

UGS I-deas 12 NX Series

Although development is undoubtedly winding down, usage is still widespread and the good news is that I-deas 12 has just been released – so for those actively using the system, what's new and improved?

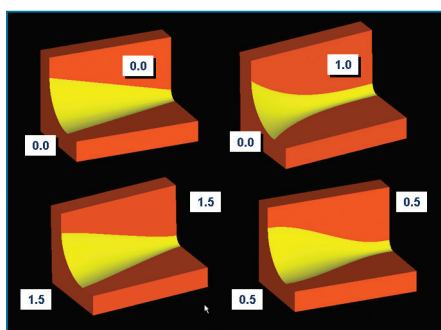
Written by Al Dean

MUCH OF THE WORK for I-deas 12 seems to revolve around updating and improving the existing functions and operations, although that said, there are a few new features. There are a number of tools that focus on making existing models more efficient when you're making design changes or modifying existing product data. A primary example is the work done on Blend Curve, a spline that transitions between two curves or edges and maintains tangency. This now handles design changes where previous versions would have seen some serious flipping, but in V12, the system retains the design intent and shifts the blend curve to accommodate the changes.

Another example is the integration of the material side definitions into the revolve feature. As with the other examples we've discussed, this allows you to predefine where downstream features need to add or remove material and helps to ensure that design changes are made without too much remodelling.

There has also been a lot of work done to fine tune the filleting tools within I-deas, with attention on the creation of specific forms of geometry. I think it's amazing that despite the fact that the system has been under development for fifteen years, I-deas users are still demanding more flexible filleting tools even that this stage in the game – still, the updates made show the advanced level of production use.

These filleting improvements include handling of complex occurrences, such as Tight Knee situations, where a mathematically correct fillet would fail rather dramatically. Now, the system has the ability to slightly fudge the surface, to create required form by slackening off the math control slightly. The ability to fillet complex geometry is also made more flexible in cases where you have a three surfaces meeting. The manner in which the system handles end (either by using a cutting plane to cap the Fillet or a side surface extension) and middle surfaces have been improved to ensure that you can get result you want, rather than relying on what the system can give you or what you can get out of the surface



Tangency control for variable radii fillets

manually. There has also been work done to provide the user with more control over variable radius fillets. In previous releases, you had a lot of control over the fillet radii at the end points of the fillet and points between, but tangency was either on or off along the whole thing. In I-deas 12, you can now set control magnitude of tangency at any end of a fillet chain and specify tangency influence at any vertex in the interior where radius function tangent is zero.

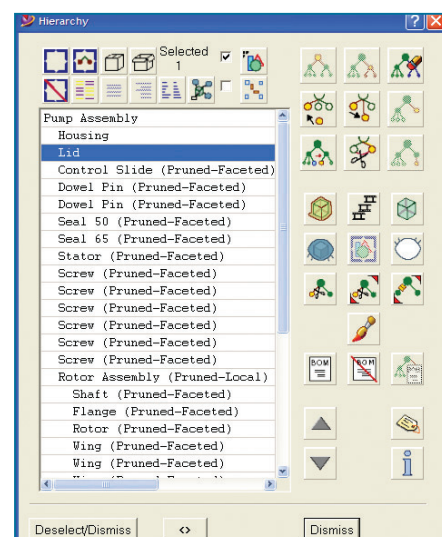
Finally for filleting, the development team has done some work that's in the polar opposite direction to the work currently under way by many other vendors. At present within I-deas, all of the filleting operations are available from within a single operation, but following the wishes of users, it's been decided that some specific, commonly used fillet functions are to be separated out to make them more accessible. You now have the main fillet command, then the Fillet Between and Three Surface Fillet available directly within the modelling interface, rather than having to pick through menus or dialogs.

