

Sheet (4)
CNC M/C Tools
Part program – Milling Machine

Assume any missing data

Construct the part program for machining the part shown in figs below. The CNC milling machine has the following format:-

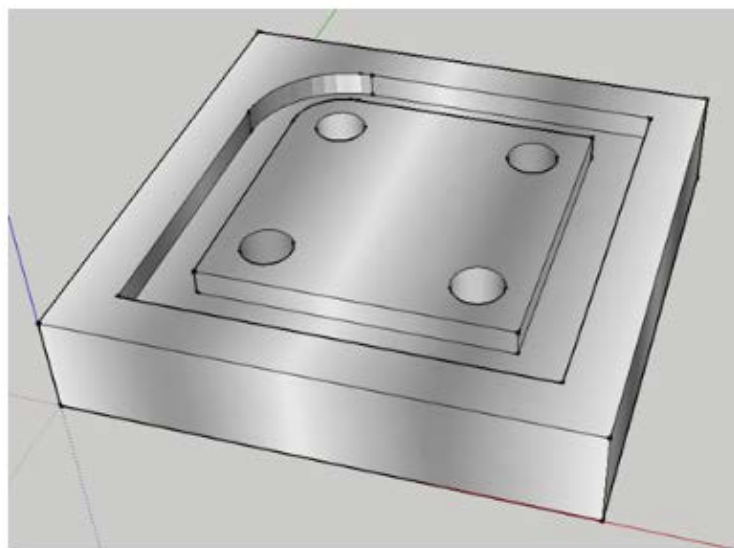
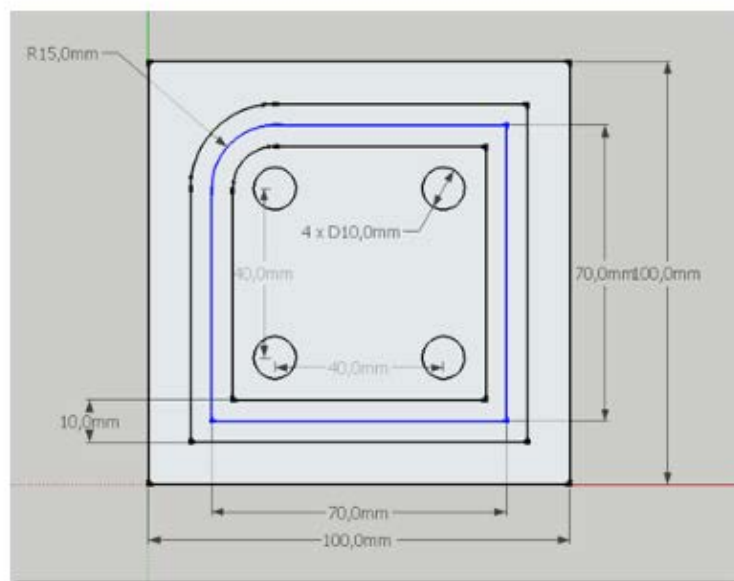
N4 G2.1 XY Z±3.3 IJK±3.3 F4 S4 T4 M2

- Suggest the required tooling for machining the part in a form of table and the proper machining values.

- Assume that the machine support the tool length and diameter compensation

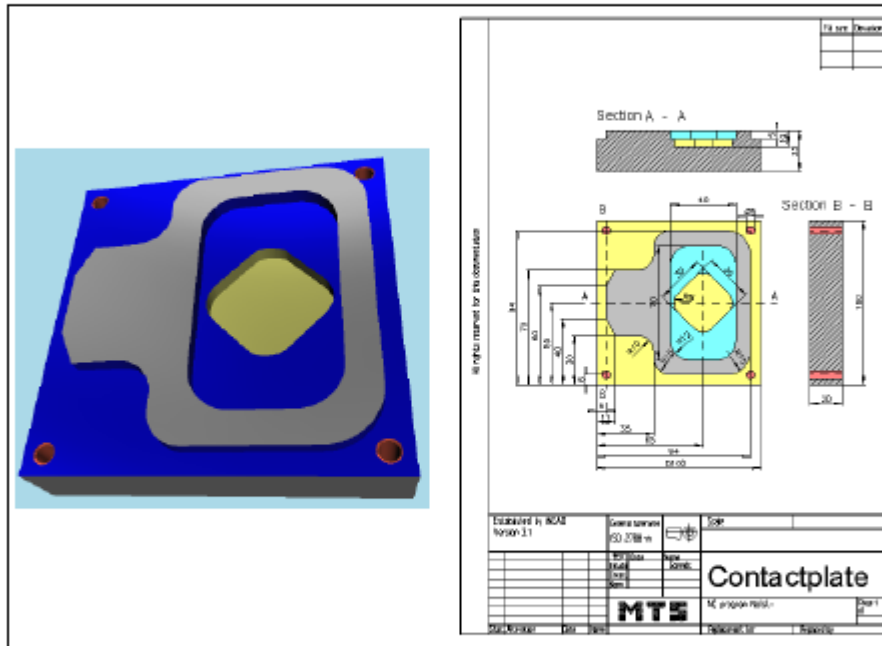
A)

