Training Document for
Integrated Automation Solutions
Totally Integrated Automation (TIA)

Module S01
Fundamentals of CNC Programming
with SinuTrain
This document was prepared by Siemens AG for training purposes for the project ‘Siemens Automation Cooperates with Education’ (SCE).
Siemens AG does not guarantee the content of this document.

Passing on this document, copying it and using and sharing its content is allowed within public training facilities and continued education facilities. Exceptions require written approval by Siemens AG (Michael Knust michael.knust@siemens.com).
Offenders are held liable. All rights reserved, including translation rights, particularly in the event a patent is granted or a utility model or design is registered.

We thank the Michael Dziallas Engineering corporation, the teachers of vocational schools others for their support during the preparation of this document.
# Table of Contents

1 **Preface** ......................................................................................................................... 5

2 **Introduction** .................................................................................................................. 7

   2.1 Development Phases of CNC Technology .............................................................. 7

   2.2 Controller Requirements for the New Millennium ............................................... 7

   2.3 Advantages of CNC Programming with SinuTrain SHOPTURN ....................... 7

3 **Programming a Turned Part in G-Code** ................................................................. 8

4 **Programming a Turned Part according to the Machining Step Procedure** ........ 9

   4.1 Starting SINUTRAIN .............................................................................................. 9

   4.2 Selecting the Machine .......................................................................................... 10

   4.3 SINUTRAIN Basic Display ............................................................................... 10

   4.4 Generating the Turning Program ....................................................................... 11

   4.5 Setting Up a New Directory/Program Generation ........................................... 11

   4.6 Setting Up a New Workpiece Program ............................................................. 12

   4.7 Program Header .................................................................................................. 13

   4.8 Program Editor ................................................................................................... 14

   4.9 Tool Selection ..................................................................................................... 14

   4.10 Face Turning ...................................................................................................... 15

   4.11 Workpiece Contour .......................................................................................... 17

   4.12 Stock Removal Cycle Roughing ................................................................... 22

   4.13 Finishing the Outside Contour ...................................................................... 23

   4.14 Thread Undercut ............................................................................................ 23

   4.15 Thread Cutting ............................................................................................... 24

   4.16 Making Grooves ............................................................................................. 25

   4.17 Simulation ......................................................................................................... 26

5 **Programming a Milled Part in G-Code** ................................................................. 28

6 **Programming a Milled Part according to the Machining Step Process** .......... 31

   6.1 Start SINUTRAIN ............................................................................................. 32

   6.2 Selecting the Machine ...................................................................................... 32

   6.3 SINUTRAIN Basic Display ............................................................................ 32

   6.4 Setting Up a New Directory/Program Generation ......................................... 33

   6.5 Setting Up a New Workpiece Program ......................................................... 34

   6.6 Program Header .................................................................................................. 35

   6.7 Outside Contour .................................................................................................. 36

   6.8 Roughing the Outside Contour ...................................................................... 37

   6.9 Selecting the Tool ............................................................................................ 37

   6.10 Finishing the Outside Contour ...................................................................... 38

   6.11 Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contour .......................... 39