You'll be glad to know that filleting isn't the be all and end all of I-deas 12. One interesting new feature, which while retaining that data modification ethic that's part of this release, sees some of the technology traffic that's occurring from I-deas to NX flow back the other way. The new Associative Offset Curve on Surface operation is a direct port of technology (it shares the same algorithm as NX) that allows you to create an offset surface from the edge of a surface (or group of). In specifics, the system doesn't just use a dimension, but rather allows you to define a distance either as a chord height or 'along' the curve. The surface quilts don't have to be smoothly connected, rather just held as a single group. The offset can either be within the surface form or can be externally offset, which uses an extension of the surface sets to create the correct form.

Another area that's been worked on is the handling of silhouette curves in tangency situations, more specifically, where the difference in alignment between two faces is less than 3 degrees. In previous releases, fillets and such would fail and the tangency wouldn't hold or you'd get a discontinuity or fault at the intersection. From now on, the system should be more capable of dealing with such problematic forms as the mathematics behind it is more robust (and can handle geometry down to around 0.05 degrees).

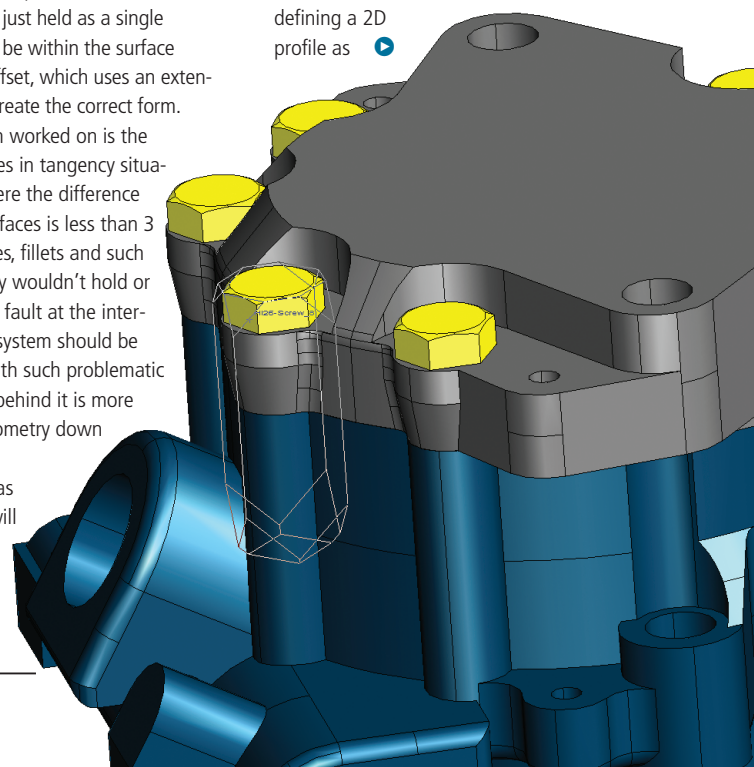
For those long time I-deas users, the Section Builder will be come as a useful new tool – and for those outside

