

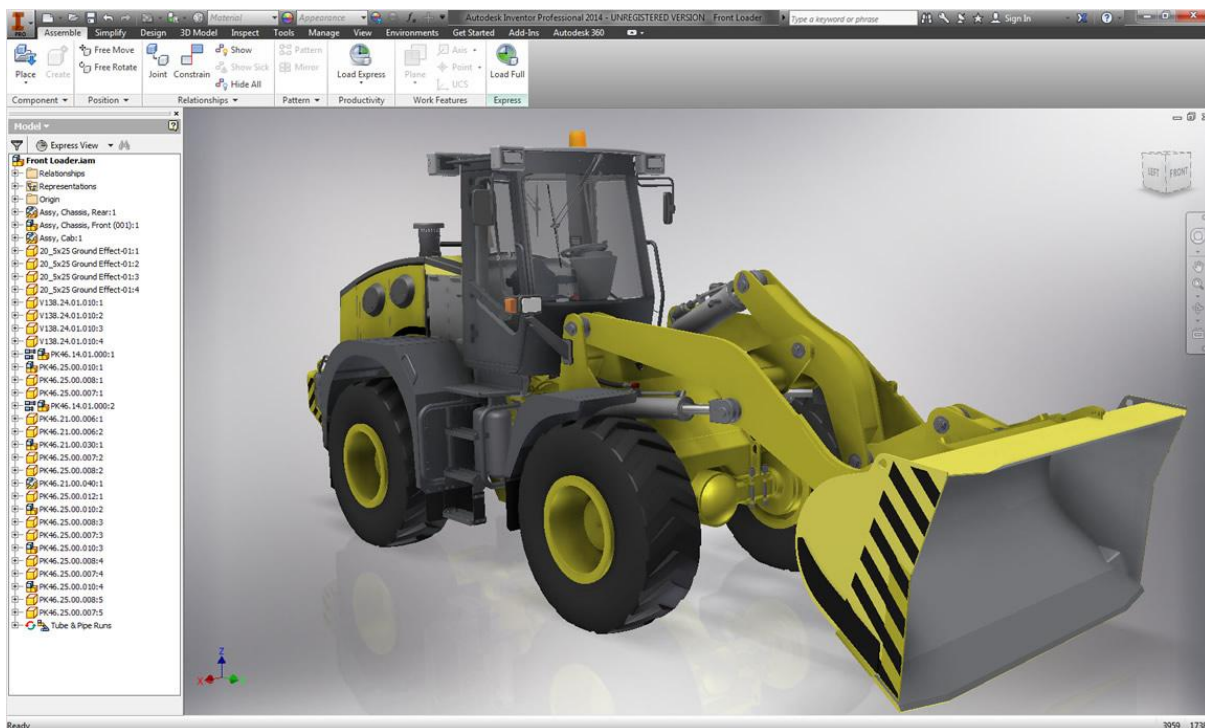
Guide by David Gate

What's New in Inventor 2014

A summary of the new features available in Autodesk Inventor 2014.

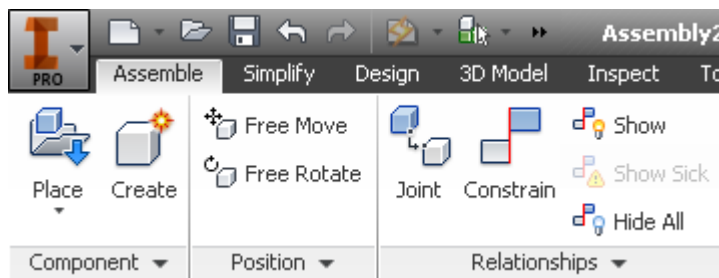
All Inventor Users, Engineering and CAD Managers, Purchasers

Autodesk Inventor 2014 has a raft of new features. First off they have worked hard on improving the performance of Inventor 2014. It runs more efficiently and utilises your hardware a lot better and of course has been made to support Windows 8. All in all this should greatly improve your experience when working on larger or complex assemblies.



The other improvements are mainly based around other assembly improvements but there are also some nice part modelling improvements as well, some of these are based around customer input, so your ideas do count. Below you will find a list and explanation of some of the new features and enhancements.

Assembly Tools



You will notice the new relationships panel. This now contains your constraint and assemble commands as well as a few new commands detailed below.