   6.12 Stock Removal of Contour Pocket Roughing ................................................... 41

   6.13 Removing Residual Material from the Corners ............................................. 42

   6.14 Finishing the Base .......................................................................................... 43

   6.15 Finishing the Edge .......................................................................................... 44

   6.16 Roughing the Circular Pocket Diameter 60................................................... 45

   6.17 Finishing the Circular Pocket Diameter 60..................................................... 45

   6.18 Roughing the Circular Pocket Diameter 30.................................................... 46

   6.19 Finishing the Circular Pocket Diameter 30.................................................... 46

   6.20 Centering and Drilling taking Obstructions into Account ............................ 46

   6.21 Machining Step Centering .......................................................................... 46

   6.22 Machining Step Drilling ............................................................................... 47

   6.23 Inputting the Line of Holes to the Left ........................................................... 47

   6.24 First Obstruction ............................................................................................ 47

T I A Training Document Page 3 of 53
Status: 01/2010
Fundamentals of CNC Programming with SINUTRAIN
<table>
<thead>
<tr>
<th>Section</th>
<th>Task Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.25</td>
<td>Inputting the Line of Holes to the Right</td>
<td>48</td>
</tr>
<tr>
<td>6.26</td>
<td>Second Obstruction</td>
<td>48</td>
</tr>
<tr>
<td>6.27</td>
<td>Drilling the Full Circle</td>
<td>48</td>
</tr>
<tr>
<td>6.28</td>
<td>Third Obstruction</td>
<td>48</td>
</tr>
<tr>
<td>6.29</td>
<td>Drilling a Single Hole</td>
<td>48</td>
</tr>
<tr>
<td>6.30</td>
<td>Simulation</td>
<td>49</td>
</tr>
<tr>
<td>7</td>
<td><strong>Additional Tasks for Turning</strong></td>
<td>50</td>
</tr>
<tr>
<td>7.1</td>
<td>Step Shaft</td>
<td>50</td>
</tr>
<tr>
<td>7.2</td>
<td>Shaft02</td>
<td>51</td>
</tr>
<tr>
<td>8</td>
<td><strong>Additional Tasks Milling</strong></td>
<td>52</td>
</tr>
<tr>
<td>8.1</td>
<td>Flange</td>
<td>52</td>
</tr>
<tr>
<td>8.2</td>
<td>Exercise11</td>
<td>53</td>
</tr>
</tbody>
</table>
1 Preface

This training document ‘Fundamentals of CNC Programming’ is the prerequisite for working with the additional modules with SINUTRAIN.

Today, CNC controllers are considered the most essential part of any automation. Depending on the problem definition, the most varied tasks in the areas of turning, milling, lasering, and grinding as well as in many other areas can be carried out economically with the controllers ShopMill and ShopTurn.

Training objective:

The CNC S01 module is structured in a way that you have before you a completely programmed workpiece in G-code programming (DIN 66025). The same workpiece is generated step by step with this document according to the machining step programming. In this way, you will learn the simple operation of this software, and control and optimize it with the simulation.
Required Hardware and Software

1. PC, operating system Windows XP Professional starting with SP1 with 500 MHz and 256 MB RAM, free disk storage approx. 400 MB, of that 50 MB on the system drive, 1GB for installing all products, MS Internet Explorer starting with 6.0

2. Software SINUTRAIN 802D/810D/840D/840Di/Programming &Training, SinuTrain/JopShop
2 **Introduction**

2.1 **Development Phases of CNC Technology**

- At the beginning of the eighties, first CNC machines with simple controllers
- In the middle of the eighties, more powerful controllers with cycles because of faster processors as well as machine tools with greater processing speed
- At the end of the eighties, machine tools with 5 and more axes and special software tools for external programming by using CAD/CAM systems
- At the beginning of the nineties, flexible manufacturing systems with extensive supplementary functions such as palette systems and multiple clamping with multiple spindle drives
- In the middle of the nineties, continued development of tool systems and the use of special tools for processing complex workpiece contours with only one tool
- End of the nineties: central programming systems for programming several different controllers at different machine tools

2.2 **Controller Requirements for the New Millennium**

- Openness: It is to be possible for the machine manufacturer or the user to configure and expand controllers according to their own requirements
- Independence: Programming by means of a uniform controller interface for the most varied CNC processing
- Equality: All machine data is to be available also at the external programming units. Programming at the external programming units is the same as on the machine tool.
- Saving programming time: With graphic machining plans and help displays, it is to be possible to generate complex workpiece contours very easily and quickly
- Editing capability: Extensive editing functions provide for fast and simple program changes/program expansion

2.3 **Advantages of CNC Programming with SinuTrain SHOPMILL, SHOPTURN**

The controller is continuously optimized and can be adapted any time to the individual requirements of the machine manufacturers. Moreover, cycles and functions can be integrated later.

Regardless of whether turning, milling, or any other type of processing is performed, always the same program interface and the same menus or functions are used.

Retrofit: This means: Siemens can retrofit also older CNC machines to ShopMill and ShopTurn. Advantage: Operating the software and the menu structure has to be learned only once.

By transferring the machine data to the programming system of SINUTRAIN, programming at the external programming unit is the same as on the machine tool.

