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SEPTEMBER 2010 }

FEATURING

MANUFACTURING ENGINEERING

BENEFITS OF AN INTEGRATED
APPROACH WITH NX

NX FOR TURBO MACHINERY

SPECIALIST TOOLS FOR
ADVANCED PRODUCTION

SYNC TECH FOR MANUFACTURING

USING DIRECT MODELLING
TO SUPPORT PRODUCTION

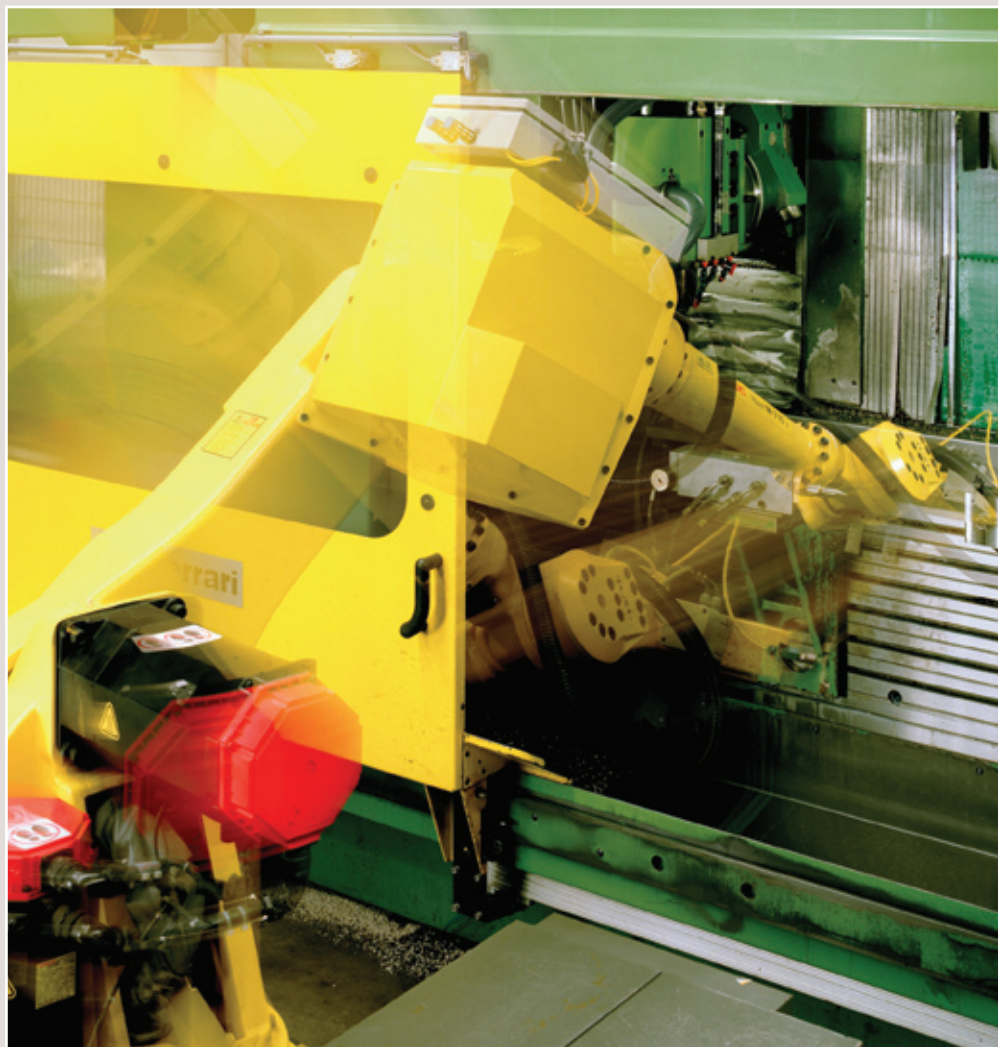


IMAGE COURTESY OF STORK TURBO BLADING

NX FOR MANUFACTURING ENGINEERING

WHEN IT COMES TO MANUFACTURING ENGINEERING AND PRODUCTION, THERE ARE ALL MANNER OF SPECIALISMS, NICHES AND REQUIREMENTS THAT NEED TO BE ADDRESSED. WITH THE NX PLATFORM THE WHOLE REALLY IS MORE THAN THE SUM OF ITS PARTS

As a product development system, NX is a giant. From basic part design and assembly creation to simulation and into the realms of production preparation with tooling design and machine tool-path creation, it has a wide range of tools to suit almost everyone involved in product development. At its very core, the NX design system allows the user to create digital models of products under development. However, in addition to the model being geometrically accurate, it can also include sophisticated information about how it behaves structurally (using analysis tools), how it behaves in use (using kinematic simulation), how it should be produced (using a wealth of tooling and CAM tools), what tolerances it should be made

(stored using Product Manufacturing Information - PMI) and how it should be installed, serviced and maintained.

