19MCE401 - PROCESS PLANNING AND PRODUCT DEVELOPMENT

STUDY NOTES

UNIT 2 - PROCESS PLANNING ACTIVITIES

TOPIC 6 – GANTT CHARTS

Handled by:

Dr. T. Prakash M.E., Ph.D.,

Professor & Head,

Department of Mechatronics Engineering,

SNS College of Technology,

Coimbatore - 35.

Process Parameter Calculation:

Introduction:

The process planning involves the various activities such as drawing interpretation, material evaluation and process selection, selection of machines and tooling, setting process parameters, selection of work holding devices, selection of quality assurance and inspection methods, cost estimating and then documenting the details using route sheets.

• The process planning activities—drawing interpretation, material evaluation and process selection and selection of machines and tooling were discussed in detail.

• In this unit, the remaining process planning activities—setting process parameters, selection of work holding devices (i.e., jigs and fixtures), selection of inspection/quality assurance methods and economics of process planning will be described in detail, one by one.

Process parameter calculation:

The three important process parameters to be calculated for each operation during process planning is:

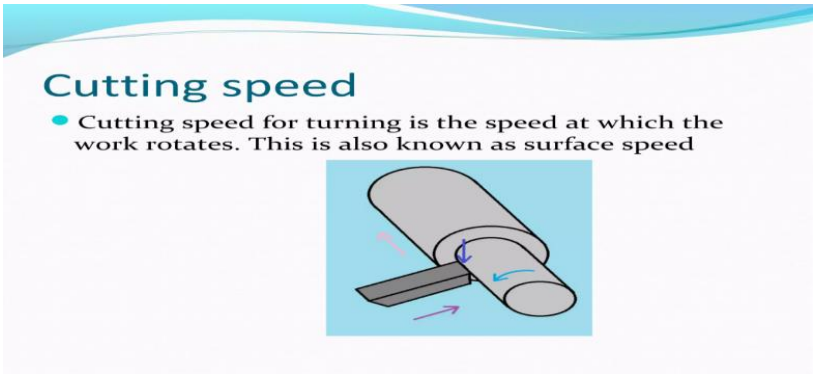
1. Cutting speed

2. Feed rate

3. Depth of cut

Cutting Speed:

The cutting speed also known as surface cutting speed or surface speed, can be defined as the relative speed between the tool and the workpiece. It is a relative term, since either the tool or the workpiece or both may be moving during cutting. Unit: It is expressed in meters per minute (mpm).



Factors Affecting the Selection of Cutting Speed:

The major factors considered for selecting cutting speed are as follows.

1. Nature of the. Cut

 Continuous cut like turning, boring, drilling, etc., are done at higher cutting speed. Shock initiated cuts in shaping machine, planning machine, slotting machine, etc. are done at lower cutting speed.

 Intermittent cuts, as in milling, hobbing. etc., are done at quite lower speed for dynamic loading.

2. Work Material (type, strength, heat resistance, toughness, chemical reactivity, etc.)

 For example, harder, stronger, heat resistant and work harden able materials are machined at lower cutting speed.

 Soft, non-sticky and thermally conductive materials can be machined at relatively higher cutting speed.

3. Cutting Tool Material (type, strength, hardness, heat and wear resistance, toughness, chemical stability, thermal conductivity, etc)

 For example, HSS tools are used at within 40 m/mm only in turning mild steel whereas for the same work cemented carbide tools can be used at the cuffing speed of 80 to 300 m/min.

4. Cutting Fluid Application

 Proper selection and application of cutting fluid may increase in cutting speed by 20 to 50%

5. Purpose of Machining

 Rough machining with large material removal rate (MRR) is usually done at relatively low or moderate cutting speed.

 Finish machining with small feed and depth of cut is usually done at high cutting speed.

6. Kind of Machining Operation

 Unlike turning, boring. etc., the operations like the threading, reaming, knurling. etc., are carried out at much lower cutting speed.

7. Capacity of the Machine Tool

 Powerful, strong, rigid and stable machine tools allow much higher cutting speed, if required and permissible.

8. Condition of the Machine Tool

 Cutting Speeds (metre/minute) for different combinations of operation and material

Material	Operation					
	Turning and boring	Drilling	Reaming	Shaping, slotting and planing	Milling	Grinding
Aluminium	300	120	120	25	200–300	20
Brass	45–75	50	25	12–15	40	22
Cast iron	20	15	10	10	50	12
Copper	30	50	15	10	40	22
Mild steel	30	25	12	20	20	15

 Cutting Speed ranges for different combinations workpiece and tool material

S.No.	Workpiece Material	Cutting Speed (m/min)	
		HSS	Carbides
1.	Low-carbon steels	20 — 110	60 — 230
2.	Medium-carbon steels	20 — 80	45 — 210
3.	Steel alloys (Ni-based)	20 — 80	60 — 170
4.	Grey cast iron	20 — 50	60 — 210
5.	Stainless steels	20 — 50	55 — 200
6.	Chromium nickel	15 — 60	60 — 140
7.	Aluminium	30 — 110	60 — 210
8.	Aluminium alloys	60 — 370	60 — 910
9.	Brass	50 — 110	90 — 305
10.	Plastics	30 — 150	50 — 230

FEED AND FEED RATE

Feed is the distance through which the tool advances into the workpiece during one revolution of the workpiece or the cutter. Feed rate is the speed a: which the cutting tool penetrates the workpiece. Unit: Feed rate is usually expressed in millimeters per spindle revolution (mm/rev) or millimetres per minute (mm/mm).

Factors Affecting Feed Rate

The factors that are considered during selection of feed value are:

I. Work material (type, strength, hardness, etc.)

II. Capacity of the machine tool (power, rigidity, etc.)

III. Cutting tool (material, geometry and configuration)

IV. Cutting fluid application

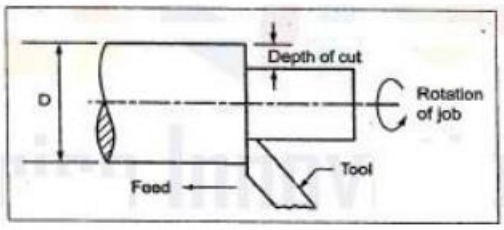
V. Surface finish desired

VI. Type of operation

VII. Nature of cut

DEPTH OF CUT

Depth of cut is the thickness of the layer of metal removed in one cut or pass, measured in a direction perpendicular to the machined surface. The depth of cut is always perpendicular to the direction of feed motion. Unit: It is generally measured in mm.



Illustrates the terms feed and depth of cut

The feed and depth of cut for a particular operation depend on the material to be machined, surface finish required and tool used.

Selection of Depth of Cut

1. Depth of cut for Turning and Boring

The general guidelines for turning and boring recommend a depth of cut of 6mm for roughing and 0.4mm for finishing

2. Depth of cut for Milling

The maximum depth of cut for milling is generally considered to be half the cutter diameter

3. Depth of cut for Drilling

The maximum depth of cut for drilling is generally considered to be half the feed rate of the tool and minimum considered to be 0.3mm

4. Depth of cut for Shaping and Planning

In general, the recommended depth of cut for shaping and planning are in the range of 1-4mm

5. Depth of cut for Grinding

The general recommendations for depth of cut for surface and cylindrical grinding is equal to the values for feeds selected in mm/pass.

Machining Time Calculations

The important reasons for selecting / calculating the process parameters – cutting speed, feed rate and depth of cut are to determine the machining times. Because the data for cutting speed, feed rate and depth of cut for the processes will be used to calculate the machining times.