AU 2020
Ask the Inventor Developers Q&A Addendum

With the large turnout for the Ask the Inventor Developers Roundtable at AU 2020 we were unable to answer all the questions that were asked. This document is a summary of all captured questions and either a repeat of the answer given live or an answer given post-class.

Speaker: Steve Dennis
Co-Speaker: Yun Chen
Co-Speaker: Michael Schmidt
Co-Speaker: Gerald Hochenauer
Co-Speaker: Tom Sturtevant
Co-Speaker: Shiva Sundaram
Moderator: Chris Mitchell

A few of these questions were raised in the online comments section of the class page, https://www.autodesk.com/autodesk-university/class/Ask-Inventor-Developers-2020, most were asked in the class (see the recording), and a few were received via email after the class conclusion. The questions are in some cases summarized for brevity.

Please keep in mind that any answer given in the class or here does not convey a commitment on Autodesk's part. The answers are the opinions of the speakers and while we do have some influence, other inputs are used for prioritization and actual engagement on these issues such as business or resource concerns.

Jason Mayes - A part or sub-assembly pattern cannot be moved into an existing folder. Are you aware and if so, are you looking into this?

Yes, we are aware of this and every year we take a new look at old browser folder restrictions. Component patterns should be able to exist in a folder seeing as they do not have any compute order issues related to browser order. We would hope to be able to address this.

Ivan Peeters : When doing casting design I really miss the folder possibility's from assemblys as the features in the browser is always a very long list. This makes it very hard to find your way in the structure

Andrew Burnett : If you create the folder and have the features still in order it would help with some of our complex parts.

JoeMorin : SolidWorks allowed folders for features in parts but the order was still kept the same. It could be confusing because the folders could be moved around, making the feature order look different from reality...

Robert Windsor : folders in part tree would be awesome! would be able to group features per iPart versions...

Matt : Browser Folders in parts: https://forums.autodesk.com/t5/inventor-ideas/browser-folders-in-part-environment/idi-p/3780620

Folder is parts is a long-requested item that has been scoped because of the compute ordering that our parametric world imposes from the browser order. We have tried multiple times (I have personally tried twice) to supply a browser that does not have any link to compute order and we always run into insurmountable problems. One suggestion that some users agreed with (even in this class) was that maybe we could still give you folders in parts (for features) but the overall order of the features must remain the same, such that you could not move all the fillets to the top for example. This might be a first step in allowing folders in parts.

Niels Schotten – Asked one question about doing the fit and finish on some areas of Inventor that have not had much work done in recent years like bolted connection. Another question was about the open/save dialogs returning to the previous folder location even if the project had been switched and he suggested several solutions to help with this issue.

We would like to stress that suggestions like this are best served by being placed on the Inventor Ideas Forum (https://forums.autodesk.com/t5/inventor-ideas/idb-p/v1232). We explained that while the Ideas Forum may appear overwhelmed and underutilized, we do use it as a primary input in our prioritization of smaller projects for every release of Inventor. There is no magic vote count to obtain before we will implement a suggestion, but it is used in our decision making process for future projects. We discussed also other avenues such as resellers, product support, or events like this class for making your suggestions known and visible.

Niels Schotten – Also asked about Tabs and Slots in sheet metal and non-sheet metal parts.

This is a project that is becoming more visible to us based on customer interactions and is being considered for a future release but will need to be prioritized against other deliverables based on cost and usage.

Pascale Brassat – Why are some Inventor features or entire AddIns such as frame generator not accessible via the API.

A bit of history, when R1 (and maybe R2) of Inventor was rolled out we had NO API at all! We are aware of the holes in our API and every release we try to close some of the holes. As Inventor Automation continues to grow we will need to close these holes for sure. The reason they were not part of the initial releases for example probably come down to cost of implementation (deliver the feature without an API in Release X or don't release it at all?) and low utilization. We fully expect automation to become more important to all users and we will address these missing APIs.

JoeMorin - What does it mean when the Adaptive entry on the context menu is grayed out? Why can't I break or set that to be Adaptive?

Adaptivity controls how a part (or parts of a part) can be computed in the context of the assembly variational solver. While a part can be inserted into an assembly multiple times and into multiple assemblies only one instance can control the adaptivity of a part, having more than one try to control would result in contention and/or infinite loops! Sometimes depending on the workflows used (delete an assembly that has the adaptivity control) a part might also not have anyone in control but it still knows it was made adaptive. So the adaptive command is grayed out when someone else has already set that part to be adaptive. If the part is marked adaptive but you are unable to find the controlling parent assembly with the adaptive instance you can clear the part adaptivity by going to document settings:

You can uncheck that and you will then be allowed to set another single instance to be the one controlling this part's adaptivity.

Ethane – Why does the recover function require so many clicks? More than one issue returns you to the start and requires just as many clicks

> *abauman - Why does Inventor generally require so many CLICKS IN GENERAL!*
>
> *Matt : On people speaking of a lot of clicks being required in general, is there a way to make a space or enter stand in for a right-click "continue" or "apply" when mid-command.*

We believe this (ethane) is referring to the design doctor functionality and there is a lot of room for improvement here, including some auto expansion as pointed out by [Sbix26 - Errors always display with the details collapsed-- typically takes four clicks of very tiny + marks to expand to the final details about what went wrong. Why not expanded by default?].