Joint

The new joint connection command adds a simple way to describe the motion of a component. This allows you to fully describe a components location and motion in one step. These new connections work by selecting and associating end, mid or center points of components.

These connections will appear in a new relationships folder, along with the constraints, in the model browser (whether in model or assembly view, a really nice change).

Rigid (Welded)

Removes all degrees of freedom (DOF). Welded and bolted connections are examples of a rigid connection.

Rotational (Revolution)

Specify one rotational DOF. Hinges and rotating levers are examples of a rotational connection.

Slider (Prismatic)

Specify one translational DOF. A slide block moving along a track is an example of a slider connection.

Cylindrical (Cylindrical)

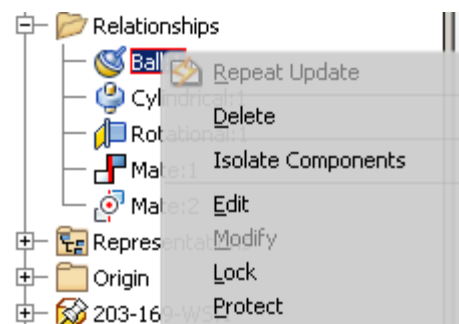
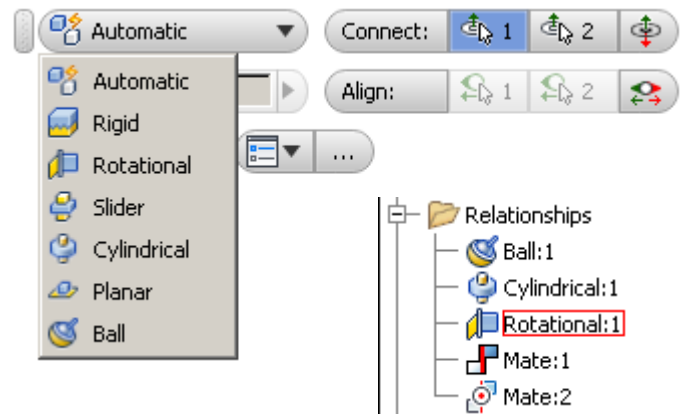
Specify one translational and one rotational DOF. A shaft in a tube is an example of a cylindrical connection.

Planar (Planar)

Specify two translational and one rotational DOF perpendicular to the linear. To place a component on a planar face, use this connection type. The component can slide or rotate on the plane.

Ball (Spherical)

Specify three rotational DOF. A ball and socket are an example of a ball connection.



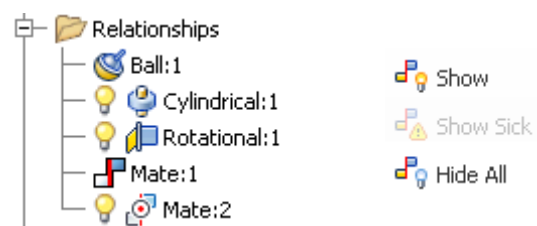
For joints that have one or more degrees of freedom, two options are available on the browser context menu:

Lock maintains the current position, but is different from grounding a component. Lock eliminates all motion, but allows the component to change position when related components move. Grounding eliminates all degrees of freedom, and fixes the component position in space.

Protect alerts you if an added relationship violates the required degrees of freedom.

Relationship Display Commands

You also have the option to turn on and off the visibility of all your relationships. Which can be done in the browser or with the handy show and hide commands on the ribbon.

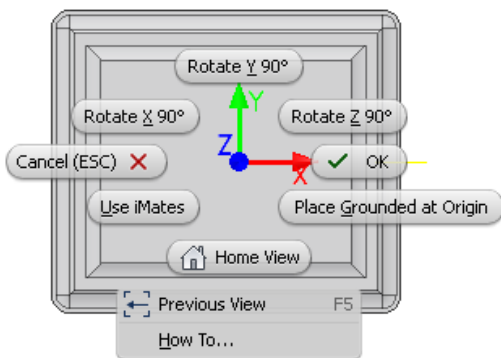
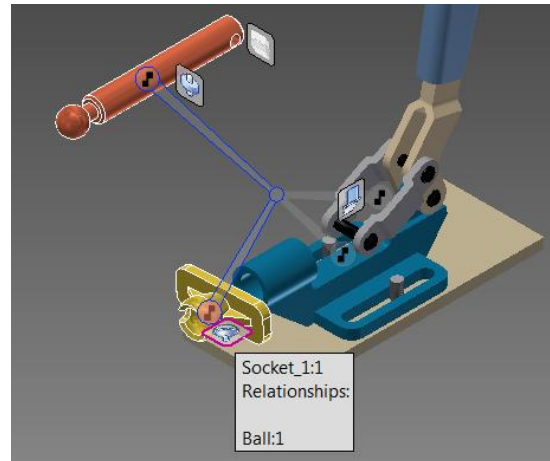


Free Move

You can now free move a component and still see the connections that exist between that component and others in your assembly. This works with both joints and constraints. A really nice way to see how things are connected.

