing Parts List

A Bill of Materials (BOM) is a listing of the components contained within the assembly. It is important as it defines what makes up the assembly. To display this BOM information within a drawing a Parts List is created. The Parts List can be thought of a snap shot of the BOM showing only the information relevant to the drawing. The information contained within the Parts List can be tweaked to override it specifically for the drawing.

Within Inventor drawings you can place Parts Lists. Changes made to the assembly BOM (or component iProperties) will be reflected in the drawing parts list.

The initial view of the parts list, including the font used, sizes, and columns shown are controlled by the current active style and active parts list style.

A drawing view is NOT required to create a parts list. When placing the parts list you need to select an assembly (or Presentation file .ipn). The assembly can be selected either by browsing, selecting from the list of currently open documents, or by selecting an existing drawing view.
Once the document is selected four options are available to set the “BOM View”. Each one (Structured, Parts Only, Structured (legacy), or Parts Only (legacy)) have different options.

The options “Structured” and “Parts Only” correspond with the Assembly BOM types. If the assembly BOM types have not been activated Inventor will prompt to enable these columns before generating the parts list.

Prior to Inventor 9 there was no BOM data stored within the assembly. In these earlier releases the parts list was used to capture and display the iProperty information of the components in the assembly. This legacy method of creating parts lists is still available; Structured (legacy) and Parts Only (legacy). These options should not be needed as the legacy options do not use BOM structure, BOM quantity overrides or the BOM Item Numbers. Additionally the automatic part number merging of components is not available.

Parts lists using the legacy options can be converted but are not automatically converted.

**Editing Parts Lists**

You can access the assembly BOM dialog from the Parts List by selecting the parts list, right-clicking, and selecting Bill of Materials. Changes made to the BOM will be reflected in the parts list, unless the parts list value has been set to static (see below).

The table size, column widths, and row heights of the parts list can be modified in the drawing by clicking and dragging the lines to the desired size.

To edit the parts list contents and formatting select it, right-click, and select Edit Parts List. Alternatively a double-click on the parts list will open the Parts List editor.
The cell data can be overridden, which will toggle the cell to be static. Static values (except for item numbers) can-not be saved back to the assembly BOM and is used to override the values to make them different from the assemblies BOM.

To restore the cell value back to the BOM value right-click on the cell and turn off the Static Value option.

To save item number overrides to the assembly BOM right-click the cell and select Save Item Overrides to BOM.

Individual rows can be turned off by right-clicking on the row and turning off the visibility option. The row will still be displayed.

Using the Column Chooser dialog you can apply a style override to the parts list displaying additional iProperty information. Additionally you can reorder the columns within the dialog using the “Move Up” & “Move Down” buttons.

Rows in the parts list can be dragged-and-dropped to reorder.

Similar to the BOM editor, the Parts List editor provides tools to sort and renumber the items. Using the renumber tool within the parts list will override the item number and toggle the value to static as it will no longer represent the BOM item order.
Figure 27: Clicking “Save Item Overrides to BOM” will save all item renumbering to the BOM database.

Accessing the Table Layout dialog allows you to override the style settings and adjust the general appearance of the parts list including the title, text styles used, and the data direction.

Custom components can be created for items that will not be modeled but yet near to appear in the Parts List, for example lubricant, paint, assembly instructions, etc. Right-click anywhere on the rows and select Insert Custom Part to add a new row.

Figure 28: Inserting Custom Parts
Once the custom row is added a balloon can be created within the drawing using this new item.

Review

- Changes to the BOM are automatically applied to the Parts List
- The Parts List can be inserted into the drawing without any drawing views
- The initial setup and look of the Parts List is controlled by the drawing styles and multiple styles can be configured
- Changes to the formatting can be applied after the Parts List is inserted
- Data adjusted in the Parts List will become “Static” and turn blue to signify that it is not the same as the BOM. Reverting this back to non-static will restore the BOM value
**Parts List Column Formatting**

The formatting of each column is customizable. This is accessible by right-clicking on the column heading and selecting **Format Column**. Within the Format Column dialog the heading caption, justification, and unit formatting can be adjusted. This includes switching from decimal to fractional with options to control the fractional format.

![Format Column Dialog](image)

**Figure 29: Column Formatting**

The substitution tab is used to populate the values of the column with a different iProperty. This can include substituting the sum of values opposed to the value of one instance.

This is valuable for standard type components like steel shapes, piping, and tubing where each type has a defined stock number (i.e. how it is ordered) which does not change regardless of the length of the part. The part number however is typically unique for each instance and with steel shapes and piping there could be various instances of the same type of component, just at different lengths. Using substitution the stock number would be assigned to any item which does not have a part number, simplifying the process as individual part numbers do not need to be assigned.

