Product Review

Dr. Charles Clarke

NX CAM TURBOMACHINERY MILLING PRODUCT REVIEW

Introduction

CAD/CAM technology has progressed to a point where functions that were once considered advanced, like 5-axis programming, are now commonplace. However, technology advancements allow software vendors to push further into more specialist areas, thus providing higher levels of value to their customers. The result is new CAD/CAM extension applications (often referred to as add-ons) that address specific requirements in specific industries with a high degree of precision and performance. These add-on applications reduce the need to use specialist, standalone applications that operate outside of standard company systems.

This review looks at an example of these new advanced applications with the NX Turbomachinery Milling module that Siemens PLM has introduced to their line-up of part manufacturing solutions in NX 7.

The new software has been developed very closely with key customers – it is a market that Siemens PLM dominates from a generic CAD/CAM perspective with companies like Pratt and Whitney, GE Aircraft Engines, Rolls-Royce and many of the major aircraft and power generation engine manufacturers and suppliers being significant customers for Siemens. Also in-house divisions like Siemens Power Generation are significant users of the software.

Dr. Charles Clarke

Charles Clarke is a recognised consultant and writer in CAD/CAM/CAE and PLM and related topics. With a background in engineering, the CAD/CAM industry and with direct experience of planning and implementing these systems for high profile companies, he has good experience on which to base product assessments and reviews. Dr. Clarke is a regular contributor to a number of industry magazines both in the UK and internationally.

Market drivers

The latest turbine systems offer high levels of efficiency in many applications from power generation to aircraft propulsion. This is one reason for the continuing increase in demand for the multi-blade components that are critical elements of turbine engines.

A development over the last decade is a move to aero-engine turbine discs (often referred to as blisks) being made as one component rather than assembled from a set of separate blades which are then attached to a relatively complex hub. The single part is typically lighter and this contributes to extra efficiency. The trend to use these has been moving from military to commercial applications, leading to the extra volume.

However, machining multi-bladed blisks or impellers puts extra demands on the NC programming needed to drive the kind of advanced multi-axis machine tool used for this work. Although many CAM vendors choose these demanding components to illustrate their 5-axis software the real tests come in productive programming. Unfortunately in many cases standard 5-axis software is not just multi-axis, it's also intended to be multi-purpose. Typically generic 5-axis CAM software does not offer the special operations designed especially for easy and effective programming of turbomachinery components.

Typically generic 5-axis CAM software does not offer the special operations designed especially for easy and effective programming of turbomachinery components.

Existing specialist applications

For some time the specialist task of programming and machining the more complex turbine blades, blisks and impeller type components has been achieved using specialist machine tools and NC programming software provided either with the machine tool, or purchased separately from a very specialist software vendor. For many companies this is not an ideal scenario, at least from the software perspective, as they would rather use an application that is consistent with the rest of their operations. Apart from the having to deal with multiple vendors, and deal with data transfer or translation, it's also a data management issue and also a revision control issue. For larger companies it's a concern that such one off applications tend to reside outside the Product Data Management (PDM) system adding to the cost of effective data management. In addition, these very unique and specialist packages can be very expensive to buy and maintain.

A new generation of software that provides flexibility

Example of a blisk model that can be programmed using the NX CAM Turbomachinery Milling application.

With the latest release of NX, Siemens PLM Software has introduced a specialized extension application for their established NX CAM software that is totally focused on the machining of turbomachinery components. When Siemens PLM talks about turbomachinery, they are not just talking about blisks and impellers, the application could also be used for propellers or other multi-bladed rotational parts. The general marketing term for all of these areas is turbomachinery.

A key objective of the new NX Turbomachinery software is to allow users to do this complex programming as easily as possible. The system provides purpose-built options that put the user in the context of turbomachinery. The idea is to simply select the geometry and tell the software the kinds of blades you are interested in and the operation you need. You then press the 'go' button and the system will come back with completely gouge-free, collision-free tool paths for the entire component with no further editing or modifications required. With generic software it would probably take hours to generate a tool path just between two blades with a splitter. With NX Turbomachinery Milling it takes about 2 minutes.

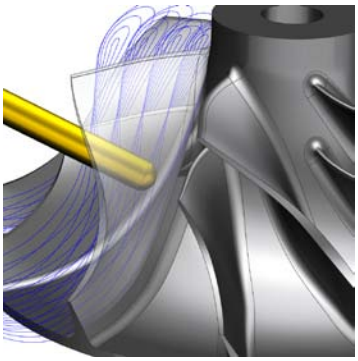
Because all manufacturers tend to do things differently, the idea with NX Turbomachinery Milling was to keep the blade machining module as flexible as possible. There are some CAM software applications where you can only machine between the blades as long as the blades are straight. If the blades curve from the shroud to the hub, the system won't be able to machine them. There are no such restrictions in NX CAM. On some impellers there are smaller blades called splitters in-between the main impeller blades. Some systems won't allow you to specify any splitters, and generally even the most powerful ones stop at one splitter. You can specify as many as six splitters in NX.

Specialized operations for blisks and impellers

With the new NX application the whole system is built to make it easy and intuitive for the programmer who interacts with menu terms like blade, splitter and hub. It is expecting to process certain types of geometry, so all you have to do is specify where these types of geometry are and the system does the rest.

If you have a splitter in-between the main blades you have to tell the system where they are. You then specify that you want a roughing operation and you specify the tool you want to use. You need to identify whether the feed rate and step over are appropriate and whatever material you want to leave on the part. The specification process takes about a minute or two and then you tell the software to go and calculate the tool path. Processing time is another minute or two and you've got your tool path.

Controlled roughing



Example of a roughing tool path.