Place Component

The first component placed in an assembly is no longer grounded. To ground any component at the origin, use the context menu. The menu contains new options to rotate components in 90-degree increments about the X, Y, or Z axis, before you place them in the assembly so you can orientate the model first before grounding or placing it into your assembly. Don't worry if you want Inventor 2014 to work like before, there is a tick box to turn the first part placed to be grounded in the application options.



Assembly Copy & Paste

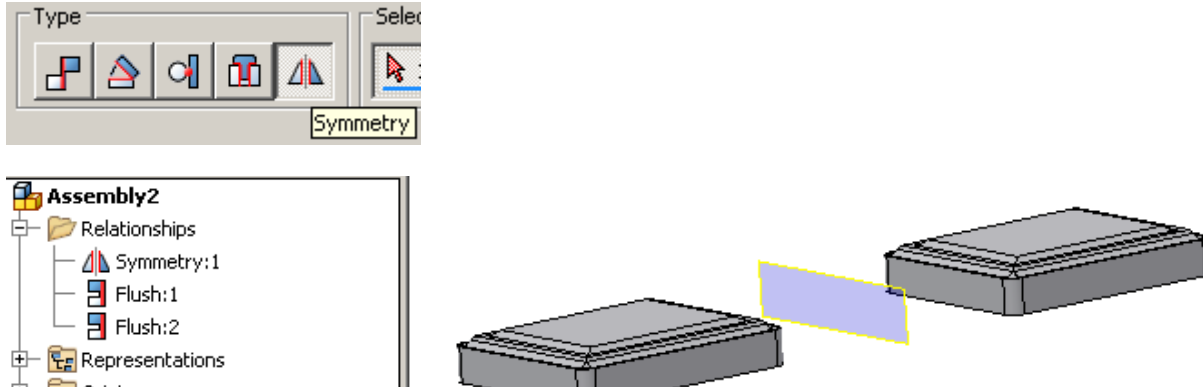
In previous releases of Inventor, a copy and paste operation did not retain component connections or current orientation.



The enhanced copy and paste operation produces a duplicate of the copied components with connections and orientation intact. A huge time saver.

Symmetrical Constraint

Two components can now be constrained in an assembly as symmetrical about a plane/face.



Express Mode

Inventor now includes something called Express Mode. If you work with large assemblies then this will be a must use feature. Once turned on and the unique threshold set, an assembly will open in a fraction of the time (4x-6x quicker) compared to any previous version.



Express mode uses a lightweight representation of your full assembly, allowing you to load it super fast, one example assembly with 7500 parts and 1500 of them unique, used to take 2mins in Inventor 2013, with Inventor 2014 it only took 11secs! You can constrain and apply joints between components while in express mode and then if you need any other functions you can use the 'Load Full' to exit Express mode and enable all commands.

Improved Graphics Performance

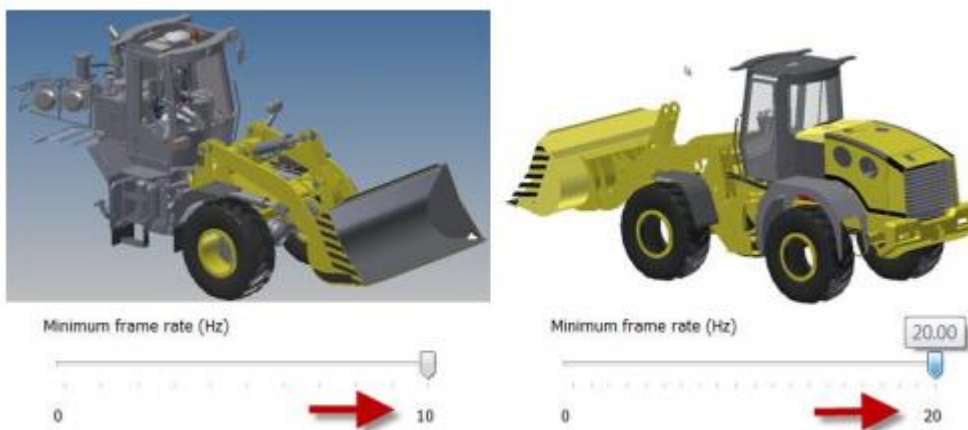
The following graphics enhancements improve the performance of pan, zoom, and rotate of large models in both Full and Express mode.

