Enrico Bonvicini Leonardo Mantuano Group#11 CAM PROJECT 2023/24

Final Report - CAM Lab

1. Notation and Formula

MILLING

Feed per tooth $[mm/rev.\cdot tooth]$	fz
Cutting diameter[mm]	D
Axial depth of cut[mm]	ap
Radial depth of cut[mm]	ae
Number of teeth	z
Tool main entering angle[degrees]	Kr
Spindle speed[RPM]	$n = \frac{1000*vc}{\pi*D}$
Feed velocity $[mm/min.]$	$V_f = z * f_z * n$
Slab milling – Chip Thickness $[mm]$	$h_{d,\theta} = h_d(\theta) = fz * \sin\theta$
Slab milling – Chip Area $[mm\text{^}2]$	$A_{d,\theta} = h_{d,\theta} * a_p$
Face milling – Chip Thickness $[mm]$	$h_{d,\theta} = fz \cos\theta * \sin kr$
Face milling – Chip Area $[mm\text{^}2]$	$A_{d,\theta} = fz \cos\theta * ap$
Cutting Force $[N]$	$Fc(\theta) = kc * Ad$
Cutting Torque $[Nm]$	$Tc = \sum_{i=1}^{z} \frac{F_c(\theta i)*D/2}{1000}$
Number of working teeth	$z = \frac{\varphi}{\varphi_0}$; $\varphi_0 = \frac{2\pi}{z}$
Cutting Power[W]	$Pc = Tc * \omega$; $\omega = \frac{1000*vc}{60*D/2}$
Axial(face milling) or radial(slab milling) rake angle[radians]	Υ_0
Engagement angle[degrees]	φ
Specific cutting force[N/mm^2]	$K_c = K_{c1} * h_d^{-mc} * (1 - \Upsilon_0/100)$
Average chip thickness[mm]	$h_{d,\theta} = \frac{f_z*2*a_e}{\varphi*D}$
Average Roughness[μm]	$Ra = \frac{f_z*10\text{^}3}{4*(\tan^{-1}k_r' + \tan^{-1}k_r)}$

HELICAL RAMPING

Ramp Leading angle[degrees]	α
Corner radius[mm]	RE
Parallel land of the insert[mm]	bs
Max machined hole diameter[mm]	Dmax=D*2
Min machined hole diameter[mm]	Dmin=(D-(RE-bs))*2
Depth of cut per revolution[mm]	$P = \pi * D * \tan\alpha$
Cutting Power[kW]	$Pc = \frac{a_e*a_p*v_f*k_c}{60*10\text{^}6}$

DRILLING

Point angle[degrees] ε
Helix angle or axial rake angle[radians] Υ_0

Chip thickness[mm] $h_d = f_z * \sin(\frac{\varepsilon}{2})$
Chip area[mm^2] $A_d = f * D/4$
Specific cutting force[N/mm^2] $K_c = K_{c1} * h_d^{-mc} * (1 - \Upsilon_0/100)$

Cutting force[N] $Fc = Kc * Ad$
Cutting torque[N*m] $Tc = \frac{F_c * D/2}{1000}$
Cutting Power[W] $Pc = Tc * \omega$

TAPPING

Radial rake angle GAMF
Helix angle or axial rake angle Υ_0
Chip thickness[mm] $h_d = pitch * \sin(GAMF)$

Specific cutting force[N/mm^2] $K_c = K_{c1} * h_d^{-mc} * (1 - \Upsilon_0/100)$

Cutting torque[N*m] $T_c = \frac{pitch^2 * D * k_c}{8000}$
Cutting power[W] $Pc = \frac{T_c * 2 * \pi * n}{60}$
Feed rate[mm/min] $v_f = Pc * n$

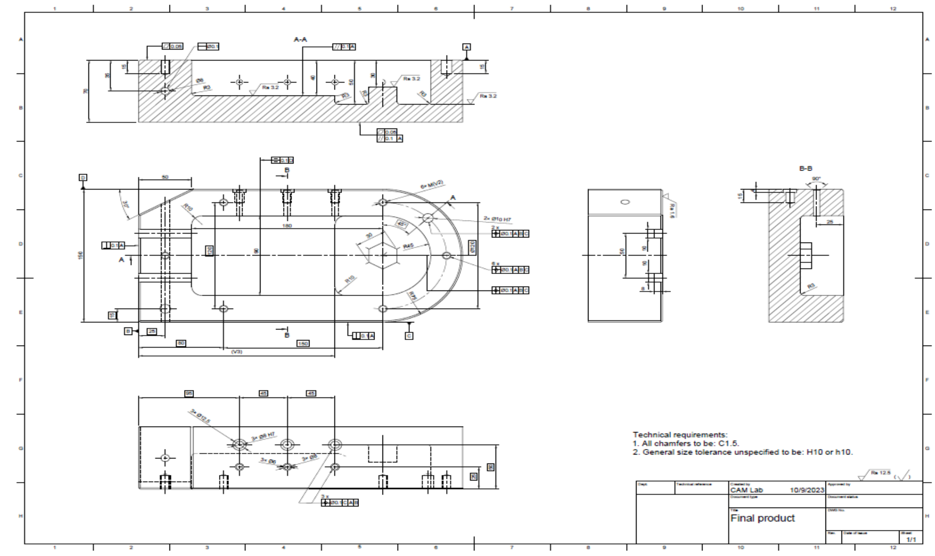
REAMING/COUNTERBORING/COUNTERSINKING

Chip area[mm^2] $A_d = f_z * \frac{\frac{\frac{Dext}{2} + \frac{Dint}{2}}{2}}{2}$
Cutting Force[N] $F_c = K_c * A_d$

Cutting Torque[N*m] $A_d = Z * F_c * \frac{\frac{\frac{Dext}{2} + \frac{Dint}{2}}{2}}{1000}$
Cutting Power[W] $T_c = T_c * \omega$

2. Introduction of Project Product

2.1 Inputs and Your Variables



Each group was provided by a 2D sketch of the part that we want to produce starting from a stock. Some parameters of the sketch and the size of the stock were different for each group, influencing the final choices we've taken.