We would like to see some specific suggestions on the Ideas Forum. The original design of Inventor included usage of backspace, spacebar, enter, etc. to "navigate" a command. This design always suffered from focus issues such that the keypress was not received correctly to be used. So there is no

grand unified way of having this work today, but default buttons on a dialog or panel should respect the Enter key like any Windows application.

Matt - Have you considered adding Bolted Connections to "File Naming Defaults" in Application Options. Have you considered adding access to these new file naming defaults via the API?

Short answer yes and yes. Each of the last few releases have seen new file naming functionality and since they are popular we will continue to add them, please help us prioritize the areas you would like to see by utilizing the Ideas Forum.

RobZazuliak - In an idw why does "copy properties" not work on hole dimensions?

This sounds like a bug or oversight. Our QA and CS teams will try to assess this for inclusion in a future release.

to7176 - In Mechanical Desktop you could generate a body of the collision between two parts. It was possible to work with this collision body like with a part. You could generate a part and you could measure all dimensions. In inventor it is not possible to generate such a body with collision analyses. Additional sometime it is not easy to find the collision bodies in a big assembly if they are very small.

This was on our list of improvements when we did our interference projects a few releases ago and due to cost and prioritization did not make the cut for inclusion. Part of the technical issue is that we don't support local bodies in an assembly currently. The logical solution to this is to have a local body only exist in that assembly that represents the collision and we are not currently architected to support that.

Joe Kniffin - As a cad manager I have to keep track of the templates for the different drawing styles and sizes. Is there any plans or even a current way to use Vault to maintain these?

Dan Silver : I have my templates and design data vaulted and it works well

Many users use Vault today to maintain their entire Design Data folder (XML files, templates, etc.)

to7176 - If you color by Representation (not by Material) it is not possible to identify the different parts in a sectional view at the cut surface.

We clarified that he was referring to the cut looking graphics we put in the graphics. Today you can do the default gray mottled appearance we do today or you can specify your own bitmap image to use. I do

see the component color is opaquely applied to the bitmap, maybe he was referring to the face color which is not available in assembly today. We need to investigate this more but we also suggested this was a fine Ideas Forum post.

David Coulam - Favorite bug: accidentally clicking the big "I" twice closes the application.

I thought we had addressed this in a previous release but if we did it's back! This is standard Windows behavior for any application that does have an icon/menu in the upper left corner (see notepad, Visual Studio, etc.) Some applications do not put an icon/menu there to avoid this it seems (Outlook, Word) but even if the icon is not there double clicking in the upper left blank area (to display the move, resize menu) still closes the application.

Justin Perry - Aligned breaks would be helpful.

This is actually one of the higher vote getters on the Ideas Forum (https://forums.autodesk.com/t5/inventor-ideas/add-quot-aligned-orientation-quot-option-to-break-view/idi-p/4422149) We are aware of this and due to resourcing it never seems to make the final list for inclusion into a release. We are very aware of this limitation.

Anirudh Ashok Kumar - Is there a way add a feature that allows the user to project a folded view from a flat pattern view on a sheet metal drawing?

There is nothing on our radar to address this (treat a flat pattern view as a projected view from the folded view or vice versa). We did discuss some better workflows we could create for making the flat pattern view in the first place and/or linking the two base views together.

abauman - Is there any work being done on better developing the sketch tools?! The Trim feature is pretty horrendous - sketch relations will get deleted - especially when using Offset tool and then trimming afterwards.

We continually evaluate improvements to sketch as well as the integration of sketch into the other tools like extrude, etc. Specific troublesome workflows can be relayed to us via the usual means (forum, reseller, etc.) Be as specific as possible for those improvements that would give you the most benefit.

Martin Winkler - Will there be enhancements and improvements in the Inventor Parameter Manager? A wish of mine is to be able to open the manager in non-modal mode and to switch between documents.

> *Matt - Parameter manager improvement is one of my top choices as well. Better equation editing support, especially for long equations.*
>
> *Matt : This applies to dimension editing as well. Providing more space for editing equations, as well as possibly a side panel of selectable in-document parameters would also speed up dim editing.*

A modeless parameter manager has not been considered for a while and keeping dialogs/panel such as that up to date are a challenge. There are Ideas Forum posts on the parameter manager where this can be discussed and clarified. This is not on our near-term radar. We do fully understand the issue here though. As to edit boxes that might not be long enough to accommodate all of the needed text please let us know specific cases.

to7176 - I am using Inventor by another language then English. For following explatations/tutorials in English it would be useful to have a language switch in the program, which translates the buttons into English and back after a switch.

We are not considering an in-session language switch to the best of our knowledge, you may install multiple language packs and restart Inventor with a different language as often as you like.

Anirudh Ashok Kumar - Why is it not possible to change dimension style on a model sketch? It would be helpful when retrieving dimensions on a drawing

There is currently no Styles UI in parts or assemblies. As 3DA/MBD/PMI becomes more accepted in the industry this will certainly be on our list of improvements. Styles improvements also tend to be expensive resource wise.

Henrique - I would really appreciate something inside the workspace that would help keep track of the dimensions I am working with. Something like a Bic Pen to compare my model with.