Bringing all of this together in a single modular platform has many advantages and adds real value to the downstream manufacturing process. So let's take a look at the advantages this brings and the cornerstones of the NX Manufacturing suite.

A COMMON ENVIRONMENT

By defining a product on the same platform that will be used during manufacturing preparation, there are several advantages. The first is associativity. Working with intelligent native data, design changes can be propagated through to manufacture much more quickly than could be done when using disparate systems. Part design

CASE STUDY: UNIVERSAL STAMPI

Universal Stampi specialises in the design and production of die-cast moulds for aluminium, magnesium and zamak. The Italian firm also produces gravity die-casting and plastic injection moulds. Its key strength lies in the fact that it keeps the entire mould-and-die production process in-house. This translates into significant cycle time reduction and high product quality.

In the typical development process at Universal Stampi, the customer foundry submits an item to be moulded, which can be more or less industrialised and production-ready.

Mould construction starts in the design phase, as Stefano Masieri, designer and business partner,

explains. "We work concurrently in design and manufacturing. This approach implies some risks, of course, but it is a distinct way to generate added value for our customers."

Universal Stampi manufactures 125 moulds annually – highly complex tooling mostly addressed to the automotive industry in Europe, China and India. Mould models are submitted for different types of analysis and simulation using specific software, and then transferred to the factory, where 26 machining centres with advanced CAM software await.

Design software plays a key role and the company recently introduced NX.

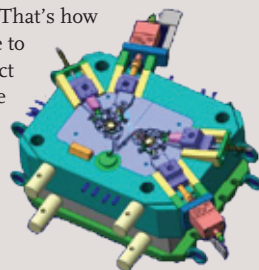
The first advantage that was noted was parameterisation, which is useful for making adjustments on the fly. "A customer may ask to change the size of a die or boss," explains Masieri, "and in such cases, parameterisation offers huge benefits."

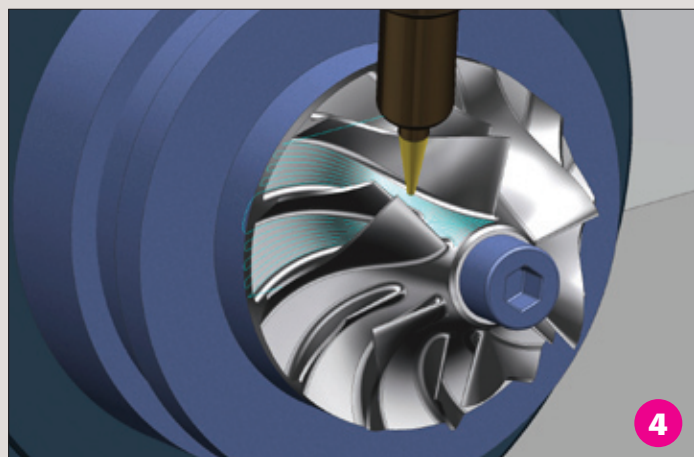
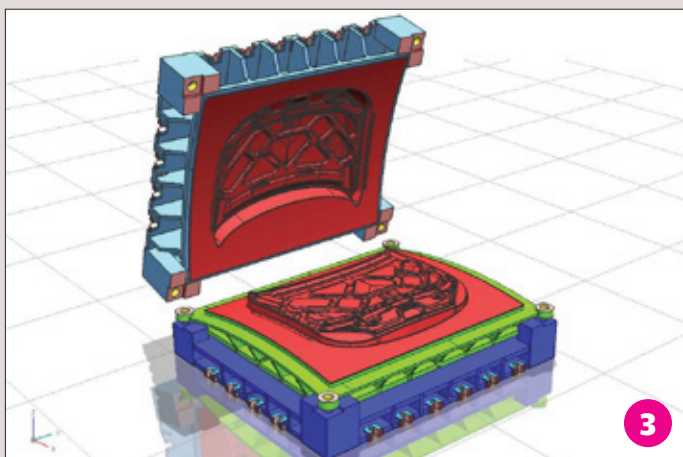
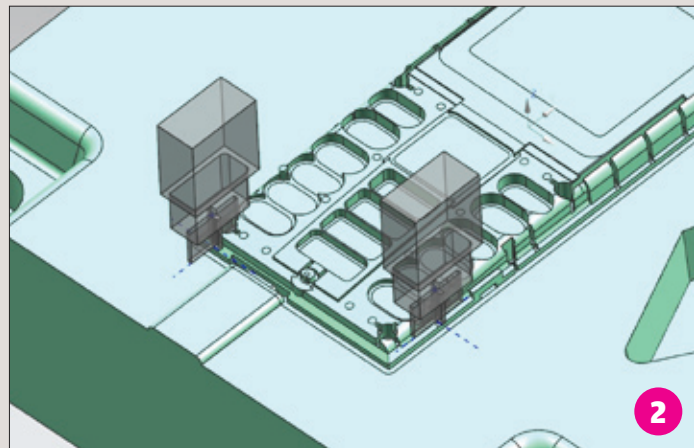
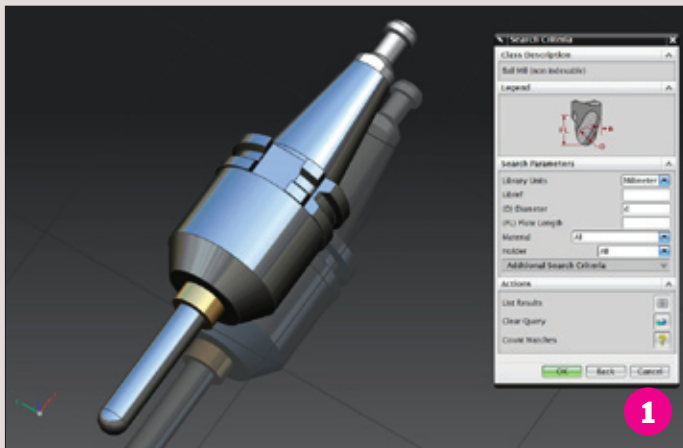
Universal Stampi licensed both NX and the NX Mold Wizard application. "With NX, we can manage the entire project in 3D with fully associative drawings, leveraging the real capability to develop a part concurrently with the corresponding mould," explains Masieri. Each modification to the part is propagated to the mould. This is very important for Universal Stampi's varied development needs,