-Threaded holes : according to the metric ISO our specification for these holes was M7x1.0

-A design parameter, the distance between the flat short side of the workpiece and the midline of the main pocket, in our case equal to 154mm.

-Stock size: x=311mm ; y=156mm ; z=76mm

Then, groups were also differentiated according to some specifications about the workpiece materials, which will affect mainly the CAM part of the project. In our case:

-Workpiece material P1.1.Z.AN where:

P->steel

1->Material group is unalloyed steel

1->Material sub-group for carbon content <=0.25% C

Z-> Manufacturing process : forged/rolled/cold down

AN->heat treatment, annealed, supplied by hardness parameter 125HB

-Specific cutting pressure Kc1=1500 N/mm^2
-mc=0.25 is a parameter that we'll use in the Kronemberg relationship

2.2 Features and Operations

Feature	*Requirement*	*Relation*	*#operaiton*	*operation*
Top Surface	*Ra 1.6*	*Datum A* *⏥0.05*	*1* *2*	*Rough Milling* *Finish Milling*
Long lateral. surface		*Datum C* *⊥0.1A*	*3a* *3b*	*Rough Milling* *Rough Milling*
Short straight side		*Datum B* *⊥0.1A*	*4*	*Rough Milling*
Bottom Surface		*//0.1A* *⏥0.05*	*5*	*Rough Milling*
Thread holes	*M7*	*⌖Ø0.1 ABC*	*6* *7*	*Drilling* *Threading*
Top Blind holes	*H7*	*⌖Ø0.1 ABC*	*8* *9*	*Drilling* *Reaming*
Blind holes with Counterbore	*H7*		*10* *11*	*Drilling* *Reaming*
Counterbore	*H10*	*⌖Ø0.1 CAB*	*12*	*Drilling*
Through holes with countersink	*H10*		*13*	*Drilling*
countersink		*⌖Ø0.1 CAB*	*14*	*Drilling*
Deep through hole	*— -- Ø0.1*		*15* *16*	*Pilot drilling* *Deep drilling*
Pocket A	*Ra 3.2*	*⌖Ø0.1 ABC*	*17* *18*	*Rough milling* *Finish milling*
Pocket B	*Ra 3.2*	*//0.1A*	*17* *18*	*Rough milling* *Finish milling*
Pocket C	*Ra 3.2*		*17* *18*	*Rough milling* *Finish milling*
Slot			*19*	*Rough milling*
Chamfer A(45)	*C1.5*		*20a* *20b*	*Rough milling* *Rough milling*
Chamfer B(45)	*C1.5*		*21*	*Rough milling*
Chamfer C(30)	*C1.5*		*22*	*Rough milling*

Chamfer D(75)	*C1.5*		*23*	*Rough milling*
External Cylindrical surface			*24*	*Rough milling*
Short inclined straight side			*25*	*Rough milling*
Fillet			*26*	*Finish milling*

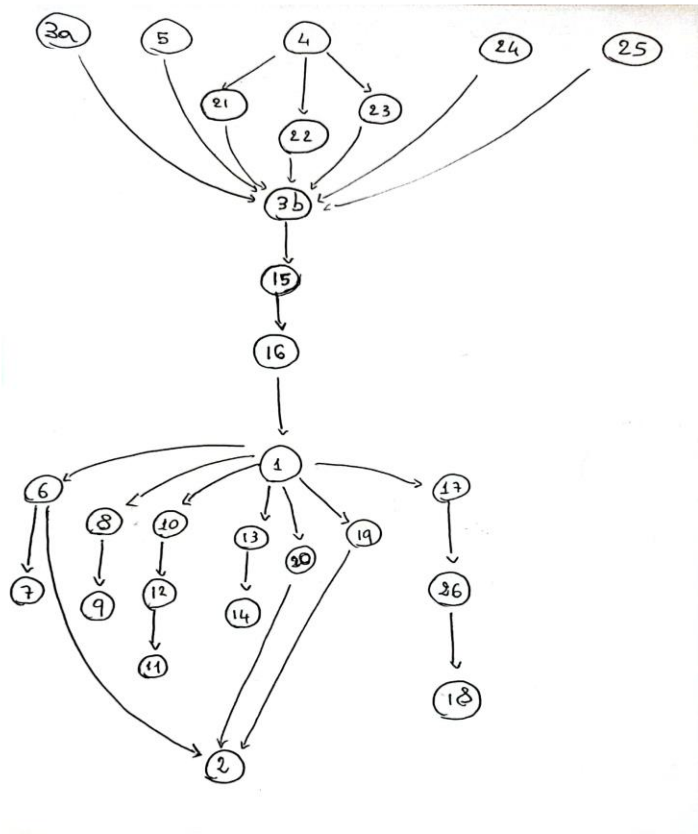
2.3 Precedence Graph

We decided the order for the operations according to some rules:

-First surfaces and then their pocket/holes

-First roughing and then finishing

-The operations must follow the order determined by setups to guarantee technical requirements



3. Detailed Final Solution to Realize Your Product

3.1 Throughput of the Product

Fusion 360 already provides us the machining time required to perform the whole sequence of operations regarding one setup already considering the tool changing time, therefore to compute the throughput we need to consider for each setup also the time required to reposition the axis of the spindle and the time to load and unload the part.

Setup 1:

Ts1 = tm1 + spindle repositioning + load + unload = 13’42’’ + 5’’ + 60’’ + 60’’= 947s

Setup2:

Ts2 = tm1 + spindle repositioning + load + unload = 3’43’’ + 5’’ + 60’’ + 60’’=348s

Setup3:

Ts3 = tm1 + spindle repositioning + load + unload = 23’09’’ + 2*5’’ + 60’’ + 60’’= 1519s

