Training Notes

Skills for Competitiveness (S4C)

ADVANCED TECHNICAL TRAINING FOR MASTER TRAINERS

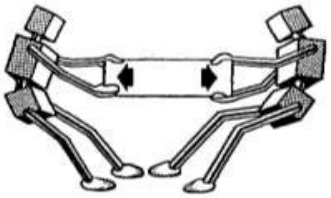
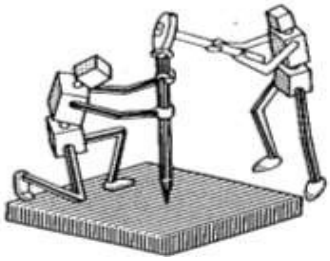
METALLURGY & CAD/CAM for CNC TURNING

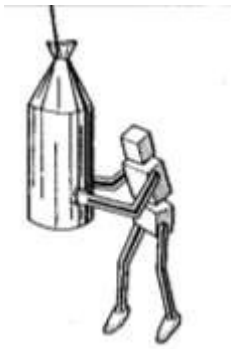
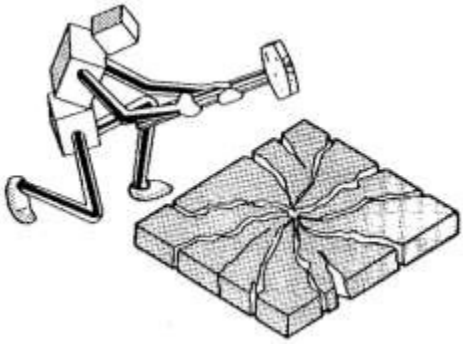
1.0_MECHANICAL PROPERTIES OF METALS

1.1 Physical Properties of Metals

Properties of metal refer to the way metals behave when acted upon by external forces.

Common Terms

	**Ductility** • It is the ability of a metal to permanently deform without breaking. • The deformation is in the direction of the force applied.
	**Tensile Strength** • It is the maximum amount of pull a metal can withstand before breaking.
	**Compressive Strength** • It is the ability of a metal to withstand heavy compressive (pressing) forces.
	**Hardness** • It is the ability of a metal to resist penetration by force • Cannot be scratched, drilled, milled etc

	**Toughness** • It the property of a metal to withstand shock or impact.
	**Malleability** • It is the property of a metal which allows it to be hammered or rolled into shapes.
	**Brittleness** • It is the property of metal which does not allow permanent distortion before breaking.

1.2 Machinability

It describes how easy or difficult a material can be machined. Some of the factors that affect machinability are:

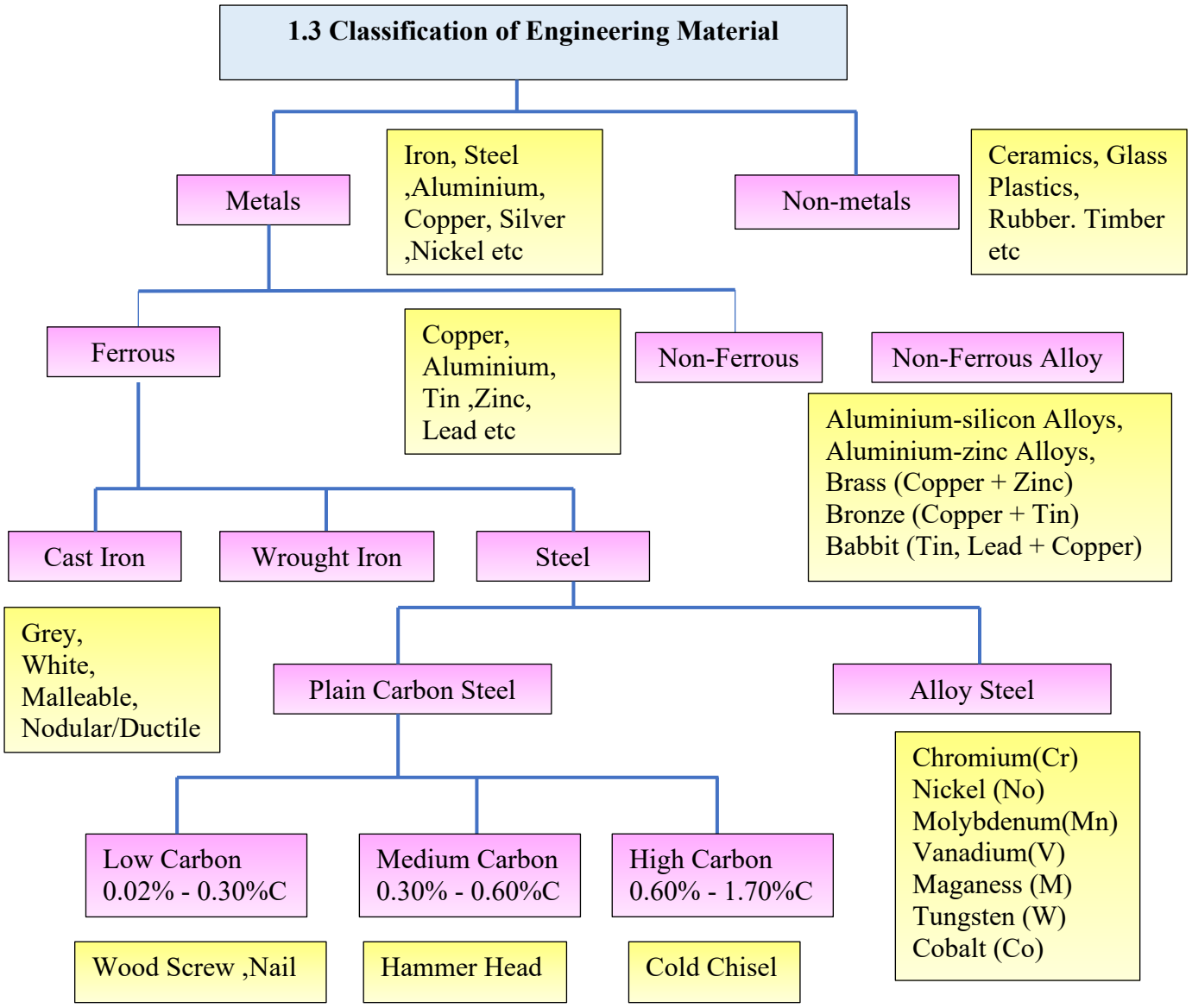
- Type of tool material used
- Types of work material used
- Shape of tool
- Hardness of work material
- Shape and size of work

Ferrous Metals and Non-ferrous Metals

Ferrous Metals	Non-ferrous Metals
There are metals or alloys which contain iron.	There are metals or alloys which do not contain iron.
Mild steel, high-carbon steel, high-speed steel, stainless steel and cast iron are ferrous metals	Copper, brass bronze , aluminum, tin and zinc are non-ferrous metals

Metals and Non-Metals

Metals	Non-Metals
They are mostly opaque, hard, heavy, and ductile . Example : Mild Steel	They are metals or alloys which do not contain iron.
They are good conductor of heat and electricity. Example :Copper	They are less ductile.
Alloys are obtained by mixing or melting,two or more metals in order to improve properties of materials. Example: 6061 Aluminum Alloy Al(97.7%)+Cu(0.25%)+Magnesium(1%)+Maganese(0.15%)+Silicon(0.6%)+Chromium).2%)	There are poor conductor of heat and electricity.


1.3 Classification of Engineering Material
Metals
Iron, Steel ,Aluminium, Copper, Silver ,Nickel etc
Non-metals
Ceramics, Glass Plastics, Rubber. Timber etc
Ferrous
Copper, Aluminium, Tin ,Zinc, Lead etc
Non-Ferrous
Non-Ferrous Alloy
Aluminium-silicon Alloys, Aluminium-zinc Alloys, Brass (Copper + Zinc) Bronze (Copper + Tin) Babbit (Tin, Lead + Copper)
Cast Iron
Wrought Iron
Steel
Grey, White, Malleable, Nodular/Ductile
Plain Carbon Steel
Alloy Steel
Chromium(Cr) Nickel (No) Molybdenum(Mn) Vanadium(V) Maganess (M) Tungsten (W) Cobalt (Co)
Low Carbon 0.02% - 0.30%C
Medium Carbon 0.30% - 0.60%C
High Carbon 0.60% - 1.70%C
Wood Screw ,Nail
Hammer Head
Cold Chisel

1.4 Types of Iron

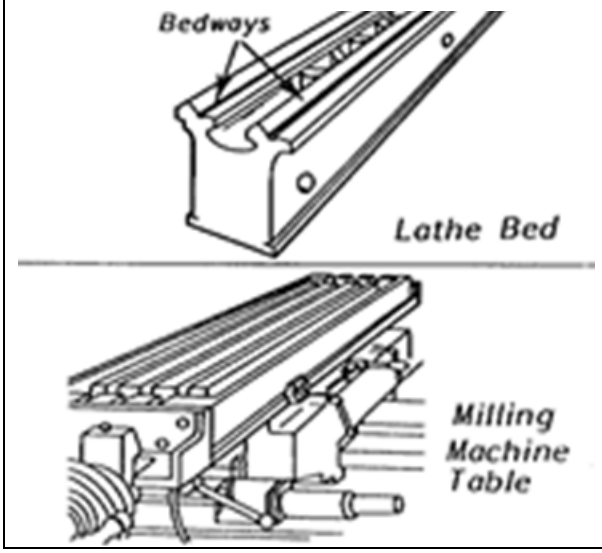
Pig Iron

It is the first product from iron ore.

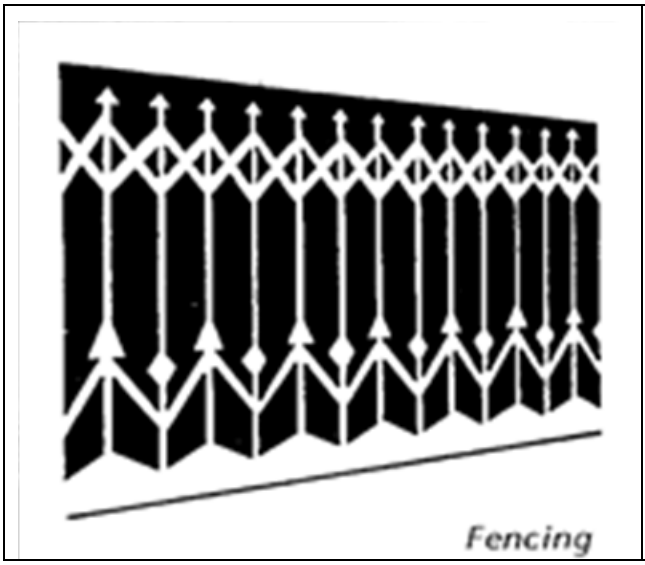
It contains impurities.

It is used for making steel and casting.

Cast Iron

	It is remelted pig iron with carbon added. Lathe beds and milling machine tables are made of cast iron.

Wrought Iron

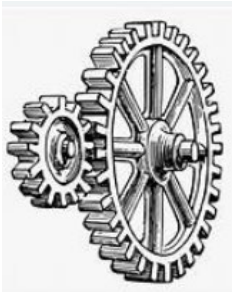
	It is basically purified pig iron with a very low carbon steel. It is tough and malleable. It can be easily formed into any shape in the cold state. Generally, it is used for making ornamental objects, e g gates and fencing

Plain Carbon Steel

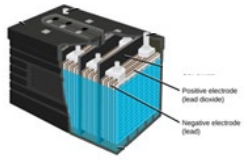
Basically , it is made of iron and carbon without major alloying elements.

It is classified according to the carbon content in the steel.

Classification and Uses of Plain Carbon Steel

Type	Carbon Content	Characteristics	Some Uses
Low-carbon Steel	0.02% - 0.30%	Cannot be through hardening. Can be case-hardening. Very ductile and malleable. Can be cut and welded easily.	Car body panels 
Medium-carbon Steel	0.3% - 0.60%	Can be through hardening. Less ductile. Has greater tensile strength. Ideal for steel forging.	Gears 
High-carbon Steel	0.6% - 1.70%	Can be heat-treated to high hardness Has the highest tensile strength among the carbon steel. Generally, used for hand tools.	Railway track 

1.5 Characteristics and Uses of Common Non-ferrous Metal

Non-ferrous Metal	Color	Characteristics	Some Uses
1.Copper 	Reddish brown	• High electrical conductivity. • Resistant to corrosion. • Very ductile.	• Electrical cables • Copper tubing.
2.Aluminum 	Greyish white	• Resistant to corrosion. • High electrical conductivity. • Ductile and malleable. • Light in weight.	• Household utensils • Outdoor transmission lines • Doors and windows frames
3.Tin 	Silvery white	• Can be rolled into veery thin foil. • Resistant to corrosion.	• Coating food containers • Essential metal for alloy of bronze, solder and bearing materials
4.Zinc 	Bluish white	• Resistant to corrosion. • Can be mixed with other metals to form alloys.	• Used in die-casting • Padlock • Toy cars
5.Lead 	Silvery but turns grey quickly when exposed to air	• Soft but heavy. • Resistant to corrosion. • Low melting point. • Low tensile strength. • Can be mixed with other metals to form alloys. • Can be added to steel to improve machinability.	• Battery plates • Solder

2.0_ALLOYS

An alloy is a mixture of two or more metals.

Alloying is used to produce a metal by mixing a base material(e g carbon steel) with smaller amounts of other metals(e g tungsten, nickel and chromium.

2.1 Purposes of Alloying

Adding alloying elements may change one or more of the following properties to steel.

- Increase tensile strength
- Increase hardness
- Increase toughness
- Increase wear-resistance
- Increase rust-resistance
- Impart red hardness
- Improve machinability

2.2 Effect of Alloying Elements on Steel

Alloying Element / Effect	Carbon	Chromium	Cobalt	Lead	Nickel	Tungsten
Increase tensile strength	√	√			√	
Increase hardness	√	√				
Increase toughness		√			√	
Increase wear-resistance	√	√			√	√
Impart red hardness			√			√
Improve machinability				√		

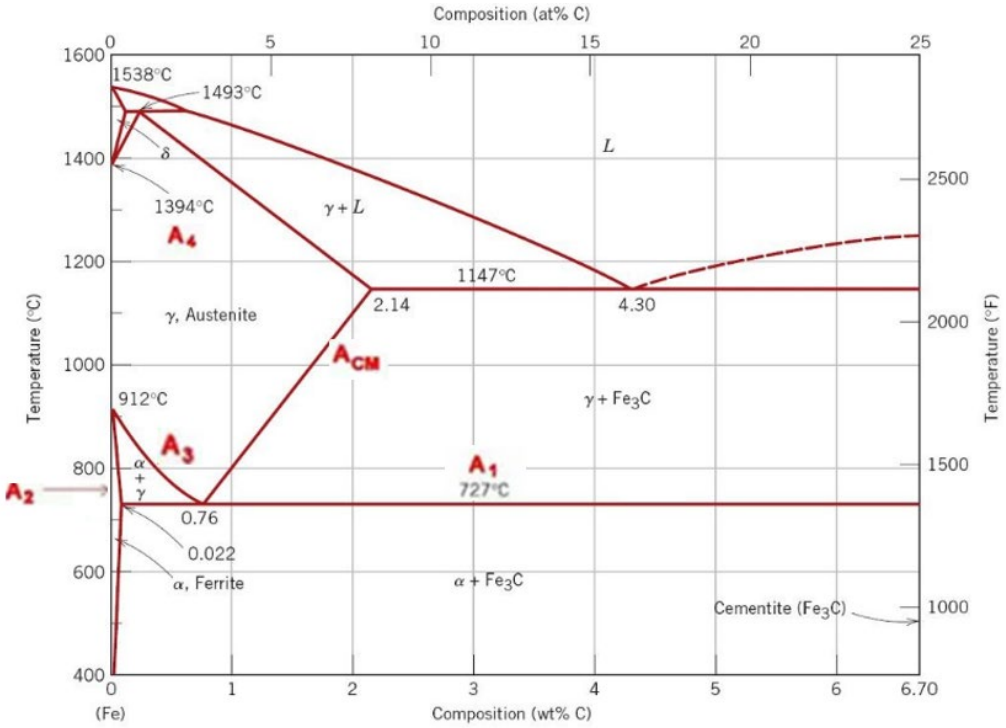
2.3 Characteristics and Uses of Common Non-ferrous Alloys

Non-ferrous Alloys	Metals Present	Characteristics	Common Uses
1.Aluminum-silicon Alloys	Aluminum Silicon	• Easily forged. • Easily cast.	Automotive pistons Marine fitting
2.Aluminum-zinc Alloys	Aluminum Zinc Magnesium Copper Manganese Chromium	• High tensile strength. • Resistant to corrosion.	Aircraft structure parts
3. Brass	Copper Zinc	• Resistant to corrosion. • Easily cast.	Taps Bolts and nuts Door hinges
4. Bronze	Copper Tin	• Resistant to corrosion. • Strong & Tough.	Bush bearings Tubing
5. Babbit	Tin Lead Copper	• Strong. • Malleable.	Bush bearing in motor vehicles Pewterware

3.0_IRON AND CARBON EQUILIBRIUM DIAGRAM

3.1 What is Iron-Carbon Diagram

The Iron-Carbon diagram is a graphical representation of the phases and microstructures that occur in iron-carbon alloys(steel and carbon) at different temperatures and carbon contents. It shows the relationship between percentage of carbon in the alloy and temperature at which the alloy is heated or cooled.



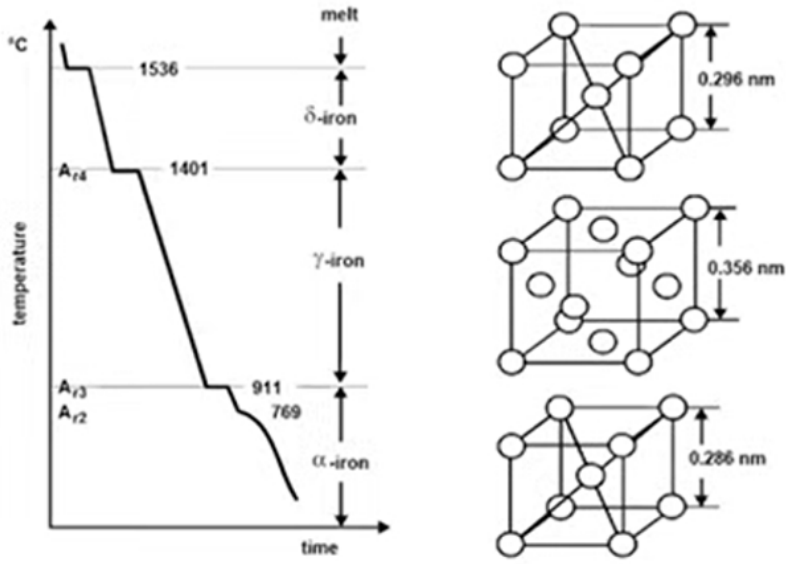
The diagram highlights various phases, including

- Austenite(γ)
- Ferrite(α)
- Cementite: Iron carbide(Fe3C).
- Pearlite :Mixture of ferrite and cementite.

Symbols are used to denote the critical temperature

- Ac = arrest during cooling
- Ar = arrest during heating
- A0 = This is the critical temperature of cementite i.e. 210° C.
- A1 = It is the lower critical temperature point where austenite to pearlite transformation occurs .
- A2 = It is the critical temperature (768° C) of iron where it turns into a paramagnetic substance. It is also known as Curie temperature.
- A3 = It is the boundary temperature between γ austenite and (austenite+ ferrite).
- Acm = It is the boundary temperature between γ austenite and (austenite+ cementite)
- A4 = At this temperature α-ferrite turns into δ-ferrite

3.1.1 Different phases of Pure Iron (Fe)



Crystal iron changes at 3-arrest point

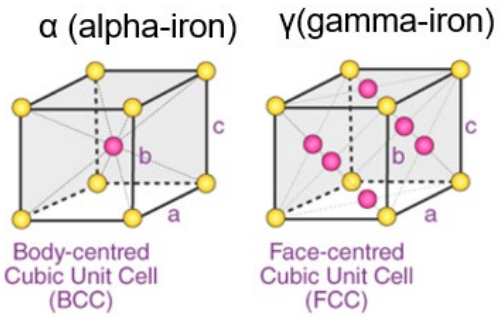
Delta-iron
δ-iron
BCC crystal structure
at 1536 °C

Gamma-iron
γ–Fe (Austenite)
BCC crystal structure
at 1401 °C –Non Magnetic

Alfa-iron
α–Fe (Ferrite)
BCC crystal structure
at 911 °C - Magnetic

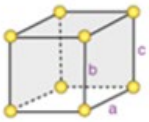
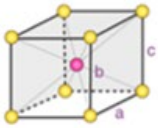
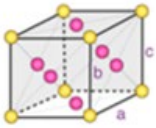
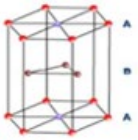
The temperature stops rising for a short time (arrests) and then continues climbing once the crystal change is complete.

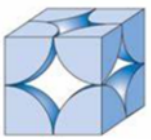
Face-Centered Cubic & Body-Centered Cubic



The carbon in iron is an interstitial impurity .The alloy may form a face-centered cubic (FCC) lattice or a body-centered cubic (BCC) lattice. It will form a solid solution with α,γ and δ phases of iron.
The alpha-iron (low temperature) has no spaces for carbon atoms to reside, while the gamma-iron (high temperature) is open to the free movement of small carbon atoms.

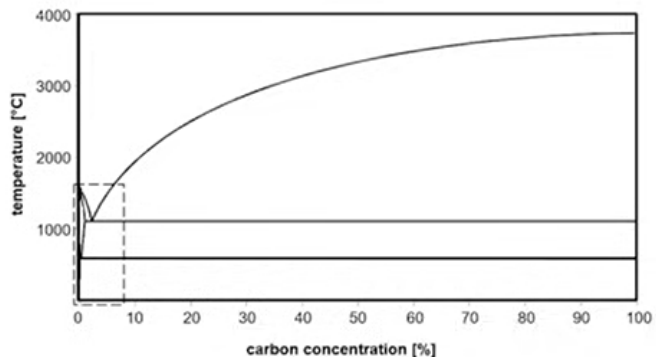
3.3 Cubic Crystil Structure

Bravais lattice	Primitive cubic	Body-centered cubic	Face-centered cubic	Hexagonal close packed (HCP)
Pearson symbol	*cP*	*cI*	*cF*	
Unit cell	 one lattice point	 two lattice point	 four lattice points	 Seven lattice points
Packing Fraction	52%	68%	74%	74%
Coordination number	6	8	12	12
Lattice point	$(\frac{1}{8} \times 8)$	$(\frac{1}{8} \times 8)+1$	$(\frac{1}{8} \times 8)+(\frac{1}{2} \times 6)$	(2+3+2)




SC	Brittle BCC	Ductile FCC	HCP
Polyolefin(PO)	Chromium Iron(α) Molybdenum Tantalum Tungsten	Aluminum Copper Gold Lead Nickel Platinum Silver	Cadmium Cobalt Titanium(α) Zinc

3.4 The Complete Iron-Carbon Phase Diagram

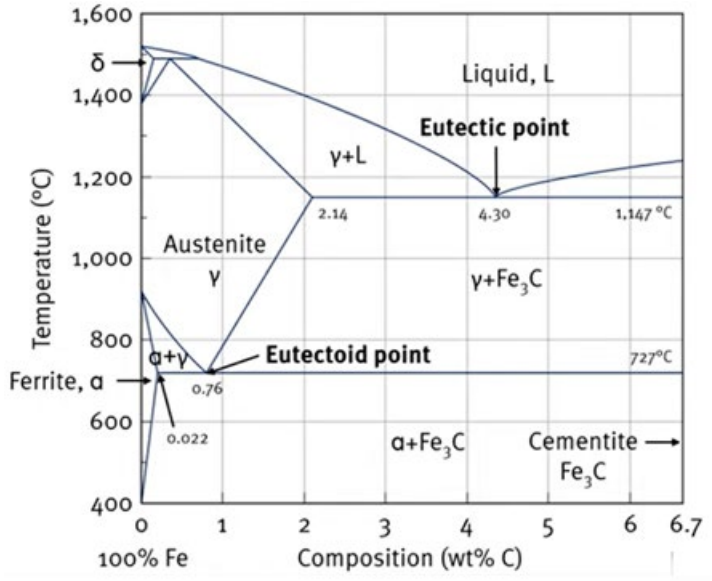


Iron at higher carbon concentration contain mainly graphite.

Graphite is not suitable for constructional material.

Iron contain lower 6.7%C is useful as engineering material as shown in dotted lines.

3.5 Eutectic Point & Eutectoid Point



Phase changes occur at different temperatures (vertical axis) for different compositions (horizontal axis).

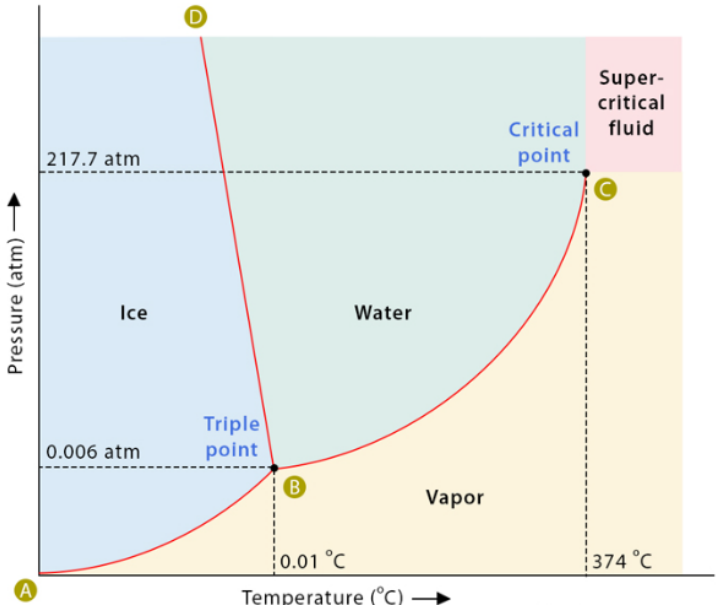
Eutectic point - Corresponds to the composition and temperature of lowest melting point at 4.3 wt% C.

Eutectoid point - The point in a phase diagram indicating a solid is in equilibrium with two other solid phases at 0.76 wt% C.

3.6 Phase Diagrams of Water

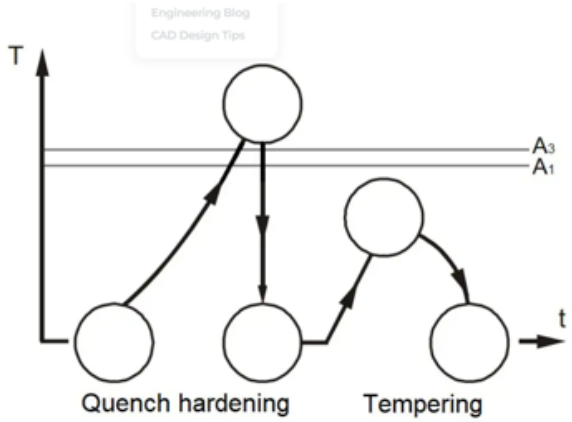
Example of the water phase diagram describes a point (triple point of water) where water can coexist in three different phases at the same time(equilibrium). This happens at just above the freezing temperature (0.01°C) and 0.006 atm(standard atmospheres).

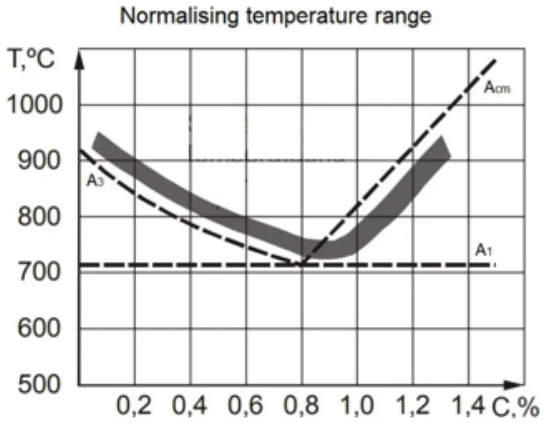
Phase Diagram of Water



3.7 Uses of Phase Diagrams

- Development of new alloys based on application requirements.
- Production of these alloys.
- Development and control of appropriate heat treatment procedures, such as tempering, annealing, normalizing to improve the chemical, physical, and mechanical properties of these new alloys.
- Troubleshooting problems that arise of these new alloys.

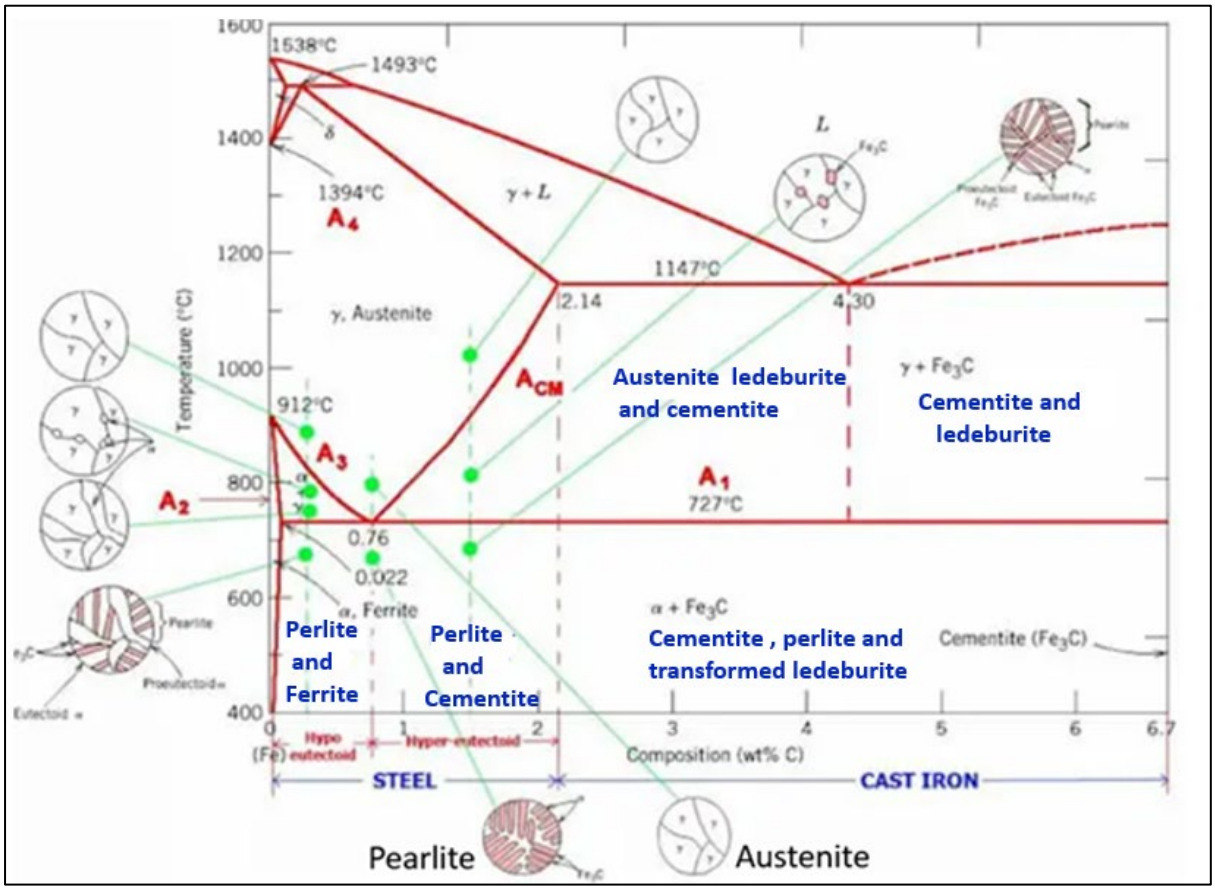


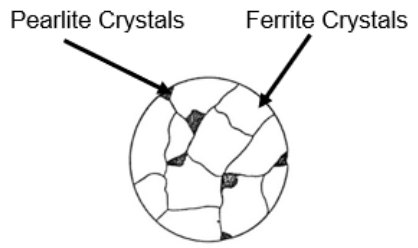
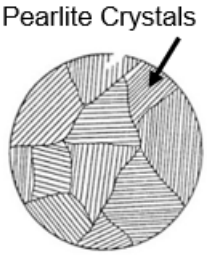
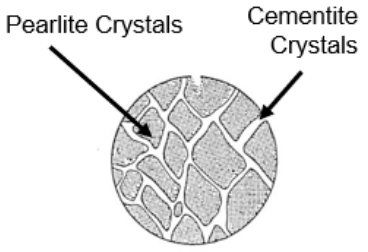


Advantages of Phase Diagrams

- Help to prevent overdesign for applications.
- Help to develop alternative alloying elements.
- Help metallurgists understand which phases are thermodynamically stable, metastable, or unstable in the long run. Appropriate elements can then be chosen for alloying to prevent machinery breakdown.
  Example: Material for exhaust piping if not chosen properly, may lead to a breakdown at higher temperatures.

3.8 Iron-Carbon Phase Diagram & Microstructure of Steel



Low Carbon Steel(Mild Steel)	Medium Carbon Steel(0.76 wt% C)	High Carbon Seel
		

Properties of Microstructure of Steel

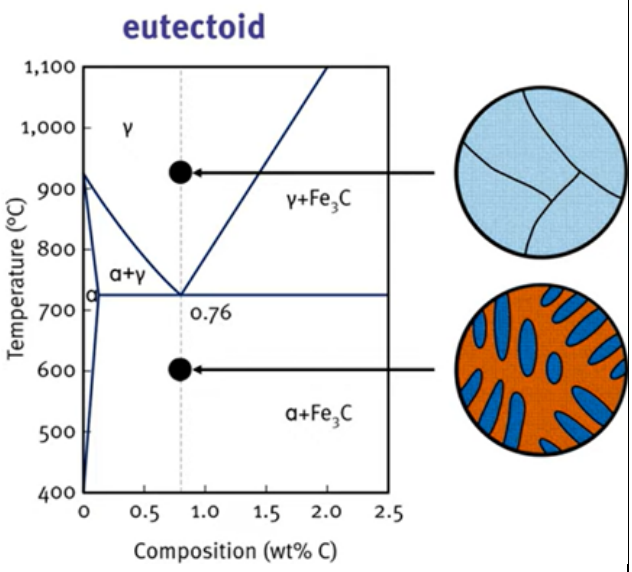
Microstructure of Steel	Properties
Austenite (γ)	Soft and non-magnetic
Cementite (Fe3C)	Hard and Brittle
Ferrite (α)	Soft and ductile
Pearlite	Alternating layers of cementite and ferrite

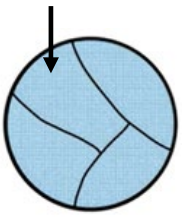
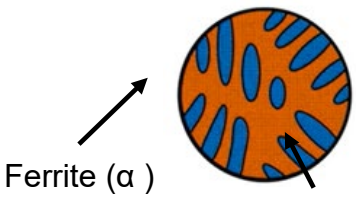
3.9 The eutectoid composition

Phase transformation of eutectoid steel:

1. Heating above critical temperature (A1) forms austenite which is soft and non-magnetic
2. Upon slow cooling, austenite transforms into pearlite (ferrite + cementite) at eutectoid temperature (0.76% C).
3. Microstructure: Consists entirely of pearlite (ferrite + cementite)
4. Properties: Good balance of strength, hardness, and ductility.
5. Applications: Used in various applications, such as rails , axles , high-strength wires

Eutectoid steel has a uniform microstructure of pearlite, which provides a good balance of strength and ductility.

The eutectoid composition Austenite (γ)- Soft and non-magnetic Pearlite [α+Fe3C] -Eutectoid structure • Alternating layers of cementite and ferrite. • Not a single phase but 2 phase arrangement Cementite (Fe3C) - Hard and Brittle Ferrite (α) - Soft and ductile	

Microstructure of eutectoid steel	
Austenite (γ) 	Pearlite  Ferrite (α) Cementite (Fe3C)

3.10 Hypo- eutectoid composition

In hypo-eutectoid steel, the carbon content is less than 0.76% C. This results in a microstructure that consists of the following ferrite and pearlite.

1. Ferrite: A soft , ductile phase that forms during cooling.
2. Pearlite : A mixture of ferrite and cementite that forms from the remaining austenite

The proportion of ferrite to pearlite depends on the carbon content. Hypo-eutectoid steels are known for their:

- Good balance of strength and ductility
- Suitable for applications requiring toughness and formability.

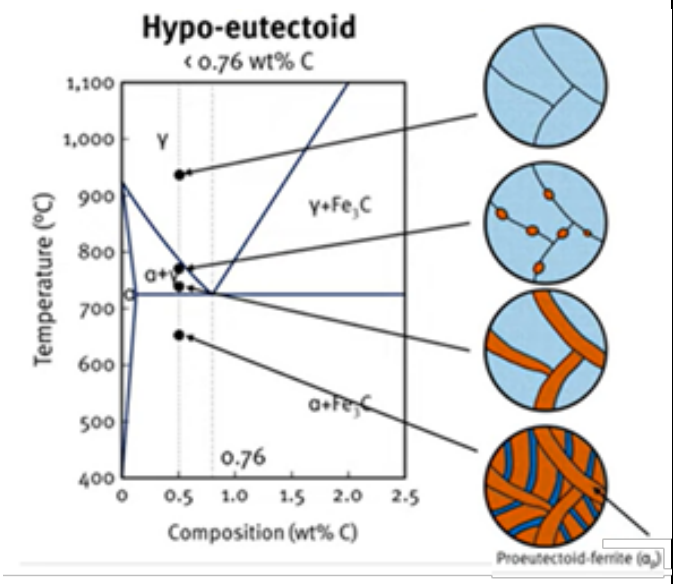
Examples of applications include structure components, machinery parts and automotive components.

<table>
<tr>
<td>Hypo- eutectoid steel < 0.76wt% C
• Pearlite grains with proeutectoid ferrite.

Phase transformation:

○ Austenite (γ)

○ Ferrite (α) + Austenite (γ)

○ Ferrite (α) + Pearlite</td>
<td> 
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<tr>
<td colspan="2">Microstructure of hypo-eutectoid steel</td>
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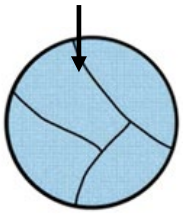
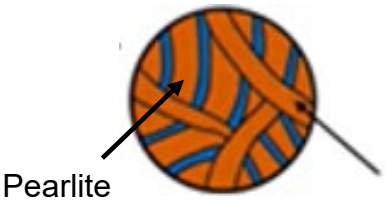
<tr>
<td>Austenite (γ)
  </td>
<td>Ferrite + Pearlite
 
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3.11 Hyper- eutectoid composition

Phase transformation of hyper-eutectoid steel:

In hyper-eutectoid steel, the carbon content is between 0.76% to 2.14 %C. This result in a microstructure that consists of the following pearlite and cementite.

1. Heating above critical temperature (Acm) forms austenite.
2. Upon slow cooling cementite forms first, precipitating out of austenite. Remaining austenite transforms into pearlite (ferrite + cementite) at eutectoid temperature.

The presence of excess cementite affects the steel hardness, brittleness, and wear resistance**.**

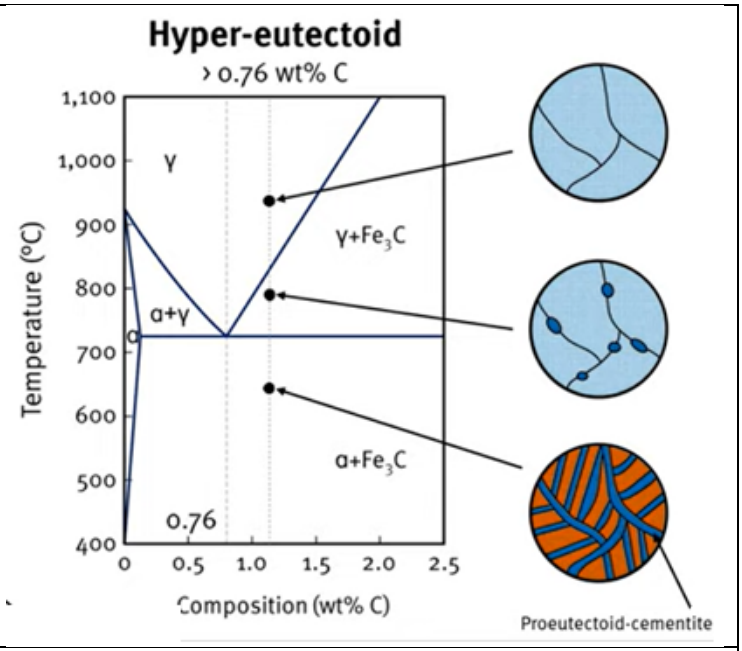
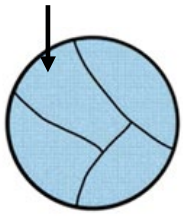
- Microstructure: Pearlite + Cementite.
- Properties: High hardness and wear resistance, but can be brittle.
- Applications: Used in high-strength, wear-resistant applications, such as cutting tools and bearing.

<table>
<tr>
<td>Hyper- eutectoid steel > 0.76wt% C
• Pearlite grains with proeutectoid cementite.

Phase transformation:

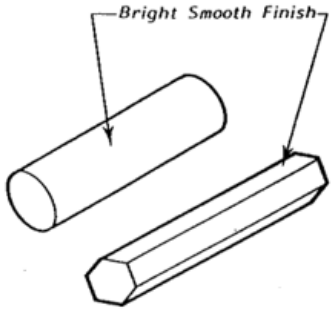
○ Austenite (γ)

○ Austenite (γ) + Cementite (Fe3C)

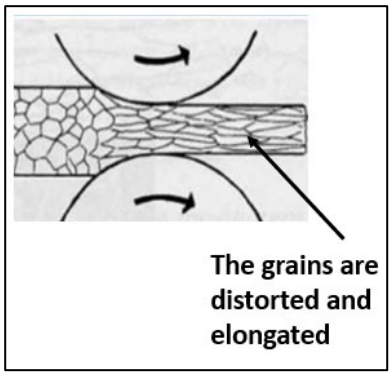
○ Pearlite + Cementite (Fe3C).</td>
<td> 
</td>
</tr>
<tr>
<td colspan="2">Microstructure of hyper-eutectoid steel</td>
</tr>
<tr>
<td>Austenite (γ)
  </td>
<td>Pearlite + Cementite (Fe3C)
 
</td>
</tr>
</table>

4.0_COLD WORKING AND HOT WORKING

4.1 Cold Working



Bright Drawn Mild Steel



Cold Working

It is the process of rolling, twisting or bending metals at room temperature.

<u>Effects on Metal</u>

- The metal will have a bright surface finish.
  Example:
  Bright-drawn mild steel is cold-rolled steel.
- The grains are compressed and deformed.
- The metal becomes harder.
- Residual stresses are set up in the metal.

<u>Advantages</u>

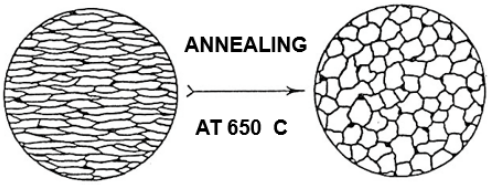
Stock material can be accurate produced.

It is used where hardness and higher tensile strength are desired.
Example: In the manufacturing of wire.

It is an ideal process for manufacturing thin metal sheet.

Relieve residual stresses

Changes of Microstructure after annealing



Causes recrystallisation of the distorted Ferrite

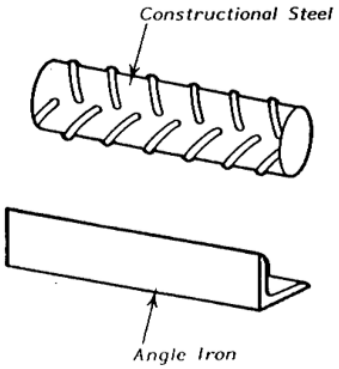
Relieve residual stresses from Cold Working :

Sometimes it is necessary to reduce stresses in a metal which has been cold-worked to restore ductility and facilitate further processing.

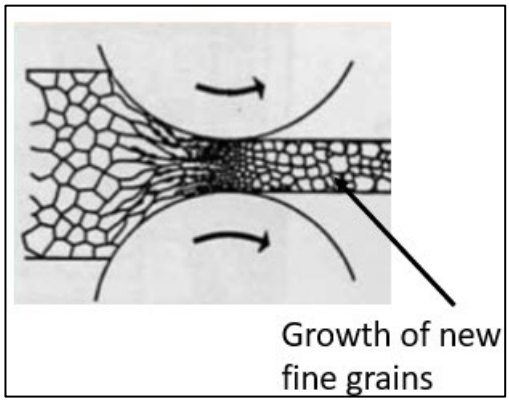
This is done by heating the metal to slightly higher that the re-crystallization temperature but lower than its melting point and cooling it down slowly.

The internal structure will then return back to normal

4.2 Hot Working



Black Mild Steel



Hot Working

It is the process of forming or bending metals at above re-crystallization temperature.

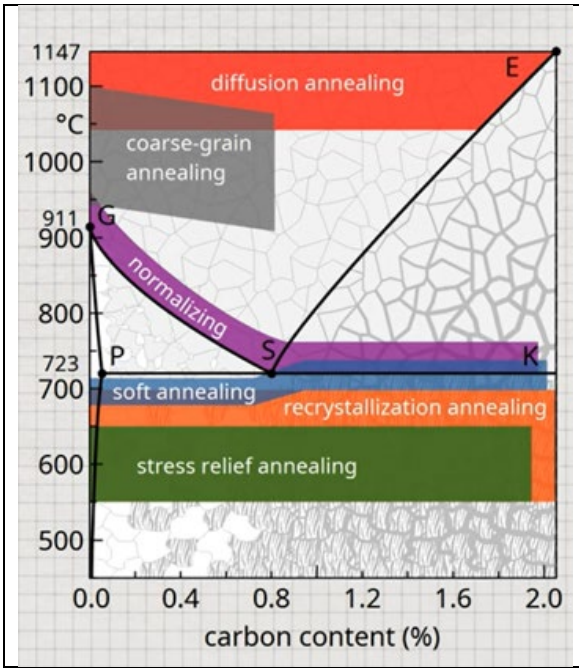
Effects on Metal

- "Scale" will form on the metal surface.
  Example:
  Black mild steel is a hot-rolled steel.
- If the temperature is too high, the metal will have a weak structure because of grain growth.

Advantages

- Less energy is needed in production.
- It is cheaper to produce.
- No residual stresses.

Heat treatment of steel & purpose



Purpose	Heat Treatment
Better formability	• Soft annealing
Better machinability	• Soft annealing, • Coarse grain annealing
Adjustment grain size	• Coarse grain annealing • Recrystallization annealing, • Normalizing
Homogenization Of alloying elements	• Diffusion annealing, • Solution annealing
Reduction of residual stresses	• Stress relief annealing

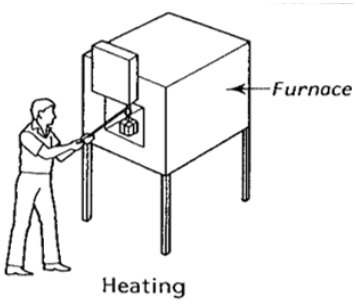
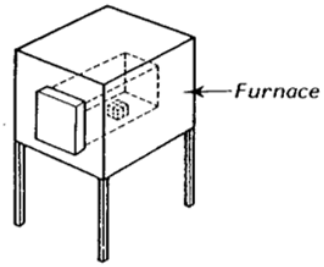
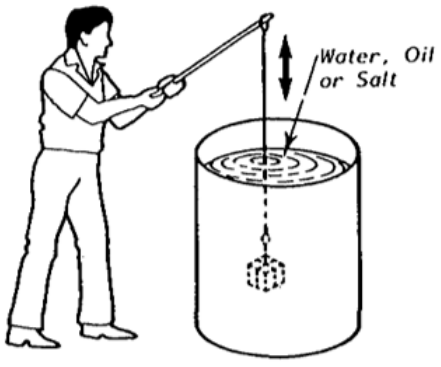
5.0_COMMON HEAT TREATMENT PROCESSES

During heat treatment, metals are heated and cooled to change their mechanical properties. The common heat-treatment processes are as follows:

- Full Annealing
- Normalizing
- Hardening
- Tempering

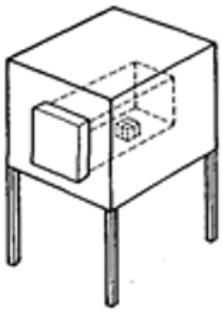
5.1 Understanding the Processes

All heat-treatment processes involve 3 main steps.

 Heating	**Heating:** Heating the metal to the required temperature
 Soaking	**Soaking:** Soaking(holding) the metal at a specific temperature for a period of time.
 Cooling	**Cooling:** Cooling(quenching) the metal at the required rate. <u>General Guidelines:</u> **Steel**:15-60 minutes for hardening,1-2 hours for tempering. **Alloy Steel:** 30-120ninutes for hardening,1-2 hours for tempering.

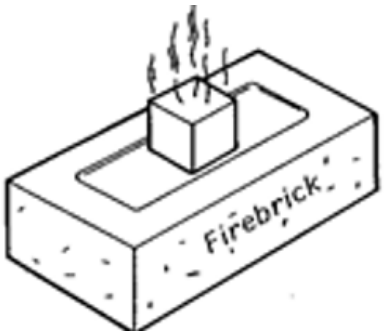
5.2 Purpose of Full Annealing

- To soften hardened steel
- To improve ductility of steel
- Full annealing leads to the formation of coarse pearlite

Cooling Work	**Full Annealing of Carbon Steel**
 A. Inside Furnace  B. Inside Sand, Lime or Other Medium	It is a process which involves the following steps: 1. Heating the work slightly above the upper critical temperature (A3 line) and lower critical temperature (A3,1 line). 2. Soak the work until it has reached the annealing temperature throughout. 3. Cool the work slowly. Switch off the furnace and leave the work inside furnace ,or bury the work in sand, lime or other suitable medium. (This process is to relieve the internal stress for metals that need subsequent treatments like shaping, bending ,rolling or welding)

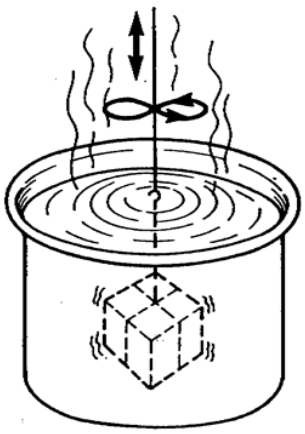
5.3 Purpose of Normalizing

- To remove stresses in steel.
- To improve the machinability of steel.
- To refine grain structure leads to the formation of finer pearlite structure.

Cooling Work in Air	**Normalizing**
 	It is a process which involves the following steps: 1. Heat the work slightly above the upper critical temperature (A3 or Acm line). 2. Soak the work until it has reached the normalizing temperature throughout. 3. Cool the work in air.

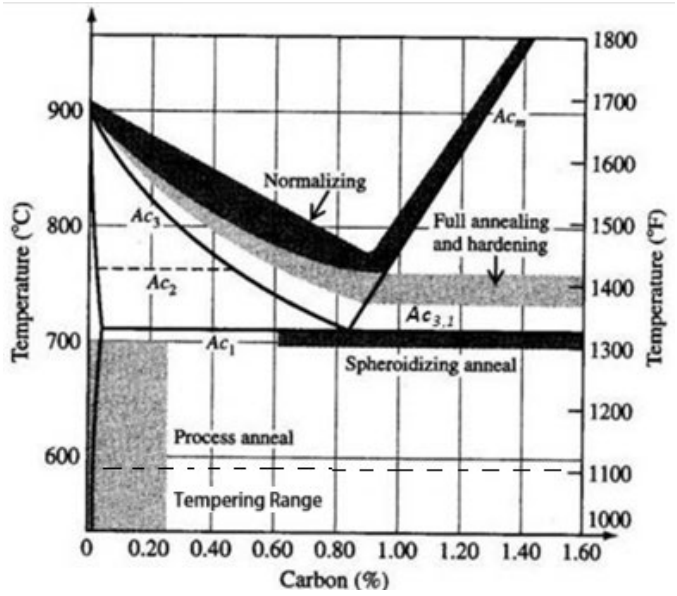
5.4 Purpose of Hardening

- To increase the hardness in steel.
  Example :
  Cutting tools , gauges and bearings must be hardened after being machined before they can be used.

Quenching Work	**Hardening of Steel**
	It is a process which involves the following steps: 1. Heating the work slightly 50° to 100° C above the upper critical temperature (A3 line) for hypo-eutectoid steel and slightly above lower critical temperature (A3,1 line) for hyper-eutectoid steel. 2. Soak the work until it has reached the hardening temperature. 3. Quench the work in a suitable medium. (Due to rapid cooling, carbon freezes at its location, and microstructure appear. The microstructure is called martensite, which is very hard and brittle)

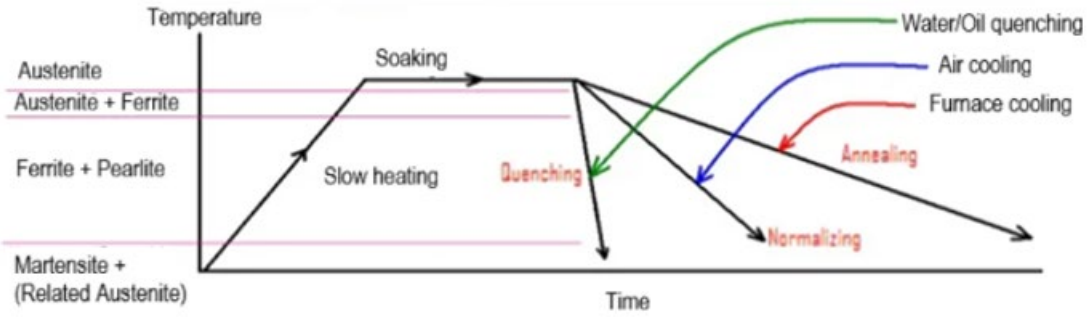
5.5 Purpose of Tempering

Heating of hardened metal to reduce brittleness and improve toughness

Basic Guide to Heat Treatment of Carbon Steel	**Tempering of Carbon Steel**
	It is a process which involves the following steps: 1. Heat the hardened work to tempering temperature. 2. Soak the hardened work until it has reached the tempering temperature. 3. Cool the hardened work in air. .

Overview of Heat treatment Process & Tempering Temperature

Heat treatment	Quenching Medium	Cooling Rate
Hardening	Water/Oil quenching	Fast/Medium
Normalizing	Air cooling	Slow
Annealing	Furnace cooling	Very Slow
Tempering	Air cooling	Slow



Metal	Tempering Temperature Range (°C)
Carbon Steel	200-550
Stainless Steel	300-600
Tool Steel	500-700
Aluminum	150-250
Copper	250-400
Brass	300-500
Bronze	300-600

Hardening and Strengthening of Steel

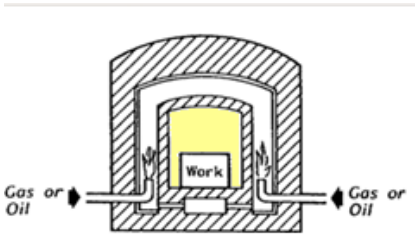
- High-alloy Steels (e.g.X153CrMoV-12) are usually completely hardened over the entire cross-section
- High-alloy Steels require lower critical cooling rates as compared to medium carbon steel (e.g. C45)

	Hardening	Strengthening
Steel	X153CrMoV-12	C45
Heating	1020°C	840°C
Quenching	Air	Water/Oil
Tempering	300°C	550°C / 660°C
Application	Cutting Tool	Spindle, Gear

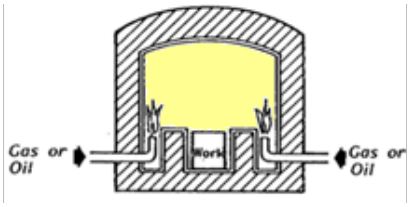
6.0_HEAT TREATMENT FURNACES

Gas, oil or electricity is used to provide in furnaces for heat-treatment

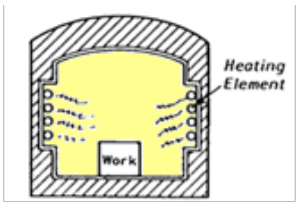
6.1 Types of Furnaces



A. Muffle Furnace



B. Open Furnace



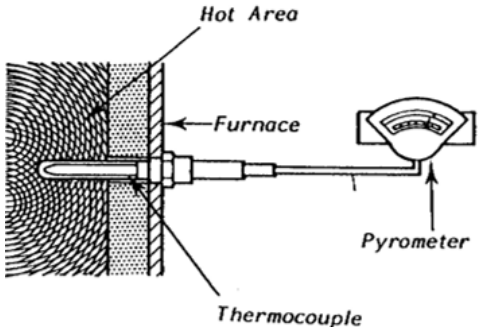
C. Electric Furnace

They are box-shaped ovens lines with heat-resistant bricks.

Gas, oil or electricity are used to heat the furnace and the work.

A temperature of up to 1000° C is easily obtained.

6.2 Measuring and Indicating Furnace Temperature



Measuring Temperature in Furnace

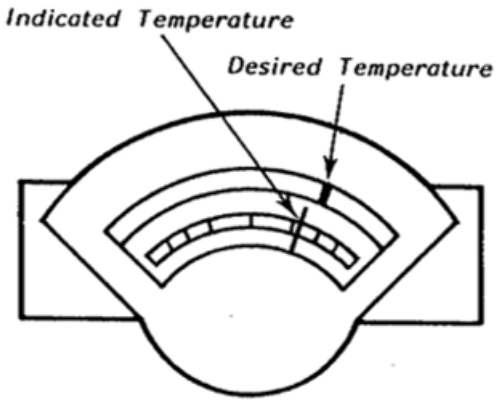
The thermocouple is mounted in the furnace.

It measures the temperature inside.

The pyrometer is placed outside the furnace and is connected to the thermocouple by electrical cables.

The pyrometer indicated the temperature of the furnace.

Control Furnace Temperature



Pyrometer

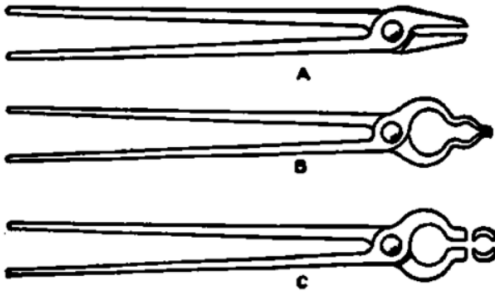
Set the desired temperature on the pyrometer.

When the temperature in the furnace reaches the desired level set on the pyrometer, the current supply cuts off automatically.

When the temperature drops below the desired level, the current supply resumes automatically.

6.3 Safety Precautions in Heat Treatment

- Do not place flammable materials near the furnace.
- Use a face shiel, heat-resistant gloves and proper attire for protection against intense heat.
- Always use dry tongs when putting or removing work from the furnace.
- Splattering will result when water or moisture comes into contact with the work or carburizing compound at high temperatures.

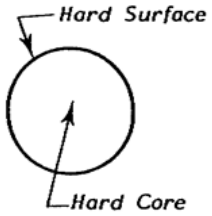
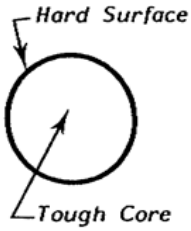


Types of Tongs

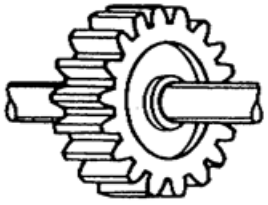
7.0_CASEHARDENING OF LOW CARBON STEEL

Casehardening steel has a hardened layer and a tough core.

7.1 Difference Between Casehardening and Through Hardening

 **A. Through Hardening**	A through-hardened part has a hard surface and hard core.
 **B. Casehardening**	A case-hardened part has a hard surface layer but a tough core.

7.2 Purposes of Casehardening

 Bearing  Gear	**Purposes of Casehardening** To make parts that require a wear-resistant surface and a tough core. E.g. Gears and bearing

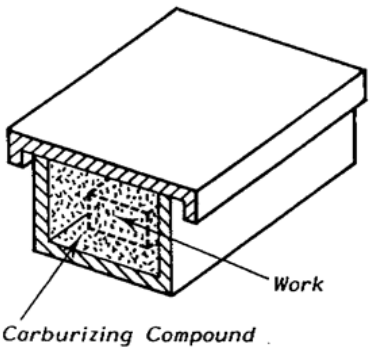
Casehardening Methods

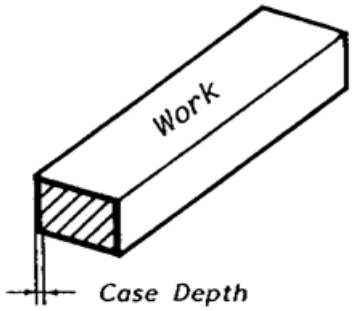
Generally, carburizing is one of the common casehardening processes.

There are 3 methods of carburizing:

- Pack carburizing
- Gas carburizing
- Liquid carburizing

7.3 Pack Carburizing of Low-carbon Steel





1. Place the work in a steel box and pack it with carburizing compound.

   The carburizing compound contains substances that are rich in carbon.

   The carburizing compound comes in a form of grains or pellets.

2. Place the steel box in the furnace and heat it to about 950° C (Carburizing temperature).

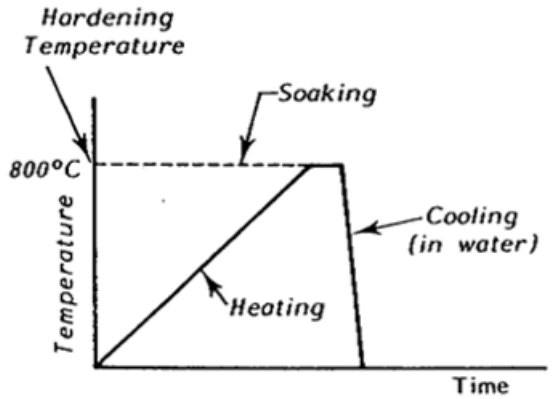
3. Soak the steel box at the carburizing temperature for 15-60 mins, depending on the required case depth.

   A case depth of 0.1mm-0.5mm can be obtained in this way.

4. Remove the work with a pair of tongs.

5. Cool the work in air.

Hardening and Tempering of Carburized of Low-carbon Steel



Hardening of Carburized Low-carbon steel

Hardening of Carburized Steel

1. Heat the work to the hardening temperature (800°C - 820°C).

2. Soak the work until it has reached the hardening temperature.

3. Quench the work in a suitable medium.

Tempering

4. Heat the work to tempering temperature (200°C - 550°C).

5. Soak the work until it has reached the tempering temperature.

6. Cool the hardened work in air.

8.0_TENSILE STRENGTH TEST

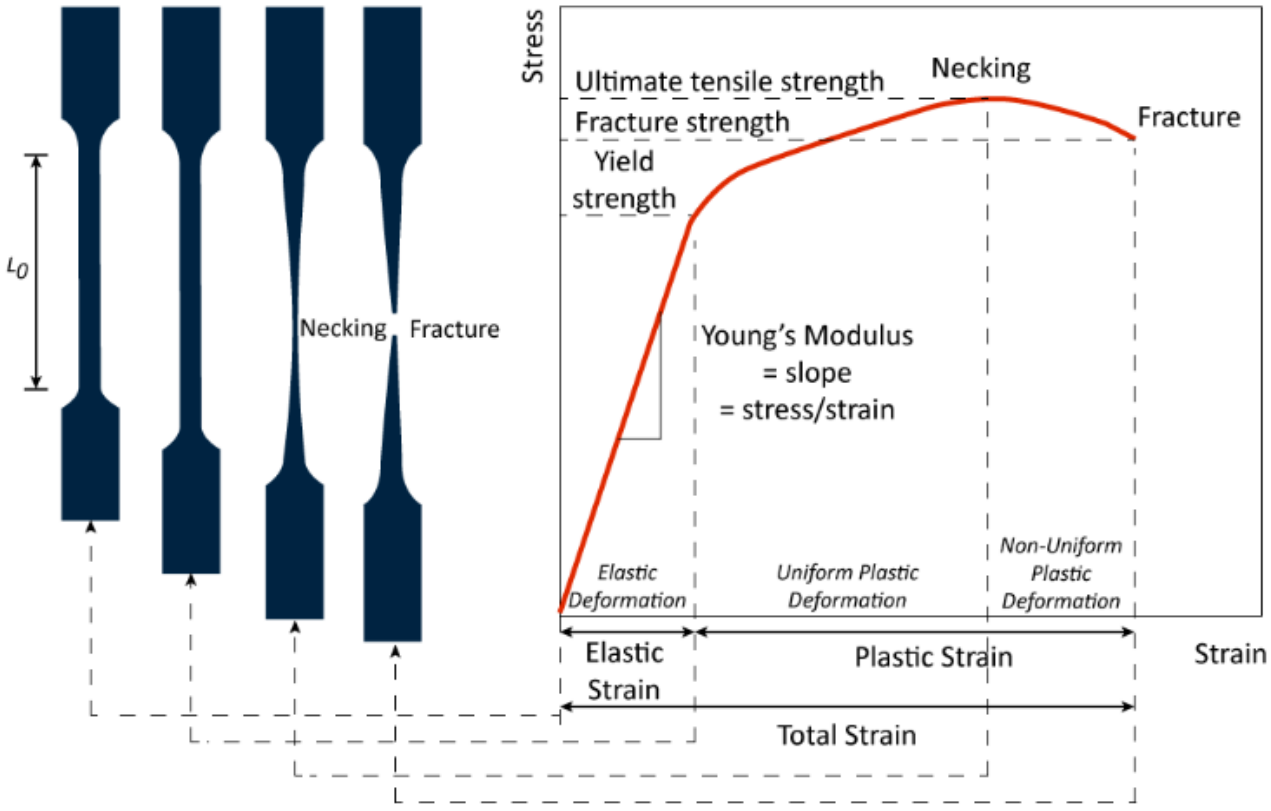
8.1 Purpose of Tensile Strength Test

Measures a material's resistance to tensile forces, determining its strength and ductility.
A sample is stretched until it breaks, recording stress and strain data.

Key Results:

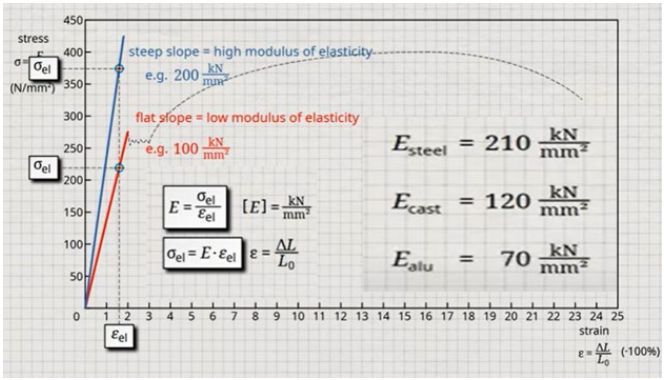
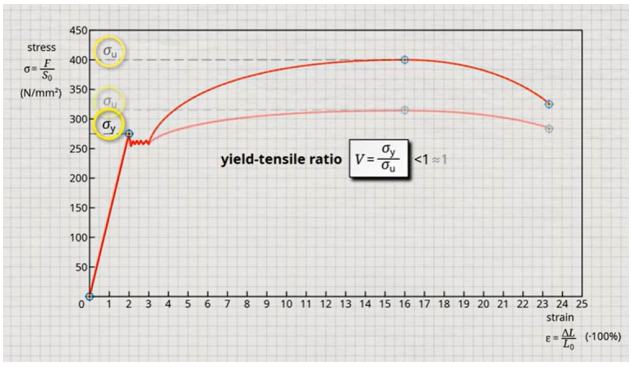
- Yield Strength: Stress at which material begins to deform plastically.
- Ultimate Tensile Strength (UTS): Maximum stress before failure.
- Elongation: Measure of ductility, calculated as percentage increase in length.

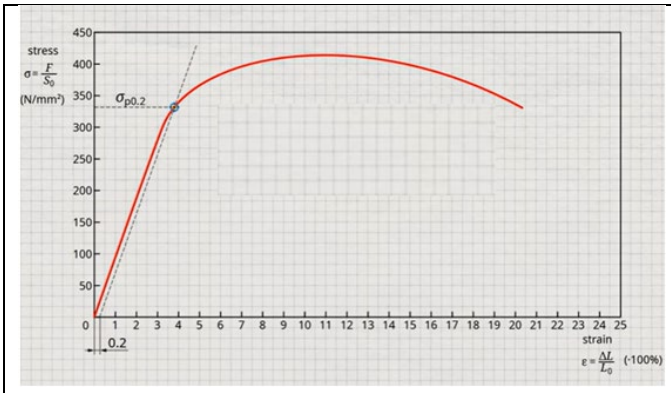
Applications:
Used to evaluate material properties, ensure quality, and inform design decisions in various industries, including construction, aerospace, and manufacturing.

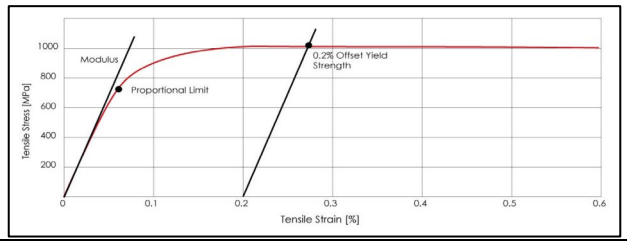


Elastic Deformation
Elastic deformation is a temporary deformation, which is recoverable. When the applied load is removed, the object returns to its original shape.

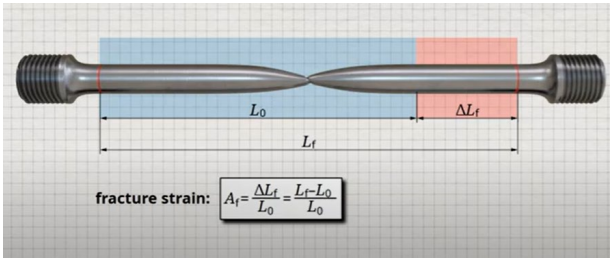
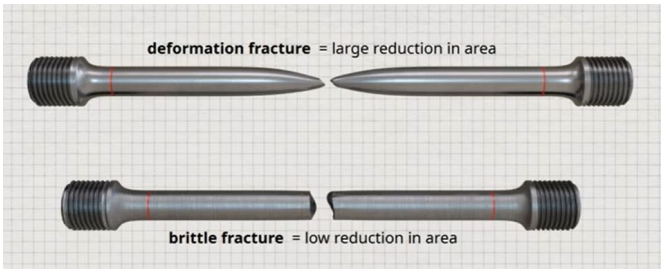
Plastic Deformation
Plastic deformation is a permanent deformation, which is non-recoverable. Plastic deformation gives rise to strain hardening, necking and fracture.

Strength Parameter	Meaning
Yield Strength (Yield Point)	• Stress at which material begins to deform plastically. • It starts to change shape permanently.
Ultimate Tensile Strength(UTS)	• Maximum stress before failure. • The maximum stress a material can withstand while being stretched or pulled before failing or breaking.
Elastic Modulus(Young's Modulus)  	• Measure of a material's stiffness or rigidity. • It represents the ratio of stress to strain within the material's elastic limit. ***Young's Modulus*** Steel iron = 210 kN/mm² Cast iron = 120 kN/mm² Aluminum = 70 kN/mm²
Yield-tensile Ratio(YTR)  A measurement of strain hardening up to the tensile strength $YTR = \frac{\text{Yield Strength}}{\text{UTS}}$	The **yield-tensile ratio** is a measurement of strain hardening up to the tensile strength. The yield strength ratio thus indicates how much tensile stress margin is available until material failure. Low Yield Tensile Ratio: Indicates high ductility and ability to absorb energy before failing. High Yield Tensile Ratio: Indicates low ductility and potential for brittle failure
Off Set -Yield Strength	**Off Set- Yield Strength** • The stress at which a certain permanent strain remains in the material.





- A line which is parallel to a specified modulus of elasticity line. 0.2% offset yield strength is calculated to distinguish the elastic and plastic region. It is defined as the stress value corresponding to the 0.2% plastic strain.

Deformation Parameters	Meaning
 e.g. Car metal bumper should design to highly deformable to absorbed as much energy in event of accident should have high fracture strain.	**Fracture strain** (elongation at break) measure the ductility and high tensile strength capacity .High value indicates high deformity.
 **Percent Elongation:** Pure Aluminum = 35% Aluminum Alloy = 11%.	**Deformation fracture** Deformation fracture is desirable because it has large reduction in area. The percentage elongation value is higher. **Brittle fracture** Brittle fracture is undesirable because it has limited deformation. The percentage elongation value is lower , exhibit minimum necking, break part occurs without warning in machine.