be they starting from scratch to make a new mould or re-engineering an existing product, which is more often the case.

"The greatest benefit of NX is its huge flexibility," confirms Diego Tavelli, chief designer. "With NX you can do virtually anything and you never end up in a blind alley. You always find a way around any issue, a solution to any problem, because the designer is not constrained to a rigid approach. That's how we have been able to reduce our product development cycle time by 20 to 25 percent."

universalstampi.it





changes can be pushed to NC tool-paths, and changes can also be incorporated into tooling and fixture design much more seamlessly.

TOOLING OF ALL STRIPES

There are a wide range of 'tooling' options for the NX platform, but perhaps the most commonly used are the injection moulding tool design modules, with core and cavity development and standards-based mould base design that allows users to create a fully detailed and documented mould stack. There's an electrode design tool which enables geometry to be extracted and patched in; electrode designs to be created; reuse positions to be identified; tool-paths for cutting the electrodes to be created and all of their use documented. There are also more specialised tools to assist with progressive die development (in itself a highly complex process) and stamping die design tools that are focussed on the large scale dies used in the automotive industry.

MACHINING PAR EXCELLENCE

When it comes to machining, CAM and NC programming, NX has a formidable reputation and the range of tools on offer is extensive. From basic 3-axis and 5-axis milling to high-speed machining and into the realms of mill/turn and wire EDM, there is something for everyone. The benefit of having tight integration with the NX platform is that users have a wealth of tools to assist with preparing data for production. Further development work will help make even more use of intelligent NX product models.

Tools introduced into NX 6 allow the generation of machining operations based on pre-defined geometry features and can be automated using the associated PMI 3D annotation sets. Extending the use of PMI, NX 7.5 sees the introduction of a dedicated module for CMM programming that extracts the information to create inspection processes.

CLOSING THE LOOP WITH INSPECTION

This latest NX 7.5 release sees the introduction of the NX for CMM module that brings automation tools to inspection

programming while retaining the ability to fine-tune the process to individual requirements. This closes the loop on the process, as it allows users to gauge the quality of manufactured components against the requirements of the design data.

Alongside this, modules such as Shop Documentation allow users to create the documentation required on the factory floor. This includes setup sheets, operations sequence information and tool lists, all of which need to be supplied along with production information. With this module, NX CAM can automatically generate shop documentation and output it in a range of formats, including ASCII text or HTML Web format, ready for printing or direct browser viewing on the shop floor.