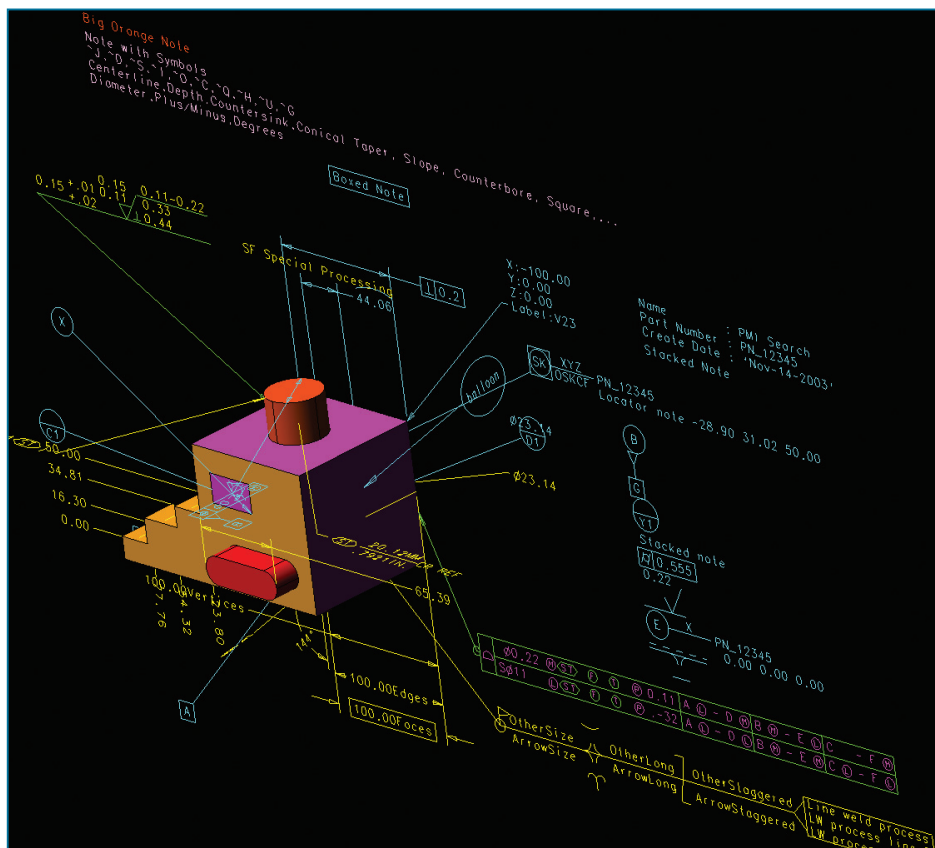
Mixing true CAD geometry with JT Faceted data



Pruned items displayed as JT faceted data

of the I-deas community, you should note that Section, in I-deas parlance, refers to a profile or sketch onto which 3D features are built. The Section Builder works similarly to the Surface Chaining options displayed in other functions, but kicks into play when you're using either wireframe or existing model edges to define a 'section' for an operation. It allows you to automatically select profile geometry either by auto-chaining tangent or fillets, and has options that will stop selections at intersections or force you to use planar sections only. The reality is that it's a method of capturing design intent on which to base further features, so rather than defining a 2D profile as





Example of various PMI annotation types and styles

- specific entities, you're really just indicating where the profile exists and letting the system build on top of that geometry. The reason that this is undoubtedly a 'Good Thing' is that if you make changes earlier in the history tree and the generating section changes, the downstream features have a much better changes of adapting to those changes.

The final modelling update I wanted to discuss were the enhancements made to the Feature Copy feature. In existing releases, this allows you to take subset of geometry and create associative copies elsewhere within a part or assembly, onto which you can then build other features or parts etc. These copies can be either made as separate entities (allowing individual changes) or can maintain an associative link back to the originating features. But the limitation was that you could only copy geometry. Well, from 12 onwards, you can use the same operations too, but also select specific reference geometry (including co-ordinate systems as well as reference planes, curve, line, and points) to include in the feature copy as well. This provides greater control and easier updating for complex models.

3D PMI

Before we get onto the drafting updates for I-deas 12, one area I wanted to touch on was 3D PMI (or 3D Product Manufacturing Information), which links the two sections rather nicely. For those unfamiliar with the term, this is UGS' nomenclature for 3D annotation, but considering that I-deas probably has the most highly developed 3D annotation tools in the market, I'll allow them this one acronym. 3D annota-

tion allows an organisation to store all manner of geometry, tolerance and dimensioning information directly within the 3D model, rather than a 2D drawing medium. Some vendors use the term 3D drawings, but it's a horrible mangling of the purpose. What you need to realise is that 3D PMI, 3D annotation whatever you want to call it, is a replacement for drawings, but uses many of the same annotation and documentation methods established and standardised on in industry. Yes, you can present the information in specific views, i.e. drawings, but it's not about that. Anyway, rant over. This release of I-deas sees the software's support for 3D annotation extended to further improve its compliance with the US ASME Y14.41 standard. Although probably first in the standards community, this standard is also being replicated across the globe and standards for 3D annotation are being drawn up (if they haven't already) by BSI, ISO etc. In specifics, the system now allows you to display different decimal places for dimensions and tolerances, use a dot as Leader terminator, an All Around, Between and Circle U symbols. Also, on the presentation of 3D data, there has also been work done on 3D section views. You can now create sections of sections and create internal "Partition" sections

