Master the challenges of programming multi-function machines

SIEMENS

White Paper

By combining multiple machining operations such as milling and turning in a single machine, multiple-function machine tools reduce the number of setups required to process parts. Saving a considerable amount of time in the overall process, it also reduces positioning errors possible in multiple setups. One machine is able to do the work of multiple, separate machines. The cost comparison of multiple simpler machines, each with an operator, to a multi-function machine can be highly positive in favor of the more advanced machine.

However, multi-function machines make extra demands on CNC programming, postprocessing and part program validation in order to make the most of these more challenging machines. Mastering this complexity is key to achieving the additional levels of productivity and realizing an effective return on investment.

This paper discusses ways to address some of the key challenges in programming and utilizing complex multi-function machines with the latest CAM and simulation software.

Contents

Introduction

More and more companies are looking at multi-function machines, realizing they can combine multiple manufacturing steps into a single machine with a single setup. This consolidation can often achieve substantial financial savings by reducing overhead costs and increasing productivity.

It can be a challenge to generate a computer numerical control (CNC) program for a machine tool that has multiple machining devices with combinations of milling heads and turning turrets. These machines may have live tooling in a turret and may just as easily have a turning tool in a multi-axis head. They can often handle multiple workpiece positions such as two lathe spindles that can also be programmed as milling rotary tables and which can allow an in-process part to be transferred from one work-holding position to another. Seven or more programmable axes that are operational within a fairly limited work envelope are not uncommon, with machine tool components passing within very close proximity to each other.

To maximize the benefits, it is desirable to have multiple cutting tools simultaneously removing material. Simultaneous and synchronized cutting becomes even more challenging as cutting tools can often reach more than one part location or spindle at a time. Keeping track of the current state of the workpiece between cutting operations, as well as from spindle to spindle, can be critical for generating efficient cutter paths.

A separate G-code program is also usually required for each machining device (sometimes referred to as a programming channel). There is a separate coordinate system for main and sub spindle adding to the programming and postprocessing complexity. Putting this all together on the shop floor for a first run or prove out can take a significant amount of time. And time spent testing is time not cutting metal.

In the following sections we will consider some of the possible ways to address these challenges to take advantage of these highly productive machine tools. We will illustrate the solutions with examples from the Siemens NX™ CAM system and the Siemens Virtual Machine, but the general principles apply to whatever system or technology is being used to prepare jobs for these advanced machine tools.

Programming multi-function machines

CAM processors for multi-function machines

Multi-function machines cover a wide range of possible configurations. Clearly a CAM system needs to offer the range of capabilities available on the machine in one software package. Thus a milling CAM system won't be of much use if the machine offers milling and turning on the same machine – as many do. Depending on the machine configuration, programs might require any combination of turning, 3-axis milling, 3+2 positional milling and full 5-axis milling. Even prismatic parts can require this more complex level of processing on a mill-turn machine tool.

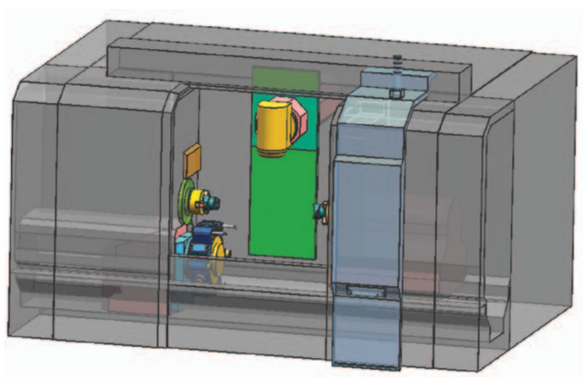

Figure 1: These complex mill-turns can have a lower turret and an upper turret or a B-axis head which can handle turning and milling operations on both the main and an optional sub spindle.

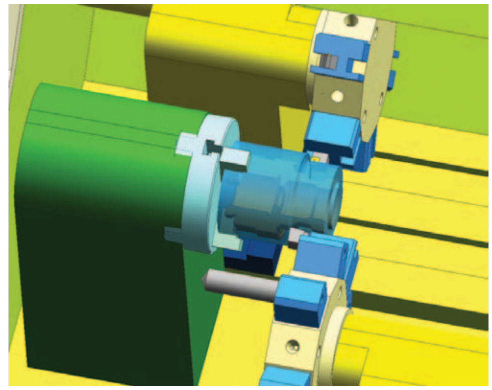

Figure 2: Merged (pinch) turning.