AVAILABILITY FOR THE SMALLER SHOP

While all this talk of highly integrated part-to-production processes is fascinating, for many small organisations it is likely to remain the stuff of dreams. Fear not, firstly the Siemens NX offering can be implemented with just one application - thousands of small shops have just one license of NX CAM for example. The NX CAM software is available in modular packages so you can go for just a fixed axis milling package, add machining simulation and so on. You can start with any of the key applications such as mould design, CMM programming or CAM and expand as needed later.

Also, to address those customers using or buying the Solid Edge CAD product, or who really want a modular, standalone CAM solution with no specific CAD flavour, Siemens has taken its CAM software and released it as CAM Express within its Velocity Series Portfolio. CAM Express has enough built in CAD capability for basic model editing but, unlike NX CAM, it is not offered with options for more complete sets of NX CAD functions.

The Express role developed for CAM Express boosts the ease of use, adds tutorials and makes for a faster start for new customers. It is also wrapped into every package of NX CAM so everyone benefits from this valuable, faster start up addition.

1 A good example of the benefits of integrated CAD/CAM is the ability to accurately model tool-holders and cutters then add manufacturing information

2 NX for Electrode Design module offers a step-by-step creation process which makes use of data and physical electrodes

3 Stamping Die Design for the automotive industry

4 NX for Turbomachinery adds in special terminology, intelligence and knowledge-based workflows to NX's existing NC programming tools

SIEMENS PLM SOFTWARE NX FOR TURBOMACHINERY

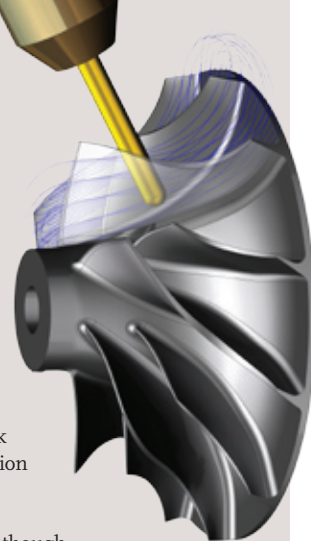
FOR THE EFFICIENT MACHINING OF TURBOMACHINERY COMPONENTS SPECIALISED TOOLS CAN GIVE USERS AN EDGE. NOW, WITH NX 7, SIEMENS PLM SOFTWARE HAS INTRODUCED A DEDICATED TOOL FOR THIS COMPLEX PROCESS. WE GIVE AN OVERVIEW OF THE SOFTWARE.

NX has a solid reputation for its high-quality NC programming tools which are used in all areas of industry. And while many of its tools are general-purpose, solving machining problems for all users, there are some industries that can benefit from something a little more specialised. NX 7 introduces a range of dedicated tools for machining turbomachinery components (blisks, impellers etc) so let's explore things in more detail.

Industry terminology: The NX for Turbomachinery module presents all of the various operations and options using language familiar to those working in the industry. Terms like blade, splitter, and hub allow the user to define geometry as specific components, then the operations pick up those definitions and reuse them where appropriate.

Controlled roughing: When roughing, a limit can be set for the volume that is removed in any tool path operation. For example,

the top 50% of the blade can be machined in one operation. Impeller blades tend to be long so the extents of the blade can act as a long cantilever and significant deflection can be experienced as it is machined. Stiffening the blades with the bulk of the blank will minimise deflection during machining.



Flexibility in pattern selection: Although automation is important for shorter, easier NC programming, flexibility is important and there are many options to tailor the process. For example, while NX will automatically find the leading and trailing blade edges the user can adjust these positions to refine the path. It is possible to specify: which corner will start the machining pattern; the type of pattern, whether that's climb or zigzag; whether the stepover is from the left blade to the right blade or vice versa; or if it is to be in 20 passes, or defined by giving a scallop height of 0.2 mm or 6 mm depth of cut or 20% of the tool etc.

Rest milling: This procedure is particularly strong as NX manages an active 'in-process' model of the work-piece and is aware of what has been machined and where material remains. For example, when material is left by a larger cutter between the base of a splitter blade and a main blade, NX CAM is able to target this uncut material automatically saving programming time and giving a much more efficient machining process. The in-process work-piece also works between turning and milling which is valuable in turbomachinery where both processes are commonplace.

Tool axis control: This is critical when trying to balance the rigidity of the cutter (using as short length as possible) against the depth required to reach the material, while ensuring good surface finish and removing any potential collision. Within NX it's possible to define the tool axis as required and then specify how much the tool should lead or lag. This is especially important when using ball and end mills, where the bottom tip does not have a cutting edge, so a leading edge can be created by tilting the tool. The user can specify the lead and lag for the leading edge and the trailing edge separately and a tool axis gets interpolated in-between. These parameters can be specified at the shroud and the hub and the software interpolates the increments required to transition from one to the other.

