

## Exporting and importing user settings and templates

It is possible to export all user-specific files such as user, quality, templates, NC codes, jobs and automation templates as well as all saved settings and registered values. This data can then be re-imported to another computer.

Under the DCAMCUT tools there are the buttons Export user files... and Import user files... buttons. This allows you to transfer all settings and user templates from one computer or user to another computer or user.

Setting file is located in the user directory (dcamcut.default\_values)

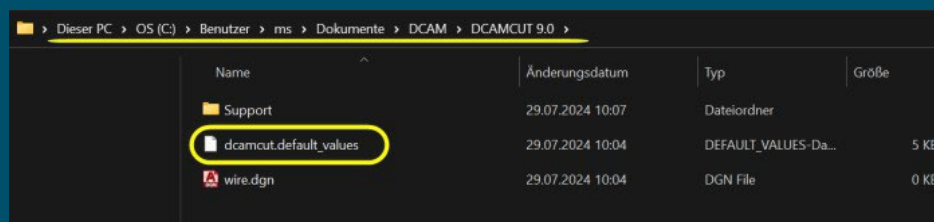
Therefore, settings are now also user-dependent (each user has their own settings)

Export ends up in documents (service set folder)

### Structure:

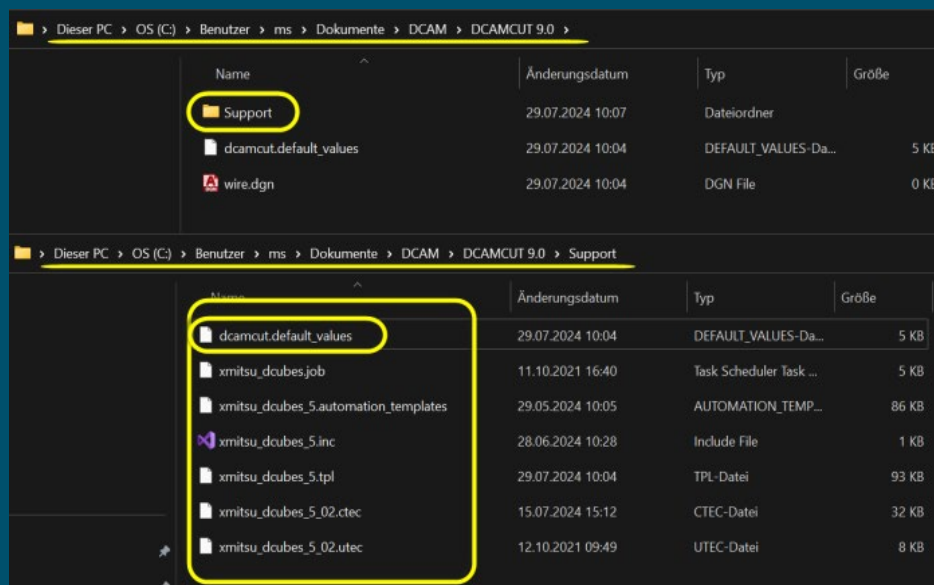
C:\Users\ms\Documents\DCAM\DCAMCUT 9.0 -> dcamcut.default\_values

All settings are saved here.