Ttot = Ts1 + Ts2 + Ts3= 2814s

Throughput = $\frac{3600}{2814}$ = 1.279 parts/hour

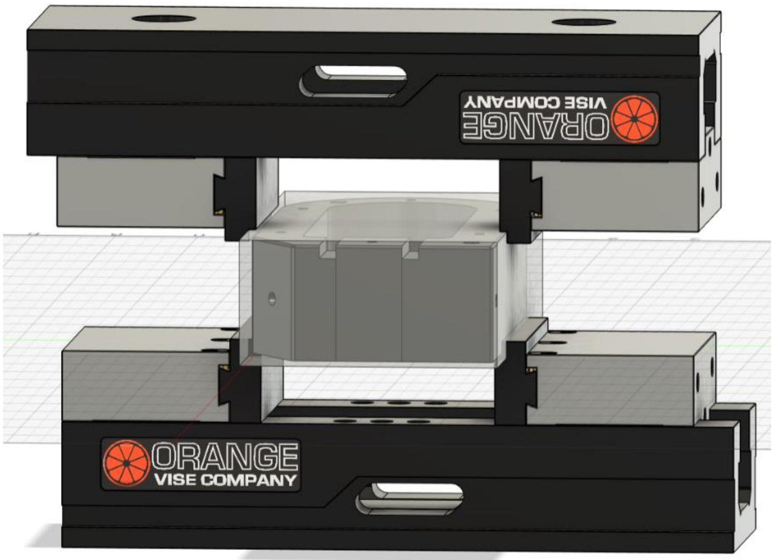
3.2 Detailed Information for Your Setups

The choices taken for the setups were made considering the constraints provided.

First, the fact that the spindle can change its main axis in one additional direction beside the z axis(perpendicular to the fixturing), which could be $\pm y \; or \; \pm x$.

Then also the technical requirements were relevant to decide the setups configuration.

We ended up with just two fixturing, one clamping the stock from the top(first clamp over the stock) and one from the bottom(now clamping directly the part), both clamping form the lateral straight sides of the part.



Setup1: We decided to start machining the bottom surface because it hasn't roughness requirements specified, therefore we can clamp it directly in the following setups without the worry to ruin the surface finish, using it as reference to create the datum reference system, preserving the parallelism and perpendicularity constraints.

Therefore, we can start machining the features that will compose our datum reference system and those which don't have technical requirements associated to the datum system.

Thus, in this setup we machine the bottom surface, half of the lateral straight surfaces, the external cylindrical surface, the inclined short straight side and also the datum B since we can change tha main direction of the axis of the spindle once per setup. Then we can also start creating chamfers for that surface.

Setup 2: We directly clamp the part from the bottom since we have removed the stock and never touch again the upper surface with the fixture since it's the one with most restrictive roughness requirements.

So, we can finish the roughing of the lateral surfaces and also create the deep though hole to use all the possibilities of the setup changing its axis.

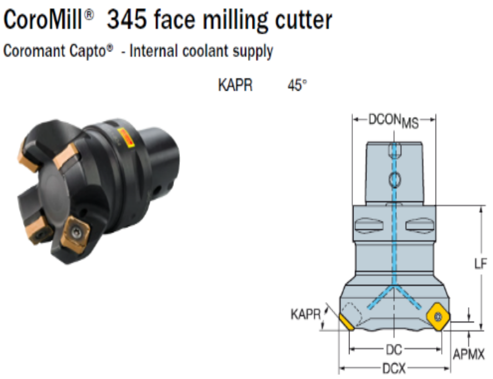
Setup3: The first operation to perform from this setup is the roughing of the top surface, so that we have our datum reference system (notice that we left a stock of 0.3mm to perform as last the finishing of this surface to preserve it).

Then we are able to machine all the remaining features such as holes, pockets, chamfers and so on since we have created our reference, and we can guarantee the technical requirements.

Lastly, we finish the pocket and the top surface to achieve the roughness required.

3.3 Manufacturing Resources

1 - CoroMill 345



Order number: 345-063C5-13M → medium milling conditions

DC=63mm

APMX=6mm

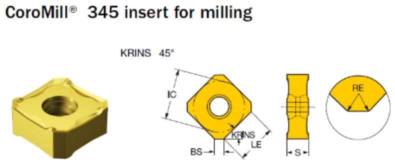
DCX=77.1mm (maximum cutting diameter)

Z=5

DCONms=50mm (diameter at connection, machine side)

LF=60mm (functional length)

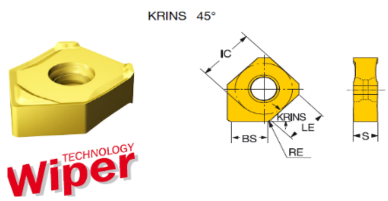
Kr=45°; γ_0=19 rad;



Insert for roughing operations, ordering code: 345R-1305M-PM

First choice for steel: P4330

IC=13;   LE=8.8; S=5.60;   BS=2;   RE=0.8 [mm]

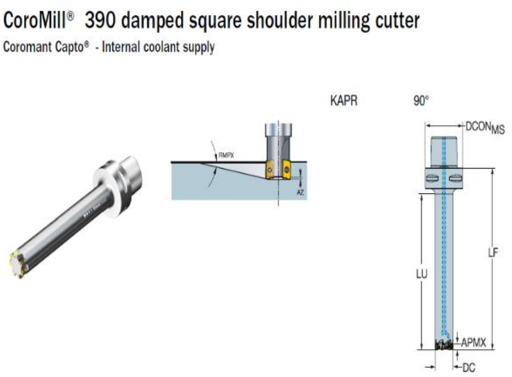


Insert for finishing operations, ordering code: 345R-1305M-PM

Our choice for steel: P4330

IC=13;   LE=8.8; S=5.60;   BS=8;   RE=1   [mm]

2-CoroMill 390



Order number: R390-020C5D-11L145 → heavy milling

DC=20mm

APMXffw= 10mm (max radial depth of cut)

APMXefw= 5.5mm (max axial depth of cut)

Z=2

DCONms=20mm (diameter at connection, machine side)

LF=145mm (functional length)

LU=120mm (usable length)

Kr=90°

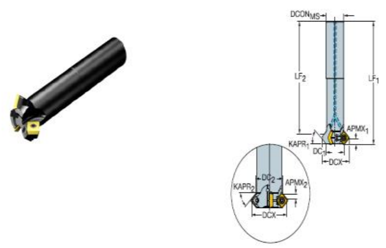
RMPX=2°; γ_0 = 10.27°;

For rough milling
ordering code : R390-11T310M-PH
Choice for steel : P4340
W1=6.8; LE=10; S=3.59; BS=1; RE=1 [mm]
For finishing (pocket)
Ordering code: R390-180616H-PTW
First choice for steel: P1130
W1=11 ; LE=15.4 ; S=6.33 ; BS=8.6 [mm]