CASE STUDY: STORK TURBO BLADING



Based in Sneek, Netherlands, Stork Turbo Blading is part of the Stork Power Services division of the industrial Stork Group, providing a broad range of turbine dimensions, from 50 to 1200 mm, and producing 80,000 blades a year.

"We do that with eleven engineers, six quality assurance specialists and 40 production staff," explains Gerrit Mulder, engineering manager. "But in spite of our significant manpower, we never would have been able to reach this production and the level of quality if we had not figured out better working methods for engineering, work preparation, production and the associated tooling." By tooling, Mulder is

not just referring to the 4-axis and 5-axis milling machines, "The real advantage lies in the efficiency of the process before the data goes to the machines," Mulder says. "And this process can only be achieved with the right software."

Stork Turbo Blading uses NX to engineer the blades and generate NC paths. "It fits within our organisation. The functionality is what we want. And the applicability to our products is very good."

It is important for Stork that multiple engineers are able work on a design simultaneously. Based on the STEP model (created from a reverse engineered model), the base and blade are developed

separately. According to Mulder, "We also find it important to be able to develop the fixtures in parallel. This is possible thanks to the NX working method based on a 'master model.' In addition, the perfect integration between CAD and CAM ensures that modification cycles become extremely short."

"Before NX, we would spend on average three days generating NC paths after an adaptation of the design. Thanks to the integration of NX, this is now done in half an hour," he adds. "However, time saving is not the only benefit. The risk of errors is lower because the generation of NC paths is almost an automatic process."

www.storkpowerservices.com

SIEMENS PLM SOFTWARE NX FOR CMM

NX 7 INTRODUCES A NEW SUITE OF TOOLS THAT HARNESS THE POWER OF MODERN METROLOGY EQUIPMENT AND IN COMBINATION WITH 3D ANNOTATION AND AUTOMATION TECHNIQUES HELP MAKE THE INSPECTION PROCESS MORE EFFICIENT AND MORE REPEATABLE

Inspection, metrology or quality assurance - call it what you will - is the process of measuring production components and checking them against their originally defined dimensions and tolerances. The techniques and technologies used for this process have changed dramatically over the last few decades and manual measurement tools have largely been replaced with more advanced and efficient tools. One such tool is the Co-ordinate Measuring Machine (CMM), a technology that has advanced greatly in the last few years. There is now much more automation which allows a larger numbers of points and references to be taken in a shorter timeframe and leads to a much greater understanding of the variation of parts. The downside of this is that the programming required for the inspection processes becomes more complex and when a machine is capable of measuring 100s of points in minutes, traditional manual methods won't suffice.

The good news is that with the increased use of 3D annotation data (PMI – Product Manufacturing Information) within a 3D product model it is now possible to reuse this data to drive inspection programming and dramatically reduce implementation time. Acknowledging this trend, Siemens PLM Software has introduced a new module for the NX 7 release that enables all this and more. It's called NX CMM Inspection Programming. So let's take a look at how it can enhance a typical quality assurance workflow.

The new NX CMM Inspection Programming module is fully integrated into the NX environment, building on

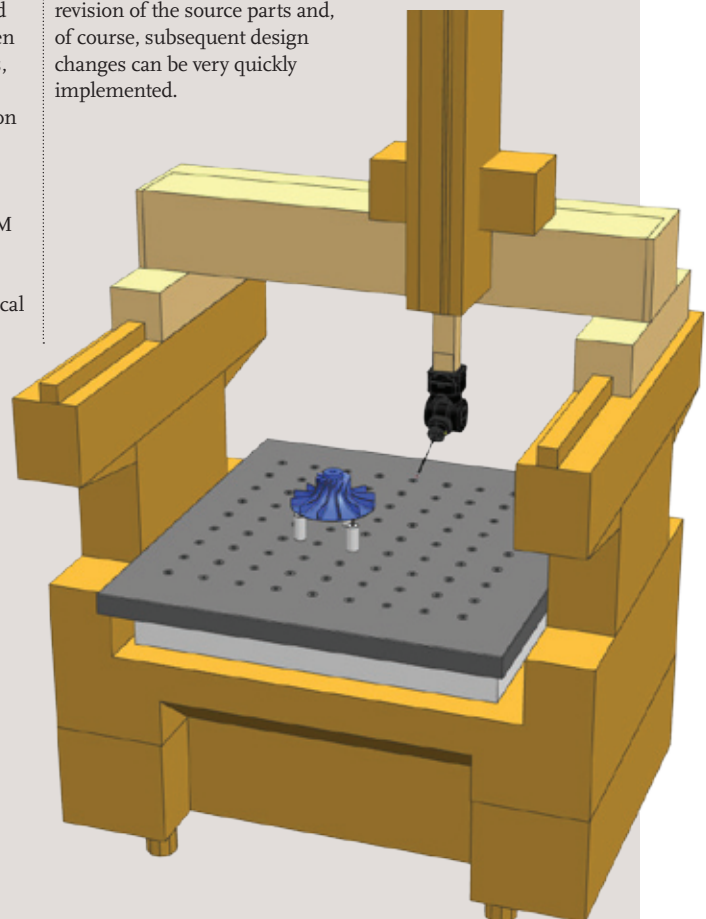
the existing geometry and PMI editing and creation tools. This also allows the team to take advantage of the existing tools for NC machine programming and simulation on which much of the new module is based. The process begins by selecting the measuring machine from a library. This contains a geometric model of the machine as well as knowledge of its operating limits, in terms of movements. Into this model, the part that is to be inspected is brought in and positioned and the 'Link PMI' command activated.