I don't understand this request but sounds like something you can add to Ideas Forum for others to vote on.

FlexDRG - This also is problematic with demote. it won't take bolted connections and patterns.

Matt : Demoting a pattern is something we've been wanting at my company for a while as well.

Matthew Zawiski : Any content center files cannot be demoted as of 2020

We did not discuss bolted connections and I suspect they won't be allowed to be demoted given the nature of them. We did successfully demote a component pattern in the class, to avoid failures you need to take any components that have directions defined on them as well. Likewise, any parameters used would have to exist in the destination sub assembly or you will get errors.

Content center files have always been allowed to be demoted unless they are part of a design accelerator like gears, bolted connections, etc. If you have a specific case please send it out way.

Steinar - After 20 years With Inventor, and still not be able to save face colors in my ipt seperated in different Design Views is the bug of all years. When did will this functionality finally be added? All parts is machined somewhere and we want to show it shaded

There actually is a technical reason for this, face color as it was implemented in parts (before design views showed up in parts) computes like a feature (internal architecture). Design view "things" like color and visibility specifically cannot be allowed to cause a compute of the assembly or part, they are aesthetic attributes only. To implement this, we need to change how face color works, migrate the old ones and move forward. Having said that we are revisiting face color as part of a future project.

abauman - OH! Why can't I drag Work features in an assembly to the top of a tree? I can manually move all parts below it to move it up, but I can't drag the feature in the tree to better locate assembly working feature components.

We are trying to improve upon the browser dragging whenever possible, several years ago I fixed the bug where virtual components could not be dragged and this is a similar oversight that needs correcting. You have identified the workaround but that just proves there's really no problem with supporting dragging on the work feature itself.

AaronBreaux - Horizontal constraint; If I am in the XY plane a horizontal constraint works as I would expect, along the XZ plane but if I am in the YZ plane then horizontal constraint acts as vertical constraint. Intended or bug?

abauman - I just wish the sketch plane triad would always match the assembly or part triad - I hate having 2 different triads - why are the different? Make them the same in the part file...

Greg Messer : I just wish the sketch plane triad would always match the assembly or part triad - I hate having 2 different triads - why are the different? Make them the same in the part file...

What is actually happening is the horizontal constraint always aligns with the X axis of the SKETCH coordinate system and that is not always the same as "horizontal" on your screen. You can redefine the sketch coordinate system, use UCS to define your sketch plane, or change Tools -> App Option -> Display -> Look At Behavior -> Align with Local Coordinate System. The last application option will leave X in the X direction but it might be pointing to the left rather than the right. We have tried to address this multiple times and we never seem to get it right for all users.

Alan Acheson - Favorite bug, Sorting fractions, need to switch to decimals to sort.

This is clearly a bug. We will take note of it.

Dan Silver : I have a drawing template in which I populate some of the iproperties such as Revision Number. However when I use this template the revision number field is empty. Is this expected behaviour?

We tried this live in the class and it seemed to work although my 2021 version was ahead of Dan's 2020. He will retry this in 2021 himself.

ethane : INV issue: In sketch, sometimes INV will think it is fully constrained when it is NOT. Other times, it will change color to say something is constrained, and it will allow it to move by drag. This is very confusing.

For a case like this you need to get the case to the development team via the forum, product support, reseller or direct contact. If we have your data showing this bug it makes it much easier to find and fix.

Andrew In't Veld - Why does the browser "lock up" sometimes when an iLogic rule throws up an error?

Likewise, we need to see this as a reproducible case. Lock up can mean many things and each one is unique.

Steinar - When will Cable Harness module be freshed up and modernized?

We have done some modernization in Frame Generator based on usage statistics and cable and harness will be evaluation for future improvement.

Steinar - When will Inventor Studio be freshed up due to render performance?

I am personally unaware of any work being done to Inventor Studio.

Andrew In't Veld - Why does it take so long for the styles and standards editor window to come up in the drawing environment?

We are locating and loading multiple XML files, parsing the XML (likely the largest issue), migrating the XML if necessary etc. It does just take time especially if you have your design data on a network drive. Moving them locally should help somewhat.

Matthew Zawiski - In a DWG is there a way to copy either note text and/or sketch symbols from one sheet to another so that they remain in the exact same location from sheet 1 to sheet 2? Simarily revit has a feature where you can copy a symbol or text box to the clipboard and then go to your next sheet and paste it so that it is in the same spot as the previous sheet.

I just saw some changes to how views are copied from sheet to sheet and about how we use the position from the source sheet or the cursor position, but I am unaware of anything like this for notes or symbols. Please add an Ideas Forum post for this. You can also copy the entire sheet and maintain those positions.

JoeMorin - Why do I see phantom lines of hidden items in assemblies when I zoom out? (in shaded with edges mode) Those edges disappear when I zoom in to a certain level...

Please record a video showing this to us, it could be an out of date driver or a bug that we can address, we have not heard of or seen this.

Martin Winkler - The functionality for evaluating point clouds is currently very limited in Inventor. Will be more functionality implemented in future? For example, detection of pipes or steel beams

We are not aware of any near term improvements here but we are doing some refocused work on interoperability with buildings via Revit, etc. I am not sure if this will result in point cloud improvements or not for Inventor.