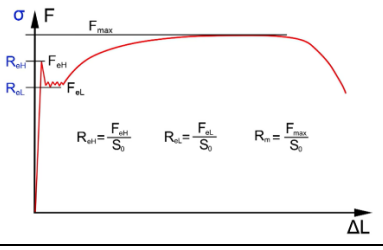
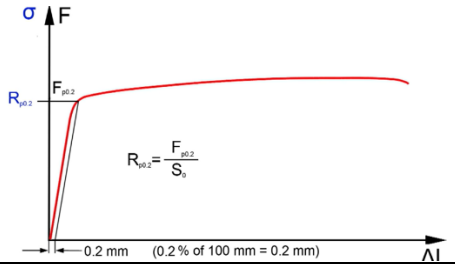
8.2 Example of Carbon Steel Grade

Properties	Carbon Steels	Alloy Steels	Stainless Steels	Tool Steels
Elastic Modulus (GPa)	190-210	190-210	190-210	190-210
Density(10000 kg/m3)	7.85	7.85	7.75-8.1	7.72-8.0
Tensile Strength(MPa)	276-1882	758-1882	515-827	640-2000
Yield Strength (MPa)	186-758	366-1793	207-552	380-440
Percent Elongation(%)	10-32	4-31	12-40	5-25
Hardness (Brinell 3000kg)	86-388	149-627	137-595	210-620

10 bar = 1 MPa =0.001 KN/mm2

Tensile Test

https://www.youtube.com/watch?v=D8U4G5kcpcM

Material with yield point phenomenon	Material without yield point phenomenon
Plain Carbon Steel	Aluminium alloy
Deformation fracture High percentage elongation	brittle fracture Low percentage elongation
	

Summary

The tensile test is testing methods to determine material properties:

<u>Strength Parameters</u>

- Yield strength
- Ultimate Tensile strength(UTS)
- Elastic Modulus (Young's modulus)
- Yield-tensile Ratio(YTR)
- Off Set-Yield Strength

<u>Deformation Parameters</u>

- Fracture strain(elongation at break)
- Deformation fracture
- Brittle fracture

9.0_CREATE 3D MODEL OF COMPONENTS

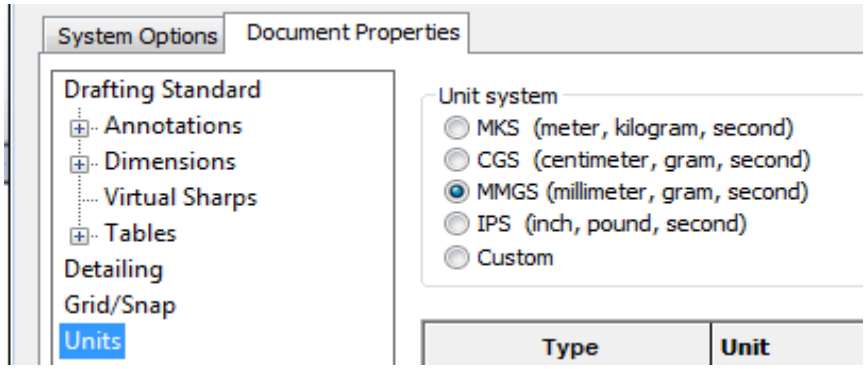
9.1 Advantages of using CAD system

- Precise.
- Computerized model to scale.
- Can resize easily.
- Reduced cost of revision
- Increased drawing value
- Reduced storage and creation of a standard filing system
- Ability to maintain consistent level of quality for projects.

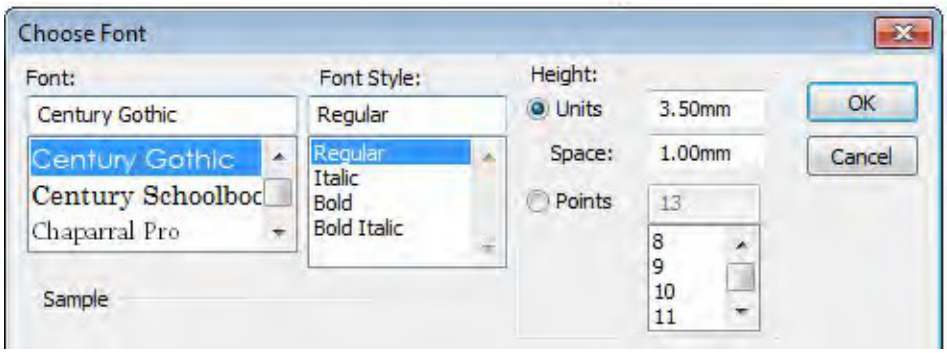
9.2 Setting of Document properties

Document properties can access through the Tools > Options menu settings for the model or click from icon .It allow you to customize your work environment.

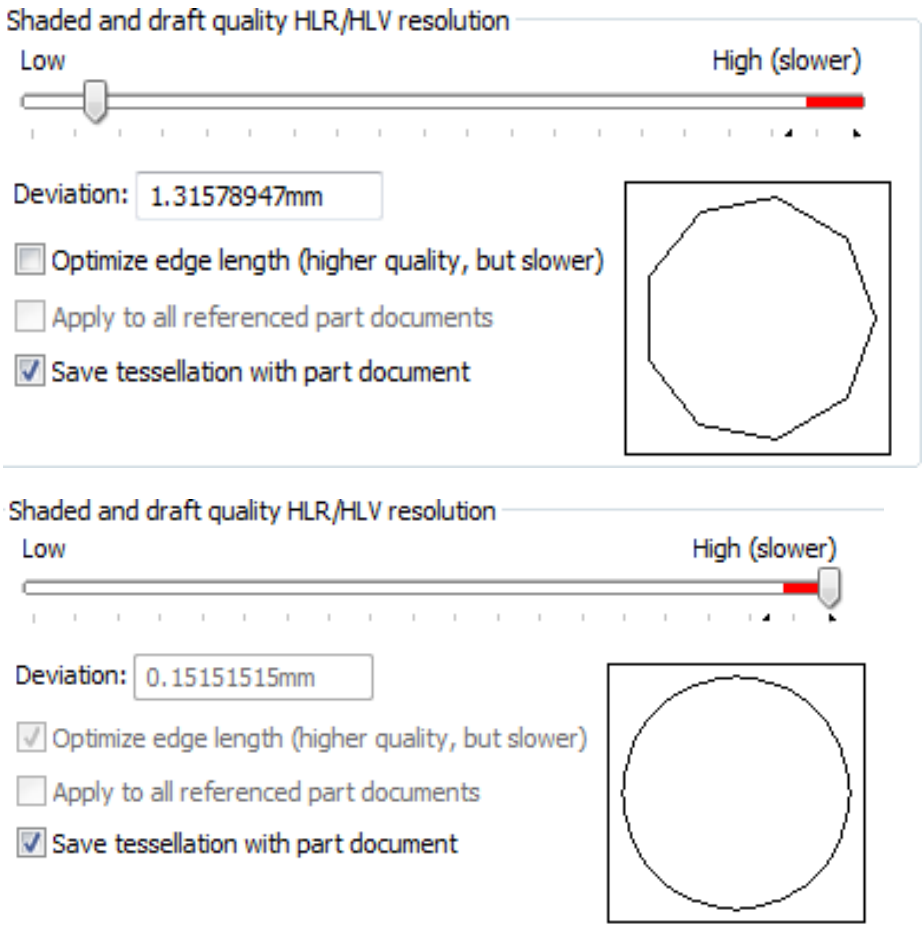
9.2.1.1Unit – English (Inches) or Metric (Meter, centimeter & millimeter)



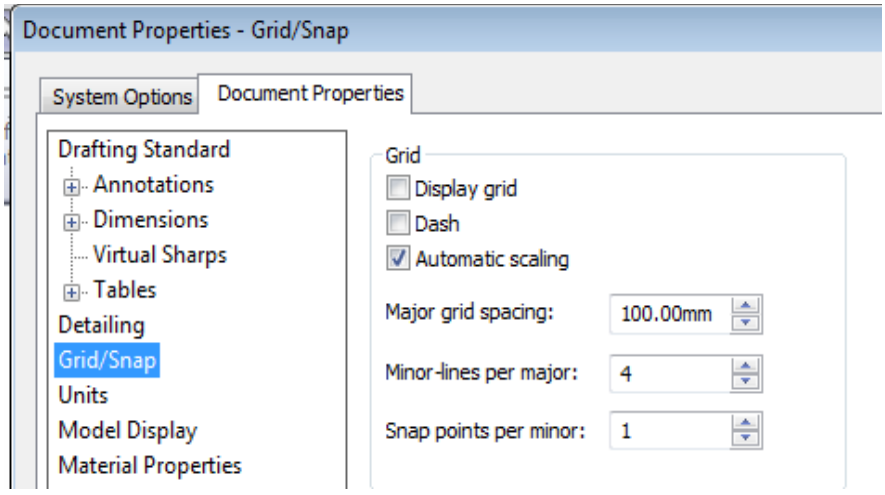
9.2.2 Annotation Font: Control all the text font style and height.



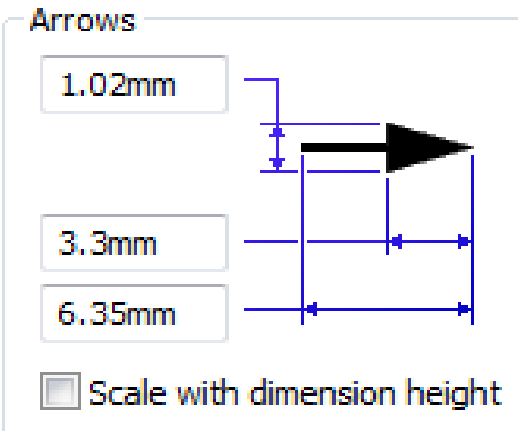
- Image quality


Shaded and draft quality HLR/HLV resolution
Low
High (slower)
Deviation: 1.31578947mm
Optimize edge length (higher quality, but slower)
Apply to all referenced part documents
Save tessellation with part document
Shaded and draft quality HLR/HLV resolution
Low
High (slower)
Deviation: 0.15151515mm
Optimize edge length (higher quality, but slower)
Apply to all referenced part documents
Save tessellation with part document

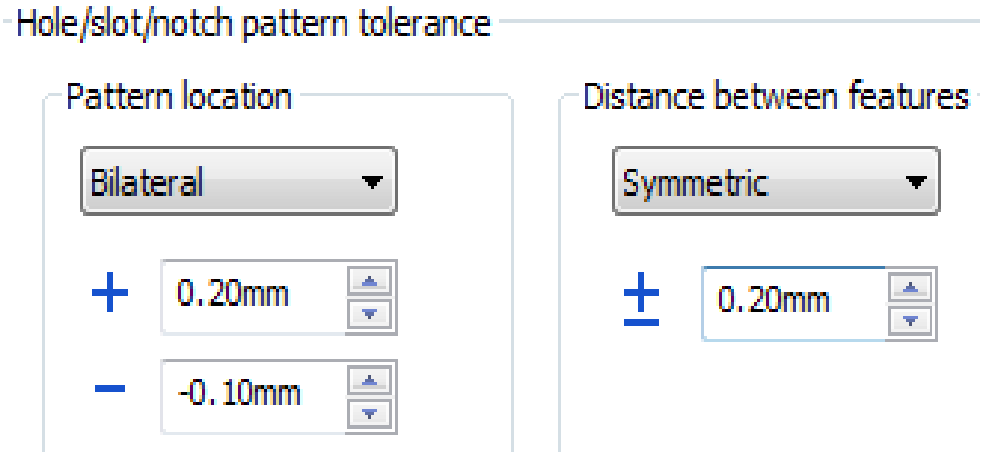
- Grid/Snap


Document Properties - Grid/Snap
System Options
Document Properties
Drafting Standard
Annotations
Dimensions
Virtual Sharps
Tables
Detailing
Grid/Snap
Units
Model Display
Material Properties
Grid
Display grid
Dash
Automatic scaling
Major grid spacing: 100.00mm
Minor-lines per major: 4
Snap points per minor: 1

- Arrows Style and Size



- Types of Tolerances



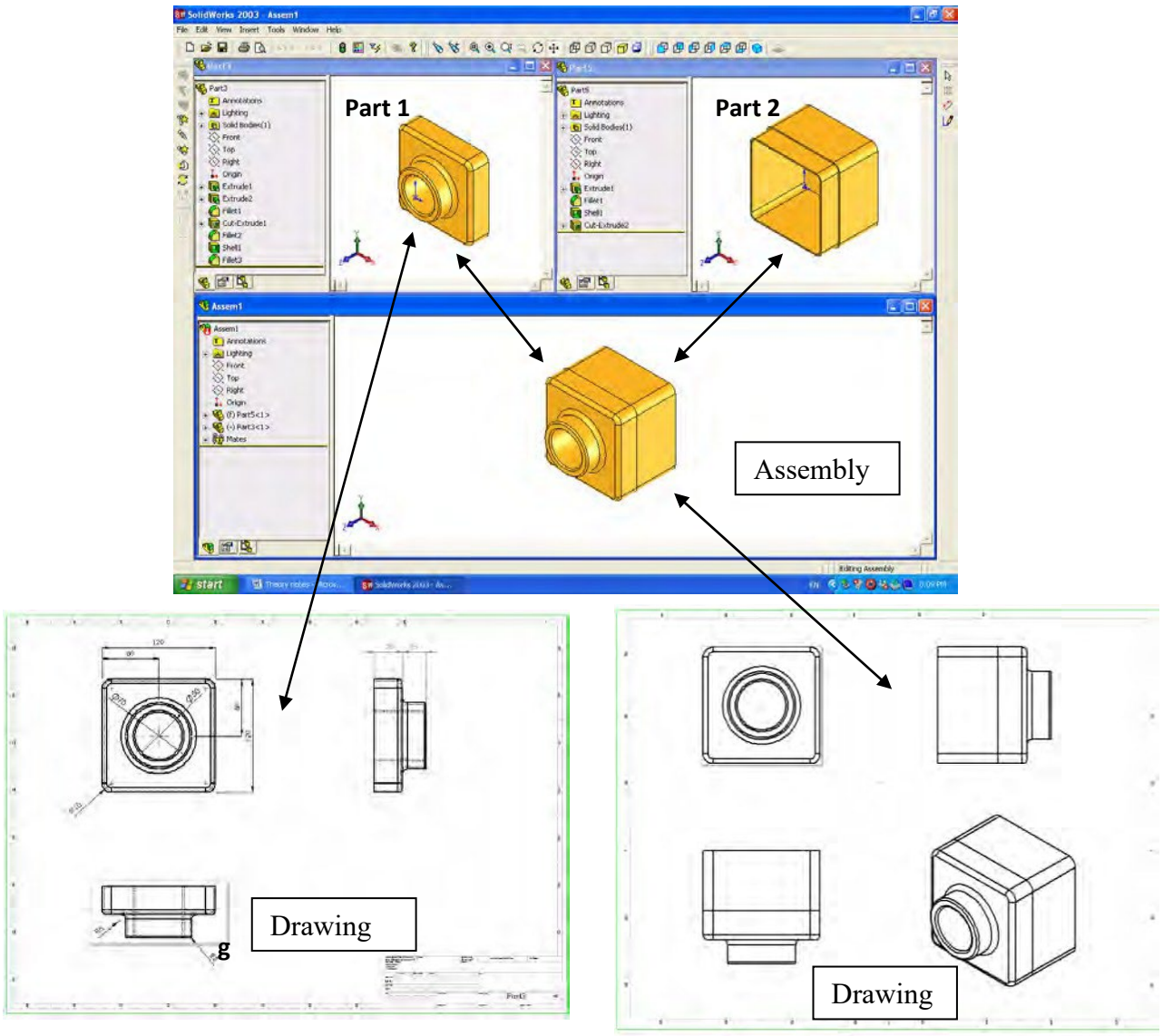
9.3 Procedures and techniques used to produce 3D models

The SolidWorks model is made up of:

9.3.1 Parts

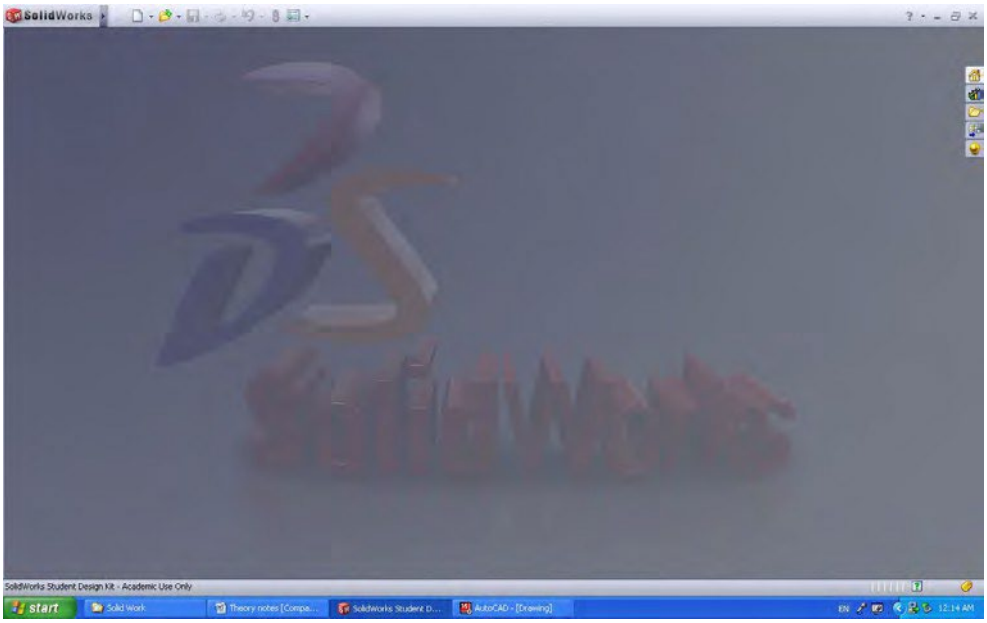
9.3.2 Assemblies

9.3.3 Drawings.



9.4 To start SolidWorks

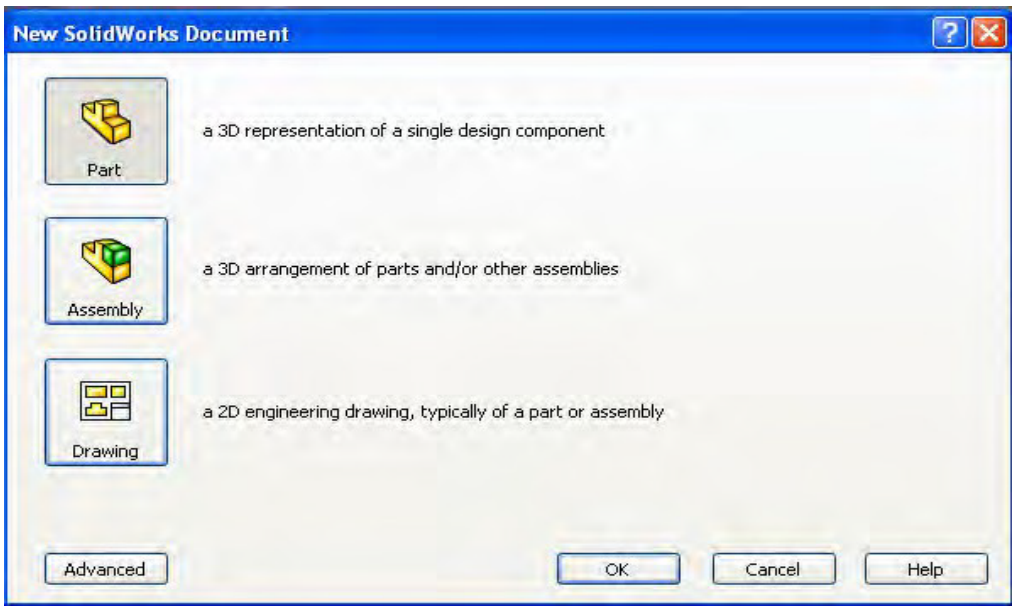
Double-click on the desktop shortcut  , Solidworks window appeared.



Creating New Files Using Templates

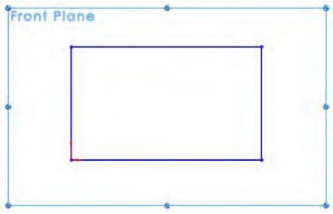
9.4.1 Click **New**  on the Standard toolbar.

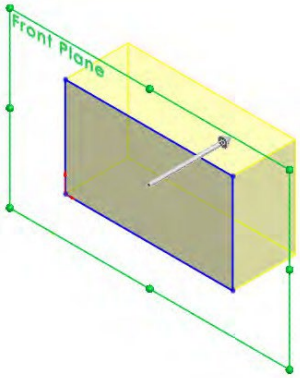
9.4.2 Select a document template: Part, Assembly or Drawing

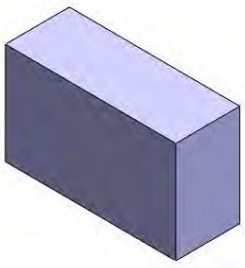


To Create an 3D model:

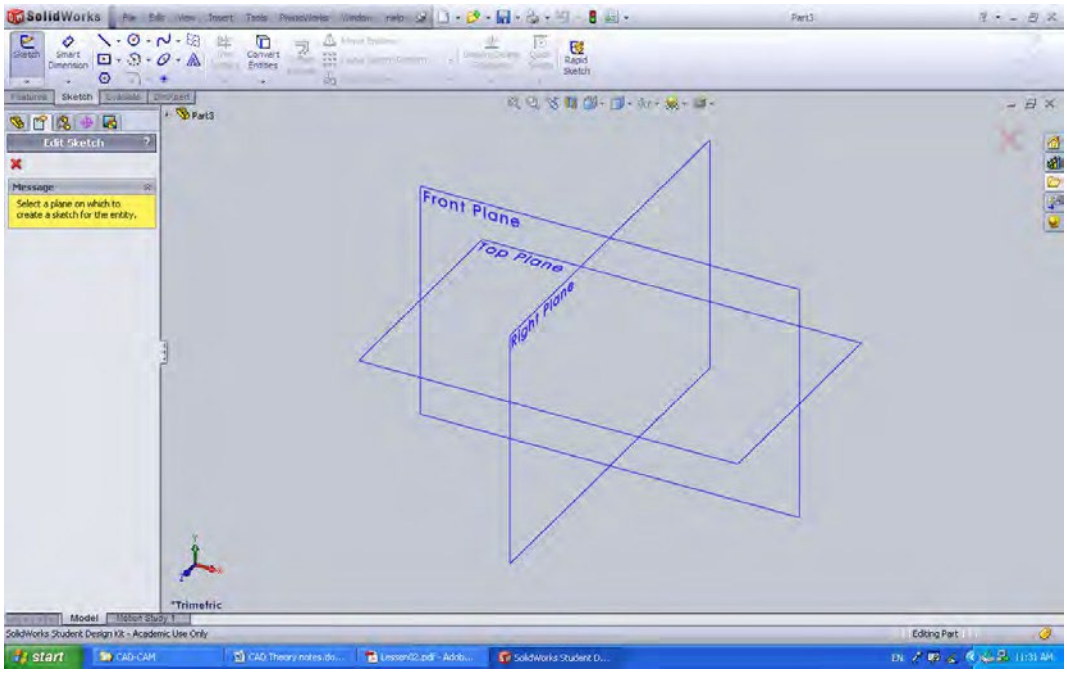
- Select Part template.
- Select a sketch plane.
- Sketch a 2D profile.
- Extrude the sketch perpendicular to sketch plane.



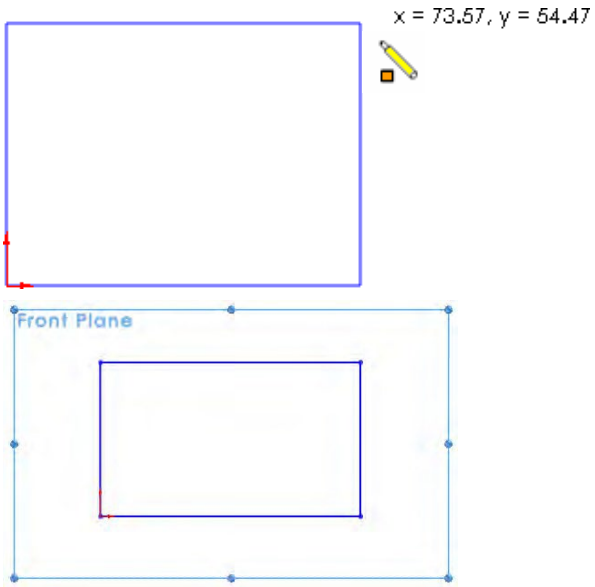




Select a **sketch plane**. The default sketch plane is **Front**.



- Click **Sketch**  on the Sketch toolbar.
- Click Rectangle  on the Sketch Tools toolbar.
- Move the pointer to the Sketch Origin.
- Click the left mouse button.
- Drag the pointer up and to the right.
- Click the left mouse button again.

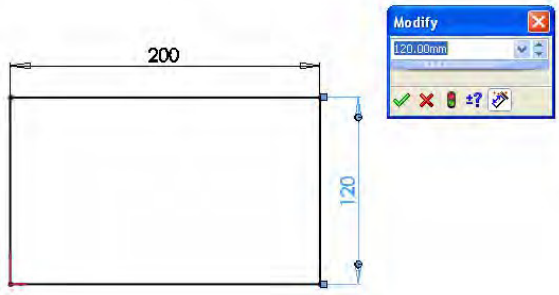


Adding Dimensions

Dimensions specify the size of the model.

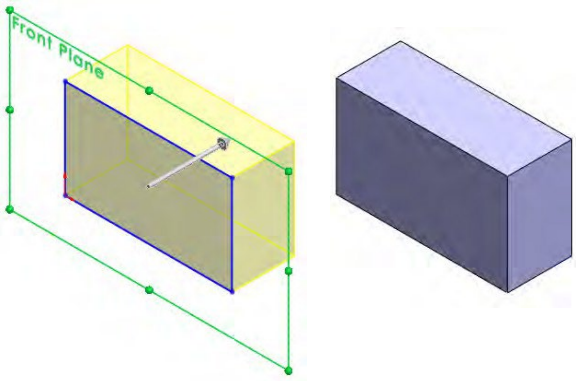
To create a dimension:

- Click Dimension  on the Sketch Relations toolbar.
- Click the 2D geometry.
- Click the text location.
- Enter the dimension value.



Create Solid

Click Feature, then select Extruded Boss/Base  and key in the width you wanted to extrude, 3D model formed.



9.5 Purpose of a coordinate system

SolidWorks uses a **system** of **coordinate systems** with origins. A part document contains an original origin. Whenever you select a plane or face and open a sketch, an origin is created in alignment with the plane or face.

An origin can be used as an anchor for the sketch entities, and it helps orient perspective of the axes. A three-dimensional reference triad orients you to the X, Y, and Z directions in part and assembly documents.

 Sketch origin (one for each new sketch)

 Reference triad in part and assembly documents

You can define a coordinate system for a part or assembly.

Coordinate systems are useful:

- With the **Measure** and **Mass Properties** tools
- When applying assembly mates.

9.6 Predefined 3D objects commonly available in CAD system which are use to develop surface models and solid models.

Sketched Feature : use to develop surface models

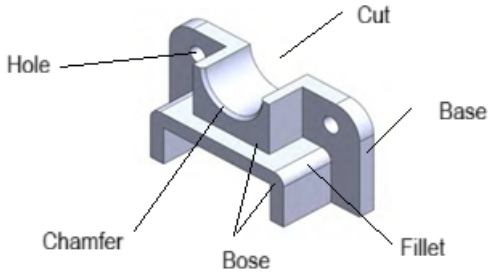
- Shape features have sketches.
- Sketched features are built from 2D profiles.

Operation Features : Use to develop solid models.

- Operation features do not have sketches.
- Applied directly to the workpiece by selecting edges or faces.

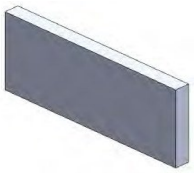
Feature

- Features are the building blocks of the part.
- Features are the shapes and operations that construct the part.



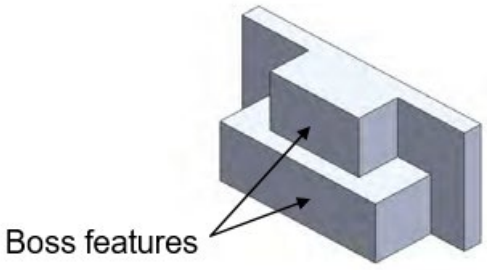
i. Base feature : Shape features have sketches

- First feature in part.
- Created from a 2D sketch.
- Forms the workpiece to which other features are added.



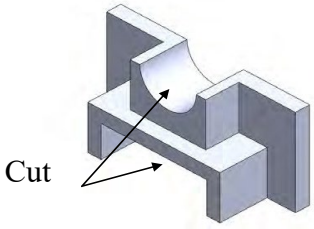
ii. Boss feature : Shape features have sketches

- Adds material to part.
- Created from 2D sketch.



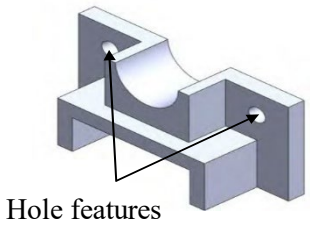
iii. Cut feature : Shape features have sketches

- Removes material from part.
- Created from a 2D sketch**.**



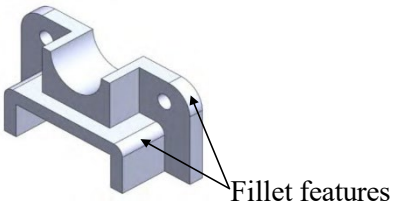
iv. Hole feature : Operation features do not have sketches

- Removes material.
- Works like a more intelligent cut feature.
- Corresponds to process such as counter-sink, thread, counterbore.



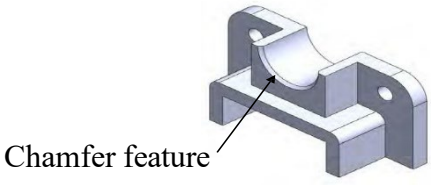
v. Fillet feature : Operation features do not have sketches

- Used to round off sharp edges.
- Can remove or add material. (outside edge convex fillet removes material.)



vi.Chamfer feature : Operation features do not have sketches

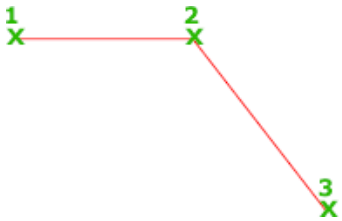
- Similar to a fillet.
- Bevels an edge rather than rounding it
- Can remove or add material.



9.7 Basic CAD software commands

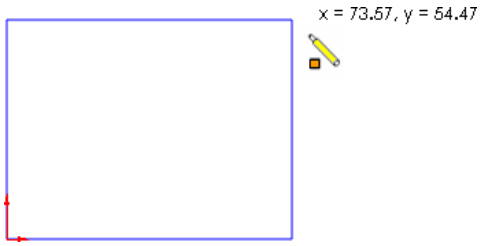
Line 

With Line, you can create a series of contiguous line segments. Each segment is a line object that can be edited separately.



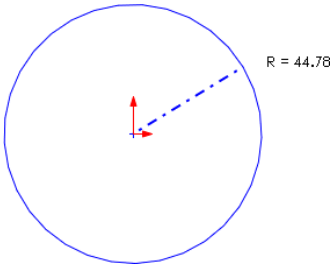
Rectangle 

Creates a rectangle using the area and either a length or a width.



Circle 

Draws a circle based on a center point and a diameter or a radius.
Defines the radius of the circle. Enter a value, or specify a point

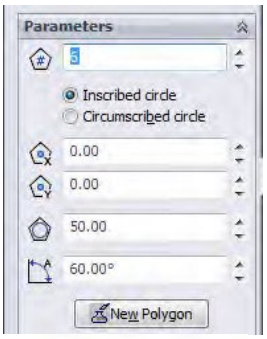


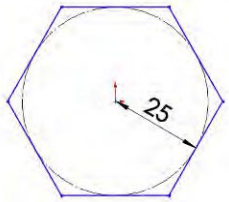
Polygon 

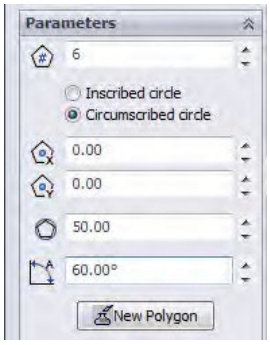
Defines the center of the polygon.
Enter an option [Inscribed in circle/Circumscribed about circle]

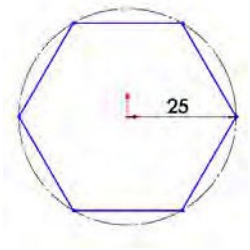
Inscribed in Circle	Circumscribed circle
Specifies the distance from the	Specifies the radius of a

circle on center of the polygon to the mid which all vertices of the polygon lie. points of the edges of the polygon.





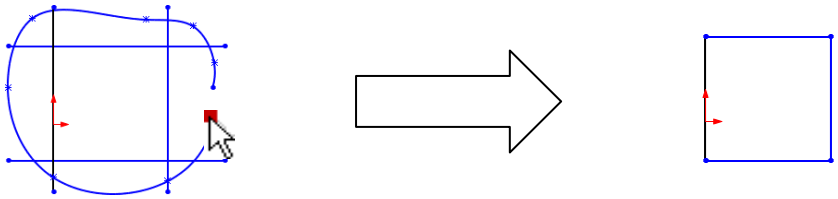




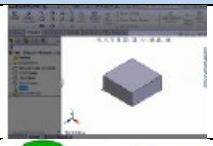
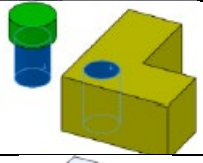
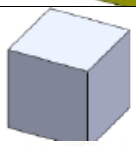
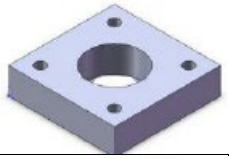
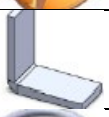
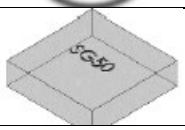
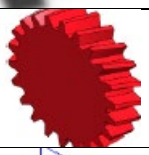
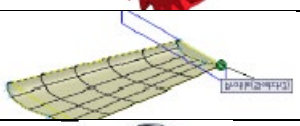
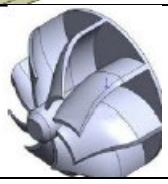
Trim

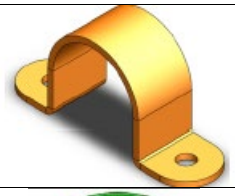
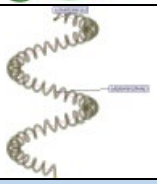
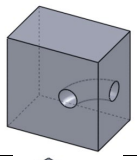
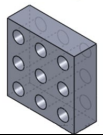
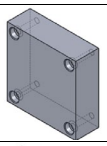
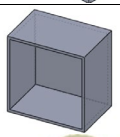
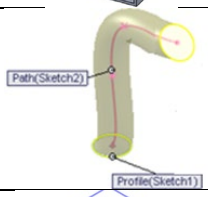
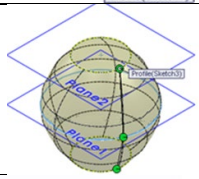
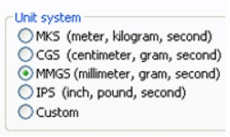
Select the trim type based on the entities you want to trim or extend. All trim types are available with 2D sketches and 2D sketches on 3D planes

- Click Trim Entities (Sketch toolbar) or Tool > Sketch Tool > Trim
- In the property Manger, under options, select power trim
- Click in the graphics area next to the first entity, and drag across the sketch entity to trim. The pointer change to as it crosses and trims the sketch entity. A trail is created along the trim path
- Continue to hold down the pointer and drag across each sketch entity you want to trim.
- Release the pointer when finished trimming the sketch then click



10_SOLIDWORKS CAD TUTORIAL

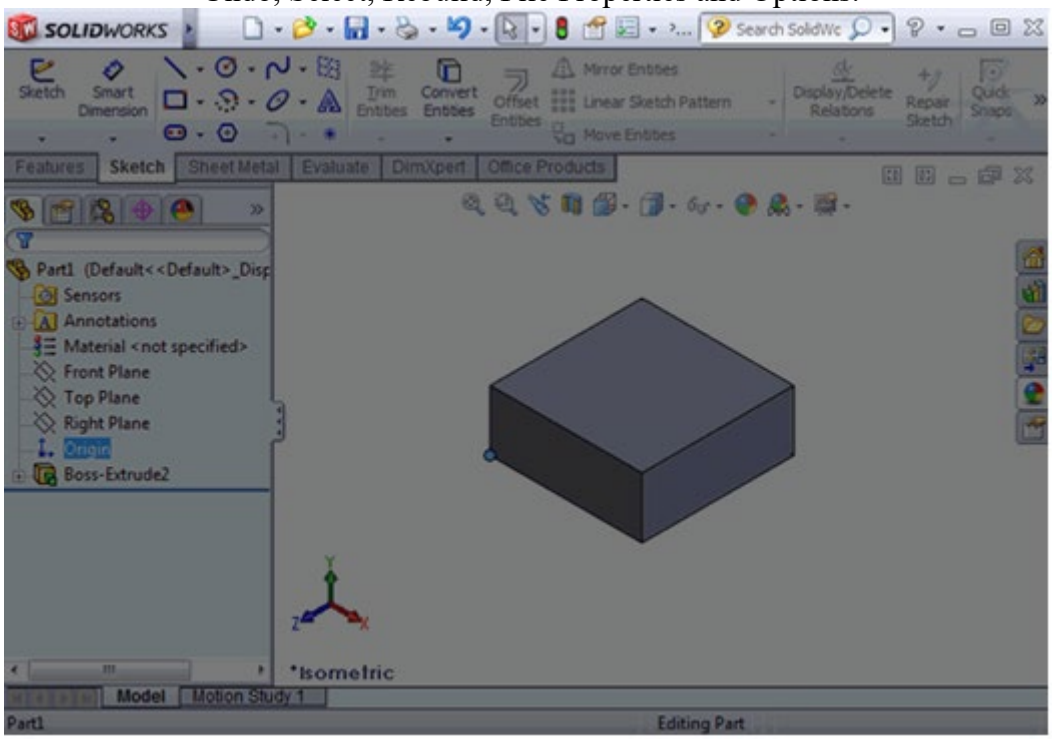
Tutorial	*Title (Part 1)*	*Model*	*Page*
Tutorial 1	SolidWorks User Interface		
Tutorial 2	Introduction to SolidWorks		
Tutorial 3	How to create simple box		
Tutorial 4	How to create simple plate		
Tutorial 5	How to create Allen key		
Tutorial 6	How to create 17" car wheel		
Tutorial 7	How to create simple sheet metal bend		
Tutorial 8	How to create spring		
Tutorial 9	How to engrave text		
Tutorial 10	How to create hexagonal head bolt		
Tutorial 11	How to create helical gear		
Tutorial 12	How to create aero plane wings		
Tutorial 13	How to create turbo fins		

Tutorial 14	How to create U bracket		
Tutorial 15	How to create bottle cap		
Tutorial 16	How to twist phone cord		
Tutorial	**Title (Part 2)**	**Model**	
Tutorial 17	How to use Revolved Boss Base		
Tutorial 18	How to use Revolved Cut		
Tutorial 19	How to use Linear Pattern		
Tutorial 20	How to use Scale	Scale down to 1/2 .	
Tutorial 21	How to use Hole Wizard		
Tutorial 22	How to use Shell		
Tutorial 23	How to use Swept Boss/Base		
Tutorial 24	How to use Lofted Boss/Base		
Tutorial 25	How to change to metric units		

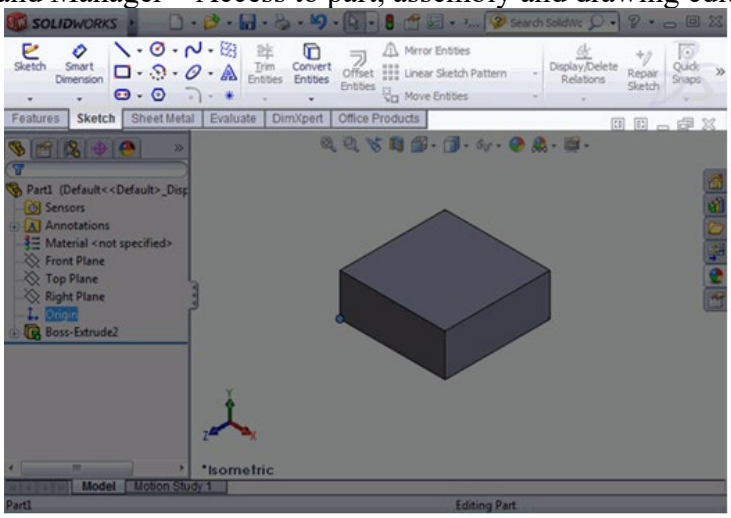
1. SolidWorks User Interface

SolidWorks User Interface is pretty simple and straight forward. There is 6 main area of interface you normally work with.

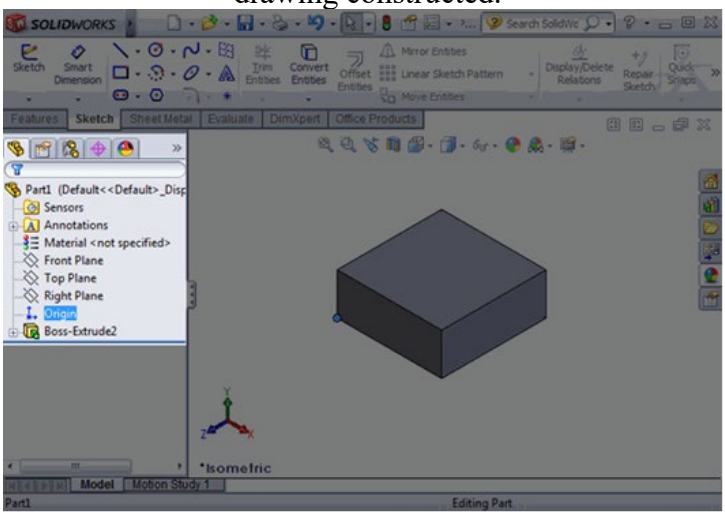
1) Menu Bar – Top most of the application, executing New File, Open File, Save, Print, Undo, Select, Rebuild, File Properties and Options.



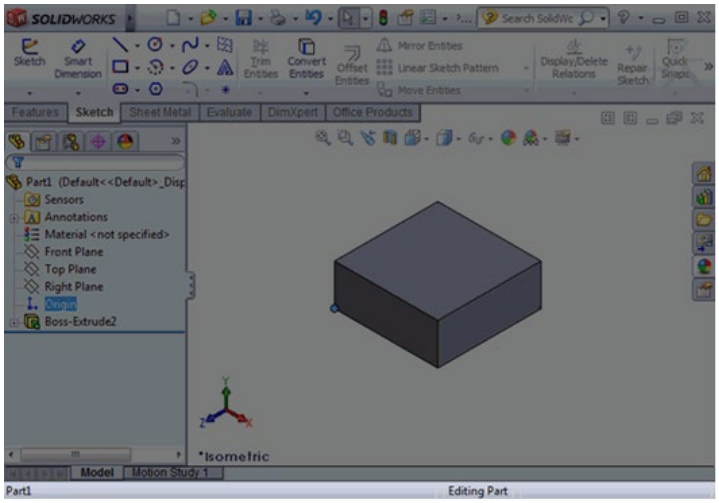
2) Command Manager – Access to part, assembly and drawing editting tools.



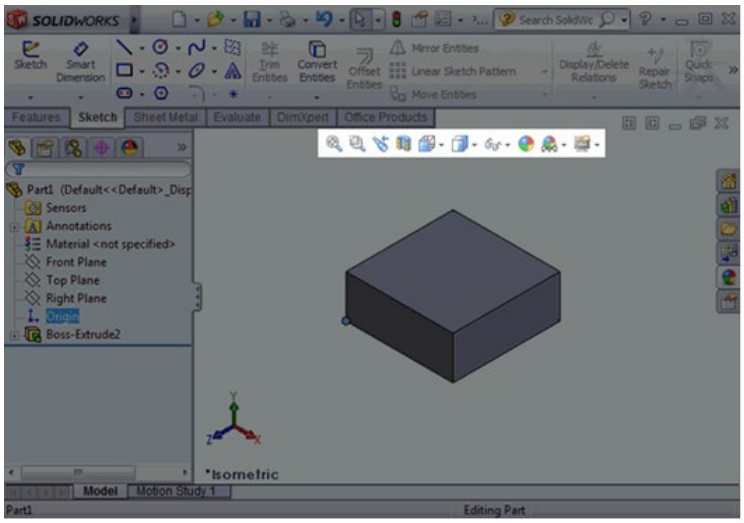
3) Feature Manager design tree – Outline overview how your part, assembly and drawing constructed.



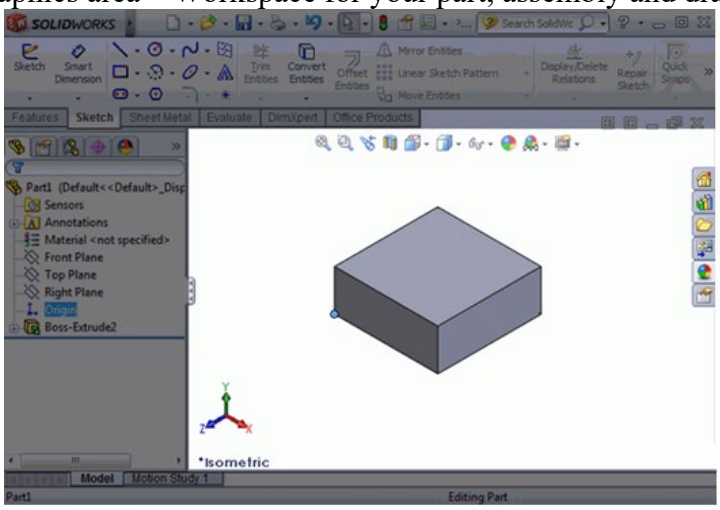
4) Status bar – Provide an information about your part, assembly and drawing.



5) Head up view toolbar – View tools such as zoom, pan, zoom plane and section view.



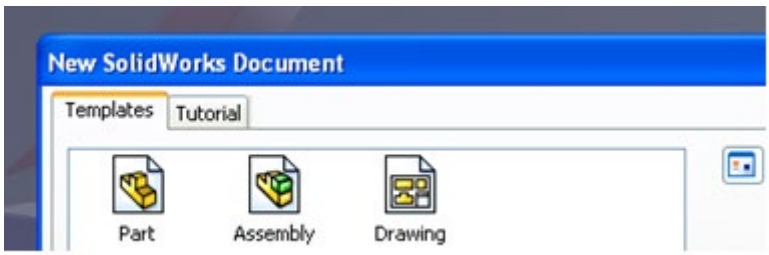
6) Graphics area – Workspace for your part, assembly and drawing.



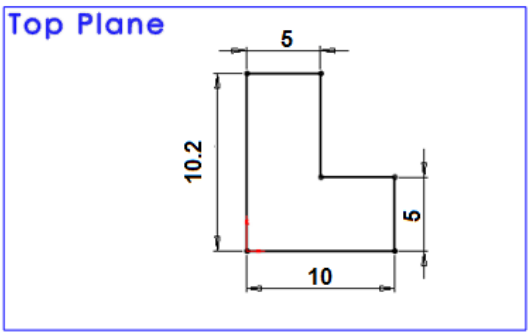
2. Introduction to SolidWorks

SolidWorks Overview

Solidworks main idea is user to create drawing directly in 3D or solid form. From this solid user can assemble it directly on their workstation checking clashes and functionality of it. Creating drawing is pretty easy just drag and drop the solid to drawing block.

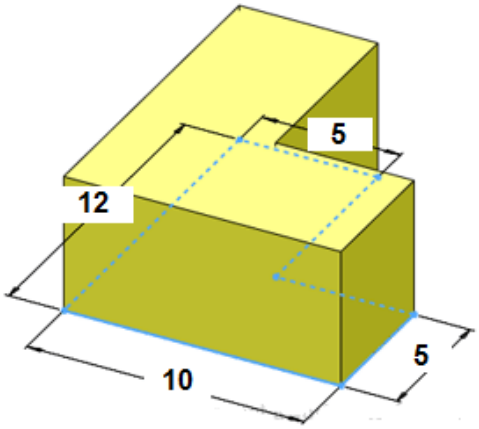


Part

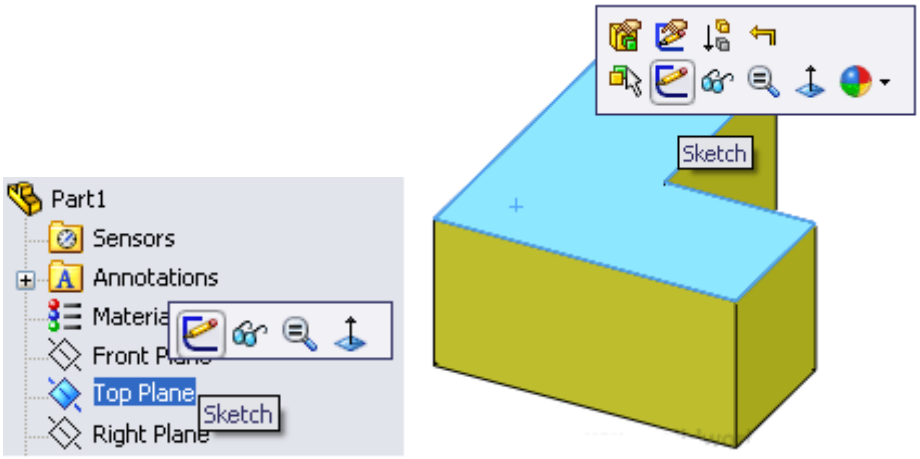


Part is created by sketch.

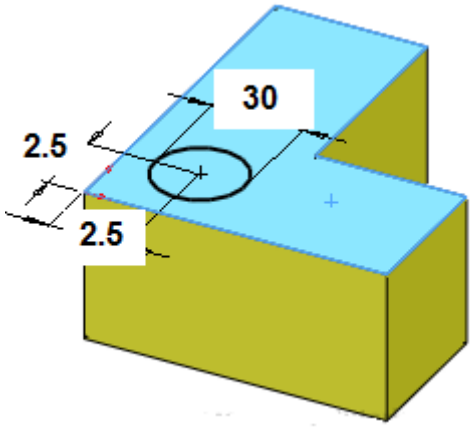
Sketch is the base to define your part, form and features.



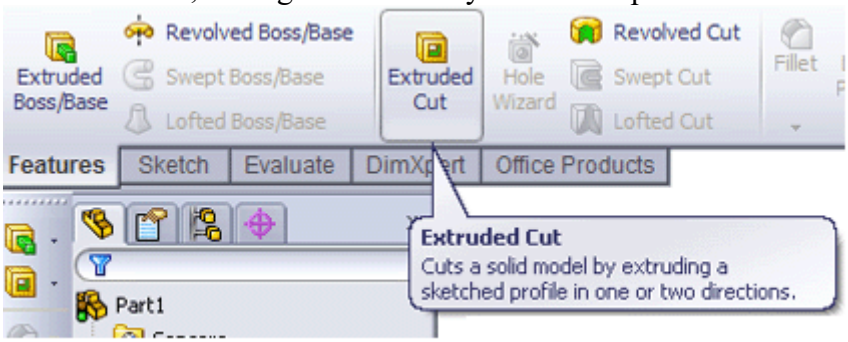
Before you start creating sketches you must select plane or face where the sketch will be place on.

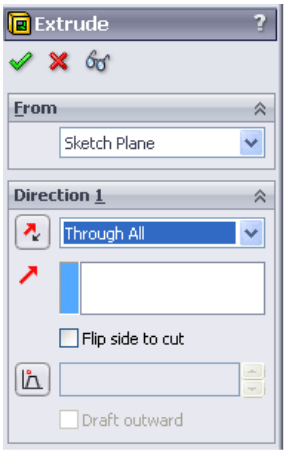


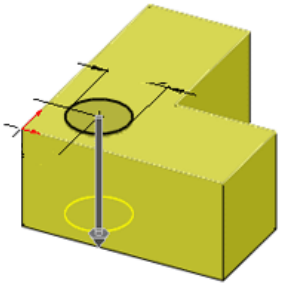
After select plane or face the sketch will be, sketch on it!



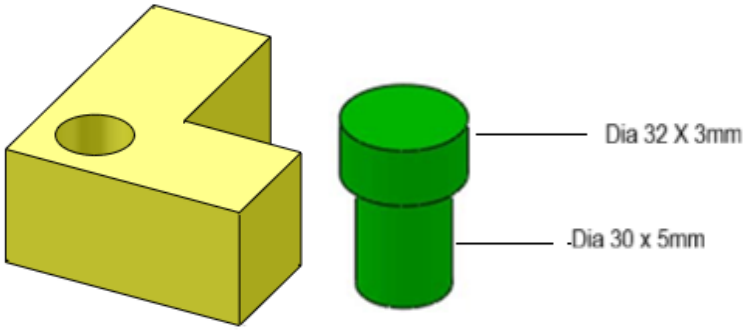
When you done with sketch, adding features it is your next step. Select Feature>Extruded Cut





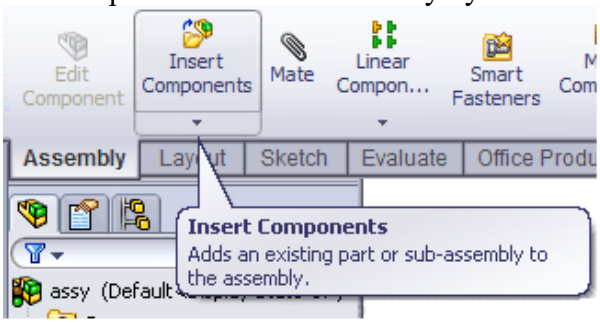


Select Through All and OK.

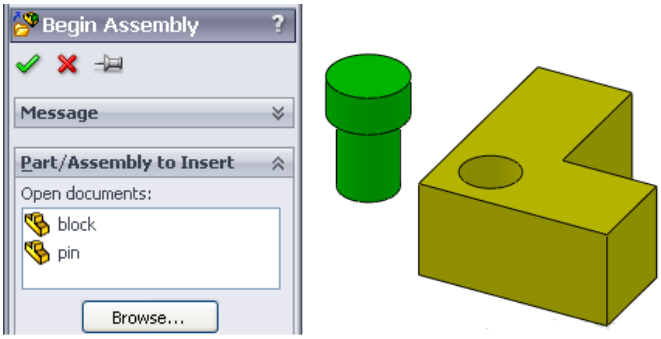


Assembly

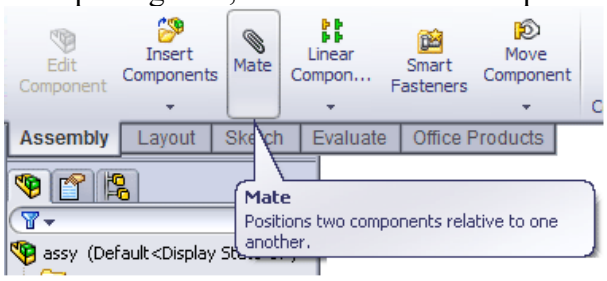
Assembly is how all parts works together in assembly, checking for clashes and it functionality. First all parts inserted in assembly by Insert Component tool.

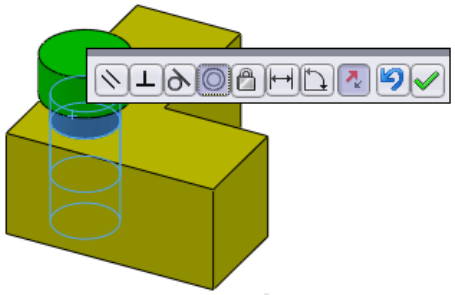


When all parts inserted into workspace, Mate is command to define how parts mate with each other.



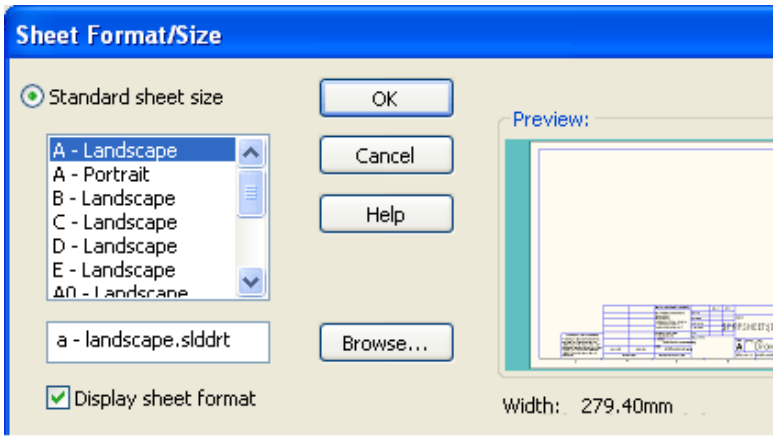
Let’s mate this block and pin together, click Mate and select pin face and hole face, OK.



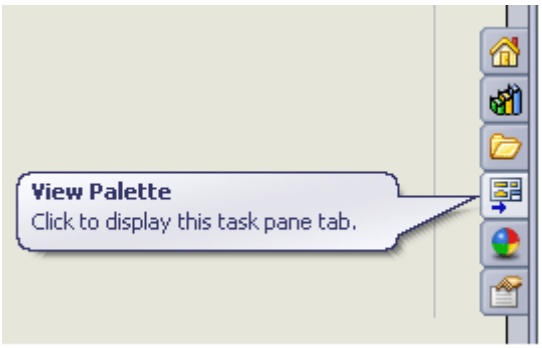


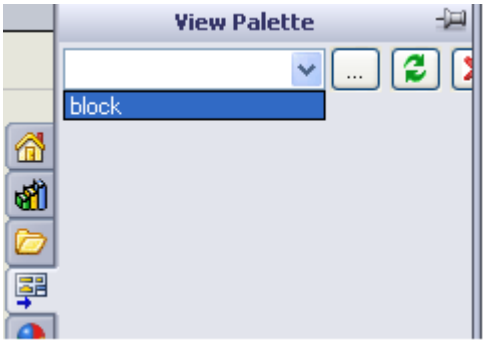
Drawing

Drawing is use for detailing part by adding dimension to it. To create a drawing first you need to select drawing block.

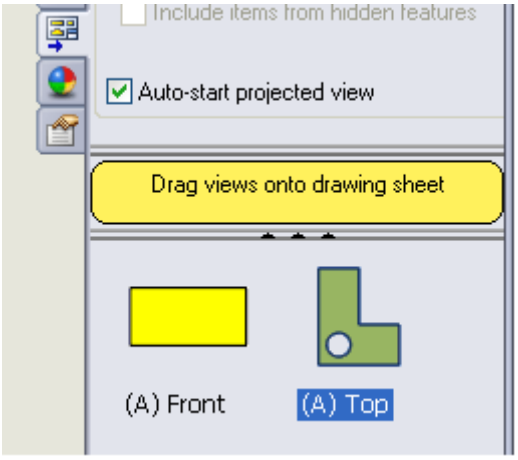


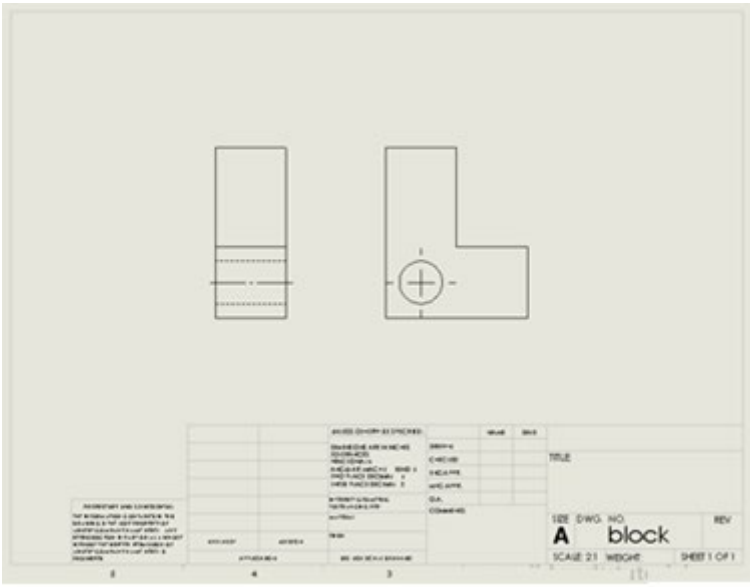
When block inserted, select click view palette to add drawing view.



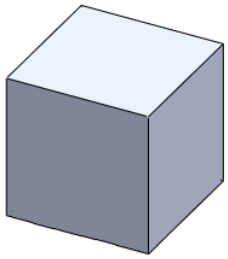
Choose the part you wish to make drawing. 

Now just drag and drop the part view on drawing block and add dimensions.



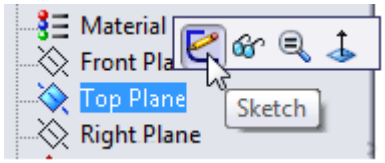


3. How to create simple box

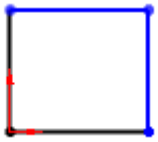


1. Click **New** , Click **Part** and **OK**.

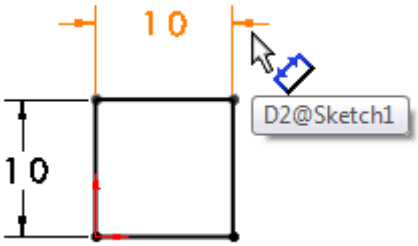
2. Click on **Top Plane** and click **Sketch**.



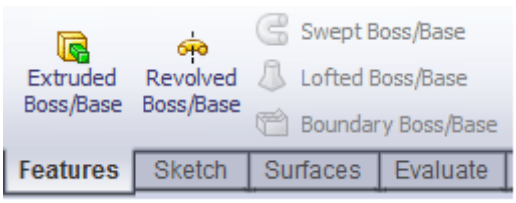
3. Click **Rectangle** , sketch a rectangle start from origin.



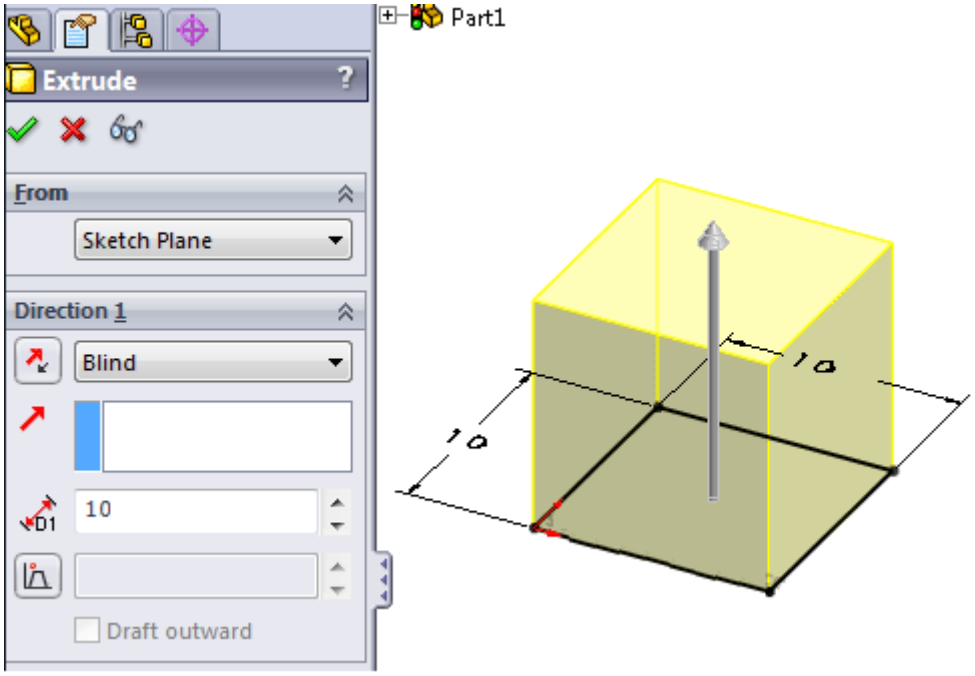
4. Click **Smart Dimension** , click side edge and click top edge to dimension it as **10mm x 10mm**.



5. Click **Features>Extruded Boss/Base**

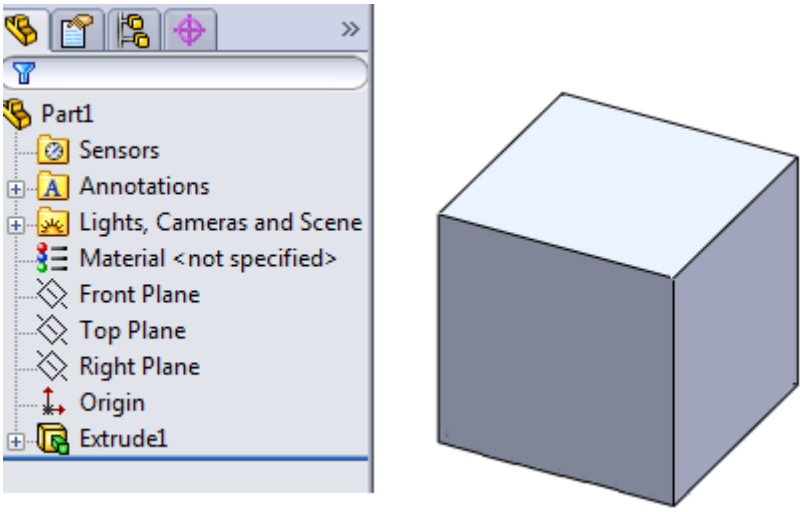


set **D1** as **10mm**

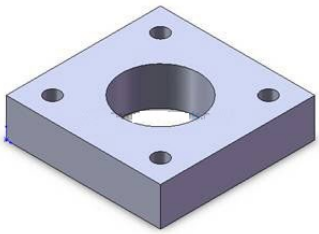


and click  .

6. It’s done.



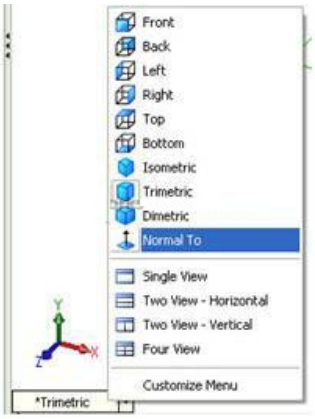
4. How to create simple plate



1. Click New  (File>New) , click Part  , OK.

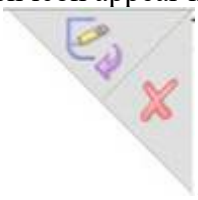
2. Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK.

3. Select Top Plane , from lower left menu select Normal To.



4. Click Sketch 

in Command Manager, click Rectangle 

. As you can see on upper right corner sketch icon appear indicate that you're on sketch mode 
.

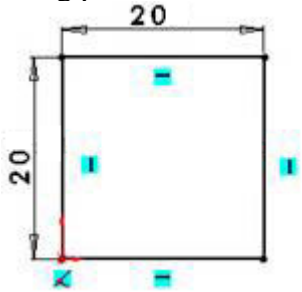
5. Pick Origin point as starting point, drag to right hand side


no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.

Note: There is two type line generated by your sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined.

6. Define sketch with dimension. Click Smart Dimension 

, and start dimensioning pick vertical line and set to 20mm , pick horizontal line and set to 20mm.



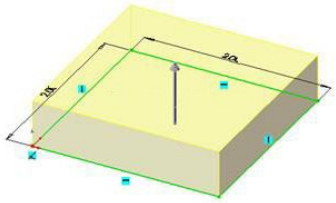
. Press keyboard ESC to end smart dimension

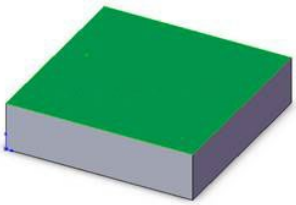
7. Build feature from sketch, click Features 

and activate features menu. Click Extruded Boss/Base 

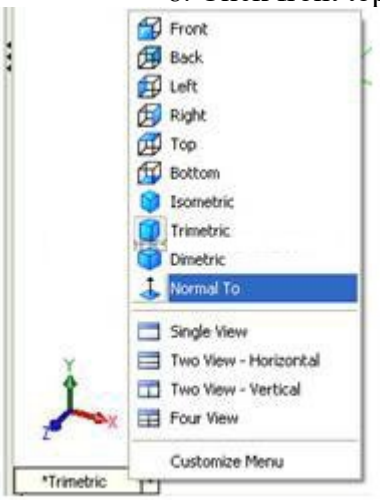
and set D1 to **5mm** 

and  .





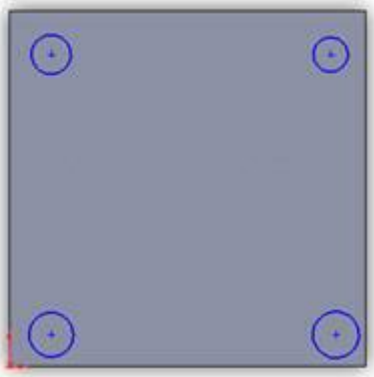
8. Click front top face , click Normal To



. Activate sketch menu by click Sketch 

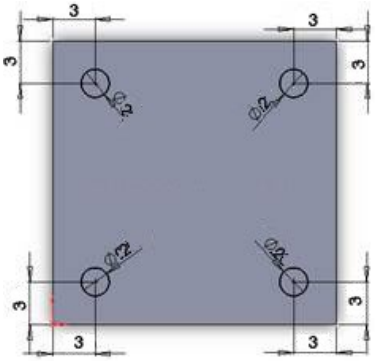
and select Circle 

. Sketch 4 circle at four edges.



9. Define new circle sketch, click Smart Dimension 

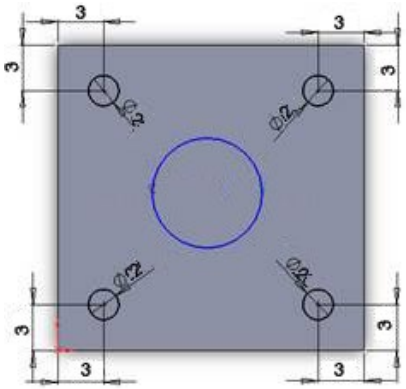
, set diameter circle to **2mm** .

Select distance for edge set to **3mm** 

.

10. Click Circle 

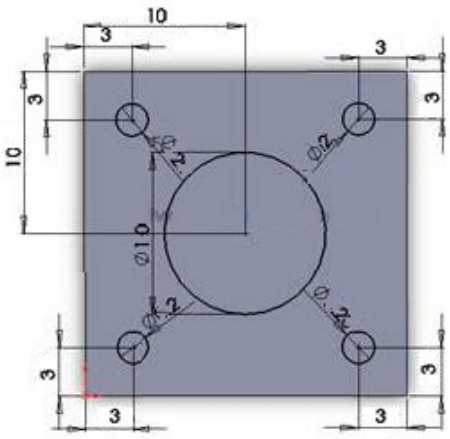
and sketch one circle at center

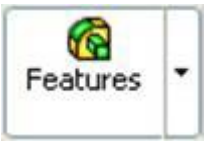


.