By using contour calculators and CAD readers, simple programming is possible without technical terms. By directly entering technological values, no external calculations have to be made beforehand. The integrated contour calculator is able to process all conceivable dimensions, yet is very simple to handle. Through work step programming and many online help functions, extensive programming tasks can be solved very quickly.

Convenient programming is possible with functions such as Copy, Cut, and Insert. Since the program is generated in the editor as a graphic machining plan by means of individual work steps, all editing steps are provided in a straightforward arrangement.
3 Programming a Turned Part in G-Code

<table>
<thead>
<tr>
<th>Schrupper = roughing tool</th>
</tr>
</thead>
<tbody>
<tr>
<td>Schlichter = finishing tool</td>
</tr>
<tr>
<td>Gewindestahl = threading tool</td>
</tr>
<tr>
<td>Stecher = grooving tool</td>
</tr>
</tbody>
</table>

```
T="SCHRUPPER_00"
G96 S250 LIMS=5000 M4 M8
G18 G54 G90
G0 X80 Z2 I1
G1 X-1.6 F3
G0 X80 Z2 I1
CYCLE95(‘KONTUR’,3.5,0,1,0.5,0.1,0,3,0.15,,1,0,0,1)
G0 X200 Z2 I1
T="SCHLICHTER_35"
G96 S300 M4 M8
G0 X-1.8 Z1 I1
G42 I1
G1 X8 Z2 I1
Kontur_Schl1 I1
G40 I1
G0 X200 Z100 I1
T="GEWINDESTAHLE"
G95 M3 M8 I1
G0 X30 Z2 I1
CYCLE97(1.5,.,0,-17,30,0,0,0,92,0,03,30,0,2,1,1,0)
G0 X200 Z100 I1
T="STechER"
G96 S120 P.07 M4 M8 I1
G0 X82 Z-50 I1
CYCLE93(60,-56,4,3,0,0,0,0,0,0,0,0,0,1,0,1,0,1,0,1,3,0,2,11,0)
G0 X200 Z100 M30 I1

==eof==
```
4 Programming a Turned Part according to the Machining Step Procedure

Using this drawing, we will generate a program in the machining step procedure.

4.1 Starting SINUTRAIN

To start SinuTrain, double click on Start SinuTrain, or start SinuTrain using the start bar.
4.2 Selecting the Machine

After booting up the computer, a menu is displayed with a selection of machines with the respective version number.

1. Select turning machine with >> ShopTurn Open V 06.04 <<
2. Click the Start button

4.3 SINUTRAIN Basic Display

After selecting the machine, the basic display of SINUTRAIN appears. From here, all SINUTRAIN actions are started.
4.4 Generating the Turning Program

4.5 Setting Up a New Directory/Program Generation

Various actions can be performed in the Program Manager (for example: New, Rename, Copy, etc.)

By pressing the softkeys **[MINI][SELECT]**, the available directories are opened in the operating area **DIRECTORY <<VERZEICHNIS>>** under ShopTurn.

With the softkey **Neu** , we are now setting up a new directory with the name "Work pieces" <<Werkstücke>>.

Confirm the input with **OK**.
The directory with the name “Work pieces“ is set up.

By pressing the arrow key on the CNC keyboard or with the cursor key toward the right, the selected directory opens.

**Note**
The cursor keys can be used for navigating in SinuTrain.

### 4.6 Setting Up a New Workpiece Program

After the directory is opened, existing programs are displayed, or new programs can be set up.

By pressing the softkey and the selection Shopturn or G-Code (vertical softkeys), a new program name “Name of the machining plan” is entered, in our case “Shaft”<<Welle>>.

Confirm input with **OK**.
4.7 Program Header

The program header of the new program opens automatically. In the program header, the following is entered: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.

With the softkey you can now call the associated help display.

With the softkey you can alternate, for example, between different zero points, units of mass, mm, inch, raw part shapes, cylinder, pipe, rectangle, corner, etc. The blue line above always shows you the current input type.

Make all inputs as indicated in the display above and accept the input into the machining plan.

Note

With the softkey you can alternate, for example, between different zero points, units of mass, mm, inch, raw part shapes, cylinder, pipe, rectangle, corner, etc. The blue line above always shows you the current input type.
4.8 Program Editor

Entered/edited in the program editor in the form of machining steps.