Henrique : From the performance point of view, is it really better to use professional graphic cards than consumer graphic cards?

Our graphics card expert said that we don't actually do performance testing of graphics cards and the variability in architecture, GPU memory, cores and speed would obviously affect that tremendously. If the cards are roughly equivalent in all those things we don't expect any performance differences really. For Inventor, we don't limit the cards to the professional cards, any consumer graphics cards meeting our minimum system requirement can be used for running Inventor. However, we just don't list them in the certified website, since the vendor doesn't certify them.

So, from vendor's perspective it will always be good to run with professional cards, and it always can get better support if there's any issues in driver side, but your budget and card specifications can drive your decision.

Mike Hilvers - Why is it when you have an IAM open and then in the Vault program itself, not the Add In, find a file an ipt and drag the file into Inventor and the IAM, it actually creates a Temp file and gives you no warning that it's out of your Project folder structure. You find it when someone else opens the IAM or you clean out your temp files. Is there something with that scenario we can do to keep guys from doing that, or to make it not drop the file in or at least warn us.

Using this workflow is roughly the same as dragging in a file from file explorer. Our Vault experts suggest not using this workflow and instead focus on other workflows like:

1. Assumption that active Inventor project is a Vault project
2. With the assembly open, select the "Place from Vault"
3. Log into Vault
4. In the "Select File From Vault", perform a search or navigate to the desired file
5. Click "Open"
6. Place file in desired location

OR

Perform a Get operation on the file inside the Vault. Back to Inventor, use the "place" option and navigate to the file.

Lars van der Greft - You can set a default template for "Make Components", but it doesn't work with a sheet metal template.

Sounds like a bug (I see you emailed me the bug # 😊). This is not currently marked for our current release. Passed onto appropriate developer.

Mhanson - Can we please get some iLogic snippets to work with our files in Vault?

We are always evaluating adding snippets.

Matt - Speaking of adaptivity, has there been discussion of managing adaptivity from an assembly level for all child components? Much like a representation?

No this is the first time I have heard such a request. We would need to understand the desired workflow better.

Alan Acheso - Tube and pipe can we get some more training about the workflow each industry uses, e.g rigid welded spools, formed piping fittings, from part creation to assembly creation down to detailing drawings. The whole work flow understanding is a mammoth undertaking.

I would strongly suggest you reach out to T&P users on the forums. Several of the Expert Elite use T&P extensively. One, Chris Benner, is now even an Autodesk employee who helps manage the forums.

AaronBreaux : Can we get fusion's slick feature browser window? :D

Probably not anytime soon! We are continuing to see what synergy we can get between Fusion and Inventor but that is a big topic we can't cover here.

David Coulam : Why are hole dimensions the only dimension type that can group dual values together in dual format dimension styles? (Styles and Standard Editor/Dimension Style/Alternate Units/Dual Format)

Other dimensions should allow this if you edit the style to allow it. If you cannot edit your local dimension style to allow this please let us know.

Lars van der Greft : Will it ever be possible to edit parts directly from the assembly? Like Solidworks or the Autodesk app "featureMigrator" from CoolOrange. So you only need one hole feature to make a hole in multiple parts

We have talked about allowing this in Inventor where you can create an assembly feature, select it and "push" the definition down to all of the affected participants. The problem that I can see with this is two of the participants are just two instances of the same part and the hole hits the part in two different places due to rotation/translation of the instances, isn't that a conflict? Which hole gets put into the part? Both? I'd be interested to see what SWx does in this scenario. BTW, after the push the assembly feature ceases to exist or maybe we just make this an option at creation time and the assembly feature is auto deleted after it computes for the first time. This also might be a problem if one of the affected parts is adaptive, we might have a race condition.

Sbix26 - Errors always display with the details collapsed-- typically takes four clicks of very tiny + marks to expand to the final details about what went wrong. Why not expanded by default?

> *ethane : so, let's say that assembly constraints have failed...there are several clicks to go through simply to fix it.*

This is a good point Sam! We could at least expand the first error by default, there is a lot of room for improvement here. The design doctor has not been modified in a long time.

Matt : When using tools with an eye dropper function (such as the new framing generator commands), holding the ALT key should temporarily show the new eyedropper tool.

Good suggestion.

glenn : I'm trying the new Inventor and Revit integration feature. When edit part sketch in an assembly, can sketch snap on or set up constrain relationship with the imported Revit model objects? like vertex or surface? It doesn't seem work for me.

We are not seeing this problem Glenn. You may need to project the geometry into a sketch but the Revit data is no different than any other AnyCAD data which has been working for several releases now. Please reach out in the forum or beta forum for specific help on this.

Matt - Has there been any high level talk of implementing part/assembly design variations within a file? Sort of like git feature branch workflows. E.g. you want to explore a design variation... You could create a new branch, test it out, and either merge it in or delete it.

Not git like branching but our beta does contain Model States that you can try this out to a certain extent.

Pepe Martin Bagüeste - A fairly common problem for me is in iparts. when I modify the parent of the ipart I have to re-generate all the child files and if they are in very different assemblies it is a hard task. is there an easy alternative without the content center?

You should be able to open the iPart factory file and execute the Generate All Members command. Than each assembly that uses them will need to be updated on open to accept the changes to the child files.

