



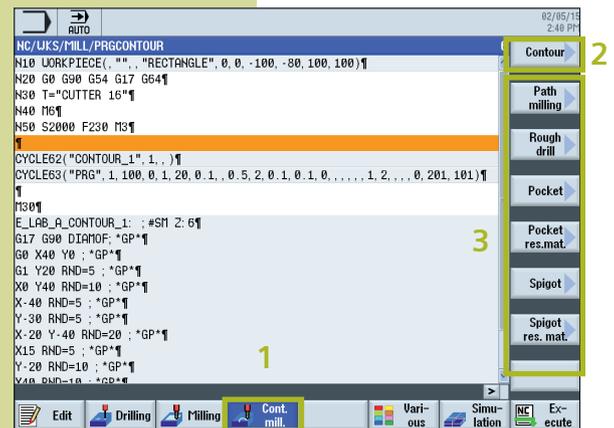
## Contour programming in programGUIDE

# Effectively program complex contours

Contour programming is the solution for effectively programming complex contours in turned and milled workpieces. With the aid of the integrated contour editor and the corresponding machining cycles in Sinumerik Operate, contours can be created and processed directly on the CNC control system. We briefly demonstrate the possibilities using the example of contour programming during milling in conjunction with programGUIDE. <

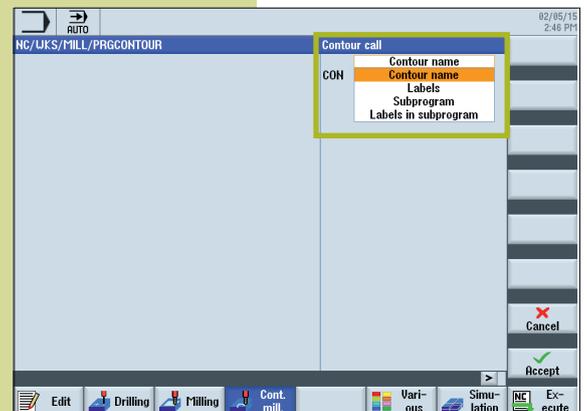
### Contour milling

1. The contour definition and machining are available in the program editor via the "Contour Milling" soft key.
2. The "Contour" soft key defines the contour and programs the contour call-up.
3. The other soft keys represent the cycles for the defined contour, for example, the cycles for path milling and for pocket and stud machining.



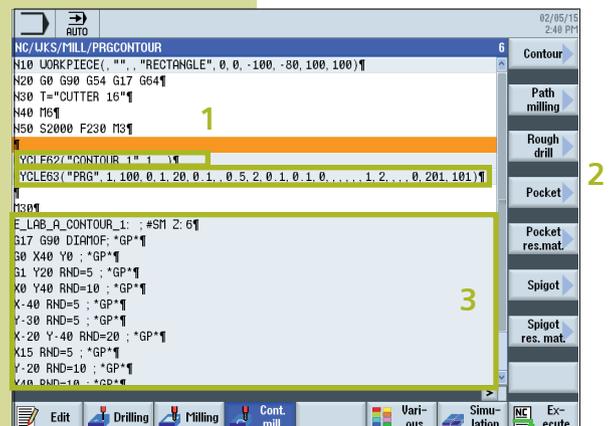
### Contour definition and contour call-up in the program

Several options are available for contour definition. The contour is contained either in the main program or in the subprogram. The contour call-up also varies accordingly: a contour created in the main program is created in a label, whereas a contour in a subprogram is created within a label in the subprogram.