C:\Users\ms\Documents\DCAM\DCAMCUT 9.0\Support

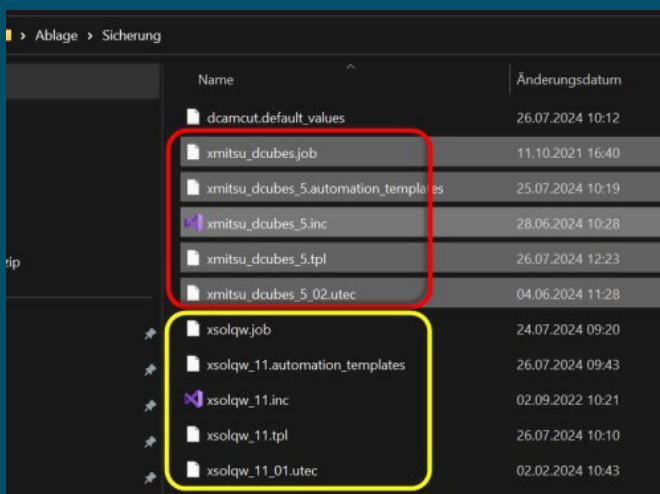
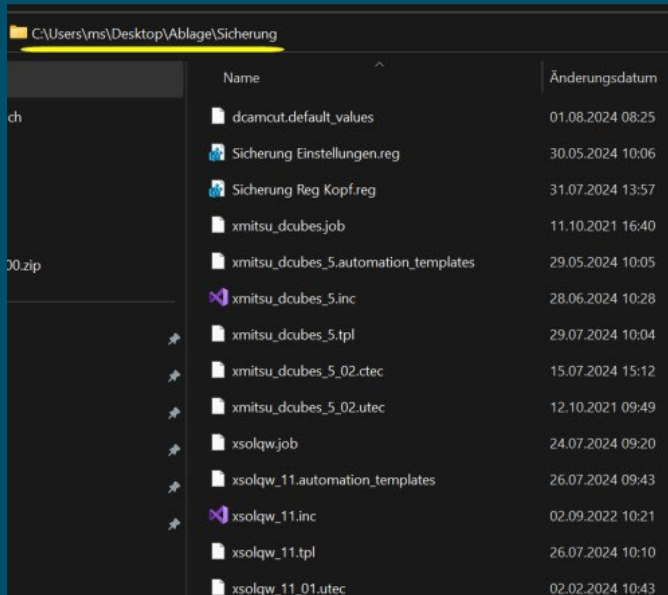
All files required for export are stored here.



(These can be saved and stored separately)

Files stored in the Backup folder on C:\Users\ms\Desktop\File  
When importing, specify the folder in which all files created during export are stored.

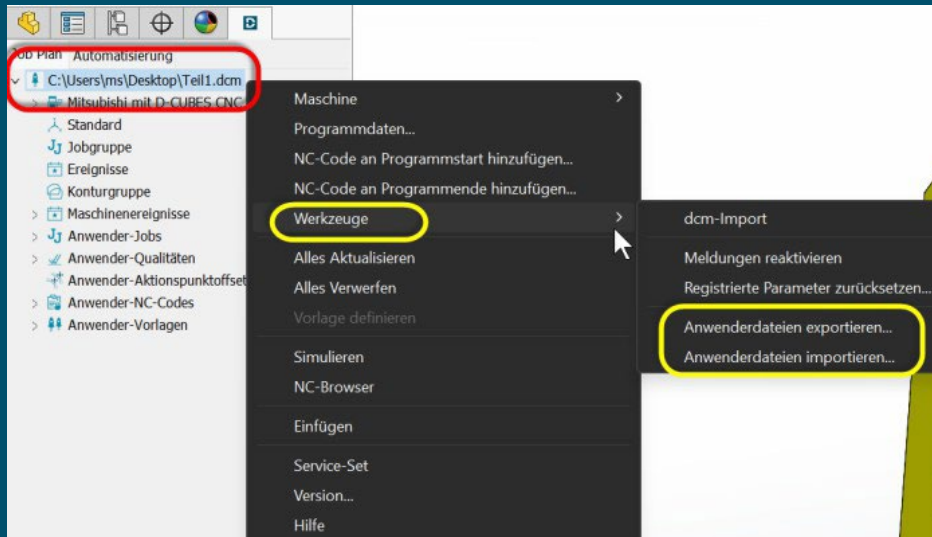
In this example, C:\Users\ms\Desktop\Folder\Backup



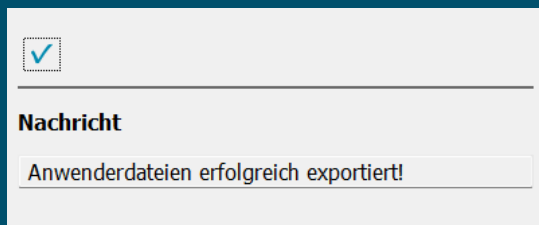
## Procedure:

A file is open and wire cutting is activated. Machine is selected. There are templates for user qualities, user NC codes and other user-specific settings and templates.

Open the context menu on the 1st root node and select the Export user files... option under Tools.



The message Files created successfully appears.



Copy the displayed user files from the open Explorer and save them in a desired path.

