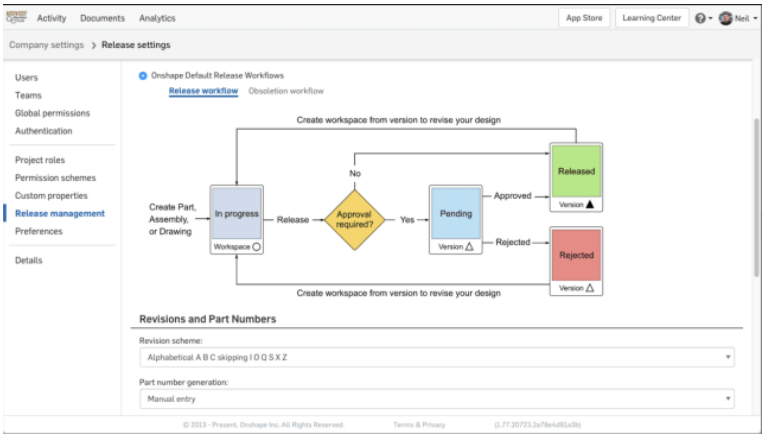
(https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/16672/Onshape-Adds-Release-Management-Revamps-BOMs.aspx#disqus_thread)



Onshape now offers a formal release workflow process. (Image courtesy of Onshape.)

"We set out to build a new generation of modern CAD," proclaimed Jon Hirschtick, co-founder and CEO of Onshape, the six-year-old start-up. This statement could be dismissed as hot air were it not uttered by the founder SOLIDWORKS, the current generation of CAD. From its roots as the first cloud-based CAD platform, Onshape has shown it's not afraid to implement new ways of doing things. Earlier this year, Onshape unveiled its own spin on parametric modeling (https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/16587/Whats-the-Difference-Between-Parametric-and-Direct-Modeling.aspx), which it calls Parametric Modeling 2.0 (https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/16304/We-Just-Remade-Parametric-Modeling-Says-Onshape.aspx). Now, Onshape has announced its take on another important aspect of CAD: design data management.

Onshape refers to its data management tools as a "modern reboot," and has styled this reboot as Design Data Management 2.0. The concept incorporates six key aspects, four of which have been in Onshape for a while:

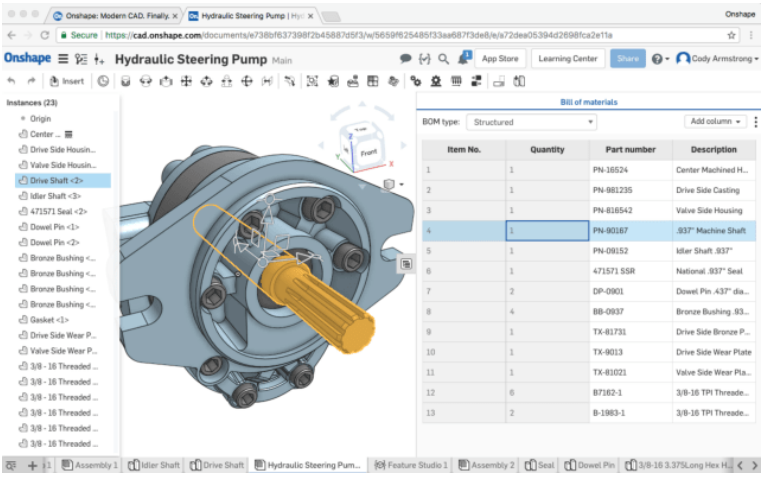
1. Search
2. Versions and History
3. Branch, Compare and Merge
4. Share and Comment

And two of which are new additions to the platform:

5. Simultaneous BOMs
6. Formal Release Management

All six of these aspects will seem familiar to CAD users experienced in product data management (PDM) tools, so why does Onshape consider them to be a reboot? Let's explore that with a deeper dive into the two newest tools of Onshape's Design Data Management 2.0.

Simultaneous BOMs



Onshape's new Simultaneous BOM alongside a hydraulic steering pump assembly. (Image courtesy of Onshape.)

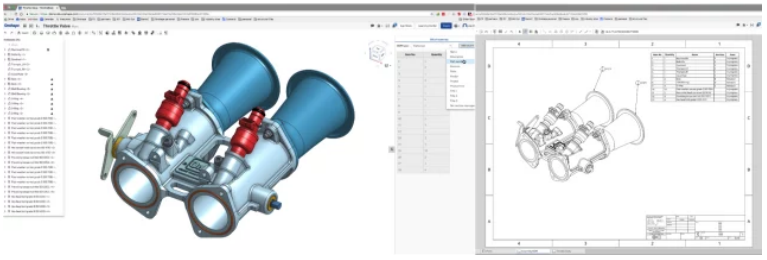
A bill of materials (BOM) is a necessary component of any design intended for production, as it itemizes exactly what parts are needed and in what quantity, along with other useful information such as a description, part number, etc.

Though Onshape has had BOM functionality in the past, it suffered from what Hirschtick claims is a problem of most CAD systems: BOM tends to be nothing more than a report generated after the fact. While he admits that some external PDM tools have additional BOM capabilities, CAD systems typically only offer BOMs as an output artifact on a drawing.

Onshape's new Simultaneous BOM tool reimagines the BOM not as simply an artifact of a CAD assembly, but as a first-class object—hence the "Simultaneous." Onshape now allows users to edit the CAD geometry and BOM together in a fully associative way.