- The maximum frame rate is increased from 10 Hz to 20 Hz.
- The largest objects are drawn first (now when you rotate smaller parts will disappear first if performance starts to degrade rather than any part at random).
- The graphics system now uses multiple cores to provide the best possible performance.

- A new background operation called consolidation is added. Consolidation draws objects have the same appearance as a single group, improving the GPU usage. So setting the appearance of your components can have an effect.

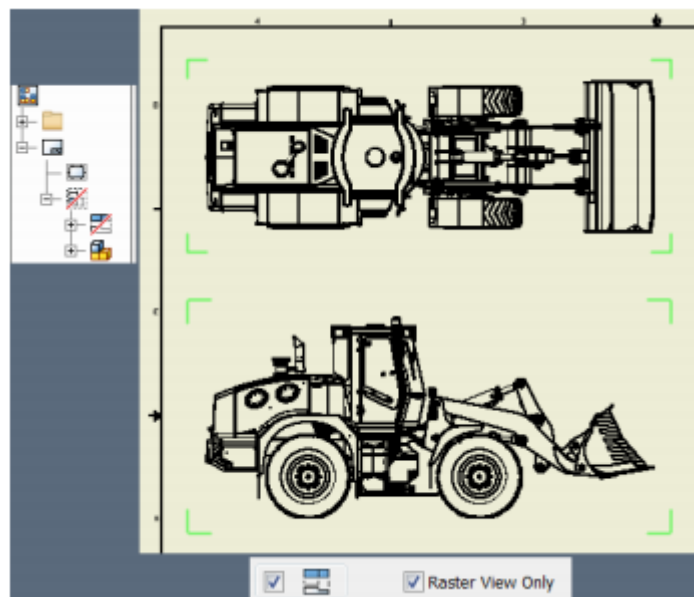
The following images, the 2013 model on the left displays the results of rotating the model using the maximum of ten frames per second. Increasing the frame rate increases the speed, but the choice of components to draw is not intelligent.

The 2014 model on the right shows the results of rotating at the faster maximum frame rate of 20 Hz. The model remains recognizable, and all view operations are faster.



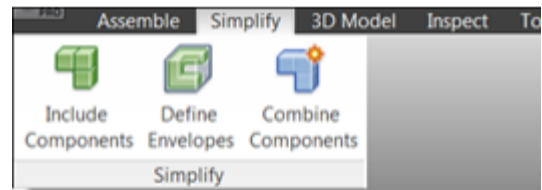
Large Assembly Drawing Performance

When placing views in a drawing Inventor they will start them as Raster Views (you may not have noticed the little green corners to your views if they load quickly). But these views will load and fill out and become more precise until they have fully loaded. Sometimes you don't need the full detail and you don't want it loading constantly in the background so you now have the choice to leave these views permanently as Raster Views.