Assembly

Alongside the part modelling updates, there has also been some work done on the assembly modelling within I-deas and again, much of this work concentrates on either making the existing tools more usable or enabling the efficient re-use, modification and

updating of design data. A good example for making new tools more efficient is the ability to use a Multiple "Select All" operation within the hierarchy browser. When you're looking to select parts of a similar nature, either for hiding, deleting (perhaps for FEA purposes) and such, then at present, you can select a single entity and the system can find all those that match – for example, select a single M5 machine screw and the system can find all the others. From 12 onwards, you can now select multiple types (in our example, M5, M6, Nuts, washers etc) and the system will find each instance of each of those selected. Also, within the same form, you can also select parts using a more powerful entity name filter that allows you to use wildcards and logic operations (such as AND, OR etc). On the data modification handling front, new abilities to update a number of configurations in one hit will make a massive difference for those users dealing with large datasets and managing product variants as will.

Another area that's been worked on is the Make Unique operation. When you use patterns of parts within an assembly, you've always had the ability to break the link for one or more instances within that pattern, so that you can edit it individually. The new version of the tool ensures that if you have drawings associated with the original part, it can optionally be copied and the associated with the new unique part – ultimately saving you a lot of work.

Multi-CAD environment

I-deas has always had a number of ways of working with multiple geometry types, but for the I-deas 12 release, this was again an area in which the development team has put in a lot of effort – presumably to assist those users out there looking to use the system in conjunction with other CAD systems on specific projects and such.

One area within this rather wide subject that I wanted to discuss was the potential for using JT data for a great deal more than just data viewing. Within I-deas, you have always been able to take advantage of Pruning when checking assemblies out from the data management system (whichever one that was). What this does is to extract just the information within an assembly that you need to edit another part. The disadvantage was that the other surrounding parts couldn't be viewed at the same time. For small products, this thought of not checking out a whole assembly might seem trivial, but when you're looking at Automotive data, checking out an entire sub-system just isn't going to fly. What this release brings is the ability to do exactly the same, namely, selectively get specific I-deas parts from the library, but then also have the other parts around those displayed as JT data. This reduces both the file sizes and the time required to extract it from the data management library. This alone is pretty impressive, but when you look at the manner in which the JT format has been integrated into the system, the picture becomes much clearer.

JT is primarily a faceted format for displaying 3D design and engineering data, but you can, if needs be, also hold b-rep data (referred to as Precise data) within those parts at the same time. Here, you get the benefits of a quick display (as the faceted



The I-deas 11 NX series release cycle (including service packs and incremental updates) saw the introduction of some core new functionality. This ranged from support for NX Nastran 3 and 4 from within I-deas, support for the NX Nastran Advanced Non-Linear solve and a rework of the integration, which now matches Model Solution. Also, considering the eventual move away from I-deas towards the NX platform, the 11 series saw the ability added to translate a solution set from I-deas Model Solution to NX Nastran, allowing you to cross compare results and conduct post processing work in a single environment.

For the I-deas 12 release cycle, there have been a number of core updates made to the existing FEA tools that should make its use more efficient and more productive – through automation of some specific workflows. A prime example is the new tools introduced that allow you to create bolted connections very quickly using CAD feature recognition. The system identifies all potential bolted connections between two volumes or single connections between two circular edges and generates the connection based on user beam, bolt type (threaded or through hole). The use of geometry to define the bolted connections is key to the whole process as you're using design intent to define the bolt connection, rather than FEA entities (such as elements and nodes). For example, aside from user defined tolerances, the system uses geometry references to define the FEA parameters. The beam cross section table sets diameter, when dealing with through holes (a bolt with a nut), the geometry sets the length, a bolt in a tapped hole's is defined by the Bolt definition. The result is the system should be much more capable of propagating design changes (even those which alter surrounding geometry). That said, the operation does require a mesh to be already generated, but with the meshing tools within I-deas, that's not a major problem. To aid in creating large quantities of these features and to assist when reusing data, I-deas also allows you to store the bolt definition and these remain available and usable when you need them.