Any PMI information required for the inspection routine that is contained within the model is then extracted, including tolerances, dimensions, features, and datums. This data is then used to generate the movement of both the CMM and the inspection device based on the feature being inspected (the system currently handles 3-axis scanning and up to 5-axis touch-trigger probing). It is thought that this can remove 80%+ of the programming time.

Of course, not every organisation uses PMI, so the good news is that the system also allows users to dive in and create each measurement manually, but using built in knowledge about the process and requirements to make it much more efficient. A menu shows the inspection features and operations creation tools and the user simply grabs the pre-defined inspection method, selects the points or features on the model geometry and NX generates the appropriate path.

In addition to the programming features of the software, NX for CMM also includes a raft of tools for both simulation and verification. When dealing with multiple movements of the machine, the probe and, in particularly complex processes, the part itself, there's a huge potential for collision. Benefiting from the software's background in NC programming, these tools are available to check the program before it gets anywhere near the inspection office. Once complete and verified, the system outputs the code to drive the measuring machine, using a range of post-processors in a number of different languages, such as DMIS (Dimensional Measurement Interface Specification) or the more proprietary GeoMeasure or Calypso.

Finally, for those using Teamcenter, all of this information and knowledge can be managed: templates, tools, probes, inspection rules, as well as inspection programs and output files. These can be linked to the most up-to-date revision of the source parts and, of course, subsequent design changes can be very quickly implemented.



“ WITH THE INCREASED USE OF 3D ANNOTATION DATA WITHIN A 3D PRODUCT MODEL IT IS NOW POSSIBLE TO REUSE THIS DATA TO DRIVE INSPECTION PROGRAMMING ”

A NICHE FOR MILLING

THE BIG ONE

AN END-TO-END PRODUCT DEVELOPMENT SOLUTION FROM SIEMENS PLM SOFTWARE IS HELPING STREAMLINE THE DEVELOPMENT OF VERY LARGE MILLING MACHINES THAT ARE CAPABLE OF MILLING STRUCTURES UP TO 100 FEET IN LENGTH

As a family business with more than 100 years of tradition, Fooke GmbH has created a niche in the machinery industry that suppliers in Europe, India, China and the USA can't match: very large milling machines that are sold as complete, customised systems. In addition to the machine itself, the system includes the clamping solution and tools as well as measuring and NC programs. These machines can: mill aluminium railway structures up to 100 feet long; perform high-precision processing of vertical tail units; create high-precision skins made of CRP, GRP and aluminum; perform high-speed milling of models for the automobile industry; and address many other specialised applications.

The steadily increasing demand for these machines in the world market, as well as the increasingly complex technical requirements, motivated this innovative enterprise with approximately 170 employees to upgrade its product development process. In particular, management wanted employees of different divisions to work more effectively as integrated project teams. They also wanted to be able to combine heterogeneous

IT systems into a complete package for the customer (a five-axis, high-speed milling machine, clamping solution, NC programs and measuring programs as well as comprehensive documentation for worldwide deployment). In addition to requiring durable production machinery, customers have come to expect more comprehensive after-sales services such as retooling, expansion, maintenance and warranty claims.