This is the part we are going to make:



B)

The contact plate on the drawing is to be produced on a CNC vertical milling machine from a blank of AlMg1 dimensioned 100 x 100 x 25 mm. Prepare, test and correct the manufacturing process with the MTS CNC Milling Simulator. Define the workpart zero, work out the process layout, set-up form and NC program.

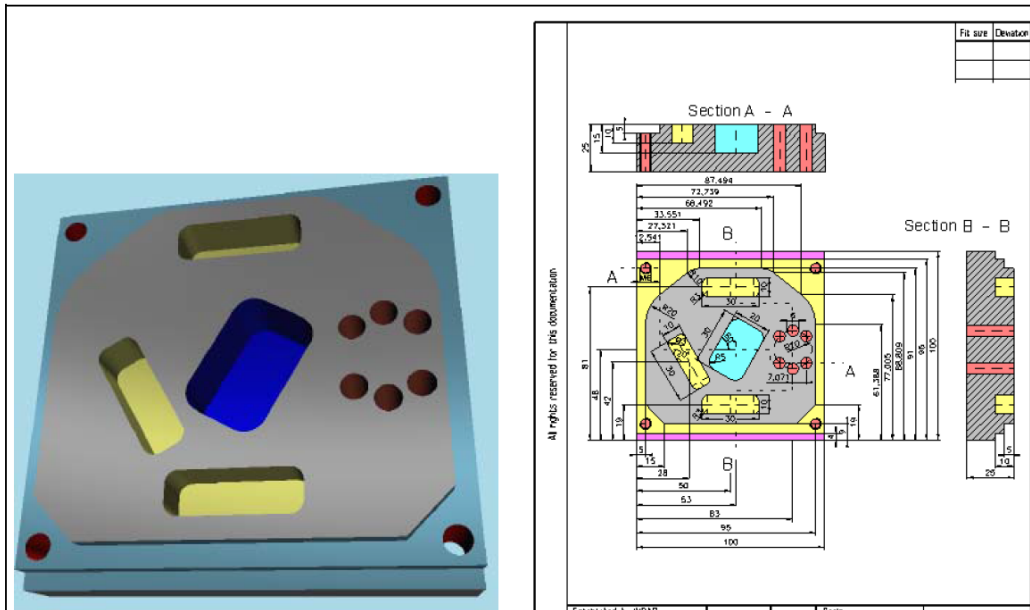


Learning Objective: He is capable of milling a workpart of medium complexity He writes and tests an operable CNC program.

Special Training Elements: Cutter radius compensation, canned cycles. For drilling operation shift the origin and rotate. Use a subprogram with repetition. Cancel rotation and zero shift.

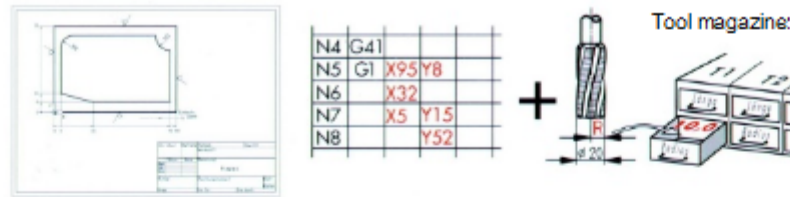
C)

The punch plate on the drawing is to be produced on a CNC vertical milling machine from a blank of AlMg1 dimensioned 100 x 100 x 25 mm. Prepare the production on the MTS CNC Simulator, work out the process layout and set-up form.