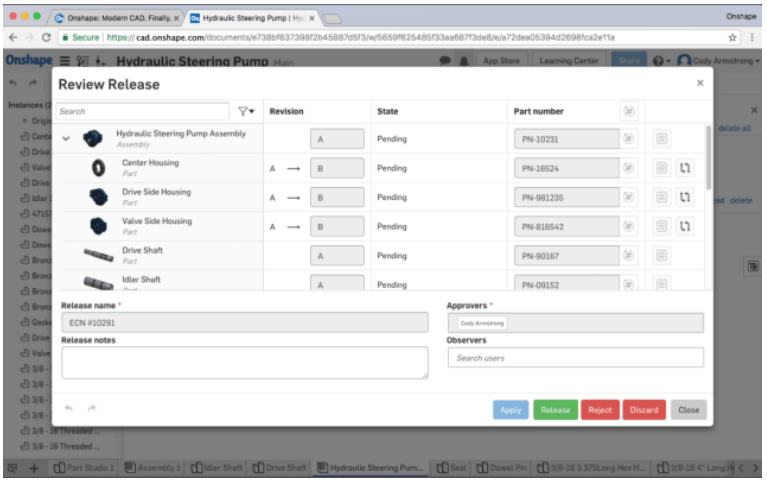
"In Onshape, for the first time, we allow you to work on the geometry of the assembly and the BOM simultaneously in one window, and all the operations made to either are associative in both," Hirschtick said. "When you're creating the assembly geometrically, you can not only see the BOM but also use it to edit and structure your assembly information in a way that we think will make people much more productive at creating assemblies that are not just geometric assemblies but have all the structure the metadata that you want in the BOM."

One of the most common uses for BOMs is in drawings, so Onshape naturally enables drawings to incorporate the Simultaneous BOM. The BOM is associated with annotations like part number balloons, and any and all changes to the BOM are automatically updated in the drawing.



Onshape's Simultaneous BOM is automatically updated in drawings to reflect changes. (Image courtesy of Onshape.)

Formal Release Management

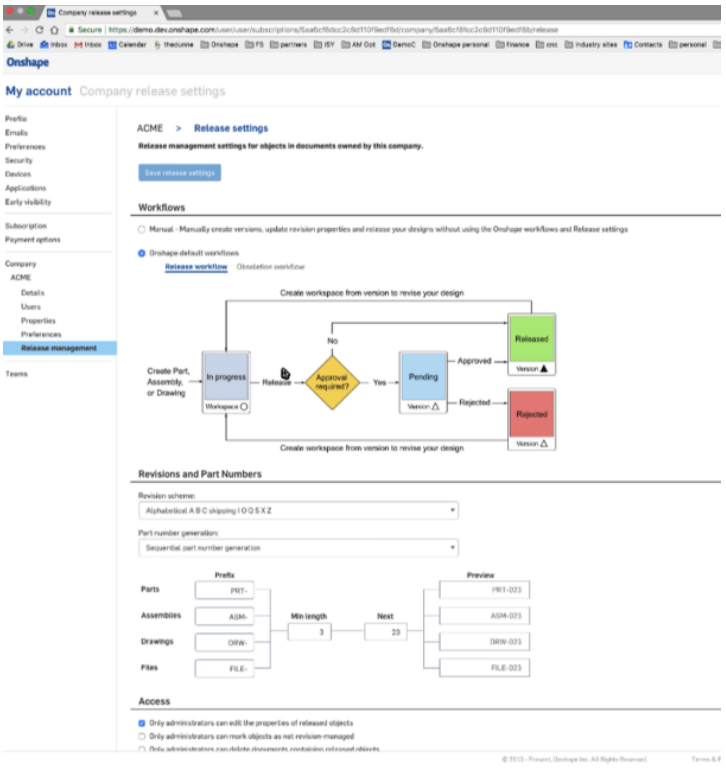


Reviewing a release in Onshape's new Formal Release Management tool. (Image courtesy of Onshape.)

Onshape is offering a new tool catered to professional CAD users requiring a robust release process: Formal Release Management. This tool gives Onshape users fine control over formal releases, including revision states, revision control, and a new approval workflow with notifications.

Although Formal Release Management is similar in functionality to Onshape's existing version control—which is basically GitHub (a software development platform) for CAD—it offers a professional level of rigor for companies that require a formal release process, with automatically generated part numbers, approver and observer settings, and more.

"This is separating the boys from the men," said Onshape's Joe Dunne. "People using this in production need this. Over the last year or two this is what our production users have been asking for."



Example company release settings for Onshape's new formal version control. (Image courtesy of Onshape.)

The 2.0 element of Formal Release Management, true to Onshape's cloud-based approach, is that the release management process doesn't impede active development. Designers need not wait on approval of a release, as they can simply branch a model and merge changes later. In Hirschtick's opinion, this approach combines the best of two worlds.

"It provides the freedom of uncontrolled data with the rigor of controlled data," he said.

Onshape Levels Up

While the new Simultaneous BOM feature will be available to all Onshape users, the Formal Release Management tool is planting a flag for Onshape in the form of a new subscription tier. Until now, Onshape has been available at two levels: free, in which all features are available, but all documents are public, and paid, in which commercial users can create private documents.

Now, Onshape is introducing a second level of subscription: a premium tier targeted at companies that require administration tools like Formal Release Management, which will be exclusive to premium users. The new tier is $2100/year, $50/month more than the $1500/year standard. If Onshape has correctly gauged how important Formal Release Management is to its users, the price hike should be well worth it.

You can read CEO Jon Hirschtick's blog post about Design Data Management 2.0 here (https://www.onshape.com/cad-blog/introducing-design-data-management-2.0?hs_preview=syjPlaCN-5644750138). To learn more about Onshape's approach to CAD, read We Just Remade Parametric Modeling, Says Onshape (https://www.engineering.com/LinkClick.aspx?link=%2fDesignSoftware%2fDesignSoftwareArticles%2fArticleID%2f16304%2fWe-Just-Remade-Parametric-Modeling-Says-Onshape.aspx&tabid=6551&portalid=0&mid=429).