JoeMorin : SUGGESTION: Constraints – show a list of all constraints, with the ability to filter that list. For example, list all constraints, then filter to show projected relationships, or locked, or whatever. Then CTRL-click or SHIFT-click to delete some or all of that TYPE of constraint.

Matt : OMG. A constraint pallet would be killer. It could list the constraints for a selected object(s). These could be selected and modified from that palette.

This should go on the Ideas Forum.

JoeMorin : SUGGESTION: Dimensions – add the feature to scroll through dimensions with mouse wheel or scroll bar below dimension

Please add more specifics to Ideas Forum.

Todd Harris : Any insight into user IV to create drawings of Fusion parts? We use both PDMC and Fusion. Fusion assemblies need to be dropped into an Inventor assembly rather than creating drawing on the Fusion file. It makes BOM's tricky.

We are looking into Fusion <-> Inventor workflows constantly. Keep an eye on some synergy here in the future.

Alan : Ilogic is there more options coming for Event triggers. Example place a rule in a part to open a configurator form. When the part is opened in an assembly all part rules fire on file assembly file open, this can be alarming if many forms are used however when you open the part through the assembly/edit the rule does not fire and user is unaware that the configuator form exists.

We are always evaluating new triggers based on user input. Our iLogic expert says your first issue can be avoided by adding this to the top of each rule:

If ThisDoc.Document IsNot ThisApplication.ActiveEditDocument Then Exit Sub

He also added a story for a new event trigger for the second one!

Mhanson : We can't figure out how to select a part that is inside another part in an assembly after you turn the outer part transparent. Any ideas other than turning visibility off?

Use "Select Other", transparency in Inventor does not work the same as that other software with respect to picking.

JoeMorin : Is there a way to auto-delete empty sketches on sketch exit?

Not currently, could be a nice option. Please add this to Ideas Forum.

Rob Luecht : Is it possible to be able to use TAB to "select other" and toggle through close objects or overlapping lines?

We have added Ctrl-A as the default for Select Other. I am not sure if TAB can be used as a shortcut but I suspect not as that is the Windows default for focus change of controls.

Lars van der Greft : Suggestion: When using "Project Geometry" make it possible to drag a window to select edges and points

Another good suggestion that can go on Ideas Forum. Adding window select for commands can sometimes be tricky but is certainly doable.

JoeMorin : Is there a way to show entity count when sketch entities are selected? This would be great to get rid of overlapping lines in sketches or lines VERY near each other, or to verify there aren't overlapping lines. In SW I used crossing select (window select from right to left) and it would show this easily.

There is no current way to do this but it is possible.

Henrik : when linking to excel the filter outfilter .xlsm files, plan for enable macro based excel files ?

I highly doubt this due to the security risks.

Nickmiester : any plans to make point clouds meshable? This used to be part of recap but has been removed

I am unaware of any such plans.

JoeMorin : Is there a way in an assembly to make each part a different color, or at least a different shade of gray, to make it easier to see which parts are which? Grey on grey, next to grey, and very difficult to see what is what...

We have not heard of this requirement and would be impractical on large assemblies where each color would differ by very very little.

Matt : New feature: parametric sketch symbols!

Probably not any time soon! (Just being honest)

Matt : New feature: drag and drop a model/assy file onto a drawing window to place a base view

> *- there is at least one Idea Station request for this. It hasn't received a ton of votes* *https://forums.autodesk.com/t5/inventor-ideas/drag-and-drop-parts-from-model-browser-in-idw/idi-p/7170251*

I like this idea. Please vote for it in the Ideas Forum. Makes total sense.

Tim Mueller : Design View reps - Part and assembly inconsistency. Very Clunky and confusing to end users

  • Part Level - can save work feature Visibility state in the locked "master" rep
  • Assemblies - all work features visible in Master. Must use "Default" or non "master" in order to save changes to visibility states

This stems from the fact that parts did not initially have design views but they still had visibility for many years and it worked differently than assembly visibility. We had to keep the legacy part way of handling visibility but make it work in a design view manner. What we ended up with is a bit inconsistent between parts and assemblies but was done for legacy reasons, we didn't want to change the way visibility worked with the addition of design views to parts.

There are a few other inconsistencies that will likely never be rectified.

Lars van der Greft : It would be great if reference and user parameters from a flat pattern (sheet metal) can be used within iLogic. Now they are handled as two seperate things.

They must remain separate or you will have name collisions for legacy data, i.e. they have always been separated even inside Inventor. A flat pattern is "sort of" like a part within a part (very simplified view!) We could scope params or give iLogic wrappers around them to support them. Please try an Ideas Forum post for this so we can gauge the interest.

Matt : New feature: from an assembly, right click a part/assembly and show "geometry" popout submenu in context menu. This allows the user to select geometry for commands, or toggle visibility if no commands are active. That way you don't have to dig into the model browser to get to the geometry you need to select.

I'm not sure what this is trying to accomplish exactly. Please clarify it with an Ideas Forum post.

cmlane : I am a experienced Mechanical Autocad user but a newer Inventor user, I appreciate the Save Reminder pop up but I don't always see it when I am engrossed in my work, is there a possibility of having an auto save feature in the future along with a recovery feature like what is in Autocad , so as if/when my Inventor crashes for no reason that I don't lose all my work because I forgot to save it.