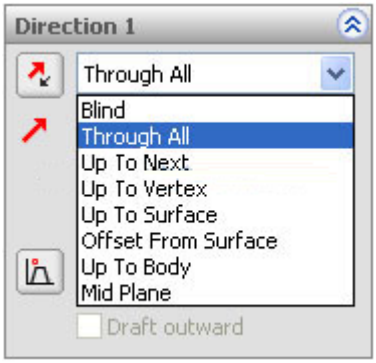
11. Define new circle sketch, click Smart Dimension 

, set diameter circle to

10mm . Select distance for edge set to **10mm.** 

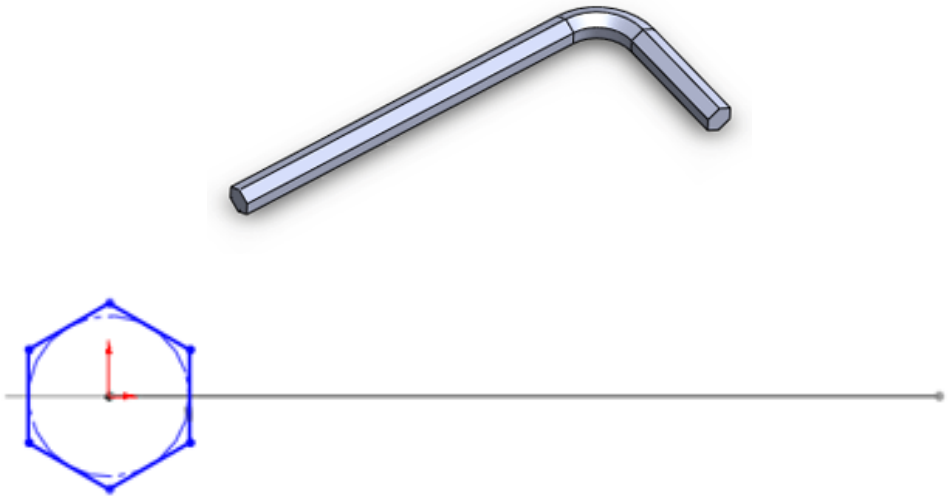
12. For cut click Features 

, click Extruded Cut 

, under Direction 1, Through All 

and 
.Done

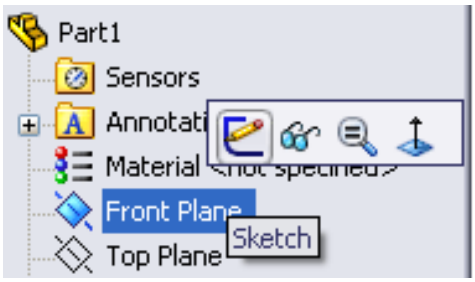
5. How to create Allen key



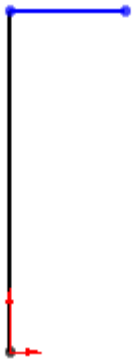
In this solidworks tutorial, you will create simple allen key.

1. Click **New**.  Click **Part**,  **OK**.

2. Click **Front Plane** and click on **Sketch**.

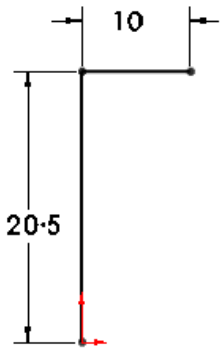


3. Click **Line**, skecth a L shape.

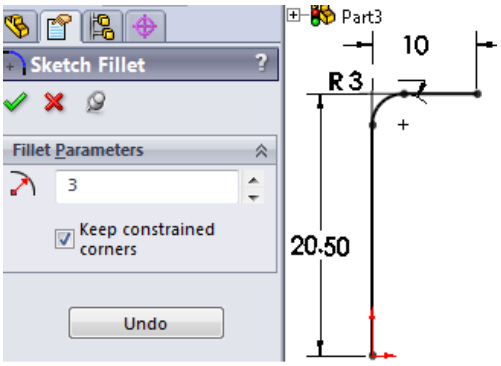


4. Click **Smart Dimension**, 

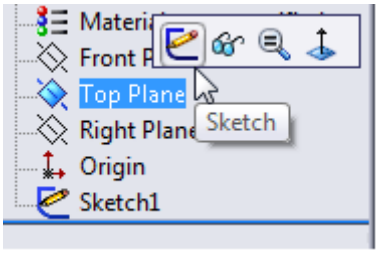
and dimension sketch as **25mm** and **10mm**.



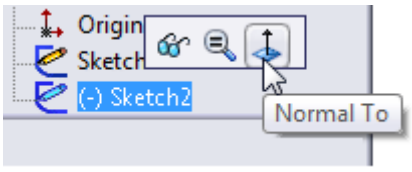
5. Click **Sketch Fillet**,  add **3mm** fillet at L corner.



6. **Exit sketch**,  click on **Top Plane** and click **Sketch**.



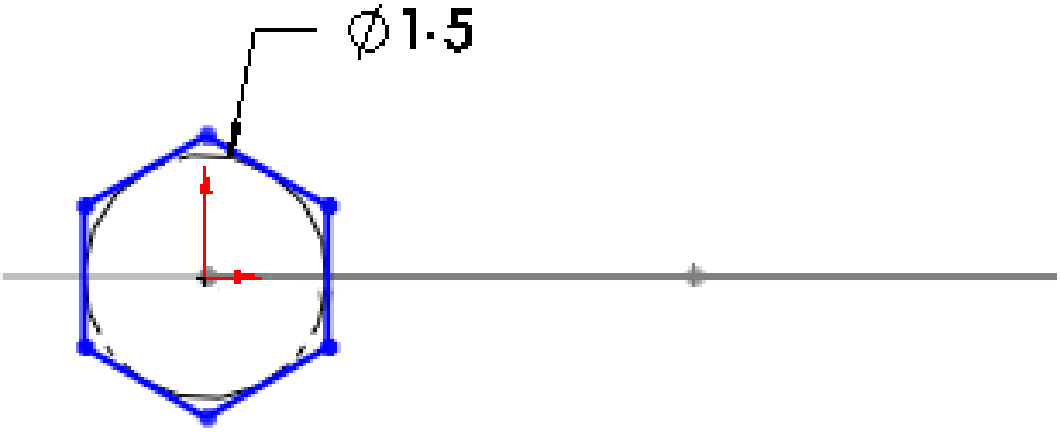
7. Click on **Sketch2** and click **Normal To**.

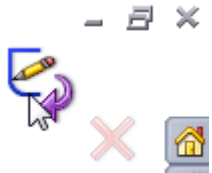


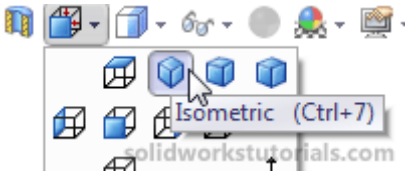
8. Click Polygon,  sketch a polygon at origin.

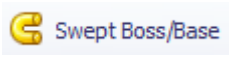
9. Click **Smart Dimension**, 

and dimension sketch diameter to **1.5mm**.

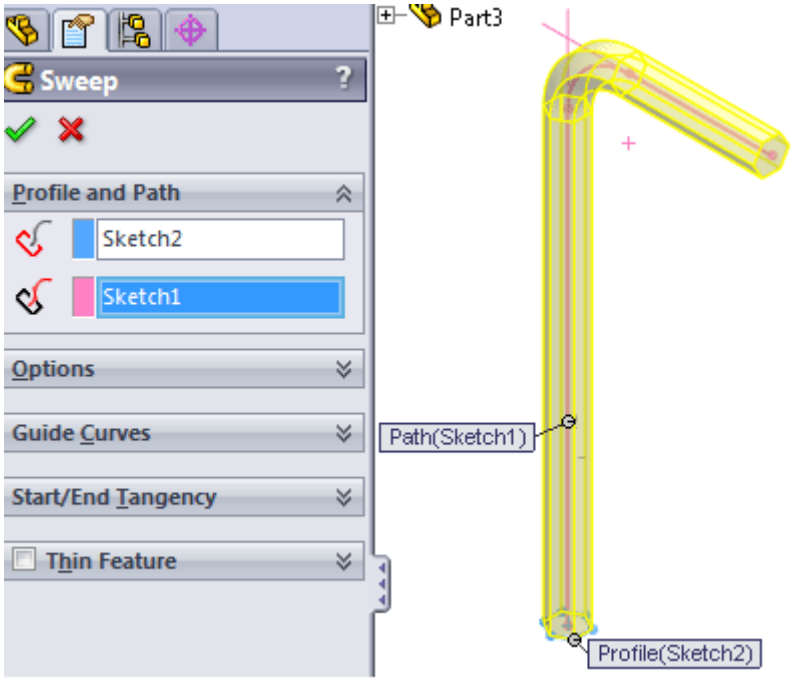


10. **Exit sketch**,  click on **Isometric** view.



11. Click **Features>Swept Boss/Base**, 

for profile click on **Sketch**2 and for path click on **Sketch1** and OK.

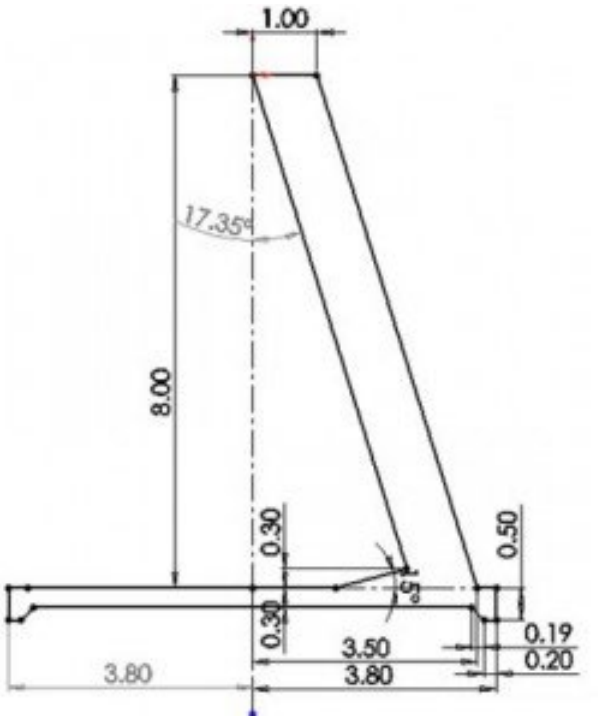


You’re done!.

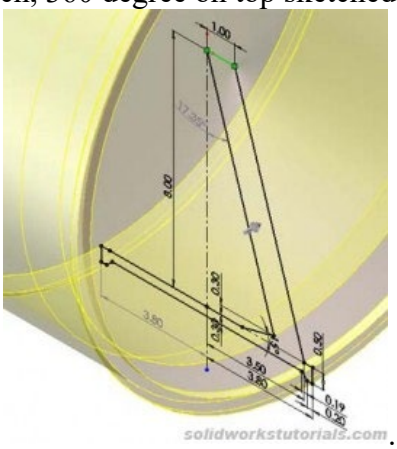
6. How to create 17 inch car wheel

Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK.

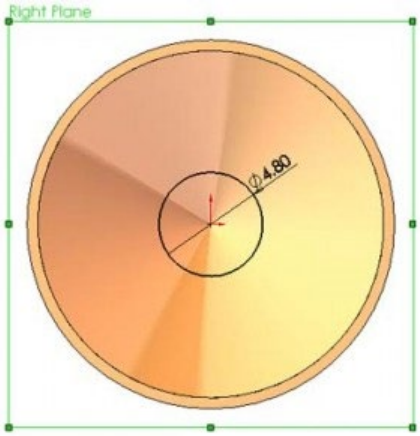
1. Create a skecth as show on Front Plane.



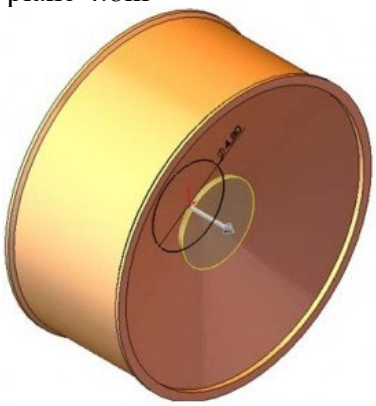
2. Revolve  sketch, 360 degree on top sketched line 



. OK.

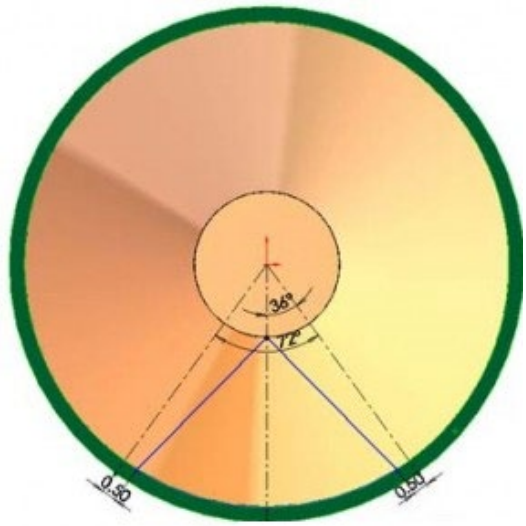


3. Create circle skecth, on right plane 4.8in , extrude 

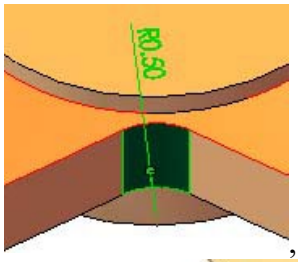


2in OK.

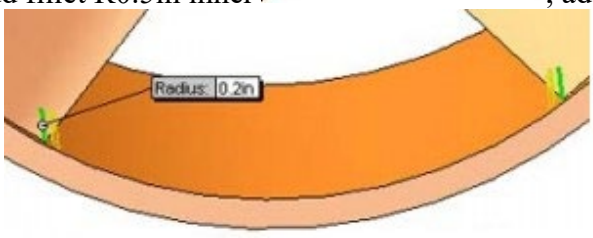
4. Insert sketch on edge wheel face, skecth for arm hole



, extruded cut , through all, OK.

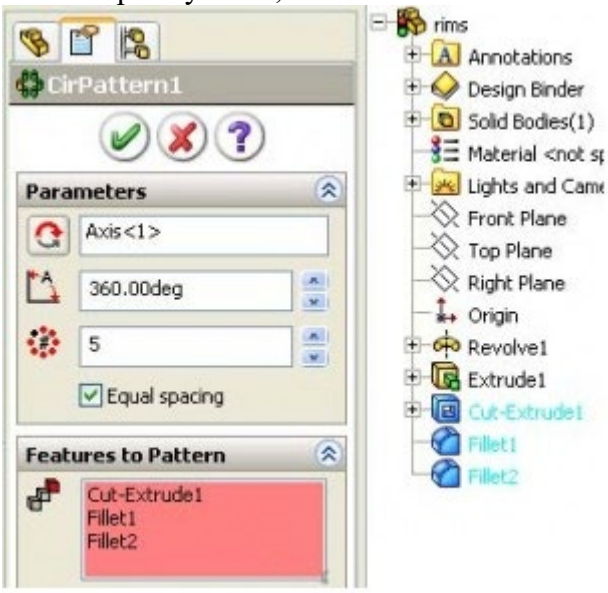
5. Add fillet R0.5in inner 

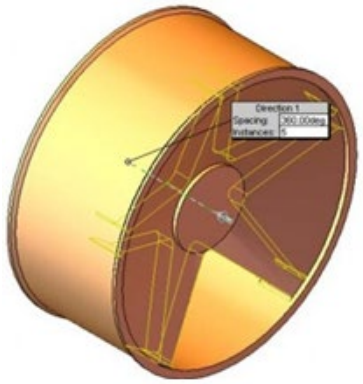
, add fillet 0.2in



OK.

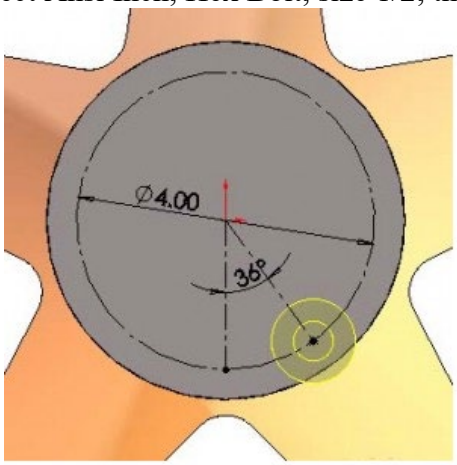
6. Click Circular Pattern  , click View>Temporary Axes, select center axis as rotation axis. 360 degree and #5 equal spacing





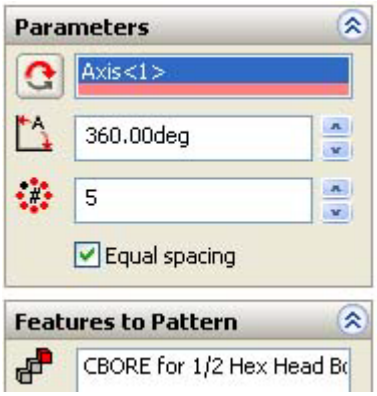
. Select Cut-Extrude1, Fillet1 and Fillet2 as a Features to Pattern. OK.

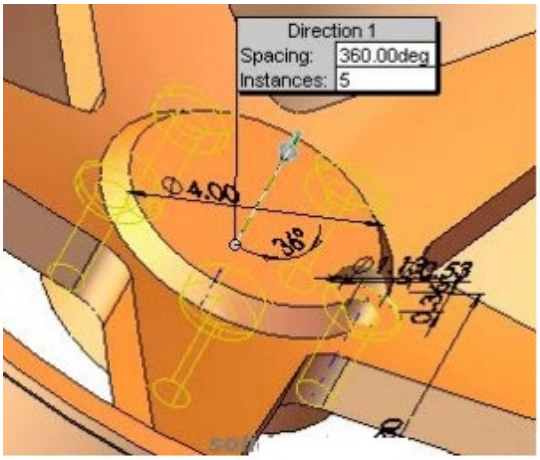
7. Select hub face, click Hole Wizard  , select Ansi Inch, Hex Bolt, size 1/2, through all.

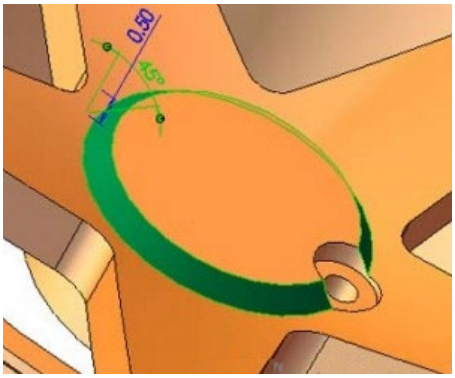


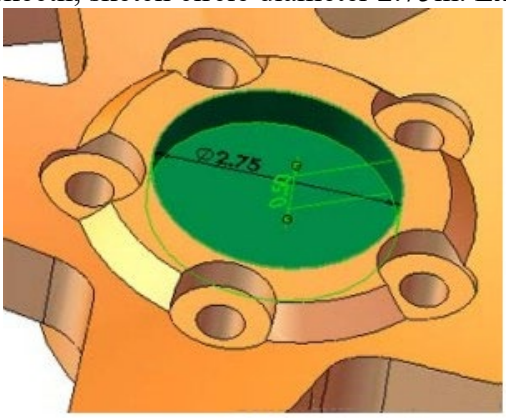
Position point at diameter 4in and 36 degree . OK.

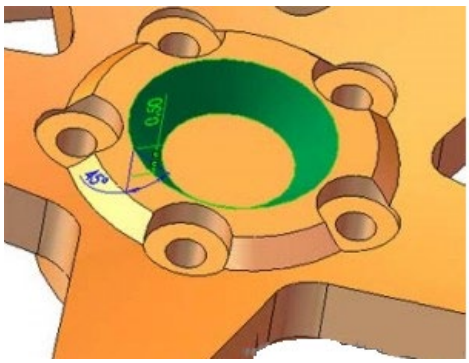
8. Click Circular Pattern  , select center temporary axis, 360 degree and #5 equal spacing. Select CBORE for 1/2 Hex Head Bolt as Features to Pattern. OK.

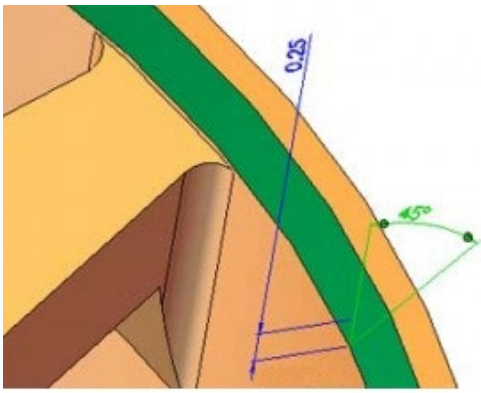




9. Add chamfer 0.5in to hub side. 

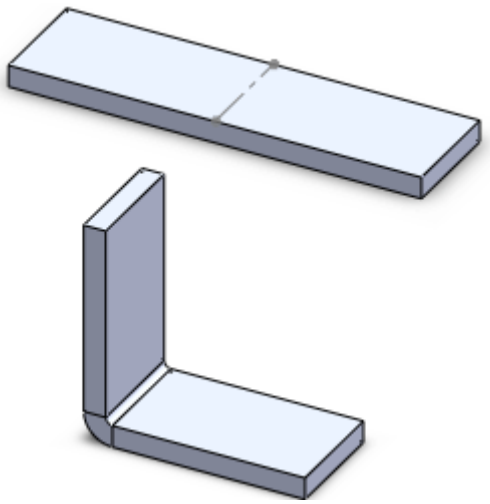
10. Click on hub face, insert skecth, sketch circle diameter 2.75in. Extrude Cut  to 0.5in deep. 

11. Add chamfer 0.5in to inner cut 

and add chamfer 0.25in to wheel edge 

, OK. Done.

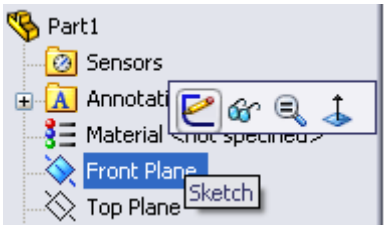
7. How to create simple sheet metal bend



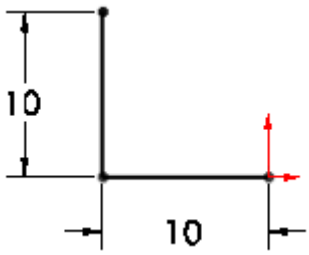
In this tutorials you will learn how to utilize sheetmetal tool such insert bend and flaten.

1. Click **New**.  Click **Part**,  **OK**.

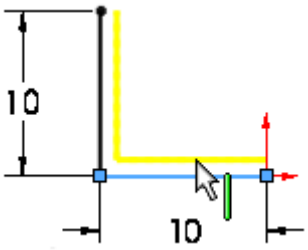
2. Click **Front Plane** and click on **Sketch**.



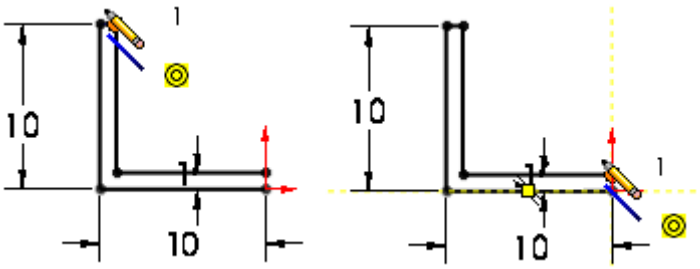
Use **Line**  , sketch L shape. Dimension sketch with **Smart Dimension**  as **10 x 10mm**

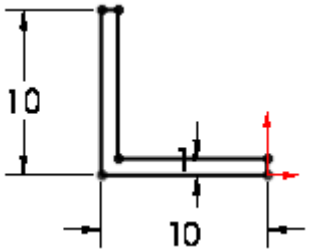


3. Click **Offset Entities**  and click L sketch. Set offset distance as **1mm**.

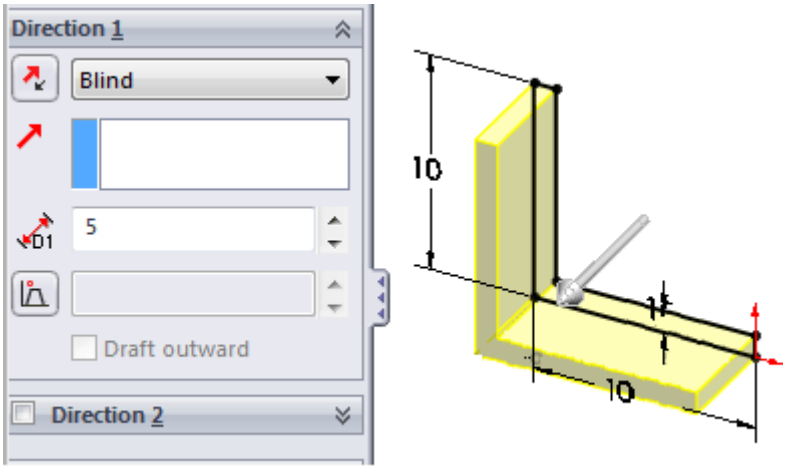


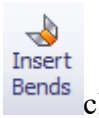
4. Use **Line** , sketch and connected open end of this sketch and make it close both end.



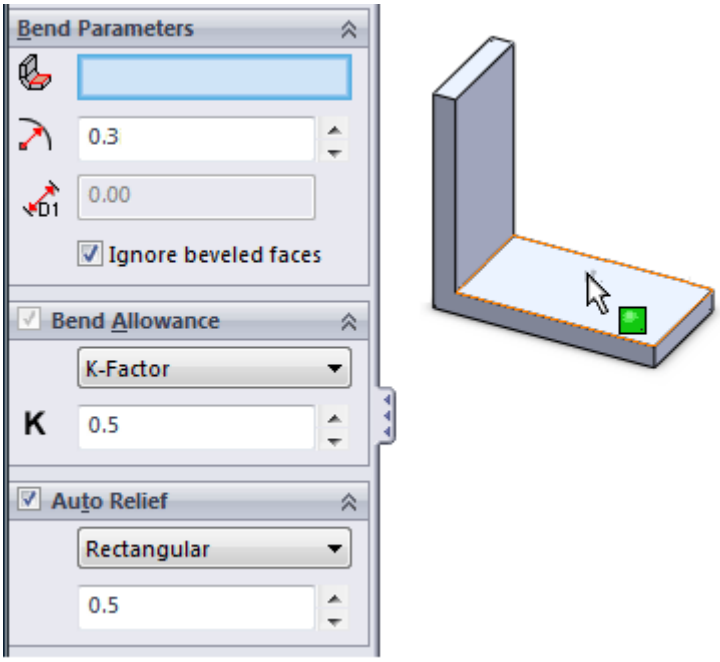


5. Click **Features>Extruded Boss/Base** set **D1** to **5mm** and **OK**.

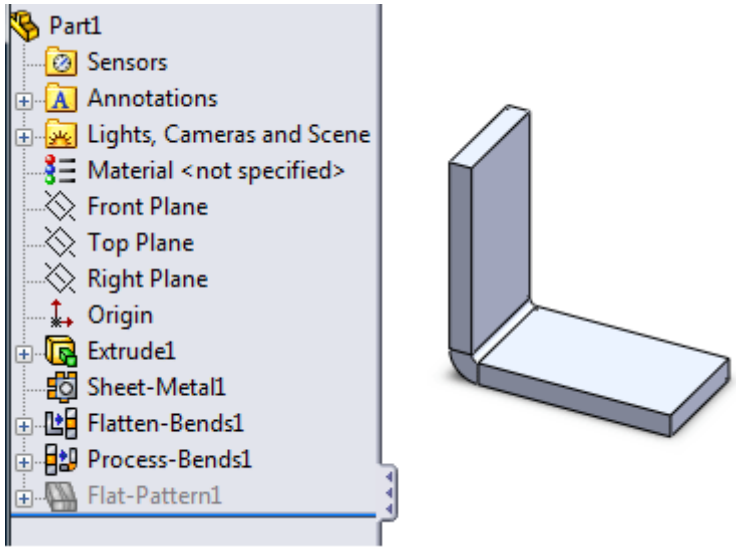


6. Click **Sheetmetal>Insert Bends**, 

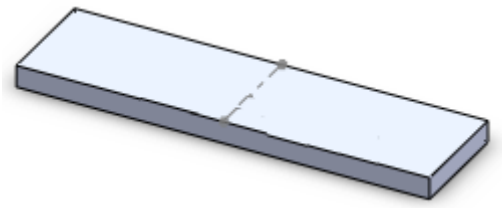
click flat face as reference when it flatten. Set bend radius to **0.3mm** and **K factor 0.5** and **OK**.



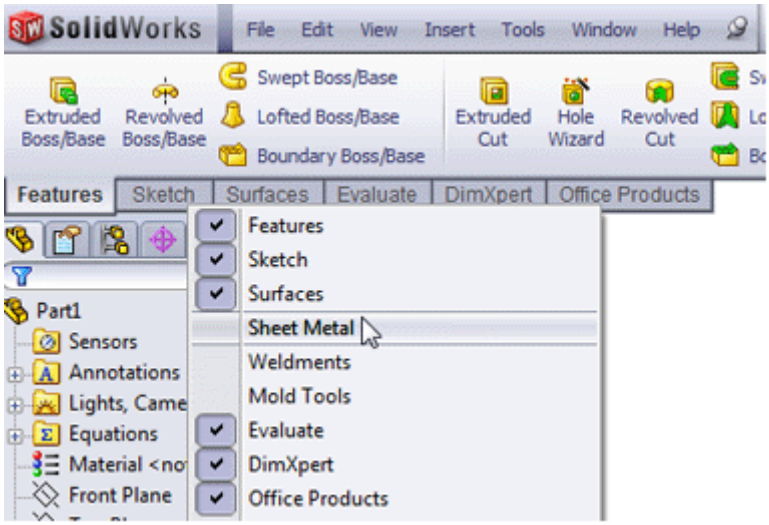
7. Your simple sheetmetal bend is ready. Look at part tree.



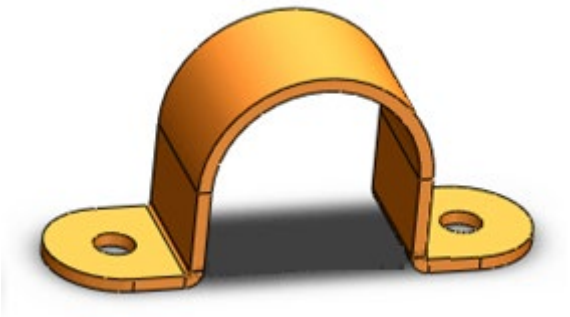
8. To view this part in flatten form click **Sheetmetal>Flatten**.



Have fun.. If you cannot find the sheetmetal tool in you main tool menu, you can right click on main menu tab and check Sheetmetal option.



You know the basic, try model this bracket.

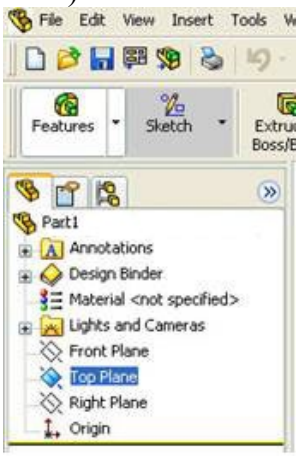


8. How to create spring

1. Click New  (File>New) , click Part  , OK .

2. Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select MMGS (millimeter ,gram , second) OK

3. Select Top Plane , from lower left menu select Normal To.

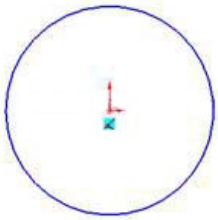


4. Click Sketch 

in Command Manager, click Circle 

. As you can see on upper right corner sketch icon appear indicate that you're on sketch mode 
.

5. Pick Origin point as starting point, drag to right hand side

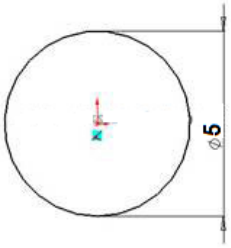


no need to be exact the size will define in later step. Press keyboard ESC to end circle sketch.

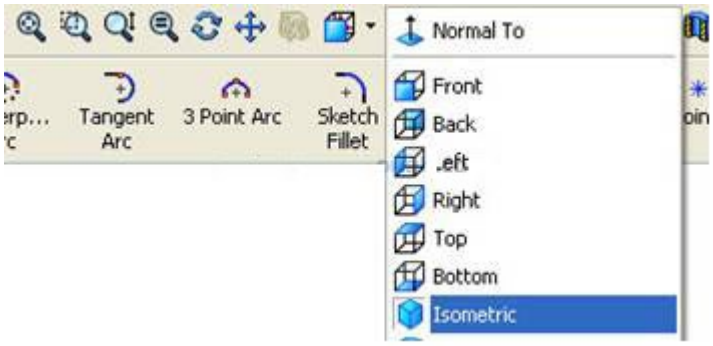
Note: There is two type line generated by in sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined..

6. Define sketch with dimension. Click Smart Dimension 

, and start dimensioning

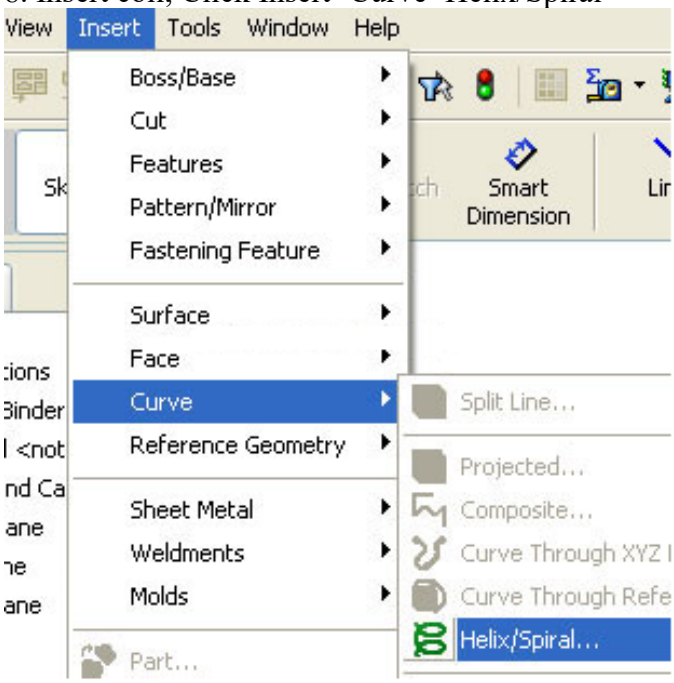


pick circle edge and set to 5mm . Press keyboard ESC to end smart dimension.



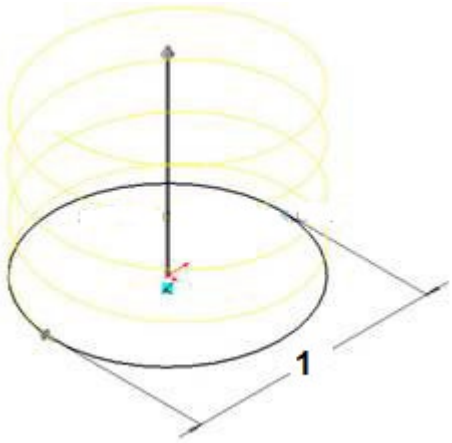
7. Change display to Isometric view.

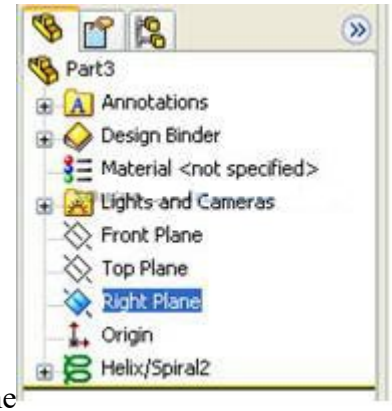
8. Insert coil, Click Insert>Curve>Helix/Spiral

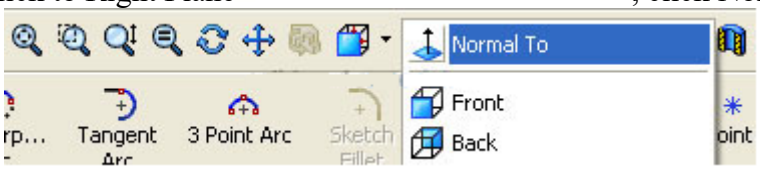


.

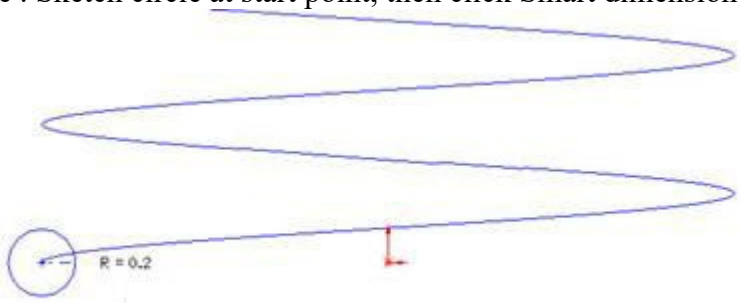
9. Press F to zoom fit, set Parameters Constant Pitch , Pitch 1mm Revolutions 4 , Start angle 0.0deg and  .

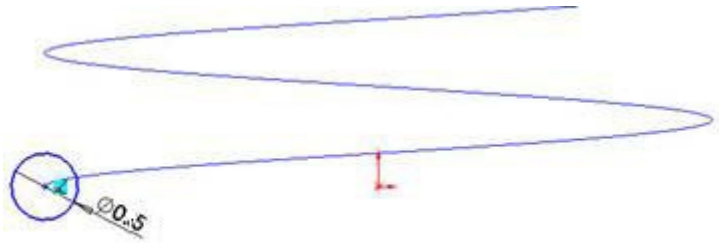


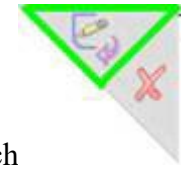
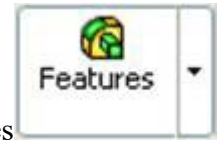
10. Click to Right Plane 

, click Normal To 

.

11. Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension set 

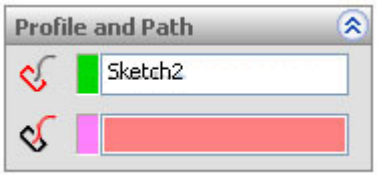
circle diameter to 0.5mm . 

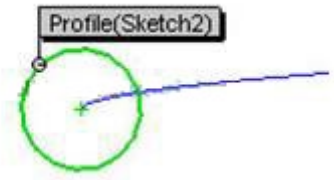
12. Click exit sketch  . Click Features 

and activate features menu.

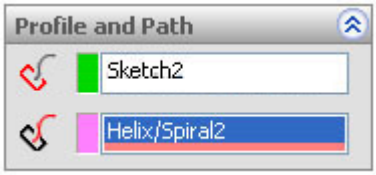
Click Swept Boss/Base 

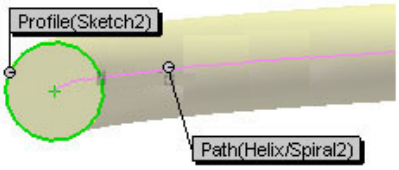
and set Profile to Sketch2 by click on circle sketch





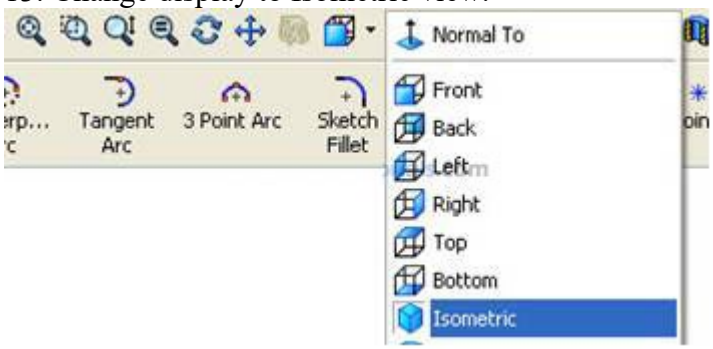
and set Path by click helix path





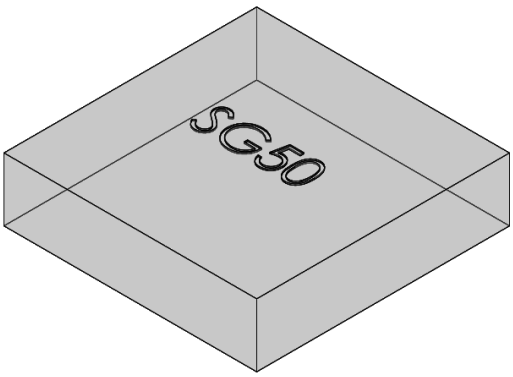
and  .

13. Change display to Isometric view.



14. Press F to zoom fit.

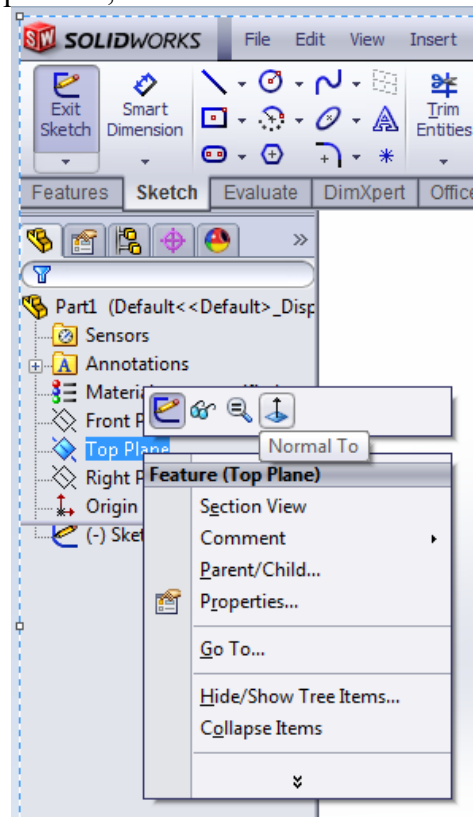
9. How to engrave text to part



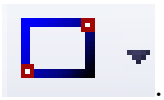
1. Click New  (File>New) , click Part  , OK .

2. Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select MMGS (millimeter,grm,second) OK .

3. Select Top Plane , from lower left menu select Normal To.



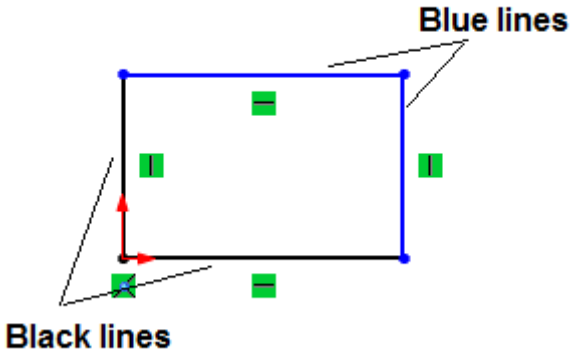
4. Click Sketch 

in Command Manager, click Rectangle  . As you can see

on upper right corner sketch icon appear indicate that you're on sketch mode .

5. Pick Origin point as starting point, drag to right hand side

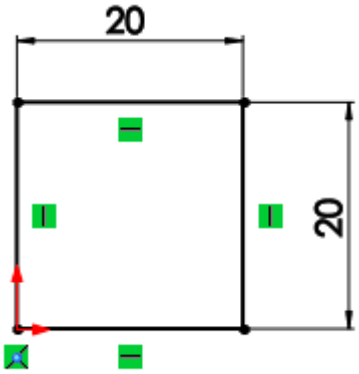
no need to be exact the size will define in later step. Press keyboard ESC to end rectangle sketch.



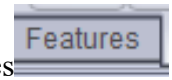
Note: There is two type line generated by your sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined.

6. Define sketch with dimension. Click Smart Dimension 

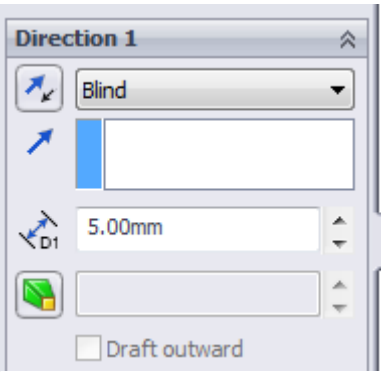
, and start dimensioning pick vertical line and set to **20mm** , pick horizontal line and set to **20mm**



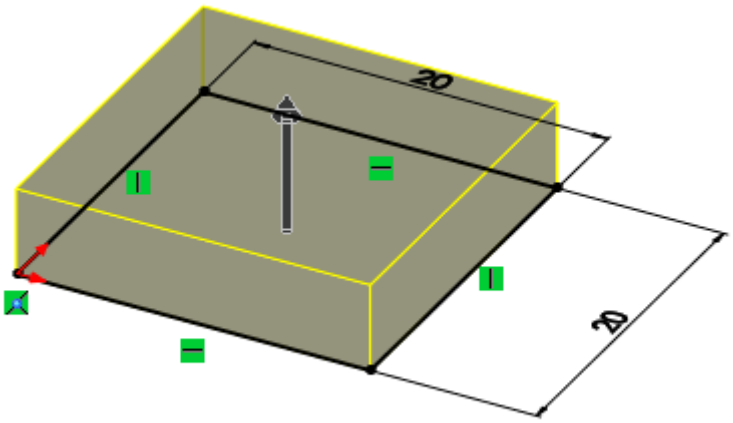
. Press keyboard ESC to end smart dimension.

7. Build feature from sketch, click Features 

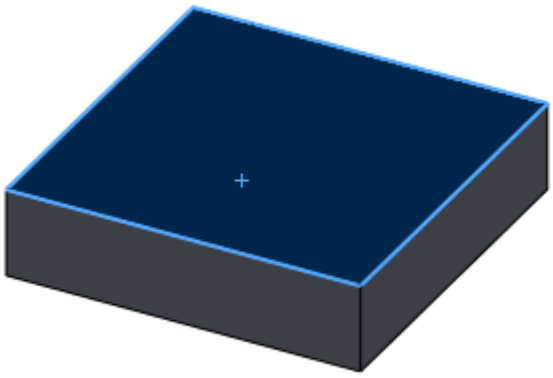
and activate features menu. Click Extruded Boss/Base 

and set D1 to 5mm 

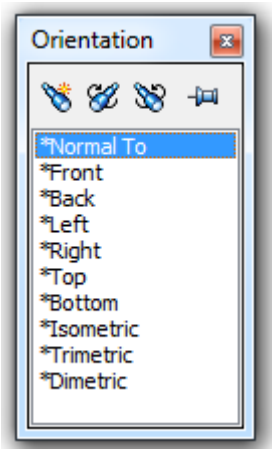
and



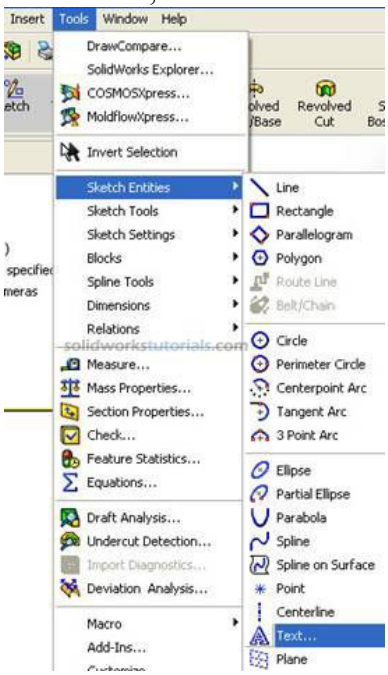

.

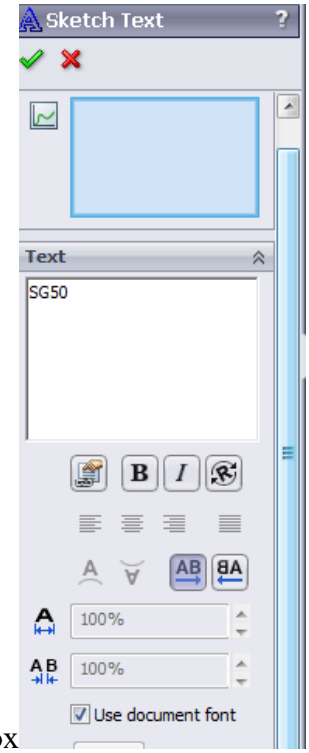


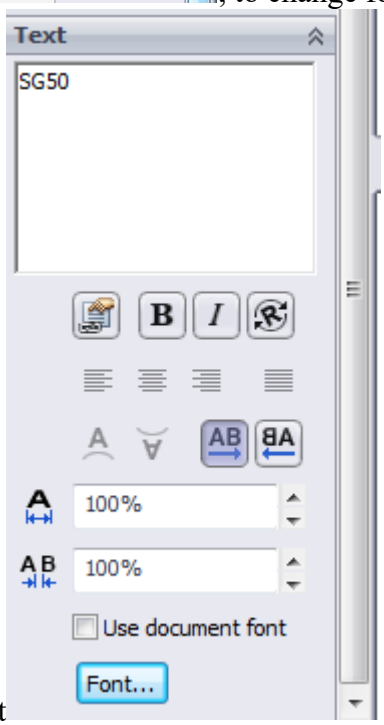
8. Click front top face , click Normal To



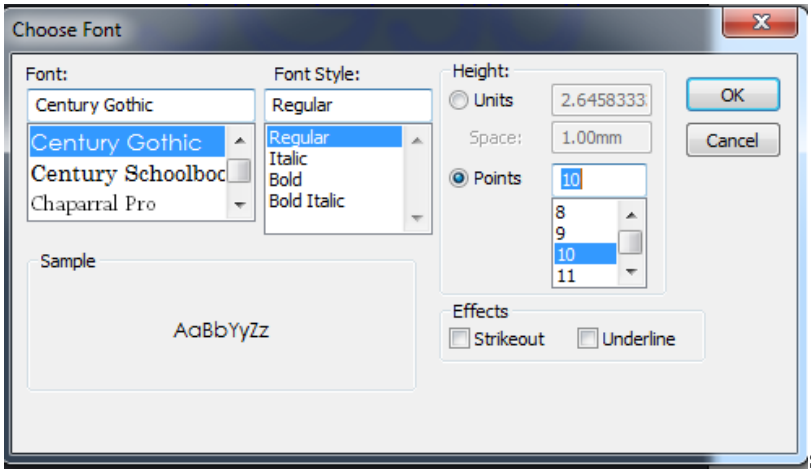
. Click Tools>Sketch Entities>Text…



9. Input text in text box 

, to change font type and size uncheck use document font 

. Click Font…

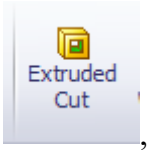


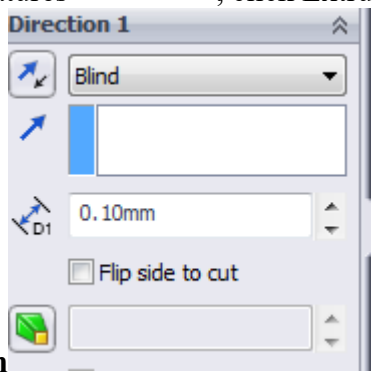
set height to Points 10 OK.

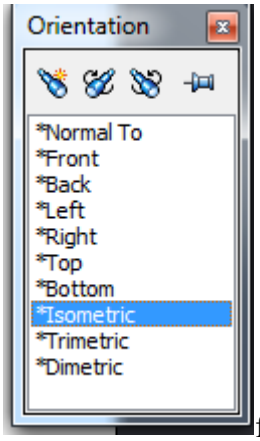
10. Click to part face to relocate text to center 

.

11. To engrave the text, click Features 

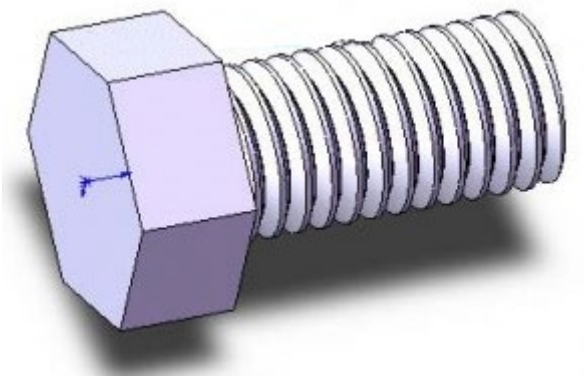
, click Extruded Cut 

, under Direction 1 Blind, set D1 to **0.1mm** 

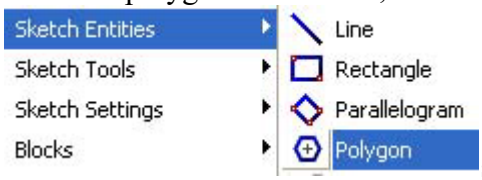
and  . Click Isometric 

from lower left view menu.

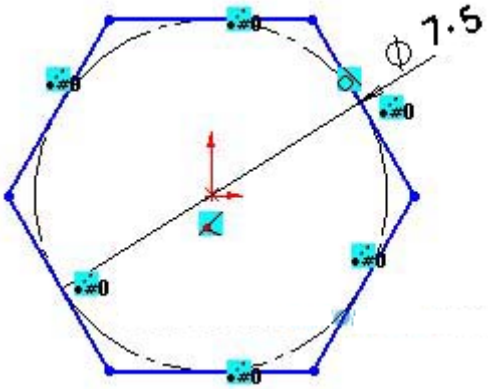
10. How to create hexagonal head bolt

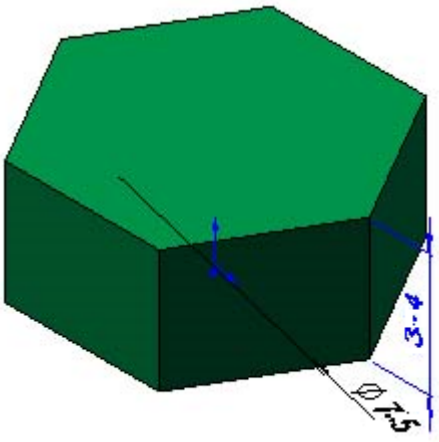


1. Sketch a polygon with 6 side, Tools>Sketch Entities>Polygon



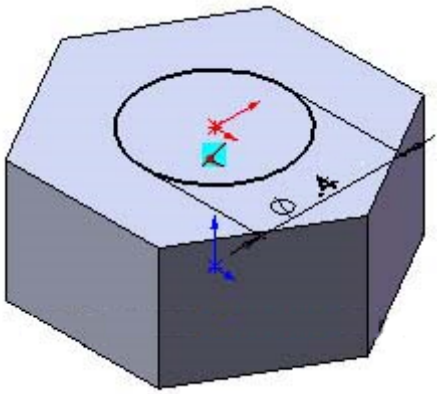
set diameter to 7.5mm.



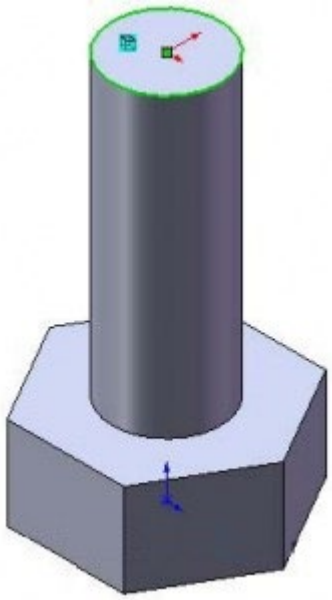


2. Extrude  sketch to 3.4mm

3. Create minor diameter for thread, sketch circle on top face, set diameter to 4mm

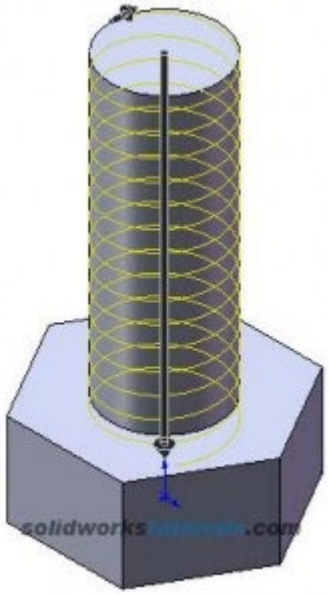


4. Extrude sketch to 11mm



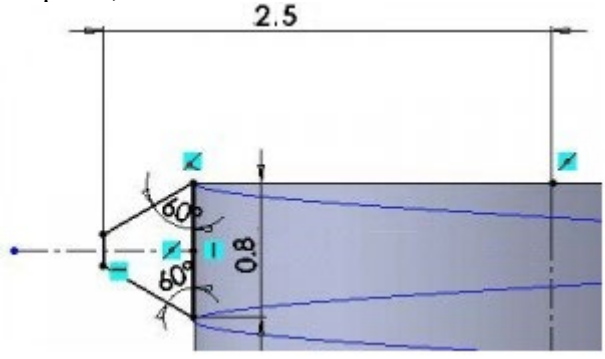
5. Click end edge of thread shaft, click convert entities  .

6. Select Helix/Spiral feature  set height to 12mm, pitch 0.8mm



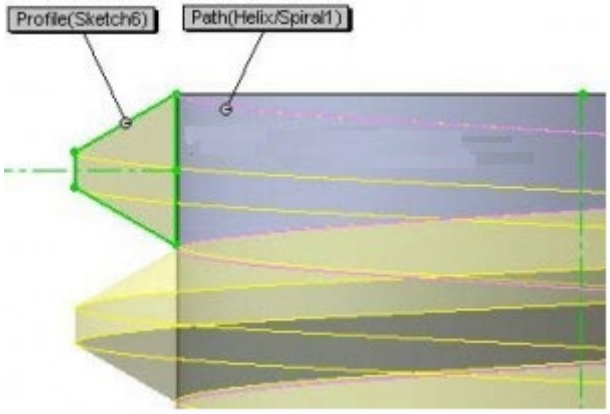
Ok.

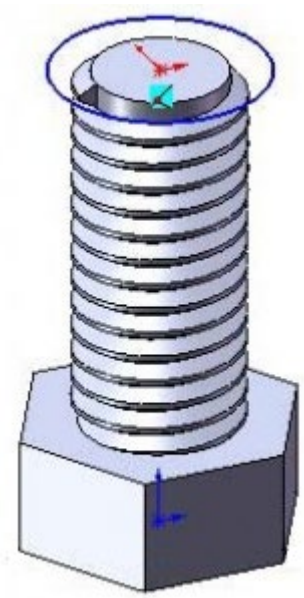
7. Right click on Front plane, Insert sketch  sketch thead profile.



8. Click sweep feature  , 

select sketch profile as sketch and helix as a path, OK.

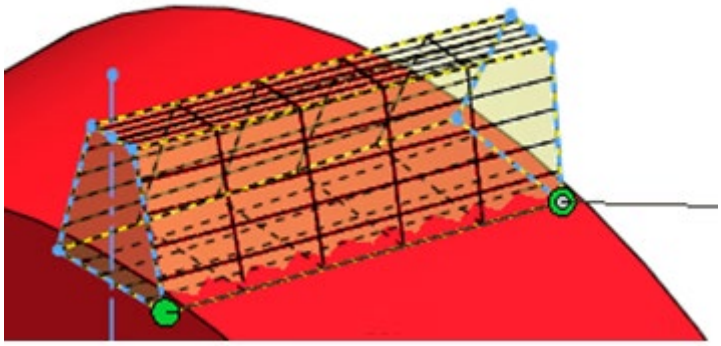




9. Create skecth a circle on the end shaft, extrude cut 1mm. 

10. Finish.

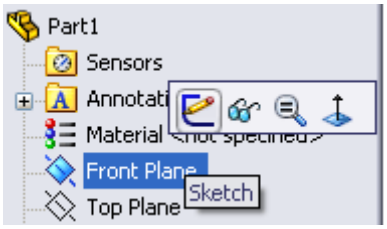
11. How to create helical gear



In this solidworks tutorial, you will create helical gear.

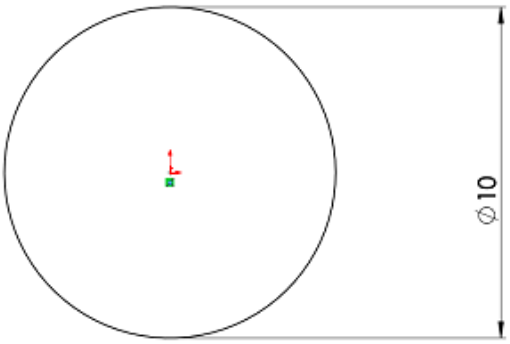
1. Click **New**.  Click **Part**,  **OK**.

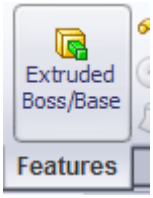
2. Click **Front Plane** and click on **Sketch**.

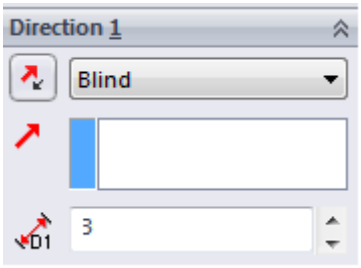


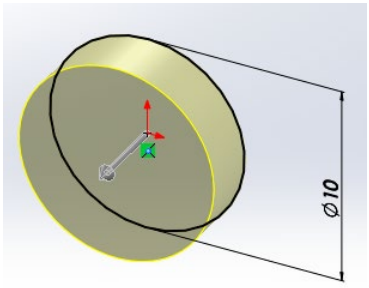
3. Click **Circle**  and sketch a circle center at origin. Click **Smart Dimension**, 

click sketched circle and set it diameter to **10mm**



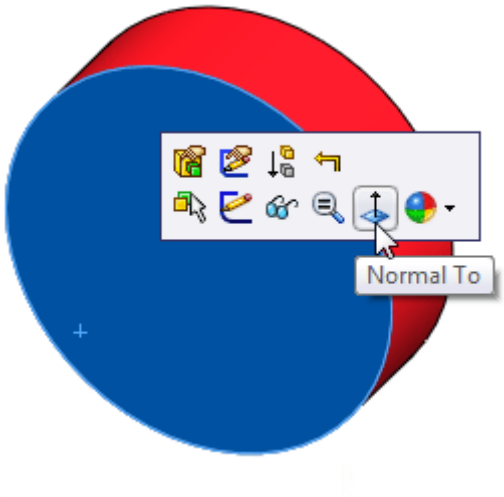
4. You just completed your sketch, let's build feature from it. Click **Features>Extruded Boss/Base**. 



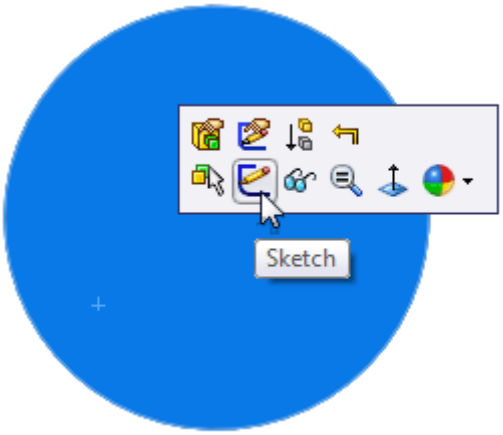
Set **D1** to **3mm** 

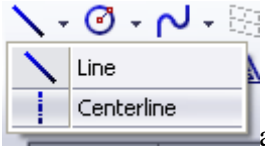
and  .

5. Click on front face and click **Normal To**.

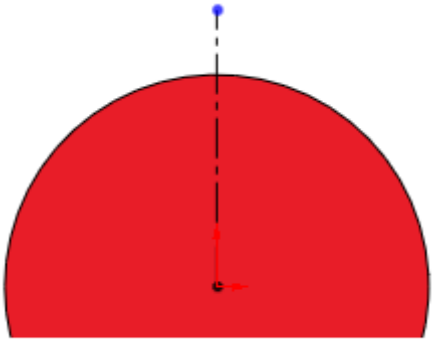


6. Click on front face and click **Sketch**.

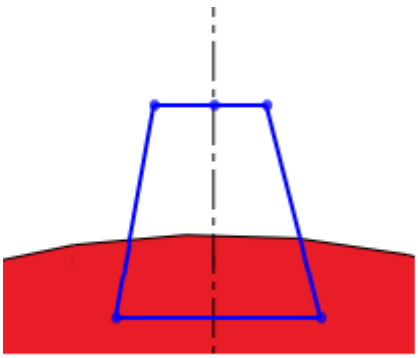


7. Click on **Centerline** 

and sketch vertical Centerline.

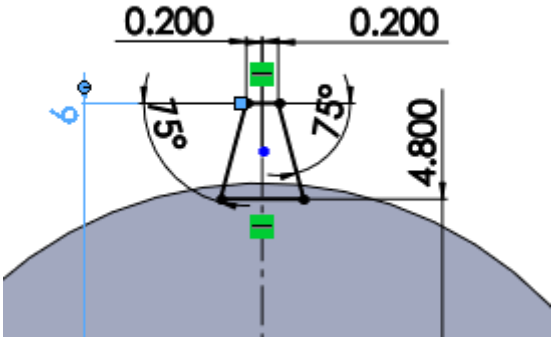


8. Click **Line** and sketch gear teeth profile.

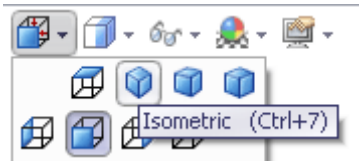


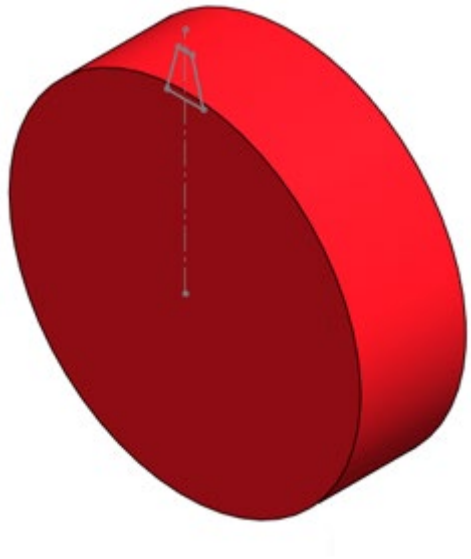
9. Click **Smart Dimension**, 

dimension sketch as sketched below.

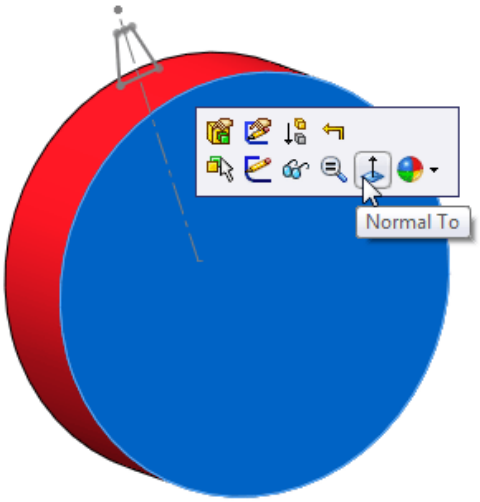


10. Click **Exit Sketch**,  change view to **Isometric**.

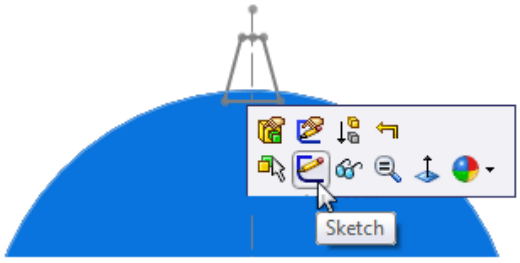




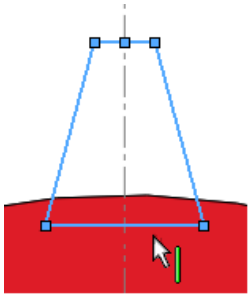
11. Click scroll mouse button and rotate the part to back side.



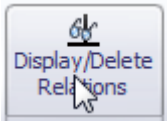
Click the back face and select **Normal To.** Click on this face again and click **Sketch.**



12. We will trace last sketch to this face, while holding **CTRL** click **all sketched line**

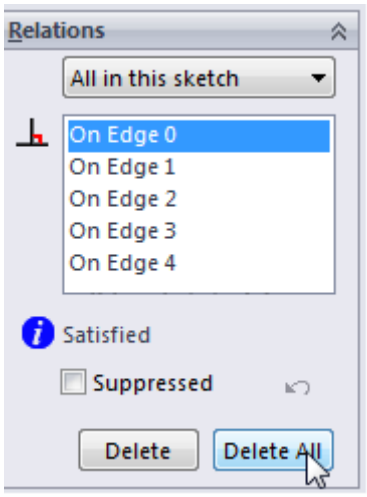


and click **Convert Entities** 

. Now we need removed all relation between this sketch and the other sketch, click **Display/Delete Relations** 

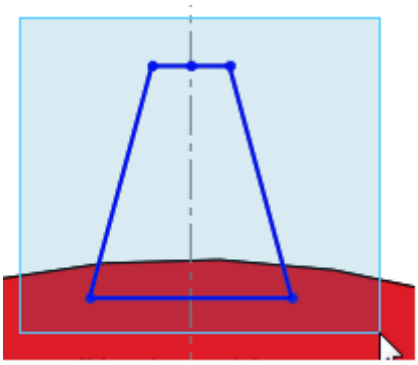
click **Delete All**

(Alterative method to use Project Curve (Sketch on faces) > Select line drawn from Top Plane > and Select face diameter)

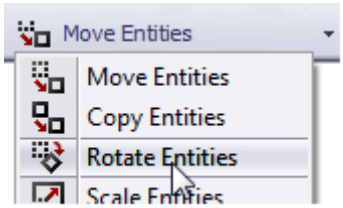


and  .

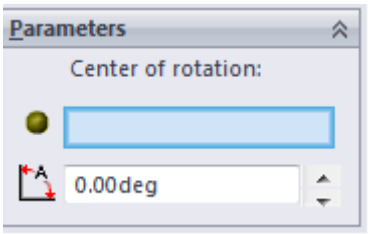
13. Click and drag select all the sketch line.



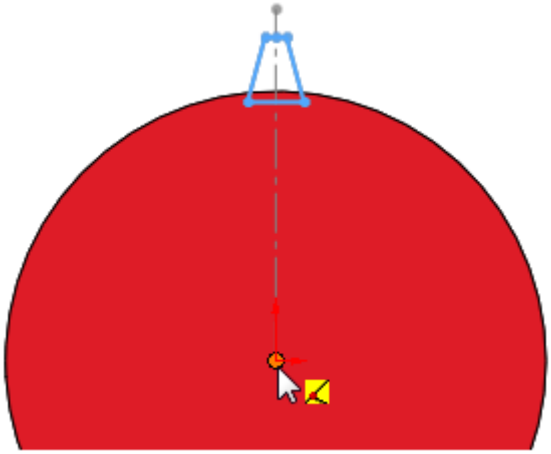
Click on **Rotate Entities,**



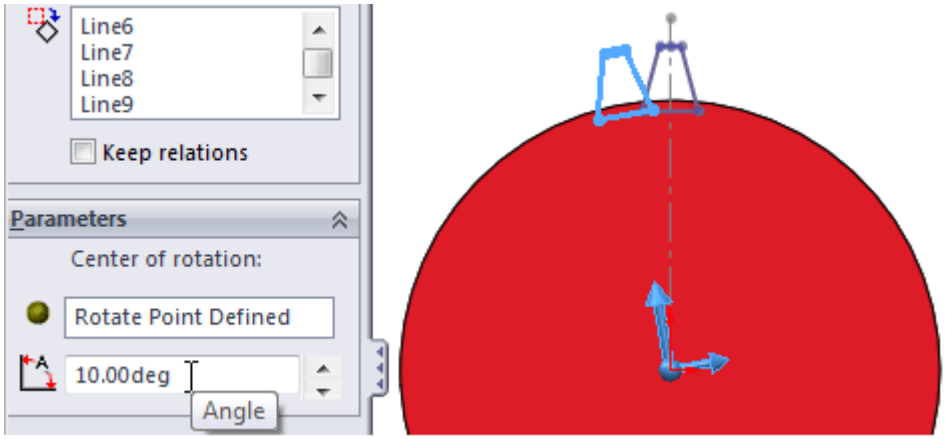
Click **Center of Rotation box**



and click **origin (center part).**

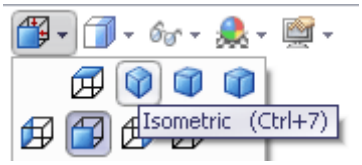


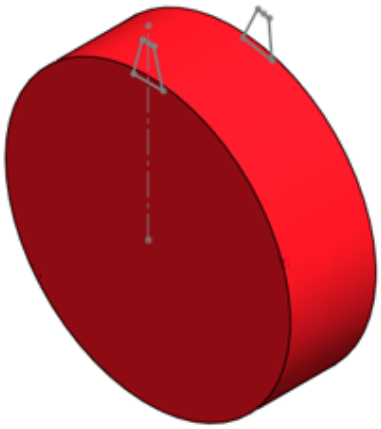
On Parameter option enter **10 deg** rotation.



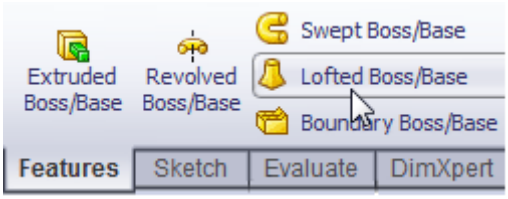
and  .