4.2.1.3 Cutter radius path compensation (G41 / G42 - G40)

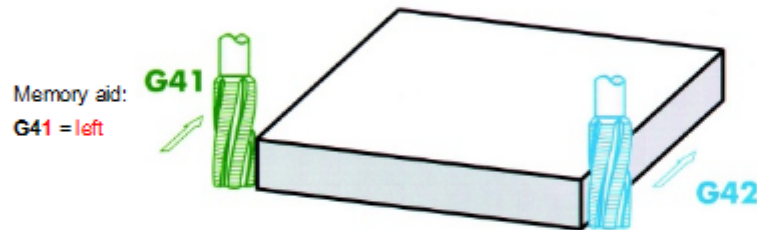
Thanks to the cutter radius path compensation, it is possible to program the **workpiece dimensions**.



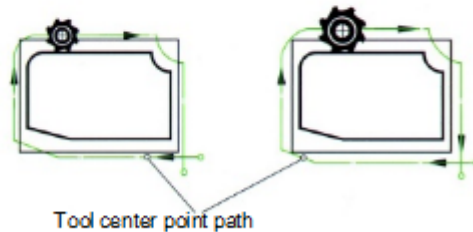
The **workpiece dimensions** are programmed, and the control system will calculate the **cutter center path** using the appropriate **radius value R**.

To enable the control system to calculate the cutter center path from the program data and the tool magazine, it must be told where the tool is to mill.

To this end, 3 G functions are provided:



- G41** = cutter radius path compensation left of the milling contour
- G42** = cutter radius path compensation right of the milling contour
- G40** = cancelation of the cutter radius path compensation



Using the cutter radius path compensation, it is possible to use different tools for the same workpiece contour. The called tool will follow the contour along a tool path offset by the tool radius (=equidistant).

Please note:

- The command (selection) of the G41 or G42 function must always be provided before approaching the contour and is only effective if the tool traverses with feed.
- The command "Cancel G41 or G42 by G40" must always be provided outside the contour.
- When using G40, G41 and G42, the distance of the cutter to the workpiece should be at least 1 x cutter radius.

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise

Format

N... G02/G03 X... Y... Z... I... J... K... F...

or

N... G02/G03 X... Y... Z... R... F...

X, Y, Z ... End point of the arc (abs. or incr.)

I, J, K Incremental circle parameter
(distance from start point to the centre point, I is related to X, J to Y, K to Z)

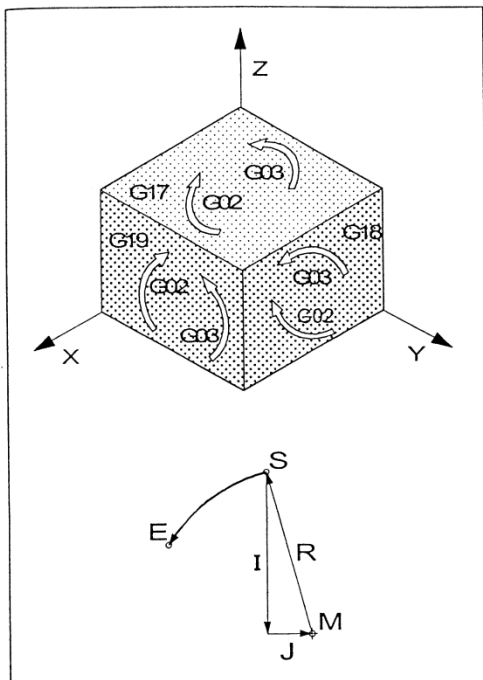
R..... Radius of the arc (arc < semicircle with +R,
> semicircle with -R), can be programmed
instead of the circle parameter I, J, K

The tool will be traversed along the defined arc with
the programmed feed F.

Notes

The circular interpolation can be proceeded in the
active plane only.

Programming the value 0 for I, J or K can be omitted.
The observation of G02, G03 occurs always vertical
to the active plane.



Rotational directions of G02 and G03