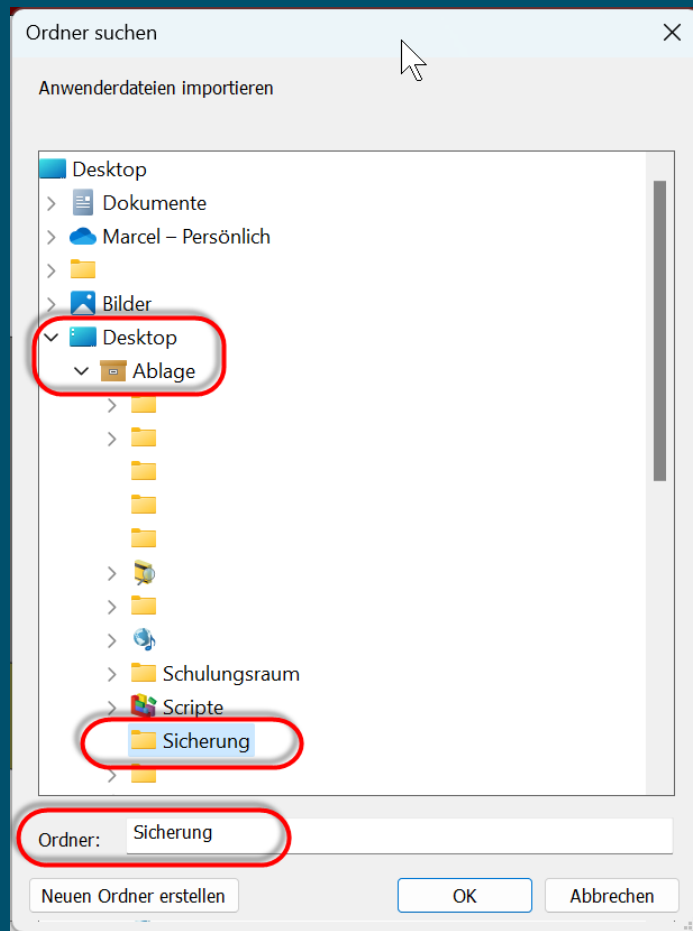
Close the file and close DCAMCUT.

A new computer is set up with a fresh DCAMCUT installation. Or a new user is added to the current system.

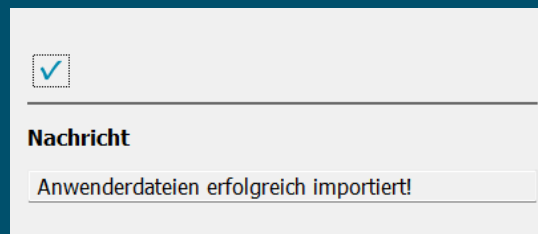
Open a file either with or without editing. If necessary, activate wire cutting in this part.

Now open the context menu in the 1st root node again and select the Import user files... option under Tools.

The Explorer opens and asks you to specify the folder containing the user files.



Select the folder and confirm with OK. The message Files imported successfully appears.



Close DCAMCUT.

When reopening, the corresponding machine is opened and all user files such as templates, qualities and user NC codes as well as all registered values in dialogs have been transferred.

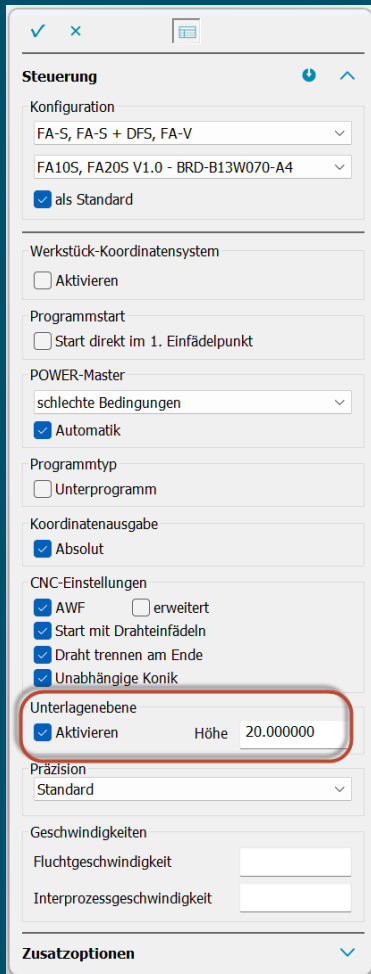
## Additional document level for Mitsubishi Advance and DCUBES machines

When programming the machining process, the exact clamping situation of the workpiece on the machine is often not known. Programming is usually carried out for a clamping operation directly on the machine table.

The solution created enables the machine operator to react easily to a changed clamping situation, such as clamping on a bar, by changing the parameter value H400 in the NC program header.