### Call-up using contour name

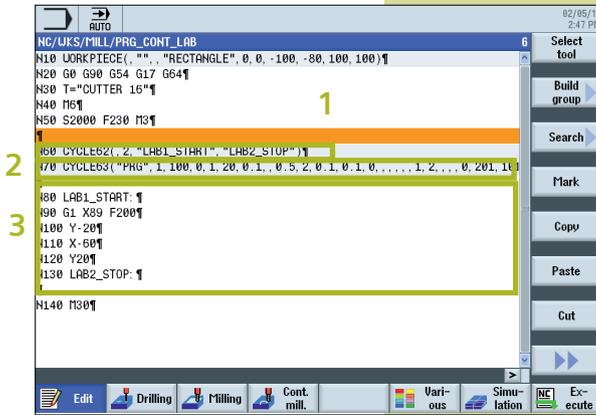
1. Call-up of the contour named "CONTOUR\_1" using CYCLE62 (call-up cycle).
2. Machining cycle (e.g., CYCLE63 milling studs); this relates to the previous CYCLE62.
3. Contour defined using the contour editor; this may also be before M30.





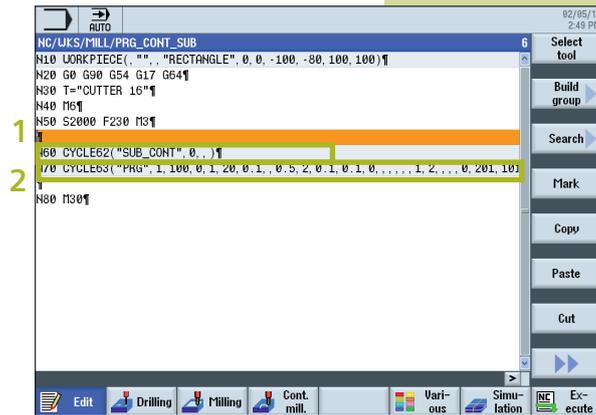
## Call-up using label

1. Call-up of the program code in the label.
2. Machining cycle (e.g., CYCLE63 milling studs); this relates to the previous CYCLE62.
3. LAB1\_START: and LAB2\_STOP: form the start and end point of the contour.



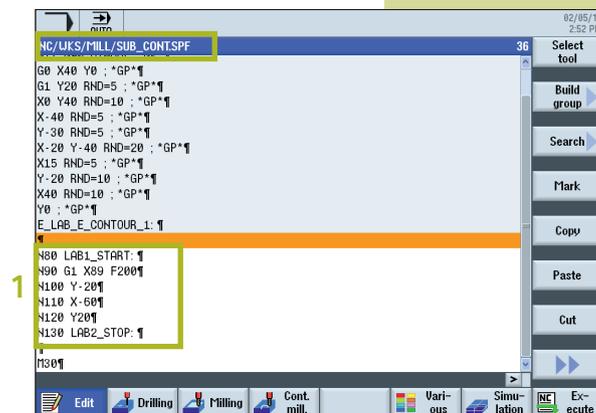
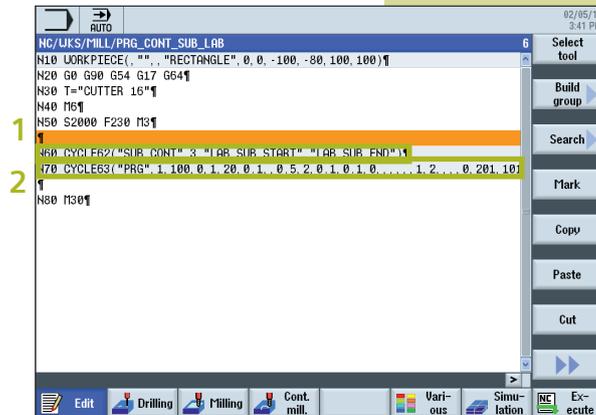
## Call-up using subprogram

1. Call-up of the program code (or contour) contained in the subprogram "SUB\_CONT".
2. Machining cycle (e.g., CYCLE63 milling studs); this relates to the previous CYCLE62.



## Call-up using label in subprogram

1. Machining of the contour contained in the subprogram "SUB\_CONT" between the two labels "LAB\_SUB\_START" and "LAB\_SUB\_END".
2. Machining cycle (e.g., CYCLE63 milling studs); this relates to the previous CYCLE62.



INFO AND CONTACT

[siemens.com/cnc4you](http://siemens.com/cnc4you)