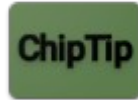# ChipTip

Milling package

Milling package is designed to save your time in workshop when working on milling machine.
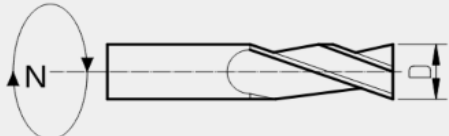
All screens of this package are located inside «Milling» main menu folder. These are:

- Cutting parameters — calculate spindle speed, table feed rate and more;

- Cylindrical groove — calculate how to do cylindrical grooves using ordinary end mills;

- Flange with holes — calculate how to locate holes (or similar features) evenly on some circle. Calculates distance between holes and coordinates of holes centers;

- Single gear dimensions — simple calculation of standard profile straight tooth gear. Find outer diameter and depth of in-between-teeths recesses to know how to machine the gear;

- Dividing — allows to create, change and store sets of settings for your indexing/dividing devices. Divides circle on even parts with direct, indirect, differential methods. Also includes very powerful «Error averaging» method which allows dividing to practically any number without using change gears!

"Cutting parameters" screen

This screen allows to calculate cutting parameters for milling. Its possible to calculate:

- mill diameterD based on spindle speed N and cutting speedV;
- spindle speedN based onD andV;
- cutting speedV based on D andN;
- table feedbased on N, number of mill flutesZ and feed per tooth value.

Parameters D, N andV interconnected with equation:$V = (PI * D * N)/1000$

Knowing any two of them allows you to calculate third one pressing "CALC" button near the unknown parameter. Table feed equation is: *table_feed= feed_on_tooth* * Z * N Once you select

"Mill material" and"Cutting material" the program will put typical values for "V" and «Feed per tooth». These are starting values to find optimal cutting parameter in your particular case. Use it with caution!

Normal screen usage sequence

1. Choose thick and short (thus rigid) end mill cutter which is suitable to do the job.
2. Choose "Mill material" and "Cutting material". Fields V and «Feed per tooth» will be filled with typical values. You can correct it manually if you want.
3. Choose "With cooling" / "Without cooling" and "Finishing" / "Roughing".
4. Press CALC for N — spindle speed will be calculated. Set nearest lower speed on your mill.
5. Press CALC for "table feed". Set nearest lower table feed on your mill.

Unusual usage sequence

Each CALC button requires next input to make its calculations:

- CALC D -> N, V
- CALC N → D, V
- CALC V → D, N
- CALC «table feed» → «feed per tooth», Z, N

Put in to the program all you know and press CALC for unknown value. If enough information is entered then calculation will be done. Otherwise missing information fields will be colored in kind of red color.

"Cylindrical groove" screen

This screen allows to calculate cylindrical grooves in two situations:

1. When you have spherical end mill and you want to calculate depth of the groove with given width or width with given depth.
2. When you have only ordinary cylindrical end mills. In this case you can pivot vertical milling head of the mill and cut elliptical cylinder groove which is very close in shape to round cylinder groove.

Input fields:
- R — groove curvature radius
- L — groove width
- H — groove depth
- D — mill cutter diameter
- angle α — angle between mill cutter and part feed direction.

Linear dimensions units could be any (e.g. mm or inches) but be aware that all information on the screen is in same units.
Last calculated value highlighted with green background color. Incorrect value highlighted with red background color.

Calculations for spherical end mills
If you have spherical end mill with the diameter twice larger then needed R then you can calculate L based on H. Enter R, H and press «CALC» near L.
If you want to calculate H based on L then enter L, set α = 90 deg and enter D. Press «CALC» near H.

Calculations for ordinary cylindrical end mills
Lets suppose that we have blueprint where groove is specified by R and H. We need to find which cutter to use (D) and how to set up milling head (angle α). Lets enter what we already know — R and H. Lets R = 15 and H=3. The mill cutter to the work should be of similar radius. Suppose we've found several mill cutters with diameters 16, 18, 20, 24 and 36. Lets sequentially put mill cutter diameters in to D field and press «CALC» for α.
  • D = 16 — program says that D can't be less then L. But we din't entered L!? The program calculated L and you could also do this by pressing «CALC» for L. L = 18. Its obvious that we can't do the job with mill cutter thinner then groove width.
  • D = 18 — program calculated α = 19 deg 28 min 16 sec
  • D = 20 — program calculated α = 32 deg 7 min 40 min
  • D = 24 — program calculated α = 47 deg 35 min 48 min
  • D = 36 — program can't calculate α, it complains on D.