**Review**

- The formatting of each column is customizable (accessible by right-clicking).
- Heading caption, justification, and unit formatting can be adjusted.
- Data formatting including switching from decimal to fractional.
- Substitution is used to substitute one value for another, say to build a cut list of items opposed to rolling the items into a single line item.
Grouping

Within the Parts List you can group items based on “keys” which represent the component iProperties. The grouping is set within the Group Settings dialog box.

For example, with frame generator assemblies you can group items based on its Stock Number (First Key) and (Base Qty) which will group items based on their material type and component length.
Review

- Use the grouping option to group components based on a specific property
- Use up to 3-keys
- Controls on what is shown in the parts list
Parts List Filters

Inventor 2009 introduced the ability to apply Parts List Filters to a parts list to control which items are displayed. These filters are very similar to what is available with Mechanical Desktop and AutoCAD Mechanical.

![Parts List Filters](image)

Figure 35: Parts List Filters

The available filters are Ballooned Items Only, Standard Content, Item Number Ranges (i.e. 1 through 10 and 20 through 30), Purchased Items, and Assembly View Representations.

The “Purchased Items” and “Standard Content” filters have options to either show only those items or to exclude those items.

![Parts List Filter Options](image)

Figure 36: Parts List Filter Options

Level of Details and the BOM

The Bill of Materials will ALWAYS show the Master BOM, regardless of the level of detail (LOD) you have active, but what if you just want to show the components of a particular LOD?

Bob Van der Donck posted to the “Being Inventive” blog the idea of using Parts List filters in conjunction with View Representations. The drawing view is configured using a view representation to just show the desired components. A part list filter is applied to only show the components in that view.

There is a catch… Level of details and View reps are not associative, meaning you need to update both when you make changes to the components in your assembly.
Vault Items

What are Items?

Like a recipe or business card, items represent things within our business. Items represent the components that the company manage, assemble, sell and manufacture.

Each item is identified by its own unique item number (or sometimes referred to as part number). Items can represent a variety of things including parts and assemblies, instructional guides, consumable goods (fluids, lubricants, etc), and any related Bill of Material. Some items might be purchased, while others are manufactured internally.

Inventor Assemblies, Vault Items, and the BOM

Inventor assemblies can be assigned to items, the Inventor assembly BOM will be replicated on the item BOM. Changes to the Inventor assembly will be reflected on the item.

Each item lists a Bill of Materials (BOM) for the files attached to the item. The BOM is a list of components, their properties, the quantities, and how the components are structured.

Similar to the Item Master the information displayed within the BOM is customizable. Right-click the column headers to access the Custom View dialog.

Alternatively you can use the Define custom views option to build saved custom views.
Editing the BOM

Adding Items
Not all components listed in the BOM have to originate in the attached file. Use the Add New Row button to add an existing item to the BOM.

Figure 39: Click Add New Row

Overriding Quantities
The proper quantities of the items are generated automatically using the same units and quantities from the CAD model. There are times where it may be desired to override this at the item level. For example you may want to increase the amount of bolts or fasteners shown on the BOM to compensate for loss when the unit is shipped.

Comparing Bill of Materials
Each revision of an item has an unique BOM as changes to the attached file can impact the BOM structure, the components contained within the assembly and the quantities. Vault Professional provides a comparison tool to compare one revision of the BOM against the other, highlighting any differences between the various revisions of the item.

In the toolbar use the drop-down to select the revision you wish to compare against and click the Compare button. The differences will be highlighted in colour and if you hover over the area a tooltip will present further information.

Figure 40: Select the Revision to compare against
Exporting Bill of Materials

The BOM data can be exported to be used in other applications (think ERP, MRP, PLM, etc) and in a variety of formats typically Microsoft Excel. The BOM can be exported in one of two ways:

- a tree hierachal view showing each sub-assembly and their components
- a flat layout with re-used items rolled up into a single row showing the total quantity (i.e. cut list).
Autodesk PLM 360

Product Lifecycle Management (PLM) is the managing of a product from its conception through to delivery including everything inbetween like design and manufacturing. Not just software PLM will integrate the data, processes, and business practices

Autodesk PLM 360 is a cloud based PLM solution which "Connect provides next-generation, cloud-based integrations that connect legacy systems, enterprise applications or other data in the cloud to your Autodesk PLM 360 platform"

Items can be synchronized with Autodesk PLM 360 using a utility from Autodesk. This will sync released items (and their BOMS) with the equivalent items in PLM 360. For more information visit the Autodesk Wiki