There is no need to intervene via the programming system or directly at many points in the NC program.

This function is available for all machining operations with the exception of those that use additional axis.



**Steuerung**

Konfiguration  
 FA-S, FA-S + DFS, FA-V  
 FA10S, FA20S V1.0 - BRD-B13W070-A4  
☒ als Standard

Werkstück-Koordinatensystem  
☐ Aktivieren

Programmstart  
☐ Start direkt im 1. Einfädelpunkt

POWER-Master  
 schlechte Bedingungen  
☒ Automatik

Programmtyp  
☐ Unterprogramm

Koordinatenausgabe  
☒ Absolut

CNC-Einstellungen  
☒ AWF ☐ erweitert  
☒ Start mit Drahteinfädeln  
☒ Draht trennen am Ende  
☒ Unabhängige Konik

**Unterlagenebene**  
☒ Aktivieren Höhe 20.000000

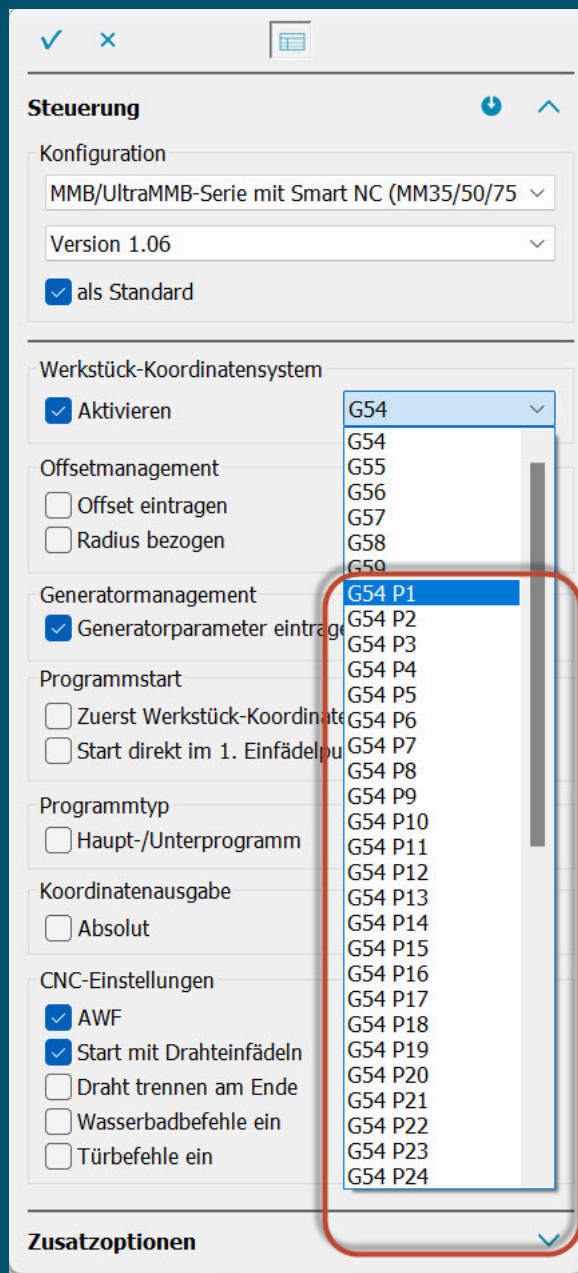
Präzision  
 Standard

Geschwindigkeiten  
 Fluchtgeschwindigkeit  
 Interprozessgeschwindigkeit

**Zusatzoptionen**

```
%
L0001 /DEMO
N00001 ()
N00002 H400=20.0
N00003 H900=0.0
N00004 G90
N00005 M91
N00006 G00 X-53.958 Y31.308
N00007 (KONTUR.ZYLINDRISCH-1-KOMPLETT-VORWAERTS-
HAUPTSCHNITT-KONIKMODUS.EIN)
N00008 G92 X-53.958 Y31.308
N00009 Z1=0.0+H400 Z2=10.0+H400 Z5=20.0+H400
N00010 M20 (DRAHT.EINFAEDELN)
N00011 M90
N00012 M102
...
```