Wiper TECHNOLOGY

3-CoroMill 495

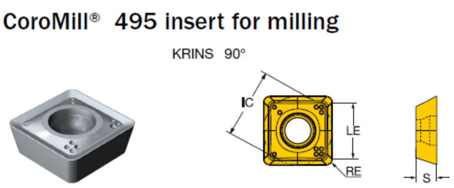
Cylindrical shank - Internal coolant supply



KAPR1	KAPR2		CZC_MS	APMX1	APMX2	CNSC		Ordering code	DCON_MS	DC1	DC2	DCX	BD	LF1	LF2	LU	BAR	NM	KG	RPMX	CICT	MID
									Dimensions, mm													
30°	60°	09	16	3.8	6.5	1	1	495-012A16-3009L	16.0	12.0	18.3	26.0	17.2	100.0	88.7		20	1.4	0.23	14400	1	495-09T3M-XL
45°	45°	09	16	5.4	5.4	1	1	495-012A16-4509L	16.0	12.0	17.7	23.4	11.2	100.0	90.8	51.0	20	1.4	0.23	14400	1	495-09T3M-XL
60°	30°	09	16	6.8	3.9	1	1	495-012A16-6009L	16.0	12.0	13.5	20.1	13.5	100.0	90.3	49.0	20	1.4	0.20	14400	1	495-09T3M-XL
75°		09	16	7.7		1	1	495-012A16-7509L	16.0	12.0		16.2	13.0				20	1.4	0.20	14400	1	495-09T3M-XL

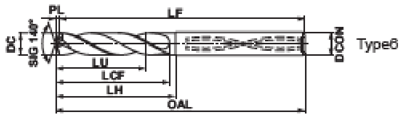
These fthree tools were all introduced to perform chamfer milling along edges to adapt to the desired slope of the chamfer according to the model of the part.

All the tools can handle the same type of insert, which will determine the feed velocity we are recommended to use.
3.1→30° Kr
3.2→45°
3.3→75°



Ordering code : 495-09T3M-PM
Choice for steel : P1130
IC=9; LE=7.4 ; S=3.51 [mm] $\gamma_0 = 20°$

4-**MVS solid carbide twist drill**



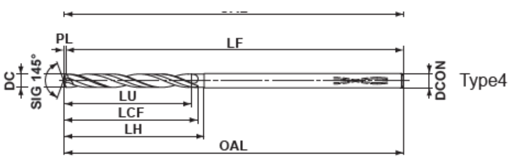
(1) Ordering number: MVS0780X05S080
L/D=5; internal coolant
DC=7.8; LU=40.4; LCF=65.4; LH=65.4; OAL=119.4; LF=118; PL=1.4; DCON=8; [mm]

(2) Ordering number: MVS0600X08S060
L/D=8; internal coolant
DC=6; LU=49.1; LCF=67.1; LH=67.1; OAL=119.1; LF=118; PL=1.1; DCON=6; [mm]

(3) Ordering number: MVS0980X05S100
L/D=5; internal coolant
DC=9.8; LU=50.8; LCF=81.8; LH=81.8; OAL=137.8; LF=136; PL=1.8; DCON=10; [mm]

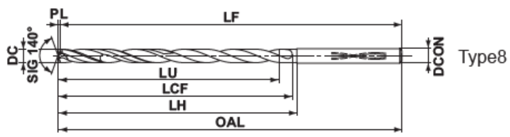
(4) Ordering Number: MVS0600X05S060
L/D=5; internal coolant; thread size M7x1.0
DC=6; LU=49.1; LCF=49.1; LH=101.1; OAL=101.1; LF=100; PL=1.1; DCON=6; [mm]

(5)



Ordering Number: MVS0800X02S080PL (for pilot hole)
L/D=2; internal coolant;
DC=8; LU=17.3; LCF=38.3; LH=38.3; OAL=80.3; LF=79; PL=1.3; DCON=8; [mm]

(6)



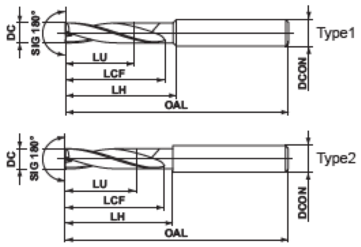
Ordering number: MVS0800X20S080 (for deep through hole)
L/D=20; internal coolant;
Z=2;

DC=8; LU=161.5; LCF=185.5; LH=188.5; OAL=242.5; LF=241; PL=1.5; DCON=8; [mm]

5-MFE flat bottom drill (counterboring)

P M K N S H

External Coolant



(inch)

	.118≤DC≤.236	.236 < DC≤.394	.394 < DC≤.709	.709 < DC≤.787
	0 / −.00047	0 / −.00059	0 / −.00071	0 / −.00083
	DCON=.236	.276, .315, .394	.472, .551, .630, .709	.787
h6	0 / −.00031	0 / −.00035	0 / −.00043	0 / −.00051

Ordering number: MFE1250X02S140

L/D=2; TYPE 2; Z=2;

DC=8; LU=25; LCF=50; LH=53; OAL=102;   DCON=14; [mm]

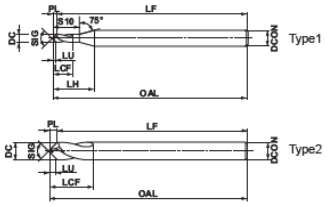
6-DLE solid carbide drill for centering and chamfering (countersink)

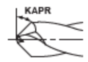
P M K N S H

METRIC STANDARD

External Coolant

■ Tip angle shape SIG 60°, 90°





(mm)

	DCON=3	3 < DCON≤6	6 < DCON≤10	10 < DCON≤16
h7	0 / −0.010	0 / −0.012	0 / −0.015	0 / −0.018

(inch)

	DCON=.1181	.1181 < DCON≤.2362	.2362 < DCON≤.3937	.3937 < DCON≤.6299
h7	0 / −.0004	0 / −.0005	0 / −.0006	0 / −.0007

Ordering number: DLE0800S080P090

SIG=90°; TYPE 2;