For example, when roughing you can limit the volume that you want to remove in any tool path operation so you can machine say the top 50% of the blade in one operation. Impeller blades tend to be long so as you approach the end of the blade it is effectively a long cantilever and you can get significant deflection on the blade as its machined. If the blade is stiffened by the bulk of the blank during these operations as much as possible then the deflection due to machining is minimised.

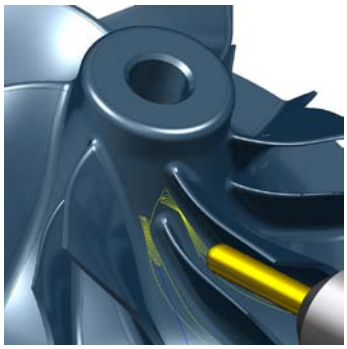
Typically users tend to rough the top 50% and then finish the top 40%. And then go on to rough the bottom 50% and finish the remaining 60%. When dealing with hard titanium alloys it is common to machine in two depths so there are two depths of roughing and two depths of finishing.

Flexibility in pattern selection

Although automation is important to shorter, easier NC programming, you need to have the flexibility to work the way you want. In NX you have access to many options if you wish to use them. For example NX will find the leading and trailing blade edges automatically but then you are able to adjust these positions to get the path you want. You can specify which corner you want to use to start the machining pattern; you can specify the type of pattern whether it's climb or zigzag; whether you want the step over to be from the left blade to the right blade; to be in 20 passes, or define it by giving a scallop height of 0.2 mm or 6 mm depth of cut or 20% of the tool etc.

Rest Milling

One area where the Siemens application appears to offer a particular advantage over some other systems is in rest milling. The NX software manages an active or "in-process" model of the work-piece so that the system knows what has been machined and what material is left. So for example, when material is left by a larger cutter between the base of a splitter



Example of a rest milling tool path.

blade and a main blade, NX CAM is able to target this uncut material automatically with the click of a button. This saves programming effort, and leads to much more efficient machining. The in-process work-piece also works between turning and milling which is also valuable in turbomachinery where the rotational parts will have turning and milling operations.

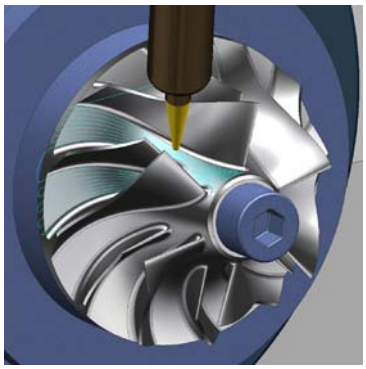
Tool axis control

You can also constrain the tool axis and specify how much you want the tool to lead or lag. This is especially important when using ball and end mills, where the bottom tip does not have a cutting edge, so you can create a leading edge by tilting the tool. You can specify the lead and lag for the leading edge and the trailing edge separately and a tool axis gets interpolated in-between. You can specify these parameters at the shroud and the hub and the software interpolates the increments required to transition from one to the other.

Most of the tool axis deflections take place as the tool transitions near the leading and trailing edges because most of their change of curvature occurs in this region. In order to minimise this you can fix the tool axis to the value close to the leading or trailing edge of the blade. You can specify that within 20 mm of the leading or trailing edge to keep the same tool axis for the remainder of the pass. What you don't want is sudden changes of the tool axis as this can translate into surface or tool damage. Sometimes a small tool axis change can mean quite a significant change at the cutting edge of the tool. Here NX helps you to manage the rate of change of tool axis change. Siemens do say that a special function for blade edge machining is one of those items they have planned and that this will make this task as fast and easy as the others already covered in the new software by a special function for blades and blisks.

Working with poor quality 3D geometry

Other CAM systems are very specific about the kinds of 3D geometry that can be used as the basis for NC programming in these complex parts. Finding and fixing problems with the CAD model mathematics can be a tedious and time consuming task. A big time saver with the new NX software is that it does not require you to resolve these geometric issues in the model before programming. NX CAM works directly with this imported geometry to create clean tool paths without the typical fixing and re-modelling – saving you hours of model preparation time prior to NC programming.



Example of g-code driven simulation using a machine tool model in NX CAM.

The value of a full CAD-CAM solution

One advantage of the NX solution is that the CAM software is a part of a complete suite of applications that can be used in manufacturing engineering where everything is connected back to a common part model. With NX, the NC programmer has access to a full NX CAD tool box that enables the creation of a 3D model from a drawing if needed. Because of the consistent model approach, if geometry is updated then the dependant data, such as the NX CAM tool path or the NX CMM inspection programs, is updated too. You can also use exactly the same CAD functions to model everything else you will need to complete the process from tools and fixtures to a complete 3D model of the machine tool with full kinematics needed for machining simulation.

Conclusion

Given the number of demo examples of CAM systems showing simulations of their 5-axis software applied to impeller type parts you might think that any CAM package could be used to program these complex parts. While the availability of multi-axis CAM software has increased enormously over the last five years, there is still a big difference across the available packages in what can be achieved once you put the demo away. This difference is greater once established systems such as NX CAM (formerly Unigraphics CAM) move into really focused applications such as turbomachinery milling.

This new release takes Siemens into a new level of capability. As always there is more that could be added and Siemens say that they have more specialized functions coming. For example, they expect to add an out-of-the-box especially designed fillet machining for blades rather than have you use the standard fillet machining options that come with NX CAM today.

Overall the fact that NX CAM was already a very broad system with in-depth and proven capabilities this application should appeal to customers that want performance in programming multi-bladed components, such as impellers and blisks, and would prefer that it came with all the extra supporting functions that a major CAD/CAM vendor like Siemens offers.