![Program Editor](image)

4.9 Tool Selection

First, you need a tool for face turning and for roughing the outside contour.

Click on Werkzeuge

![Tool Selection](image)

Then, by clicking on “Tools” again, call the tool list. Here, the tools that have already been set up are provided. But you can also create new ones.

Accept the roughing tool. The finishing tool and the threading tool are added later from the tool list. The grooving tool has to be set up, however.
Now, enter the data for the constant cutting speed and the plane.

Accept the values with the input key.

If needed, change the unit or selection with

and finally confirm your input with

The roughing tool is included in the program, with all the data.

4.10 Face Turning

Positioning for face turning in rapid feed.
By using line/circle, we can program a travel movement.

Click on

Enter X80 Z0.1 and click on

Confirm with
Next, face the front of the workpiece to X-1.6 (2 x plate radius size R=0.8) so no residual material remains in the center.

Click on and enter X-1.6 and F0.3 and mm/U

Confirm with

Now, position on X90 Z5 at rapid feed.
After accepting, you are back in the program.

4.11 Workpiece Contour

The contour is generated with the contour calculator.

Click on the button below and then in the upper right on Assign a name to the new contour; then the contour calculator opens.

The first input specifies the starting point.

Enter X0 Z0 and confirm with

**Note**

With the buttons we describe the contour.
Here, the first diameter of the finished part contour is described. A chamfer 2 x 45 degrees as transition element is appended to the end of the element.

Click on the button and enter X30 and FS 2.

Click on the button and enter Z-20.

Click on the button and enter X40 R2.5.
Click on the button and enter X50 Z-30.

Click on the button and enter Z-44 R2.5.

Click on the button and enter X60 FS1.
Click on the button and enter Z-70 and R1.

Click on the button and enter X66 and R1.

Click on the button and enter Z-77 and R1.
Click on the button and enter X82 and FS3.

The raw part is diameter 80 in the program and is run to 82, in order to gain distance from the tool. For that reason, the chamfer size has to be programmed 1mm larger.

The finished part contour has now been described, and is accepted into the program with .

After that, you are back in the program editor.
4.12 Stock Removal Cycle Roughing

The stock removal of a contour is performed in the area Contouring and Stock Removal. Using the “Help Key” brings up the help display.

Click on the button and Select the tool and enter the cutting data. Accept all other data in the picture.

Here, you should test the program with a simulation display.

Clicking on the button “End” takes you back to the program editor.
4.13 Finishing the Outside Contour

The outside contour is executed with the finishing tool contour parallel.

Click on the button and Select the tool and enter the cutting data.
Accept all other data from the figure.

4.14 Thread Undercut

We incorporate the undercut DIN 76

Click on the button and then on

Use all data from the figure and exit the menu with
We can see here that the contour during roughing and finishing is connected with stock removal. The thread undercut was generated with its own cycle.

Here, you can test your program with the simulation display.

4.15 Thread Cutting

A thread of M30 X 1.5 is to be manufactured.

Click on Drehen and on Gewinde and enter the values.
4.16 Making Grooves

Finally, we are making two grooves.

Click on **Drehen** and on **Einstich** and enter the values.

Exit menu with **Übernahme**.
We are done with our machining step program.

4.17 Simulation

Finally, test your program with a simulation display.
First in the side view.
Next in the 3 window view

and in the volume model.

Using **EXIT**, we exit the ShopTurn software.
5 Programming a Milled Part in G-Code
FORMATTED G_CODE