> *abauman : that autosave would be GREAT! - if only they would listen from the past many years of requests...*
>
> *Greg Messer : good comment cmlane on auto save feature*
>
> *Pascale Brassat : Autosave could be done in a diy way via iLogic in combination with events*
>
> *Christine Newman : I would NOT like autosave. If you add it please make it optional.*
>
> *abauman : Even if autosave had a backup of files as a temp for a period of time*

Regarding Autosave, there are quite a few idea station requests for this.

https://forums.autodesk.com/t5/forums/searchpage/tab/message?q=autosave&location=idea-board:v1232

In all honesty, we will likely never have an Autosave feature in Inventor. We have tried multiple times in the past and failed each time. The reminder came from one of these failed attempts. Inventor is a distributed system with file references, trying to autosave potentially 1000s of files would be a noticeable event.

Aaron : Please add a filter to the Inventor BOM (not the dwg BOM) to reflect current visibility state and/or LOD.

Aaron, you really need to join the beta and check out what we are working on with respect to BOM and LOD! 😊

Robert Windsor : would be nice if it added construction geo with trimming or adding fillets/chamfers in sketches

Sorry we did not get to this one during the event, I'm not sure what you are asking for. Please contact me directly or add an Ideas Forum post for this.

Joe Kniffin - is there a way to reflect when an "iproperty" is being used in the title block/revision block in the iproperties window?

We could likely do it for drawing iProperties that are used in the title block/rev block, but we could not do it for the iProperties from the part or assembly that is the first view that are used. We do not keep ANY references from a part or assembly TO the drawing. That is why "where used" is a Vault only operation, they sniff out those relationships during checkin/add to vault and store them in their own database.

Lars van der Greft - It would be nice if you can minimize an iLogic rule, without closing the rule window.

I passed this onto the iLogic expert and he indicates that we are looking into making the rule editor non-modal and allowing minimize as well. Hopefully that would address your workflow.

FlexDRG : Unfortunately this meeting sounds a lot like a "go to idea station" pitch.

ethane - How else are you supposed to get feedback without something like ideastation?

JAMERO - My team uses the Ideas station and it is easy to use and we send good ideas to our team to have them vote it up.

abauman - Better filter and combining of ideas would be great on the idea station - a lot of ideas are just "abandoned" and then more come up again that are very similar... they should start filtering by "areas" of issues or requests a little more.

JoeMorin - What is the best way to get to the idea station???

Joe Kniffin - short cut in inventor to go to the idea station? so when the idea strikes we can go submit one right from inventor.

FlexDRG - Idea station needs better 'marketing'. showing a popup every now and then, just like the save reminder...

JoeMorin : Joe Kniffin's idea to jump to idea station directly from Inventor (maybe under the help pulldown?)

Pascale Brassat : Is it "legal" to send all colleagues from our company to vote for a specific idea? it's not the real picture at the end I think?

Pascale Brassat : I recommend the Beta!! less time this special year to participate for myself (very sad about that), but it is a perfect place to share ideas!

Aaron : There is a link to the idea station right in Inventor. It's under the "Collaborate" tab But you need to have a file open.

Andrew In't Veld - Idea station is great. I have had 5 or 6 ideas that have been implemented.

Massimo Frison : ....several of my ideas are above 80, but still far to be mentioned by you :D

We understand FlexDRG's frustration but we cannot stress enough that we have 100s of thousands of users with different needs and desires. Our resources (much as we would like it) are not unlimited and we can only do so much each release. We need your help to prioritize our time so that we deliver the best overall product. We need to localize the Ideas Forum , improve its interface, as well as continue to "market" it like we are here. Pascale, there is no legality with Ideas Forum! If you have 200 users and they all vote for your idea we will consider that.

And there is no magic vote count needed to get it worked on. Votes are just one metric, resources, and other considerations go into getting an idea into the shipping product.

FlexDRG - Could we get an ideastation 'newsletter' option? getting mail updates with links to new and trending ideas?

Steve Hilvers - 👍👍👍 *idea stations email......vault too*

Many of the developers and fellow customers thought this was a great idea! Maybe weekly or monthly you see new ideas with links, etc. I will forward this idea on for consideration.

glenn : reason that the default Inventor Z Axis is pointing forward instead of pointing upward?

Ideally this would be an option but it has been this way since Inventor R1. I've seen some blog posts about this being mechanical designers vs. architects or US based vs. European...

Aaron : I need balloons in drawings to have landings like in autocad. Are there any plans for this?

I do not know of any plans for this, there does appear to be an Ideas Forum post for it though: https://forums.autodesk.com/t5/inventor-ideas/add-leader-landings-for-use-in-balloons/idi-p/8538180

Matt : Do you have input into the web-based API docs for inventor? Specifically the functionality of them. Opening links to new tabs does not work, and it would be great to fix that.

I'm seeing the same problem. I have passed this feedback on to one of our documentation guys. I noticed it does work for non-API help docs... Apparently we have already logged a defect for this.

ethane : Handles on ellipses are difficult/impossible to grab... that would be nice to see made easier

Can you show us exactly where you see this problem. Sorry we didn't try this live in the class.