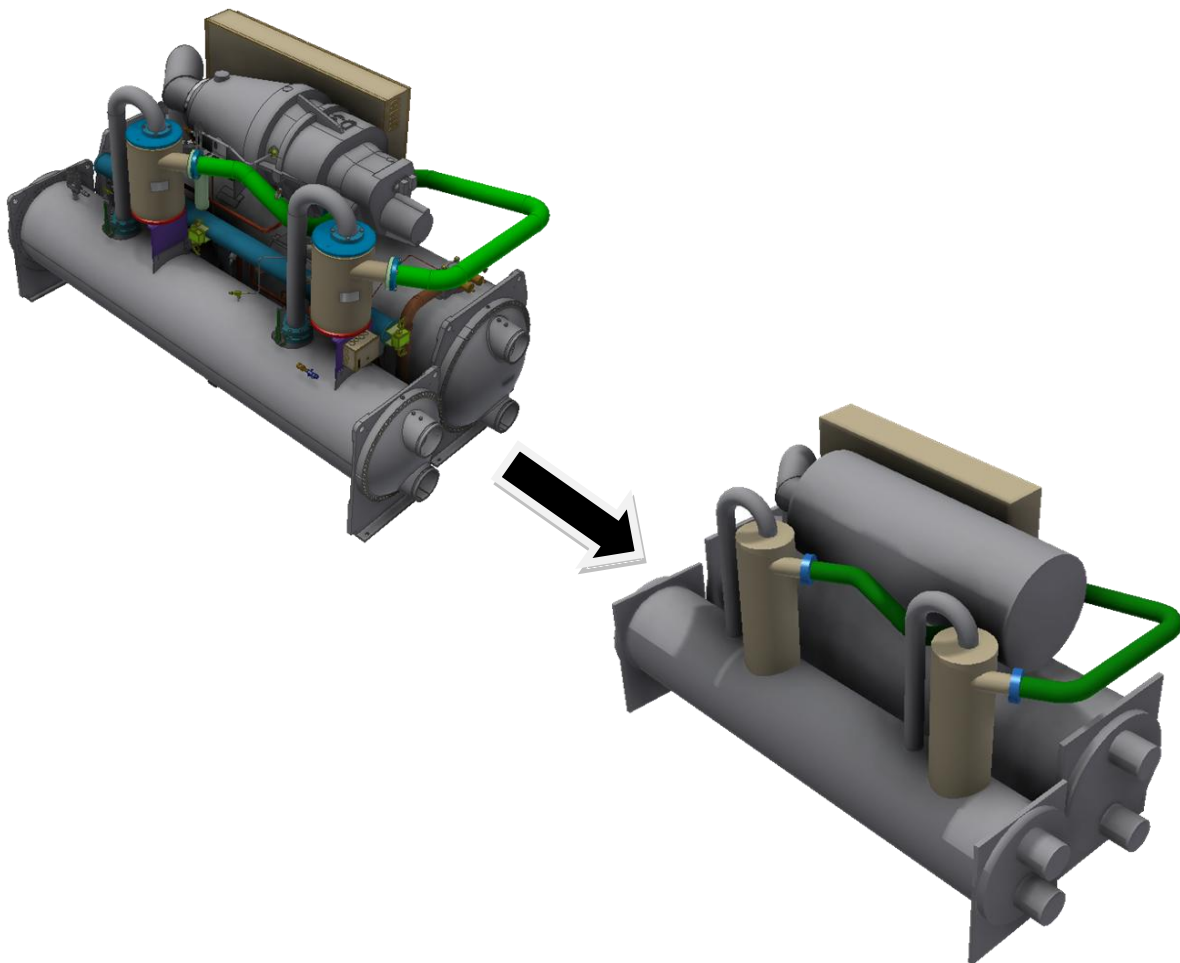
BIM Simplification

You may have used the Simplification tools available from Labs but these have now been integrated into Inventor.

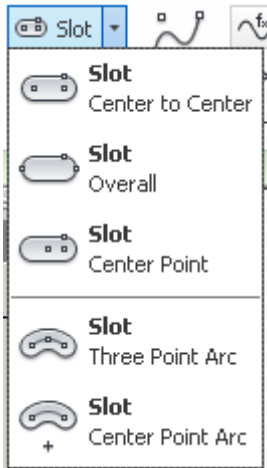


These tools allow you to simplify down your model for sending to a customer or for use a substitute, by removing unwanted detail and defining envelopes that represent simpler forms than the complex models. A great way of showing an assemblies space envelope with giving away you interlectual property.

These tools have also been expanded on and you can now take your simplified part/assembly and convert it into a native Revit Family member. This will no longer be a dumb solid but you can recognise features and parameters which can be taken into Revit and utilised. Hugely important if you are working/supply products going into buildings. In the coming years BIM is going to hugely take off and the manufacturing industry will be included.



Part Modelling Improvements



There have been a number of part modelling improvements as well as all the above assembly improvements. The most useful are again outlined below.