14. Click **Exit Sketch**, change view to **Isometric**.

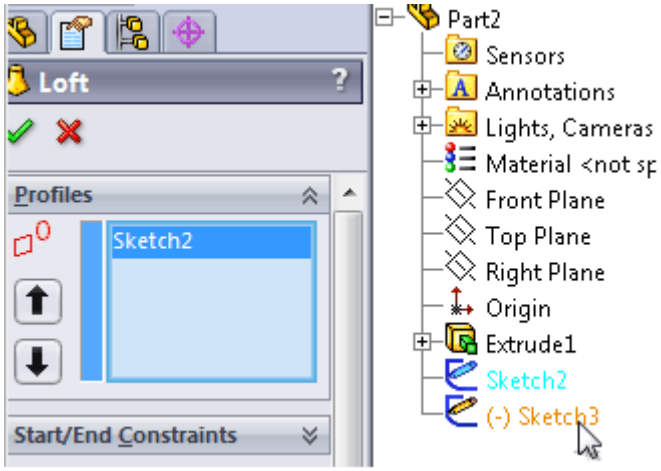


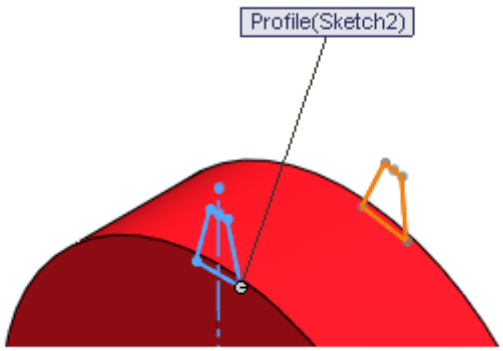


15. Click **Features>Lofted Boos/Base,**

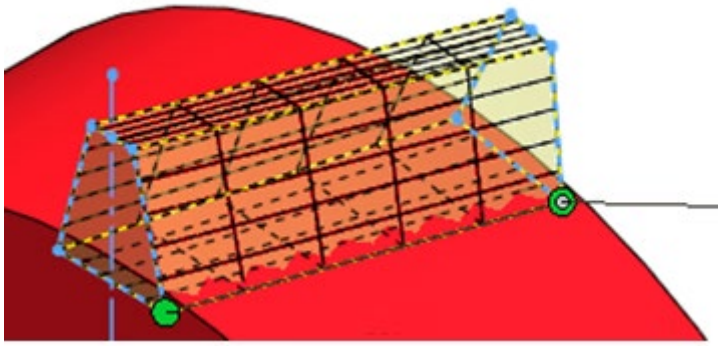


open up part tree and double click **Sketch2** and **Sketch3** to add for lofted features.

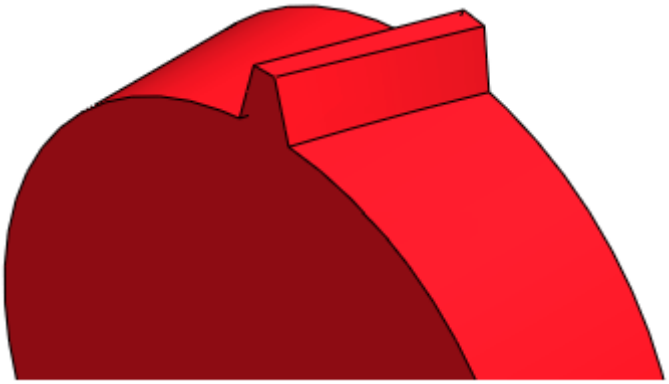


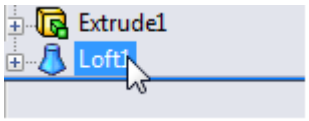


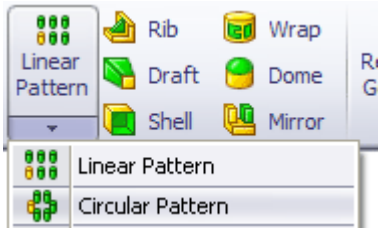
Make sure two green point is at the same edge as other sketch, if not drag and relocate it.



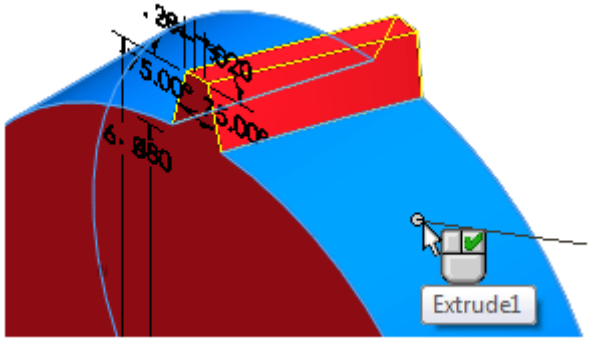
and  .



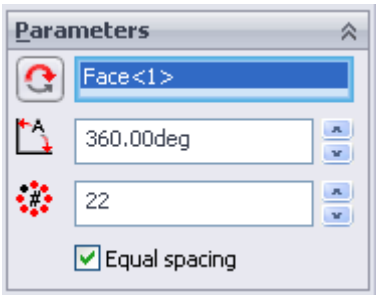
12. Click on **Loft1** (gear teeth) 

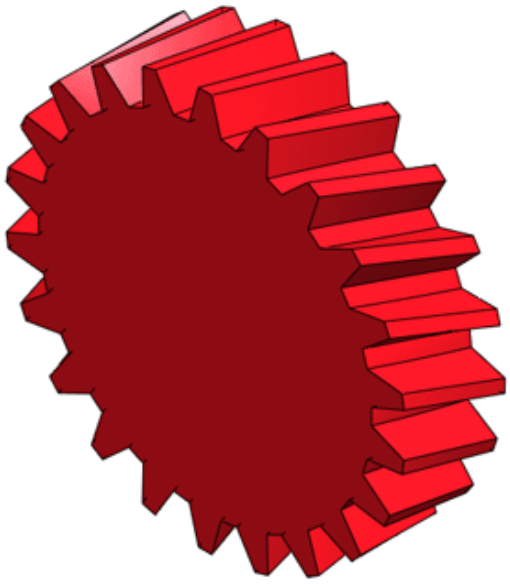
and click **Circular Pattern**. 

Click on the cylinder face as axis of rotation (or click on View>Temporary Axes select the temporary axis as axis of rotation).



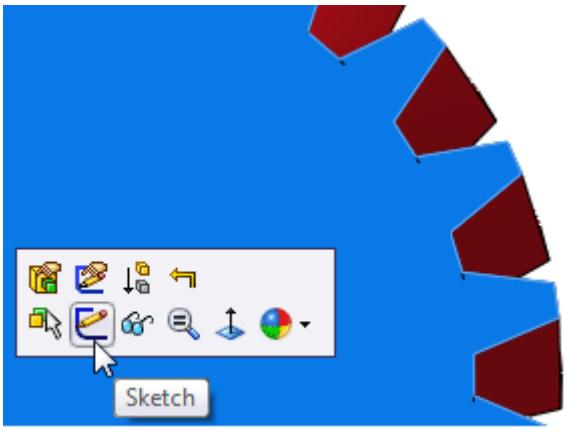
Set **Instances** to **22** and  .





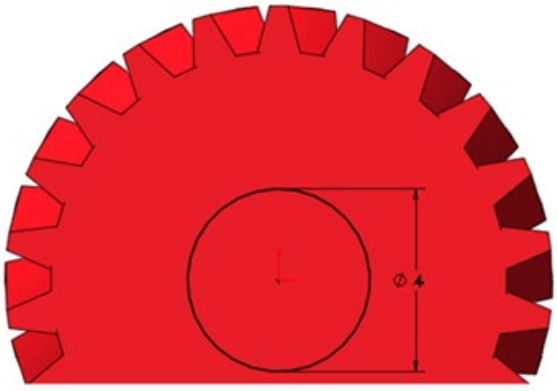
13. Click on Front face and select **Normal To**.

14. Click on front face and select **Sketch**.



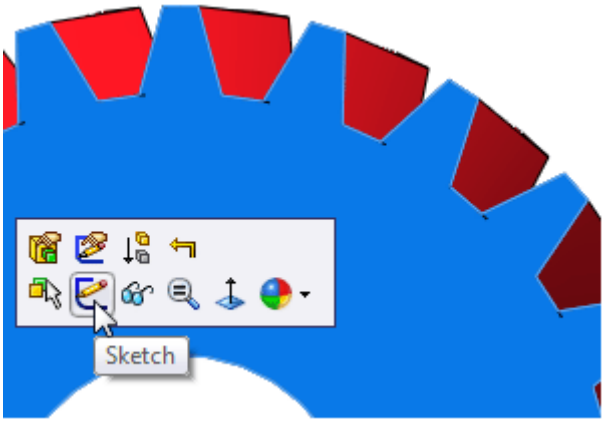
15. Sketch a **Circle**  and sketch a circle center at origin. Click **Smart Dimension**, 

dimension sketch as **4mm** circle.



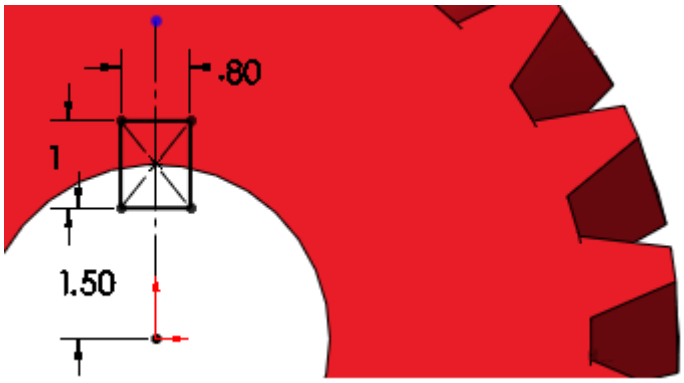
16. Click **Features>Extruded Cut**  and set **Direction** to **Through All** and  .

17. Click on front face and select **Sketch**.



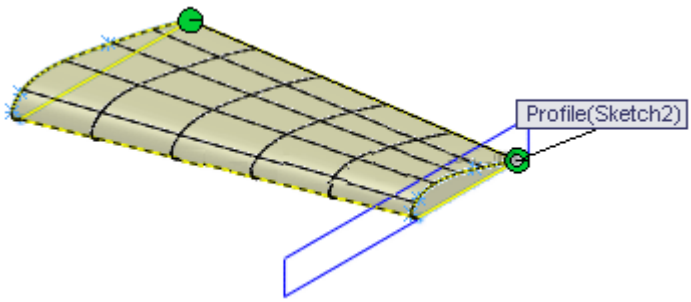
18. Click **Rectangle** and sketch a rectangle as sketched. Click **Smart Dimension**, 

dimension rectangle as skecthed below.



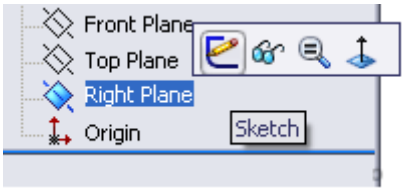
16. Click **Features>Extruded Cut**  and set **Direction** to **Through All** and  .

12. How to create aero plane wings

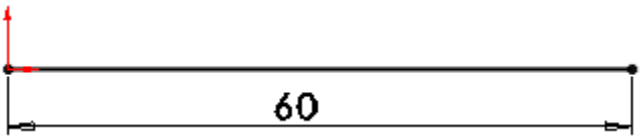


1. Click **New**, **Part** and **OK.**

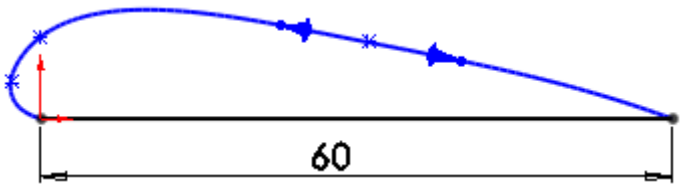
2. Click on **Right Plane** and click **Sketch**.



3. Sketch a center aerofoil profile at this plane. Click **Line**, sketch a horizontal line, click **Smart Dimension** and dimension the line as **60mm**.

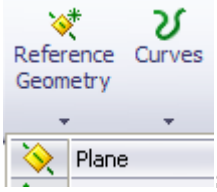


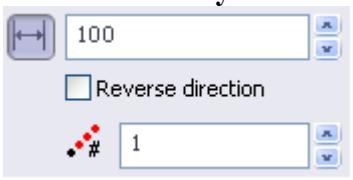
4. To create top curve of aerofoil, click **Spline**, and sketch top curve as sketched below, to end Spline press **Esc** key.



Exit the sketch. 

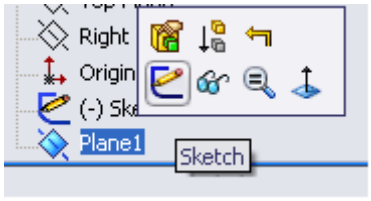
5. For another aerofoil profile at wing tip, you need to create another plane. Click on **Right Plane** 

and click **Reference Geometry>Plane** 

set distance between plane as **100mm** 

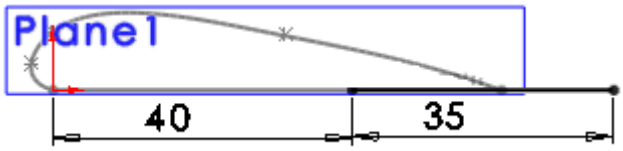
and  .

6. Click on **Plane 1** and click **Sketch**.

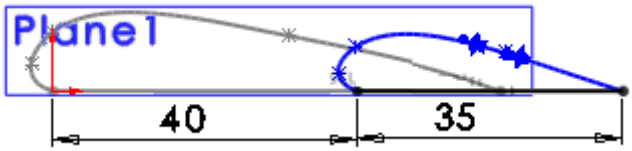


7. Click **Line**,  sketch a horizontal line on same level as first sketch a bit off set from origin, click **Smart Dimension** 

and dimension sketch as sketched below.

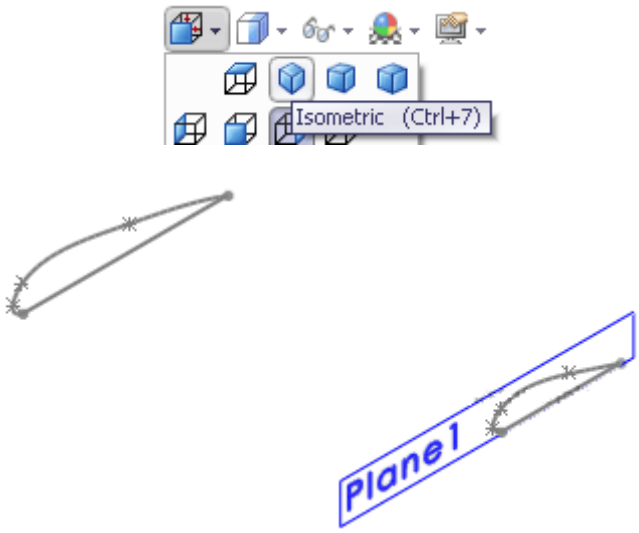


8. To create top curve of aerofoil, click **Spline**,  and sketch top curve as sketched below, to end **Spline** press **Esc** key.

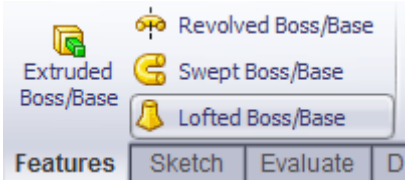


Exit the sketch. 

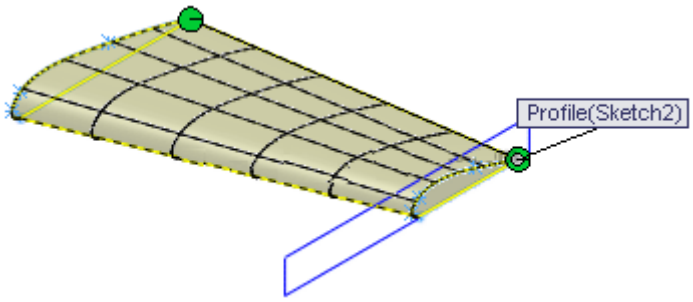
9. Click **View Oreintation>Isometric**.

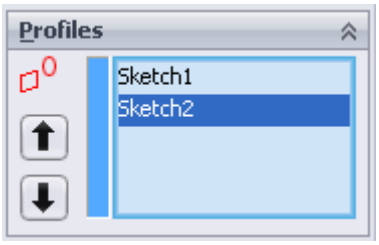


10. Click **Features**>**Lofted Boss/Base**,



click **Sketch1** and then **Sketch2**.

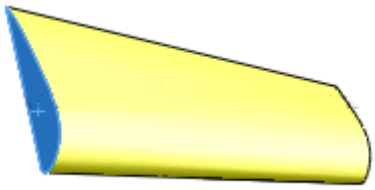




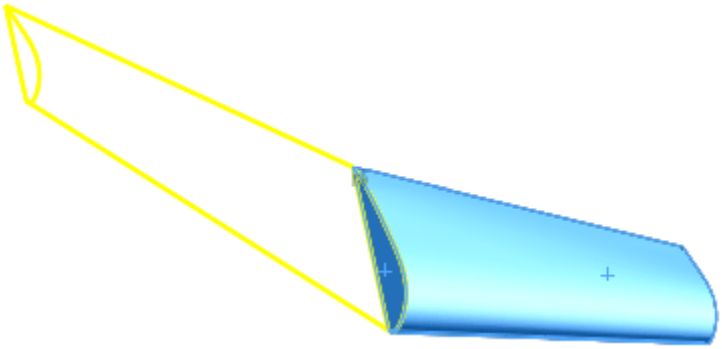
and  .

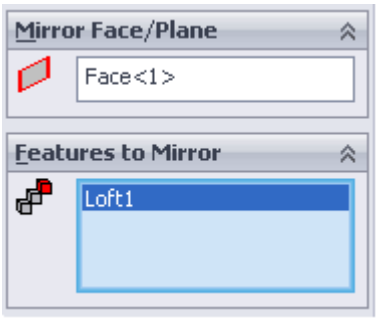
11. To hide Plane 1, click **Plane 1** and click **Hide**.

12. Now let make the full wings, click on **Mirror**.  Turn the wings to right side and **select center face** as a **Mirror Face/Plane**.



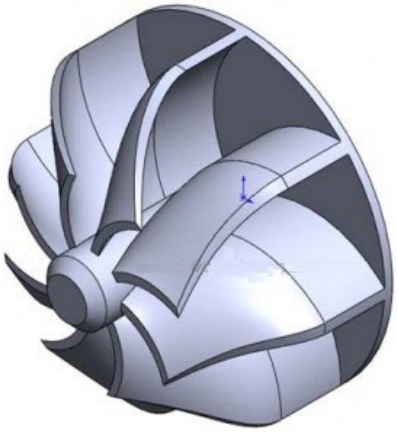
Click on wing body as **Features to Mirror**

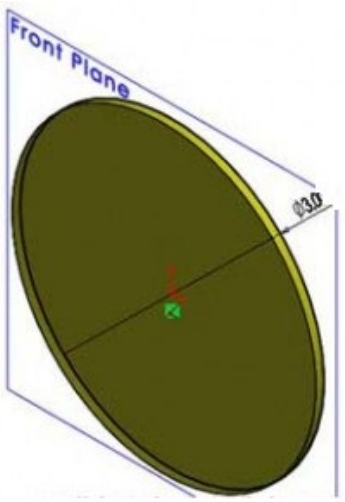




and  .

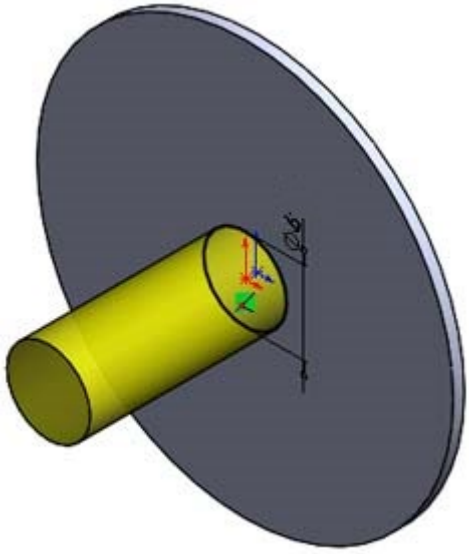
13. How to create turbo fins



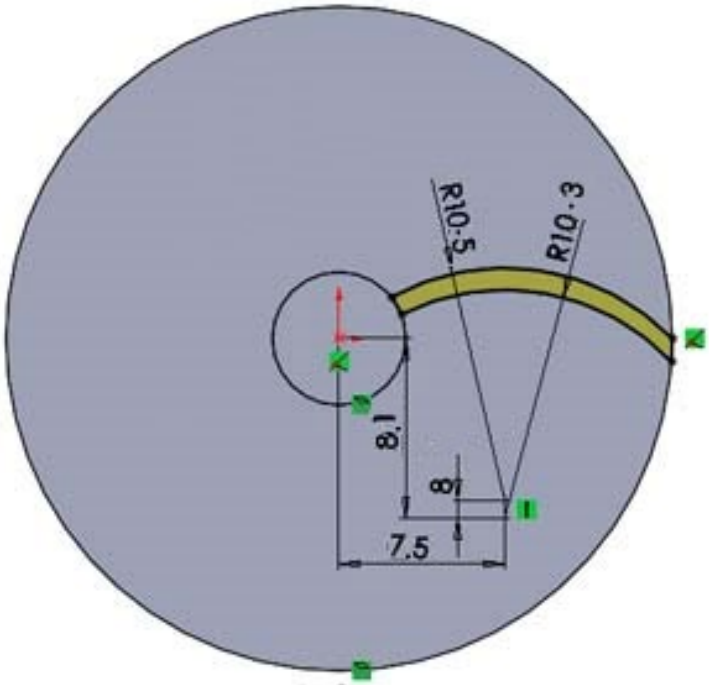


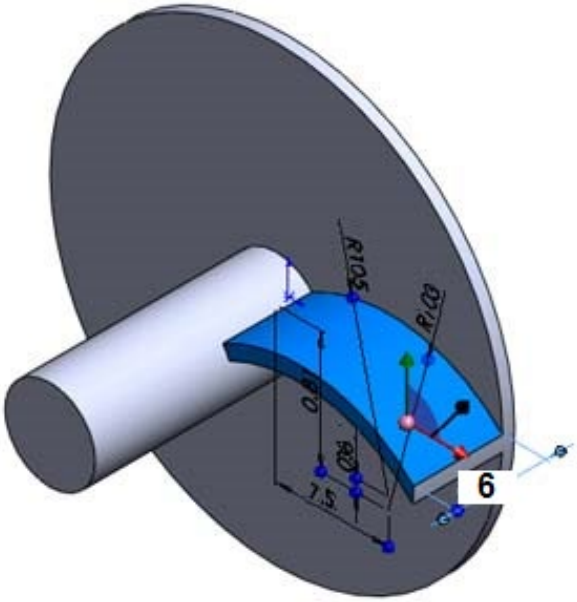
1. Skecth **30mm** circle and extrude to **0.8mm** on front plane.

2. Skecth **6mm** circle on top extruded face and exrude to **15mm.**



3. Sketch fin profile at extruded face as shown and extrude to 6mm.



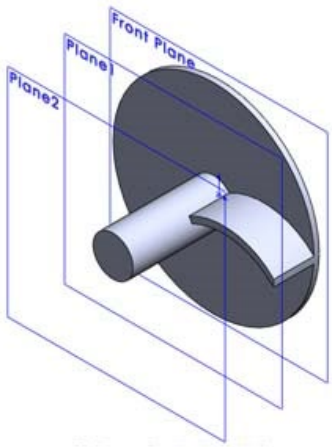


4. Add Plane 1 

with **6.7**

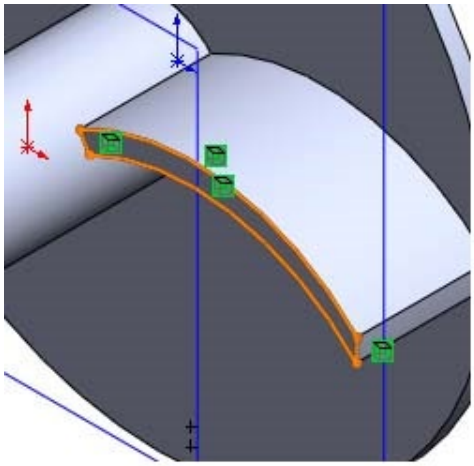
offset from Front plane and

Plane 2 with **8.5mm** from Plane 1.

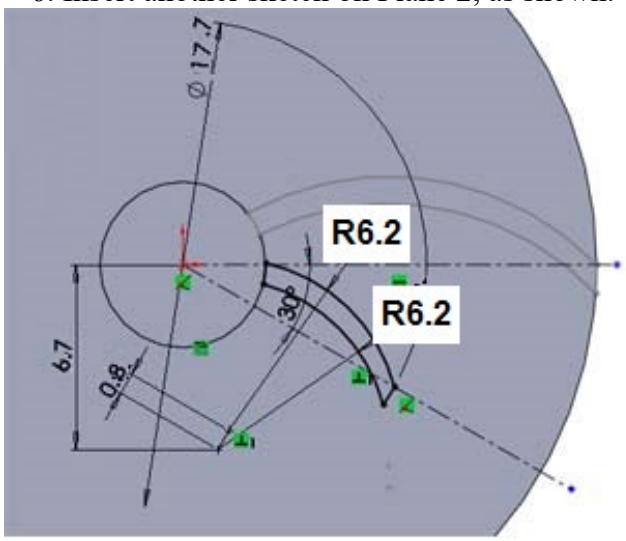


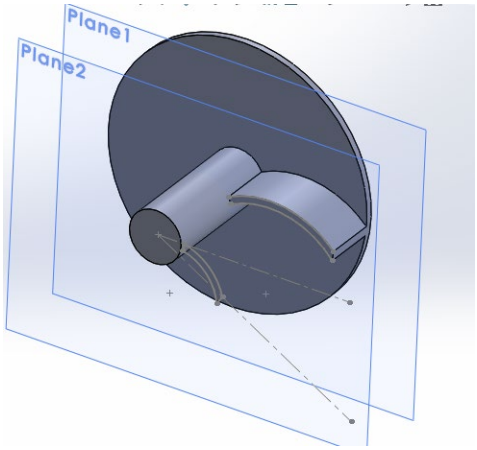
5. Insert sketch on Plane 1, select all edges to extruded fin and convert it to entities.

(Convert Entities>Entities to convert: Click the geometry>Ok>Select Plane1>OK)



6. Insert another sketch on Plane 2, as shown.

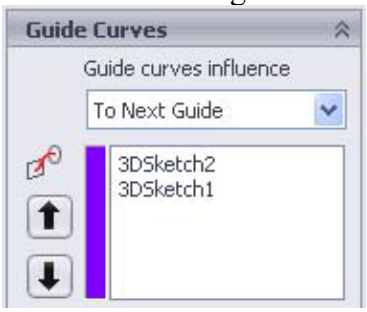




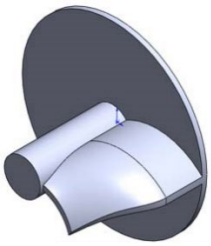
7. . Click Lofted Boss/Base 

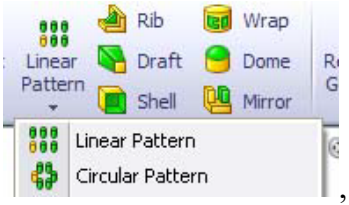
, select profile Sketch 5 and sketch 6

and for guide curves select 3DSketch1 and 3DSketch2



, OK.

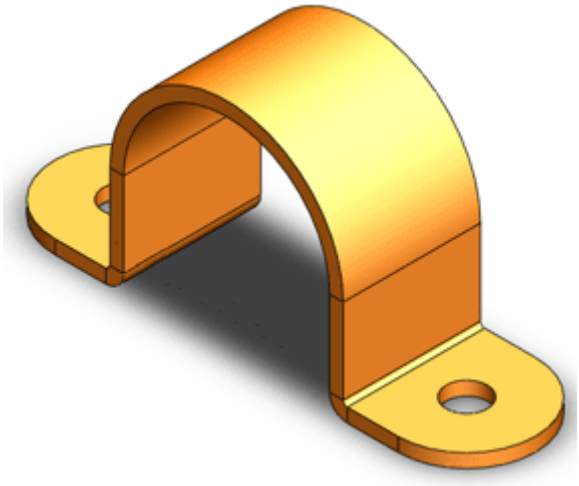


9. Click Circular Pattern 

, view temporary axis Tools>Temporary Axes. Select center axis, 360 degree, #8, Equal Spacing, OK

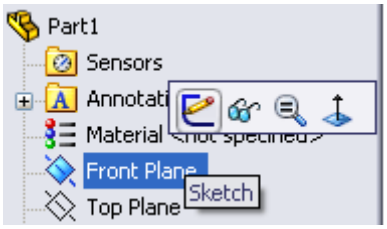
. Done!

14. How to create U bracket sheet metal

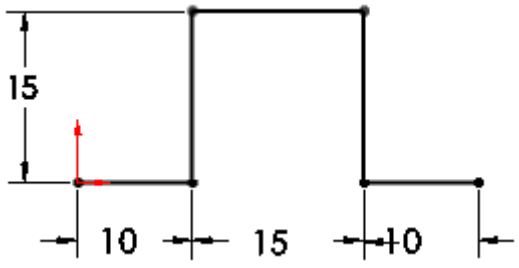


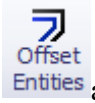
1. Click **New**.  Click **Part**,  **OK**.

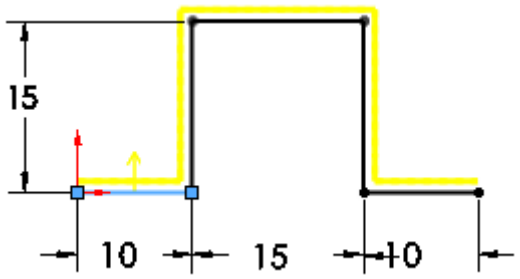
2. Click **Front Plane** and click on **Sketch**.



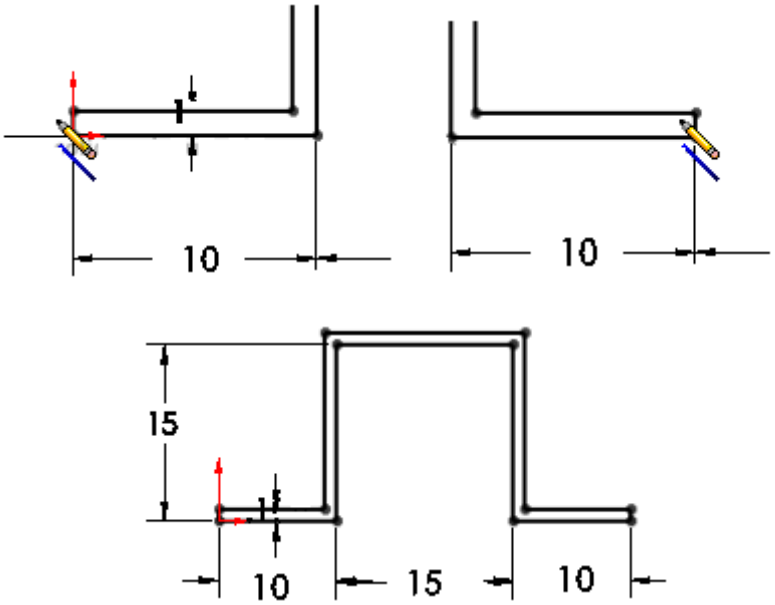
Use **Line**  , sketch U shape. Dimension sketch with **Smart Dimension**  as **10mmx 15mm x10mm and 15mm height.**

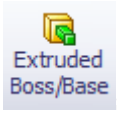


3. Click **Offset Entities**  and click U sketch. Set offset distance as **10mm,** check **Reverse** box and **OK**.

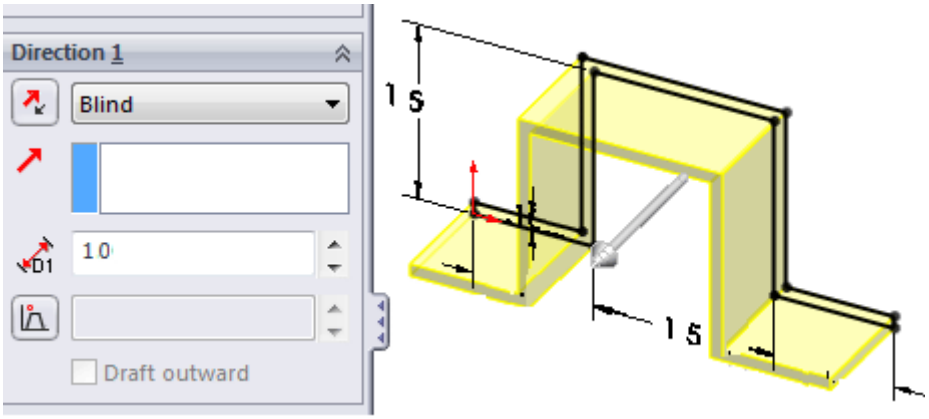


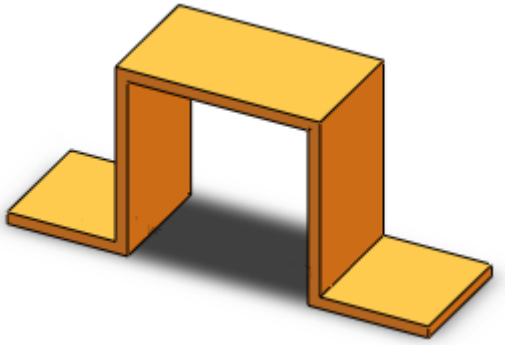
4. Use **Line**  , sketch and connected open end of this sketch and make it close both end.



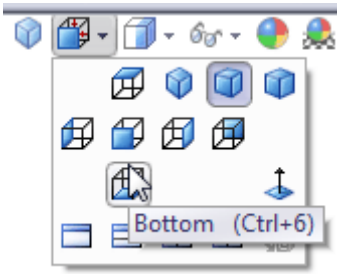
5. Click **Features>Extruded Boss/Base** 

set **D1** to **1in** and **OK**.

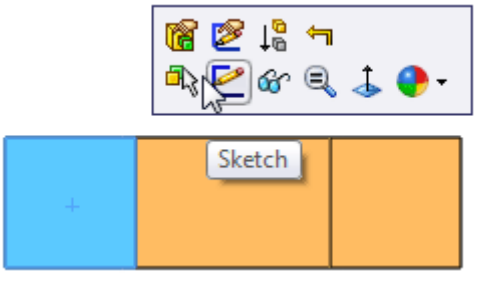




6. Click **View>Bottom**

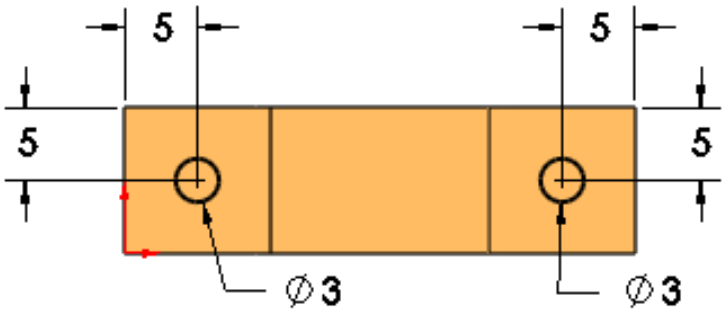


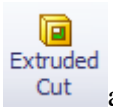
click on bottom face and click **Sketch**.



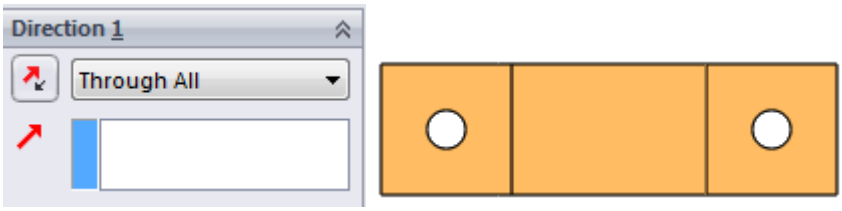
7. Click **Circle** and sketch 2 circle on bottom face each side. Use **Smart Dimension** 

to dimension this sketch as sketched below.

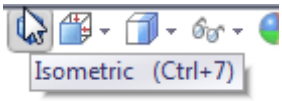


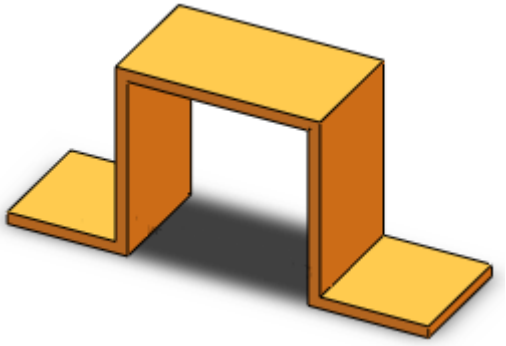
8. Click **Features>Extruded Cut** 

and cut **Through All** this circle.



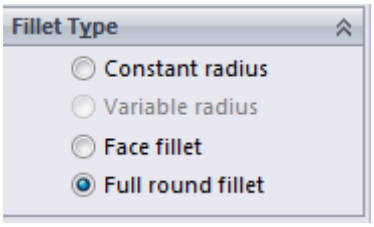
9. Click **View>Isometric.**



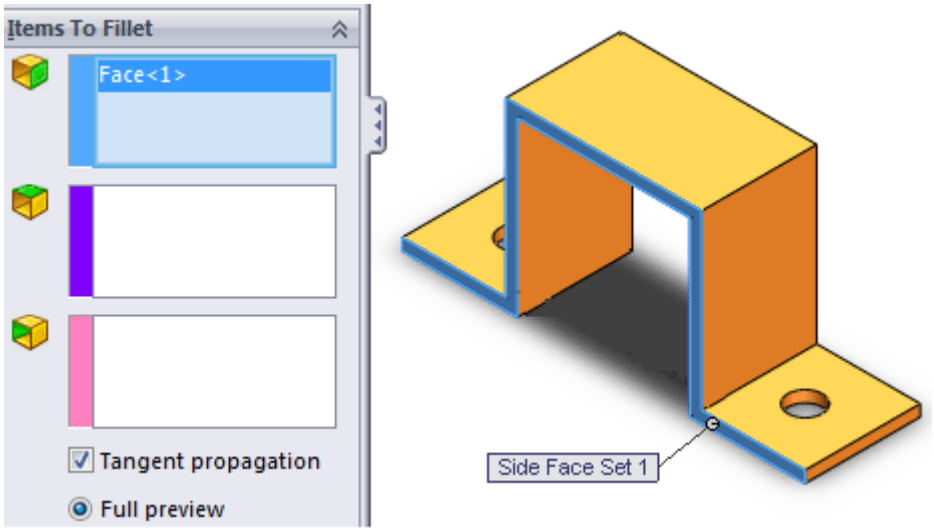


10. Click **Fillet** 

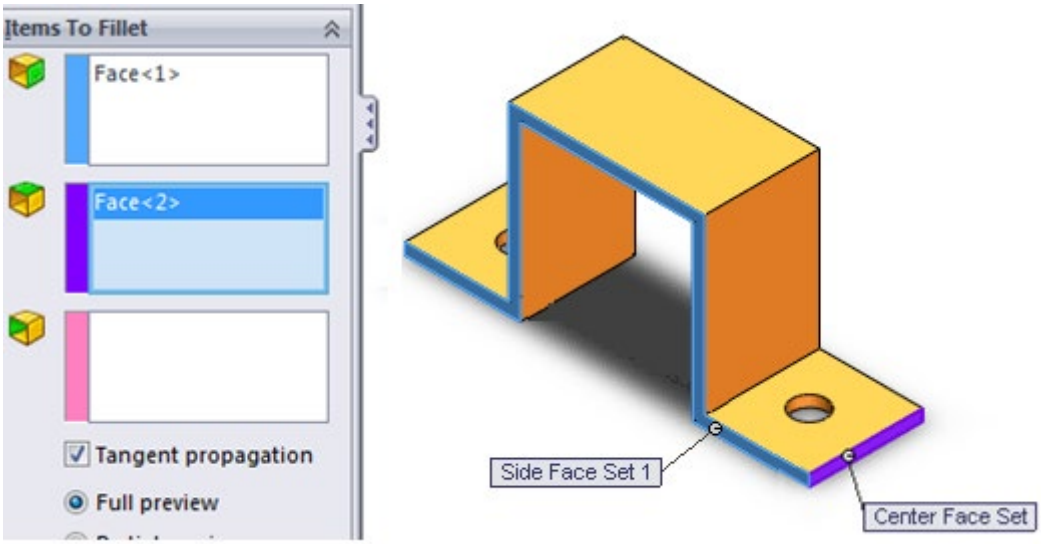
, check box **Full round fillet**.



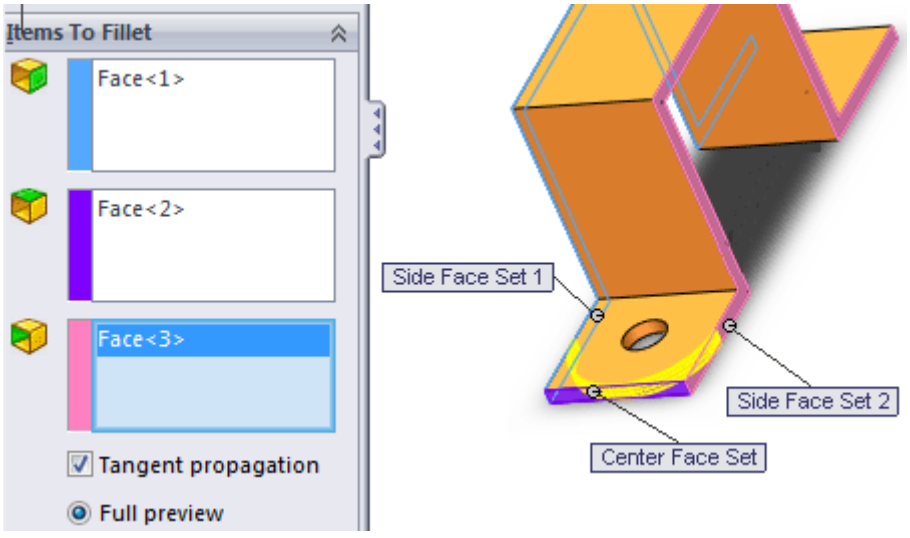
11. Click side left side face as **Side Face 1**.

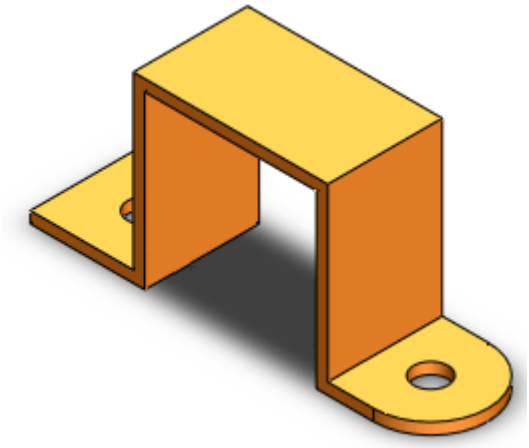


12. Click on purple box and click center face as **Center Face Set**.

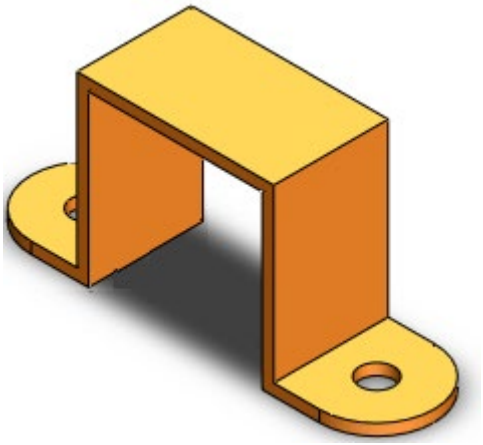


13. Click on pink box and click right side face as **Side Face Set2** and **OK**.

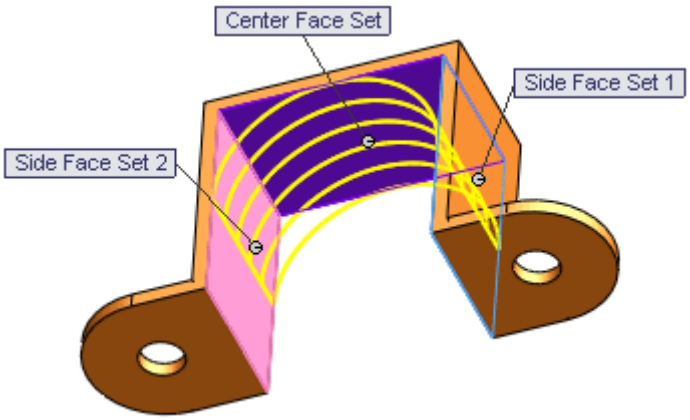


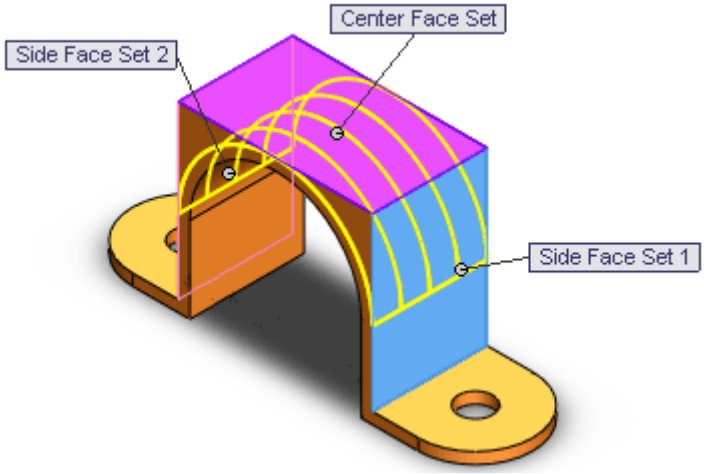


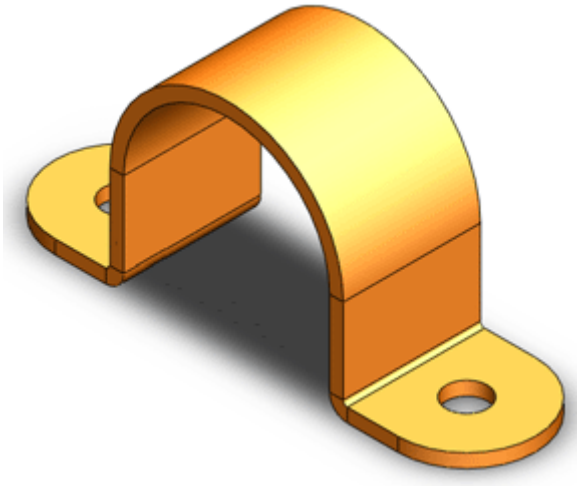
14. **Repeat step 11 – 13** for the other side.

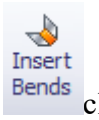


15. **Repeat step 11 – 13** for inner face and outer face of U bracket.

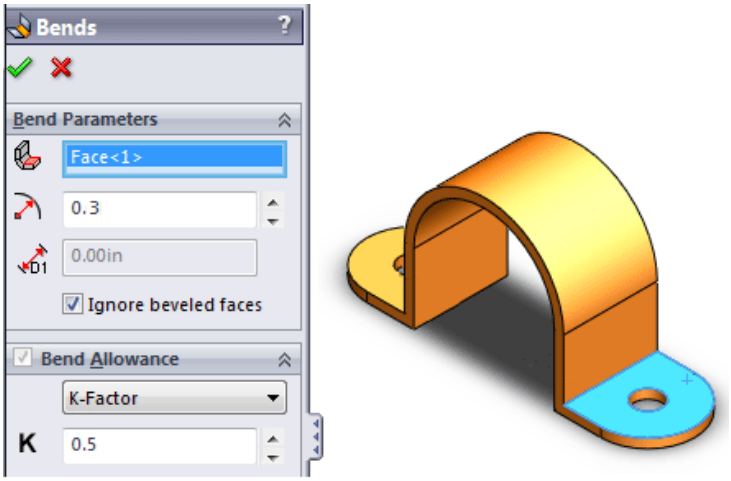




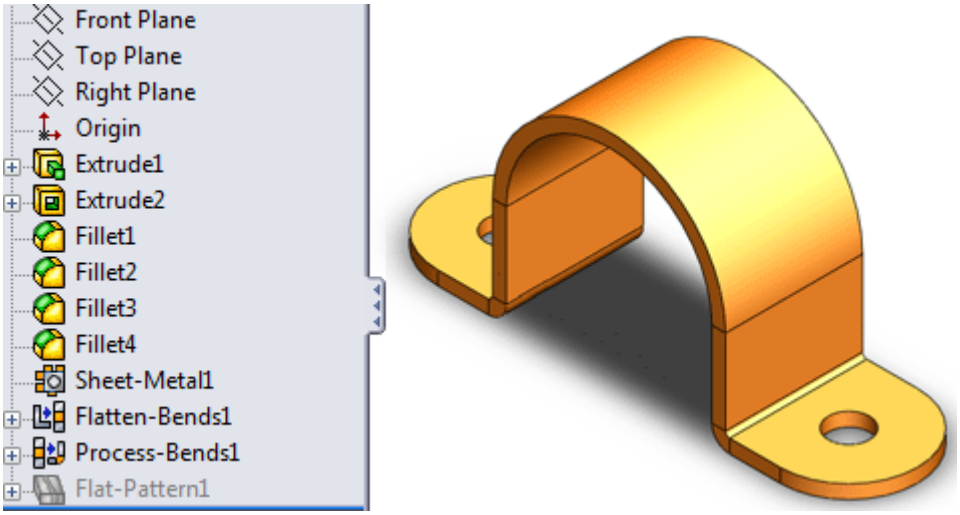


16. Click **Sheetmetal>Insert Bends**, 

click flat face as reference when it flatten. Set bend radius to **0.3mm** and **K factor 0.5** and **OK**.

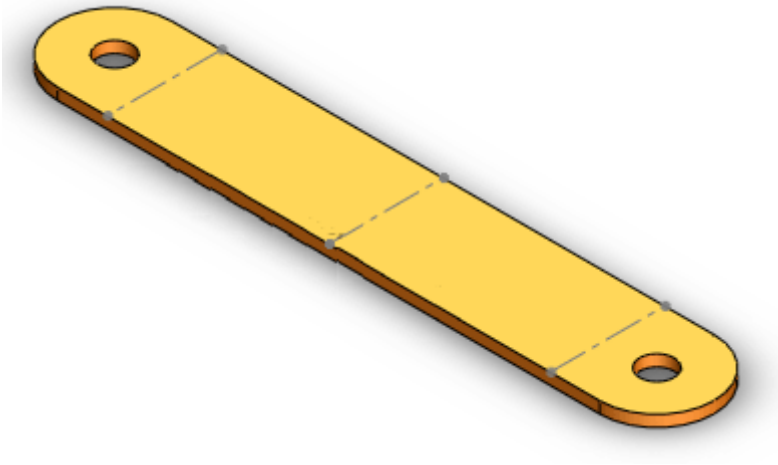


17. Your simple sheetmetal bend is ready. Look at part tree.

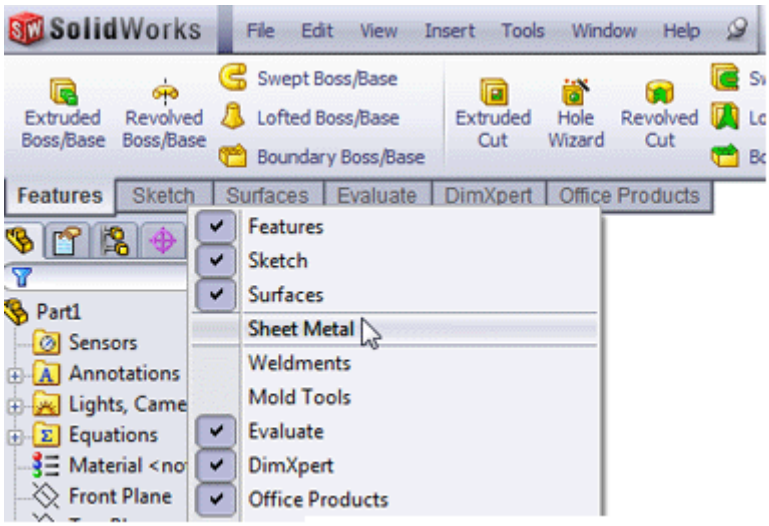


18. To view this part in flatten form click **Sheetmetal>Flatten** 

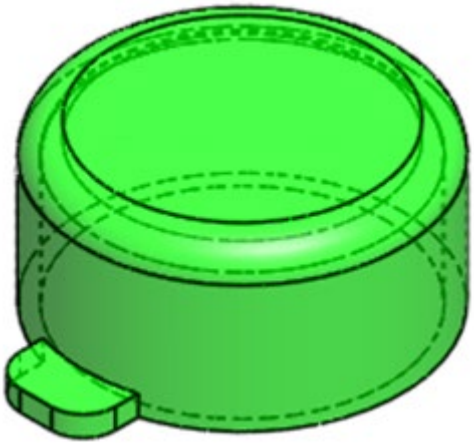
.



Note:If you cannot find the sheetmetal tool in you main tool menu, you can right click on main menu tab and check Sheetmetal option.

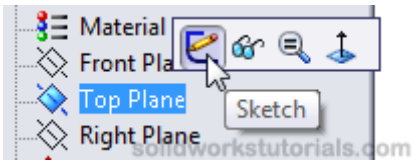


15. How to create bottle cap

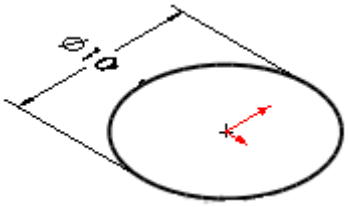


1. Click **New** , Click **Part** and **OK**.

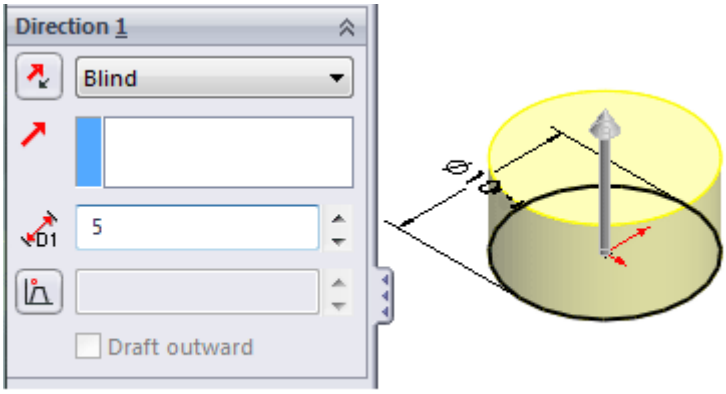
2. Click on **Top Plane** and click **Sketch**.



3. Click **Circle** and sketch start at Origin, click **Smart Dimension** and dimension the circle as **10mm** diameter.



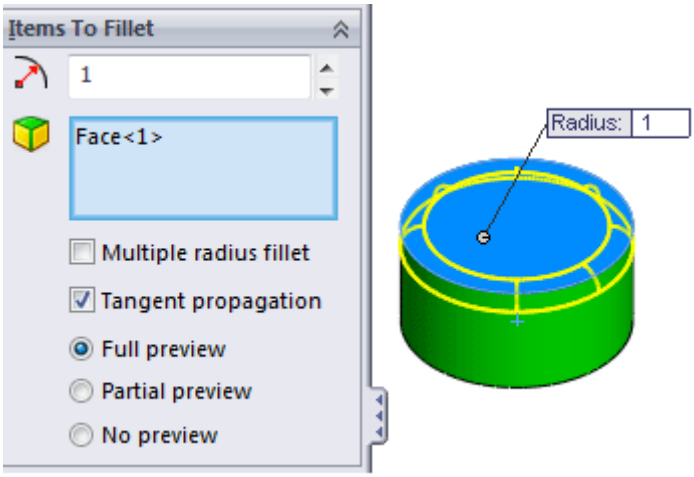
4. Click **Features>Extrude Boss/Base** set the D1 to **5mm.**



and ✔.

5. Click **Fillet** 

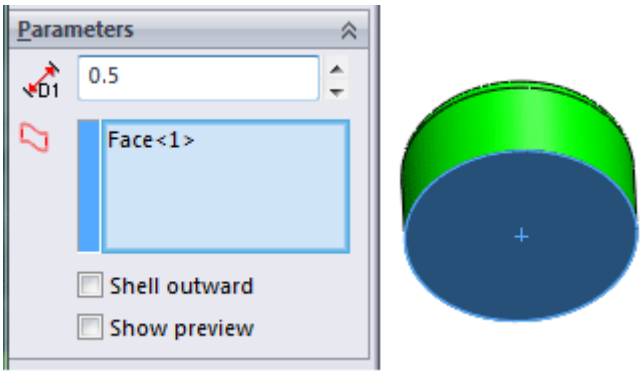
, set fillet size as **1mm**, select **top face** of the part



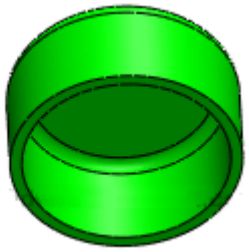
and ✔.

6. Turn the part to view bottom side, set D1 as **0.5mm**, click **Shell** 

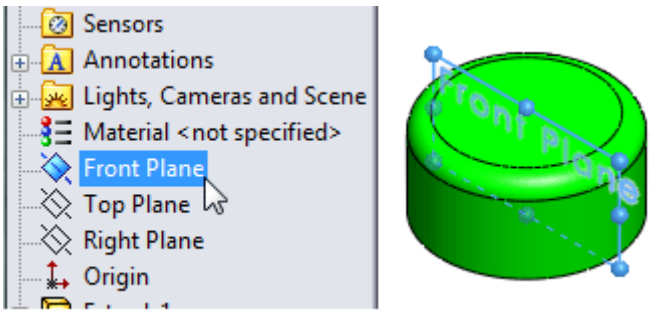
, select **bottom face**



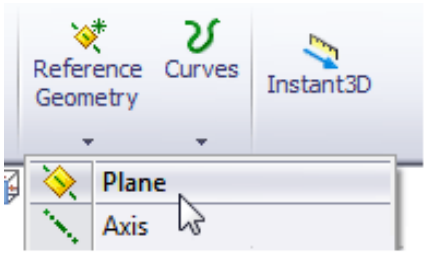
and ✔.



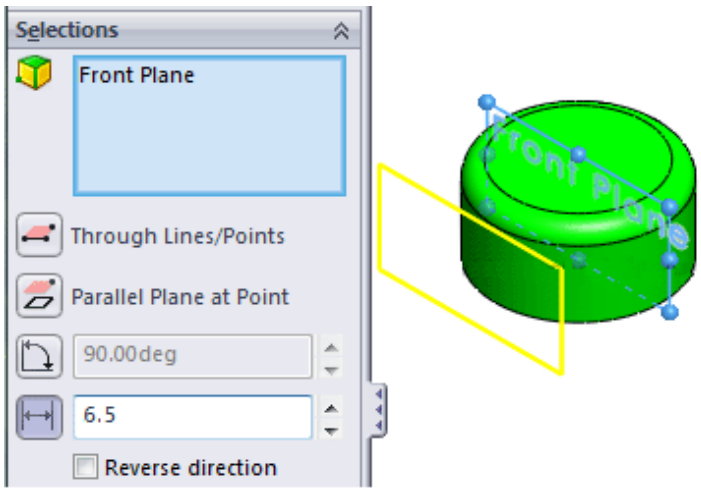
7. Click **Isometric View** , click on **Front Plane**



and click on **Reference Geometry>Plane**.

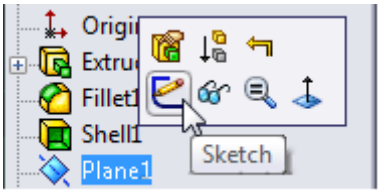


Set distance to **6.5mm**

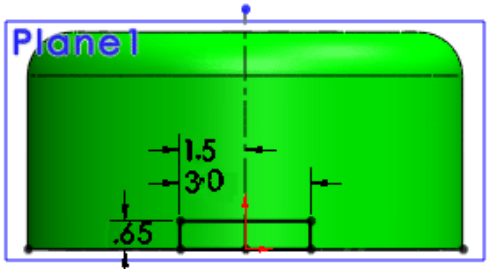


and .

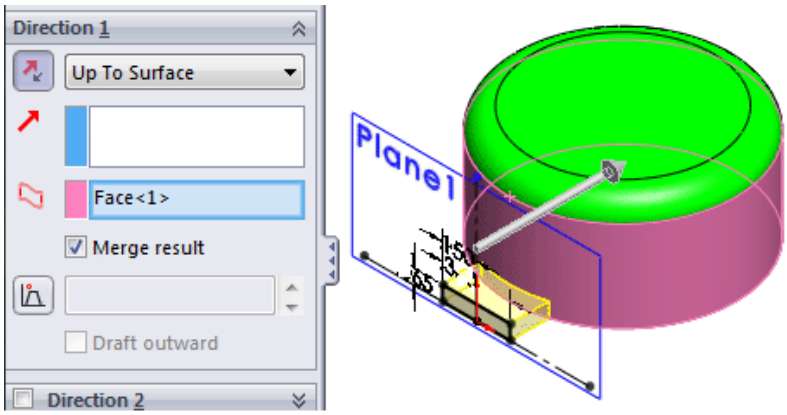
8. Click **Plane1** and click **Sketch**.



9. Click **Rectangle** , sketch on **Plane1** as sketched below and use **Smart Dimension** for your dimensioning.



10. Click **Features>Extrude Boss/Base** set the **Up To Surface**



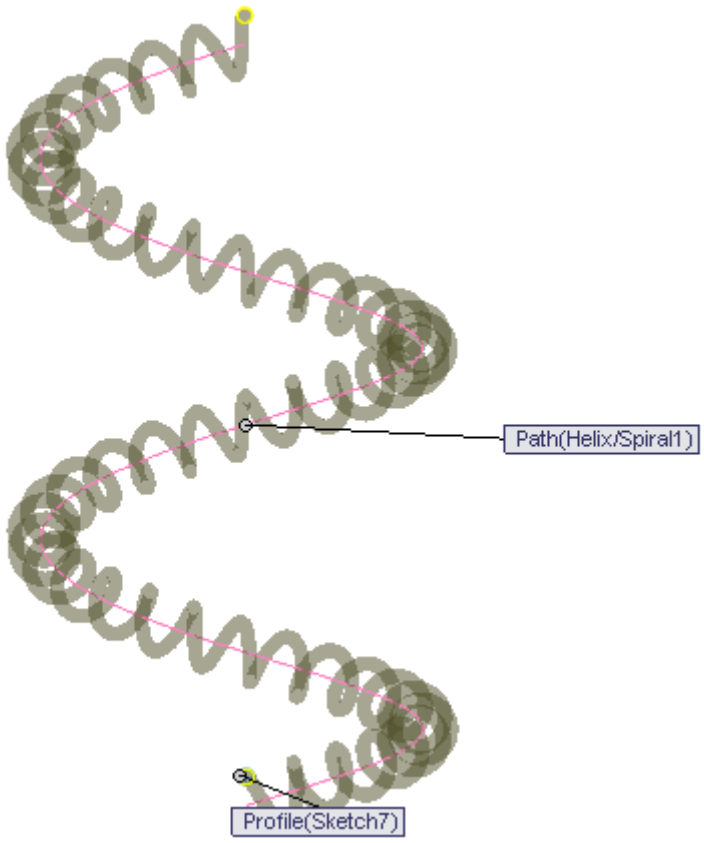
and .

11. Click **Fillet** , set fillet size as **1mm**, select **side edge of the lid**.

and .

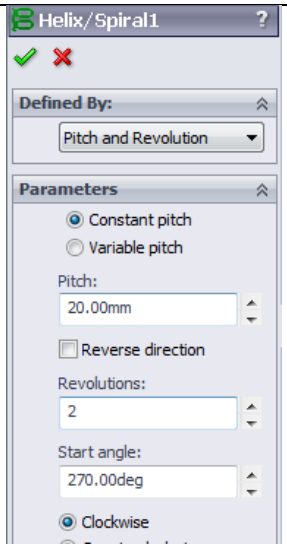
12. And you're done!

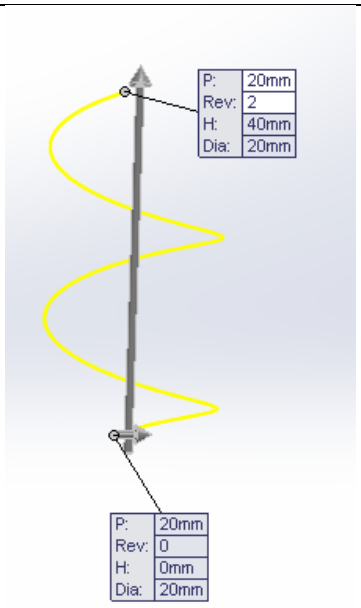
16. How to twist phone cord



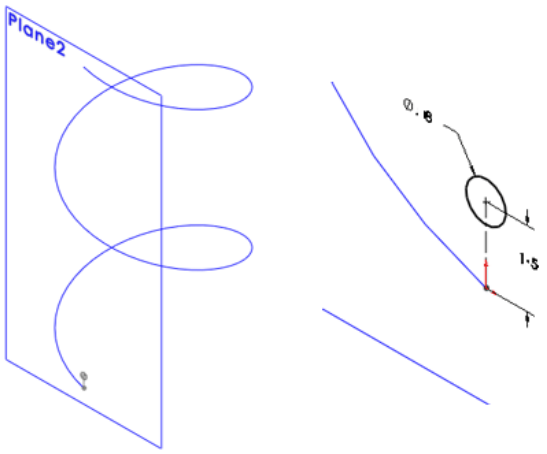
First you need to have spiral, with circle base 20mm, 2 revolutions and 20 pitch.

Refer to tutorial: How to create a simple spring helix/spiral from the right plane.

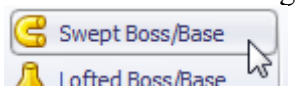




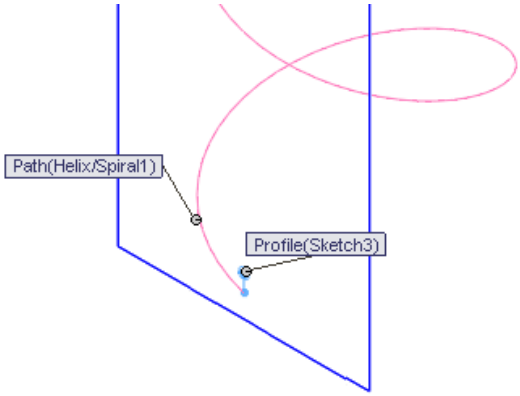
Now add a plane at end of spiral, select parallel to front plane.



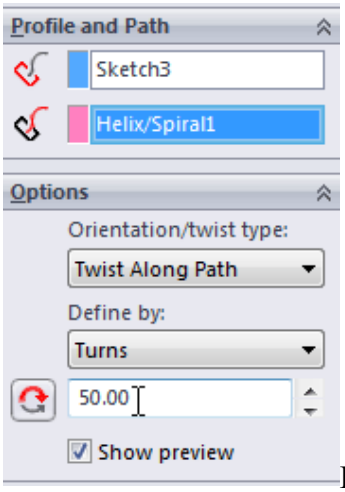
Sketch a circle on Plane2, 0.8mm and 1.5mm height. Click Swept Boss/Base.



Select Sketch3 as profile and Helix/Spiral1 as path.

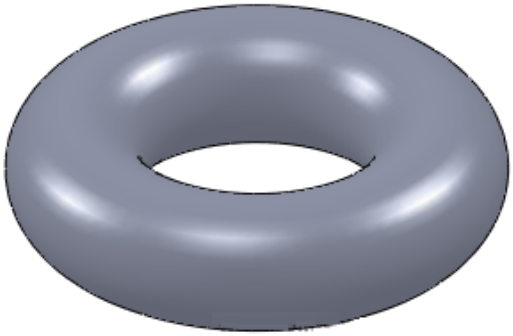


Open up Options and set Twist Along Path, define by Turns and 50 turns.



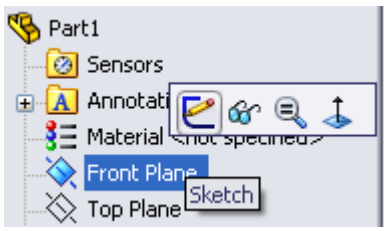
Done!

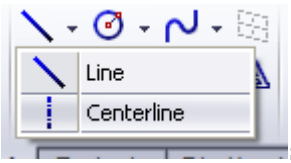
17. How to use Revolved Boss/Base



1. Click **New**.  Click **Part**,  **OK**.

2. Click **Front Plane** and click on **Sketch**.

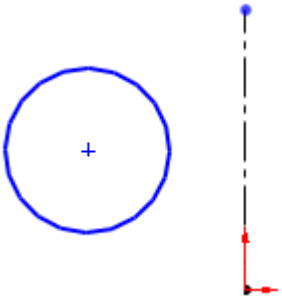


3. Select **centerline**, 

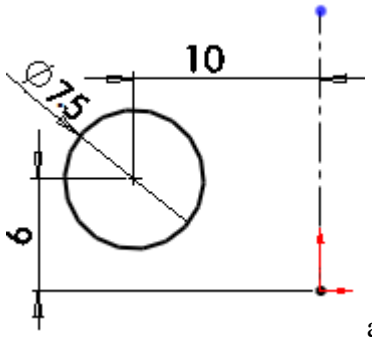
sketch vertical line start from origin, roughly

15mm. and **OK**. 

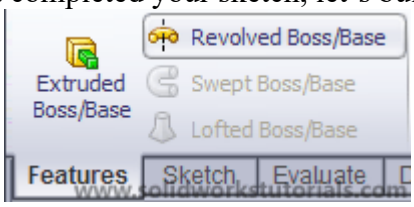
4. Click **circle**  and sketch a circle on left side of the centerline.

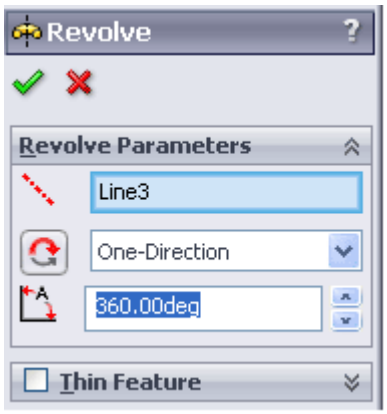


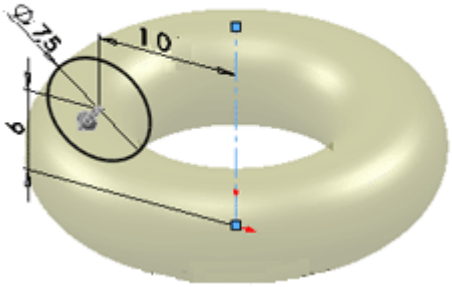
5. Click **Smart Dimension**, 

click sketched circle and set it diameter to **7.5mm** and add dimension for it location as below sketch 

and OK. 

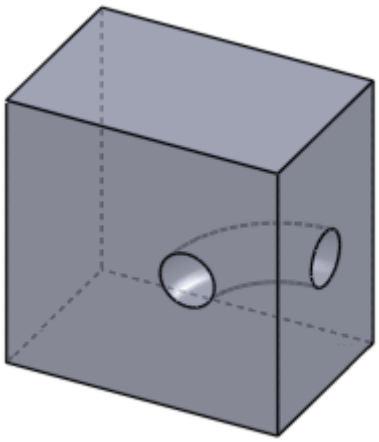
6. You just completed your sketch, let's build feature from it. Click **Feature>Revolved Boss/Base** 



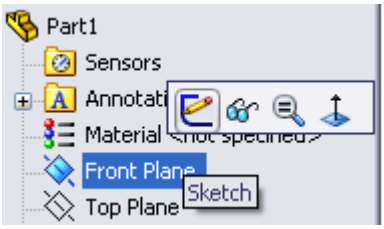


7. Click centerline as axis **OK**. 

18. How to use Revolved Cut



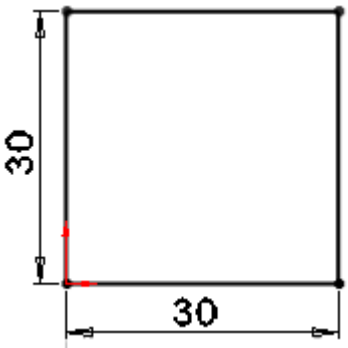
1.Click **New**.  Click **Part**,  **OK**.

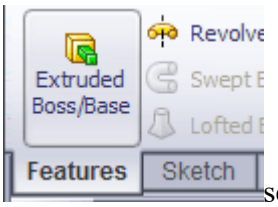
2.Click **Front Plane** 

and click on **Sketch**.