Matt : Bug: When adding threads to a surface, the currently selected options are not applied when repeating the command, even when the thread command pallet has not closed (when using the add+ button). Even currently selected presets are ignored.

I just tried this in 2022 and it does the same. I passed on this issue to our UI team for consideration. The panel is cleared out because there is no selection but if you reselect a surface maybe we can return the values you just had, especially if you selected "Last Active" preset.

Further investigation shows that SOME of the values do persist but some of the values depend on the selection, so they get reset because perhaps they don't apply to the newly picked surface. We do acknowledge some preset inconsistencies in the thread panel as well. The team is aware of your concerns.

Sbix26 : Any progress on assembly View Reps: turn a subassembly visibility off temporarily, but VR link is lost and has to be re-established every single time. Inventor should remember what the active VR was when made invisible and restore it when visibility re-enabled.

Sam is referring to the fact that when you turn off a subassembly that also has an associated view rep we display the dialog asking if you want to modify the associated view rep or remove the associativity. The issue is that this dialog is completely unnecessary because you are not trying to change the visibility of objects INSIDE the sub assembly (which is what the associated design view is controlling) you are turning off the sub assembly itself from a higher level. This was pointed out to me by another user at AU – Vegas a few years ago. We agree that this workflow needs to be fixed by removing this prompt completely and leaving the associativity alone.

Jan : I Always get those weird error Messages if I try to place an iLogic part into an Assembly when accidentally Pressing place instead of place iLogic part

Jan : Why is there such a big difference between all the "place" part function in an Assembly?

Jan : Why are there so many different place commands in an Assembly. What's so different between "place" part and "place iLogic component"

Answered in chat during class, Michael Schmidt : Regarding question about difference between "Place..." vs "Place iLogic Component..." Place iLogicComponent will create a copy of the file, and also prompt the user if there are any applicable iLogic parameters found. In effect an iLogic Component is used in a Save As workflow, sort of like a template. Here's some help for it: https://help.autodesk.com/view/INVNTOR/2021/ENU/?guid=GUID-1645DF9F-1DB5-4E61-ABE7-888B32DBF254

Massimo Frison : Alltogether votes on ideas requiring an overhaul of 2D PDF translator are above 280. Are there any plans on that? (I've asked the same question 3 weeks ago at ITF but I didn't get a real answer)

We are aware of no short term plans to change 2D PDF functionality in Inventor.

Pascale Brassat : Is there any plan to enable Virtual Reality functions like working in VR with your model. Put the Quest on and let's go?

Not that any of us are aware of.

Rodney Pudney - Is there any update to be able to import IFC into inventor. We can now import RVT files in 2021 but in order to work with IFC files we still have to have a revit licence to convert them

Autodesk just made an announcement of our licensing of IFC technology from ODA. It's too early to say how that might manifest itself in Inventor but clearly it is first and foremost a priority for Autodesk.

Brian Duffy : I have a need to fold sheet metal that has a contour on the edge of the flat part. My fold is at an angle and across the edge that has the contour. The contour does not bend with the rest of the sheet metal.

Our sheet metal expert says this is not fixed and we have investigations ongoing into cases that sound similar to yours. If you can get the data to us, we would love to see it and do some investigation specific to your case. You could also potentially get a result by using cut normal to produce the contour.

Joe Kniffin - rename patterns!! so we can easily identify what it is.

I demoed live that we can rename component patterns and we can rename feature patterns so Joe needs to post on the forum or beta or send me an email where we can't do this. I see no problems right now. (Tested using 2021, Joe was on 2020)

Pascale Brassat : Some features like: Autoroute for path from A to B according to a point cloud (collision check) would be nice.
Joe Kniffin : add water mark (similar to the vault plot tool) to prints with out marking up the drawing.

I am personally unaware of any plans to do either of these in core Inventor. Feel free to add an Ideas Forum post for this.

Aaron : Is there any plan to allow you to collect all of the same part into a folder or group like NX's Pack?

> *Matt : Is that like pack and go @Aaron?*

We did not understand this question, as Matt said we do have pack and go. Please add an Ideas Forum post if Pack and Go is not what you are referring to.

David : I've got a question about the API and sharing code. What do the devs think of a repo for extension methods for common tasks that programmers come up against when using the API?
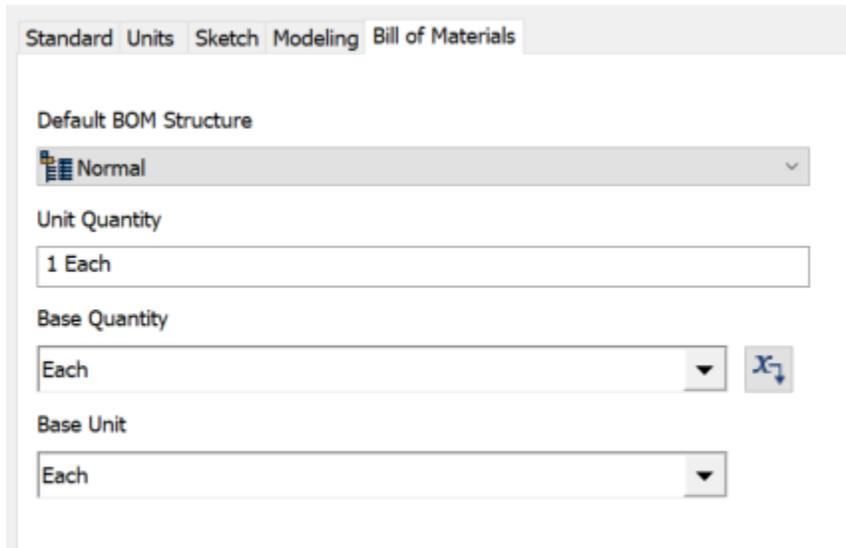
> *For instance, I recently wrote a GetNativeObjectPick method that has many steps involved but returns the native object after a user picks something, and I'd love to share that hard work. Or just methods that simplify things. I have a Document.SetiProperty("propertyName", "new value") extension method that I use all the time.*
>
> *Matt : @Dave is this something that someone just needs to throw up on GitHub? Or do you want something managed by Autodesk?*