INTEGRATED SOLUTION IS A PERFECT FIT

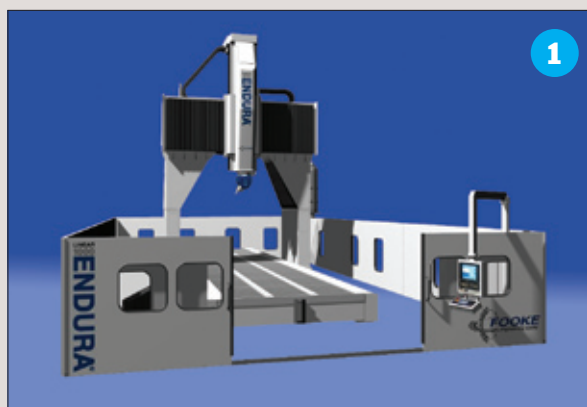
In 2004, the company began searching for a 3D CAD solution for its fifteen design engineers as well as a CAM module that supported high-speed and five-axis machining. "We looked at all the reputable systems on the market," says Hans-Jürgen Pierick, who as team leader of the systems' specialists coordinated the selection process. Five CAD systems with different CAM combinations were evaluated in conversations, demonstrations and trial installations.

Fooke chose an end-to-end, integrated product lifecycle management (PLM) approach from Siemens PLM Software that included the NX digital product development solution, the NX CAM system, the NX Nastran Finite Element Analysis (FEA) program and the Teamcenter digital lifecycle management solution. In addition, the company implemented the VNCK virtual NC system for machine-specific simulation of the Siemens 840 D CNC control. "It was a solution-oriented, unified concept that was a perfect fit for us," says Pierick.

The advantages of this solution became evident even during the pilot program. Having integrated CAD and CAM systems put an end to interface problems, savings hours of conversion labour. And having a common "language" – Teamcenter – improved cooperation among the different divisions.

ADVANCED MACHINES NOW POSSIBLE

Since 2006, all new Fooke machines have been completely developed on the Siemens platform. In particular the new models of gantry milling machines with overhead bridges –



1 The ENDURA 1000LINEAR milling machine was developed entirely using the Siemens platform



the ENDURA 900LINEAR with linear drive, and the travelling column milling machine ENDURA 1000LINEAR – show how the benefits of the new solution extend to the end user. This new generation of machines is characterised by an elevated, movable bridge. The use of FEA during the design process resulted in a bridge with higher rigidity, reliability and precision.

A machine of this type is used for the five-axis milling of the Superjet 100's outer skin, which is made of 1.5-millimetre aluminium (AlMg3) sheets. The bridge moves seven metres in the X axis, 3.5 metres in Y and 1.5 metres in Z. The A axis allows +120 to -95 degrees mobility, while the C axis allows +/-275 degrees. Innovative clamping technology consists of 200 actuators, each of which bears a vacuum suction cup and can be positioned via NC control. The positions of the individual actuators are programmed in the CAM module. The actual positions of the component are determined with Renishaw probes.

The Siemens 840 D was selected as the control system for these varied tasks. It has strengths not only in five-axis milling but also in the special applications of measuring, setting the neutral zero points and positioning the actuators. The CAM platform adds other strengths. "NX supplies a robust, open CAM system, which is expanded in a program written with Visual Studio .NET in order to output measuring and control programs for the Siemens 840 D," says Klaus Harke, CNC system specialist at Fooke. "This is followed by the programming of the five-axis contour processing."

The entire program can now be simulated using the virtual NC kernel VNCK with machine-specific parameters such as mass and inertia. As a result, for the first time it is possible to ensure that a concept solves a given problem without damaging any expensive components.

The benefits of the Siemens platform have been especially evident in this project. "Programming the machine during development made it available to the customer sooner," says Pierick. Computer simulation ruled out many of the risks associated with innovative machining techniques. In addition,

2 Fooke's travelling column milling machine, the ENDURA 1005LINEAR is one of a new breed of gantry milling machines with overhead bridges

showing the simulation to the customer created confidence in Fooke's problem-solving skills. It also facilitated implementation and training. The delivery of an entire CAM process that had been defined on one platform ensured a successful solution for the customer. Teamcenter tied together the entire package by providing immediate access to all the product information necessary for later retooling, maintenance and service.

FURTHER EXPANSION PLANNED

"The integration of the Siemens system is what brings the benefits," says Pierick. Of course, Fooke passes these benefits on to its customers. Every manufacturing facility meets customer needs with the application of key production machinery. The high performance of Fooke machines is a strong selling point, which is not to be underestimated in the capital goods business.