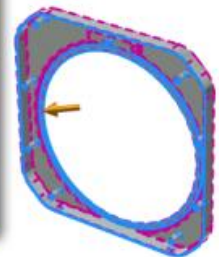
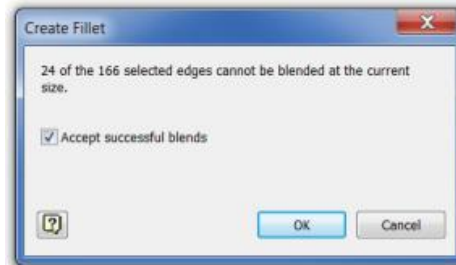
Sketch Slots

In the sketching environment you can now draw a few different types of slots quickly and fully dimensioned. These new slots should save you some time, instead of having to draw a minimum 3 geometries and dimension them you can place slots with a few clicks.

The dynamic entering of dimensions also can be done here so you can type your sizes as you go and the dimensions will be auto created for you.⁷

Fillet Improvements

The fillet command has been improved so that if one edge fails the rest of the fillet will still stay intact. This is particularly useful if you edit your base model and in the past your whole fillet may fail now only the edges causing a problem will have their fillets removed.

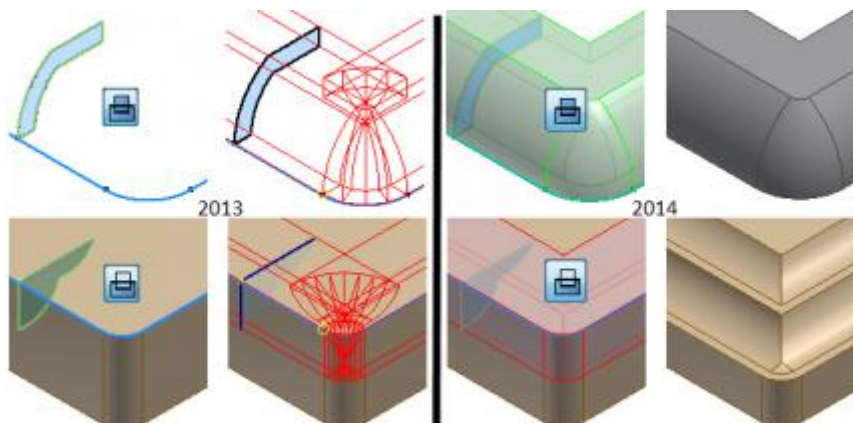


You get a nice prompt and display if any fillets do fail and you can choose to accept the failures or go back to edit the geometry.

Self-intersecting sweeps

With the enhanced sweep command, you can create sweeps that were previously considered too complex to solve. You can sweep large profiles along small fillets and bends.

The following image displays examples of previous sweep failures on the left, and successful 2014 sweeps on the right.

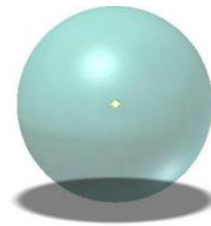


Non tangent surface path sweeps

Previously, creating a swept surface using a profile that was perpendicular to a closed G0 (not smooth or tangent) path failed. New sweep functionality lets you create a surface using a perpendicular profile along a closed G0 path.

Work Point on a sphere

In previous releases, a sphere was not a valid selection to locate a work point. You can now define a work point at the center of a sphere.



Drawing Enhancements

As already mentioned large assembly views can have their precise and raster view options controlled to enable quicker operation.

A few other options have also been added at the request of customers.

Split tables

You can split a parts list, hole, revision, or general table, and then relocate the table segments to the desired place on the drawing sheet.



To split a table, right-click text in a table row, and click Table Split Table. The table splits into two segments. In the browser, the new table segment displays as a child of the parent table. To cancel the table split, right-click a table segment, and click Un-split Table.

Edits to the parent segment of the table are promoted to child segments. Changes that you make to a child segment do not affect other table segments.

Appearance property

The model Appearance value is added as a selectable property. You can add the Appearance property to a bill of materials, or a parts list.

| PARTS LIST | | | | |
|------------|-----|-------------|----------------|--------------------|
| ITEM | QTY | PART NUMBER | DESCRIPTION | APPEARANCE |
| 1 | 1 | BP-100 | Base Plate | Metal |
| 2 | 1 | SB-101 | Slide Base | Sky Blue Dark |
| 3 | 1 | SBE-103 | Shaft Ball End | Metallic Paint |
| 4 | 1 | C-100 | Connector | Metal |
| 5 | 1 | L3-100 | Lever | Smooth - Off-White |
| 6 | 1 | S104 | Socket | Nickel |