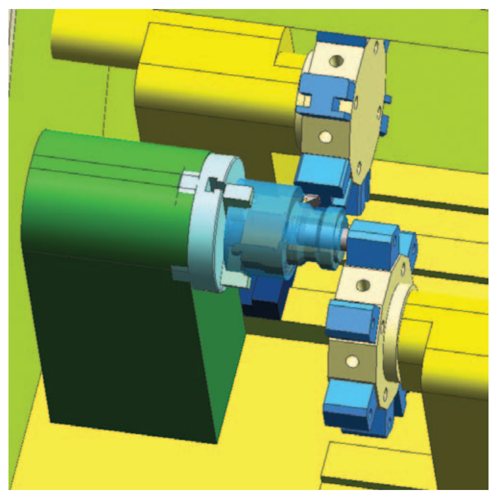

Figure 3: Synchronized turning.

Synchronized tool paths

On multi-function machines one workpiece is often addressed by more than one cutting tool at the same time, say in a pinch turning mode with turning tools tracking on the one part from a lower turret and a top head (or turret). Or completely different and independent operations may be performed by the different heads or turrets on the same or different spindles. In each case the synchronization between each operation becomes critical. In simple cases, it may be sufficient to time one operation to begin on one of the machine's channels at the same time another operation ends (or begins) on a different channel. This can be accomplished by operation-level synchronization. However, maximum productivity might be achieved by timing different motion channels at a finer level, so that one operation begins on one of the machine's channels at the point when another operation is partially complete. This would require synchronization at the G-code level.

One way to address this is to see the intended CNC code for the multiple device channels displayed in parallel and add codes called synchronization codes or marks into the code stream. With the Synchronization Manager in NX CAM from Siemens PLM Software, the CNC programmer can insert synchronization events and/or dwell events in the graphical display. At any point, the user can step through or play the simulation to see where the various multi-function machine components are going to be at any given time in the CNC program and make necessary adjustments to optimize the machine utilization. In some cases it is possible to place the synchronization codes into a CNC program at the operation level. For fine control the codes need to checked (and possibly adjusted) in the display of G-code for each parallel channel. To work with the G-code output of the postprocessor is a key issue for successful programming of multi-function machine tools.

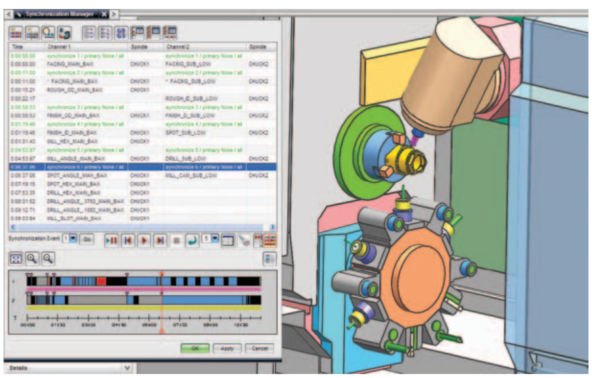

Figure 4: Multi-channel synchronization time chart.

Most CAM systems with a multi-channel synchronization capability will also provide a time chart display – like the one from NX CAM shown in Figure 4. The display shows the relative positions of the synchronization codes and periods of machining or waiting across as many channels as the machine can offer providing a graphical indication of the balance of machining operations across the available devices.

Linking the synchronization tool to the machine tool simulation display inside the CAM system can be especially useful. The connection of the synchronization to the 3D model simulation function inside the CAM system provides a much more complete view of what is moving, where and when – all in the same system. It's easier to spot errors, see displays of collisions (or near misses), for both cutting and noncutting moves.

G-code driven machining simulation

Unless the detailed synchronization at the G-code level is to be done outside of the CAM programming system as a separate and later task, it is necessary to have direct access to a view of the postprocessor output code (the "G-code") while using the synchronization application. This can be difficult if the postprocessor is a separate application or not tightly integrated within the CAM software.

The immediate access to an integrated postprocessor also allows the CAM system to offer machine tool simulation that's driven off of the output from the postprocessor and not from the internal tool path data. Machine simulations of the internal tool path are useful for basic checking and provide some level of confidence, but some CAM systems can additionally offer simulation driven by the postprocessed output. Especially on advanced multi-function machines, simulation is important, and the availability of G-code driven simulation makes sure that the effects of postprocessing are included and accounted for.

Postprocessing

Postprocessing is a critical element of the multi-function machine tool programming solution. Such a machine can often require multiple postprocessor functions to address the various combinations of milling and turning.

Channel	Main spindle	Subspindle
Upper turret milling	Post-1	Post-1
Upper turret milling	Post-3	Post-2
Lower turret milling	Post-1	Post-1
Lower turret milling	Post-3	Post-3

Figure 5: A system of individual postprocessors for particular methods or devices can be automatically connected into a linked postprocessor.

One approach is to create one large postprocessor that strings the entire set together. The challenge with this approach is that the one postprocessor is very complex, hard to write and even harder for someone else to edit. Another method is to arrange a system of postprocessors – one for each key function and device – and then connect these with a “link postprocessor”. The result is a logical structure that is easier to develop and edit later. Each linked postprocessor is associated with a particular device or method and the system takes care of selecting the correct postprocessor automatically.

In-process workpiece

In order to eliminate time wasted “cutting air”, a CAM system needs to manage the state of the in-process workpiece (IPW) from one operation to the next. As material from the workpiece is removed by a tool path, the result becomes the blank for the next operation. Cutting tools can then be driven to safely approach the part in rapid mode to within a predefined minimum clear distance before switching into a feed rate. In these complex multi-function machines with multiple work-holding positions there is a need to transfer the in-process workpiece definition from one location to another, say from the main spindle to the sub spindle. The machining operations performed on the workpiece in the main spindle, becomes the blank for the workpiece on the sub spindle.

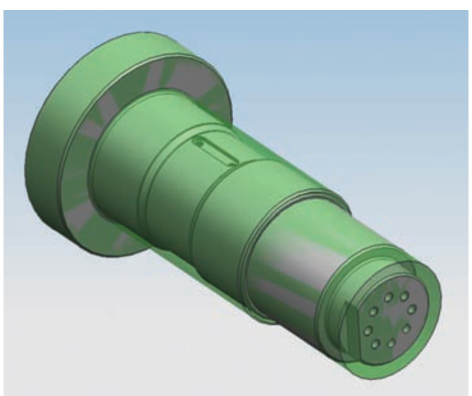

Figure 6: Workpiece and blank.

The system should keep track of the IPW, if program operations are resequenced, or the model changes. The user only needs to regenerate the program, not start over. This is a valuable aid for rapid and accurate programming.

Coordinate systems

When working with multi-function machines, it is important to be able to select the most appropriate or useful coordinate system for each set of operations. Being able to arrange a separate coordinate system for each device in the CAM setup can make it easier to program and much easier to edit and debug the CNC programs. While creating an operation, the programmer then simply chooses the correct device and each will have its own machining coordinate system (MCS) for its respective program zero.

Programming or production – Not wasting machine time

Highly skilled machinists with very detailed knowledge can and do prepare very efficient part programs right at the machine tool using the controller interface. However, this means that the machine itself can become a part of the programming and debugging system so that it ties up the machine while being programmed. But programs prepared in CAM systems are also typically validated at the machine tool, and the setup work for each new job is another lengthy task that takes up machine time.

One way to overcome this challenge is to use a virtual machine – a functional copy of the real machine tool and its controller that runs only in software, but is more complete and accurate than a typical machining simulation system. The idea is that the virtual machine can behave just like the real one – its 3D model of the machine, workpiece, tooling and any work-holding devices, are driven on the computer screen by software from the real controller.

In the Siemens PLM Software Virtual Machine, the core controller software from the Siemens SINUMERIK controller is built right into this advanced simulation system. The motion control software (drives, acceleration curves, etc.) and the user interface software (menus, dialogues and onscreen selection buttons) from the Siemens SINUMERIK controller provide simulation of the machining process that’s as close to reality as one could need for editing and checking CNC part programs.

Users of the virtual machine report up to 90% reductions in machine setup time – which of course can translate into more time for the machine to cut metal and make money.

Figure 7: Machinists can verify NC programs and machine setup for the next job by using the virtual machine without tying up the physical machine tool.

About Siemens PLM Software

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 6.7 million licensed seats and more than 69,500 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit *www.siemens.com/plm*.

Siemens PLM Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
972 987 3000
Fax 972 987 3398

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
800 498 5351
Fax 972 987 3398

Europe
3 Knoll Road
Camberley
Surrey GU15 3SY
United Kingdom
44 (0) 1276 702000
Fax 44 (0) 1276 702130

Asia-Pacific
Suites 6804-8, 68/F
Central Plaza
18 Harbour Road
WanChai
Hong Kong
852 2230 3333
Fax 852 2230 3210

www.siemens.com/plm

X3 4746 12/10 C