DC=8; LU=3.2; LCF=20; LH=234.8; OAL=74; LF=70.6; PL=3.4; DCON=8; [mm]

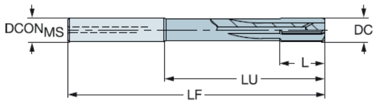
7-CoroReamer 835 (tolerance class H7)

CoroReamer™ 835 solid carbide reamer

For steel

For blind holes

	835.B..A1-PF
CNSC	1
CXSC	1



(1)

Ordering number: 835.B-1000-A1-PF

NOF=6;

DC=10;   LU=80;   LF=118.5;   L=20;   DCON=10;   [mm]

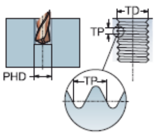
(2)

Ordering number: 835.B-0800-A1-PF

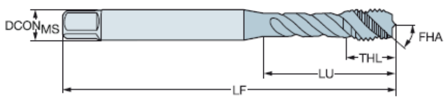
NOF=6;

DC=8;   LU=64;   LF=98.8;   L=16;   DCON=8;   [mm]

8-CoroTap 835



ULDR	2.5
FHA	45°
SUBSTRATE	HSS-PM



Ordering number:T300-XM100DA-M7

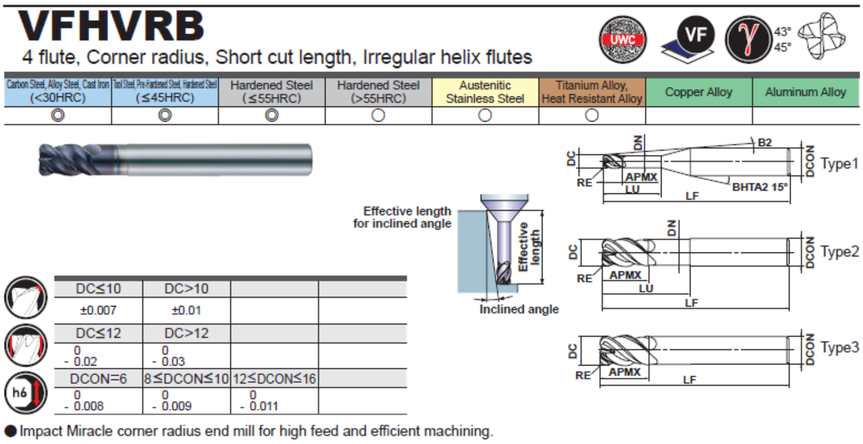
Tread Size M7 ; Grade PC110

LF=80; LU=31; THL=10; DCON=7; DT=7; TP=1; [mm]

NOF=3; GAMF=10°(radial rake angle); Thread profile angle=60°;

Premachined hole diameter PHD=6mm;

9-VFHVRB (internal shoulder/fillet)

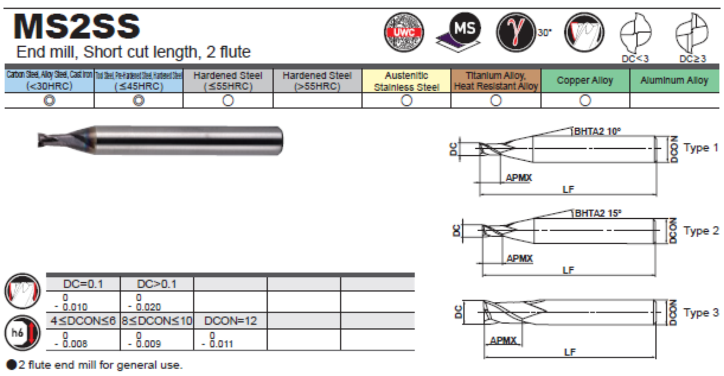


Ordering number: VFHVRBD1200R30N060

DC=12; RE=3; APMAX=18; LU=60; DN=11.7; LF=120; DCON=12; [mm]

NOF=4; TYPE 2; $\gamma_0 = 45°$;

10-MS2SS (slotting)



Ordering number: MS2SSD1000

Υ_0=30°; TYPE 3;

DC=10; APMX=15; LF=70; NOF=2; DCON=10; [mm]

3.Verification of operations

Operations will be feasible if the power requirement is below 16kW which is the spindle power given as input.

The retract federate will always be set to the rapid federate given as input equal to 42m/min.

-Bottom Surface roughing(op#5) **Tool1**

It's a face milling in a single pass operation in which the maximum depth of cut is equal to the stock left over the model of the part, therefore ap=3mm. The radial depth of cut is set to a reasonable value of ae=39.6mm.

According to the recommendations of the manufacturer the cutting speed is vc=445m/min and the feed per tooth fz=0.2mm/min.

$\varphi 1 = \arcsin\left(\frac{8.114}{31.5}\right) = 14.926°$ $\varphi_2 = 90°$; $\varphi = \varphi 1 + \varphi 2 = 104.926°$

$\varphi_0 = \frac{360}{z} = \frac{360}{5} = 72°$

$z = \frac{\varphi}{\varphi_0} = 1.457$→1<z<2→two cutting teeth symmetric with respect to the direction of motion of the tool

h_d=0.2 $* \cos 36°$*sin45° =0.1144mm

$k_c = 1500 * 0.1144^{-0.25} * \left(1 - \frac{0.19}{100}\right) = 2574 N/mm\text{^}2$

$A_d = 0.1144 * 3 = 0.3432 mm\text{^}2$

$F_c = k_c * A_d = 2410 * 0.699 = 883.49\ N$

$$T_c = 2 * \frac{883.49 * 31.5}{1000} = 55.66\ Nm$$

$$P_c = \frac{55.66 * 1000 * 445}{60 * 31.5} = 13105\ W$$

-Top and short side surfaces roughing(op#1,#4) **Tool 1**

The feasibility of these operations is ensured by the check over the feasibility of the bottom face milling because the process parameters and cutting data are the same.

A small difference should be considered in the case of the top surface, which has to leave a stock of 0.3mm to perform a further finishing operation, but this does not change the conclusion about feasibility since we are reducing the ap and therefore the power requirement will be lower and thus feasible.