N1 G0 G90 G40 G17 \\
N2 T="PLANFRAESER63" \\
N3 M6 \\
N4 G94 S400 F300 M3 M8 \\
N5 G54 G0 X-20 Y-135 Z3 \\
N6 CYCLE71(3,2,2,0,-50,-100,100,150,0,1,8,55,2,0.2,300,22,) \\
N7 G0 Z150 \\
N8 G0 X200 Y200 \\
N9 T="FRAESER32" \\
N10 M6 \\
N11 G94 S1000 M3 M6 F2000 \\
N12 G0 X-51 Y-112 Z10 \\
N13 G0 Z-10 \\
N14 G17 G90 DIAMOF ;*GP* \\
N15 G1 X-35 Y-100 G41 ;*GP* \\
N16 V35 RND=15 ;*GP* \\
N17 X35 RND=15 ;*GP* \\
N18 V-120 ;*GP* \\
N19 G40 10 \\
N20 G0 G92 Z100 \\
N21 X200 Y150 \\
N22 T="FRAESER10" \\
N23 M6 M6 \\
N24 G94 S2000 M3 M6 F2000 \\
N25 G0 X0 Y-84 Z10 \\
N26 G1 Z-15 \\
N27 G17 G90 G41 DIAMOF ;*GP* \\
N28 G1 X0 Y-90 ;*GP* \\
N29 X30 RND=5 ;*GP* \\
N30 Y-19.9 RND=5 ;*GP* \\
N31 G2 X-30 I=OC(0) J=OC(0) RND=5 ;*GP* \\
N32 G1 Y-90 RND=5 ;*GP* \\
N33 X0 ;*GP* \\
N34 G40 \\
N35 G0 Y-94 \\
N36 G0 Z10 \\
N37 _ZSD[2]=1 ;*RO* \\
N38 POCKET3(0,1,-15,60,54,5,-30,-90,0,5,0,2,0.2,200,100,0,31,6,12,)
N39 POCKET3(2,0,1,-15,60,54,5,-30,-90,0,5,0,2,0.2,200,100,0,32,6,12,)
N40 G0 Z100 \\
N41 G0 X200 Y150 \\
N42 T="FRAESER50" \\
N43 G94 S1000 M3 M6 F2000 \\
N44 POCKET4(2,0,1,-10,30,0,5,0,2,0.2,200,100,0,21,10,12,5,)
N45 POCKET4(2,-10,1,-10,30,0,5,0,2,0.2,200,100,0,21,10,12,5,)
N46 POCKET4(2,-10,1,-21,15,0,5,0,2,0.2,200,100,0,22,10,12,5,)
N47 G0 Z100 \\
N48 G0 X200 Y150 \\
N49 T="ZENTRIERER12" \\
N50 G94 S2000 M3 M6 F2000 \\
N51 G0 Z2 \\
N52 M00 MCALL CYCLE82(2,0,2,-15.3,0.1)}
N104 HOLES1(-42.5,-92.5,90,0,45,4)
N106 G0 X0 Y42.5
N109 HOLES1(42.5,42.5,-90,0,45,4)
N111 MCALL
N114 G0 Z1
N116 MCALL CYCLE82(2,0,2,-15,3,0,0.1)
N122 HOLES2(0,0,22.5,0,60,6)
N124 MCALL
N125 G0 Z100
N126 G0 X200 Y150
N127 T="BOHRER10"
N128 G94 S2000 M3 M8 F200
N131 MCALL CYCLE83(2,0,1,-24,,-15,5,3,1,3,10,1)
N135 HOLES1(-42.5,-92.5,90,0,45,4)
N137 G0 X0 Y42.5
N140 HOLES1(42.5,42.5,-90,0,45,4)
N142 MCALL
N143 G0 Z2
N146 MCALL CYCLE83(2,0,1,-24,,-15,5,3,1,3,10,1)
N150 HOLES2(0,0,22.5,0,60,6)
N152 MCALL
N153 G0 Z100
N154 G0 X200 Y150
N155 M30

PLANFRAESER = facing tool
FRAESER = milling tool
ZENTRIERER = centering tool
6 Programming a Milled Part according to the Machining Step Process
6.1 Start SINUTRAIN

To start SinuTrain, double click on Start SinuTrain or start SinuTrain using the start bar.

6.2 Selecting the Machine

After you booted the computer, a menu is displayed with the selection of machines with the respective version number.

1. Select milling machine >> ShopMill Open V 06.04 <<.
2. Click on the Start button.

6.3 SINUTRAIN Basic Display

After you selected the machine, the basic display of SINUTRAIN appears. All SINUTRAIN actions are started from here.
6.4 Setting Up a New Directory/Program Generation

Different actions can be performed in the program manager.

By clicking on the softkeys the available directories are opened under ShopMill, in the operating area DIRECTORY

With the softkey we are now setting up a new directory called "Pattern plate".

Confirm the input with OK.
The selected directory opens by pressing the arrow key on the CNC keyboard, or with the cursor key toward the right.