## Further developments and changes Seibu processors



### All SEIBU's:

- Option of output as pure subroutine removed from machine dialogs

### Output as main/subprogram for enabled machines:

- Subroutine termination changed to M02 instead of M99

### Coordinate systems for all machines from MA series onwards:

- Extended to additional group G54 P1 - G54 P48

### Machines with straightness compensation:

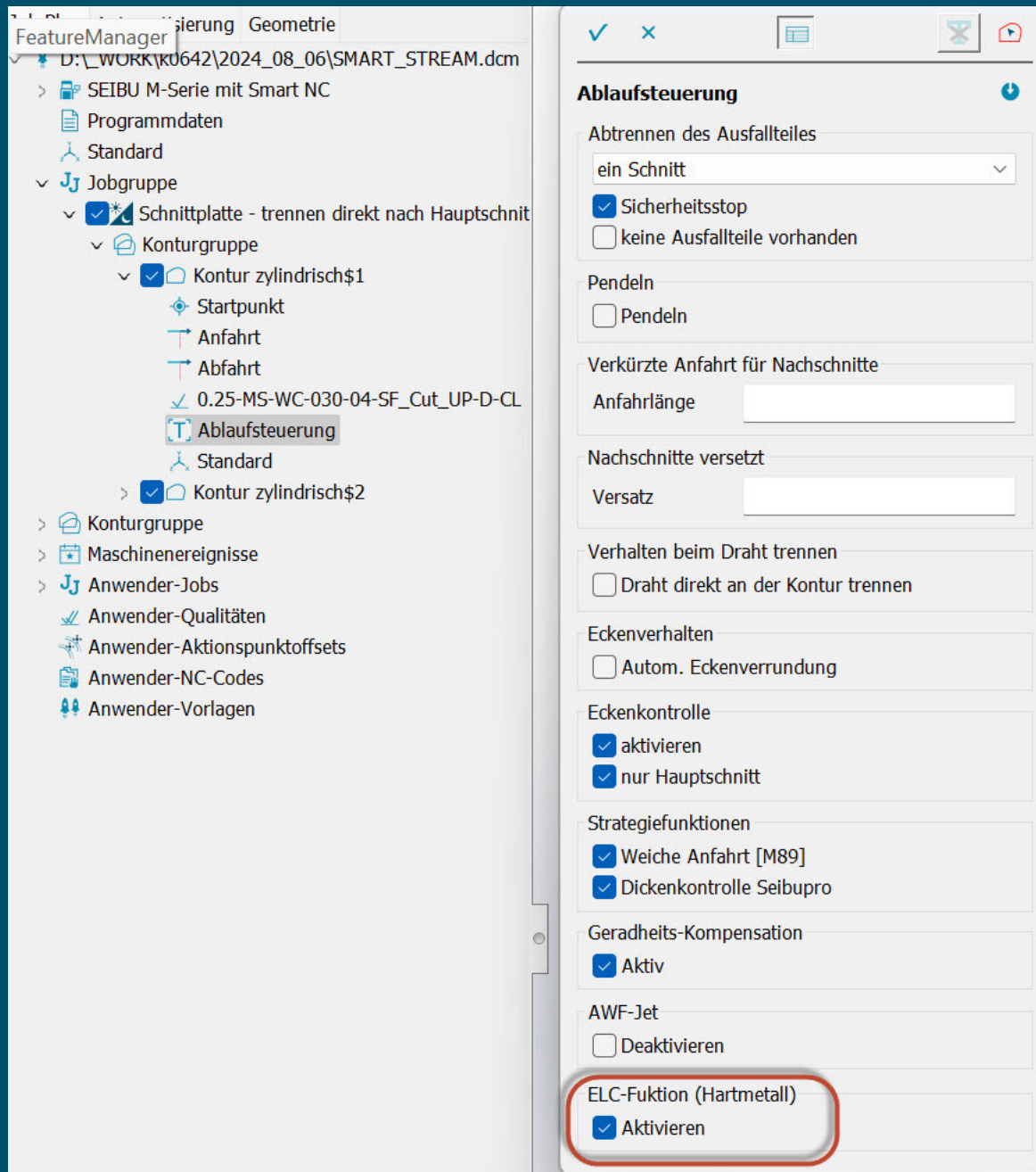
- The program level (J) for straightness compensation of 2D contours is set to the top edge of the workpiece, the height (I) is set correspondingly negative

### Strategic M commands are provided with a corresponding comment.

### Seibu with Smart NC

- Implement ELC function for last cut of carbide machining





## STOP events implemented for all Seibu processors