Now we have several options to do the job. You are free to choose most convenient e.g. 332 deg 7 min 40 min.

If the groove defined by its width and depth then enter these values and then do the same procedure as described above.

If you enter L, H, and then sequentially will change α (e.g. with 10 deg steps) with pressing «CALC» for D – you can find the range of mill cutter diameters suitable for making this particular groove.

Error in inputs
When R and L values could be calculated from each other when H is fixed. So from math point of view R and L are basically one parameter. Therefore if you enter not matching R and L the program will highlight this as an error. In this case you need to either recalculate one of these parameters or clear one input field.
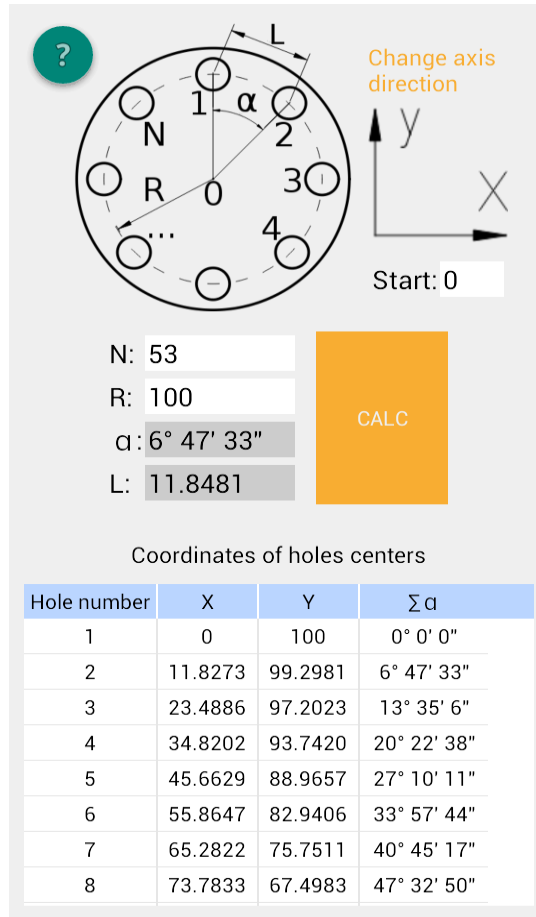Angle α must be in range 0 — 90 deg. Usually angles less then 15 deg. doesn't work.
Mill cutter diameter D must be bigger then groove width L.
The check you can do before finally setting up the milling machine: sequentially press all "CALC" buttons from top to the bottom and vica versa – all values must stay the same and no error messages should come from the program.

"Flange with holes" screen

The screen allows to calculate periodic features located on circle e.g. holes on a flange. It works in Cartesian coordinates system.

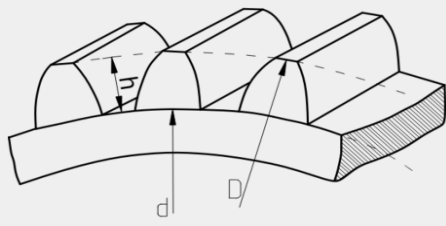The screen helps you to do two types of work:
- **Do manual layout of holes on flange-like parts**. Enter number of holes N and radius R then press "CALC". There will be calculated between centers distance L. Draw the circle of R radius on your part. Then, using for example caliper or divider, draw hole centers with L distance between them.
- **Machining using coordinate table**. If you have milling machine or drill press equipped with coordinate table you can use data from "Coordinates of holes centers" table to machine them. The table consist from: **«Hole number»** - index number of the hole, **X** and **Y** — coordinates of hole center, $\sum\alpha$ – angle on which hole center is. Dimensions of **X** and **Y** are the same as for R, which you've entered. If you assume that R is measured in millimeters then all other linear dimensions on this screen should be also considered to be in millimeters. Enter N and R. Optionally you can choose direction of coordinate system axises by tapping axises image. Also you can change zero point of coordinate system. If you put 0 in to «Start» field then zero point will be at the flange center. If you put 1 in to "Start" field then zero point will be in the center of hole #1. #1 hole is those located strictly under flange center on 12 hours. If you put 2 in to "Start" field then zero point will be in the center of hole #2 and so on.