### 6.5 Setting Up a New Workpiece Program

After opening the directory, existing programs are displayed, or new programs can be set up.

By pressing the softkey and selecting ShopMill or G-Code (vertical softkeys), a new program name “Name of the work plan” is entered; in our case “Pattern plate”.

Confirm the input with OK.
6.6 Program Header

The program header of the new program opens automatically. The following is entered in the program header: the workpiece data, zero shift, raw part pattern, distance to the chuck, retraction planes, tool change point, safety clearance incremental or absolute, and the spindle speed limitation.

With the softkey you can now call the associated help display.

Enter all data as in the figure above

and accept the inputs in the machining plan with .

**Note**

With the softkey you can alternate, for example, between different zero points and measurement units, mm, inch. The blue line above always shows the current input type.
6.7 Outside Contour

The contour is generated with the contour calculator.

Click on the button below and then in the upper right on . Assign a name to the new contour; after that, the contour calculator opens.

The first input specifies the starting point.

Complete programming the outside contour with the contour calculator.

The outside contour is now described, and is accepted into the program with .
6.8 Roughing the Outside Contour

To perform the machining step roughing on the outside, with contour compensation on the left, do the following:

Click on Behnfräsen and on Werkzeuge and enter the values.

6.9 Selecting the Tool

Here the tools that have already been set up are provided. But you can also generate new ones. Here, take the MillingTool32.

Click on ins Programm to accept the selected tool.
6.10 Finishing the Outside Contour

To perform the machining step Finishing Outside, with contour compensation to the left, do the following:

Click on \texttt{Bahnfräsen} and enter the values.

Accept all values from the figure, and exit the menu with \texttt{Übernahme}.
Here, you can test your program with the simulation display.

6.11 Stock Removal, Residual Material and Finishing of Contour Pockets and Inside Contour

The contour is generated with the contour calculator.

Below, click on the button \( \text{Kontur} \) \( \text{fräsen} \), and then in the upper right on \( \text{Neue Kontur} \). Assign a name to the new contour; then the contour calculator opens.

<table>
<thead>
<tr>
<th>Neue Kontur</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bitte geben Sie den neuen Namen ein:</td>
</tr>
<tr>
<td>FORMPLATTE_Innen</td>
</tr>
</tbody>
</table>
The first input specifies the starting point.

With the contour calculator, complete programming the inside contour.

The inside contour is now described and is accepted into the program.
6.12 Stock Removal of Contour Pocket Roughing

Click on and on and enter the values.

Select the tool and enter the cutting data.

Accept all additional data from the figure.
6.13 Removing Residual Material from the Corners

Click on \textit{Kontur fräsen} and on \textit{Tasche Restmat.} and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the figure.
6.14 Finishing the Base

Click on **Konturfräsen** and on **Taschefräsen** and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.
6.15 Finishing the Edge

Click on Konturfräsen and on Tasche fräsen and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.
6.16 Roughing the Circular Pocket Diameter 60

Click on \texttt{Fräsen} and on \texttt{Kreistasche} and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.

6.17 Finishing the Circular Pocket Diameter 60

Click on \texttt{Fräsen} and on \texttt{Kreistasche} and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.
6.18 Roughing the Circular Pocket Diameter 30

Rough the circular pocket yourself.
Important: The starting depth is no longer at “0” but at “-10mm”

6.19 Finishing the Circular Pocket Diameter 30

Rough the circular pocket yourself.
Important: The starting depth is no longer at “0” but at “-10mm”

6.20 Centering and Drilling taking Obstructions into Account

For centering and drilling, first the corresponding machining steps are programmed, and then the individual positions are specified.
The obstructions have to be taken into account when positioning.

6.21 Machining Step Centering

Click on and on and enter the values.
Select the tool and enter the cutting data.
Accept all additional data from the picture.
6.22 Machining Step Drilling

Click on **Bohren** and on **Tiefbohren** and enter the values.
Select the correct tool for drilling, enter the cutting data
and exit the menu with **Übernahme**.

6.23 Inputting the Line of Holes to the Left

Click on **Bohren** and **Positionen** and enter the values for the
“left” side.