Please feel free to start such a repo yourself! Could we formalize our API blogs, perhaps but you are the first person to suggest it. If you start it I'm sure we would contribute to it. Autodesk does not have to be the owner/operator of an open source repo such as this.

Christine Newman : I would love to be able to put a partial number as an override in an assembly bom. Let's say if I want to call up 2.3 feet of chain, or hose. Currently you can only override whole numbers (at least in 2019 version)

Even in 2019 you can change the QTY "type" for any part. Document Settings -> Bill of Materials. You can change the Base Unit to Feet and tell it which value to use (in Base Quantity). Once you do that you can than do non whole number overrides in the BOM as well.

Standard Units Sketch Modeling Bill of Materials

Default BOM Structure

Normal

Unit Quantity

1 Each

Base Quantity

Each

Base Unit

Each

to7176 : Screw holes cannot be exported into step models, because screws are not model. Do you see any way in order to give the information to these holes in another way?

Answered by another customer, JAMERO : I believe we use Custom iProperties for this. Using STEP242 you should also be able to export any MBD Annotations into the STEP file.

Massimo Frison : ContentCenter: possibility to interact with custom family tables via API? (this way some external system could populate certain fields)

> *Matt : I think you can access them via API. I thought mod the machine did a blog post on that*
>
> *Massimo Frison : this is needed directly on the idcl file*
>
> *Massimo Frison : technically the request would be "an idcl editor"*
>
> *Massimo Frison : Content Center: in several circumstances ERP systems would require a systematic way to access custom content center families tables. Any different approach will not be so effective as this one. Unfortunately Inventor does not offer this possibility, and the SQLite files are actually unreadeable. by any software other than Inventor Has there ever been demand for controlling custom content center tables systematically?*

I found the blog post that Matt was referring to:
https://adndevblog.typepad.com/manufacturing/2015/12/manipulate-family-table-of-content-center.html

Hopefully this helps.

FlexDRG : Oh... BOM list... i get a mixed list with item counts, but also with lengths. And i can't get the lengths to drop the ",000" on the end. trying to use the change filter messes with the regular item count...

Assuming you are referring to the frame generator issues we have heard about that and are aware of it but the values seem to be hardcoded. Please add an Ideas Forum post for this improvement.

igor.fuks : Is there a way now in 2021 to have default value of the prompted entry in title blocks?

Not that I am aware of but I bet you could do it with an iLogic rule that detects no input value and uses your default.

Christine Newman : When creating title blocks I would like to be able to constrain a bmp relative to the sheet. So if my title block was created on a c size sheet, if that title block is used on a A-size sheet, the bmp doesn't go off in the distance.

Please add an Ideas Forum post for this, we really need to give you ways to constrain that bmp border to the sketch geometry probably.

FlexDRG : Anyone here sometimes have trouble with selecting a face to constrain against? i can click allover the face, sometimes other faces (of other parts) and nothing gets selected. i need to rotate the view around a bunch befor the wanted face can be selected.

Michael Schmidt : @FlexDRG - have you tried using the "Select other..." context menu option? this is helpful when trying to select a specific face/edge/etc.

FlexDRG : Yes. it selects all sorts of faces exept the one my cursor is over right up front.

This sounds like a bug, if you can reproduce it for us with a data set and/or video we would be happy to look into it. UPDATE: We did receive your video and Michael will be reaching out to you directly with some follow-up questions.

JAMERO - When will Inventor be able to more fully use multiple cores on a processor and what is the optimum number of cores to get when purchasing a new Workstation?

With every release we add more and more usage of multiple cores. Inventor will never use multiple cores 100% of the time, the architecture doesn't really support it. We endeavor every release to use multiple cores for anything we can including graphics display, body computations, etc.

Steinar - Please name to selections in the Select Other meny better than Face, Face, FaCE, Edge, Edge, Edge

The trouble is what do we name them? Faces and edges do not have names in Inventor and adding an index doesn't help you locate them any easier, they should get highlighted in the view as you hover over each selection though

JAMERO - Will you be back porting the machining export for PDF to Inventor 2020?

There are no plans to do this. Assembly features are exported to 3D PDF with 2021.

Anirudh Ashok Kumar - Why no record macro option like SolidWorks or Excel?

We have no current plans to support this. Please log an Ideas Forum post for this so we can prioritize it against other projects.