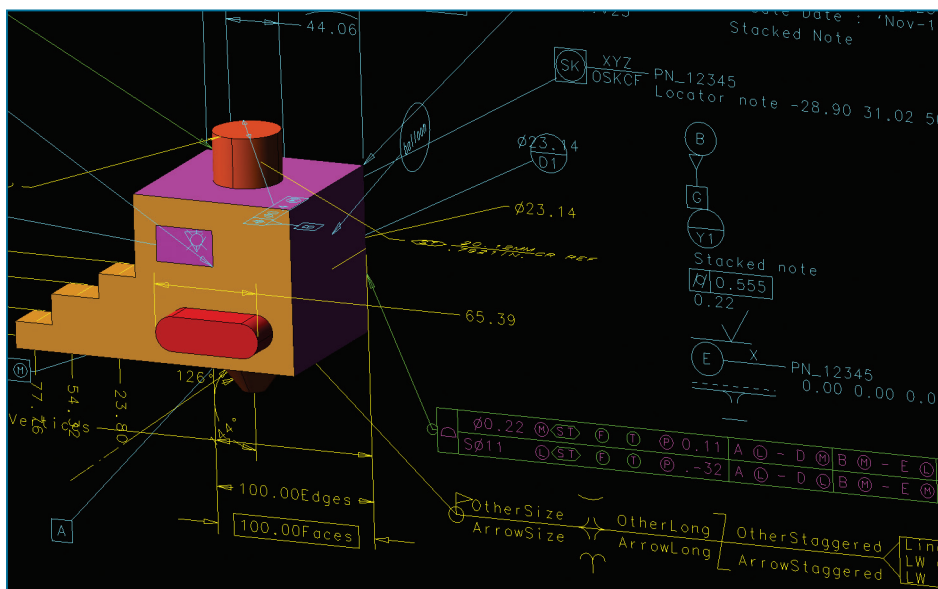
Another area in which there's been a great deal of work for R12 is in mesh splitting, particularly when you're either modelling contact surfaces with a partition or modelling coincident contact surfaces with sections (again, remember that section is I-deas parlance for profile or sketch on which a 3D feature is based). The partition generates one surface shared by the two volumes, and by default the volume meshes generated share the same nodes on that surface. This is extremely useful to model different materials bonded together. But if the user is trying to model a contact condition then the volumes need to have independent nodes on the shared surface. The new Mesh Split function creates this condition where the volumes have independent nodes on the shared surface allowing contact conditions to be modelled with ease.

It's strange watching what's happening to I-deas. I've been reviewing the system consistently for the past eight or so years and you might have been forgiven for thinking that a system that's almost fifteen years old wouldn't need much tinkering with. The work done for this release shows that although UGS has concentrated the majority of its efforts on the NX platform, which is undeniably Unigraphics-based, the I-deas community is still getting some benefit.

We can't really discuss I-deas without talking about the move to NX that will be facing more and more of its users in the coming months and years. UGS are offering some pretty compelling reasons to move across (such as free like-for-like license swaps) as well as a range of tools that can assist to make the process as painless as possible. With this in mind, it's clear that I-deas is winding down, but the development work has been done shows that there is life in the old beast yet and while this might not be the headline grabbing work done in NX 4, its still important to many users.

You should have picked up on the fact that much of this release is focused on allowing users to edit existing data more efficiently and to ensure that those changes don't have too much downstream impact. And when you look at the sort of customers that are still using I-deas in anger, it's clear this is going to be a major focus for many of those users – namely, the need to maintain master models of products and be able to adapt them within I-deas for sometime to come.

UGS has already committed itself to I-deas 13 and the product release planning is currently underway. Release cycles seem to have lengthened slightly (as you'd probably expect) so the current delivery date is around Q4 of 2006. While details are not concrete yet, it would seem that again workflow productivity and the extension of multi-CAD use are key focus areas, particularly using the JT format, together with more tools to move I-deas customers onto the NX product set.



Product	I-deas 12 NX Series
Supplier	UGS <i>www.ugs.com</i>
Price	n/a