Due to these advantages, the digital product development system is currently being expanded. The company is going to use the viewer functionality of Teamcenter to make product information available to people involved in marketing and production. And with Fooke's software supplier, Siemens PLM Software, being part of the Siemens enterprise, Fooke has a single source and integrated solution for both its internal and external manufacturing environments.

www.fooke.eu

“THE INTEGRATION OF THE SIEMENS SYSTEM IS WHAT BRINGS THE BENEFITS... IT WAS A SOLUTION-ORIENTED, UNIFIED CONCEPT THAT WAS A PERFECT FIT FOR US”

SYNCHRONOUS TECHNOLOGY FOR MANUFACTURING

WHILE NX IS WELL KNOWN FOR ITS ROLE IN PRODUCT DEVELOPMENT WORKFLOWS, THE INTRODUCTION OF SYNCHRONOUS TECHNOLOGY INTO THE HEART OF THE SYSTEM MEANS IT CAN ALSO BE USED TO GREAT EFFECT BY THOSE WORKING IN MANUFACTURING

1 Synchronous Technology allows the NC programmer to create patches over openings with surfaces that blend into the surrounding model shape

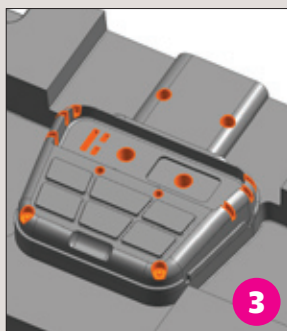
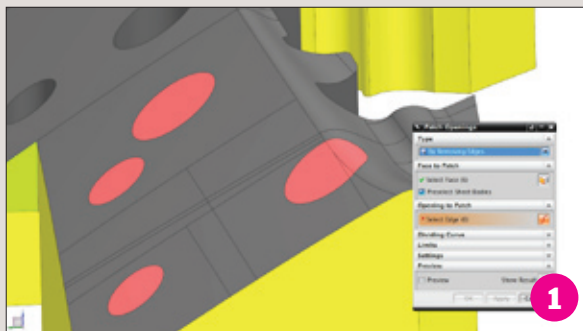
2 The move face command allows faces to be grabbed and dragged on any 3D model, maintaining any connection relationships and making the addition of machining stock much easier

3 Electrode design requires both the extract of the electrode geometry and the patching of the holes to create a contiguous surface for machining. Here the hole patterns have been selected in one operation and then deleted with the geometry closing up in a single operation

4 With Synchronous Technology it is possible to make some complex edits with a single selection on any 3D model such as this rib and adjacent fillets

There is no doubt that NX offers excellent tools for the manufacturing engineer to work with models from any source. The typical NC programmer can easily spend 20% or more of his or her time fighting with 3D models before even thinking of adding a tool-path. Radii may need to be changed, openings closed off, geometry added to drive NC tool-paths or containment volumes, or models even created from drawings.

Alongside this, manufacturing engineers also receive high volumes of third-party data that have to be translated into NX, removing intelligence. This can be hard to work with without a high degree of CAD expertise in traditional systems and a remodel is never far away once the clock starts ticking and the frustration levels rise. Even with native, fully defined 3D models containing all of the design features and parametric data, the manufacturing-focussed engineer may have no idea how it was created, may not have the tools needed to exercise those features to make the required edits, or simply not know how to use these functions.



What if the user could just click a face and patch in a hole, click a rib and drag it 10mm along the part and still leave it fully connected, fully blended as was, but in a new position, with no special modelling skills and without the dreaded remodel. This is what Synchronous Technology can offer in NX (both NX CAM and CAM Express) so let's explore a few common usage scenarios.

General model changes: There can be many reasons why a tool designer or NC programmer needs to change a model - perhaps to move a rib, to alter a draft angle, to move a hole or lengthen a pocket for manufacturability, or simply because of a good old-fashioned design change. This can happen after the part is programmed or the tool designed. No problems in NX; even if the 3D model was a translated dumb shape, with Synchronous Technology these changes are easy to do even with really limited CAD knowledge and with no idea how the model was built. Then because of the associativity that the NX system has always had - the tool design or the tool-path can be updated to suit the new geometry.

Model clean up: Many times imported models have dozens of modelling errors in them. Mismatching surfaces and gaps are common and these can throw off a CAM tool-path processor completely, often ending in a partial or complete remodel by the frustrated NC programmer, not to speak of hours wasted. NX has several levels of capability to 'repair' these broken models, finding the errors and giving the user options to heal the geometry ready for programming.

Closing openings: Finished part models don't represent the stage models that are needed to define machining operations. For example, holes in parts may often be the last features to be machined yet they appear in the finished part CAD model. For efficient machining from stock to the finished surface, pre-drilling operations, the NC programmer would like to have a model with the hole features suppressed. With Synchronous Technology the user can close off openings over holes or pockets in "dumb" models with patches that blend into the surrounding surface geometry - in a single click.

Castings & machining stock: The NC programmer or tool designer will often need a model that's larger than the finished part and more material may be required on one surface than another. With Synch Tech options the user can grab any surface and pull it into a new position with complex surfaces extending logically to keep the model intact - even on dumb geometry.

