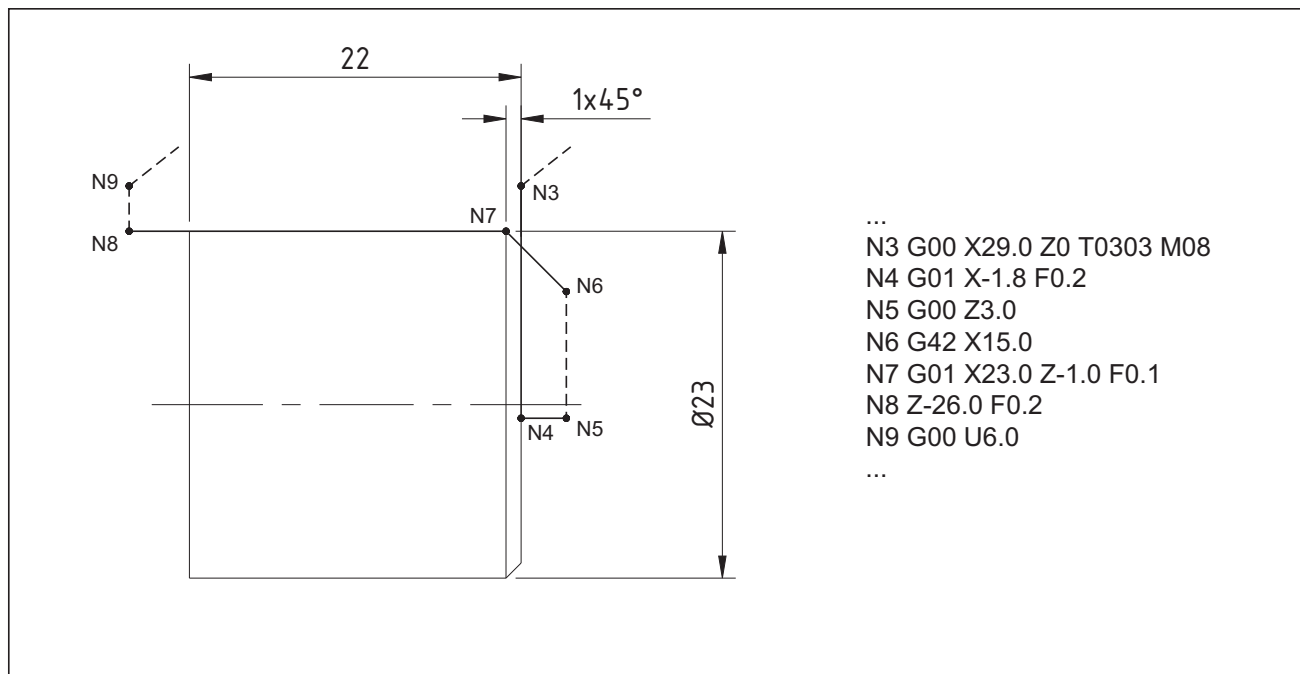


## Face - Chamfer - Diameter

The basic turning program is quite straightforward, once you follow certain rules:

- ❑ For tool nose radius compensation, any physical clearance amount must be minimum **two times the tool radius** (3 mm used in this exercise was more than necessary)
- ❑ Make sure the tool does not hit the stock, particularly at approach
- ❑ When the face is completed, move always in Z-axis only, by at least the minimum clearance (see **item 1**)

The following illustration shows the critical program excerpt with corresponding block numbers:



Here is the complete program, as required by the assignment:

```
(34-01 - FACE - CHAMFER - DIAMETER)
(T03 - OD TURNING TOOL)
N1 G21 T0300
N2 G96 S125 M03
N3 G00 X29.0 Z0 T0303 M08
N4 G01 X-1.8 F0.2
N5 G00 Z3.0
N6 G42 X15.0
N7 G01 X23.0 Z-1.0 F0.1
N8 Z-26.0 F0.2
N9 G00 U6.0
N10 G40 X100.0 Z100.0 T0300 M09
N11 M30
%
```