-Top surface finishing Tool1, wiper insert

For the finishing step we kept the same cutting data and process parameters used in the face milling of the pocket but with a much smaller ap=0.3 and a higher feed since we are using wiper inserts, therefore we decided to set the feed to the max recommended value 0.3mm/min.

hd=0.27mm;

Ad= 0.081mm^2;

Kc=2076 N/mm^2;

Fc=168.23 N
Tc= 10.59 Nm
Pc=2130 W

-Chamfers(op #20, #21, #22, #23)
All the tools we used to create chamfers along the edges can mount the same type of insert, which determines the feed velocity that we are recommended to use→fz=0.17mm/tooth
The cutting speed depends on the material properties and the selected tool , therefore we can use in all the operations vc=370m/min.

To check the feasibility we treated them as face milling operations with small engagement, where both ap and ae are equal to 1.5mm since chamfers should be C1.5.

-Kr=30°: **Tool 3.1**

$$\varphi = \arccos\left(\frac{8-1.5}{8}\right) = 35.65°$$

$$\varphi_0 = 360°$$

$z = \frac{\varphi}{\varphi_0} = 0.099$→ $h_d = f_z * cos\varphi * sinKr$=0.15 $* cos35.65 * sin30$ =0.0609

$$k_c = 1500 * 0.0609^{-0.25} * \left(1 + \frac{0.34}{100}\right) = 3008N/mm\text{^}2$$

$$A_d = 0.0609 * 5.5 = 0.09mm\text{^}2$$

$$F_c = k_c * A_d = 3008 * 0.09 = 274.78N$$

$$T_c = \frac{274.78 * 8}{1000} = 2.198\ Nm$$

$$P_c = \frac{2.198 * 1000 * 370}{60 * 8} = 1694.48\ W$$

The other operations follow the same steps so we'll provide only the final results
-Kr=75° **Tool 3.3**
h_d =0.0426mm k_c=3290N/mm^2 A_d =0.0639mm^2 F_c =210.23N T_c = 1.68Nm
P_c= **1296W**

-Kr=45° **Tool 3.2**
h_d =0.0298mm k_c=3598N/mm^2 A_d =0.0447mm^2 F_c =160.8N T_c = 1.286Nm
P_c= **991.6W**

-Pocket roughing(op#17) **Tool 2, standard insert**

The machining of the pocket has a lot of parameters to consider if we want to use a single tool since it must be long enough to avoid collisions, therefore the utile length has to be greater than 50mm and can't have a diameter that is too big otherwise it would not be able to machine all the corners a go around the hexagonal pocket.

Sandvik catalogue gives some recommendations about cutting speed depending on the engagement of a specific tool and about the feed per tooth depending on the type of insert used(in op#17 the one for roughing).

The pocket creation involves both a larger engagement (during the helical ramping) and a small one along flat path.

Helical ramping: vc=340 m/min

fz=0.08 m/min

Shoulder milling: vc=380 m/min

fz=0.15 m/min

We need to verify the feasibility of both separately using respectively the formulas from helical ramping and face milling sections.

-Ramping:

Dmax=2*DC=40mm; Dmin=(20-2.3)*2)35.4mm

→a reasonable value for the machined hole diameter is DH=36mm, therefore the helical ramp diameter, which is the one followed by the center of the tool is DH-DC=16mm, which is the parameter to set in the fusion360 ramp section of the pocketing also with the ramp leading angle $\alpha = 2°$.

We use as depth the depth of cut for each revolution of a helix, P=$\pi * 16 * \tan(2) = 1.755mm$, which is the other key parameter for the simulation.

$h_d = fz = 0.08mm$

$k_c = 1500 * 0.08^{-0.25} * \left(1 - \frac{10.27*\pi}{180*100}\right) = 2815\ N/mm\text{^}2$

$v_f = \frac{0.08*2*1000*340}{\pi*20}$=865.8 mm/min

$Pc = \frac{1.755*20*865.8*2815}{60*10\text{^}6} = 1.42kW$

-Shoulder milling:

Since we have an operation involving both axial and radial cuts the calculation of power requirements can be approached separating the axial and radial contributes and then sum them to check if the total power is lower than the power provided by the spindle.

Each pass has a maximum roughing stepdown ap=5.5mm and a max stepover ae=10mm.

Radial (radial rake angle -9.599°):

$\varphi = \arccos\left(\frac{10-9}{10}\right) = 84.26°$

$\varphi_0 = \frac{360}{z} = \frac{360}{2} = 180° = \pi$

$z = \frac{\varphi}{\varphi_0} = 0.468$→ → $h_d = f_z * sin\varphi$=0.15 $* sin84.26$ =0.149mm

$k_c = 1500 * 0.149^{-0.25} * \left(1 + \frac{9.599*\pi}{180*100}\right) = 2418N/mm\text{^}2$

$A_d = 0.149 * 5.5 = 0.82mm\text{^}2$

$F_c = k_c * A_d = 2410 * 0.699 = 1983\ N$

$$T_c = \frac{1976 * 10}{1000} = 19.83\ Nm$$

$$P_c = \frac{19.83 * 1000 * 380}{60 * 10} = 12559\ W$$

Axial (axial rake angle 10.27°):

$$\varphi = \arccos\left(\frac{10-9}{10}\right) = 84.26°$$

$\varphi_0 = \frac{360}{z} = \frac{360}{2} = 180° = \pi$

$z = \frac{\varphi}{\varphi_0} = 0.468$→ → $h_d = f_z * sin\varphi$=0.15 $* cos84.26$ =0.015mm

$k_c = 1500 * 0.149^{-0.25} * \left(1 + \frac{10.27*\pi}{180*100}\right) = 4278N/mm\text{^}2$

$A_d = 0.015 * 5.5 = 0.0825mm\text{^}2$

$F_c = k_c * A_d = 4278 * 0.82 = 353\ N$

$$T_c = \frac{353 * 10}{1000} = 3.53\ Nm$$

$$P_c = \frac{3.52 * 1000 * 380}{60 * 10} = 2235\ W$$

Ptot=14794W

-Pocket finishing(op#18) **Tool 2, wiper insert**

For the finishing step we kept the same cutting data and process parameters used in the face milling of the pocket but with a much smaller ap=0.3 and a higher feed since we are using wiper inserts, therefore we decided to set the feed to the max recommended value 0.2mm/min.

hd=0.1989mm;

Ad= 0.06mm^2;

Kc=2242 N/mm^2;

Fc=134.5 N

Tc= 1.345 Nm

Pc=908 W

-OP#3a#3b#24#25

All these operations are ensured to be feasible since we decided to use the same tool adopted for the pocketing operation since it had a good compromise between machining time(cutting speed and feed rate) and reachability of deep points, always taking in account the type of steel we are machining and thus respecting the recommendations provided by manufacturers, furthermore, this tool has a main entering angle of 90°, which is an ideal angle to create straight surfaces through shoulder milling.

More precisely, operations 3a and 3b are contouring milling through multiple depths with max stepdown of 5.5mm, vc=405m/min and fz=0.15mm/tooth.

Operations 24 and 25 remove a higher amount of stock, indeed they are pocketing operations which have a max stepover of 9mm and max stepdown of 5.5, always keeping vc=405m/min and fz=0.15mm/tooth.

Fillet(op#26) **Tool 9**

Cutting data process parameters

The fillet in the bottom edges of pockets has been machined performing a shoulder milling operation by multiple depths of ap=8mm (in fusion360 corresponds to the maximum roughing stepdown) and radial depth of 3mm. We decided to use and end mill with corner radius RE=3mm so that the fillet was automatically created during the last pass. The Mitsubishi catalogue provides the RPM and the feed rate for some values of ap and ae related to the diameter of the cutter and the usable length. Since we decided to increase the depth of cut, we reduced both the values recommended, leading to n=2400 RPM and vf=3600mm/min.

Therefore vc=90.47m/min , fz=0.375mm/tooth

To check feasibility each step was considered as a face milling operation.

$$\varphi = \arccos\left(\frac{6-3}{6}\right) = 60°$$

$$\varphi_0 = \frac{360}{z} = \frac{360}{4} = 90° = \pi/2$$

$z = \frac{\varphi}{\varphi_0} = 0.66 \rightarrow$ $\rightarrow$ $h_d = f_z * sin\varphi$=0.375 $* sin60$ =0.32476mm

$$k_c = 1500 * 0.32476^{-0.25} * \left(1 - \frac{45*\pi}{180*100}\right) = 1971.4\ N/mm\text{^}2$$

$$A_d = 0.32476 * 8 = 0.97428$$

$$F_c = k_c * A_d = 1971.4 * 0.97428 = 1920\ N$$

$$T_c = \frac{1920 * 6}{1000} = 11.52\ Nm$$

$$P_c = \frac{11.52 * 1000 * 90.47}{60 * 6} = 2896\ W$$

-Drilling

All the drilling tools were taken from the Mitsubishi catalogue following the manufacturers recommendations to estimate the L/D parameter, which besides the actual length of the hole to make, has to consider an additional length to ensure a smooth entry which has to be 1.5*DC.

The cutting parameters are assigned considering the type of material to cut P1.1.Z.AN steel and the diameter of the hole of interest.

Op#10, Tool 4.1

According to the recommendations the L/D has to be 5 since we want to create a blind hole of 7.8mm of diameter (we left 2mm for the reaming) and depth 15mm.

v_c=85m/min;

f_r=0.2311mm/rev;

Feasibility check:

$h_d = \frac{f_r}{2} * \sin\left(\frac{\varepsilon}{2}\right) = 0.115 * sin70° = 0.108mm$

$k_c = 1500 * 0.108^{-0.25} * \left(1 - \frac{0.523}{100}\right) = 2602.887N/mm\text{^}2$

$A_d = f * \frac{D}{4} = 0.45mm\text{^}2$

Fc=1171N

Tc=4.568Nm

Pc=1659W

Op#13, Tool 4.2

According to the recommendations the L/D has to be 8 since we want to create a thorugh hole of 6mm of diameter and 30mm of depth.

v_c=80.18m/min;

f_r=0.2mm/rev;

Feasibility check:

$h_d = \frac{f_r}{2} * \sin\left(\frac{\varepsilon}{2}\right) = 0.1 * sin70° = 0.09mm$

$k_c = 1500 * 0.09^{-0.25} * \left(1 - \frac{0.523}{100}\right) = 2724N/mm\text{^}2$

$A_d = f * \frac{D}{4} = 0.3mm\text{^}2$

Fc=817.28N

Tc=2.45Nm

Pc=10.9kW

Op#6, Tool 4.4

According to the recommendations the L/D has to be 4 since we want to create a blind hole of 6mm of diameter and deep 15mm. The final size of the threaded hole will be M7x1.0, therefore we first create a hole that is equal to the premachined hole diameter required by the tapping tool, which is 6mm. Furthermore, the Mitsubishi materials catalogue also specifies the thread size associated with the specific cutting tool we picked, which is hence M7x1.0.

v_c=110.03m/min;

f_r=0.2mm/rev;

Feasibility check:

$h_d = 0.18mm$

$k_c = 2270N/mm^2$

$A_d = 0.3mm^2$

Fc=681N

Tc=2.04Nm

Pc=1248W

Op#15, Tool 4.5

According to the recommendations the L/D has to be 2 since we want to create a pilot hole of 8mm of diameter and 8mm of depth. It's a first step for creating the deep through hole, we have chosen the shortest tool as recommended because the initial stage can provide a big uncertainty due to the huge length of the deep drill.

v_c=121.35m/min;

f_r=0.23114mm/rev;

Feasibility check:

$h_d = \frac{f_r}{2} * \sin\left(\frac{\varepsilon}{2}\right) = 0.1 * sin70° = 0.1086mm$

$k_c = 2604N/mm^2$

$A_d = 0.4622mm^2$

Fc=1203.49N

Tc=4.81Nm

Pc=2424W

Op#16, Tool 4.6

According to the recommendations the L/D has to be 20 since we want to create a deep through hole of 8mm of diameter and 138mm of depth.

v_c=111.118m/min;

f_r=0.2997mm/rev;

Feasibility check:

$h_d = 0.1408mm$

$k_c = 1958N/mm^2$

$A_d = 0.46mm^2$
Fc=901N
Tc=3.6Nm
Pc=1670W

-Counterbore(op#12), Tool 5
We chose a flat bottom drill to create the counterbore with best accuracy possible. The L/D has to be 2 since we want to create a hole with depth of 4mm and in this case it starts with an already existing hole inside which has to be enlarged from 7.8 to 12.5mm diameter. This operation has been performed before the reaming to preserve the final surface finish of the inner hole.
v_c=74.676m/min;
f_r=0.2mm/rev;
Formulas are in the respective section of the report, we'll provide the final results.
n=1901.6 RPM ap=2.25mm Ad=0.225mm^2
hd=0.2mm→Kc=2235N/mm^2
Fc=502.9N Tc=5.1547 Nm Pc=950.45W

-Countersink(op#14) Tool 6
According to the technical drawing the countersink should be made with a tool having a diameter of 8mm and a point angle of 90°, in order to enlarge the previously machined hole of 6mm.
v_c=80.08m/min;
f_r=0.07mm/rev;

Feasibility:
ap=1mm; Ad=0.035mm^2 hd=0.0247mm;
Kc=3754N/mm^2
Fc=3754N; Tc=0.919Nm; Pc=306.88W

-Reaming
Op#9 Tool 7.1
The top hole has to be enlarged from a 9.8mm to 10mm of diameter to reach H7 tolerance grade, provided by the tool.
v_c=180m/min;
f_r=0.8mm/rev;

Feasibility:

ap=0.1mm; Ad=0.013mm^2 hd=0.0646mm;
Kc=2943N/mm^2
Fc=38.259N; Tc=1.136Nm; Pc=681W

Op#11 Tool 7.2

The top hole has to be enlarged from a 7.8mm to 8mm of diameter to reach H7 tolerance grade, provided by the tool.

v_c=180m/min;

f_r=0.8mm/rev;

Feasibility:

ap=0.1mm; Ad=0.013mm^2 hd=0.0646mm;
Kc=2975N/mm^2
Fc=38.675N; Tc=0.916Nm; Pc=687W

Tapping(op#7), **Tool 8**

Cutting data process parameters

We chose the tool in order to satisfy the thread requirement M7x1.0 meaning that the thread diameter has to be 7mm and the pitch 1mm.

The only one from Sandvik catalogue available with cutting data recommendations for our type of steel P1.1.Z.AN and matching the chosen grade C110 and with ULDR=2.5 (usable length diameter ratio) had $v_c = 32.5m/min$. Then we can get:

$$n = \frac{1000 * 32.5}{\pi * 7} = 1477.867 \text{ RPM}$$

$$V_f = pitch * n = 1477.867mm/min$$

The premachined hole diameter for this tool is 6mm and therefore it will produce a maximum diameter of 7mm. Meanwhile in the operation setting in fusion360 we have to set the thread pithch and the diameter offset, both equal to 1mm.

The formulas were taken from the tapping section of formulas.

The chip thickness can be estimated multiplying the feed rate, which is equal to the pitch for one pass, by the sin of the thread profile angle.

$$h_d = 1 * \sin\left(\frac{60}{2}\right) = 0.5\text{mm}$$

$$k_c = 1500 * 0.5^{-0.25} * \left(1 - \frac{45*\pi}{180*100}\right) = 1769.8\ N/mm^2$$

$$T_c = \frac{1^2 * 7 * 1769.8}{8000} = 1.54857\ Nm$$
$$P_c = \frac{1.54857 * 2 * 1477.867}{60} = 76.286W$$
Feasible

Slot(op#19) **Tool 10**

The slot was machined by a single pass having a radial depth ae=10mm and an axial depth ap=8mm.

The Mitsubishi catalogue provided values for the revolutions per minute and the feed rate that we reduced respectively by 70% and 60% respectively as they recommended for higher depth of cut, thus leading to n=1920 RPM , vf=360mm/min.

Therefore vc=60.318m/min , fz=0.09375.

Verified feasibility using face milling formulas

$\varphi_0 = \frac{360}{z} = \frac{360}{2} = 180° = \pi$

$z = \frac{\varphi}{\varphi_0} = 2$→ Average approach → $h_{dav} = \frac{0.09375*2*10}{\pi*10}$=0.05968mm

$A_{dav} = 0.05968 * 8 = 0.47744mm^2$

$k_c = 1500 * 0.05968^{-0.25} * \left(1 - \frac{30*\pi}{180*100}\right) = 3018.898\ N/mm\text{^}2$

$F_{cav} = 3018.898 * 0.47744 = 1441.34N$

$$T_c = \frac{2 * 1441.34N * 10/2}{1000} = 14.41Nm$$
$$P_c = \frac{14.41 * 1000 * 60.318}{60 * 10/2} = 2897.96\ W$$
Feasible.

4. Discussion on technical requirements

Tolerances

Pocket(Ra3.2)

According to Sandvik guide, in order to achieve a good surface finish in a face milling operation, we have to keep the feed per tooth <=0.6*bs (insert feature).

We decided to keep the feed to the max recommended value 0.2mm/min.

Then, to verify that the limit of roughness has not been reached we have used the formulas from the theory in the case of face milling:

Ramax=$\frac{0.2*10\text{^}3}{\text{atan}\ (90)} = 2.238\mu m$

Ra=Ramax/4=0.559 μm →Requirement satisfied

Top Surface(Ra1.6)

We followed the same recommendations just described; therefore we took fz=0.3mm/tooth so that:

Ramax=$\frac{0.3*10^3}{\text{atan}\,(45)} = 3.38\mu m$

Ra=Ramax/4=0.845 μm →Requirement satisfied

Hole tolerances

The tolerances specified for the holes were guaranteed by the reamers used, since they are made on purpose to perform finishing operation to reach an H7 grade of tolerance, while for the H10 grade it is enough to perform a drilling operation over the hole diameter since it is not a high requirement.

The thread size requirement was also guaranteed by the choice of the tapping tool.

The geometric specification in the 2d drawing such as location, orientation and forms specifications are all guaranteed by the sequence of operation and clamping modalities(also refer to the 3.2 section of the report).

Furthermore, Fusion360 allows to highlight (after all the machining operations) those areas having extra material with respect to the final model we want to obtain, within a tolerance that we can specify, therefore if there are no blue parts highlighted and the tolerance is set for example to 0.1 it means that we should have respected the form tolerances.

.