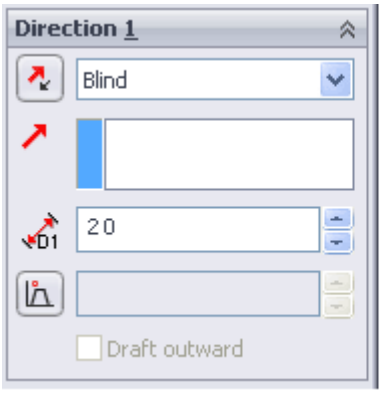
3.Click **Rectangle**,  sketch rectangular. Click **Smart Dimension**, 

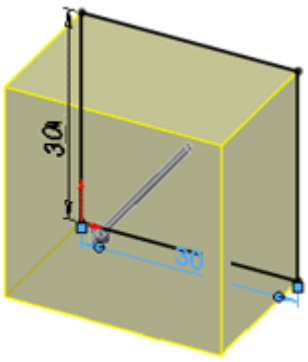
dimension rectangular **30mm x 30mm**.



4.Click **Feature>Extruded Boss/Base**, 

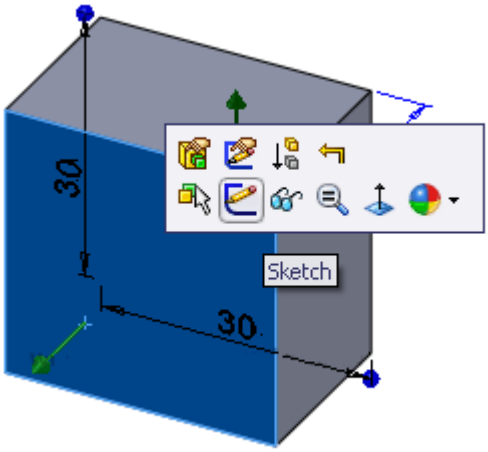
set **D1** to **20mm.**



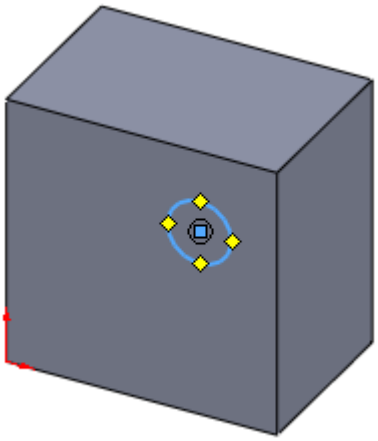


and  .

5.Click on front face and click **Sketch**.

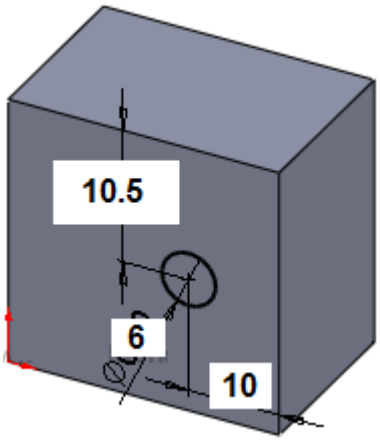


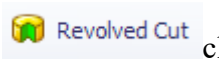
6. Click **Circle**,  and sketch a circle on front face.



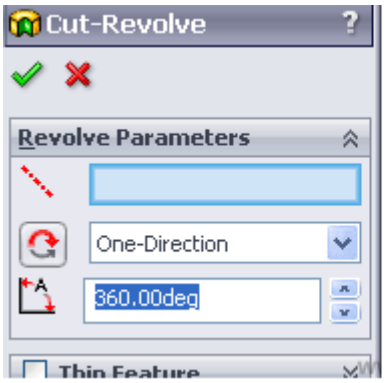
7. Click **Smart Dimension**, 

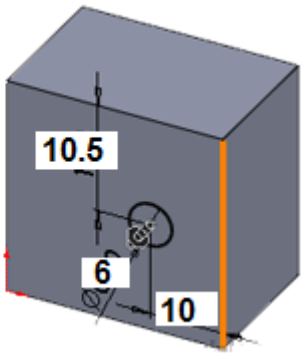
dimension sketch as below sketched.

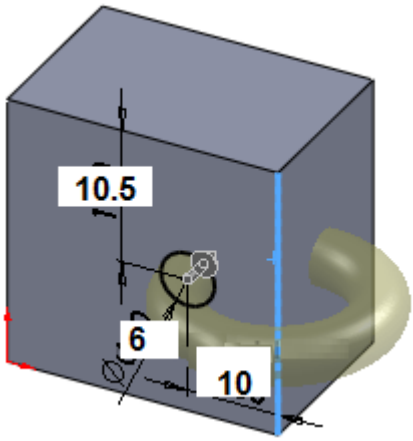


8.Click **Features**>**Revolved Cut** 

click on **right side edge** as axis of revolution,



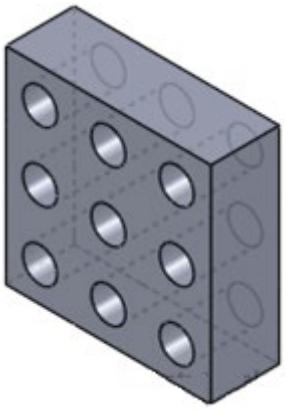




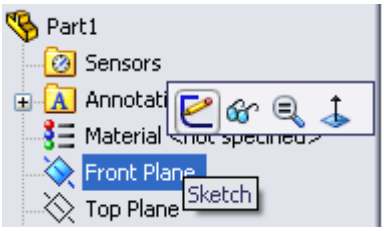
and  .

9

19. How to use Linear Pattern



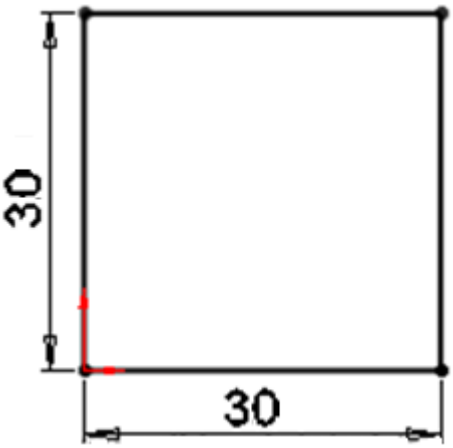
1.Click **New**.  Click **Part**,  **OK**.

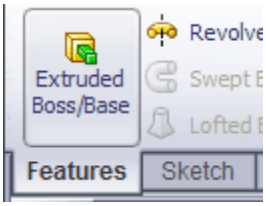
2.Click **Front Plane** 

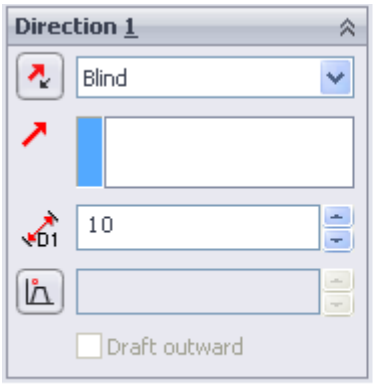
and click on **Sketch**.

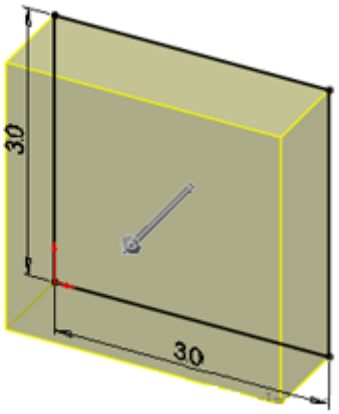
3.Click **Rectangle**,  sketch rectangular. Click **Smart Dimension**, 

dimension rectangular 30mm x 30mm

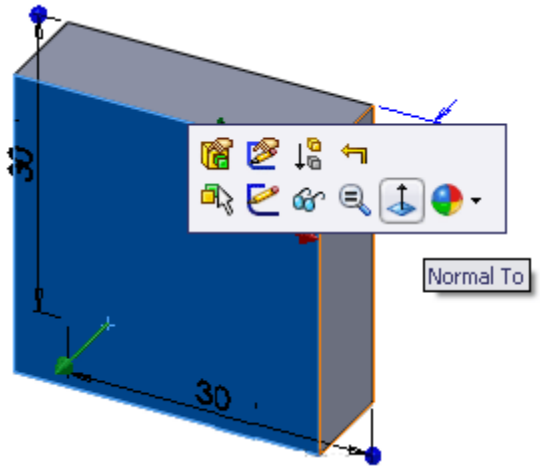
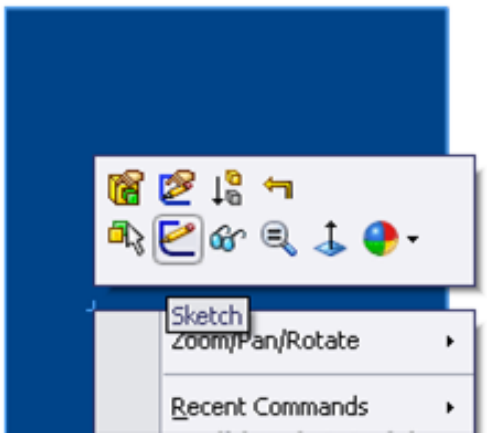


4.Click **Feature>Extruded Boss/Base**, 

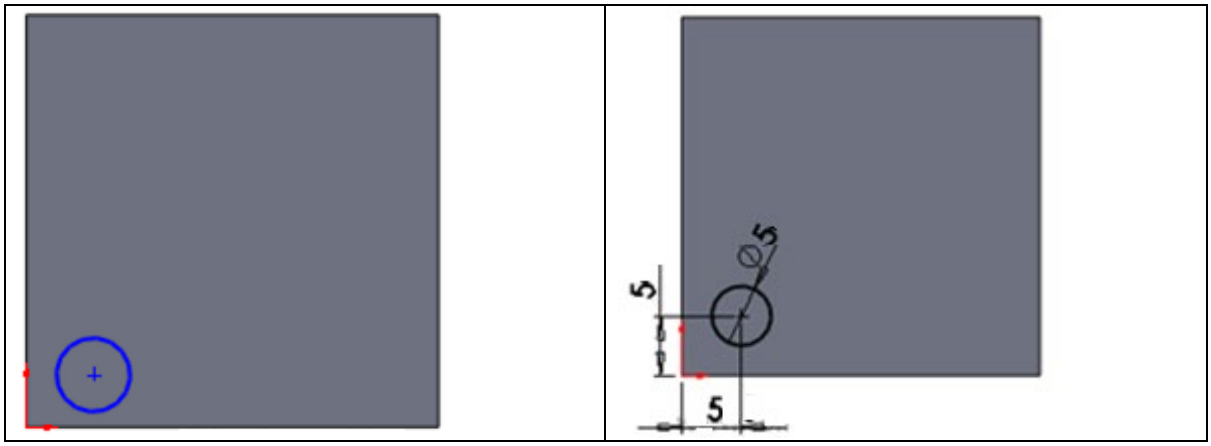




set **D1** to **10mm** and **OK**.

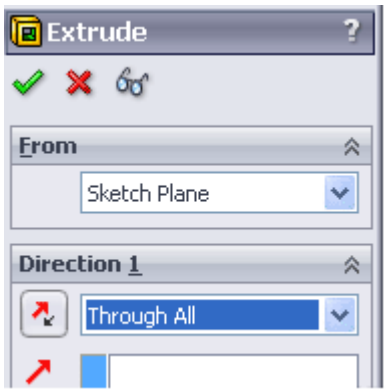
5. Click on front face and select **Normal to**.	6.Click front face and Insert Sketch.
	

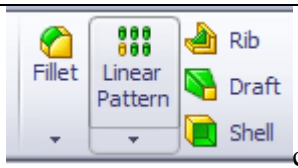
7.Click Circle, sketch circle at one edge.	8.Click Smart Dimension, dimension circle as below sketch.



9.Click Features>Extruded Cut,

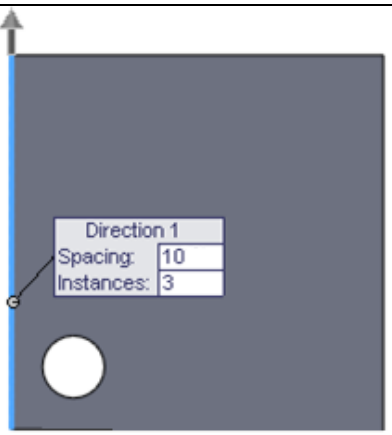
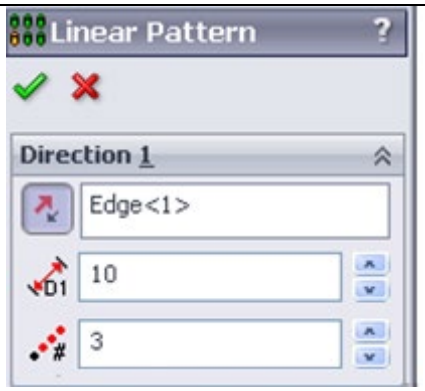
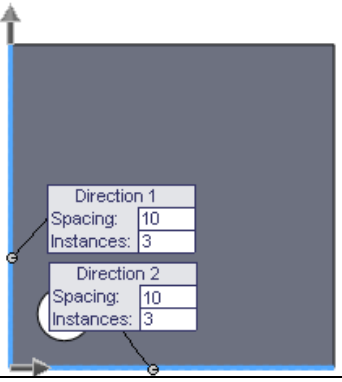
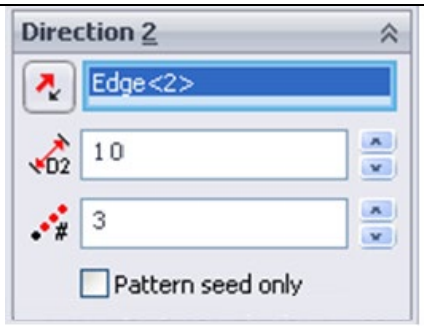
set Direction 1, Through All and OK.



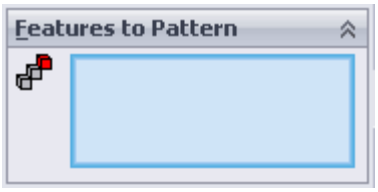
10.Click Linear Pattern, 

click left edge,

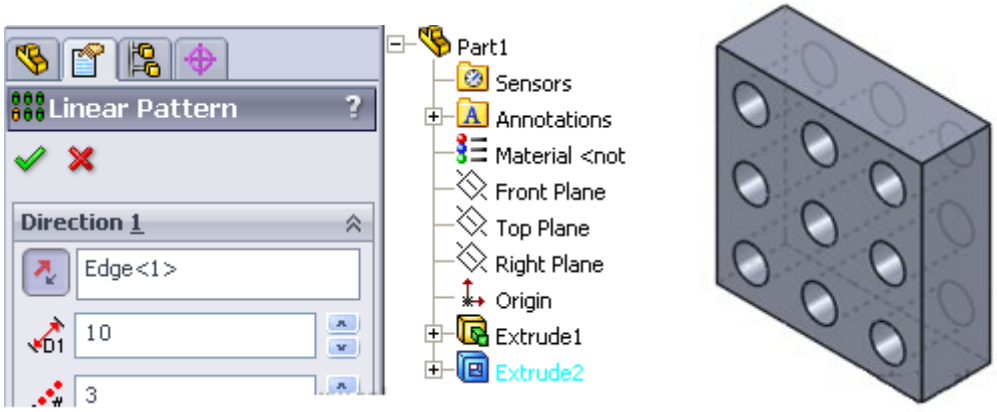
on Direction 1 set spacing D1 to 10mm and Instances # to 3.

	
11.Click bottom edge,	on Direction 2 set spacing D2 to 10mm and Instances # to 3.
	

12.Click inside white box Features to Pattern.

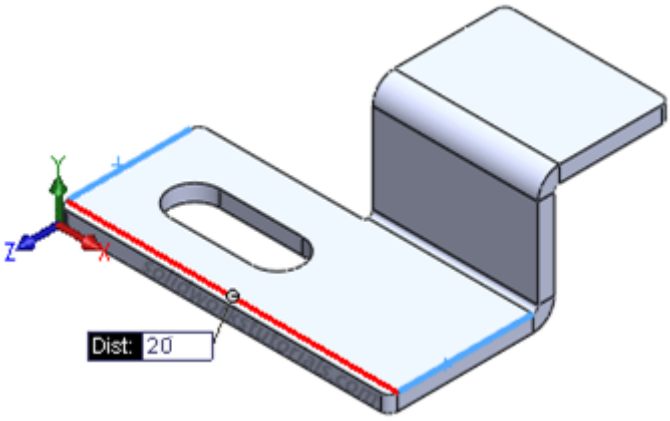


Open up part tree, select Extrude 2



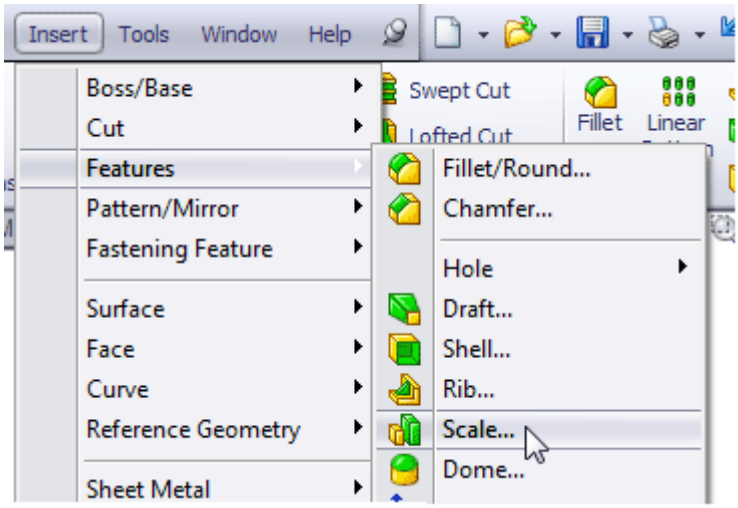
and OK.  .You're done!

20. How to use Scale

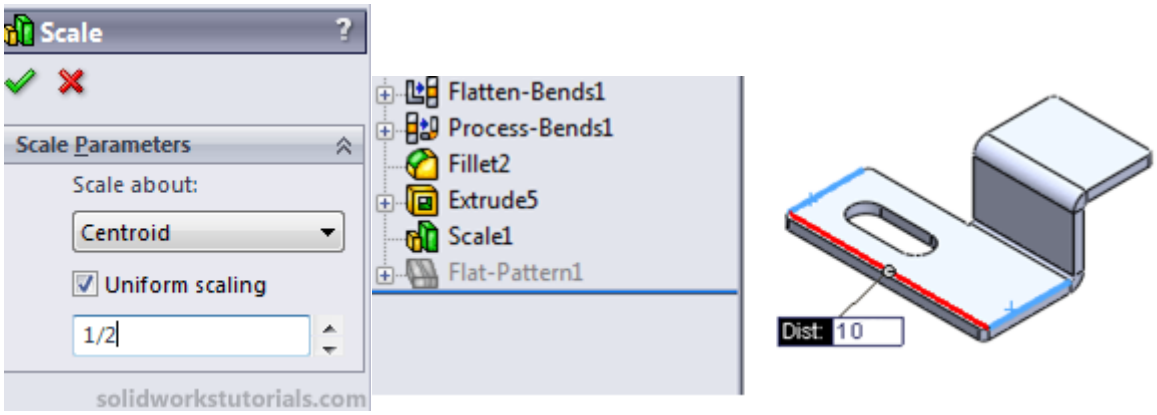


Scale down to 1/2 …

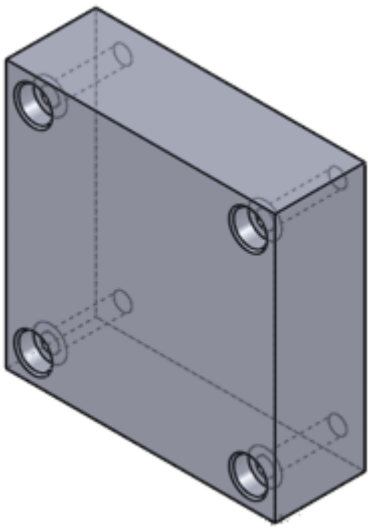
First click Insert>Features>Scale…



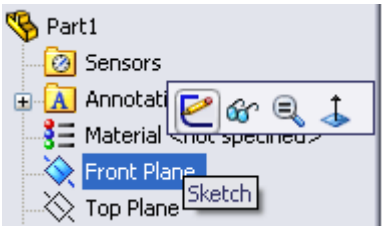
Scale down your part 1/2 of it original size and OK.



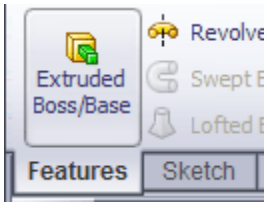
21. How to use Hole Wizard

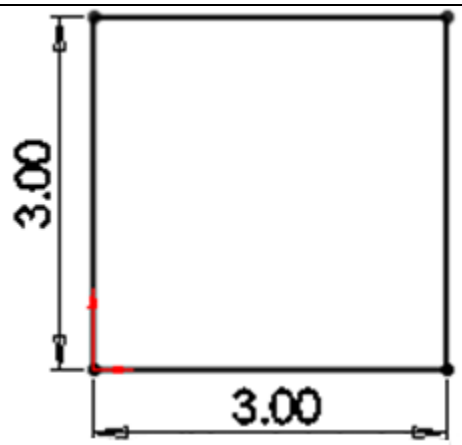
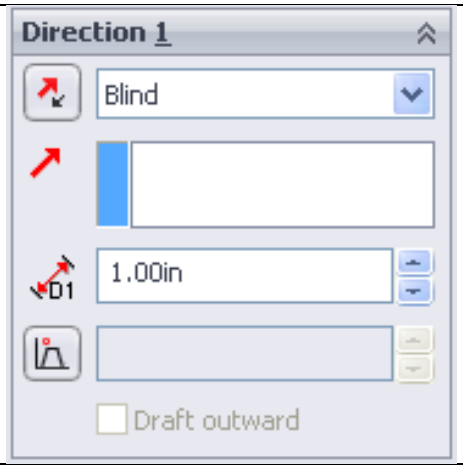
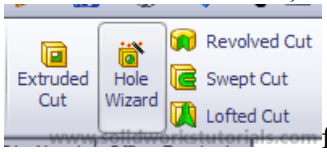


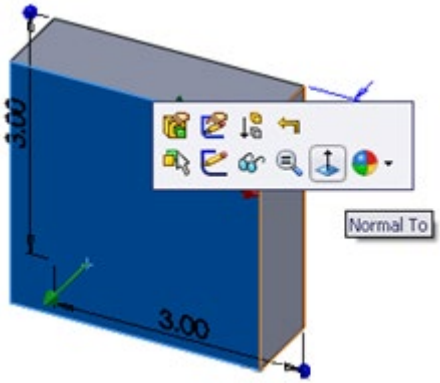
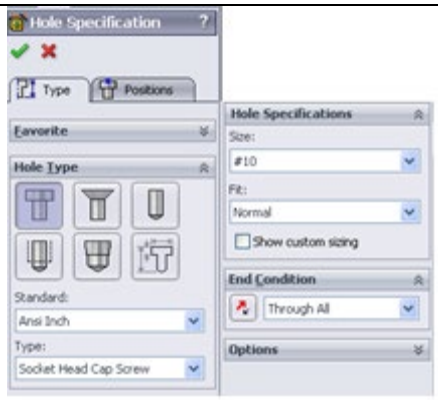
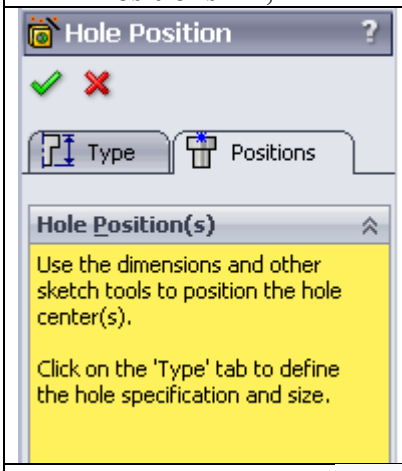
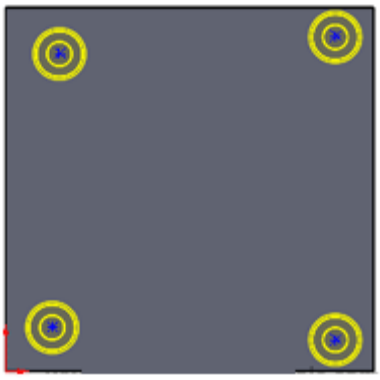
1.Click **New**.  Click **Part**,  **OK**.

2.Click **Front Plane** 

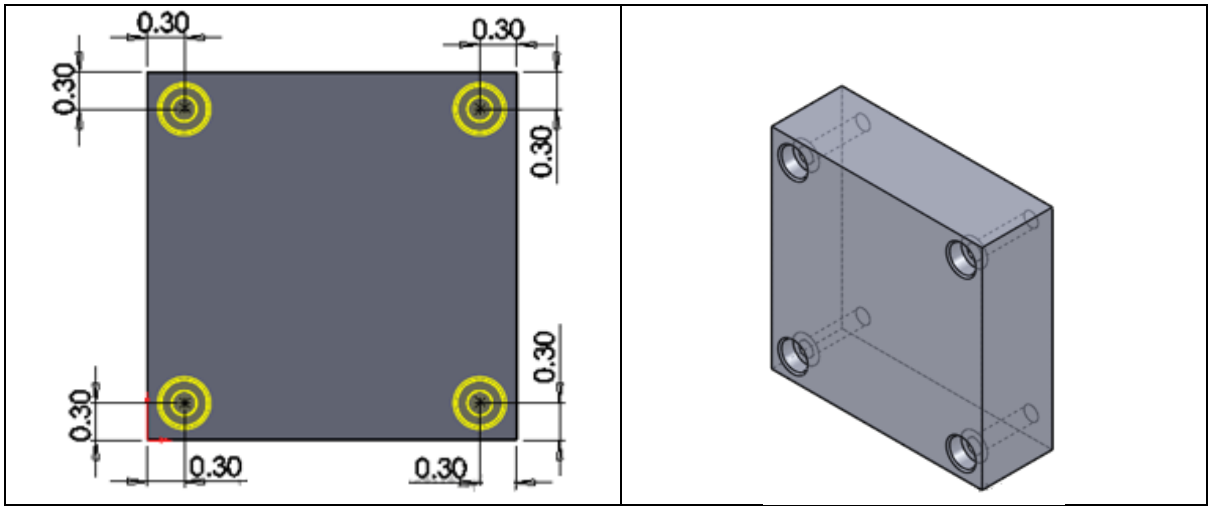
and click on **Sketch**.

3.Click **Rectangle**,  sketch rectangular. Click **Smart Dimension**,  dimension rectangular 3.0in x 3.0in.	4.Click **Feature>Extruded Boss/Base**,  set **D1** to **1.0in** and **OK**. 

<table>
<tr><td> 
</td><td> 
</td></tr>
<tr><td>5. Click on front face and select Normal to.</td><td>6. Click Hole Wizard,
 

for Hole Type select Counterbore, Standard ANSI Inch, Type Socket Head Cap Screw and Size #10.</td></tr>
<tr><td> 
</td><td> 
</td></tr>
<tr><td>Click Positions tab,</td><td>click 4 points at edges,</td></tr>
<tr><td> 
</td><td>  </td></tr>
<tr><td>click Smart Dimension 

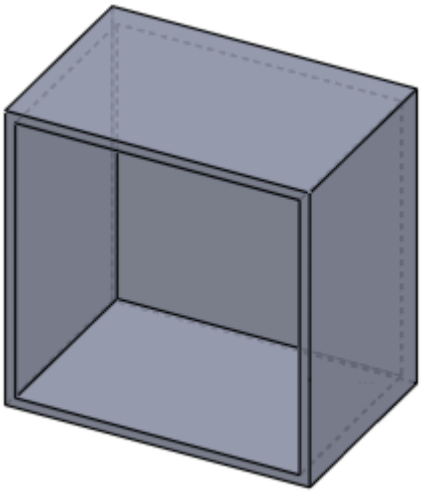
and dimension all 4 points 0.3in from edge.</td><td>Click OK.  </td></tr>
</table>



22. How to use Shell

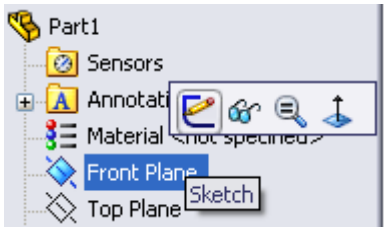
In this tutorial, you will create this part using shell 

feature tools.



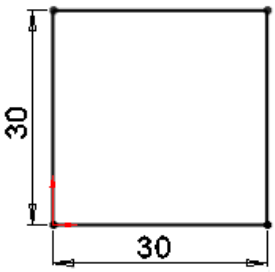
1.Click **New**.  Click **Part**, 

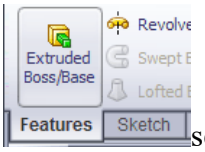
OK.

2.Click **Front Plane** 

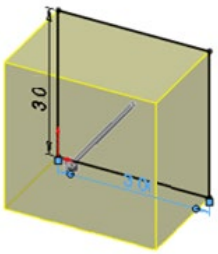
and click on **Sketch**.

3.Click **Rectangle**,  sketch rectangular. Click **Smart Dimension**,  dimension rectangular **30mm x 30mm**.



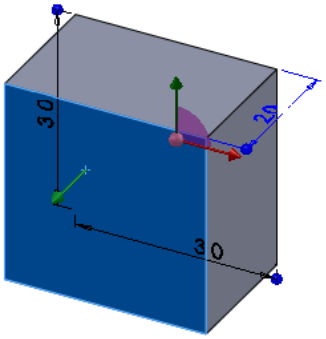
4.Click **Feature>Extruded Boss/Base**, 

set **D1** to **20mm**

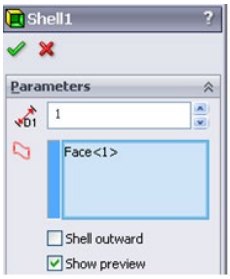


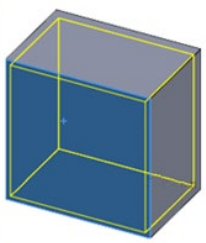
and  .

5.Click on front face,

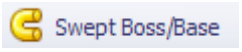


and click **Shell**  on shell **Parameter**, set **D1** to **1mm,** check **show preview** and  .

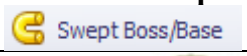
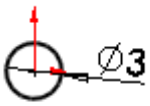
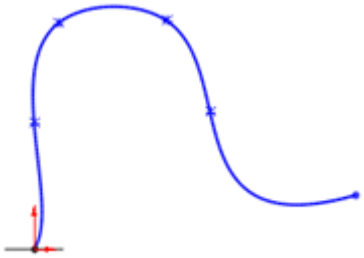
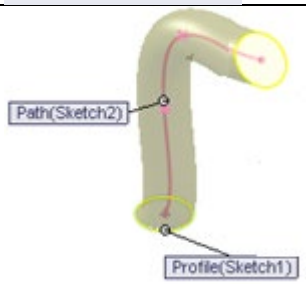




23. How to use Swept Boss/Base

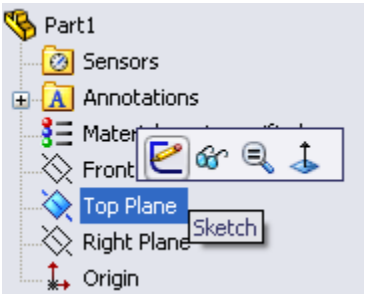
In this tutorials you will create swept part using Swept Boss/Base tool. 

Overview:

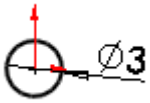
Top Plane	Front Plan , **Spline** 	**Features>Swept Boss/Base** 
		

1.Click **New**.  Click **Part**,  **OK**.

2.Click **Top Plane** and click on **Sketch**.



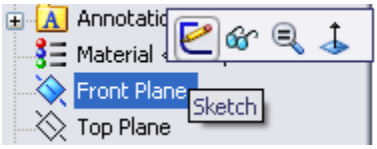
3.Click **Circle**  and sketch a circle origin as it center. Click **Smart Dimension**, 



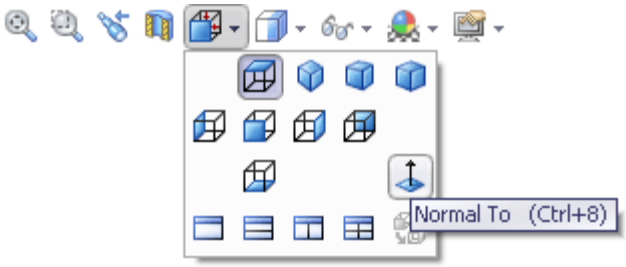
dimension sketched circle diameter as **3mm.**.

4.**Exit Sketch**. 

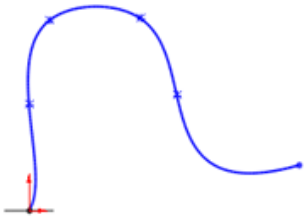
5.Click on **Front Plane** and click **Sketch**.



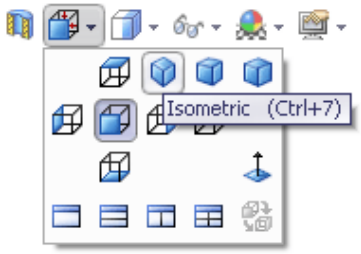
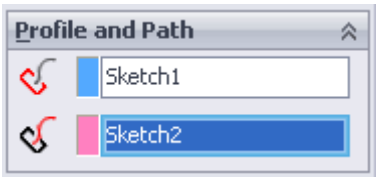
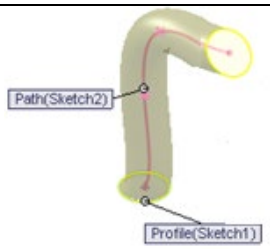
6.Click on **View Orientation**>**Normal To**.



7.Zoom out the sketch, click on **Spline** and sketch a curve as sketched below.



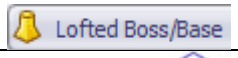
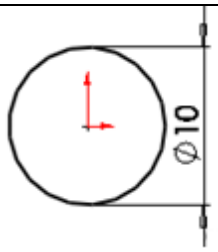
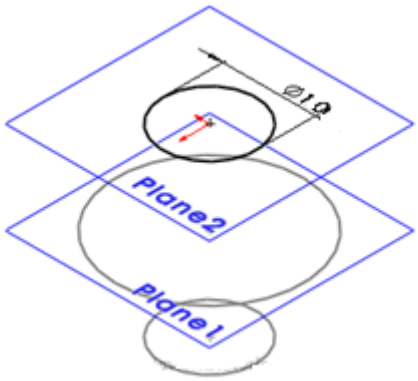
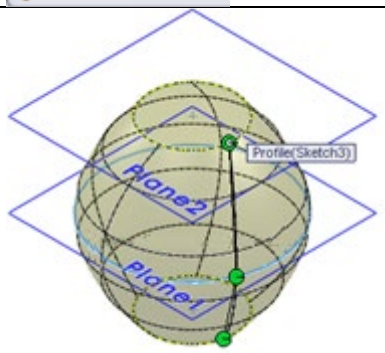
8.**Exit Sketch**.

9.Click **View Orientation**>**Isometric**. 	10. Click **Features**>**Swept Boss/Base**. Swept Boss/Base For swept profile select **Sketch1** (circle) and for path click on **Sketch2** (curve).  and .
	

24. How to use Lofted Boss/Base

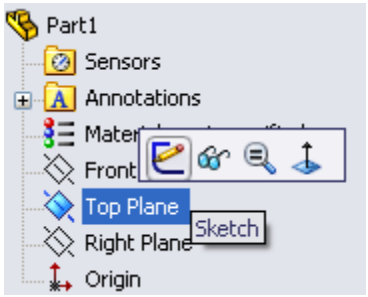
In this tutorial you will create this part using loft feature

Overview:

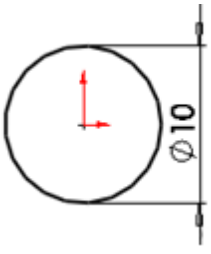
Top Plane	Create two plane 1 & plane 2	Lofted Boss/Base 
		

1. Click **New**.  Click **Part**,  **OK**.

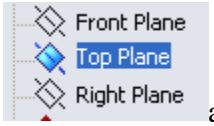
2. Click **Top Plane** and click on **Sketch**.



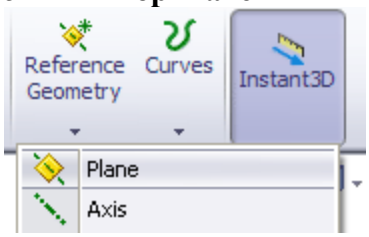
3. Click **Circle**,  sketch a circle start at origin. Click **Smart Dimension**,  click on circle and set dimension to **10mm**.

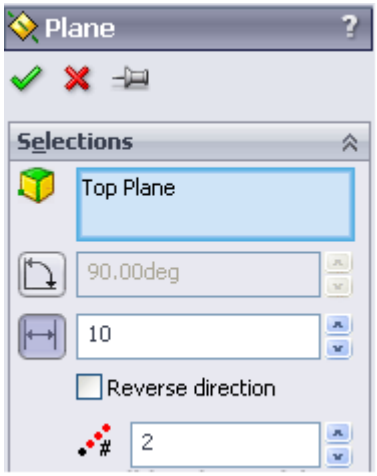


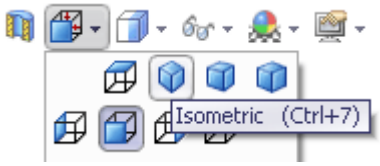
4. Exit sketch. 

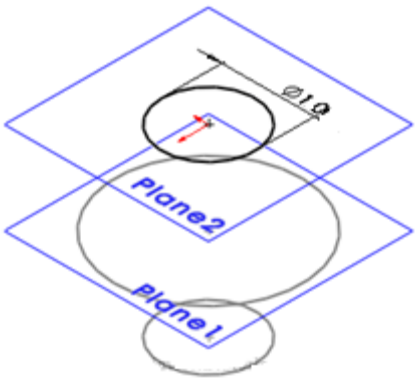
5. Click **Top Plane** 

and click **Features**>**Reference Geometry**>**Plane**.

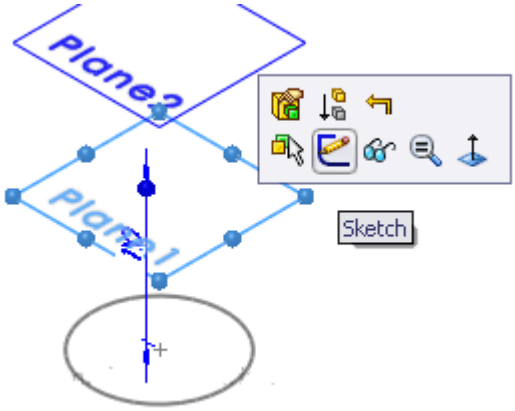




Set distance to **1**0mm apart, set # to **2** and OK.  Two more plane added, click View **Orientation**>**Isometric** 

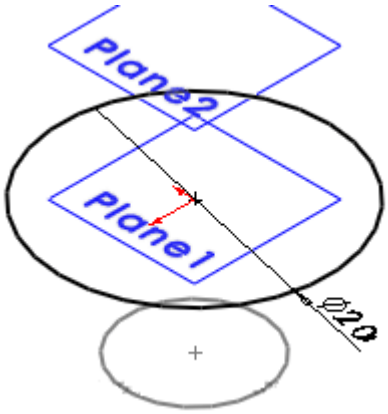


Plane 1 and Plane2.



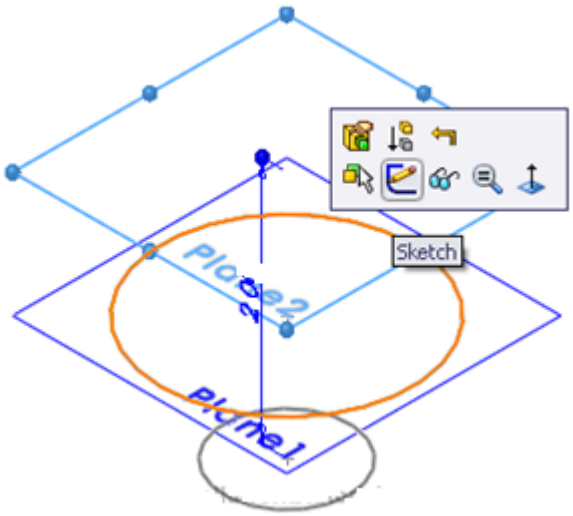
6. Click on **Plane 1**, click **Sketch**.

7. Click **Circle**, sketch a circle start at origin. Click **Smart Dimension**, click on circle and set dimension to **20mm**.

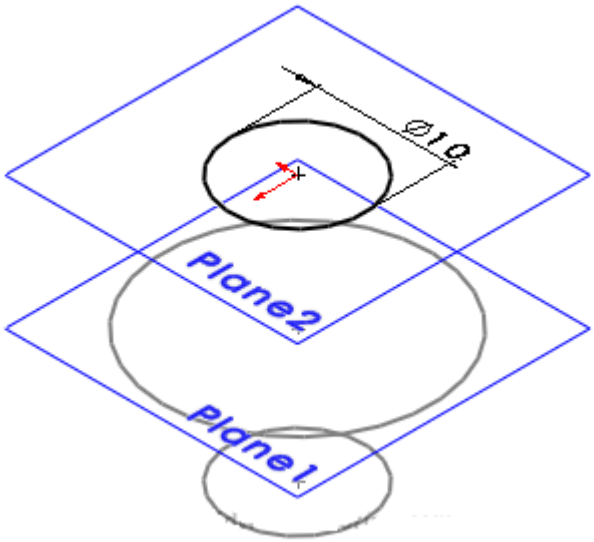


Exit sketch.

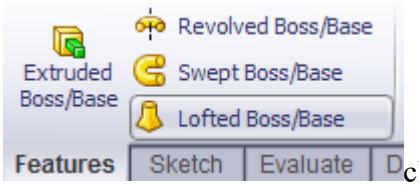
8. Click on **Plane 2**, click **Sketch**.



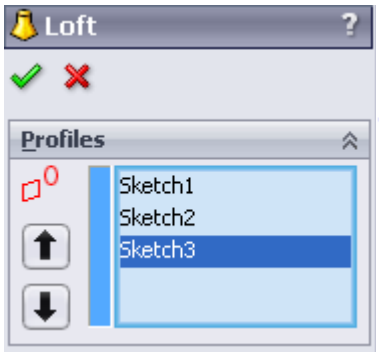
9. Click **Circle**,  sketch a circle start at origin. Click **Smart Dimension**,  click on circle and set dimension to **10mm**.

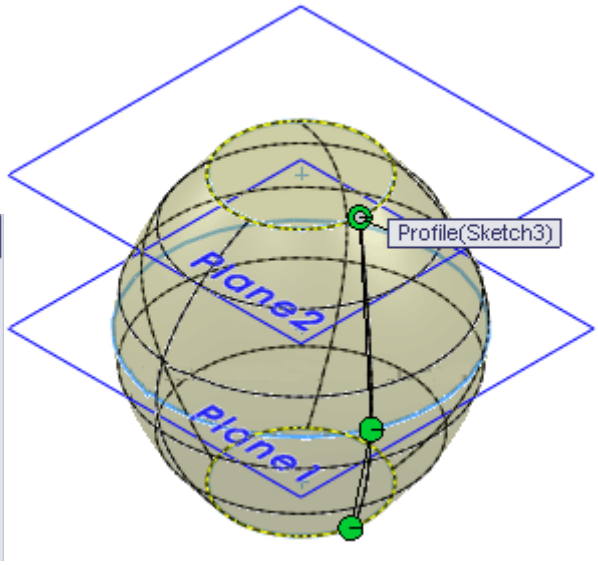


Exit sketch. 

10. Click **Features**>**Lofted Boss/Base** 

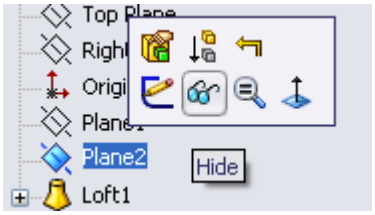
click bottom circle, middle circle, top circle





and **OK**. 

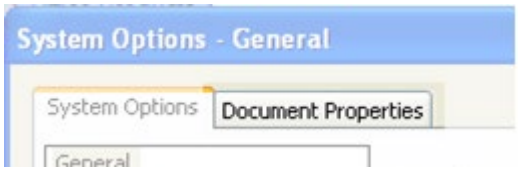
11. Hide **Plane 1**, 

hide **Plane 2**. 

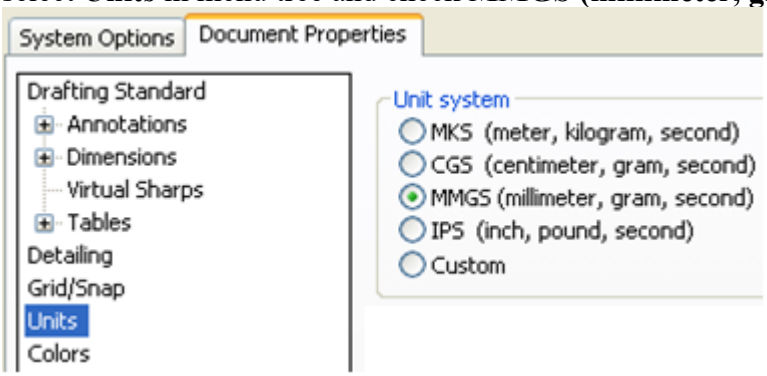
12. Done.

25. How to change to metric units

First click **Option** on top of main menu, 

open **Document Properties** tab, 

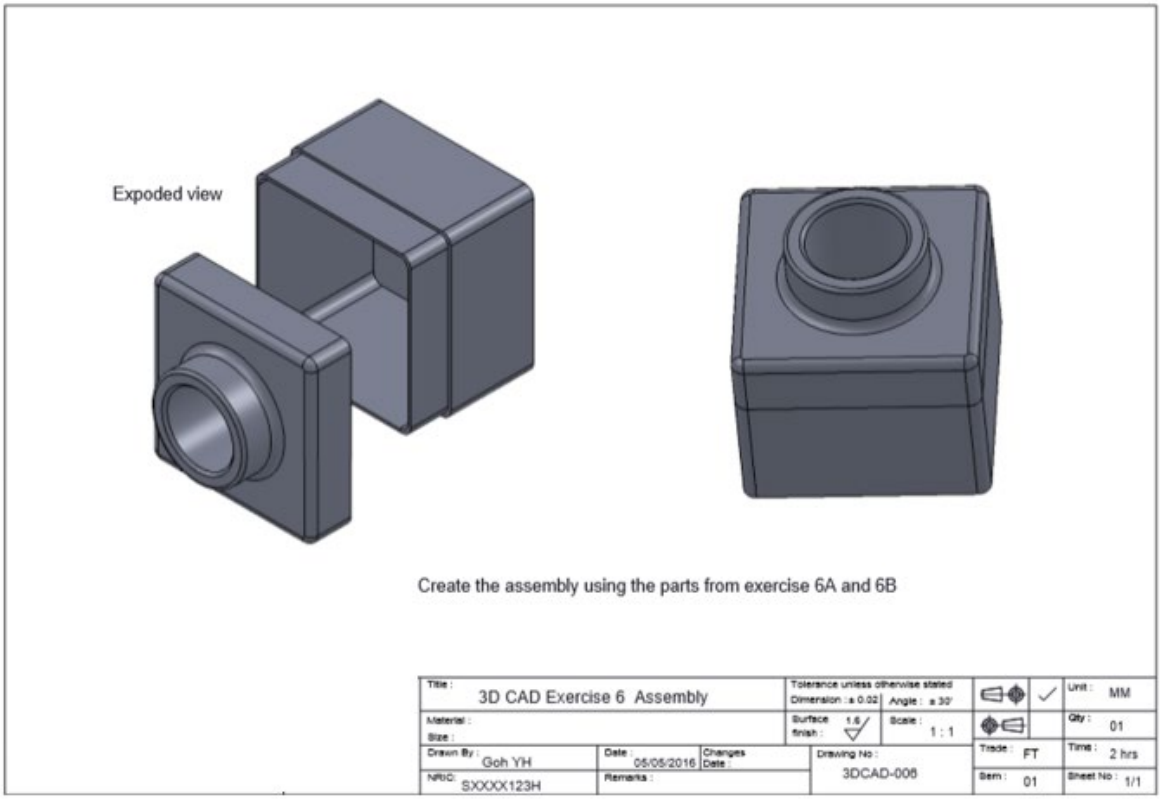
select **Units** in menu tree and check **MMGS (millimeter, gram, second)**.

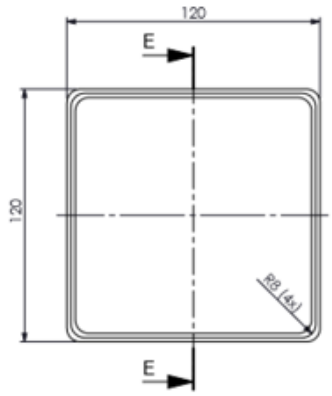


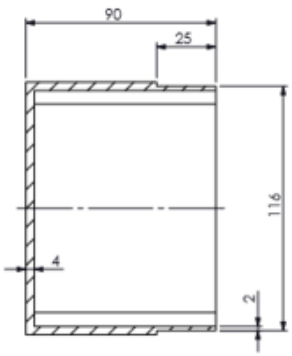
Ok, done!

11_SOLIDWORKS CAD ASSEMBLY

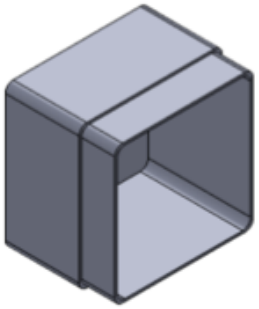
11.1. 3D CAD ASSEMBLY 1





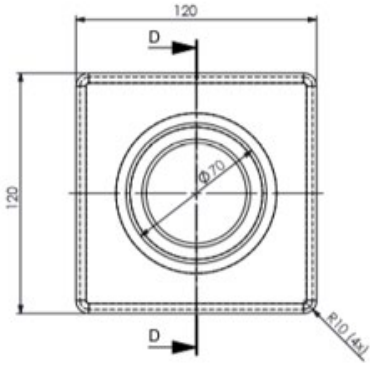


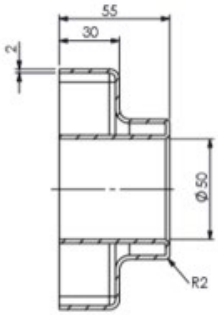
SECTION E-E



Q1 What is the mass of the part in grams?

Title : 3D CAD Exercise 6B			Tolerance unless otherwise stated Dimension : ± 0.02 Angle : ± 30'		Unit : MM
Material : ABS Plastics Size : 120 x 120 x 90 mm			Surface finish : 1.6	Scale : 1 : 1	Qty : 01
Drawn By : Goh YH	Date : 05/05/2016	Changes Date :	Drawing No : 3DCAD-06B		Trade : FT / Time : 2 hrs
NRIC: SXXXX123H	Remarks :				Sem : 01 / Sheet No : 1/1





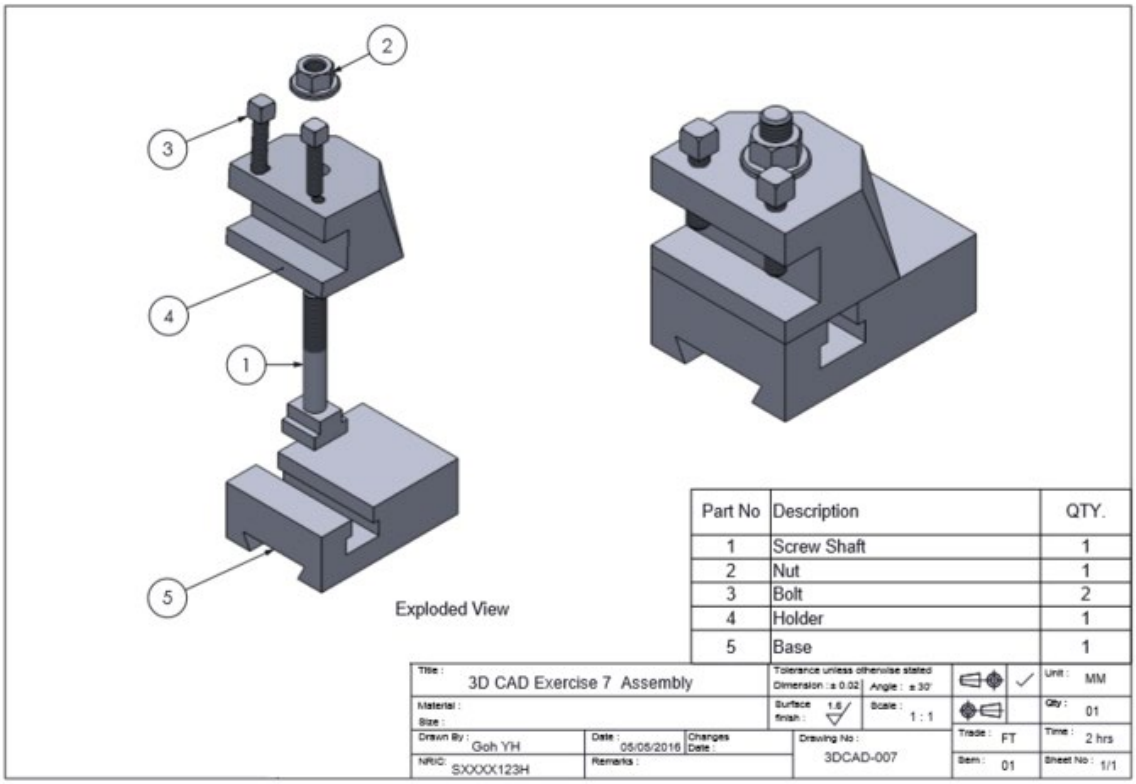
SECTION D-D

Q1 What is the mass of the part in grams?

All fillets Radius 5 mm unless stated

Title : 3D CAD Exercise 6A			Tolerance unless otherwise stated Dimension : ± 0.02 Angle : ± 30'		Unit : MM
Material : ABS Plastics Size : 120 x 120 x 50 mm			Surface finish : 1.6	Scale : 1 : 1	Qty : 01
Drawn By : Goh YH	Date : 05/05/2016	Changes Date :	Drawing No : 3DCAD-06A		Trade : FT / Time : 2 hrs
NRIC: SXXXX123H	Remarks :				Sem : 01 / Sheet No : 1/1

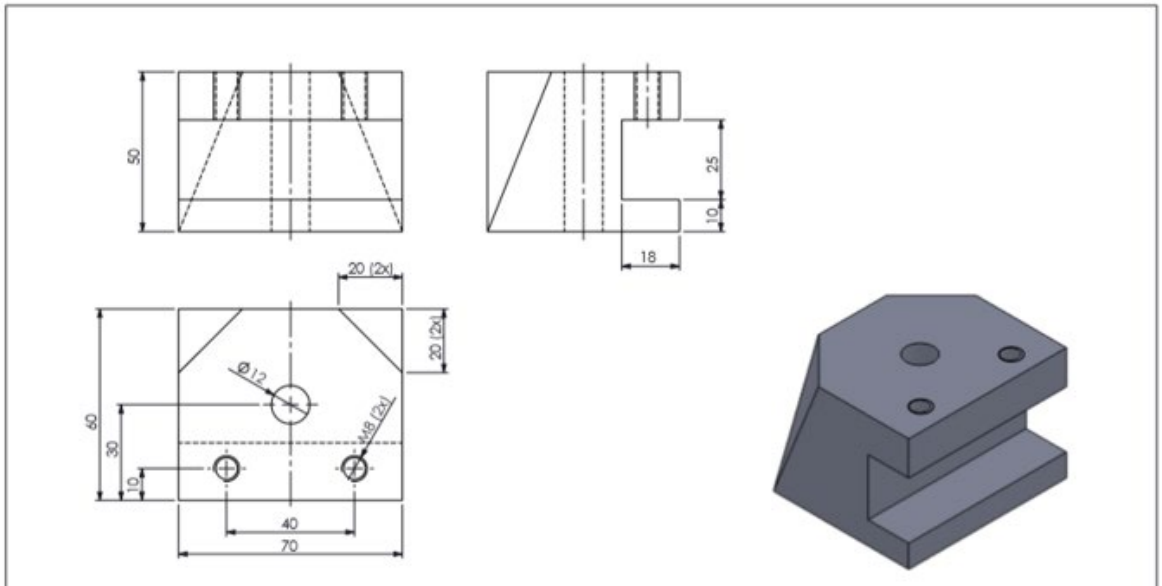
11.2. 3D CAD ASSEMBLY 2



Exploded View

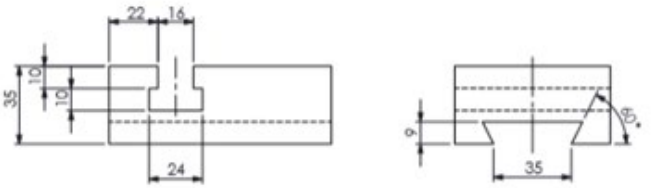
Part No	Description	QTY.
1	Screw Shaft	1
2	Nut	1
3	Bolt	2
4	Holder	1
5	Base	1

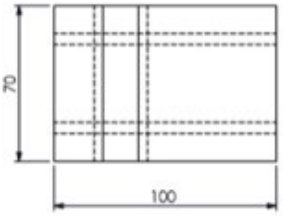
Title : 3D CAD Exercise 7 Assembly			Tolerance unless otherwise stated Dimension : ± 0.02 Angle : ± 30'		Unit : MM
Material : Size :			Surface finish : 1.6	Scale : 1 : 1	Qty : 01
Drawn By : Goh YH	Date : 05/05/2016	Changes Date :	Drawing No : 3DCAD-007	Trade : FT	Time : 2 hrs
NRIC: SXXXX123H	Remarks :			Sem : 01	Sheet No : 1/1

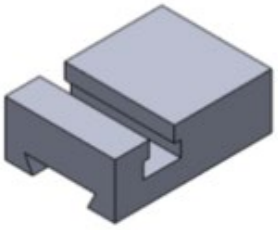


Q1 What is the mass of the part in grams?

Title : 3D CAD Exercise 7A			Tolerance unless otherwise stated Dimension : ± 0.02 Angle : ± 30'		Unit : MM
Material : Alloy Steel Size : 63 x 63 x 73 mm			Surface finish : 1.6	Scale : 1 : 1	Qty : 01
Drawn By : Goh YH	Date : 05/05/2016	Changes Date :	Drawing No : 3DCAD-07A	Trade : FT	Time : 2 hrs
NRIC: SXXXX123H	Remarks :			Sem : 01	Sheet No : 1/1

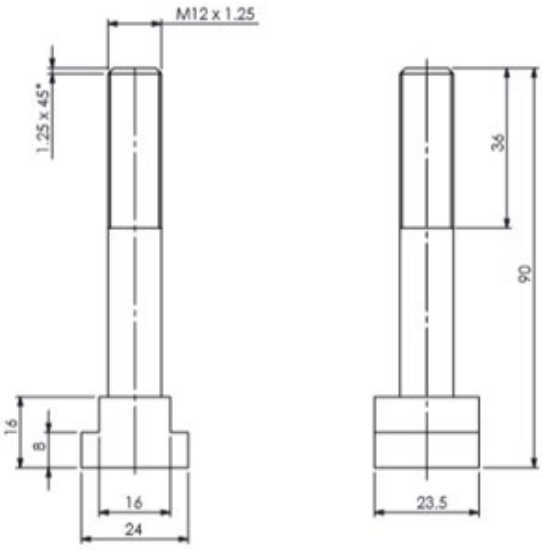






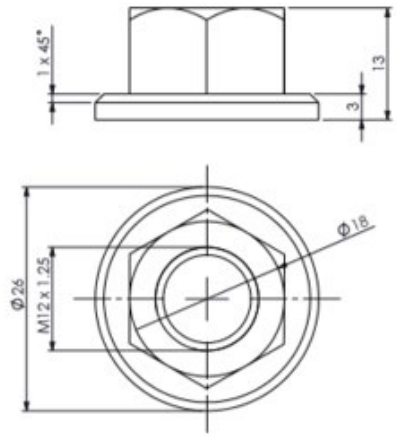
Q1 What is the mass of the part in grams?

Title: 3D CAD Exercise 7B			Tolerance unless otherwise stated Dimension: ± 0.02 Angle: ± 30'		Unit: MM
Material: Cast Alloy Steel Size: 103 x 73 x 38 mm			Surface finish: 1.6	Scale: 1 : 1	Qty: 01
Drawn By: Goh YH	Date: 05/05/2016	Changes Date:	Drawing No: 3DCAD-07B	Trade: FT	Time: 2 hrs
NRIC: SXXXX123H	Remarks:			Item: 01	Sheet No: 1/1



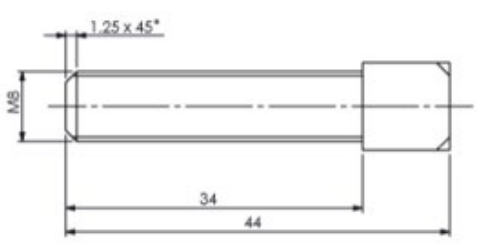
Q1 What is the mass of the part in grams?

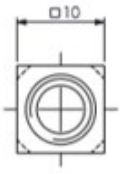
Title: 3D CAD Exercise 7C			Tolerance unless otherwise stated Dimension: ± 0.02 Angle: ± 30'		Unit: MM
Material: Alloy Steel Size: 27 x 27 x 93 mm			Surface finish: 1.6	Scale: 1 : 1	Qty: 01
Drawn By: Goh YH	Date: 05/05/2016	Changes Date:	Drawing No: 3DCAD-07C	Trade: FT	Time: 2 hrs
NRIC: SXXXX123H	Remarks:			Item: 01	Sheet No: 1/1



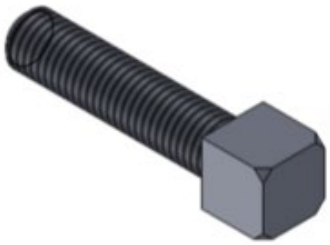
Q1 What is the mass of the part in grams?

Title: 3D CAD Exercise 7D			Tolerance unless otherwise stated: Dimension: ± 0.02, Angle: ± 30'		Unit: MM
Material: Alloy Steel			Surface finish: 1.6	Scale: 1 : 1	Qty: 01
Size: Φ28 x 16 mm					Trade: FT, Time: 2 hrs
Drawn By: Goh YH	Date: 05/05/2016	Changes Date:	Drawing No: 3DCAD-07D		Sem: 01, Sheet No: 1/1
NRIC: SXXXX123H	Remarks:				





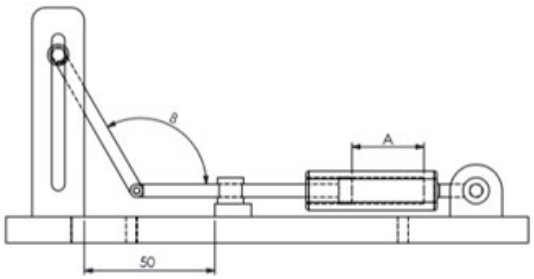
Vertex chamfer 1.5 mm

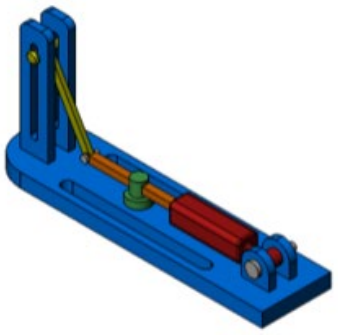


Q1 What is the mass of the part in grams?

Title: 3D CAD Exercise 7E			Tolerance unless otherwise stated: Dimension: ± 0.02, Angle: ± 30'		Unit: MM
Material: Alloy Steel			Surface finish: 1.6	Scale: 1 : 1	Qty: 01
Size: 12 x 12 x 47 mm					Trade: FT, Time: 2 hrs
Drawn By: Goh YH	Date: 05/05/2016	Changes Date:	Drawing No: 3DCAD-07E		Sem: 01, Sheet No: 1/1
NRIC: SXXXX123H	Remarks:				

11.3. 3D CAD ASSEMBLY 3





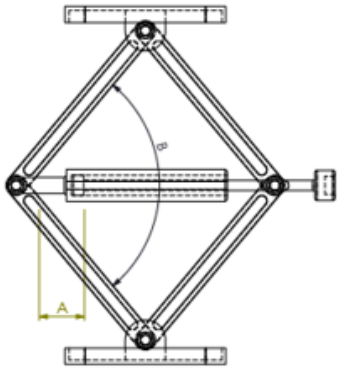
Create the assembly using the parts provided, and answer the following questions:

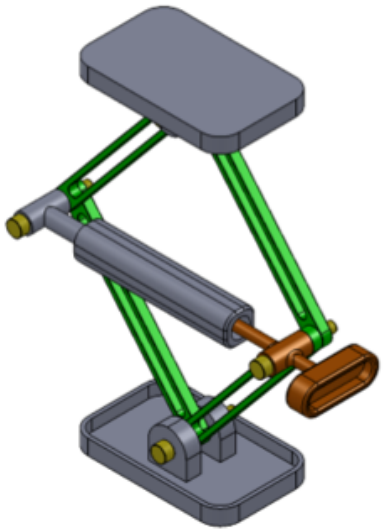
Q1 What is the centre of mass of the assembly if A = 5 mm?

Q2 What is the centre of mass of the assembly if B = 120 deg?

Title: 3D CAD Excercise 10 Assembly			Tolerance unless otherwise stated: Dimension: ± 0.02, Angle: ± 30'		Unit: MM
Material: / Size:			Surface finish: 1.6, Scale: 1 : 1		Qty: 01
Drawn By: Goh YH	Date: 05/05/2016	Changes Date:	Drawing No: 3DCAD-010	Trade: FT	Time: 2 hrs
NRIC: SXXXX123H	Remarks:			Item: 01	Sheet No: 1/1

11.4 3D CAD ASSEMBLY 4





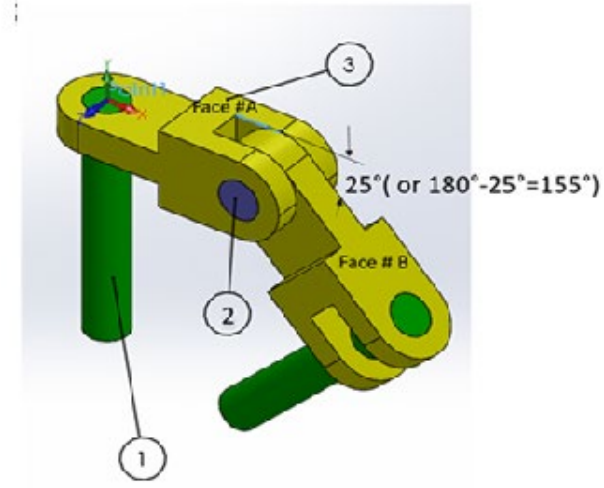
Answer the following questions wrt the coordinate system shown

Q1 What is the centre of mass of the assembly if A = 38 mm?

Q2 What is the centre of mass of the assembly if B = 61 deg?

Title: 3D CAD Exercise 12 Assembly			Tolerance unless otherwise stated: Dimension: ± 0.02, Angle: ± 30'		Unit: MM
Material: / Size:			Surface finish: 1.6, Scale: 1 : 1		Qty: 01
Drawn By: Goh YH	Date: 10/01/2015	Changes Date:	Drawing No: 3DCC- Assembly	Trade: FT	Time: 4 hrs
NRIC: SXXXX123H	Remarks:			Item: 01	Sheet No: 1

11.5. 3D CAD ASSEMBLY 5



(A) What is the centre of mass of the assembly (mm) if A=25 degrees:
(B) What is the center of mass of the assembly(mm) if A= 45 degrees:

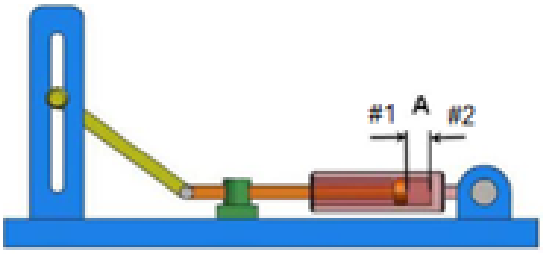
Title :		Tolerance unless otherwise stated Dimension : ± 0.02 Angle : ± 30'		Unit : MM	
Material : Size :		Surface finish : 1.6	Scale : 1 : 1	Qty : 01	
Drawn By :	Date : 10/01/2015	Changes Date :	Drawing No : Assembly	Trade : FT	Time : 1.5 hrs
NRIC: S0000123H	Remarks :			Sem : 01	Sheet No : 2/4

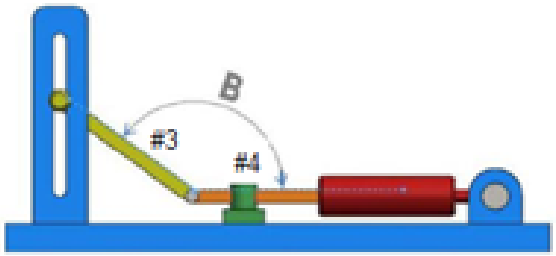
12_ INSTRUCTION GUIDE-CAD

Instruction Guide (CAD ASSEMBLY 3)

Questions:

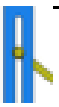
1) Define the mass of assembly when A is set at distamce **5 mm**
2) Define the mass of the assembly when B is set at an angle of **120°**





- **Assemble of parts**

  Insert > Mate.....

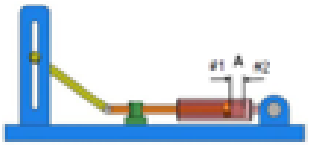
 Degrees of Freedom =1	 Degrees of Freedom =1	 Degrees of Freedom =1

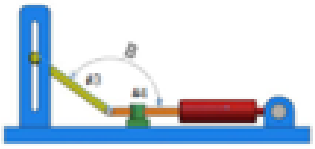
- **Locate the coordinate system**

Assembly > Reference Geometry> Coordinate System > "Locate point "

[X axis] > "Locate edge-Right" > [y axis] > "Locate edge-Left"

- **Set distance mate (A=5mm) to define the Centre of Mass**





Reference Geometry > PLANE > Select face" Side of Cylinder" > Distance(15/2) =7.5

Section view >

Mate > Select "Faces #1 & #2" > Distance: 5 > Anti-Aligned

- **Define the Centre of Mass(A=5mm)**

Evaluate > Mass properties > Select "Coordinate System" > Check !

Mass : 356.18 grams Centre of Mass : X = 25.00 Y= 94.65 Z=13.80

- **Set angle mate(B=120°)to define the Centre of Mass**

At Feature Manager Design Tree > Delete "Distance mate"

Mate > Select "Faces #3 & #4 "> Angle: 60° > FRONT

- **Define the Centre of Mass(B=120°)**

Evaluate > Mass properties > Select "Coordinate System" > Check !

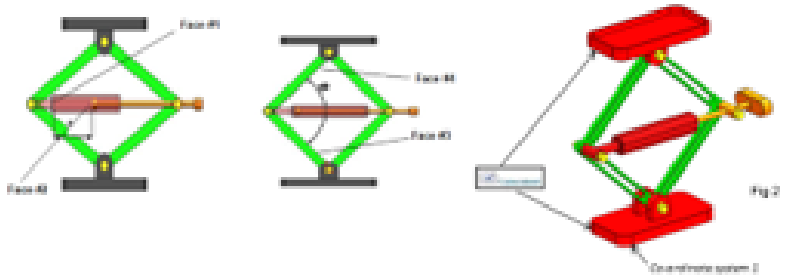
Mass : 356.18 grams Centre of Mass : X = 25.00 Y= 95.85 Z=13.91

- **Delete the PLANE**

Section view >

At Feature Manager Design Tree > Delete "PLANE"

Instruction Guide (CAD ASSEMBLY 4)



Questions:

a) What is the center of Mass when A = 38 mm?

b) What is the center of Mass when B = 61°?

1. **Assemble of parts**

   Open "Base"

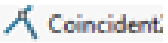
   Insert Components > Mate.....

Shift +roller =ZOOM	Ctrl +roller =PAN	Ctrl+7 =Isometric View

2. **Locate the coordinate system**

   Assembly > Reference Geometry> Coordinate System >

   [X axis] > "Locate ~~edge~~ Right" > [Y axis] > "Locate edge-up"

   Mate both faces of base end coincident  as in Fig2

3. **Set distance mate (A=38mm) to define the Centre of Mass**

   Reference Geometry > PLANE > Select face" Side of Cylinder" >

   Section view >

   Mate > Select "Faces #1 & #2" > Distance: 38mm

4. **Define the Centre of Mass(A=38mm)**

   Evaluate > Mass properties > Select "Coordinate System" > Check !

   Mass : _____grams Centre of Mass : X = _____Y= _____ Z=_____

5. **Set angle mate(B=61°)to define the Centre of Mass**

   At Feature Manager Design Tree > Delete "Distance mate" or right click to suppress it

   Mate > Select "Faces #3 & #4 "> Angle: 61° > FRONT

6. **Define the Centre of Mass(B=61°)**

   Evaluate > Mass properties > Select "Coordinate System" > Check !

   7. Mass : _____grams Centre of Mass : X = _____Y= _____ Z=_____

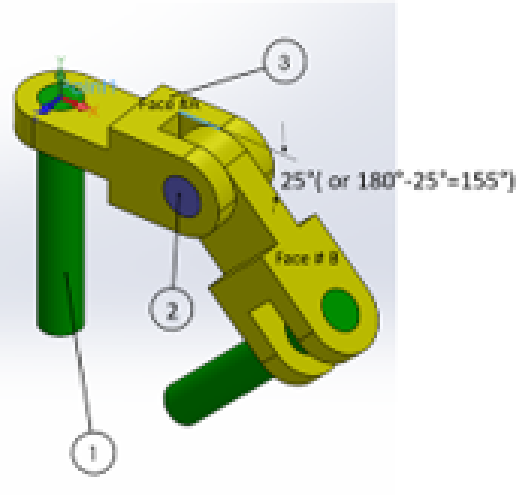
8. **Delete the PLANE**

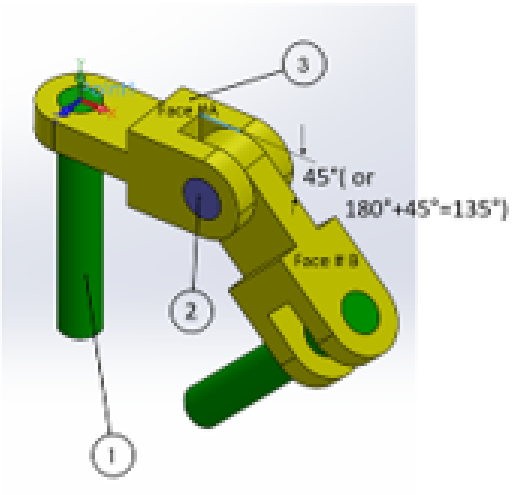
   Section view >

   At Feature Manager Design Tree > Delete "PLANE"

Instructional Guide (3D CAD ASSEMBLY 5)

Instruction Guide (3D CAD ASSEMBLY 5)





Assemble of parts

Open "Long Pin"

Insert Components > Mate.....

Shift +roller =ZOOM	Ctrl +roller =PAN	Ctrl+7 =Isometric View

Locate the coordinate system

Assembly > Reference Geometry> Coordinate System > "Locate point "

Assembly > Reference Geometry> Coordinate System >

[X axis] > "Locate edge-Right" > [y axis] > "Locate edge-Up"

Set distance mate (A=25 °) to define the Centre of Mass

Mate > Select "Faces #A & #B" > Distance: 25"

Evaluate > Mass properties > Select "Coordinate System" >

Check !

Mass : _____grams Centre of Mass : X = _____Y= _____ Z=_____

Set angle mate(A=45")to define the Centre of Mass

At Feature Manager Design Tree > Edit " Angle2 (CHAIN LINK< ..

Mate > Select "Faces #A & #B "> Angle: 45" > FRONT

Evaluate > Mass properties > Select "Coordinate System" >

Check !

Mass : _____grams Centre of Mass : X = _____Y= _____ Z=_____

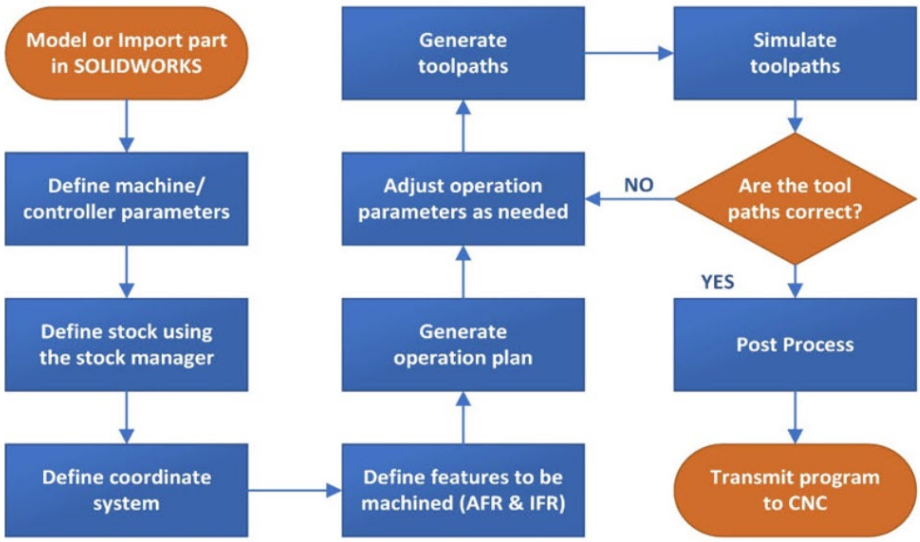
13_SOLIDWORKS CAM

SOLIDWORKS CAM is a Computer Aided Manufacturing add-in for SOLIDWORKS that allows to create G-code and M-code for CNC machining part -program.

What is special about SOLIDWORKS CAM is that it uses **Knowledge Based Machining with Automatic Feature Recognition (AFR)**. The part features can be automatically recognized from the model. By using a technology database, the CAM can create conditions that apply machining strategies to the part features. This dramatically reduces the time of having to program part operations manually or interactively (IFR). Having this power directly in SOLIDWORKS gives designers and machinists unparalleled versatility, consistency and speed within an integrated production environment.

Turning SOLIDWORKS CAM 2 and 4 axis turning software for programming CNC lathe machines includes automatic roughing, finishing, grooving, threading, cutoff and single point (drilling, boring, reaming and tapping) cycles.

SolidWorks CAM Process



Basic Steps SolidWorks CAM Process

- ❑ Define Machine (**Fanuc T2nx**)
- ❑ Define Stock Manager
- ❑ Define Coordinate System (**Work offset:G54**)
- ❑ Extract Machinable Features
  - o Generate Operation Plan
  - o Generate Tool Path
  - o Simulate Toolpaths
- ❑ Post Processer

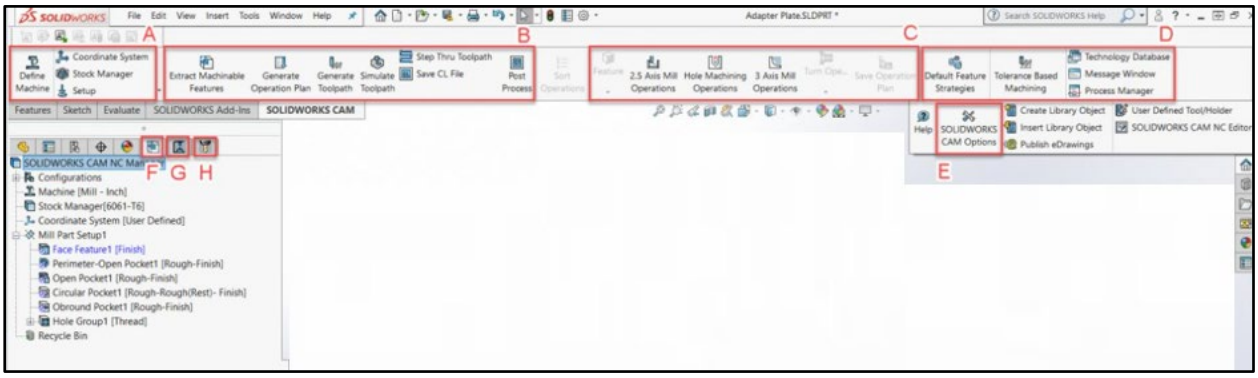
Create Single Feature (Example: OD Threading)

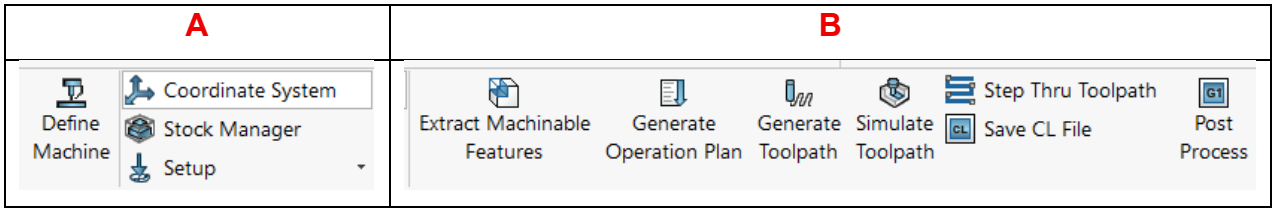
- o Turn Setup 1 > Turning Operation > Thread > Feature > Create Feature
- o Type: OD Strategy : Thread > Sketch > Sketch entity > Line D1 > OK
- o Generate Operation Plan > Generate Toolpath >Simulate Toolpath

- ❑ Post Processer

Edit Tool (Select : CAM Tool Tree)

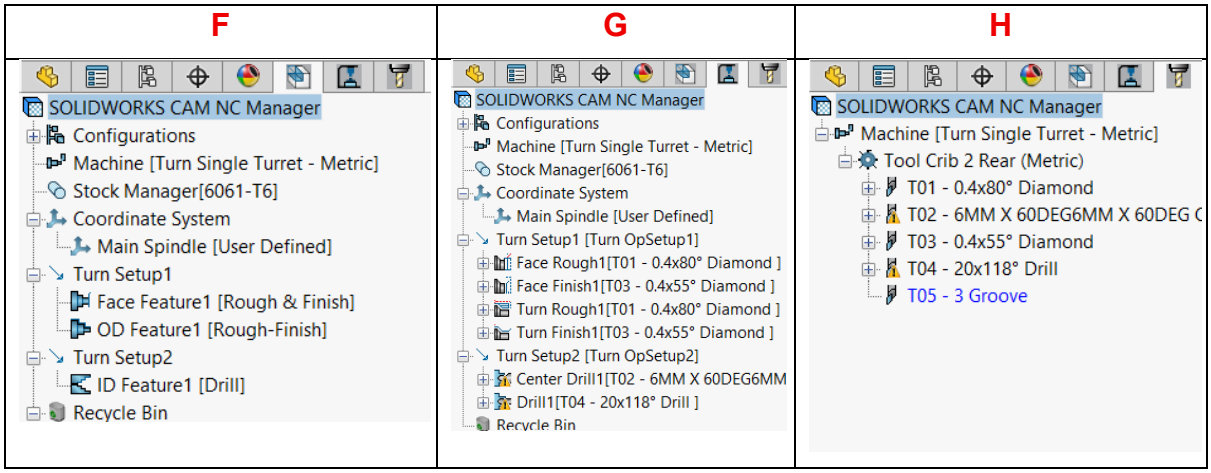
SOLIDWORKS CAM User Interface





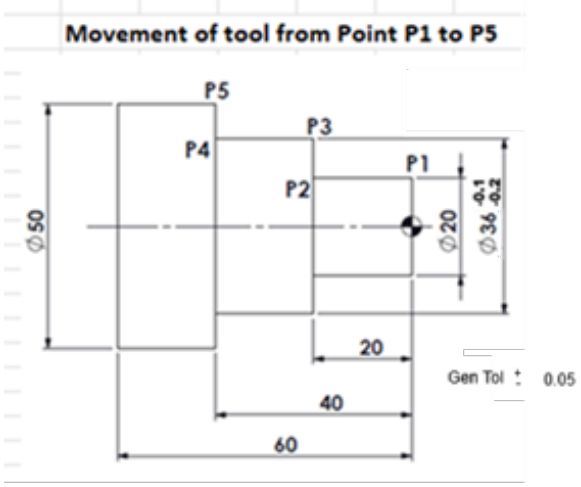
The SOLIDWORKS CAM user interface closely follows the above process.

A. **Set-up options**
B. **Program generation commands**
C. Options to add operations interactively
D. Technology Database Options
E. Options for defining features to be extracted when using (AFR)
F. **CAM Feature Tree** – Shows extracted features
G. **CAM Operations Tree** – Edit or add operations interactively
H. **CAM Tool Tree** – Editing current Tool Crib



Absolute Programming (G90)

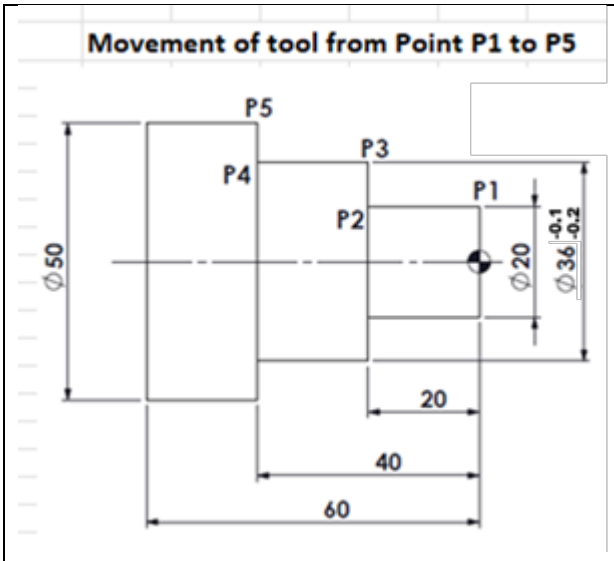
Absolute Programming (Fixed ZERO))
All co-ordinate values are taken from the work zero point
Bank Ø50 x 65 mm



POINTS	Co-ordinates	
	ØX	Z
P0		
P1		
P2		
P3		
P4		
P5		

Incremental Programming (G91)

Incremental Programming (Floating zero point)
The 'zero' point is floating and moves to the new position that the tool takes.
Blank Ø50 x 65 mm



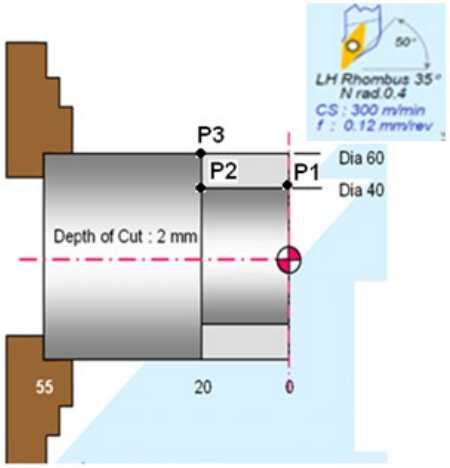
POINTS	Co-ordinates	
	ØX	Z
P0		
P1		
P2		
P3		
P4		
P5		

Longitudinal Cutting

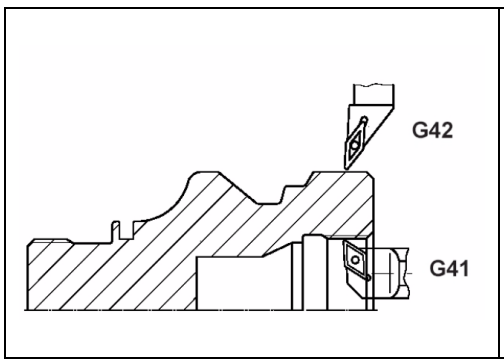
Complete the CNC part program using a Fanuc system with the following cutting conditions ;

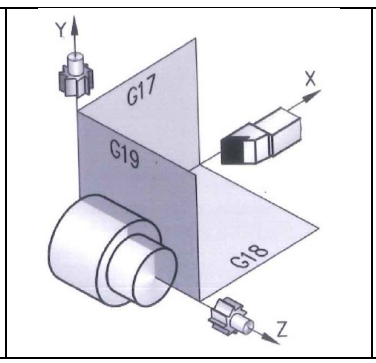
Maximum Spindle Speed	: 3000 rpm
Cutting Path	: From P1 to P3
Material size	: Ø65 X 60 mm
Cutting Speed	: 200m /min ;
Feed	:0.12 mm/rev (Finishing cut)
Tool type	:Left Hand Finishing Tool-T3
Tool angle/nose radius	:35°,NR 0.4 (ODL35.4)
Start position point	: X 70 Z10
Return position point	: X 90 Z10

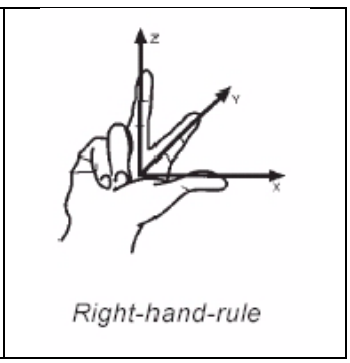
Coolant to be applied during the machining operations.



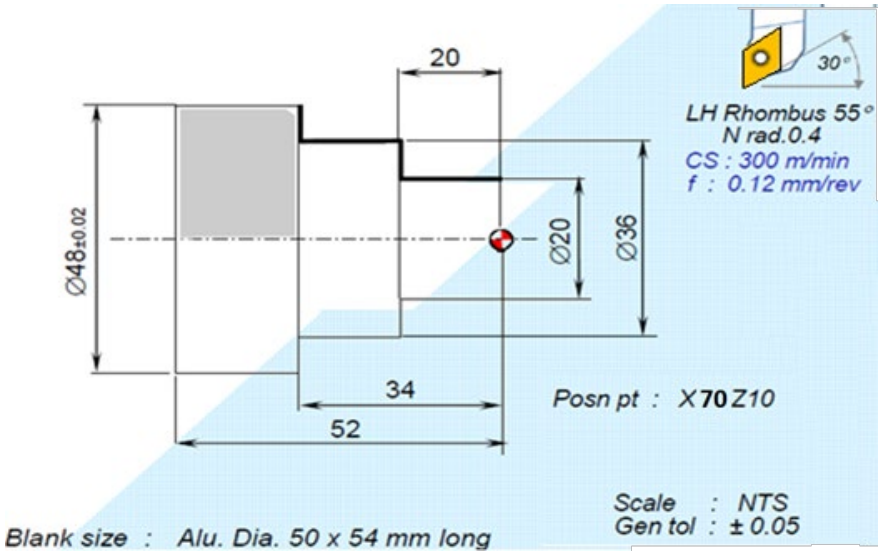

```
     [PROGRAM HEADER]
N1   G291                                    ;  Convert to Fanuc System
N2   G54 G21 G80 G40                         ;  (Work offset, Metric unit, Canned cycle cancel)
N3   G18                                     ;  (Plane XZ )
N3   G50 S3000          G90                  ;  (Spindle clamp 3000/Absolute dimension)
     G28 U0.0
N5   W0.0               M09                  ;  (Return to reference/Coolant OFF)
     [PROGRAM BODY]
N6   T0303              M06                  ;  (Tool change ,tool offset 03)
N7   G95                F0.12                ;  (Feed rate in mm/rev )
N8   G96                S200    M04 M08      ;  (Constant cutting speed in m/min /CCW/Coolant ON)
N9   G00                X70     Z10          ;  (Tool radius compensation RIGHT)
N10  G01 G42            X40     Z0           ;  (P1)
N12  Z-20                                    ;  (P2)
N13  X60                                     ;  (P3)
N14  G00 G40            X90     Z10          ;  (Tool radius compensation CANCEL)
N15  M05                                     ;  (Spindle stop)
     [PROGRAM END]
     G28 U0.0
N16  W0.0               M09                  ;  Return to reference/Coolant OFF
N17  M05                                     ;  Spindle stop, cutting Speed OFF
N18  M30                                     ;  End of program
```







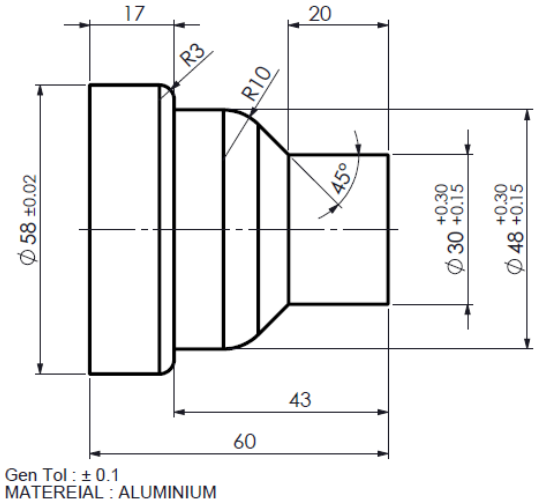
13.1. CAM EXERCISE 1



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

SIDE B & A
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

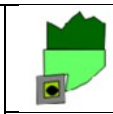
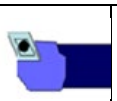
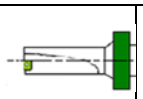
13.2. CAM EXERCISE 2



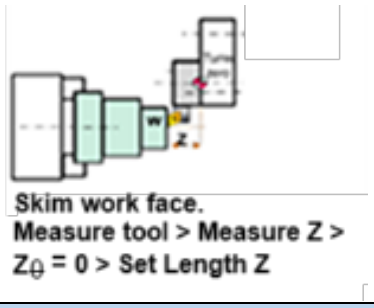
Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

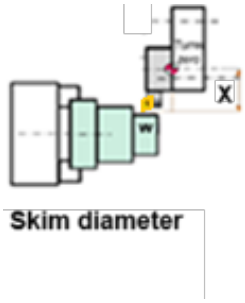
SIDE B & A
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

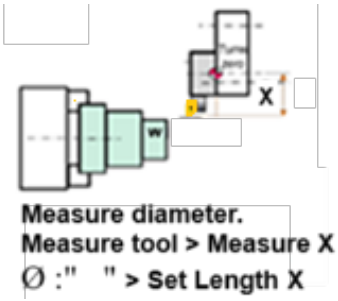
13.3. CAM EXERCISE 3

Hand of Tool	 LH Roughing Tool (OD)	 LH Finishing Tool (OD)	 LH Grooving Tool (OD)	 RH Threading Tool (OD)	 LH Finishing Tool (ID)	 Face Drill (ID)	 Twist Drill
Cutting Speed (m/min)	150-250 (150)	250-350 (250)	120-180 (120)	80-120 (80)	200-300 (200)	60-100 (62.84)	30-40 (30)
Feed (mm/rev)	*0.25-0.35 (0.25)*	*0.12-0.15 (0.12)*	*0.08-0.15 (0.08)*	*0.25 mm (0.25)*	*0.1-0.15 mm (0.1)*	*0.1-0.15 (0.1)*	*0.05-0.1 (0.05)*

Tool length offset setting



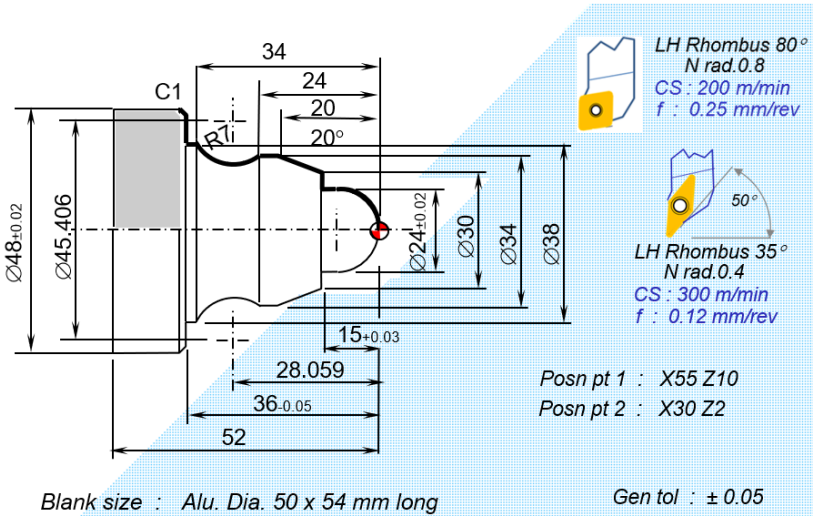




Standard Tool List & Tool Length Offset

Loc	Type	Tool Name	DP	Length X (offset)	Length Z (offset)	Radius	Working Direction	Holder angle	Insert Angle	Insert Length	Spindle rotation
1		ODL80.8	1			0.800	←	95.0	80	12.0	CCW
2											
3		ODL55.4	1			0.400	←	93.0	55	11.0	CCW
4											
5		OGL4.12	1			0.120		4.000		8.0	CCW
6		IDL55.4_16	1			0.400	←	93.0	55	9.0	CCW
7		OTR2.12	1			0.120				16.0	**CW**
8		IDL80.8_16	1			0.800	←	95.0	80	12.0	CCW
9											
10		FD20_3	1			20.00	←				CCW
11											
12											

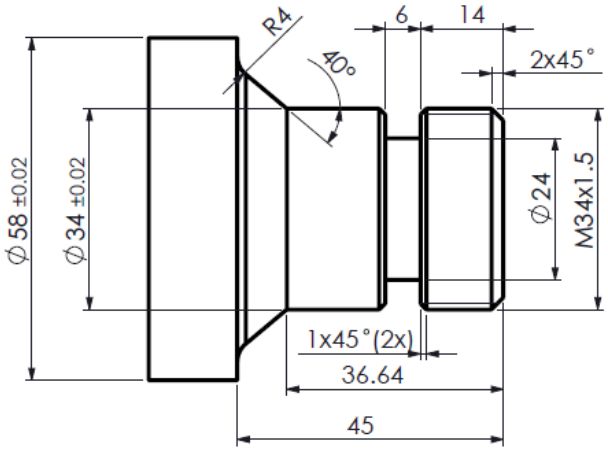
13.4. CAM EXERCISE 4



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL35.4	0.12	250	

SIDE B & A
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

13.5. CAM EXERCISE 5

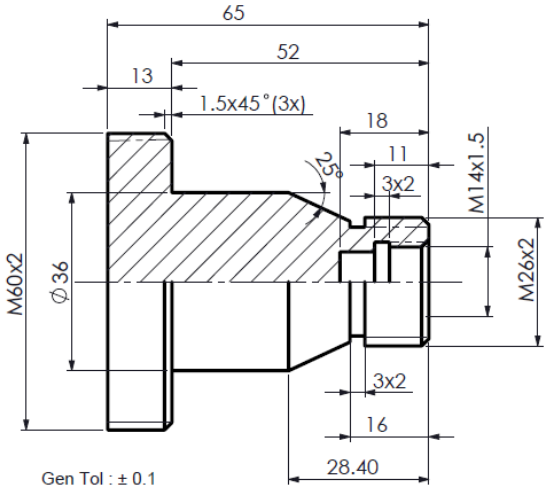


Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
5	OGL4.12	0.008	120	
7	OT2.12	1.5	80	

SIDE B
Sequence of operation:
1.Face (Rough)
2. Face (Finish)
3. OD Longitudinal (Rough)
4.OD Longitudinal (Finish)
5.Grooving
6.Threading

13.6. CAM EXERCISE 6

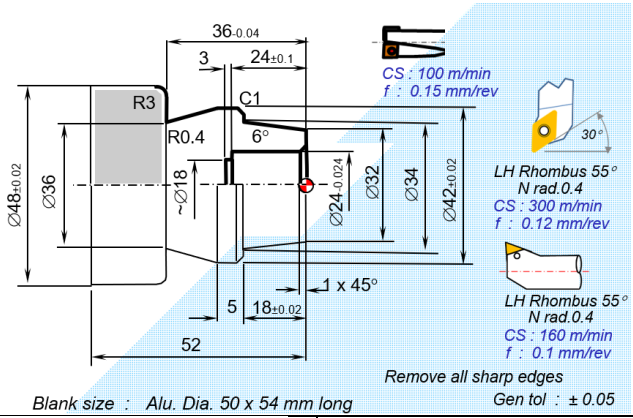


Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
2	IT1.5.12_10	1.5	80	
3	ODL55.4	0.12	250	
5	OGL3.12	0.008	120	
6	IDL55.4_10	0.1	200	
7	OT2.12	2	80	
8	IDL80.8_10	0.2	160	
10	FD20_3	0.1	62.84	
12	OGL3.12	0.008	120	

SIDE B
Sequence of operation:
1.Drill
2.Face (Rough)
3.Face (Finish)
4.OD Longitudinal (Rough)
5.OD Longitudinal (Finish)
6.ID Longitudinal (Rough)
7.ID Longitudinal (Finish)
8 ID Grooving
8.OD Threading
9.OD Threading
10.ID Threading

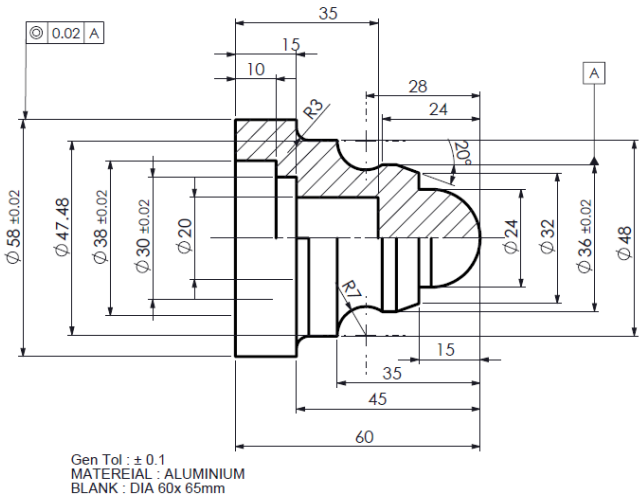
13.7. CAM EXERCISE 7



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE B
Sequence of operation:
1.Drill
2.Face (Rough)
3.Face (Finish)
4.OD Longitudinal (Rough)
5.OD Longitudinal (Finish)
6.ID Longitudinal (Rough)
7.ID Longitudinal (Finish)

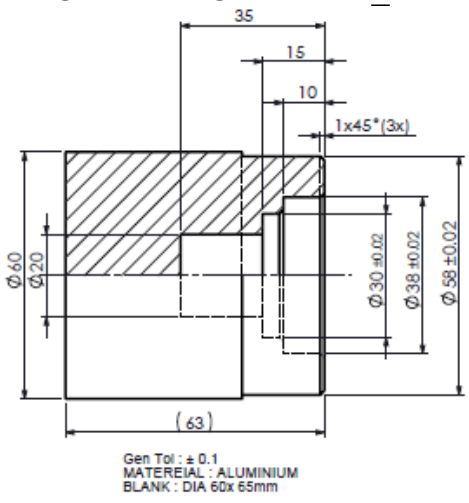
13.8. CAM ASSIGNMENT 1



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

SIDE B
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

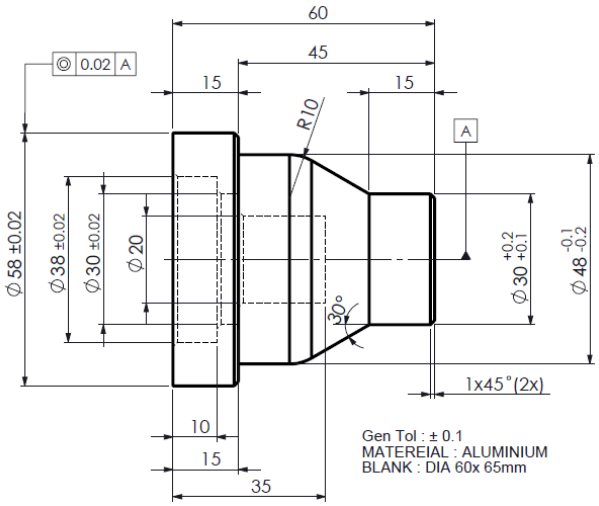
CAM ASSIGNMENT 1_A



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE A
Sequence of operation:
1.Drill
2.Face (Rough)
3. Face (Finish)
4. OD Longitudinal (Rough)
5. OD Longitudinal (Finish)
6. ID Longitudinal (Rough)
7. ID Longitudinal (Finish)

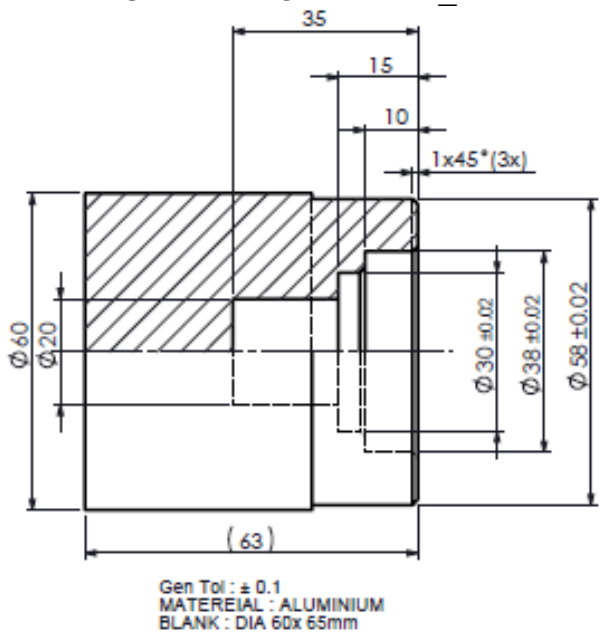
13.9. CAM ASSIGNMENT 2



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

SIDE B
Sequence of operation:
1. Face (Rough)
2. Face (Finish)
3. OD Longitudinal (Rough)
4. OD Longitudinal (Finish)

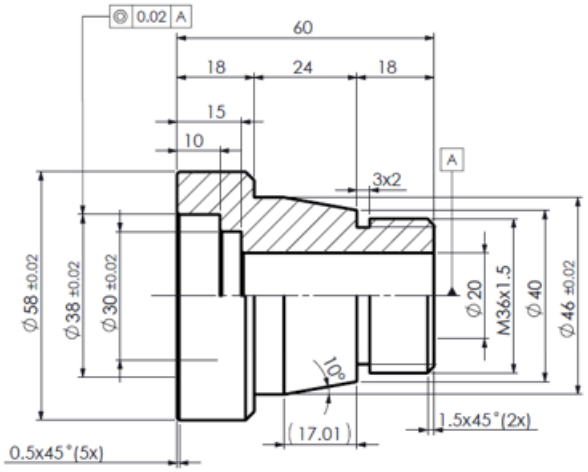
CAM ASSIGNMENT 2_A



Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE A
Sequence of operation:
1.Drill
2.Face (Rough)
3. Face (Finish)
4. OD Longitudinal (Rough)
5. OD Longitudinal (Finish)
6. ID Longitudinal (Rough)
7. ID Longitudinal (Finish)

13.10. CAM ASSIGNMENT 3

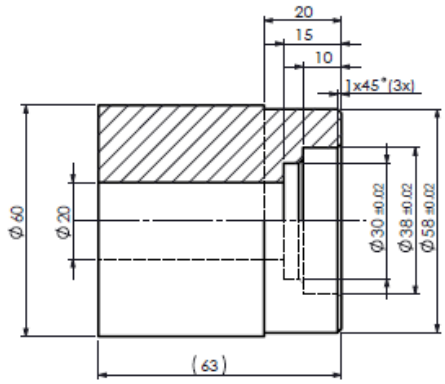


Gen Tol: : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
5	OGL4.12	0.008	120	
7	OT2.12	1.5	80	
10	FD20_3	0.1	62.84	

SIDE B
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)
5.Grooving
6.Threading

CAM ASSIGNMENT 3_A

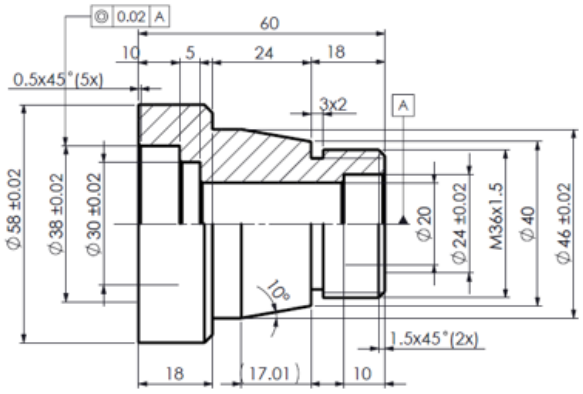


Gen Tol : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE A
Sequence of operation:
1.Drill
2.Face (Rough)
3. Face (Finish)
4. OD Longitudinal (Rough)
5. OD Longitudinal (Finish)
6. ID Longitudinal (Rough)
7. ID Longitudinal (Finish)

13.11. CAM ASSIGNMENT 4

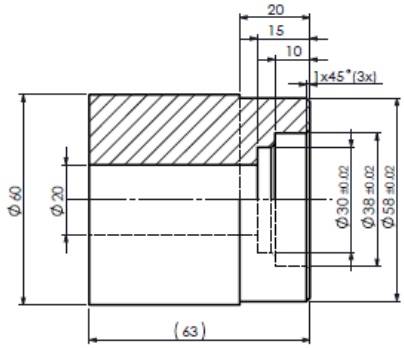


Gen Tol : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
5	OGL4.12	0.008	120	
6	IDL55.4_16	0.1	200	
7	OT2.12	1.5	80	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE B
Sequence of operation:
1.Face (Rough)
2. Face (Finish)
3. OD Longitudinal (Rough)
4. OD Longitudinal (Finish)
5. ID Longitudinal (Rough)
6. ID Longitudinal (Finish)
7. Grooving
8. Threading

CAM ASSIGNMENT 4_A



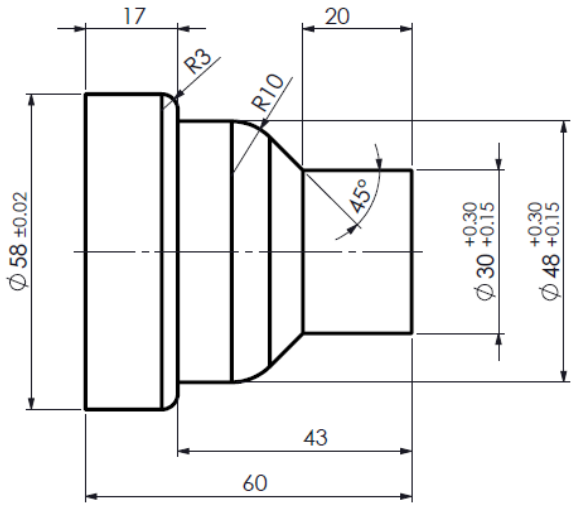
Gen Tol : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

SIDE A
Sequence of operation:
1.Drill
2.Face (Rough)
3. Face (Finish)
4. OD Longitudinal (Rough)
5. OD Longitudinal (Finish)
6. ID Longitudinal (Rough)
7. ID Longitudinal (Finish)

14. INSTRUCTION GUIDE-CAM

14.1. CAM EXERCISE 2



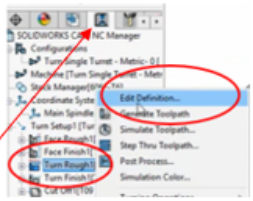
Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM
BLANK : DIA 60x 65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

SIDE B & A
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

Basic Steps SolidWorks CAM Process

- ❑ Define Machine (**Fanuc T2nx**)
- ❑ Define Stock Manager
- ❑ Define Coordinate System (**Work offset:G54**)
- ❑ Extract Machinable Features
  - o Generate Operation Plan
  - o Generate Tool Path
  - o Simulate Toolpaths
- ❑ Post Processer



Create Single Feature (Example: OD Threading)

- o Turn Setup 1 > Turning Operation > Thread > Feature > Create Feature
- o Type: OD Strategy : Thread > Sketch > Sketch entity > Line D1 > OK
- o Generate Operation Plan > Generate Toolpath >Simulate Toolpath

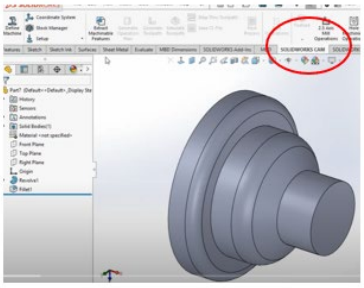
- ❑ Post Processer

Edit Tool (Select : CAM Tool Tree)

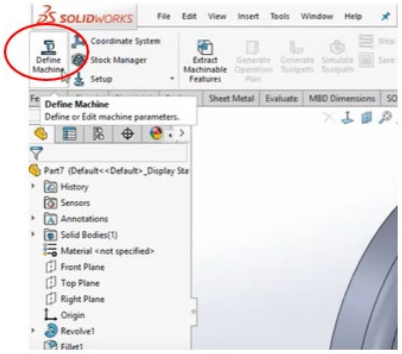
- o Turn Rough >Edit definition…

1. Activate SolidWorks CAM

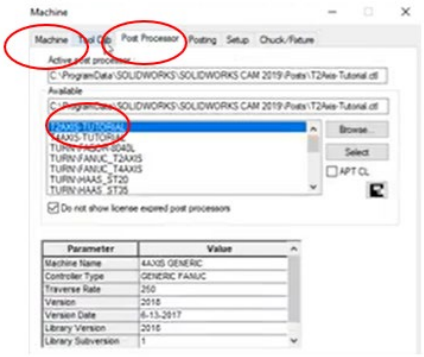
1.SOLIDWORKS CAM >Option > Add in > Activate SolidWorks CAM



2. Define Machine

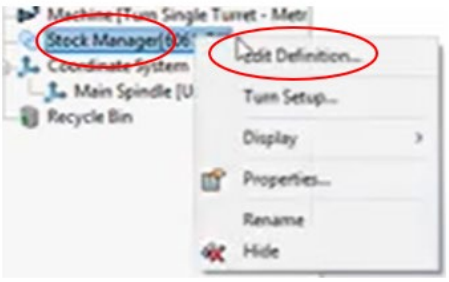


3.Machine> Turn Machine >Select Turning Single Turret
Post Process > Select Fanuc T2nx

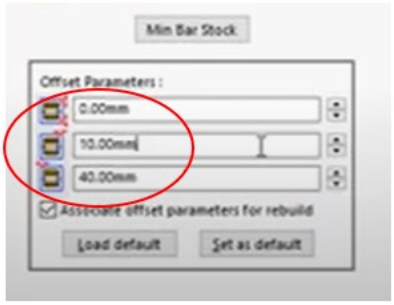


2. Define Stock Manager and Coordinate System

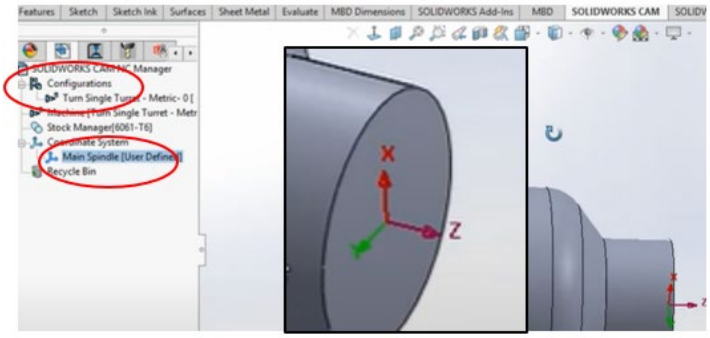
1. Right click **Stock Manager** >
Edit Definition



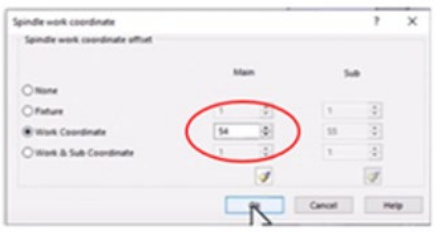
2. Offset Parameters: 2:2:3 > OK



3. Coordinates System > Main Spindle (User



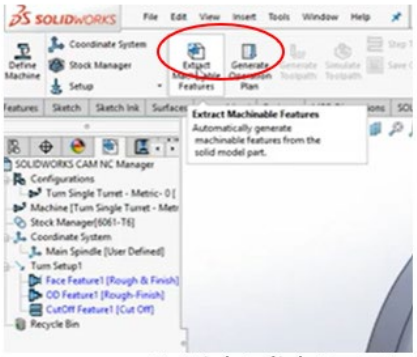
4. Work offset>Work Coordinate:**54**



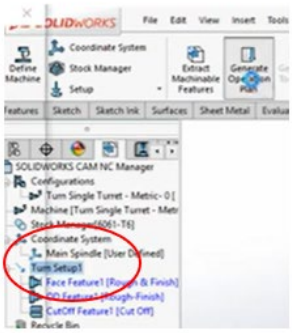
3. Extract Machinable Feature

1. Extract Machinable Features

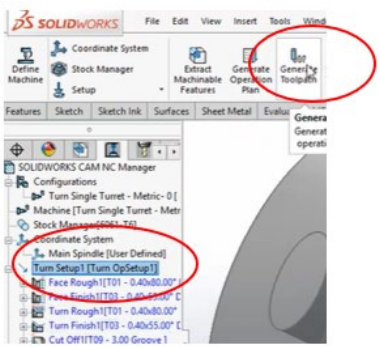
(Turn Setting 1 ..appear automatically)

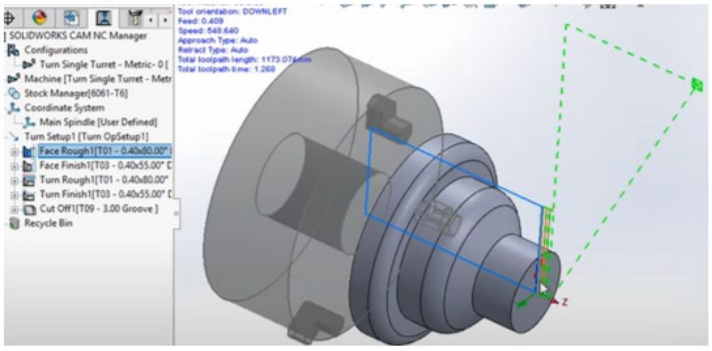


2. Right click Turn Setup 1 > **Generate Operation Plan**



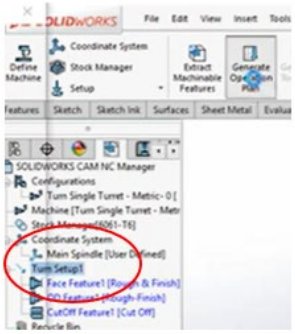
3. Right click Turn Setup 1 > **Generate Tool Path**

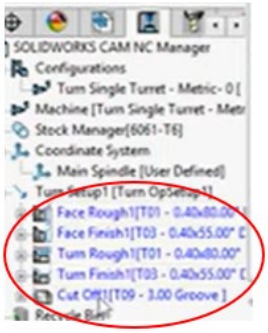




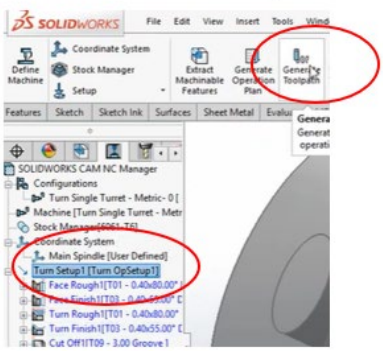
4. Generate Operation Plan & Generate Toolpath

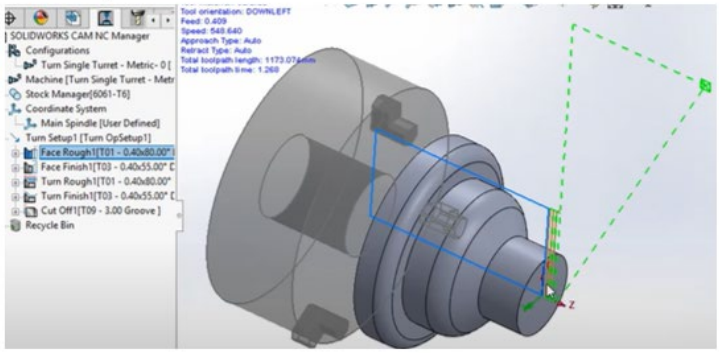
1. Right click Turn Setup 1 > **Generate Operation Plan**





2. Right click Turn Setup 1 > **Generate Tool Path**

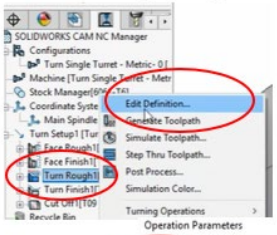




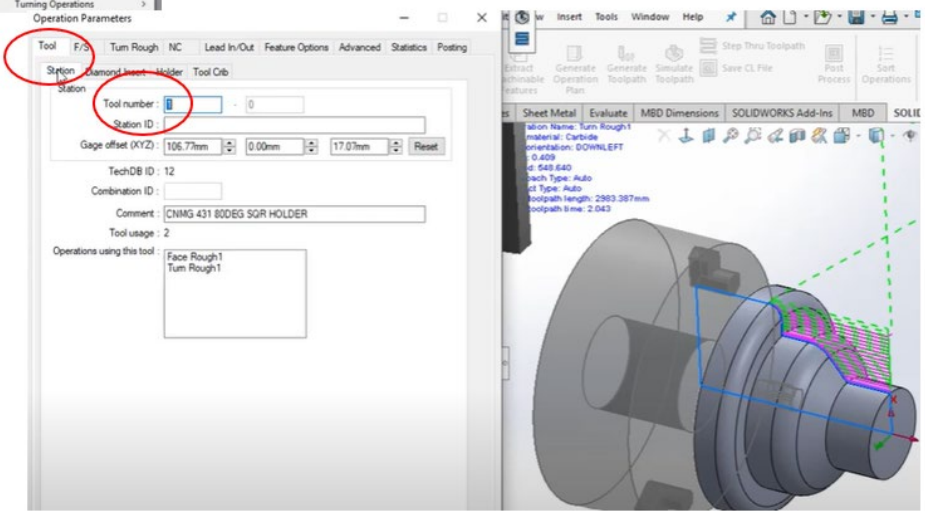
5. Edit Turn Rough (CAM Tool Tree)

Edit Turn Rough

1. Turn Rough> Right Click > Edit Definition…

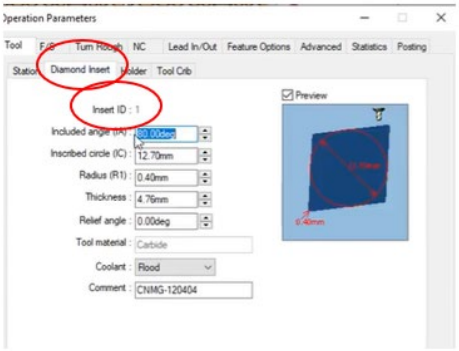


2. Tool > Station : 1

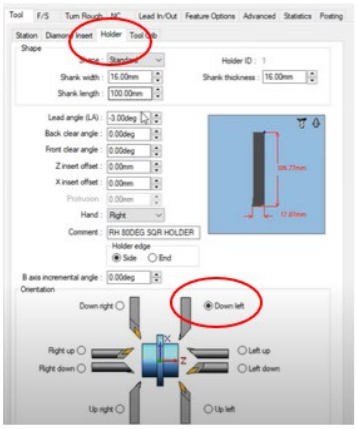


6. Check Tool Rough

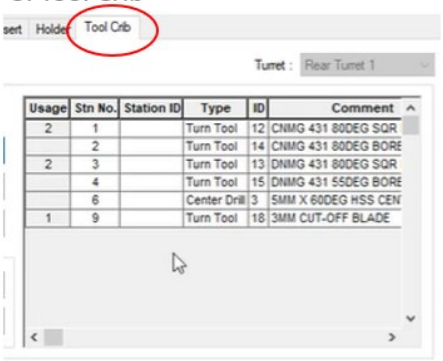
1. Diamond > Insert ID:1



2. Holder> Down Left



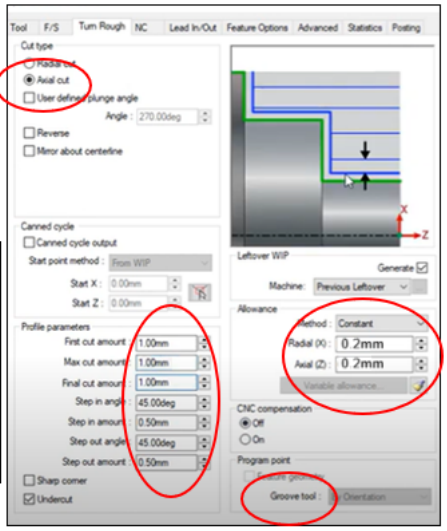
3. Tool Crib



7. Check Speed ,Feed and Rotation

1. Speed Feed(S/F) >Defined by : Library/Operation > Direction : CCW

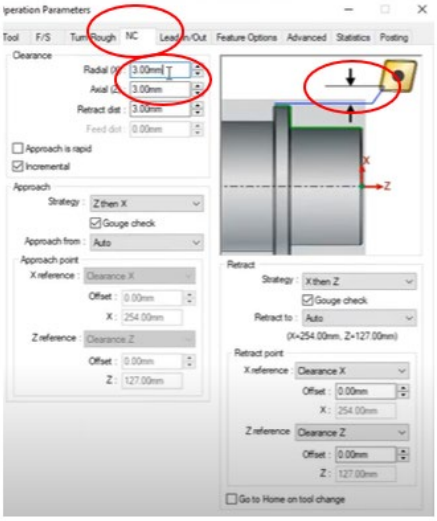
2. Turn Rough > Cut Type : Axial cut



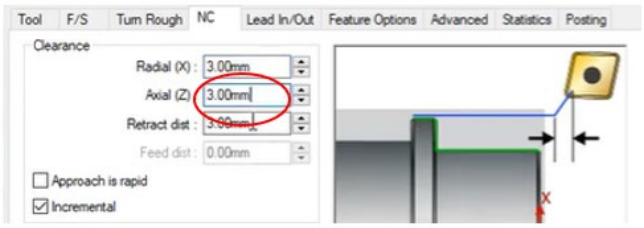
First cut amount :1.00 mm
Rough cut amount :1.00mm
Final cut amount :1.00mm
Allowance: Constant
(X) : 0.2mm
(Z) : 0.2 mm

8. Check Cutting Clearance

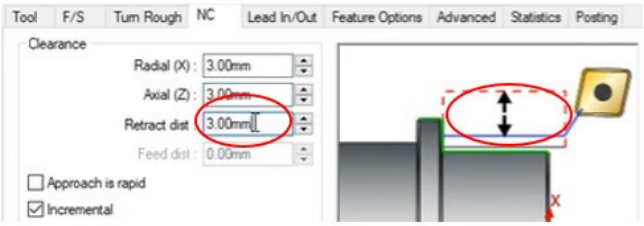
1. NC > Radial(X) :3.00mm



2. NC > Radial(Z) :3.00mm

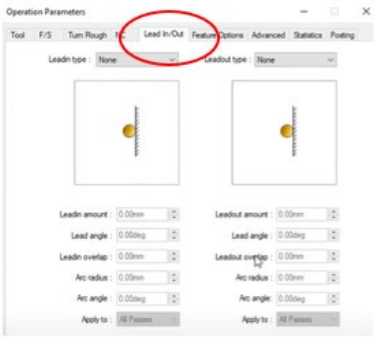


3. NC > Retract Dist.:3.00mm

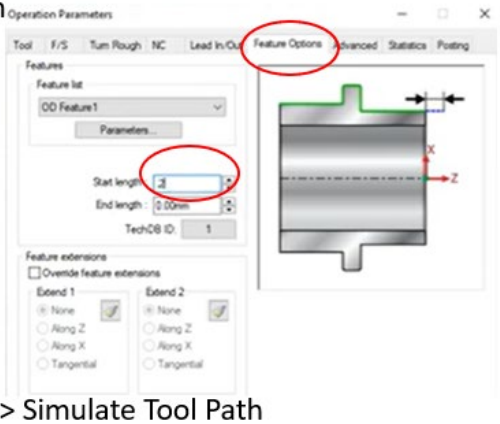


9. Generate Toolpath and Simulate Toolpath at Turn Setup 1

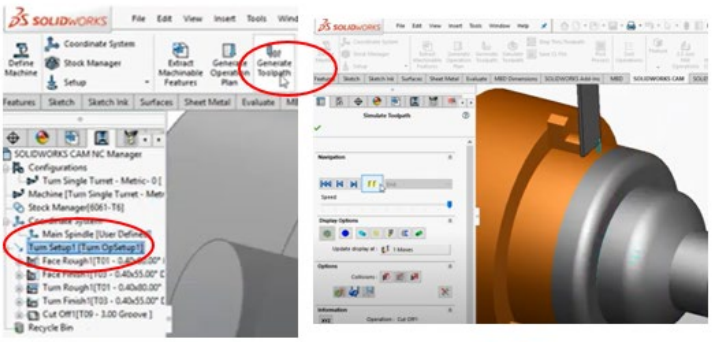
1. Lead In/Out



2. Feature Opton > OD Feature > Start length: 0.2mm

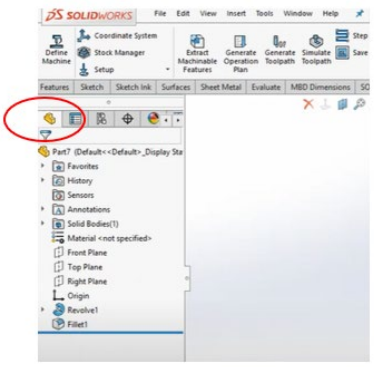


3. Turn Setup 1 > Generate Tool Path > Simulate Tool Path

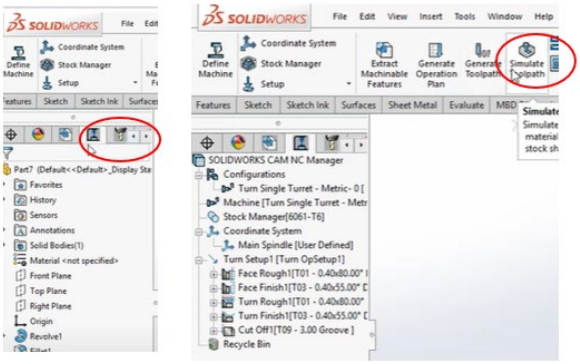


10. Simulate Toolpath and Post Process

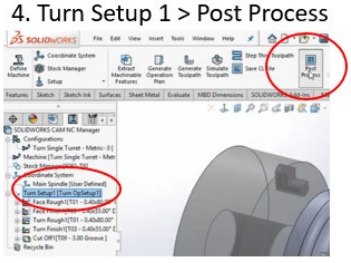
1. SolidWorks(to edit model)



2. Next Page > CAM 3. Simulate Toolpath >

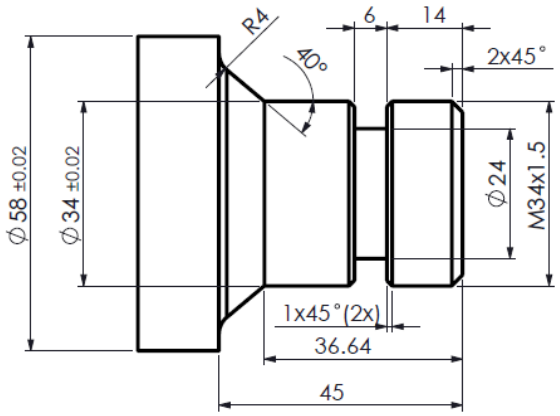


4. Turn Setup 1 > Post Process



5. Tape file > Play > Desktop

14.2. CAM EXERCISE 5



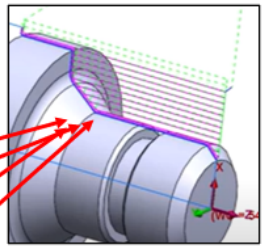
Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM
BLANK : DIA 60x 65mm

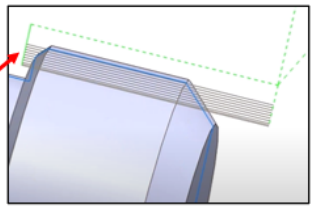
Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
5	OGL4.12	0.008	120	
7	OT2.12	1.5	80	

SIDE B
Sequence of operation:
1.Face (Rough)
2. Face (Finish)
3. OD Longitudinal (Rough)
4.OD Longitudinal (Finish)
5.Grooving
6.Threading

Basic Steps SolidWorks CAM Process

- ❑ Define Machine (Fanuc T2nx)
- ❑ Define Stock Manager
- ❑ Define Coordinate System (G54)
- ❑ Extract Machinable Features
  - o Generate Operation Plan
  - o Generate Tool Path
  - o Simulate Toolpaths
- ❑ Post Processer





Create Single Feature (Example: OD Threading)

- o Turn Setup 1 > Turning Operation > Thread > Feature > Create Feature
- o Type: OD Strategy : Thread > Sketch > Sketch entity > Line D1 > OK
- o Generate Operation Plan > Generate Toolpath >Simulate Toolpath

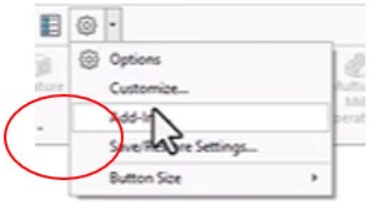
❑ Post Processer

Edit Tool (Select :CAM Tool Tree)

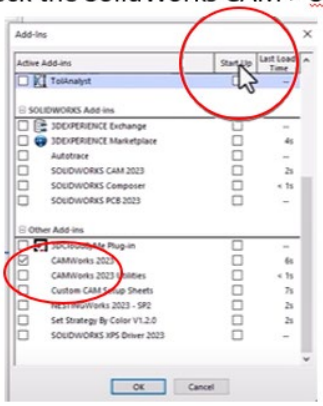
- o Turn Rough > Edit Definition….

1. Activate SolidWorks CAM

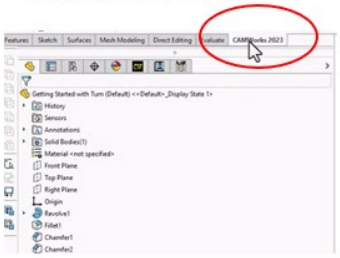
1.SolidWorks CAM > Option > Add in > Activate SolidWorks CAM



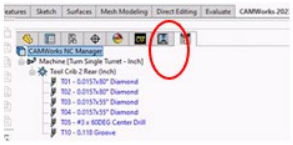
2.Check the SolidWorks CAM > StartUp



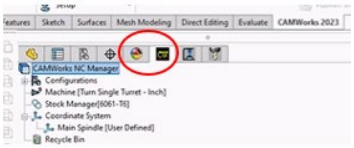
3.SolidWorks CAM



4.CAM Work Tool Tree

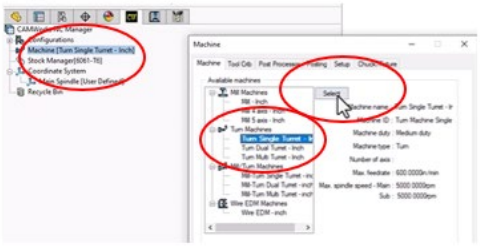


5.CAM Work Feature Tree

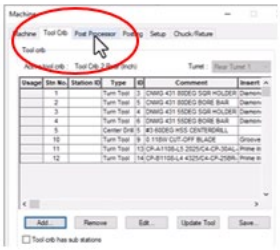


2. Define Machine(Fanuc T2nx) and Coordinate System(G54)

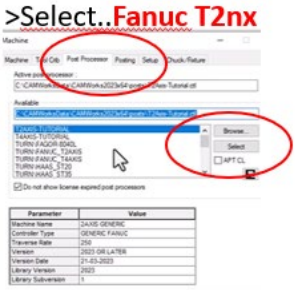
1.**Double Machine**>Turn Single Turret>Select



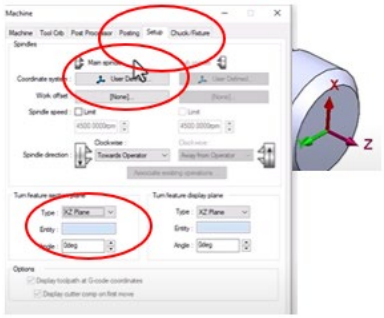
2.Tool Crib



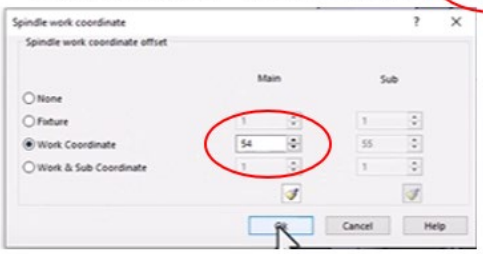
3.Click Post Processer >Select..**Fanuc T2nx**



4.SetUp >**Coordinate System**: User define



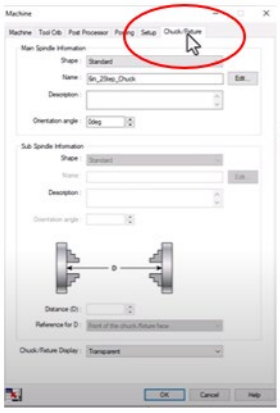
> 5. Work Offset >Work Coordinate : **54**



4.1.Ensure Feature in: XZ plane

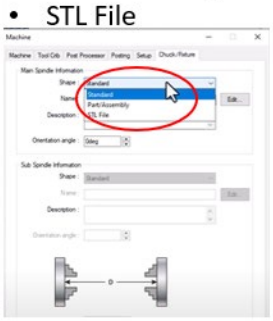
3. Define Stock Manager

1.Chuck Feature > Standard

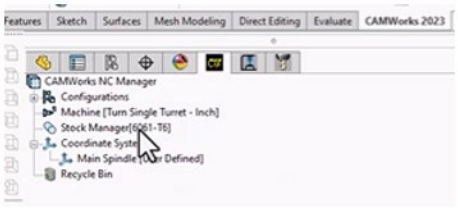


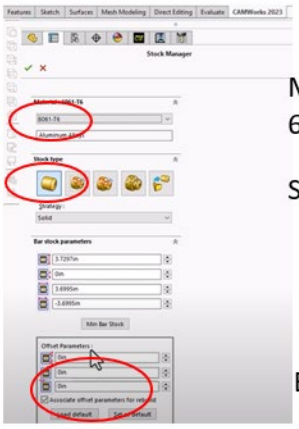
Main Spindle Information:

- Standard
- Part/Assembly
- STL File



2.**Stock Manager**[6061 T6]





Material:
6061 T6

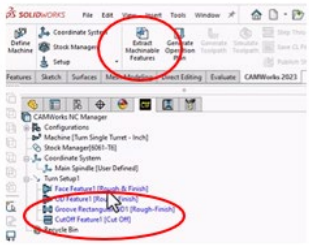
Solid Bar

Bar Stock

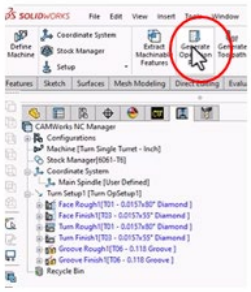
4. Process Automatically

Process Automatic and Manually CAM

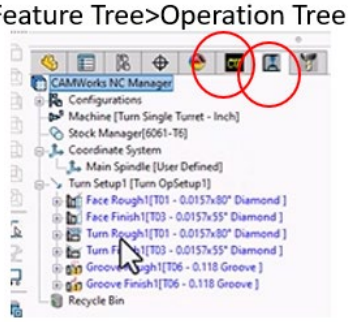
1.Extract Machinable Features >
Delete Cutoff Feature



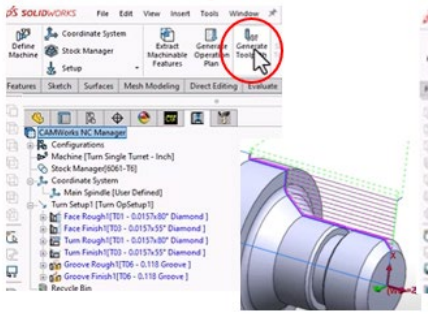
2.Generate Operation Plan



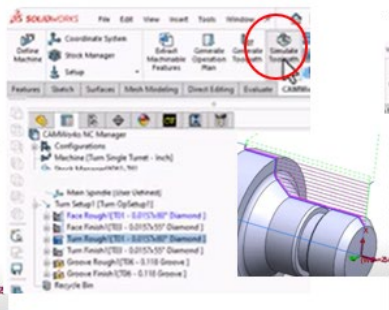
3.Feature Tree>Operation Tree



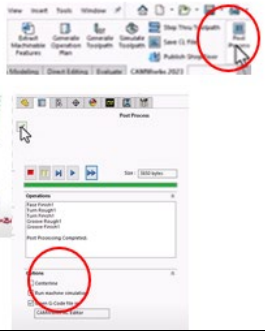
4.Generate Toolpath



5.Simulation Toolpath>Play



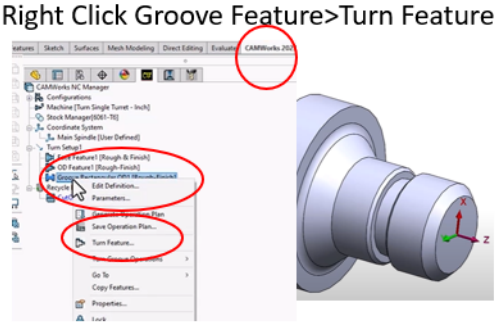
5.Post Process



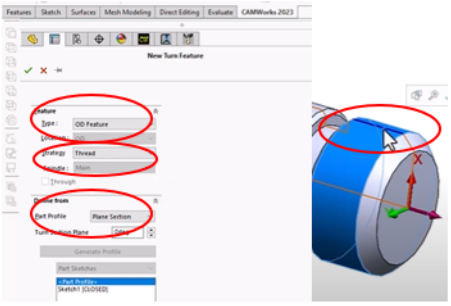
5. Process Manually for Threading

Manually CAM (Threading)

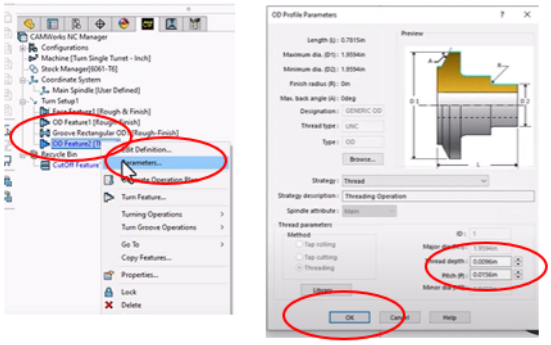
1.CAM Works 2023 >
Right Click Groove Feature>Turn Feature



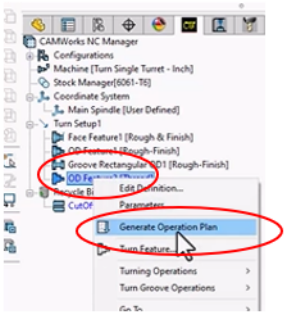
2. Type : OD Feature > Thread
Part Profile: Plan Section >Click Line ,D1



3. OD Feature Thread >
Parameter > Library >Select the depth & pitch

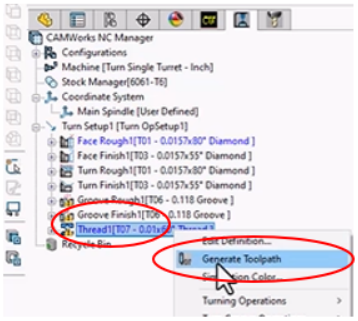


4. OD Feature Thread >
Generate Operation Plan

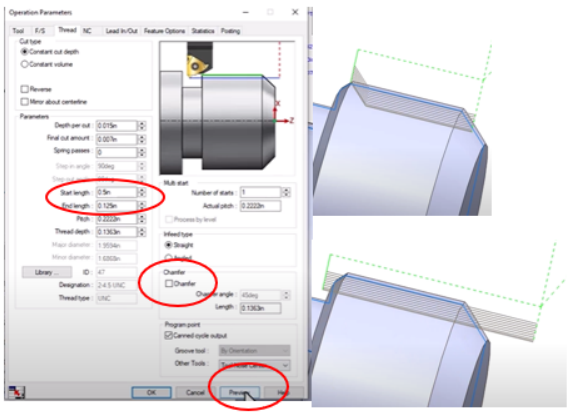


6. Generate Toolpath >Simulate Toolpath and Post Process

1. OD Feature Thread >
Generate Toolpath

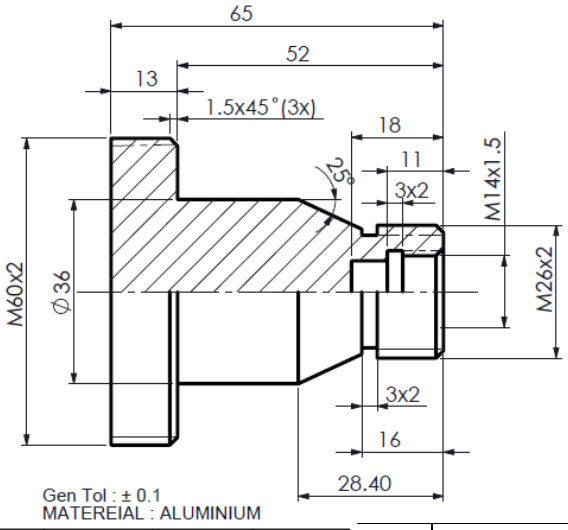


2. OD Feature Thread > Thread >Change the
Start Length: End Length: No Chamfer: :>
Preview > OK



3. OD Feature Thread >
Simulate Toolpaths >
Post process

14.3. CAM EXERCISE 6

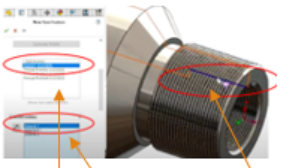


Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	
2	IT1.5.12_10	1.5	80	
3	ODL55.4	0.12	250	
5	OGL3.12	0.008	120	
6	IDL55.4_10	0.1	200	3000
7	OT2.12	2	80	
8	IDL80.8_10	0.2	160	
10	FD20_3	1.5	62.84	
12	OGL3.12	0.008	120	

SIDE B
Sequence of operation:
1.Drill
2.Face (Rough)
3.Face (Finish)
4.OD Longitudinal (Rough)
5.OD Longitudinal (Finish)
6.ID Longitudinal (Rough)
7.ID Longitudinal (Finish)
8 ID Grooving
8.OD Threading
9.OD Threading
10.ID Threading

Basic Steps SolidWorks CAM Process

- ❑ Define Machine (**Fanuc T2nx**)
- ❑ Define Stock Manager
- ❑ Define Coordinate System (**Work offset:G54**)
- ❑ Extract Machinable Features
  - Generate Operation Plan
  - Generate Tool Path
  - Simulate Toolpaths
- ❑ Post Processer



Create Single Feature (Example: OD Threading)

- Turn Setup 1 > Turning Operation > Thread > Feature > Create Feature
- Type: OD Strategy : Thread > Sketch > Sketch entity > Line D1 > OK
- Generate Operation Plan > Generate Toolpath >Simulate Toolpath

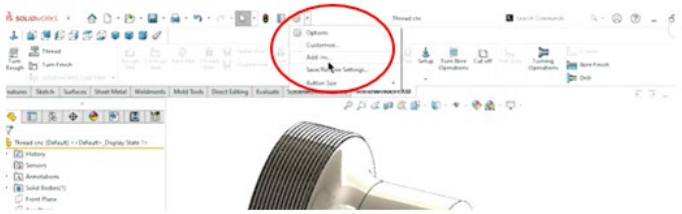
- ❑ Post Processer

Edit Tool (Select : CAM Tool Tree)

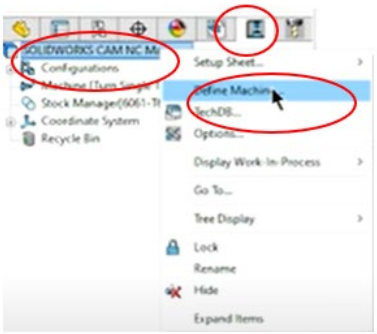
- Turn Rough >Edit definition…

1. Activate SolidWorks CAM

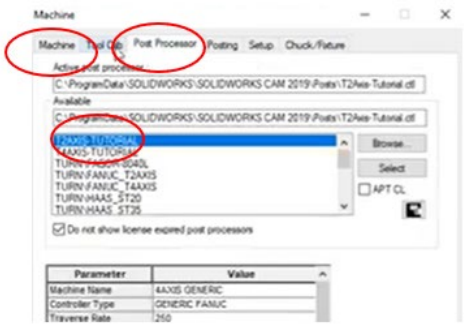
1.SolidWorks CAM > Option > Add in > Activate SolidWorks CAM



2.Right Click SolidWorks NC Manager>
Define Machine

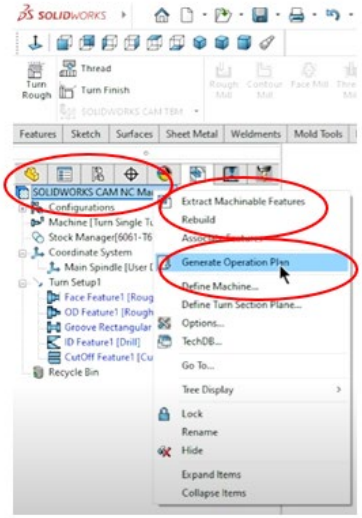


3.Machine> Turn Machine >Select Turning Single Turret
Post Process > Select **Fanuc T2nx**

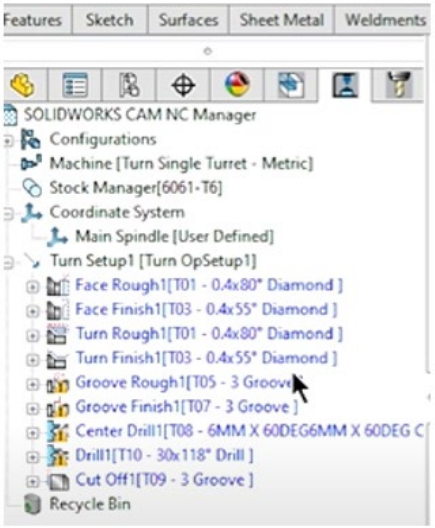


2. Extract Machinable Features, Generate Operation Plan & Toolpath

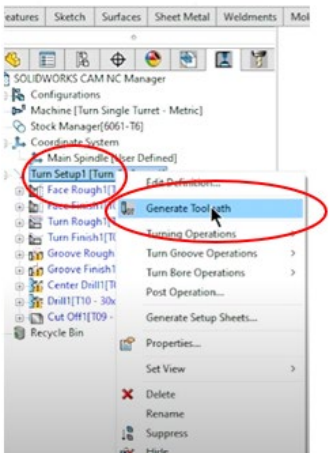
1.Select SolidWorks CAM NC >
Extract Machinable Features



2. Generate Operation Plan

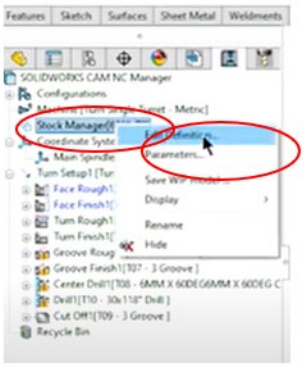


3.Select Turn Setup 1 >
Generate Toolpath

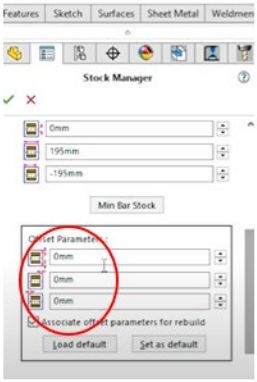


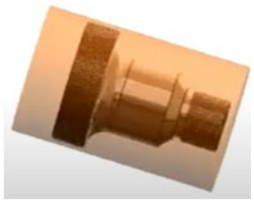
3. Define Stock Manager & Coordinates System (G54)

1.Right Click **Stock manager** > Edit Definition

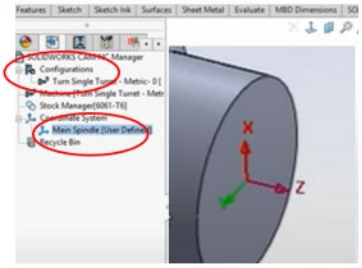


2. Offset Parameter >Dia:10,Face:10,Back :50mm>OK

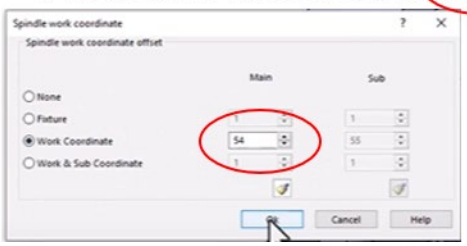




3. Coordinates System > Main Spindle (User Define)



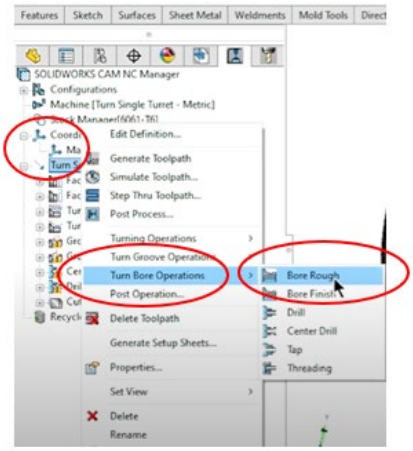
> 5. Work Offset >Work Coordinate : **54**



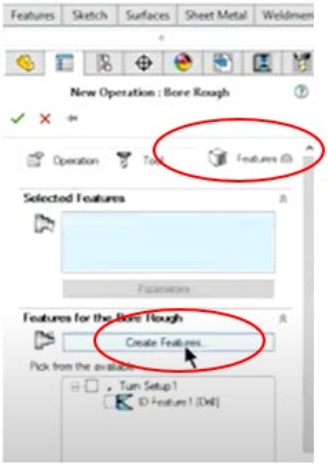
4. Create Boring Feature

Program For Boring Operation

1.Right click **Turn Setup 1**> Turning Operation > Bore Rough

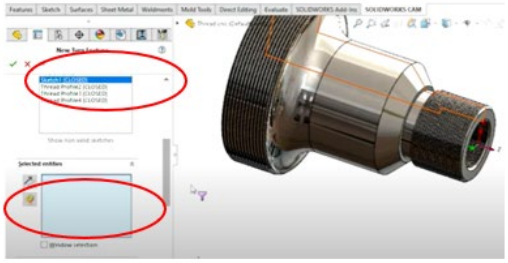


2. Feature> Create Feature

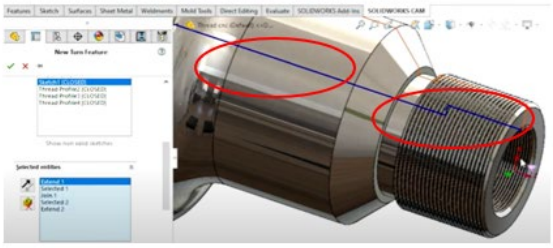


5. Select ID Feature and Create ID Tool

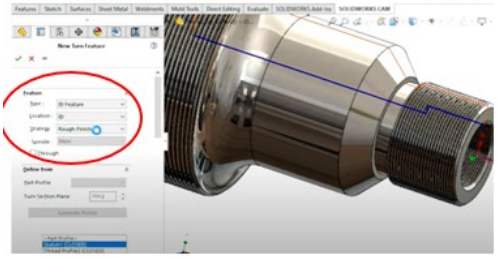
1. Select **Sketch**> Select **Sketch Entity** >



2. Click the line D1,Line D2.. > OK

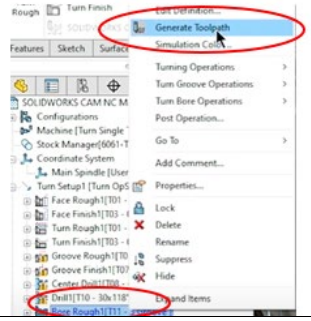


3. Type : ID Feature
Strategy : Rough Finish >



4.Tool >Add New>
ID Tool

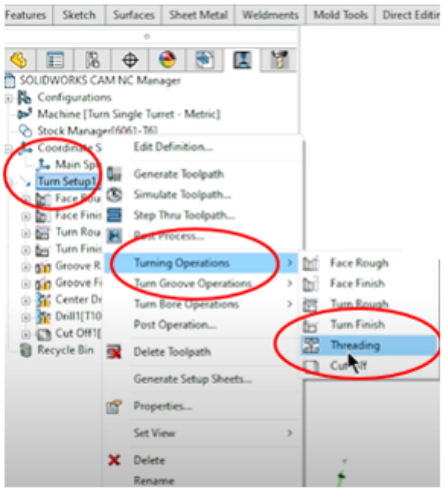
5. Drag to place the tool after drilling > **Generate Toolpath**



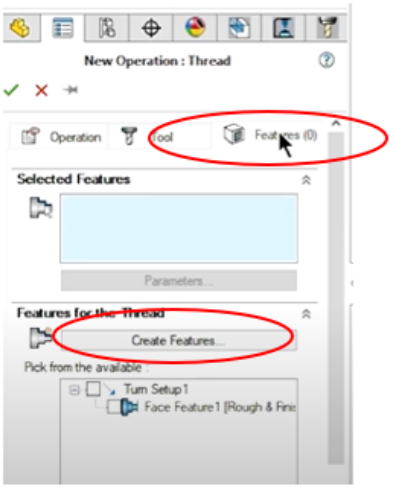
6. Create OD Thread Feature

Program For OD Threading (Pitch= 2mm)

1. Right click **Turn Setup 1**>
Turning Operation > Threading

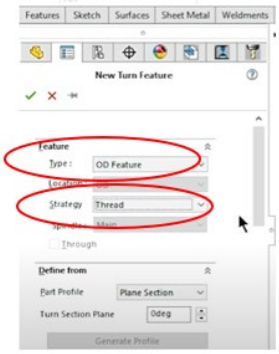


2. Feature> Create Feature

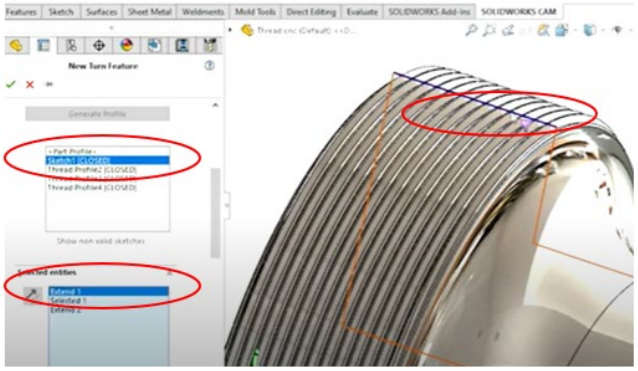


7. Select OD Thread Feature and Create Threading Tool-2mm

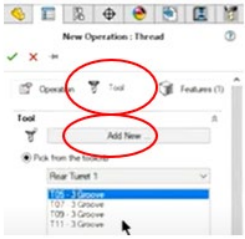
1. Type: OD Feature
Strategy : Thread



2.Select **Sketch**> Select **Sketch Entity**> Click the line D1 > OK

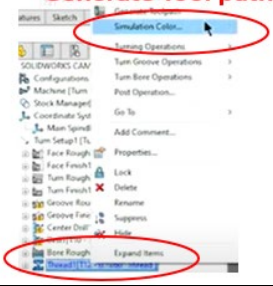


3.Tool > Add New >
Thread Tool >OK



4.Pitch : 2mm >
Depth:2mm

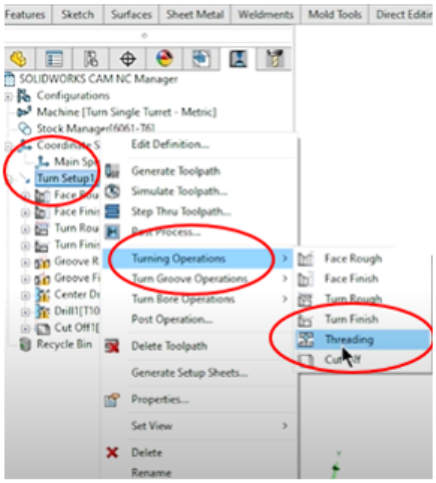
5. Right Click Thread Tool>
Generate Tool path



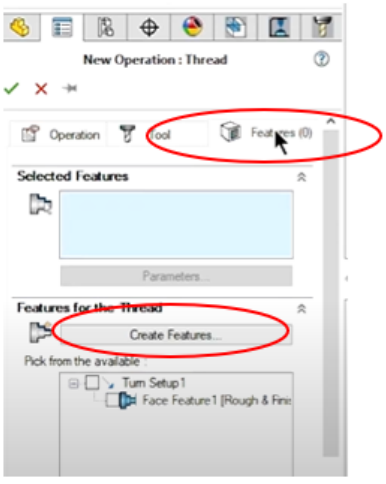
8. Create OD Thread Feature

Program For OD Threading (Pitch= 2mm)

1. Right click **Turn Setup 1**>
Turning Operation > Threading

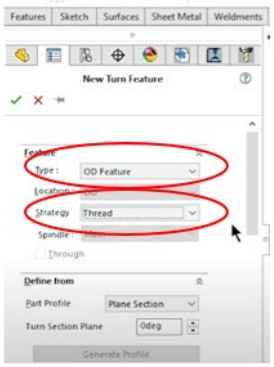


2. Feature> Create Feature

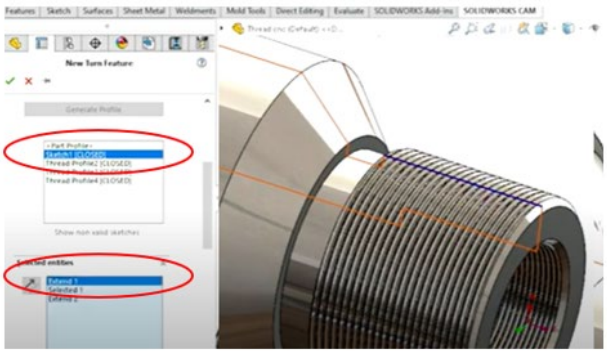


9. Select OD Thread Feature and Create Threading Tool-2mm

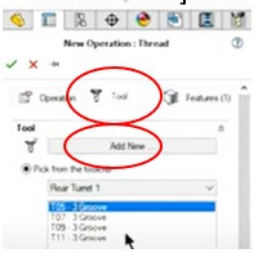
1. Type: OD Feature
Strategy : Thread



2. Select **Sketch**> Select **Sketch Entity**> Click the line D1 > OK

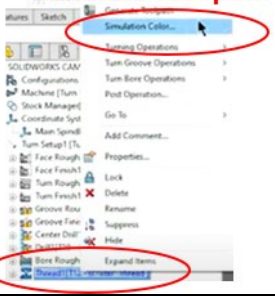


3. Tool > [Add New > Thread Tool] >OK



4. Pitch : 2 mm > Depth:2 mm

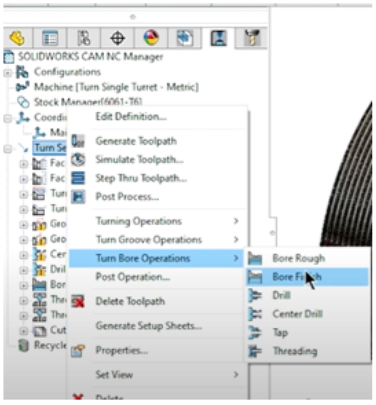
5. Right Click Thread Tool> **Generate Tool path**



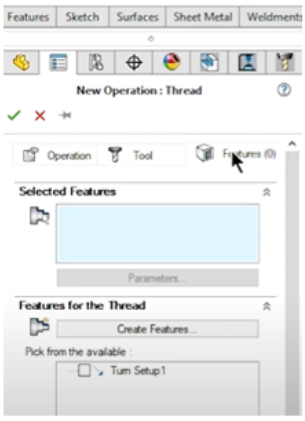
10. Create ID Thread Feature

Program For ID Threading (Pitch= 1.5mm)

1. Right click **Turn Setup 1**> Turning **Bore** Operation > Threading

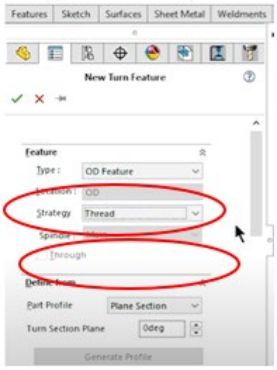


2. **Feature**> Create Feature

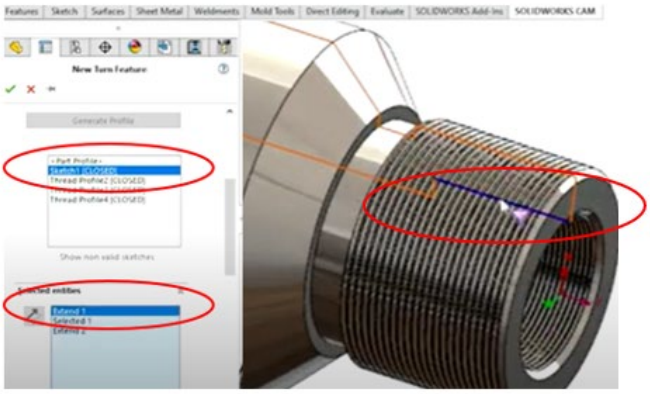


11. Create ID Thread Feature and Trading Tool-1.5mm

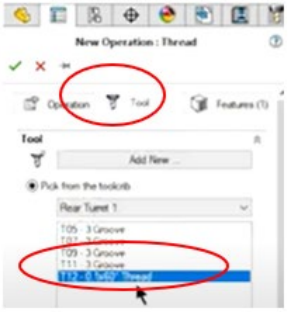
1. Type: ID Feature
Strategy : Thread



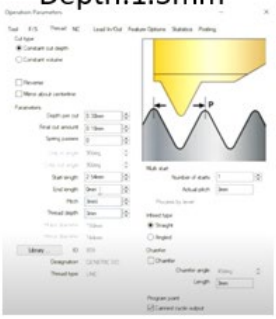
2. Select **Sketch**> Select **Sketch Entity**> Click the line D1 > OK



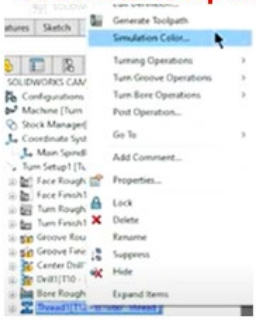
3. Tool > Add New >
Thread Tool >OK



4. Pitch : 1.5mm >
Depth:1.5mm

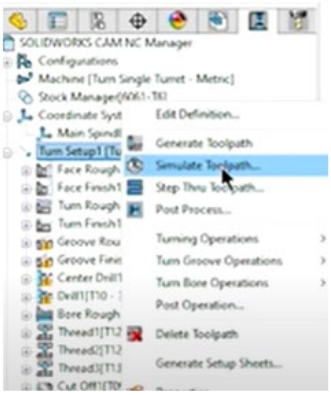


5. Right Click ID Thread>
Generate Toolpath



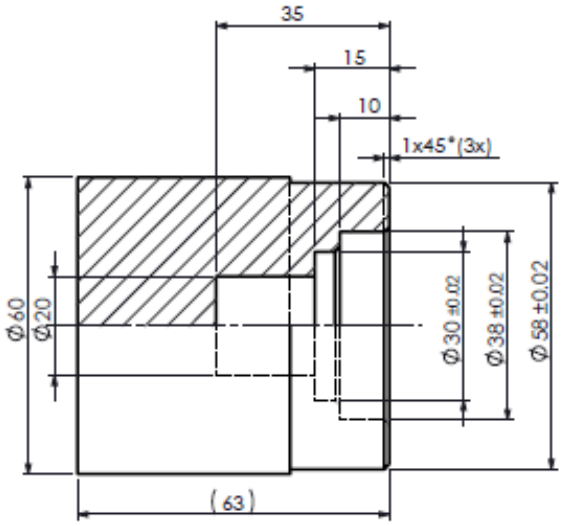
12. Simulate Toolpath

1. Right click **Turn Setup 1**>
Simulate Toolpath>



2. Play > **Simulate Toolpath>**

14.4. CAM ASSIGNMENT 1



Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM
BLANK : DIA 60x 65mm

Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	
6	IDL55.4_16	0.1	200	
8	IDL80.8_16	0.2	160	
10	FD20_3	0.1	62.84	

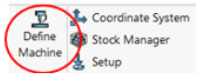
SIDE A
Sequence of operation:
1.Drill
2.Face (Rough)
3. Face (Finish)
4. OD Longitudinal (Rough)
5. OD Longitudinal (Finish)
6. ID Longitudinal (Rough)
7. ID Longitudinal (Finish)

ID BORING OPERATION

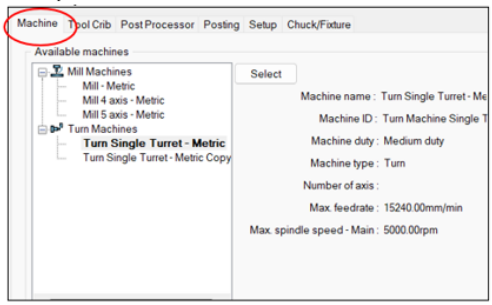
1.

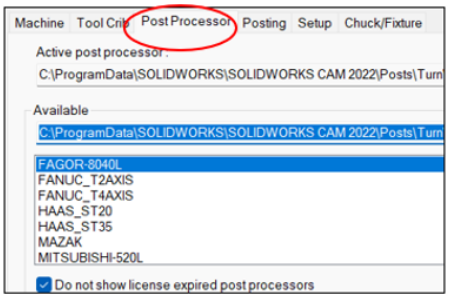
1.SolidWorks CAM > Option > Add in > Activate SolidWorks CAM

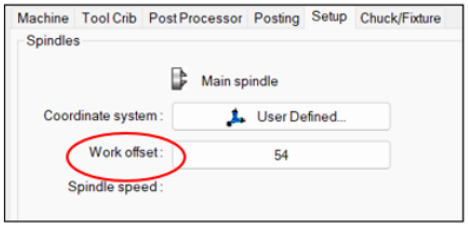
2. Define Machine



3.Machine:Select Turning Single Turret >Select
Post Process: Fanuc T2nx
Setup : Work offset:54

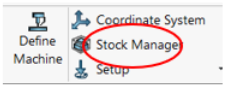




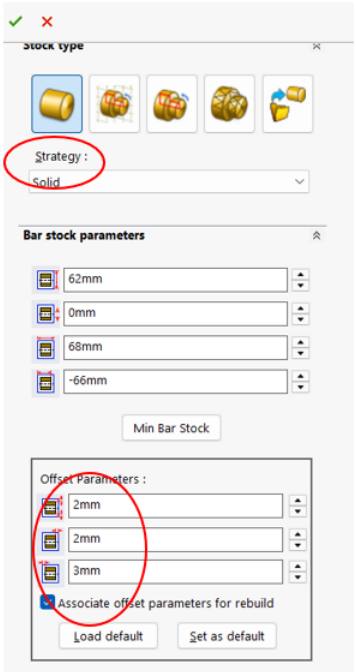


2.

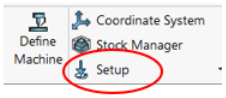
1. Define Manager



2.Strategy: Solid
Offset Parameters: 2,2,3



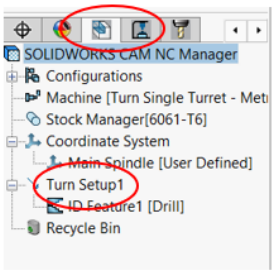
3. Setup > OK

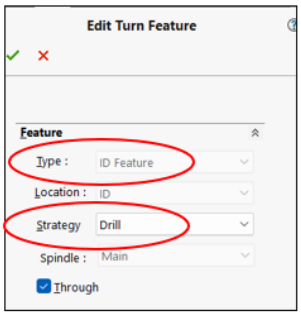


3.

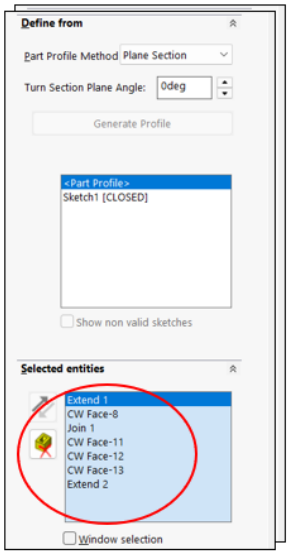
1.CAM Feature Three > **Turn Setup 1**>Turn Feature >

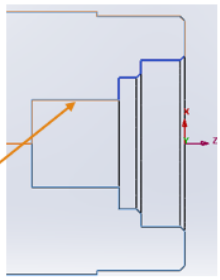
Type: ID Feature
Strategy :Drill





2.Click Selected entities >
Click Lines for drill, D1

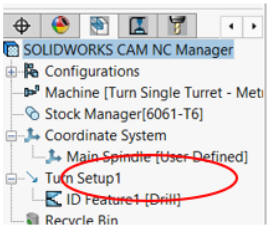




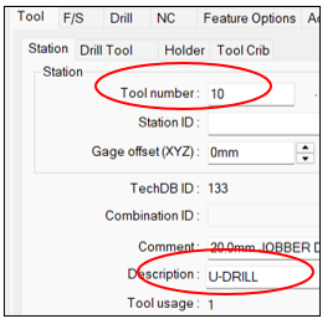
Turn Bore Operation > Drill

4.

1.ID Feature [Drill] > Turn Bore Operation >Drill

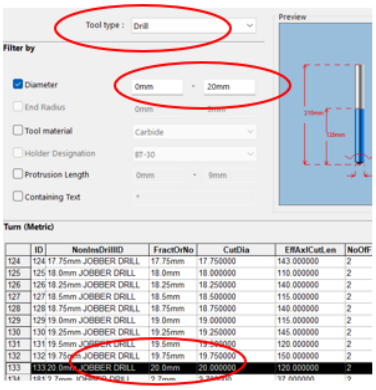


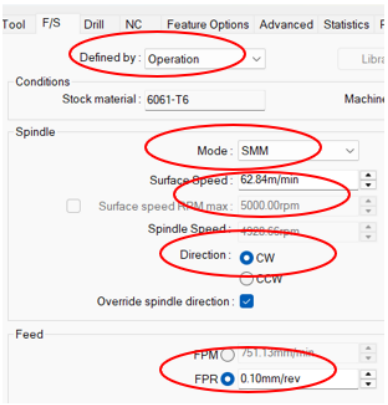
2.1 Tool >Station> Tool number :10 > Description :UDRILL



2.2 F/S>Defined by: Operation >
Mode :SMM
S Speed : 62.84 m/min >
Direction :CW
Feed :0.1 mm/rev

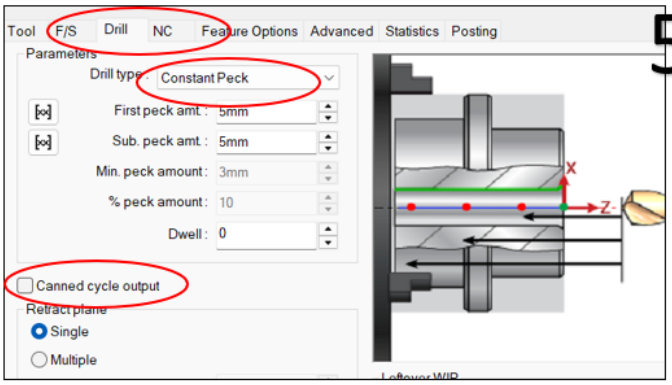
2.Add New >Tool Type
:Drill >Diameter :20mm



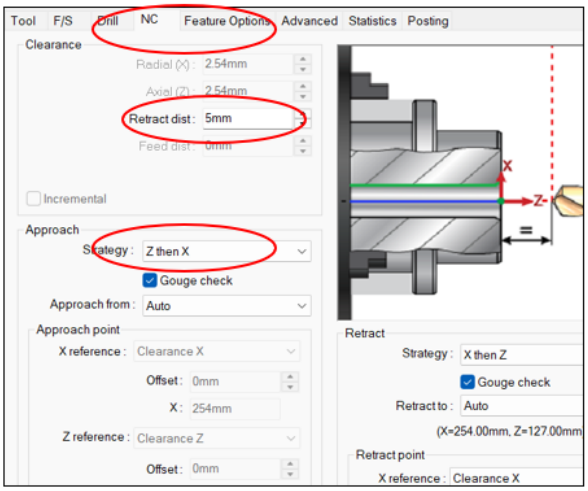


5.

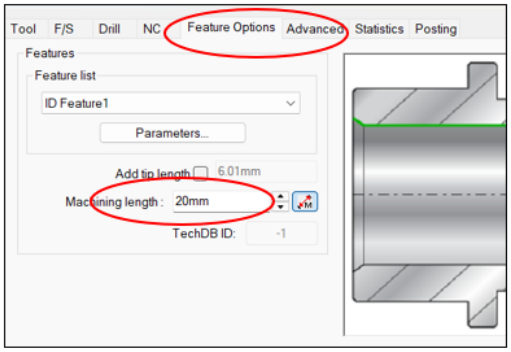
2.3 Drill >Dill type: Constant Pack >
- Canned cycle output



2.4 NC > Clearance:>
Retract dist :5mm >
Approach > Strategy :Z then X

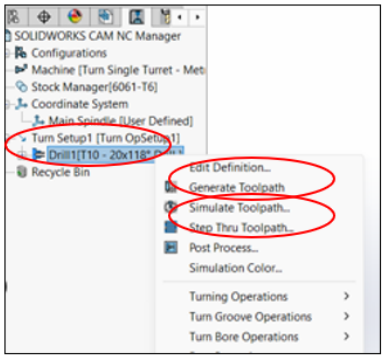


2.5 Feature Option > Machining length:20m

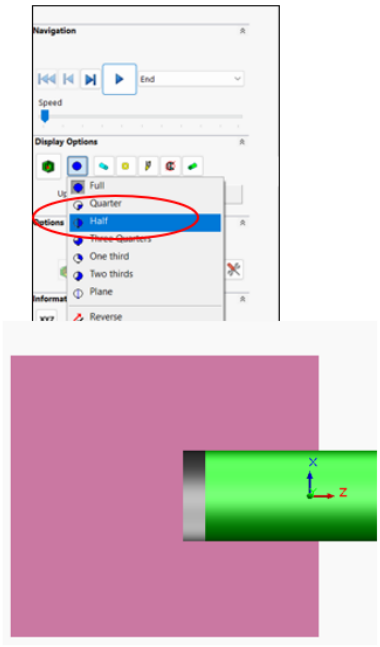


2.7 Drill > Generate Toolpath > Simulate Toolpath>

6.



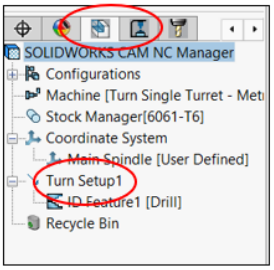
Half Section > Play

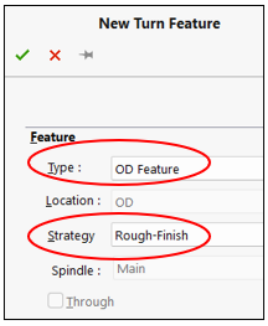


1.CAM Feature Three > **Turn Setup 1**>Turn Feature >

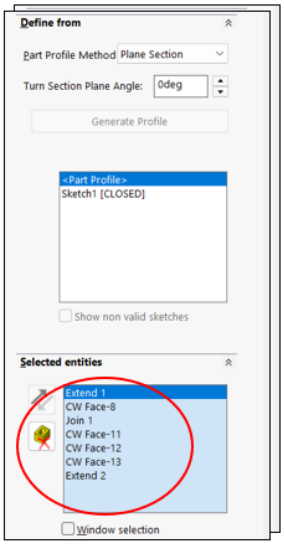
Type: ID Feature
Strategy :Rough Finish

7.

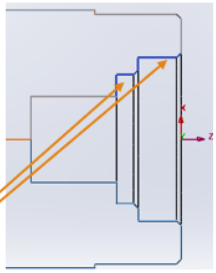




2.Click Selected entities >
Click Lines for bore, D1



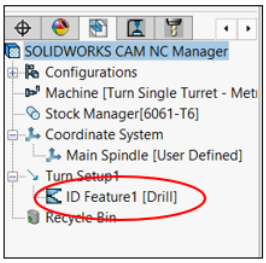
Lines, D1



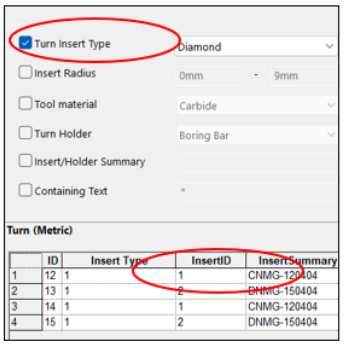
Turn Bore Operation > Bore Rough

8.

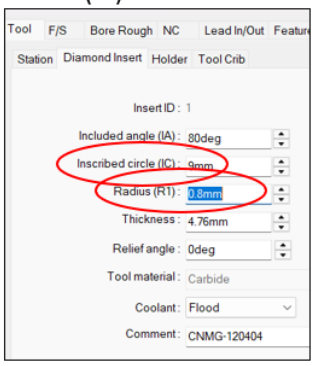
1.CAM Feature Tree >
ID Feature [Drill] > Turn Bore Operation >Bore Rough



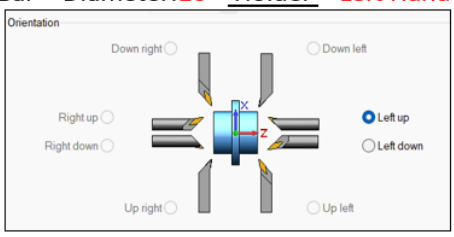
2.Add New >Turn Insert Type : Diamond > CNMG..



2.1 Tool>
Station >Tool Number:8
Diamond Insert >
Inscribed circle(IC): 9
Radius (RI) :0.8 mm

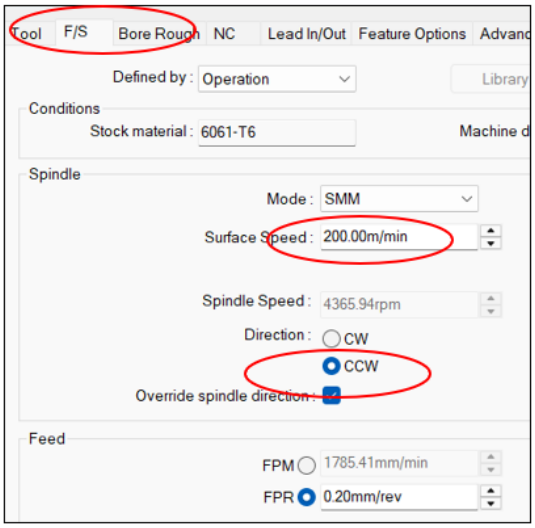


2.1 Boring Bar > Diameter:16 > Holder > Left Hand

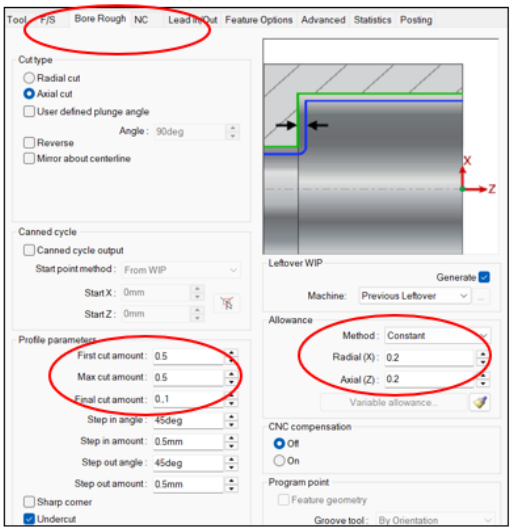


9.

2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :200 m/min
Direction :CCW
Feed : 0.20mm/rev

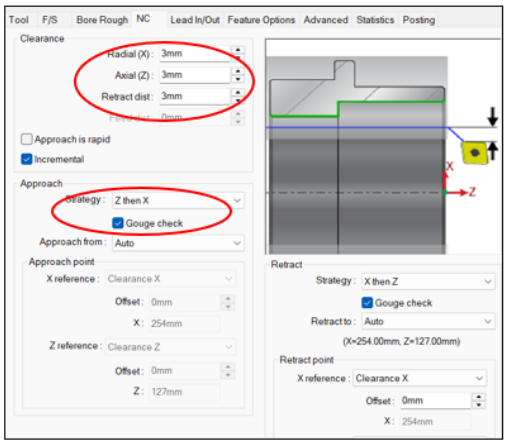


2.3 Bore Rough>
Profile Parameters:
First cut amount :0.5
Max cut amount :0.5
Final cut amount :0.1
Allowance::
Method :Constant
Radial(X) :0.2mm
Axial (Z):0.2 mm

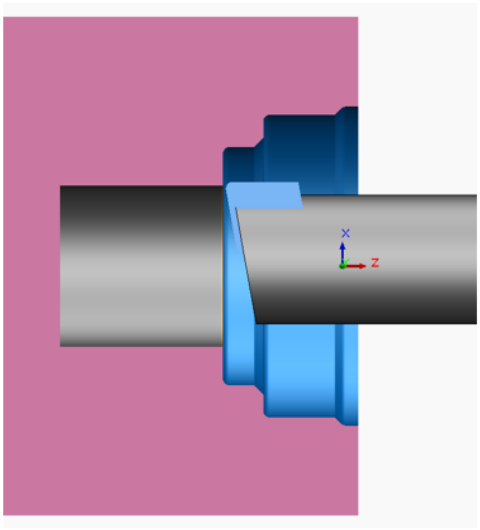


10.

2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X



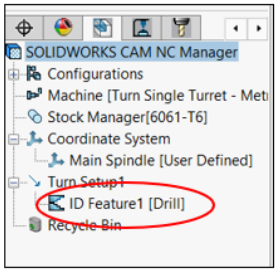
3. Turn Setup 1 >
Generate Toolpath>
Generate Simulation >
Half Section > Play



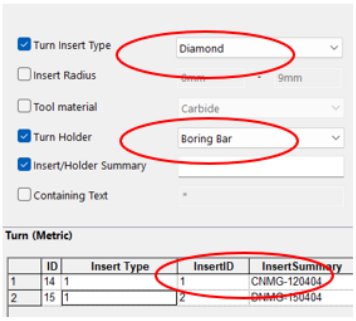
Turn Bore Operation > Bore Finish

11.

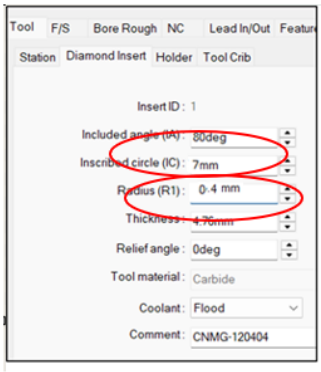
1.CAM Feature Tree >
Bore Rough > Turn Bore Operation >Bore Finish



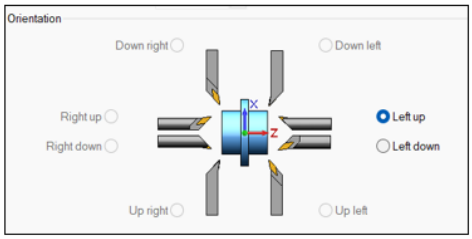
2.Add New >Turn Insert Type : Diamond > DNMG..



2.1 Tool>
Station >Tool Number:6
Diamond Insert >
Inscribed circle(IC): 7
Radius (RI) :0.4 mm

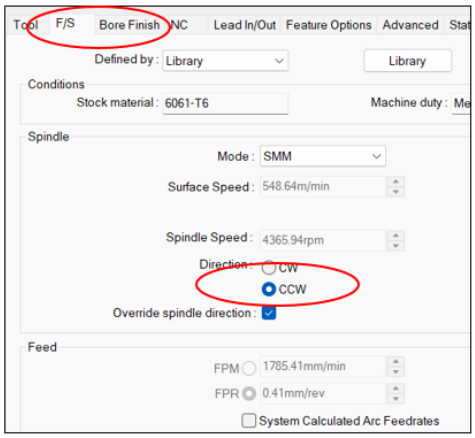


2.1 Boring Bar > Diameter:16 > Holder > Left Hand

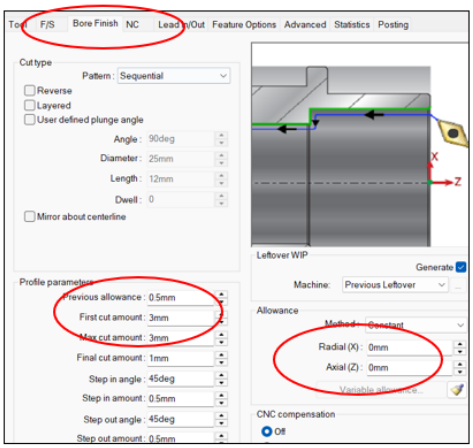


12.

2.2 F/S>
Defined by :
Mode :
Surface Speed :
Direction :CCW
Feed :

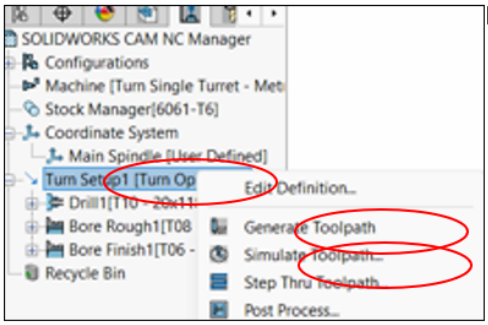


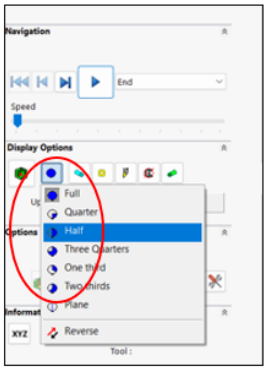
2.3 Bore Finish>
Profile Parameters:
First cut amount :0.5mm
Max cut amount :3mm
Final cut amount :1mm
Allowance::
Method :Constant
Radial(X) :0
Axial (Z): 0

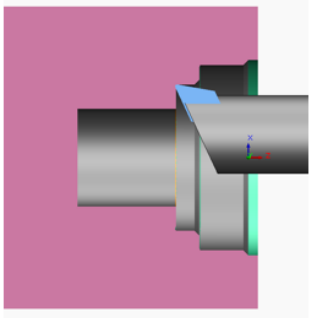


13.

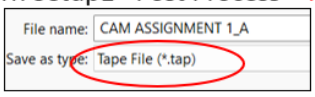
1. Turn Setup1 > Generate Toolpath > Simulate Toolpath> Half Section > Play

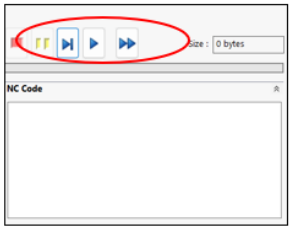


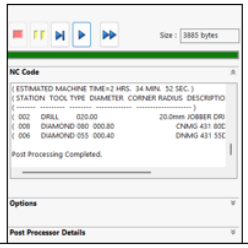




2. Turn Setup1> Post Process > Play > Save as Tape File or Text File > Play





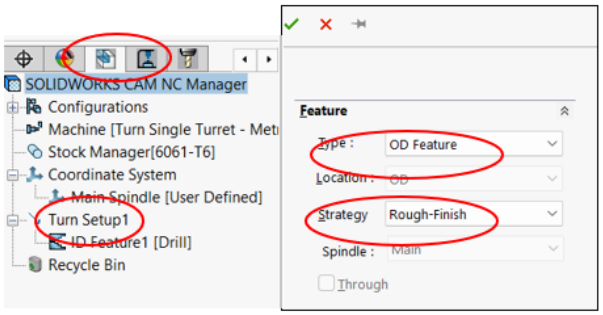


OD Operation

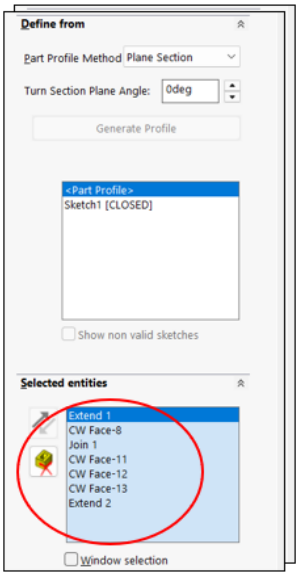
1.CAM Feature Three > **Turn Setup 1**>Turn Feature >

Type: OD Feature
Strategy :Rough Finish

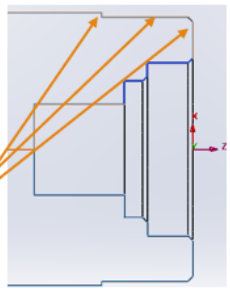
1.



2.Click Selected entities >
Click Lines for OD, D1



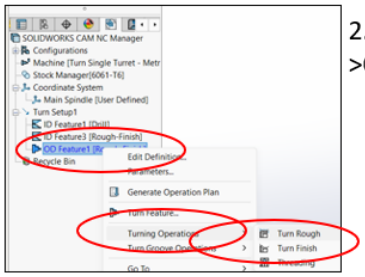
Lines, D1



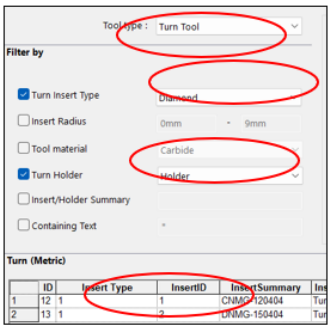
Turn Operation > Turn Rough

2.

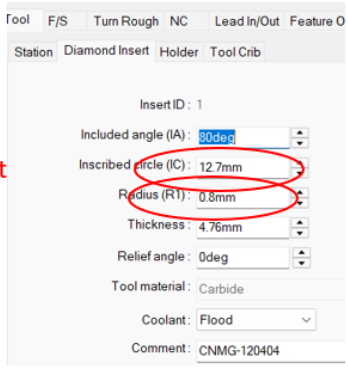
1.CAM Feature Tree >
OD Feature > Turn Operation >Turn Rough



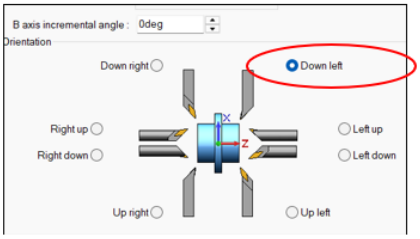
2.Add New >Tool Type>Turn Insert Type> : Diamond> Holder >CNMG..



2.1 Tool>
Station >Tool Number:1
Diamond Insert >
Inscribed circle(IC): 12
Radius (RI) :0.8 mm

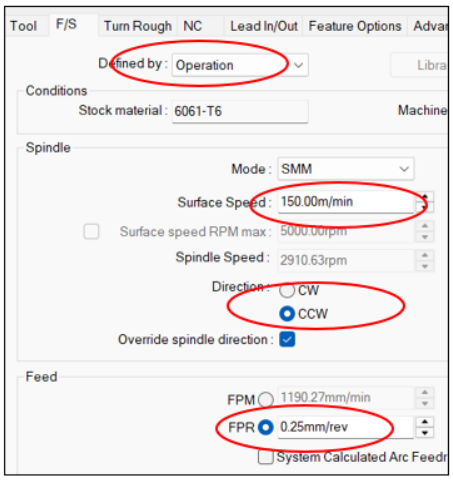


2.1 Holder> Shank With:16 > Orientation>Down Left

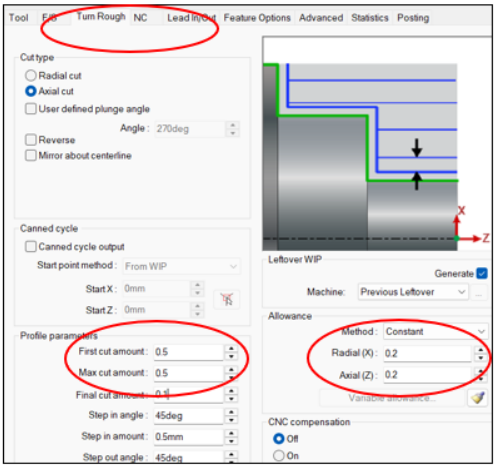


3.

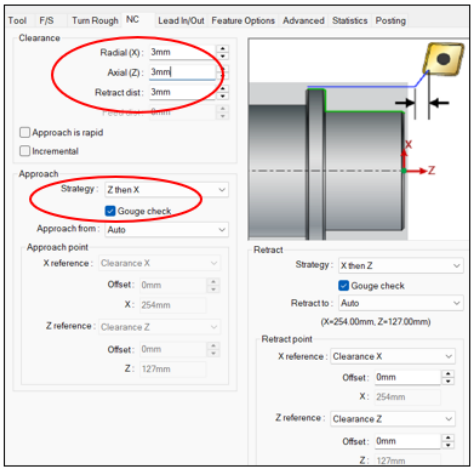
2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :150m/min
Direction :CCW
Feed : 0.25mm/rev



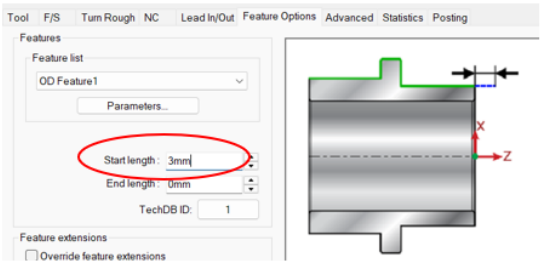
2.3 Turn Rough>
Profile Parameters:
First cut amount :0.5
Max cut amount :0.5
Final cut amount :0.1
Allowance::
Method :Constant
Radial(X) :0.2mm
Axial (Z):0.2 mm



2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X



2.5 Feature Options > Start Length : 3mm **4.**

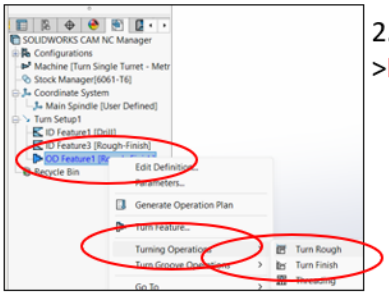


3. Move item after Facing Operation
Turn Setup 1 > Generate Toolpath>
Generate Simulation >
Half Section > Play

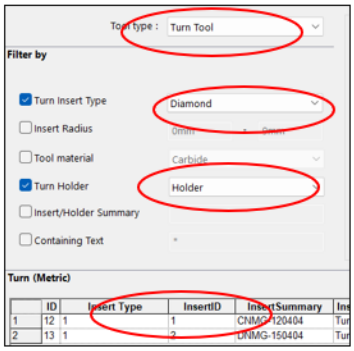
Turn Operation > Turn Finish

5.

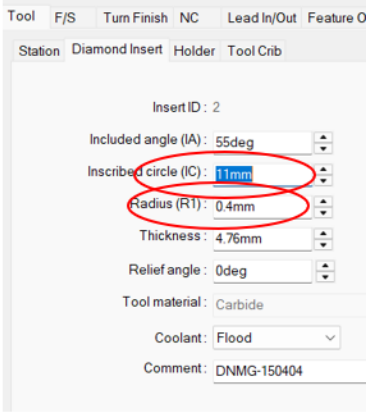
1.CAM Feature Tree >
Turn Rough > Turn Operation >Turn Finish



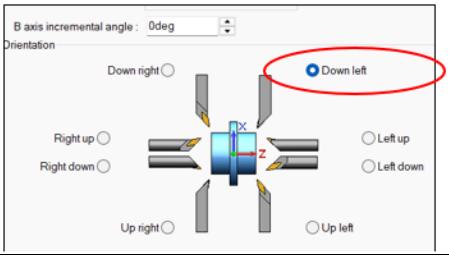
2.Add New >Tool Type>Turn Insert Type> : Diamond> Holder >DNMG..



2.1 Tool>
Station >Tool Number:3
Diamond Insert >
Inscribed circle(IC): 11
Radius (RI) :0.4 mm

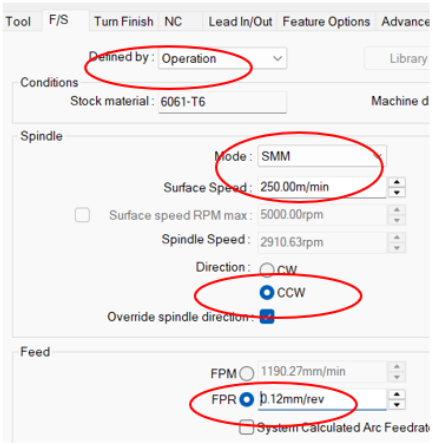


2.1 Holder> Shank With:16 > Orientation>Down Left

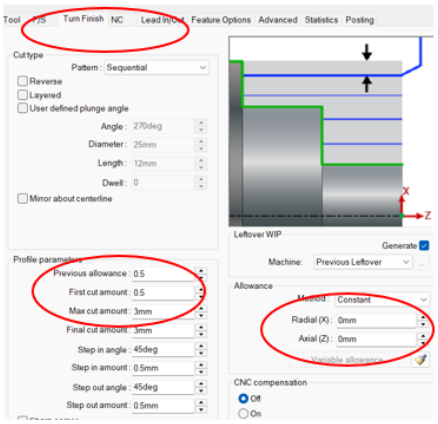


6.

2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :250m/min
Direction :CCW
Feed : 0.12mm/rev

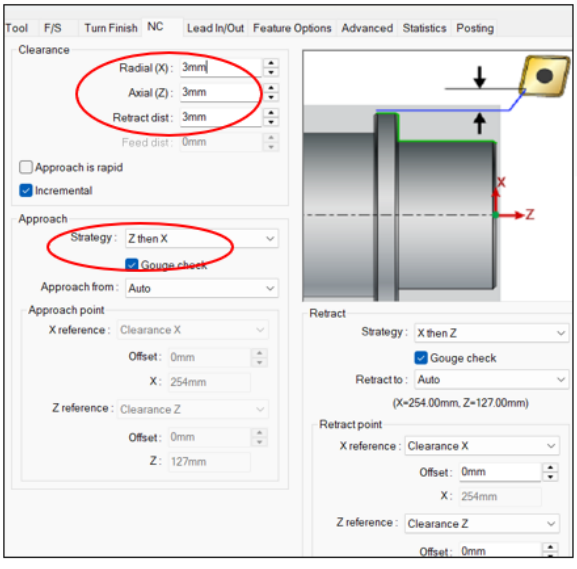


2.3 Turn Finish>
Profile Parameters:
Previous Cut : 0.5
First cut amount :0.5
Max cut amount :3mm
Final cut amount :3mm
Allowance::
Method :Constant
Radial(X) :0
Axial (Z):0

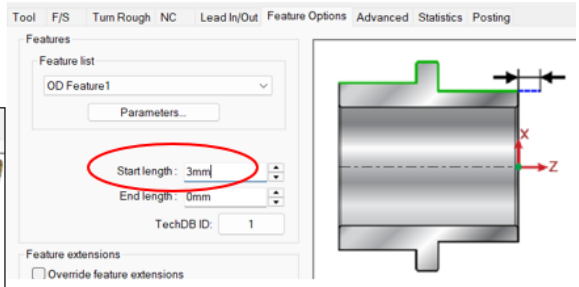


7.

2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X



2.5 Feature Options > Start Length : 3mm



3. Move item after **Turn Rough**
Turn Setup 1 > Generate Toolpath>
Generate Simulation >
Half Section > Play

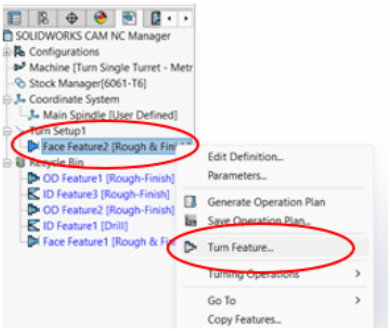
Facing Operation

Turn Operation > FACE Rough

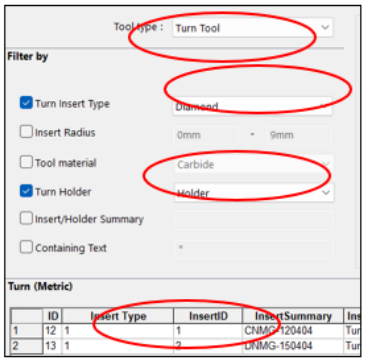
1.

1.CAM Feature Three

FACE Feature > Turn Operation >FACE Rough



2.Add New >Tool Type>Turn Insert Type> : Diamond> Holder >CNMG..



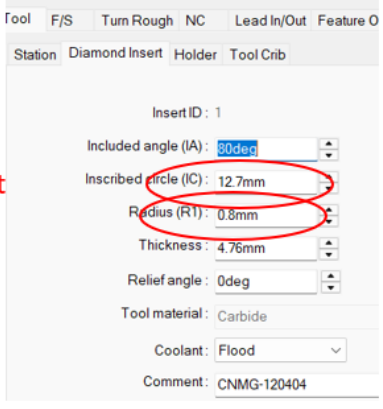
2.1 Tool>

Station >Tool Number:1

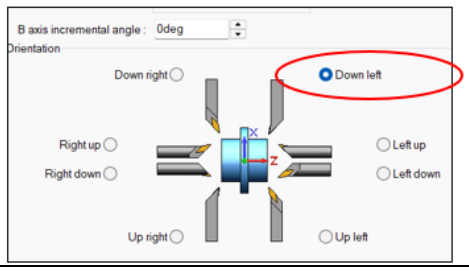
Diamond Insert >

Inscribed circle(IC): 12

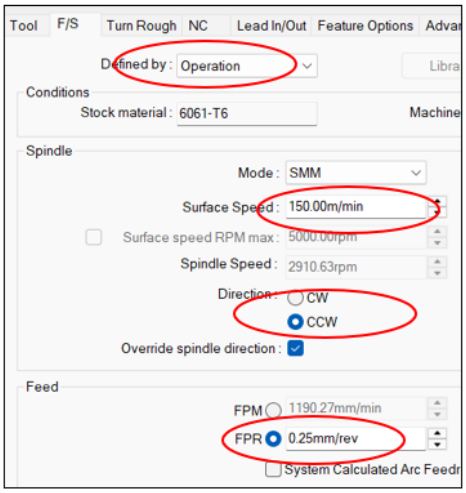
Radius (RI) :0.8 mm



2.1 Holder> Shank With:16 > Orientation>Down Left

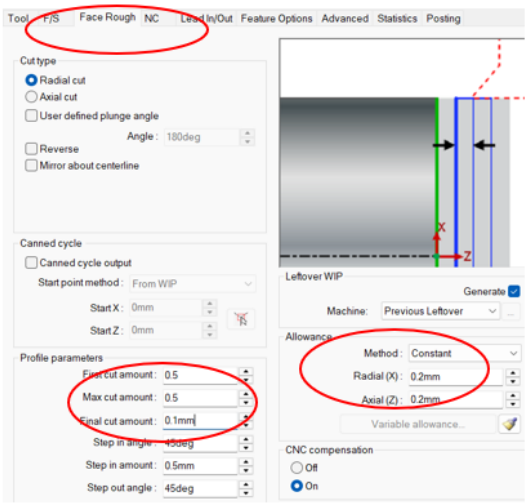


2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :150m/min
Direction :CCW
Feed : 0.25mm/rev

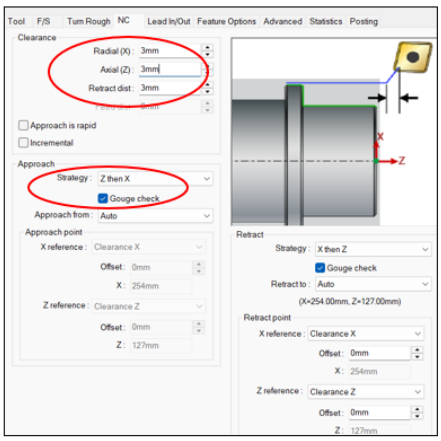


2.3 FACE Rough>
Profile Parameters:
First cut amount :0.5
Max cut amount :0.5
Final cut amount :0.1
Allowance::
Method :Constant
Radial(X) :0.2mm
Axial (Z):0.2 mm

2.



2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X



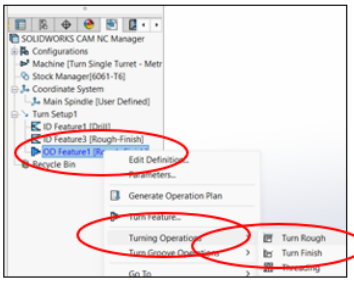
3.

3. Move item after Facing Operation
Turn Setup 1 > Generate Toolpath>
Generate Simulation >
Half Section > Play

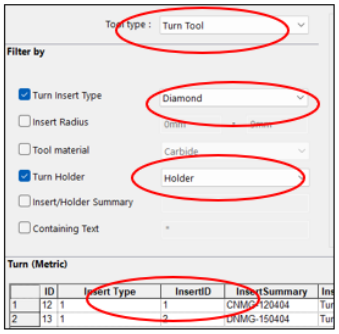
Turn Operation > FACE Finish

4.

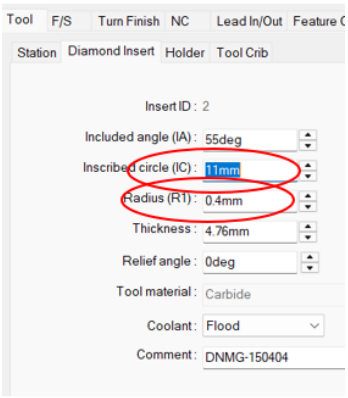
1.CAM Feature Tree >
FACE FINISH > Turn Operation >FACE Finish



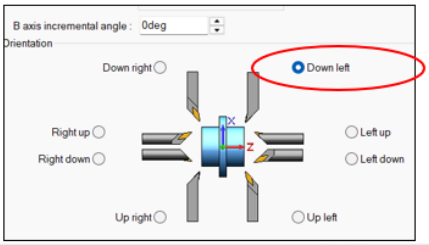
2.Add New >Tool Type>Turn Insert Type> : Diamond> Holder >DNMG..



2.1 Tool>
Station >Tool Number:3
Diamond Insert >
Inscribed circle(IC): 11
Radius (RI) :0.4 mm

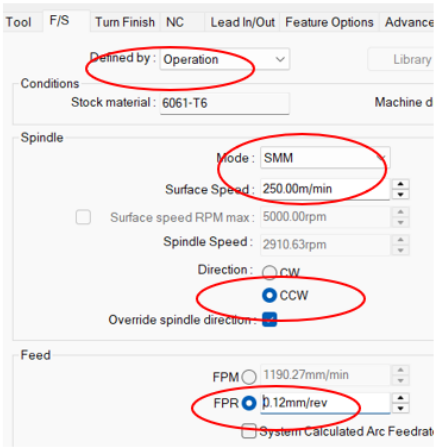


2.1 Holder> Shank With:16 > Orientation>Down Left

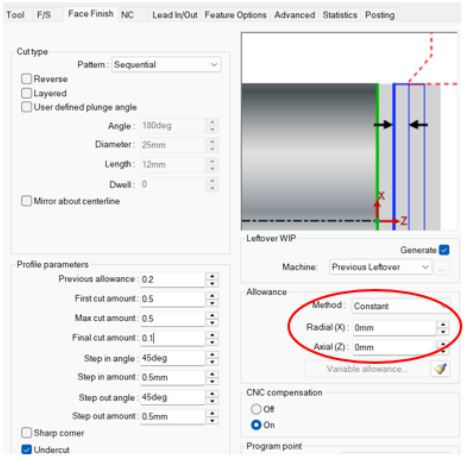


5.

2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :250m/min
Direction :CCW
Feed : 0.12mm/rev



2.3 FACE Finish>
Profile Parameters:
Previous Cut : 0.5
First cut amount :0.5
Max cut amount :3mm
Final cut amount :3mm
Allowance::
Method :Constant
Radial(X) :0
Axial (Z):0



6.

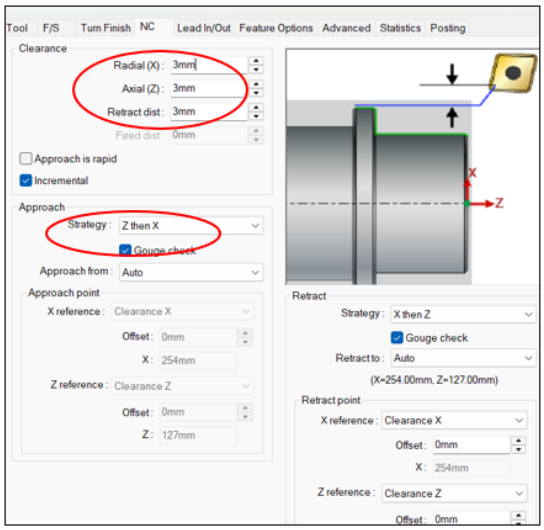
2.4 NC > Clearance >

Radial (X) :3

Axial (Z):3

Retract dist :3mm >

Approach > Strategy :Z then X



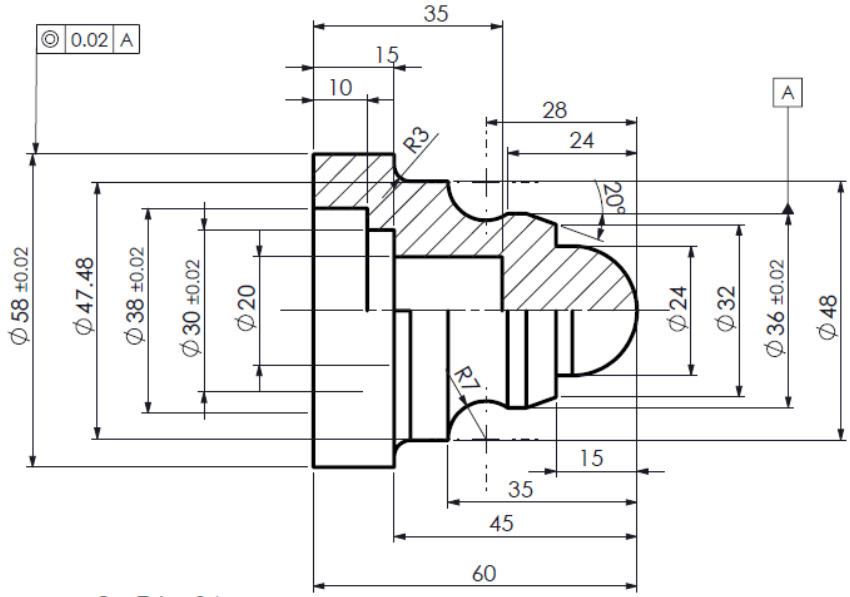
3. Move item after **Turn Rough**

Turn Setup 1 > Generate Toolpath>

Generate Simulation >

Half Section > Play

4. CAM ASSIGNMENT 1



Gen Tol : ± 0.1
MATEREIAL : ALUMINIUM
BLANK : DIA 60x 65mm

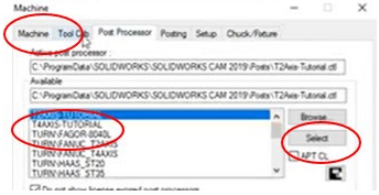
Turret No	Tool	Feed mm/rev	Cutting Speed M/min	Speed Limit RPM
1	ODL80.0	0.25	150	3000
3	ODL55.4	0.12	250	

SIDE B
Sequence of operation:
1.Face (Rough)
2.Face (Finish)
3.OD Longitudinal (Rough)
4.OD Longitudinal (Finish)

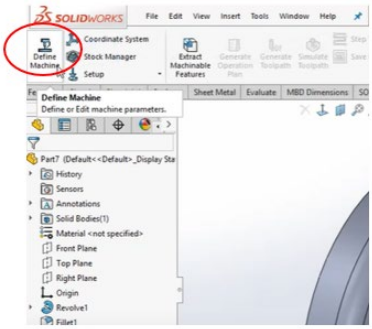
1.

1.SOLIDWORKS CAM >Option > Add in > Activate SolidWorks CAM

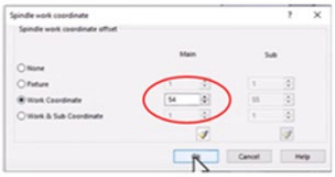
2.1 Machine> Turn Machine >
Select Turning Single Turret
2.2 Post Processor > Select **Fanuc T2nx**



2. Define Machine

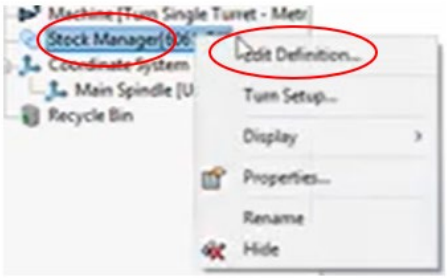


2.3 Work offset>Work Coordinate:**54**

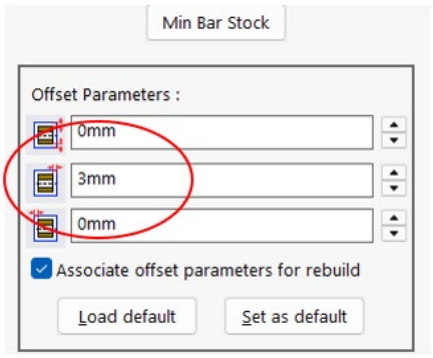


2.

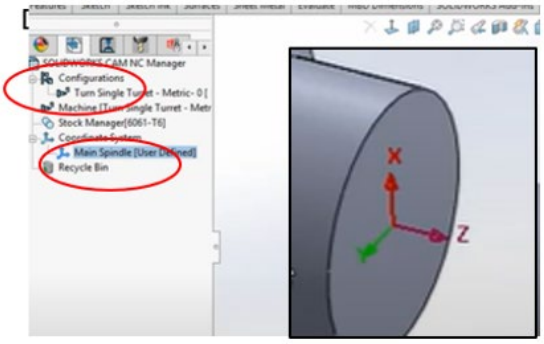
1. Right click **Stock Manager** > Edit Definition

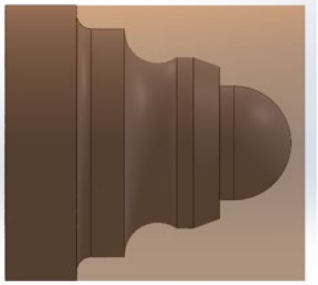


2. Offset Parameters: 2:2:3 > OK



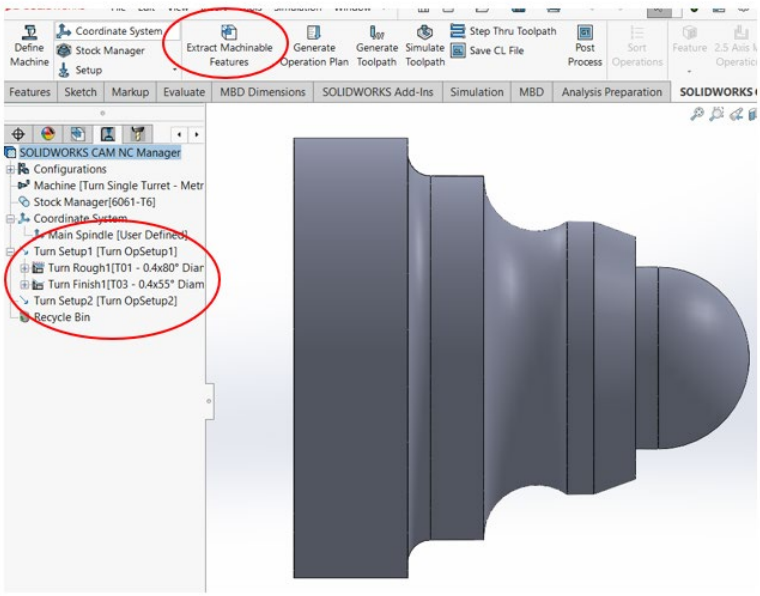
3. Coordinates System > Main Spindle (User Define)





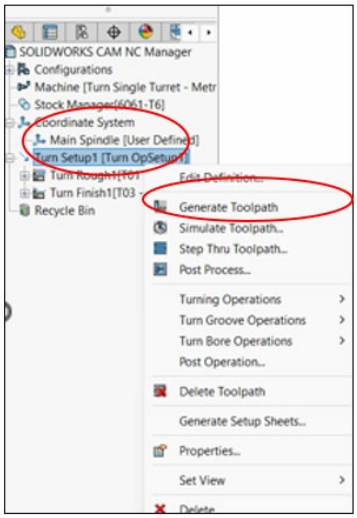
3.

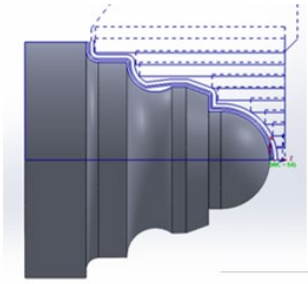
2. Extract Machinable Features (Turn Setting 1 ..appear automatically)



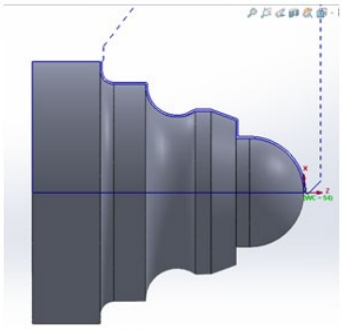
4.

1. Right click Turn Setup 1 > **Generate Tool Path**





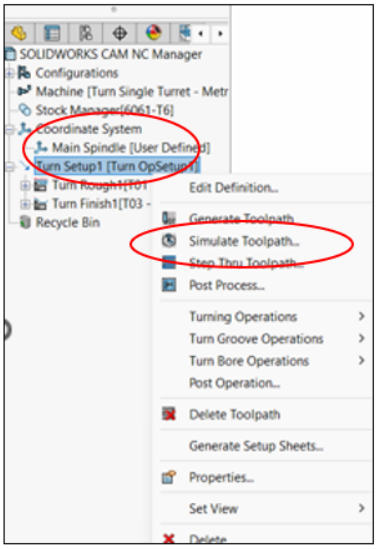
Turn Rough



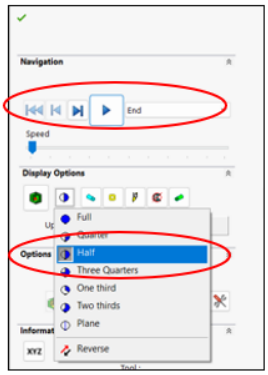
Turn Finish

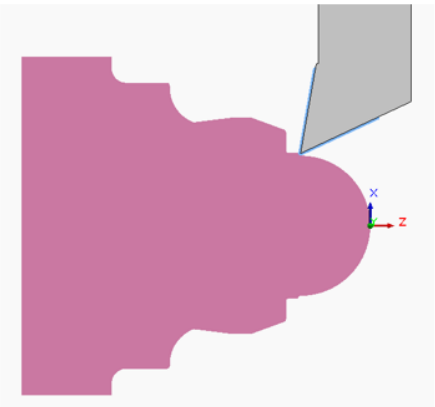
5.

1. Right click Turn Setup 1 > **Simulate Toolpath >**



2. Display half section > Play

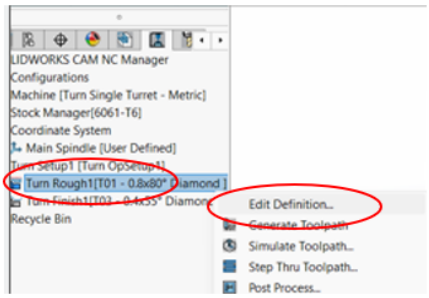




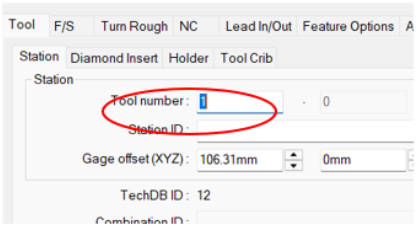
Edit Tool

6.

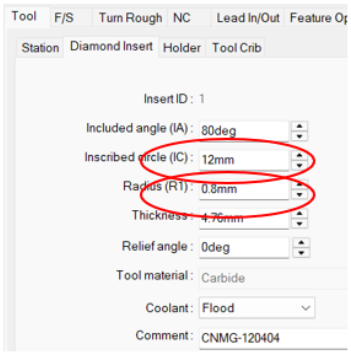
1. **Turn Rough** > Edit Definition>



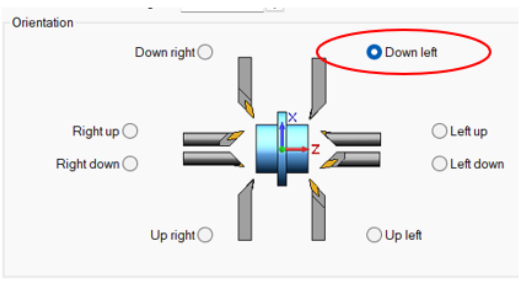
2.1 Tool>
Station >Tool Number:1



2.2 Tool>
Diamond Insert >(IC) :12 mm
(RI):0.8mm

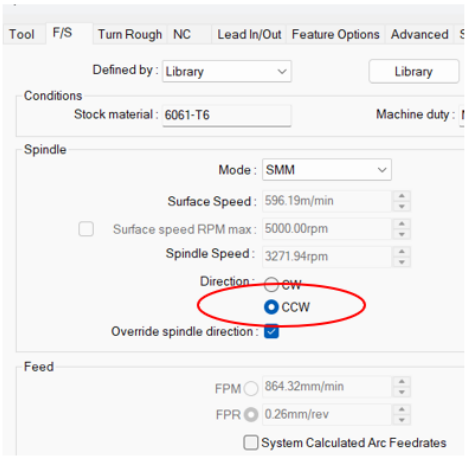


2.2 Tool>
Holder >Shank Width:16mm
Orientation : Down Left

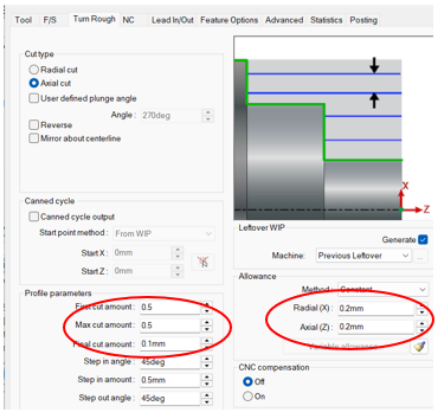


7.

2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :150m/min
Direction :CCW
Feed : 0.25mm/rev

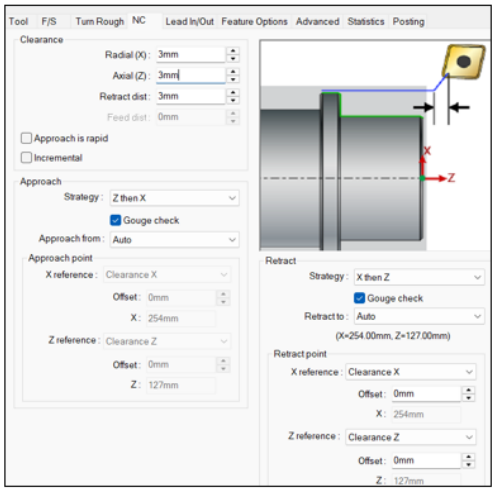


2.3 Turn Rough>
Profile Parameters:
First cut amount :0.5
Max cut amount :0.5
Final cut amount :0.1
Allowance::
Method :Constant
Radial(X) :0.2mm
Axial (Z):0.2 mm

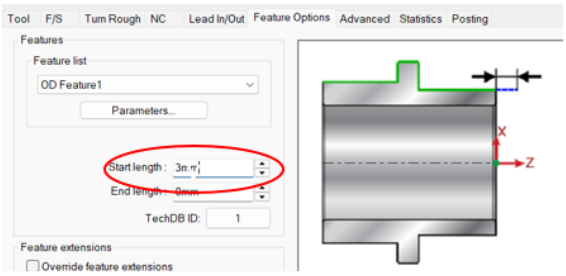


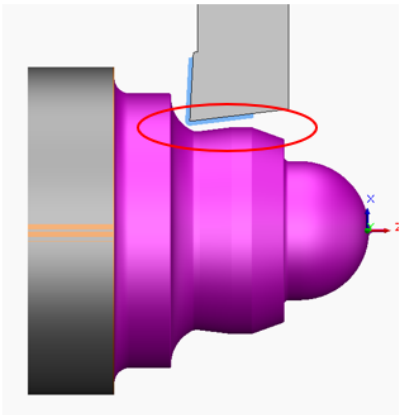
8.

2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X



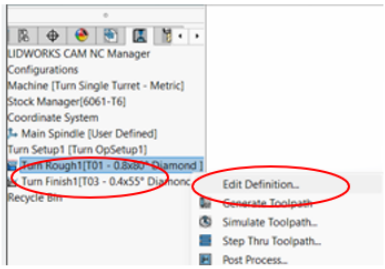
2.4 Feature Options > Start Length : 3mm



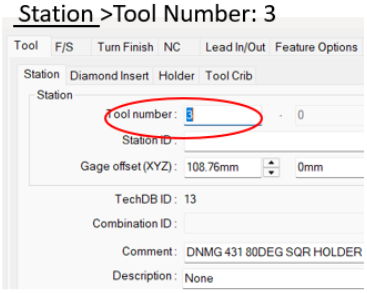


9.

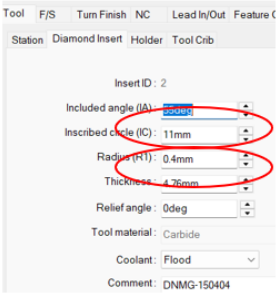
1. **Turn Finish** > Edit Definition>



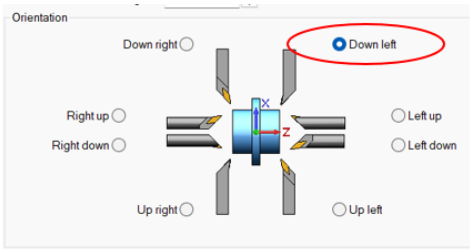
2.1 Tool>
Station >Tool Number: 3



2.2 Tool>
Diamond Insert >(IC) :11 mm
(RI):0.4mm

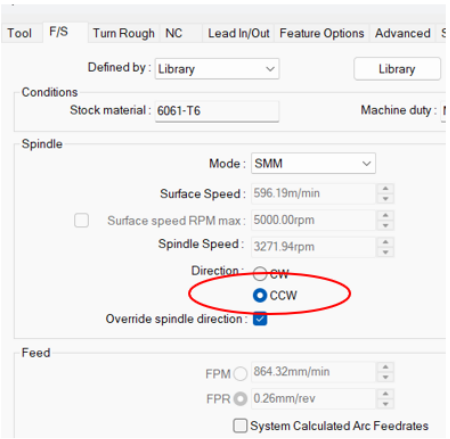


2.2 Tool>
Holder >Shank Width:16mm
Orientation : Down Left

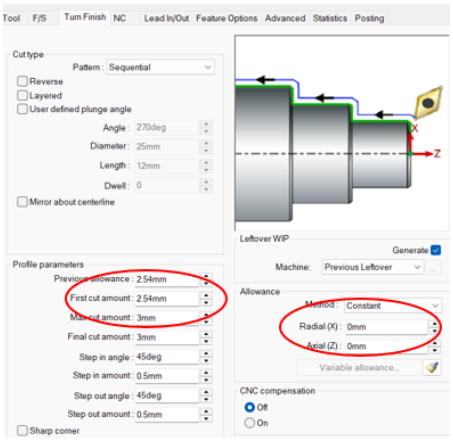


10.

2.2 F/S>
Defined by : Operation >
Mode : SMM
Surface Speed :150m/min
Direction :CCW
Feed : 0.25mm/rev

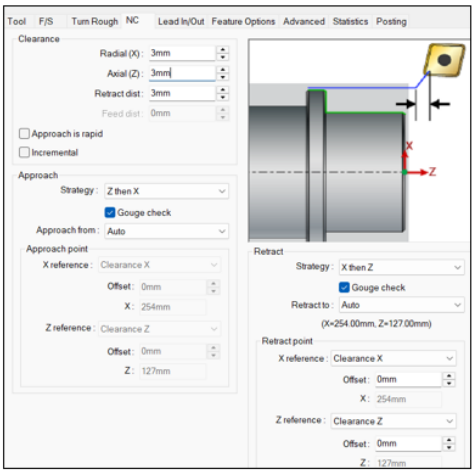


2.3 Turn Rough>
Profile Parameters:
First cut amount :0.5
Max cut amount :0.5
Final cut amount :0.1
Allowance::
Method :Constant
Radial(X) :0
Axial (Z):0

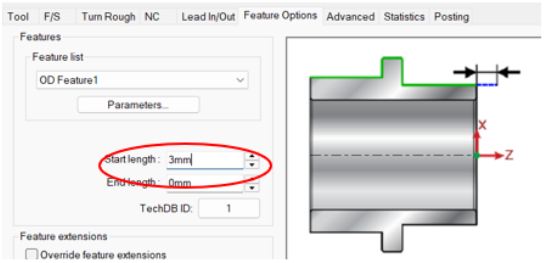


11.

2.4 NC > Clearance >
Radial (X) :3
Axial (Z):3
Retract dist :3mm >
Approach > Strategy :Z then X

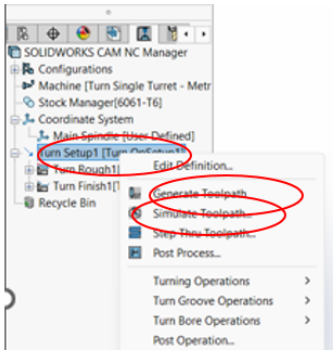


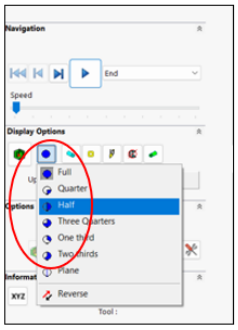
2.4 Feature Options > Start Length : 3mm

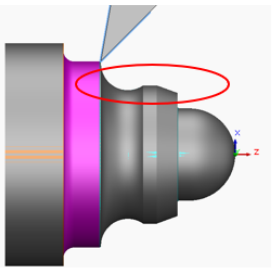


12.

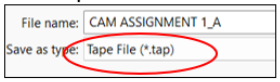
1. Turn Setup1 > Generate Toolpath > Simulate Toolpath>
Half Section > Play

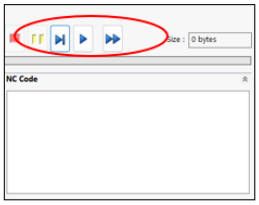


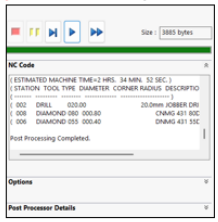




2. Turn Setup1> Post Process > Play > Save as Tape File or Text File > Play







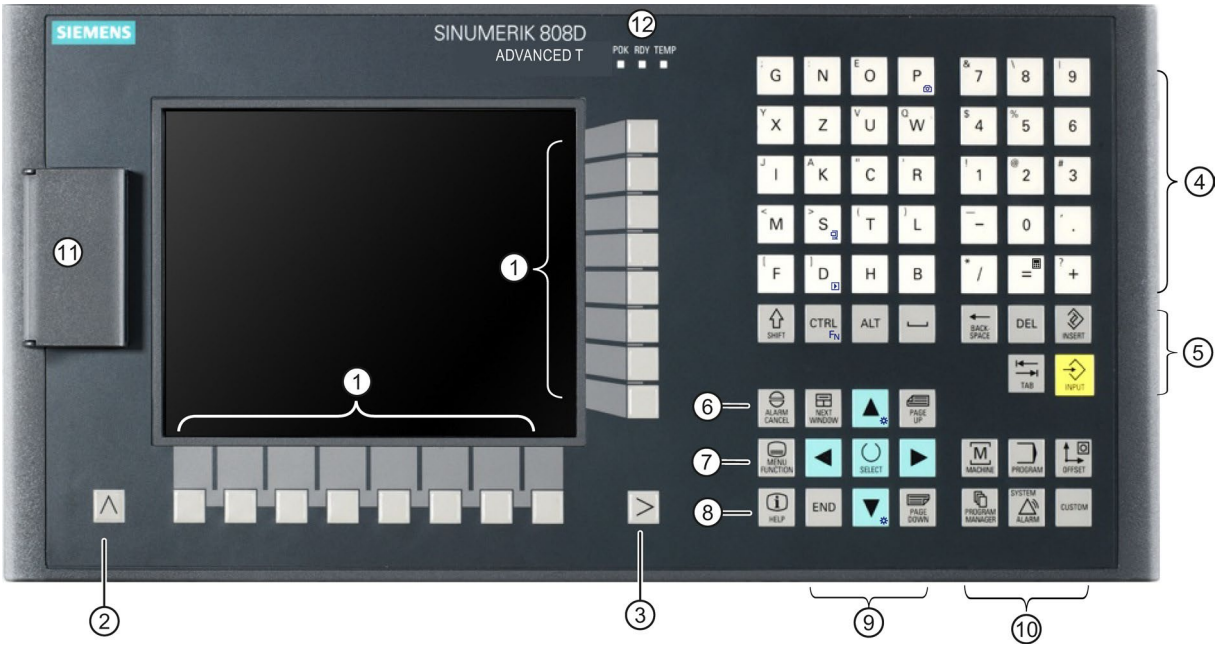
15. LIST OF G AND M CODES

Fanuc G- Code List (Lathe)			Fanuc M- Code List (Lathe)	
G code	**Description**	**Group**	**M code**	**Description**
G00	Rapid traverse	01	M00	Program stop
G01	Linear interpolation		M01	Optional program stop
G02	Circular interpolation CW		M02	End of program
G03	Circular interpolation CCW		M03	Spindle start forward CW
G04	Dwell	00	M04	Spindle start reverse CCW
G09	Exact stop		M05	Spindle stop
G10	Programmable data input		M08	Coolant on
			M09	Coolant off
G20	Input in inch	06	M29	Rigid tap mode
G21	Input in mm		M30	End of program reset
G22	Stored stroke check function on	09	M40	Spindle gear at middle
G23	Stored stroke check function off		M41	Low Gear Select
G27	Reference position return check	00	M42	High Gear Select
G28	Return to reference position		M68	Hydraulic chuck close
G32	Thread cutting	01	M69	Hydraulic chuck open
G40	Tool nose radius compensation cancel	07	M78	Tailstock advancing
G41	Tool nose radius compensation left		M79	Tailstock reversing
G42	Tool nose radius compensation right		M94	Mirror image cancel
G70	Finish machining cycle	00	M95	Mirror image of X axis
G71	Turning cycle		M98	Subprogram call
G72	Facing cycle		M99	End of subprogram
G73	Pattern repeating cycle			
G74	Peck drilling cycle			
G75	Grooving cycle			
G76	Threading cycle			
G92	Coordinate system setting or max. spindle speed setting	01		
G94	Feed Per Minute			
G95	Feed Per Revolution			
G96	Constant surface speed control	12		
G97	Constant spindle speed control cancel			

16. Step by Step Approach with CK50LX750 - SINUMERIC 808D

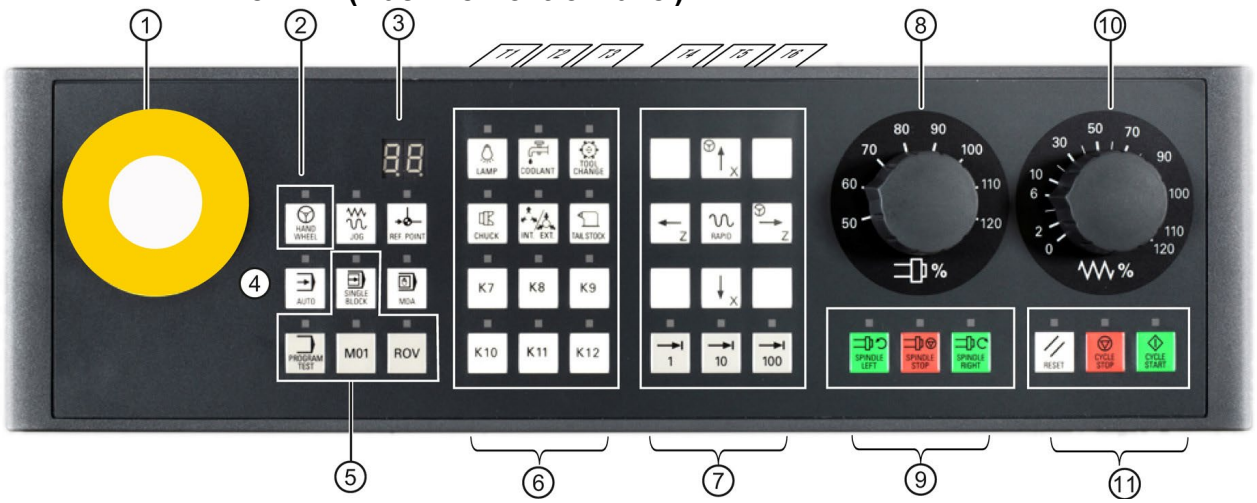
I. Turning Operator panel

1.1 The PPU (Panel Processing Unit)



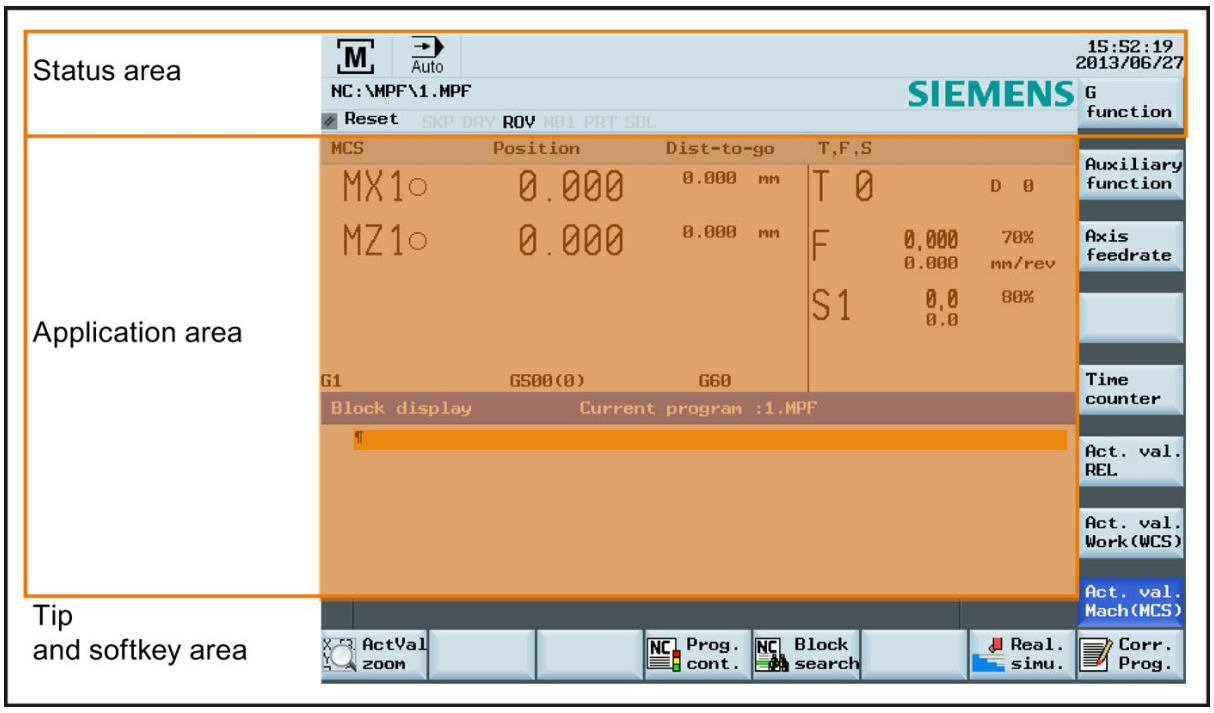
①	**Vertical and horizontal softkeys** Calls specific menu functions	⑦	**On-board wizard key** Provides step-by-step guides on basic commissioning and operation procedures
②	**Return key** Returns to the next higher-level menu	⑧	**Help key** Calls help information
③	**Menu extension key** Opens the next lower-level menu or navigate between the menus of the same level	⑨	**Cursor keys** *
④	**Alphabetic and numeric keys** *	⑩	**Operating area keys** *
⑤	**Control keys** *	⑪	**USB interface** *
⑥	**Alarm cancellation key** Cancels alarms and messages that are marked with this symbol	⑫	**Status LEDs** *

1.2 The MCP (Machine Control Panel)



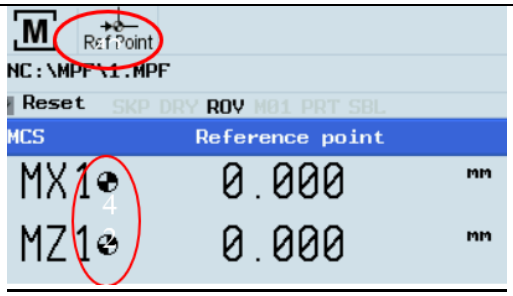
①	Reserved hole for emergency stop button	⑦	Axis traversing keys
②	Handwheel key Controls the axis movement with external handwheels	⑧	Spindle override switch (unavailable for the vertical MCP with reserved handwheel slot)
③	Tool number display Displays the current tool number	⑨	Spindle state keys
④	Operating mode keys	⑩	Feedrate override switch Traverses the selected axis at the specified feedrate override
⑤	Program control keys	⑪	Keys for program start, stop, and reset
⑥	User-defined keys *		

1.3 Screen Layout



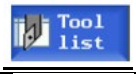
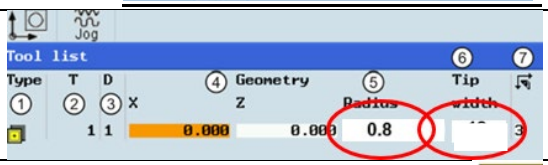
II. Turning on / Set Reference Point

1. Set to Reference Point

Operating Sequence	Action (Press button)
1.Main Power	Main switch turn-on
	Air supply-on
	Wait
2.Control Panel (MCL)	Release “EMERGENCY STOP” button
	Axis traversing keys **X** until ( ) appears
	Axis traversing keys **Z** until ( ) appears
	**Check:** The symbol ( ) appears “REF POINT” visible

III. Setting-up tools

1. Creating a new tool (e.g. T1)

Operating Sequence	Action (Press button)
	OFFSET
	TOOL LIST
	NEW TOOL>Turning Tool
	Enter Tool No : “1:” Edge position :“3”  OK
	Ener the Tool Radius”0.8” Ener the Tip Width “12” (Grooving Tool)
	INPUT (To confirm)

2. Standard tool list

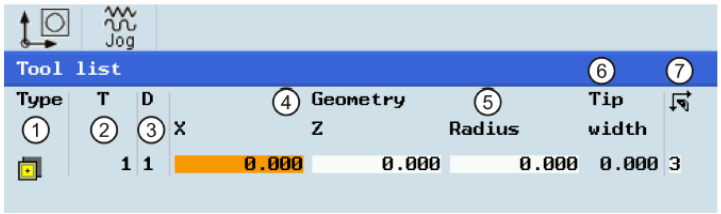
Standard Tool List

Skim work face.
Measure tool > Measure Z >
Z_0 = 0 > Set Length Z

Skim diameter

Measure diameter.
Measure tool > Measure X >
Ø :" " > Set Length X

Loc	Type	Tool Name	DP	Length X (offset)	Length Z (offset)	Radius	Working Direction	Holder angle	Insert Angle	Insert Length	Spindle rotation
1		ODL80.8	1	94.542	48.174	0.800	←	95.0	80	12.0	CCW
2											
3		ODL55.4	1	93.445	48.127	0.400	←	93.0	55	11.0	CCW
4											
5		OGL4.12	1	70.045	43.561	0.120		4.000		8.0	CCW
6		IDL55.4_16	1	-10.944	131.179	0.400	←	93.0	55	9.0	CCW
7		OTR2.12	1	63.414	48.342	0.120				16.0	CW
8		IDL80.8_16	1	-10.902	130.225	0.800	←	95.0	80	12.0	CCW
9											
10		FD20_3	1	0.00	166.554	20.00					CCW
11											
12											



① Tool type

② Tool number

③ Cutting edge number

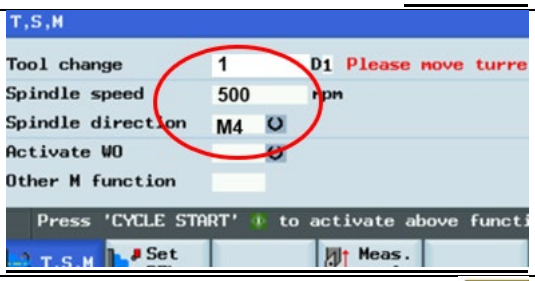
④ Tool length in the X and Z axes

⑤ Tool radius

⑥ Tip width of the cutting edge, which is only active for the grooving tool

⑦ Cutting edge direction

3. Activating the Tool (T1), Spindle Speed & Rotation (CCW)

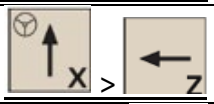
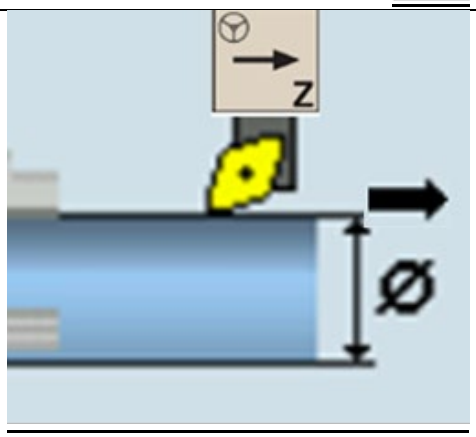
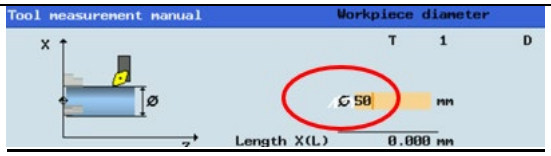
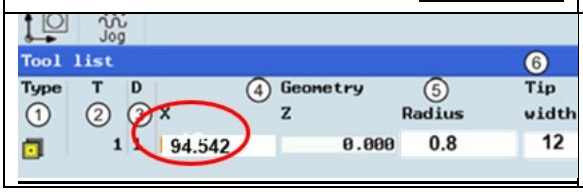
Operating Sequence	Action (Press button)
	MACHINE
	JOG
	TSM
	Enter Tool No : “1:” Spindle speed :“500” Spindle direction :”M4” CCW
	INPUT
	CYCLE START
	CYCLE STOP (To stop)

4. Assigning the handwheel

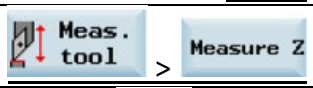
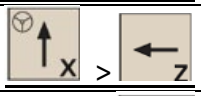
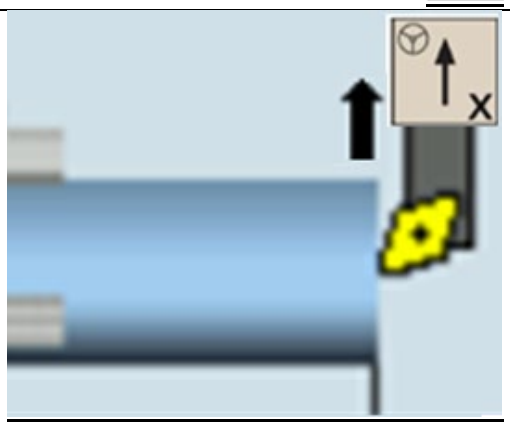
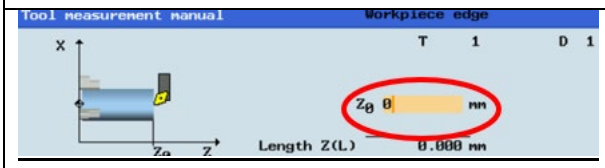
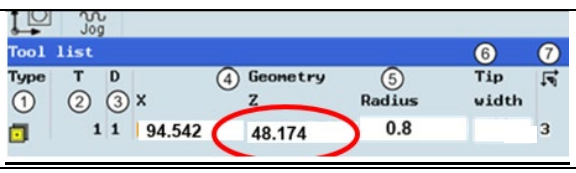
Operating Sequence	Action (Press button)
	MACHINE
	HANDWHEEL Control mode
	Use Handwheel for X travel
	Use Handwheel for Z travel

5. Measuring the Tool (manually)

A) Measuring the tool in the X direction

Operating Sequence	Action (Press button)
	Activating the tool (T1) ,500rpm & CCW
	MACHINE
	JOG
	MEASURE TO > Measure X
	Move the tool near to workpiece in X & Z directions
	HANDWHEEL Control mode
	Use handwheel to select a suitable override feedrate to make a skim cut on Ø.
	 increment in 0.001mm
	 increment in 0.010mm
	 increment in 0.100mm
	Move Z direction away from workpiece. 
	Stop spindle. Measure the machined workpiece diameter. Enter the workpiece diameter in the Ø field. (Example: “50”)
	INPUT
	Set length X
	Off set>TOOL LIST
	**Check tool data:** The length X value appears (Example. 94.542)

B) Measuring the tool in the Z direction

Operating Sequence	Action (Press button)
	MACHINE
	JOG
	MEASURE TOOL > Measure Z
	Move the tool near to workpiece in X & Z directions
	HANDWHEEL Control mode
	Use handwheel to select a suitable override feedrate to skim cut end face.
	 increment in 0.001mm
	 increment in 0.010mm
	 increment in 0.100mm
	Move X direction up away from workpiece.  >  Stop spindle
	Stop the spindle. Enter the distance between the tool tip and the workpiece face in the Z0 field. (Example: “ ”)
	INPUT
	Set Length Z
	Off set >TOOL LIST
	**Check tool data:** The length Z value appears (Example. 48.174)

6. Measuring the Tool (tool eye)

1. Select & mount tool the tool to be measured

TSM > Tool change > " eg .No 1" > 

> 

Cycle start

Open door > Mount tool > ....

2. Measure the tool length offset

2.1. 

Open door > Set Tool Eye >Close door

2.2. Position the tool in X direction (Fig 1)

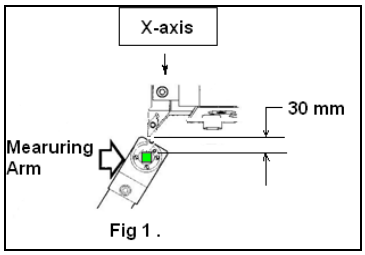


Fig 1 .

Measure tool tool >Automatic> X > 

Cycle start ....[Length X=_____]

1.3. Position the tool in Z direction (Fig 2)

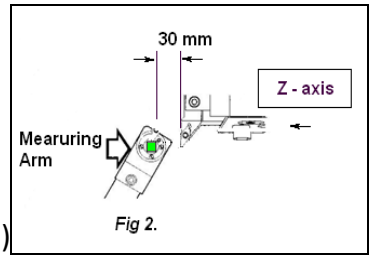


Fig 2.

Meas. tool >Automatic> Z > 

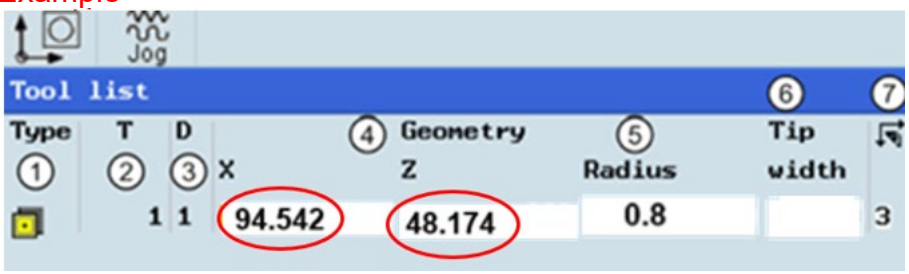
Cycle start ....[Length Z=_____]

1.4. 

Open door >Return Tool Eye > Close Door

>Select Tool > Check Length X & Z values.

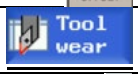
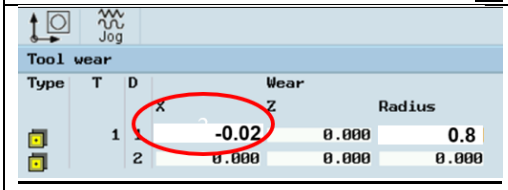
Example



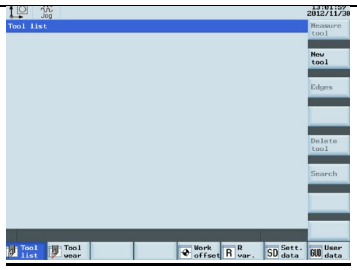
7. Verifying the tool offset result in “MDA” mode

Operating Sequence	Action (Press button)
	MACHINE
	MDA mode
	DELETE FILE
G500 T1 D1 G00 X0 Z5 or 	Enter the test program Example: G500 T1 D1 G00 X0 Z5 Or load an existing part program .
	The ROV function activates the federate override switch under the G-- function
	Increase the federate override gradually to avoid accident.

8. Modifying the tool wear data

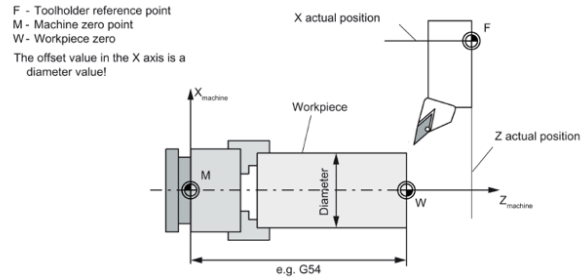
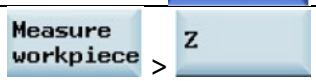
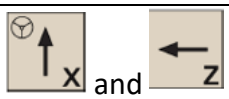
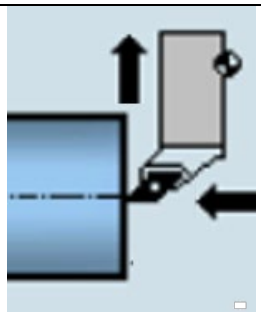
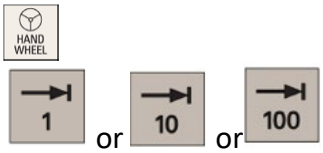
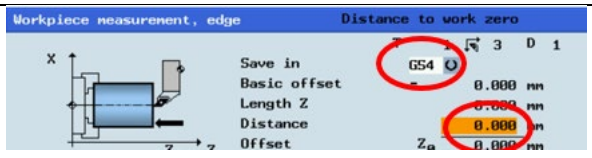
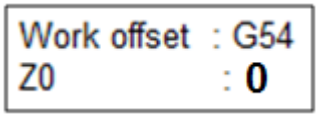
Operating Sequence	Action (Press button)
	OFFSET
	TOOL WEAR
	Use cursor key to select the tools
 INPUT	Enter the tool length wear parameter of axis X and axis Z. Positive value: The tool move away from workpiece Negative value: The tool move closer to workpiece (Example: **-0.02mm**)

9. Operating area overview

Operating Sequence	Action (Press button)
	OFFSET (Operating area overview)
	When working with the CNC ,you need to set up the machine and the tools,etc as follows: • Create the tools and the cutting edge. • Enter/modify the tool and the work offsets. • Enter the setting data.

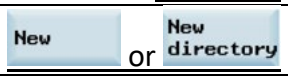
IV Setting-up The Workpiece

1. Determining the work offset Z axis

Operating Sequence	Action (Press button)
CK50LX750 slant bed CNC Turning SINUMERIK 808D ADVANCED Fanuc - G259  Work Offset Setting Skim work face. Work offset >Measure workpiece> Work offset: G54 Z:0 > Set Work offset	Machine zero, M (Machine Coordinate System) Workpiece zero ,W (Work Coordinate System) Tool holder reference point ,F
	Activating the Tool,Spindle Speed & Rotation
	OFFSET
	JOG
	WORK OFFSET
	MEASURE WORKPIECE > Z (softkey)
	Traverse the tool X and Z directions near the workpiece
	HANDWHEEL Use handwheel to perform facing  Move up tool up (X axis = Ø100)  Stop spindle.
	Select the offset plane “G54”
 	 Enter Z0:”0”

V Part Programming

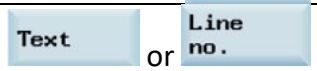
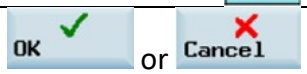
1. Creating a part program

Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	NC
	NEW > NEW DIRECTORY
	Name the new program (Example: “HARON”)
	OK
	Select the new directory with the cursor keys.
	INPUT to open the directory
	NEW FILE
	Name the new file (Example: “EX 1”)
	OK

2. Renumbering blocks

Operating Sequence	Action (Press button)
	RENUMBERING

3. Searching for blocks

Operating Sequence	Action (Press button)
	SEARCH
	TEXT or LINE NUMBER
	SELECT
	OK or CANCEL the search

4. Copying and pasting blocks

Operating Sequence	Action (Press button)
	MARK ON
	Use cursor keys to select program block
	COPY or DEL
	Place a cursor on the desired insertion point.

5. Searching for program

Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	NC
 or  or 	Select the storage medium
	SEARCH
	SELECT
 or 	OK or CANCEL

6. Copying and pasting program

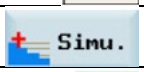
Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	Select the desired program
	COPY or DEL
	Select the target directory and paste.

7. Renaming program

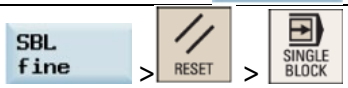
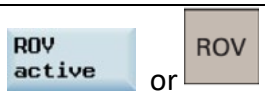
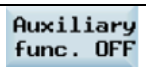
Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	Select the desired program
	To access more options
	RENAME
 or 	OK or CANCEL

VI Machining

1. Performing the simulation

Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	Select the part program for simulation
	INPUT
	AUTO mode
	SIMULATION
	CYCLE START

2. Program control

Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	AUTO mode
	PROGRAM CONTROL
Program control options	
 > PROGRAM TEST	
	First remove workpiece from the machine
 > M01	
	
 > RESET > SINGLE BLOCK	
 or ROV	The federate is overide
	The AFL(auxiliary function lock) function disables the spindle and suppresses all auxiliary functions

3. Program test

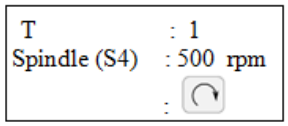
Operating Sequence	Action (Press button)
	First remove workpiece from the machine PROGRAM MANAGER
	AUTO mode
	PROGRAM CONTROL
	DRY RUN FEEDRATE
	CLOSE DOOR or Close door manually
	Feedrate override is 0%
	CYCLE START
	Feedrate override turn slowly
	RESET Stop program test

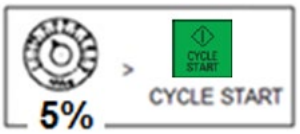
4. Starting and stopping/interrupting a part program

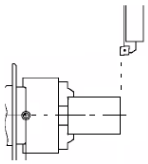
Operating Sequence	Action (Press button)
	PROGRAM MANAGER
	Select the program
	EXECUTE Change to AUTO mode
	[Select this option for more control]
	CLOSE DOOR or Close door manually
	Feedrate override is 0%
	CYCLE START Start machining
	Feedrate override turn slowly
To stop machining	
	RESET The program aborted
	CYCLE STOP The execution of program stopped/interrupted

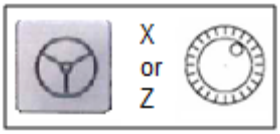
5. Machining (1st side- A)

➡ Clamp workpiece

➡ TSM > 

> 

➡  Move the tool, T1 near work, 



Facing (Use HANDWHEEL) .

 Moves cutter up (X-axis =Ø100mm) **..... [Do not move Z-axis]**

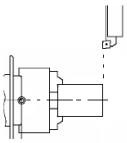
 Switch 'OFF' spindle.

Work offset> Measure workpiece > 

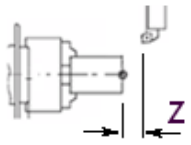
Enter Z0 = "**0**" > Set work offset

➡

➡ Close door >  Move tool, T1 away from work.
(Open Door) Use a steel ruler to check that cutting edge in-line with workpiece edge

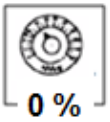
➡ 

➡ Call the program>Simulation > Execute >  Single Block > Basic Block. >

> 

> 

> Coolant 'ON'

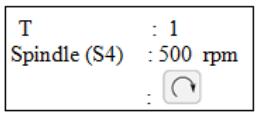
➡ Machining complete >  MANUAL/Jog >  > 

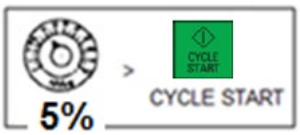
> Coolant 'OFF'

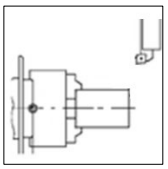
➡ Remove the machined workpiece.

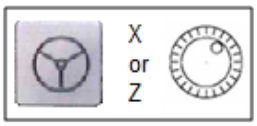
6. Machining (2nd side- B)

➡ Clamp Workpiece

➡ TSM > 

> 

➡  Move the tool, T1 near work, 

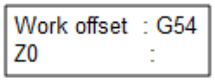


Facing (Use HANDWHEEL) .

 Moves cutter up (X-axis =Ø100mm) ..... **[Do not move Z-axis.]**

 Switch 'OFF' spindle.

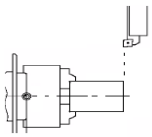
Measure the length of workpiece after facing … [Eg. 51.8 – 50(*Length shown in drawing*) = *1.8*]

Work offset> Measure workpiece > 

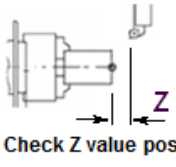
Enter Z0 value = **"1.8"** >

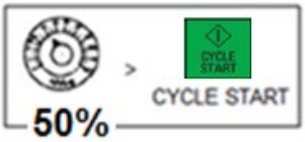
➡ Set work offset

➡ Close door >  Move tool, T1 away from workpiece.

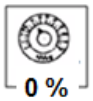
➡ Use a Steel Ruler to estimate cutting edge in-line with workpiece zero 

➡ Call the program>Simulation > Execute >  Single Block >Basic Block. >

> 

> 

> Coolant 'ON'

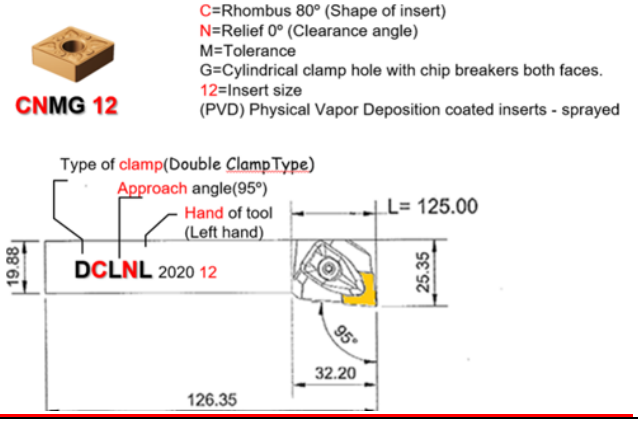
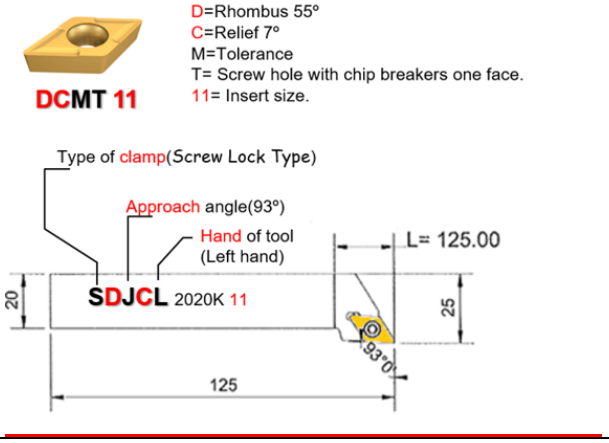
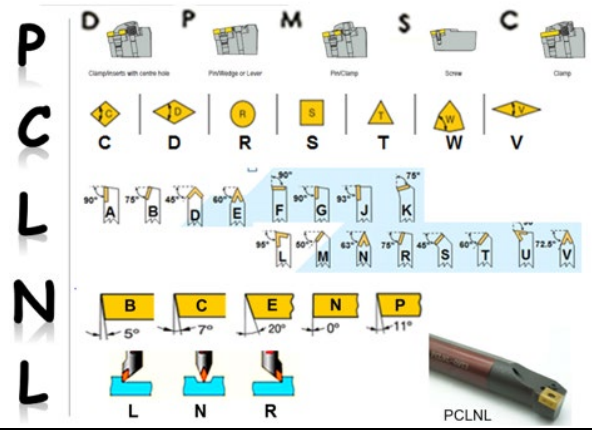
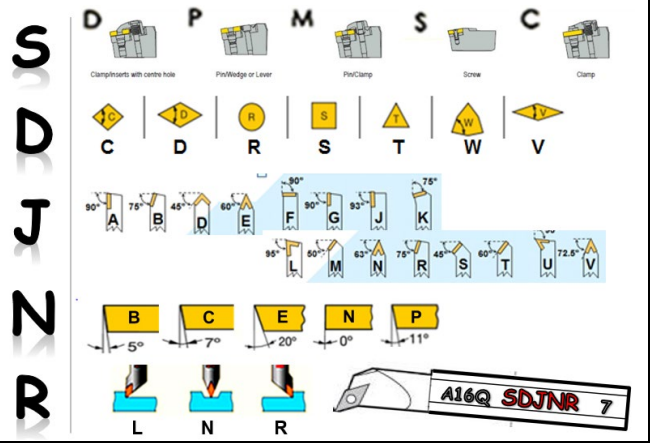
➡ Machining complete >  MANUAL/Jog >  > 

> Coolant 'OFF'

➡ Remove the machined workpiece.

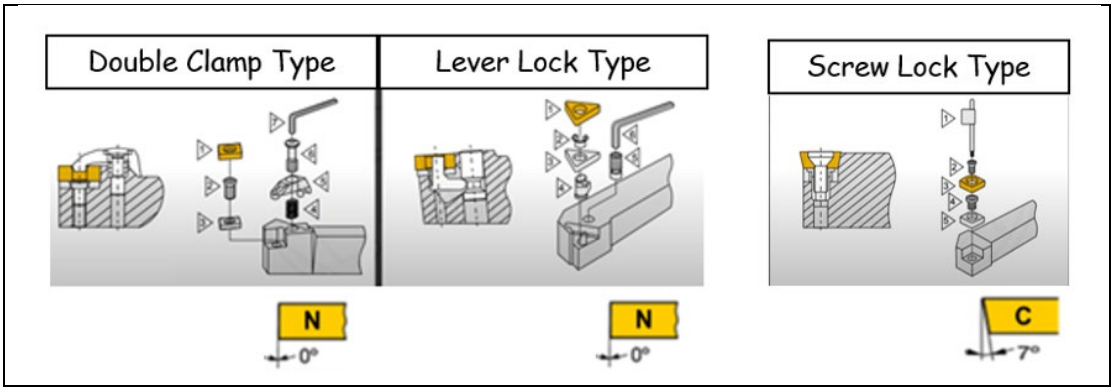
VII Miscellaneous

i. Tool Holder & Inset

Left Hand OD Rough Holder-DCLNL (ODL80.8)	Left Hand OD Finish-SDJCL (ODL55.4)
	
Left Hand ID Rough-PCLNL (IDL80.8_16)	Left Hand ID Finish-SDJNR (IDL55.4_16)
	

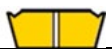
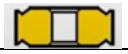
ii. Clamping System

Double clamp and lever lock types of clamping are mainly used for insert with zero relief degree.

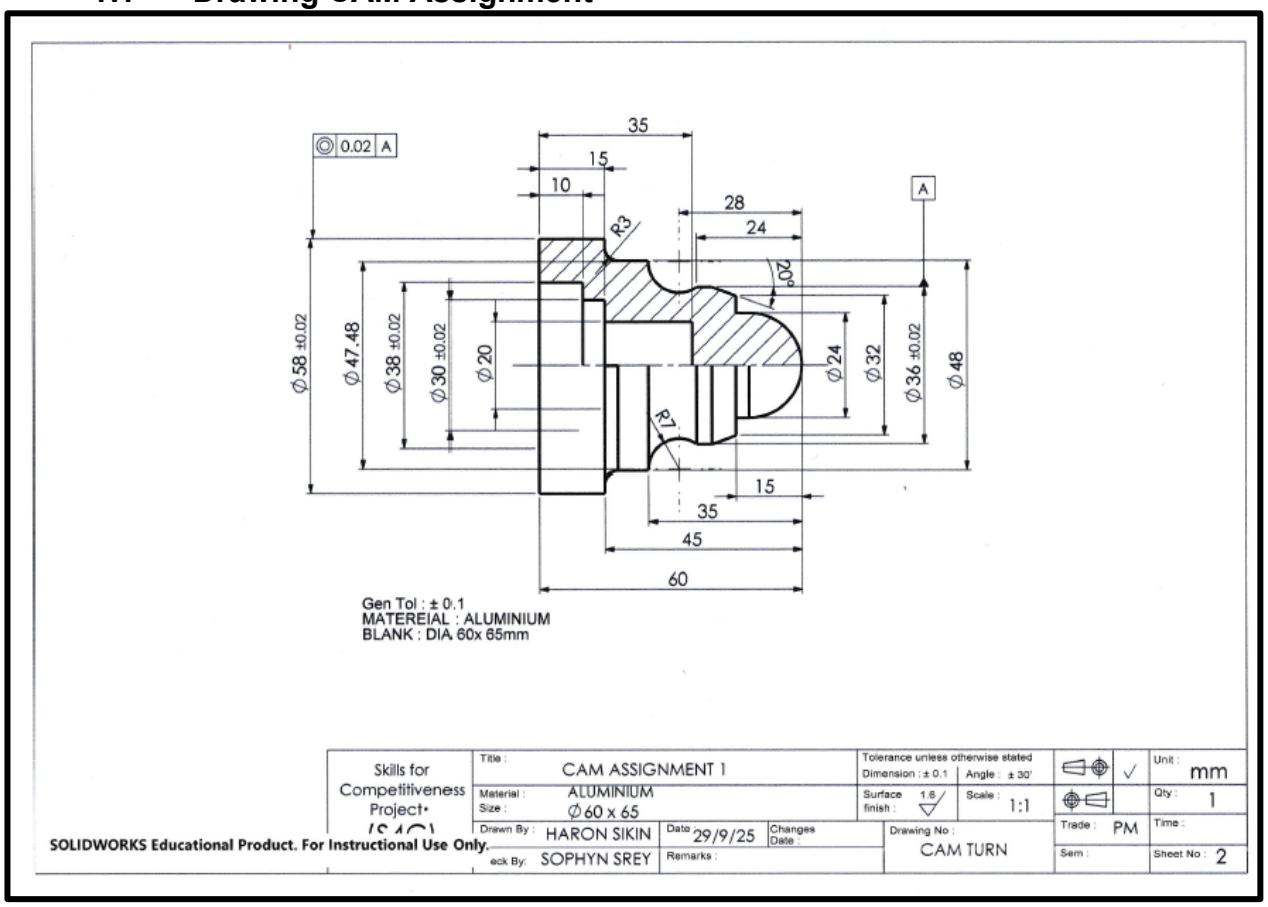


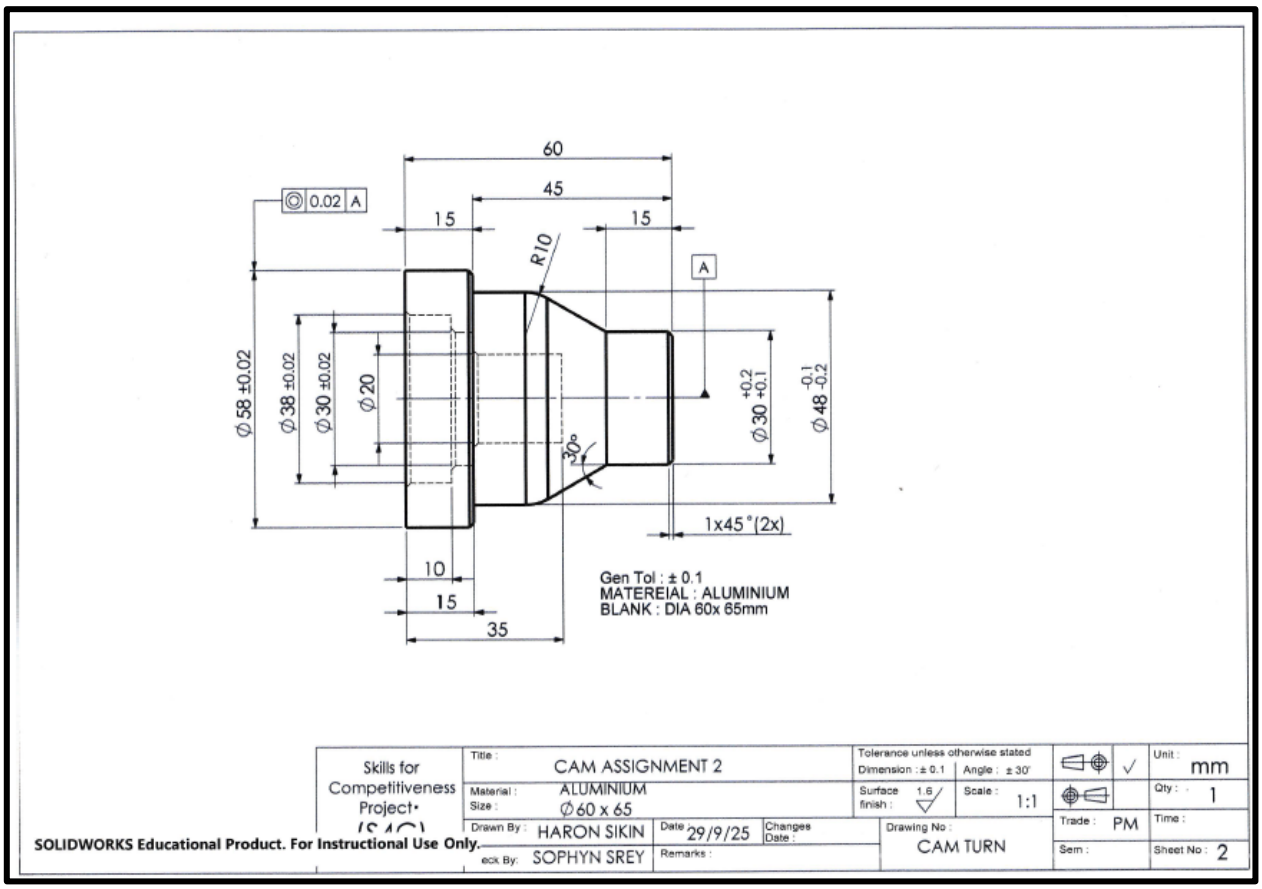
Relief angles	

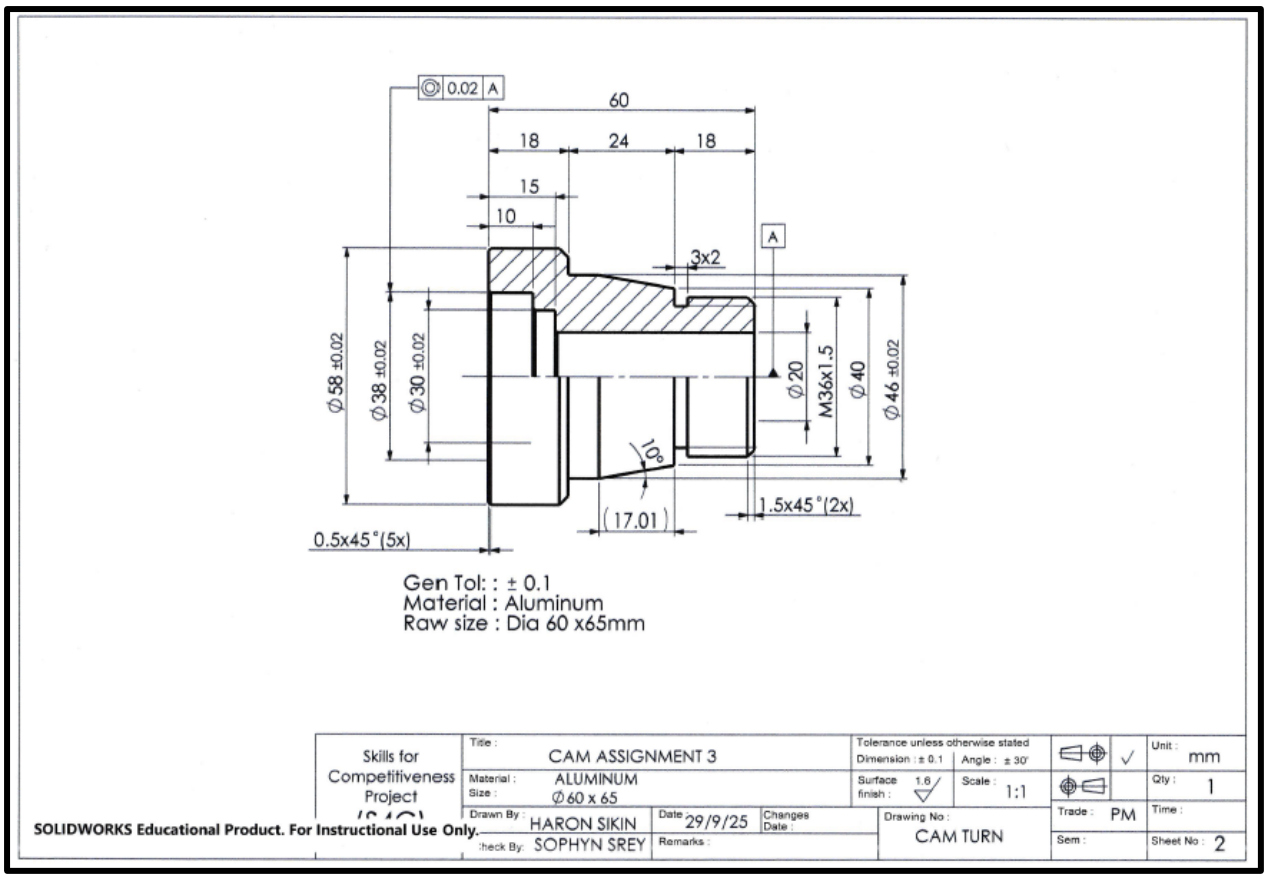
iii. Insert profile

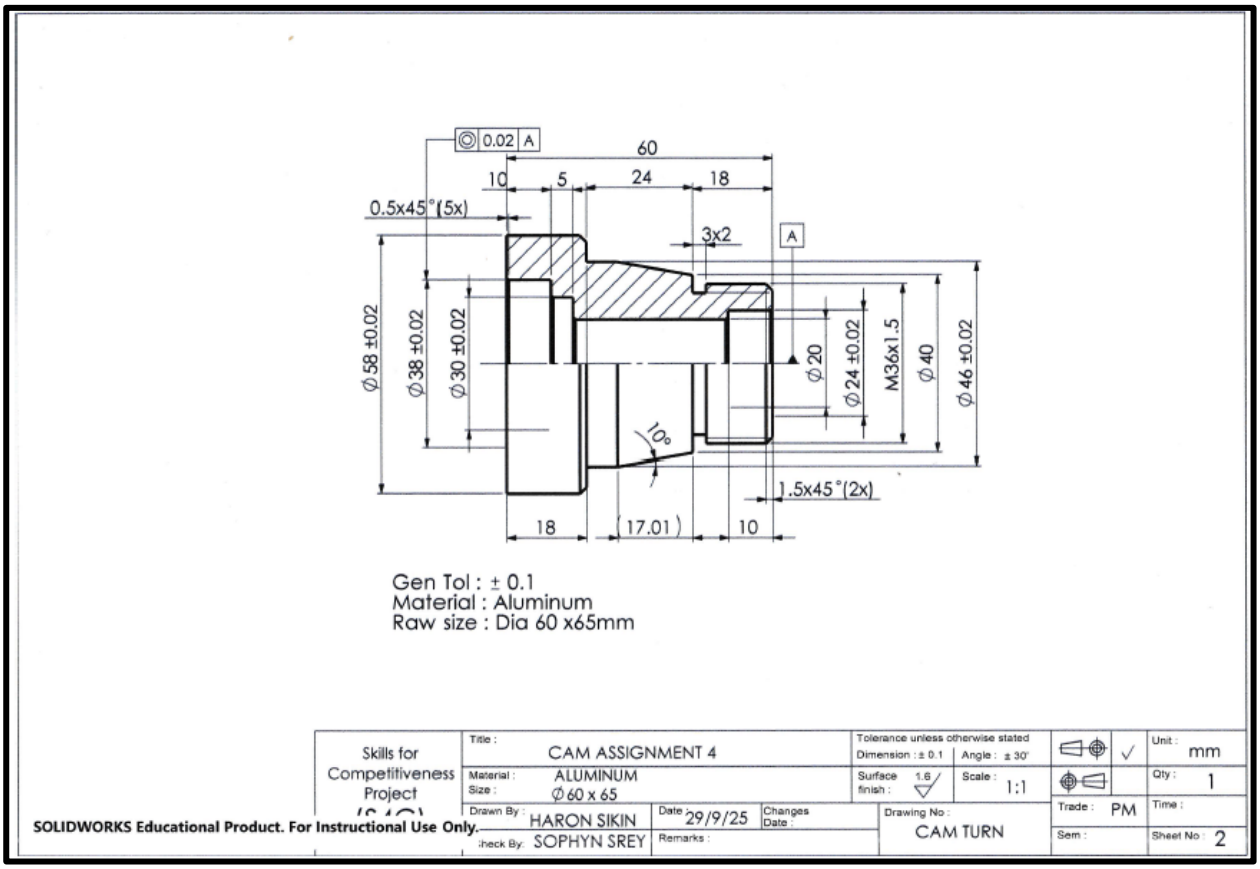
Letter	Clamp	Chip breaker	Profile
N	No Clamping Hole	X	
R		1 face	
A	Cylindrical Clamp Hole	X	
M		1 face	
G		Both Faces	
W	Screw Hole	X	
T		1 face	
U		Both Faces	

iv. Drawing CAM Assignment



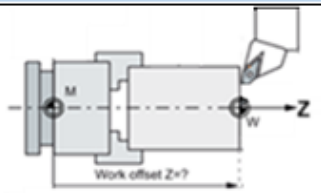



0.02 A
60
18
24
18
15
10
A
3x2
Ø58 ±0.02
Ø38 ±0.02
Ø30 ±0.02
Ø20
M36x1.5
Ø40
Ø46 ±0.02
10°
(17.01)
1.5x45°(2x)
0.5x45°(5x)
Gen Tol: : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm
Skills for Competitiveness Project
Title : CAM ASSIGNMENT 3
Material : ALUMINUM
Size : Ø60 x 65
Tolerance unless otherwise stated
Dimension : ± 0.1
Angle : ± 30'
Surface finish : 1.6
Scale : 1:1
Unit : mm
Qty : 1
Drawn By : HARON SIKIN
Date 29/9/25
Changes Date :
Check By: SOPHYN SREY
Remarks :
Drawing No : CAM TURN
Trade : PM
Time :
Sem :
Sheet No : 2
SOLIDWORKS Educational Product. For Instructional Use Only.


0.02 A
60
10
5
24
18
0.5x45°(5x)
3x2
A
Ø58 ±0.02
Ø38 ±0.02
Ø30 ±0.02
Ø20
Ø24 ±0.02
M36x1.5
Ø40
Ø46 ±0.02
10°
1.5x45°(2x)
18
(17.01)
10
Gen Tol : ± 0.1
Material : Aluminum
Raw size : Dia 60 x65mm
Skills for Competitiveness Project
Title : CAM ASSIGNMENT 4
Material : ALUMINUM
Size : Ø60 x 65
Tolerance unless otherwise stated
Dimension : ± 0.1
Angle : ± 30'
Surface finish : 1.6
Scale : 1:1
Unit : mm
Qty : 1
Drawn By : HARON SIKIN
Date 29/9/25
Changes Date :
Check By: SOPHYN SREY
Remarks :
Drawing No : CAM TURN
Trade : PM
Time :
Sem :
Sheet No : 2
SOLIDWORKS Educational Product. For Instructional Use Only.

Tool list (CK50LX750 - SINUMERIK 808D)

Turret No	1	2	3	4	5	6	7	8	9	10	11	12
Hand of Tool	 80°/NR0.8		 55°/NR0.4 (35°/NR0.4)		 Width:3 R 0.12	 Bar Ø16 NR0.4	 Pitch:1.5	 Bar Ø16 NR0.8		 Ø20 x3L NR0.4		
	LH OD Rough [ODL80.8]		LH OD Finish [ODL55.]		LH OD Groove [OGL3.12]	LH ID Finish [IDL55.4 16]	RH OD Thread [OTR2.12]	LH ID Rough [IDL80.8 16]		Face Drill (ID) [FD20 3]		Twist Drill
Insert example	CNMG 12		DCMG 11 (VNMG)		MGMN 300	DNGP 0703	16ERM1.5 ISO K908	CNMG 0903		SPMG0 60204		HSS
Holder example	DCLNL 2020K 12		SDJCL 2020K11		MGEHL 2020-3	A16Q SDZNL 7	20x20mm	S16Q PCLN L 09		TDR 2200 25T2-06		
C/S (m/min)	150-250 (100)		250-350 (200)		120-180 (120)	200-300 (200)	80-120 (80)	200-300 (200)		60-100 (62.84)		30-40 (30)
Feed (mm/rev)	*0.25-0.35 (0.25)*		*0.12-0.15 (0.12)*		*0.08-0.15 (0.08)*	*0.1-0.15 (0.1)*	*0.25 (0.25)*	*0.25-0.35 (0.2)*		*0.1-0.15 (0.1)*		*0.05-0.1 (0.05)*
Holder angle	95°		93°			93°		95°				
Insert angle	80°		55°/(35°)			55°		80°				
Insert length	12		11/(16)		8	7		9		6		
Nose radius	0.8		0.4/(0.4)		0.12	0.4	0.12	0.8		0.4		
IC Diameter(D)	12.7		9.5		-	6.35	9.525	9.5		6		
Thickness(T)	4.76		4.76		3	3.18		3.1		2.38		
Spindle rotation	CCW		CCW		CCW	CCW	CW	CCW		CW		CW

Tool offset setting (All the tools)			Work offset setting
 **Skim work face. Measure tool > Measure Z > Z_0 = 0 > Set Length Z**	 **Skim diameter**	 **Measure diameter. Measure tool > Measure X > Ø :" " > Set Length X**	 **Skim work face. Work offset >Measure workpiece > Work offset : G54 Z : 0 > Set Work offset.**