"Single gear dimensions" screen

This screen allows to calculate main dimensions for making ordinary gear with following properties:

- Tooth profile — Russian standard 16532-70 i.e. involute engagement, profile angle 20 deg., tooth head height ratio = 1, tooth boundary height coefficient = 2, radial clearance ratio = 0.25
- Profile offset ratio = 0
- Straight tooth
- Cylindrical.

Screen shot:

For calculation of gear outer diameter one should enter its Module (M) and teeth number (Z). Than press "CALC" near D. Calculated values are:

- D — outer gear diameter. Machine the blank part on lathe to this size.
- h — in-between-teeth recesses depth.
- d — recesses diameter.
- Mill # - number of the module cutting mill from the standard set of 8.
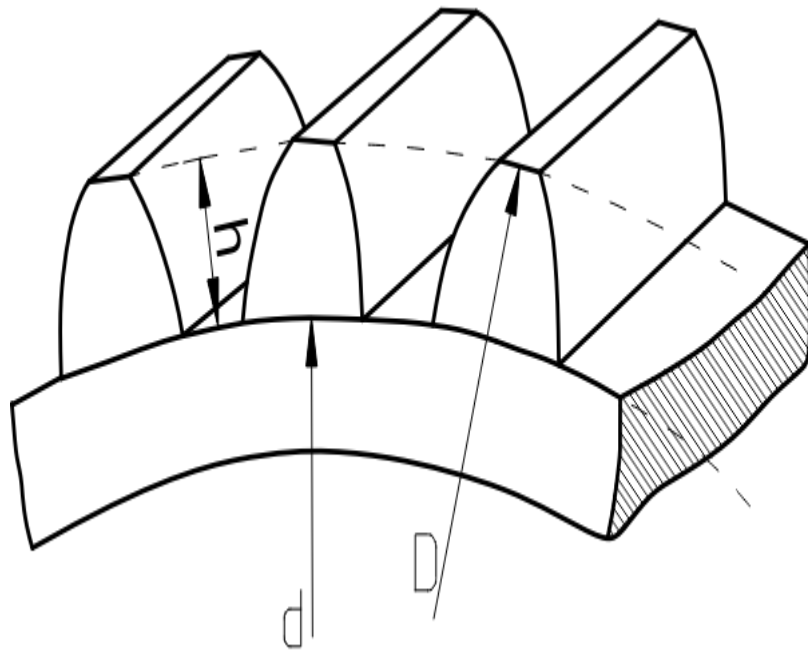
You can calculate any of M, Z and D if you know two other parameters. Just enter what do you know and press ""CALC" for unknown parameter.

"Dividing" screen

Diving screen makes it easier to work with universal dividing heads (UDH) and rotary tables as well as with other indexing fixtures.

Why to use?
- The screen allows direct dividing on any number of circle fractions (Z). It is very convenient for rotary tables which has degrees scale but lack indexing disks.
- It is possible to divide by enormous number of various Z using error averaging method (named after USSR machinist Gutsula). Using just standard indexing disks for your semi universal dividing head it is possible to get all Z numbers usually obtained with differential dividing and even more!
- Automatically choose simplest dividing method based on entered Z.
- Shows simplest options first during differential dividing. Each option contains drawing of the change gears assembly.
- It is possible to calculate dividing options for non standard indexing fixtures with handle-to-spindle gear ration other than 1:40.

UDH settings editor
You can enter up to 50 different sets of settings for your dividing heads and rotary tables.
Press «Tap to edit dividing» at the top of the screen to open the editor.

Use selector at the top editor layout to load existing UDH configurations. Chosen configuration will be loaded to editing and also it will be used for all calculations on "Dividing" screen.
Editor input fields:
- **Name** — name of this settings set;
- **Worm ratio** — handle-to-spindle gear ratio;
- **Available holes numbers on indexing disks** — put number of holes you have on your dividing/indexing plates/disks there;
- **Available change gears** — put numbers of teeth of available change gears there.

After changing any data in the editor screen don't forger to press "Save". To create new settings set press "New". To delete selected set – press "Delete".
The screen provides 5 ways to make dividing. Chose one with "Dividing type" selector.

«Auto» dividing type
When Z = 2, 3, 4, 6, 8, 12, 24 then direct dividing will be applied. When Z is 7 or greater – indirect dividing by means of dividing plates will be performed. If no suitable combinations found then differential dividing will be done.

"Direct" dividing type
This dividing method is suitable for indexing equipment with degrees scale and vernier.
Enter any Z number and press "CALC".

Tap to edit dividing head УДГН - 100	?

Dividing type: Direct ▼

Dividing number (Z): 38 | CALC

#	Degrees	Minutes	Seconds
1	0	0	0
2	9	28	25
3	18	56	51
4	28	25	16
5	37	53	41
6	47	22	6
7	56	50	32
8	66	18	57
9	75	47	22
10	85	15	47
11	94	44	13
12	104	12	38
13	113	41	3
14	123	9	28
15	132	37	54
16	142	6	19

"Indirect" dividing type

This method uses stationary dividing plate mounted on UDH.

Enter any Z number and press "CALC". No every Z could be obtained by this method. Typically with standard dividing plates one can achieve any Z < 51 and after 51 there will be only some Z available.

There could be several combinations shown as a result. You could use any of them. Columns in the resulting table has following meaning:

- **#** - sequential number of combination;
- **Number of holes** — designates particular "circle" on dividing plate. This circle have exactly this number of holes and UDH indexing pin must be adjusted to travel on this circle;
- **Full turns** — number of full turns which UDH crank handle must travel each time;
- **Holes to skip** — number of free holes between UDH sector arms. Or number of between-holes spaces which crank handle must travel each time in addition to number of full turn movements.

If no combinations found for indirect dividing then two below methods definitely will help...

"Differential" dividing type

Differential dividing is based on slight rotation of dividing plate during each crank handle turn. Dividing plate receives its rotation from UDH spindle by means of change gears assembly.

Enter any Z number and press "CALC".

#	Num holes	Full turns	Holes to skip	A	B	C	D	
0	16	0	5	100	Any	-	40	
1	33	0	12	100	55	50	25	
2	33	0	12	100	25	50	55	
3	33	0	12	50	55	100	25	
4	33	0	12	50	25	100	55	
5	16	0	5	100	80	70	35	
6	16	0	5	100	35	70	80	
7	16	0	5	70	80	100	35	
8	16	0	5	70	35	100	80	

First four columns of the resulting table has exactly the same meaning as for already described "indirect" dividing. Columns A — E describes change gears assembly. Column E is not visible on the screenshot but it exist – juts scroll the table to the right. Last column «i» shows change gears reduction ratio. Press any resulting row to see drawing of the change gears assembly for it.

If some cell in A-E columns contain number – it means number gear teeth to put on named (A-E) position in the gears assembly. If there is «Any» in the cell – it means that dummy gear with any number of teeth will do the job. If there is «-» sign – it means that no gear needed in this position.

"Error averaging" dividing type

In this dividing method after machining, say, one between-teeth recess we'll be machining next one recess located at some distance from the first recess. The UDH spindle will do many turns prior the gear (say we are cutting some gear) will be fully machined.

This method works when:
- There is no way to divide by indirect or differential methods;
- There are dividing plates of low accuracy or worm has uneven wear.

The method allows to divide (without change gears – just using standard dividing plates) on enormous number of various Z. As a trade off there will be some error introduced in cut teeth positions when using this method. From the other hand, if you have low accuracy dividing plates worn worm then this method will probably increase the overall accuracy of cut gear because of averaging all errors due to many UDH spindle rotations.

#	Num holes	Full turns	Holes to skip	Error, D100mm	Recesses to skip
0	54	4	16	-0.0895	13
1	54	6	33	-0.0218	20
2	54	8	32	-0.0895	26
3	54	12	48	-0.0895	39
4	54	13	12	-0.0218	40
5	54	15	11	-0.0601	46
6	54	15	29	-0.0031	47
7	54	17	10	-0.0895	52
8	54	17	28	-0.0384	53
9	49	10	12	-0.0931	31
10	49	11	44	-0.0757	36
11	49	13	27	-0.0625	41
12	49	15	10	-0.0523	46
13	47	4	45	-0.0780	15
14	47	9	12	-0.0298	28

There are two additional columns in the results table:
- **Error, D100mm** — shows error in millimeters for the position of last cut recess compared to its theoretical position for dividing diameter 100mm. For example, is the value is -0.0068, it means that last cut recess (and thus one border of the cut teeth) will be located 0.0068mm before it theoretically should. The error has linear dependency on dividing diameter. For 10mm diameter the error from above example will be 0.00068mm and for 200mm diameter — 0.0136 mm.
- **Recesses to skip** — informs you how many recesses will be skipped between two sequential cuts. This value is just for your information, it doesn't affect any UDH settings.

If you try do divide some Z which could be also divided by indirect dividing then error will be 0 and there will be an option in the resulting table with recesses to skip equals 1.