6.24 First Obstruction

Click on **Hindernis** and enter the Z value.
6.25 Inputting the Line of Holes to the Right

Click on **Bohren** Positio- nen and enter the values for the “right” side.

6.26 Second Obstruction

Click on **Hindernis** and enter the Z value.

6.27 Drilling the Full Circle

Click on **Bohren** Positio- nen and enter the values for the right hole circle.

6.28 Third Obstruction

Click on **Hindernis** and enter the Z value.

6.29 Drilling a Single Hole

Click on **Bohren** Positio- nen and enter the values.
6.30 Simulation

Finally, test your program by using the different simulation displays.
7 Additional Tasks for Turning

7.1 Step Shaft

![Diagram of Step Shaft with dimensions and tolerances]

<table>
<thead>
<tr>
<th>(Verwendungsbereich)</th>
<th>(Zul. Abw.)</th>
<th>(Oberfl.)</th>
<th>Maßstab: 1:1</th>
<th>(Gewicht)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Datum</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.10.01</td>
<td>KHE</td>
</tr>
</tbody>
</table>

STUFENWELLE

<table>
<thead>
<tr>
<th>Zust.</th>
<th>Änderung</th>
<th>Datum</th>
<th>Name</th>
<th>Ursprung</th>
<th>Ersatz für:</th>
<th>Ersatz durch:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### 7.2 Shaft02

<table>
<thead>
<tr>
<th>(Verwendungsbereich)</th>
<th>(Zul. Abw.)</th>
<th>(Oberfl.)</th>
<th>Maßstab: 1:1</th>
<th>(Gewicht)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Datum</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.10.01</td>
<td>KHE</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Werkstoff, Halbzeug</th>
<th>Modell- oder Gesenk-Nr</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>(Rohteil-Nr)</th>
<th>Blätter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Blatt</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>X</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ø90</th>
<th>Ø60</th>
<th>Ø34</th>
<th>Ø60</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>35</th>
<th>80</th>
<th>100</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>75</th>
<th>35</th>
<th>3x45°</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>All non-dimensioned radii</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ø90</th>
<th>Ø60</th>
<th>Ø34</th>
<th>Ø60</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>35</th>
<th>80</th>
<th>100</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>75</th>
<th>35</th>
<th>3x45°</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>All non-dimensioned radii</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ø60</th>
<th>Ø34</th>
<th>Ø60</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>35</th>
<th>80</th>
<th>100</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>75</th>
<th>35</th>
<th>3x45°</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>All non-dimensioned radii</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ø60</th>
<th>Ø34</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>35</th>
<th>80</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>75</th>
<th>35</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>All non-dimensioned radii</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Ø60</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>80</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>75</th>
</tr>
</thead>
</table>
8 Additional Tasks Milling

8.1 Flange

![Flange Diagram]

(Flange Diagram with dimensions and specifications)

<table>
<thead>
<tr>
<th>(Verwendungsbeziehung)</th>
<th>(Zul. Abw.)</th>
<th>(Oberfl.)</th>
<th>Maßstab: 1:1</th>
<th>Gewicht</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Werkstoff, Halbzug)</td>
<td>(Teil-Nr)</td>
<td>(Modell- oder Gesenk-Nr)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(Flange Specifications Table)

<table>
<thead>
<tr>
<th>Datum</th>
<th>Name</th>
<th>Bearb.</th>
<th>Gepr.</th>
<th>Norm</th>
<th>Blatt</th>
<th>Blätter</th>
</tr>
</thead>
<tbody>
<tr>
<td>06.03.02 PURR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

(Zust. | Änderung | Datum | Name | Ursprung | Ersatz für | Ersatz durch |)

(Fundamentals of CNC Programming with SINUTRAIN)
8.2 Exercise 11

Schnitt A-A

UBUNG 11

<table>
<thead>
<tr>
<th>(Verwendungsbereich)</th>
<th>(Zul. Abw.)</th>
<th>(Oberfl.)</th>
<th>Maßstab 1:1</th>
<th>(Gewicht)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Datum</td>
<td>Name</td>
<td>Bearb.06.03.02 PURR</td>
<td>Gepr. Norm</td>
<td></td>
</tr>
</tbody>
</table>

Zust. Änderung Datum Name Ursprung Ersatz für: Ersatz durch: