יום עיון בנושא חדשנות טכנולוגית בעיבוד שבבי לשכת המהנדסים והאדריכלים 2019

רונן קומריאן,
עובד בקמ"ג בתחום הפיתוח, תכן הנדסי, ובעבר גם בעיבוד שבבי, משנת 1998 .
מלמד את נושא ה – GD&T במכללה להנדסה סמי שמעון, קמ"ג סאסא פלסן

השכלה:

תואר ראשון בהנדסת מכונות, מהמכללה להנדסה SCE באר שבע.
תואר שני בהנדסת אנרגיה, אוניברסיטת בן גוריון (סיום 2019).

קורסים:

1. **ASME** - "Geometric Dimensioning & Tolerancing Fundamentals + Advanced Applications with Stacks and Analysis "- Course (2010)
2. **ASME** – "Dimensioning and Tolerancing Principles for Gages and Fixtures" - Course (2012)

הסמכות:

1. **GDTP - Geometric D**imensioning & **T**olerancing **P**rofessional – **Senior Level** - 3355
2. **GDTP - Geometric D**imensioning & **T**olerancing **P**rofessional - **Technologist Level** – 0843

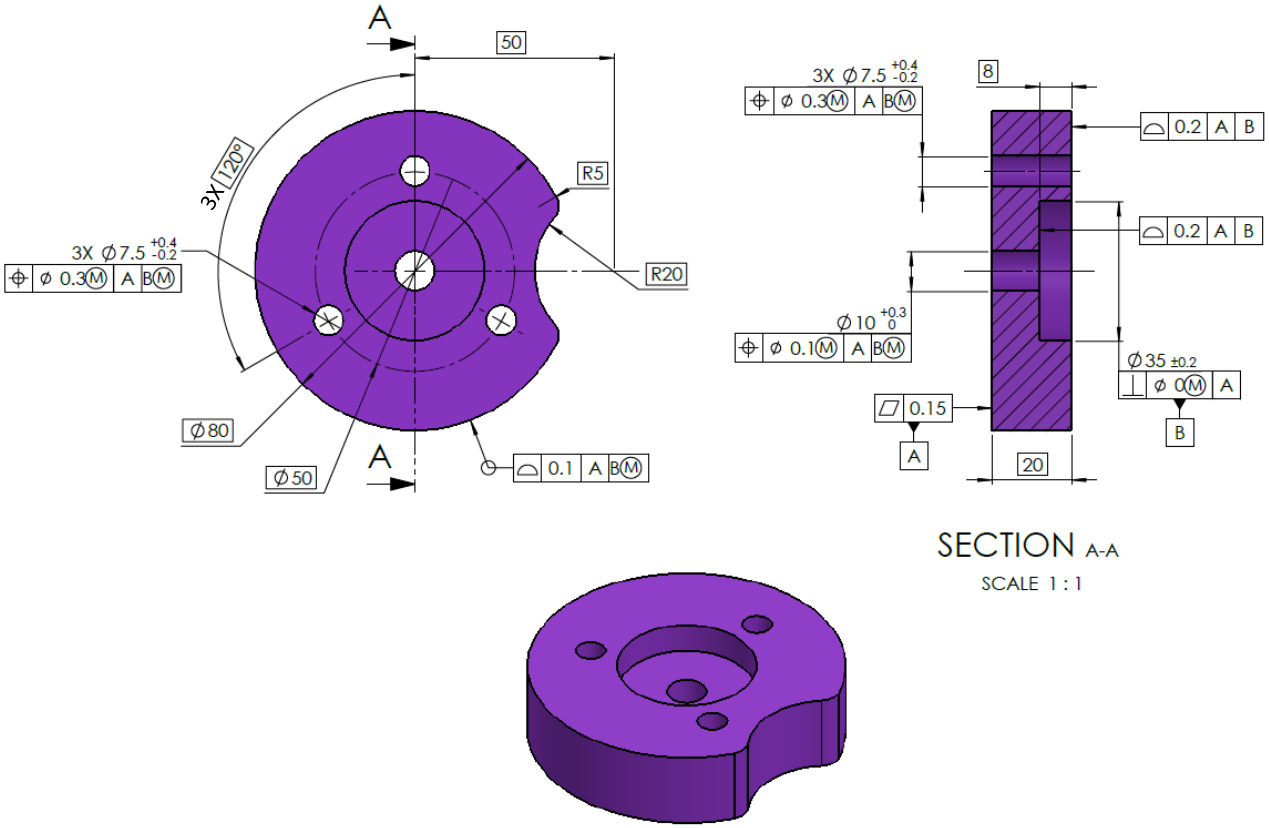
Table of Contents

Example 1

1) GD&T – international Drawing language

international Drawing language, Applied drawings to describe a mechanical parts.

- **Rules:** rule #1, rule #2…
- **Settings:** Assembly, Part, feature, datum, DRF…
- **Tools:** dimensions, shape, orientation, location…

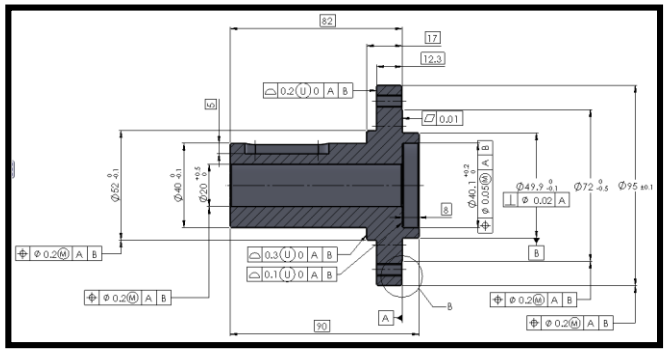


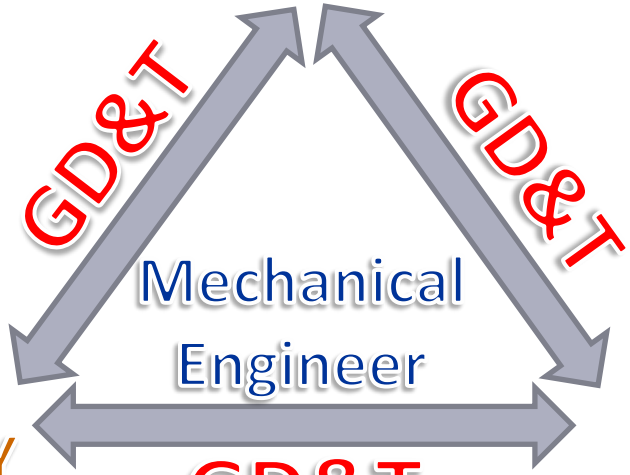
2) GD&T - precise language

Geometry dimensions and tolerances is a precise language.

DESIGN

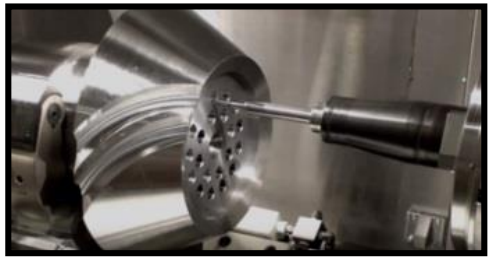
The information is define more clearly by the designer





MANUFACTURING

The part is better understood by the manufacturer



QUALITY

The Inspector measure the part as defined by the designer

What is GD&T

3) GD&T - language that provides uniformity

Uniform language

- Engineering
- Manufacturing
- Inspection
- Development
- Customer
- Assembly
- Materials
- Gage Planning
- Marketing

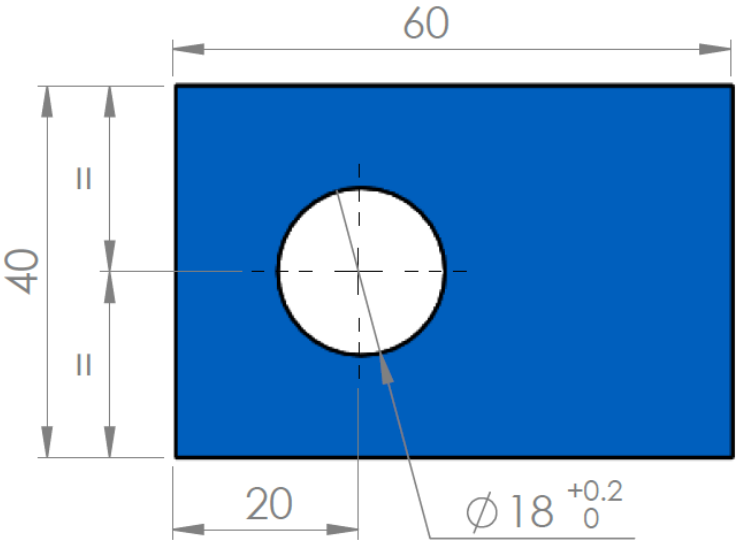
same drawings interpretation

The immediate effect is

- reduce disputes, guesswork and assumptions
- drawing clarity
- follow the manufacturing process

Size Does Not Control Interrelationship between Individual feature

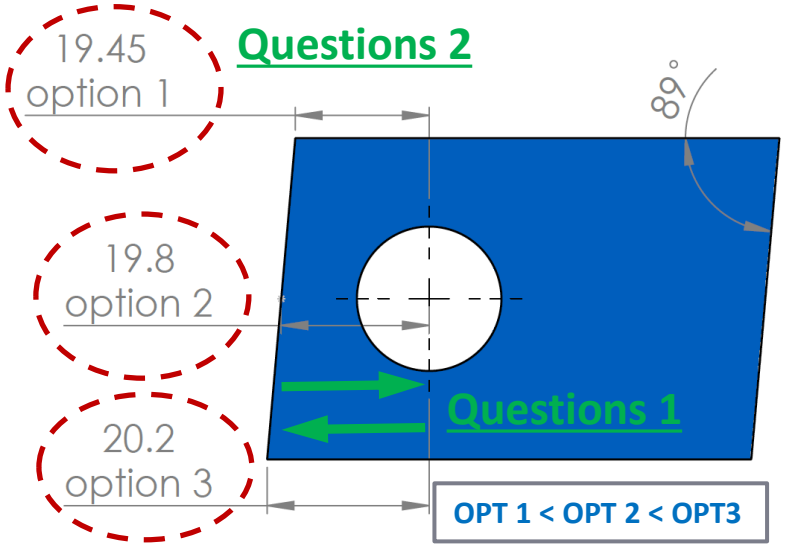
Coordinate Dimension Drawing



Notes:
General Tolerance ±0.2
General Angle Tolerance ±1°

Example 2

Means this

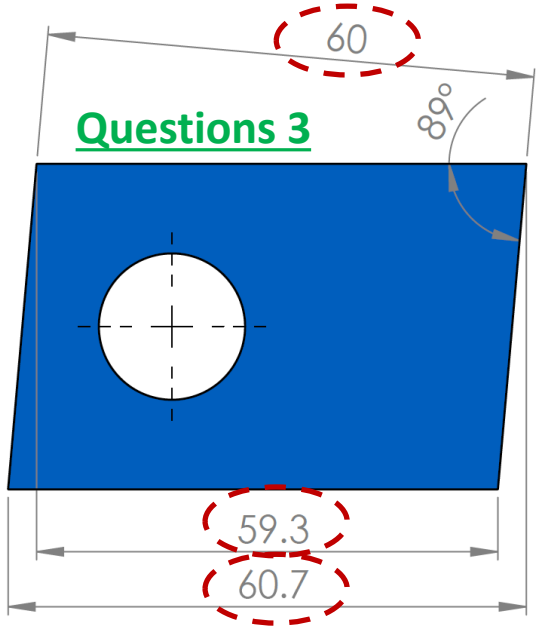


Questions

1. What is the correct measurement?
   a. to measure the hole relative to the surface?
   b. to measure surface relative to the hole?

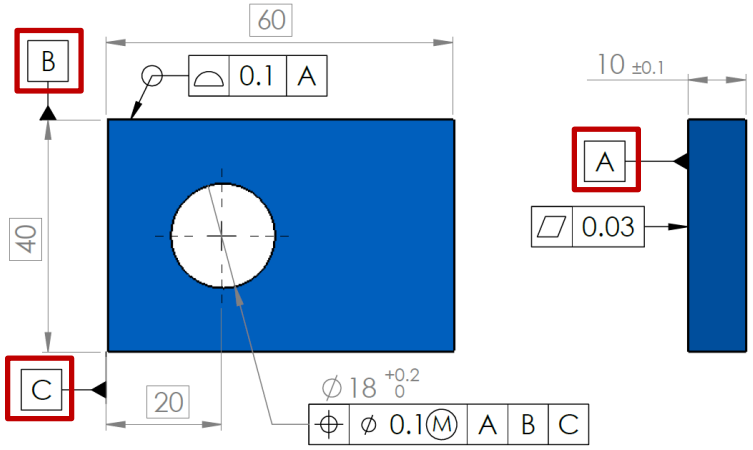
2 . Which option is the correct measure?
→ OPT 1?
→ OPT 2?
→ OPT 3?

3 . Which option is the correct measure for the external geometry?



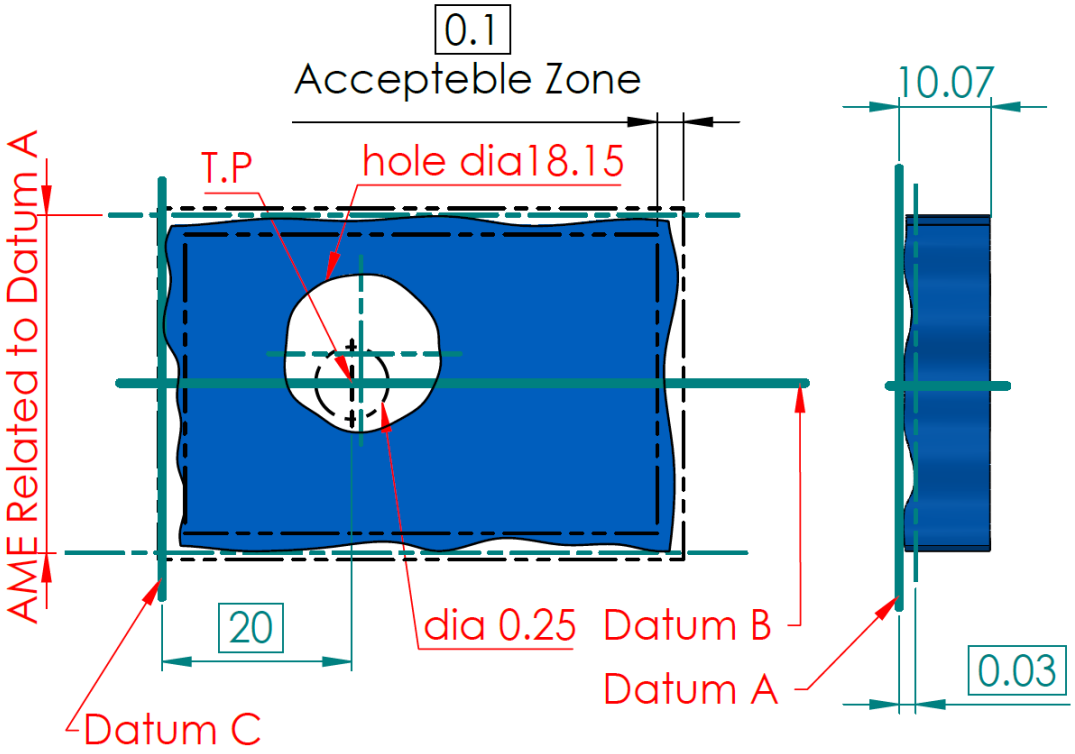
Size Does Not Control Interrelationship between Individual feature

GD&T Drawing



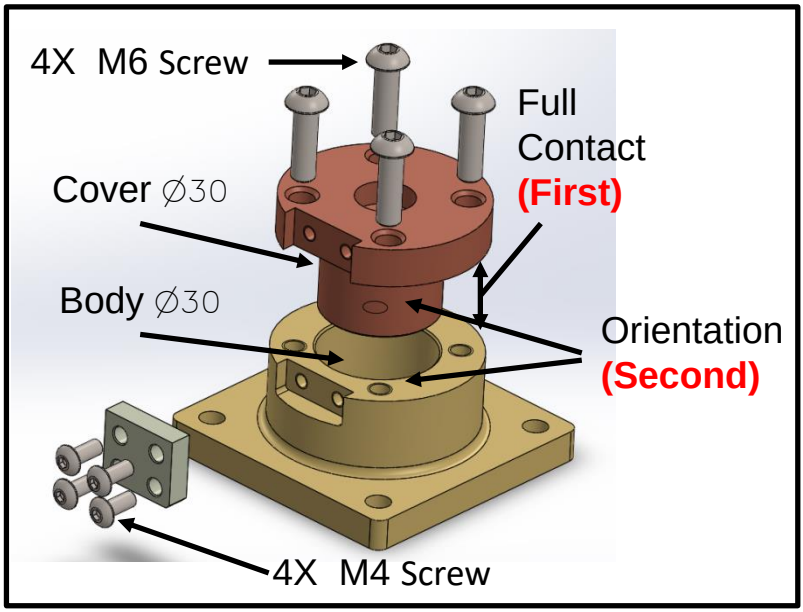
Example 2

Means this

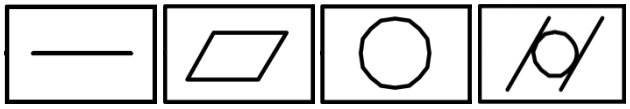


4) GD&T – Advantages

- Functional design
- Design by limits
- Maximum **match between components**.
- The ability to **calculate tolerance stuck** easily and systematically
- Process repeatability.
- Reduction of product disqualifications
- Shorter production duration.
- Reduce costs.



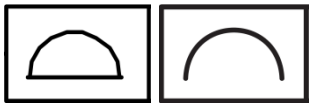
All Geometric Tolerance (2009)



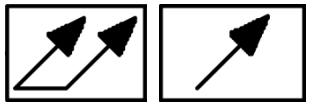
F O R M

ORIENTATION

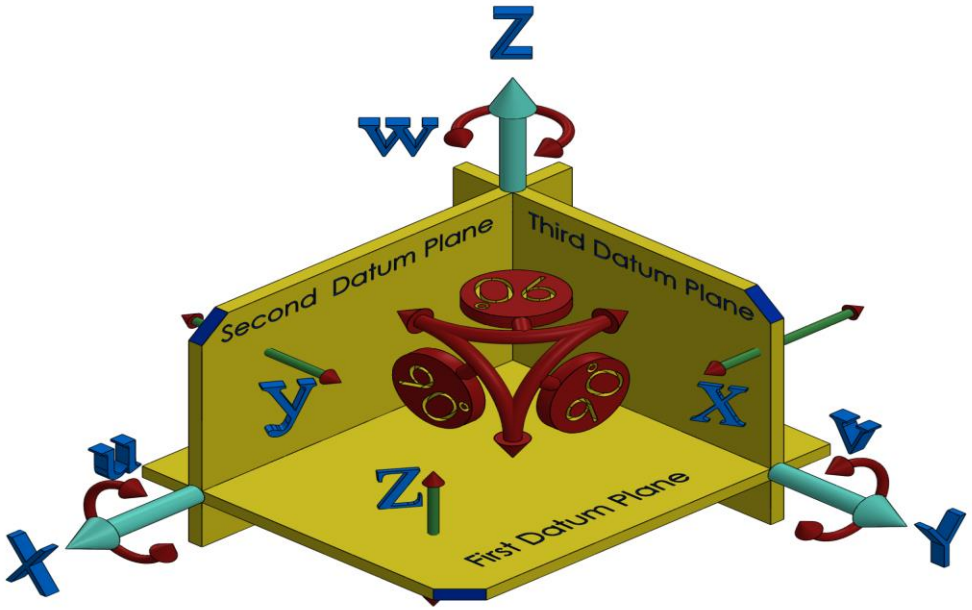
LOCATION



PROFILE

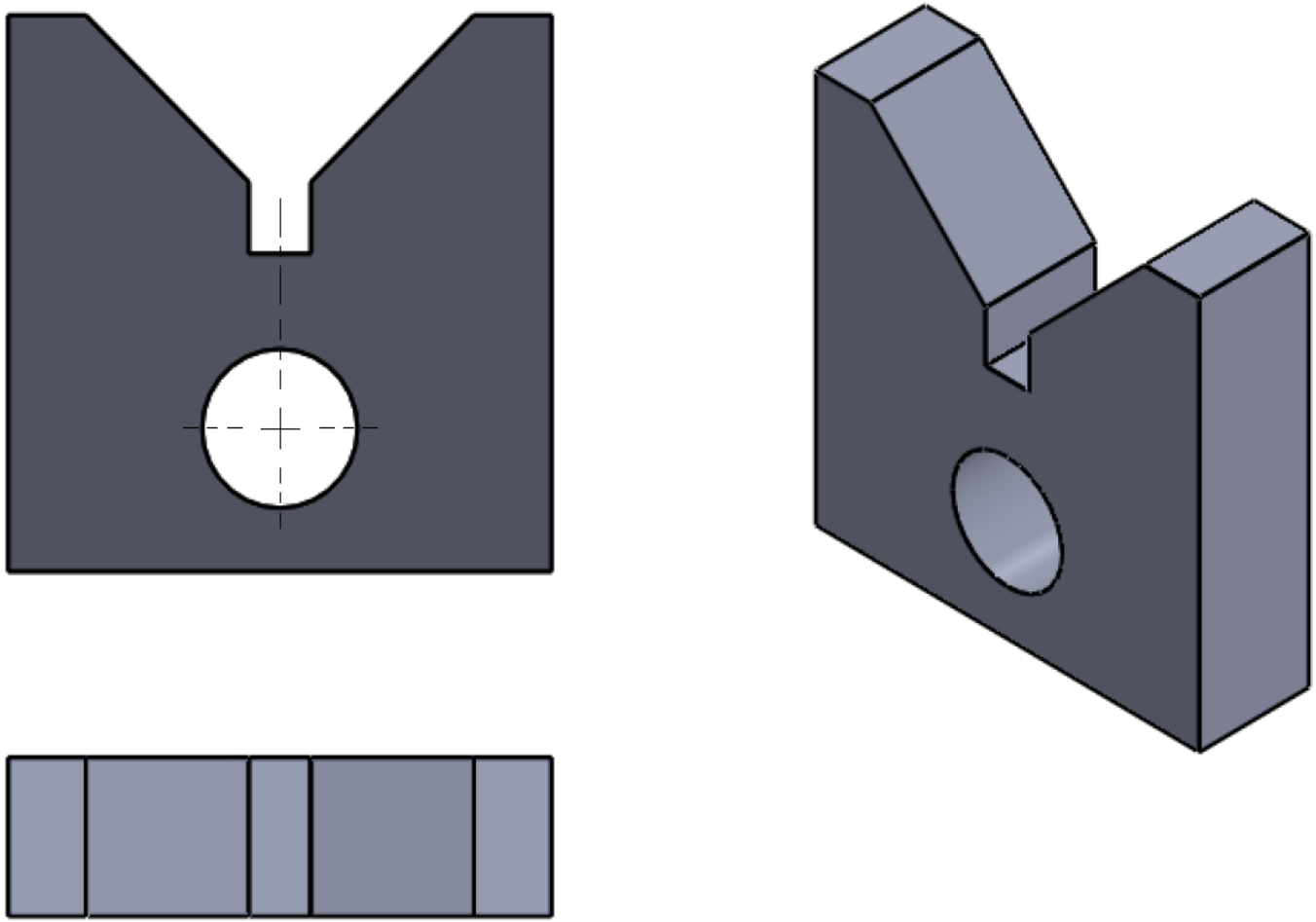


RUNOUT



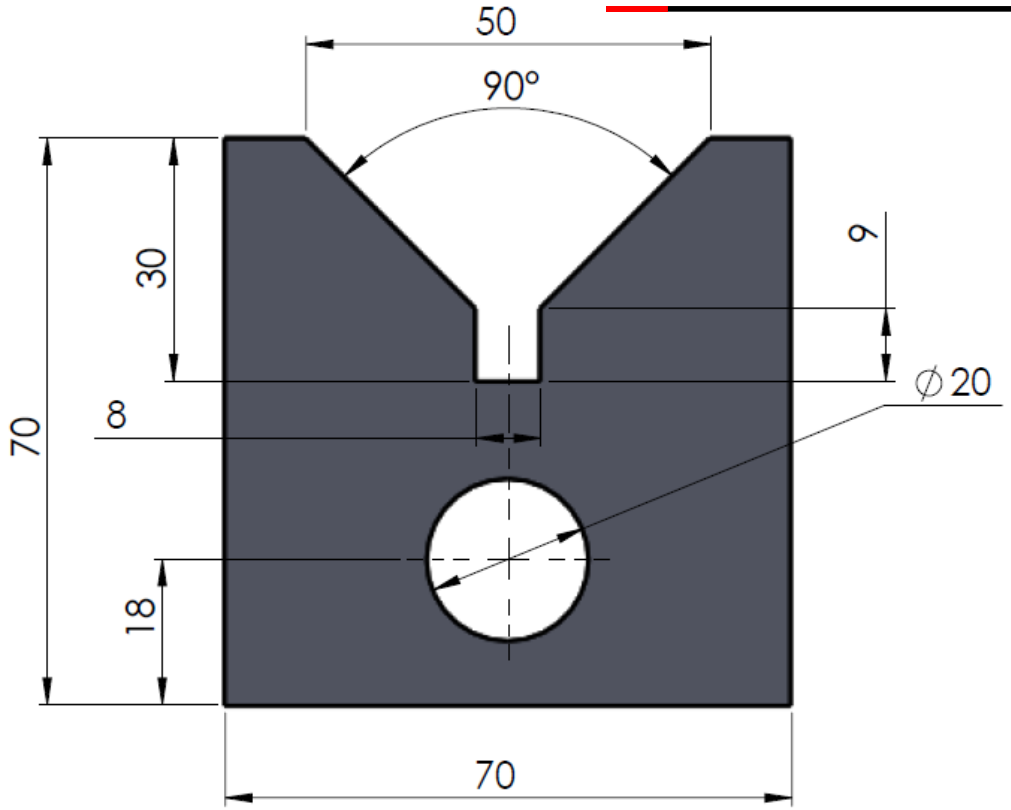
Example 3

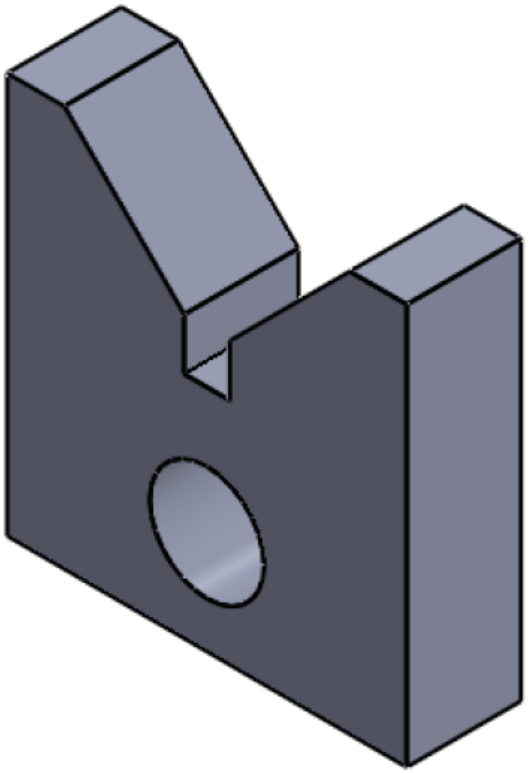
Geometric

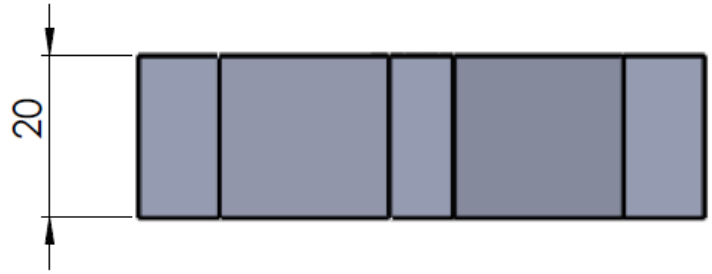


Example 3

Dimensioning



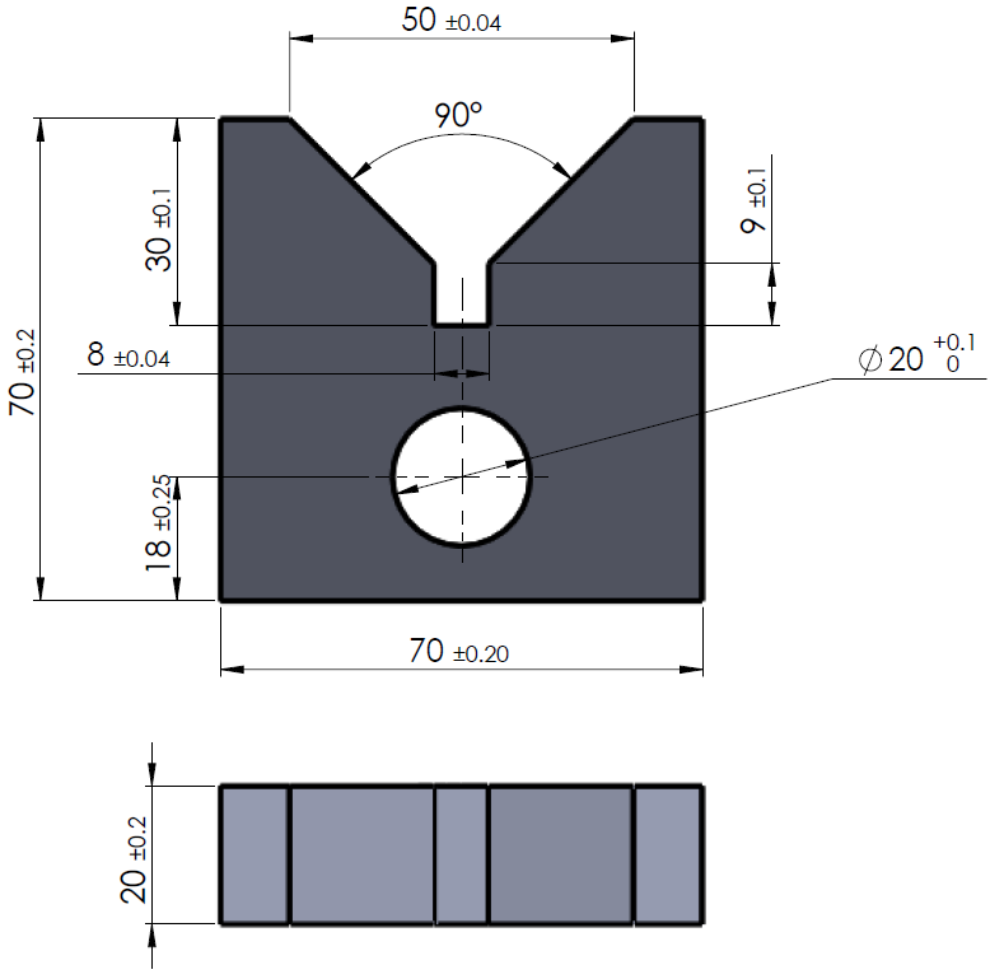


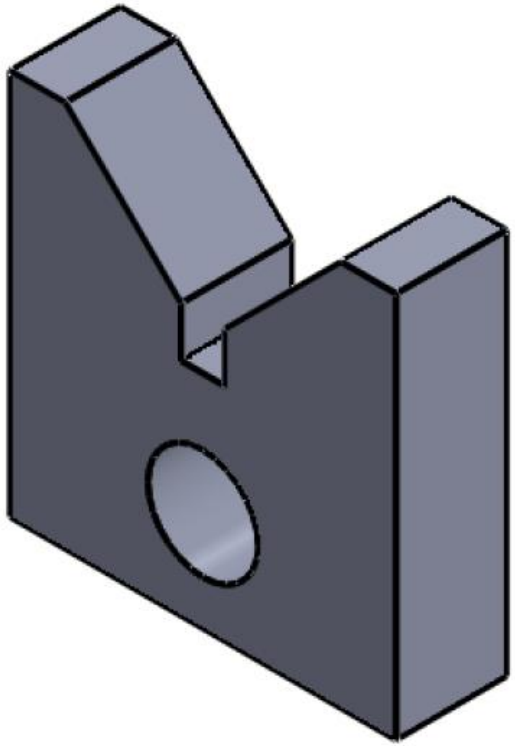


Ronen koⓂerian

Example 3

Tolerancing



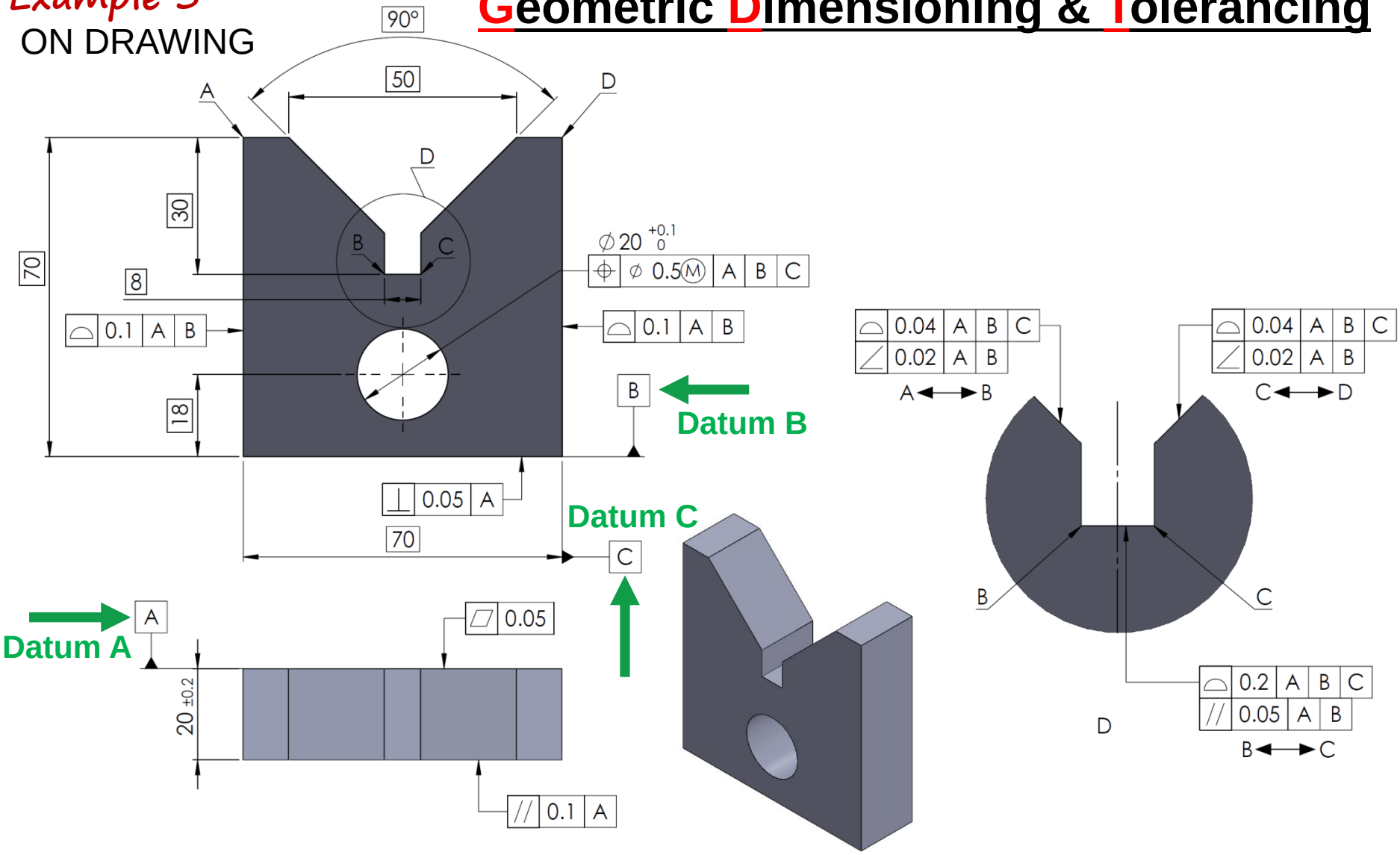


All dimensions are set! But is the geometric relationships between the various elements are defined? (perpendicularity, profile of surface , location, cylindrical, straightness,,,)

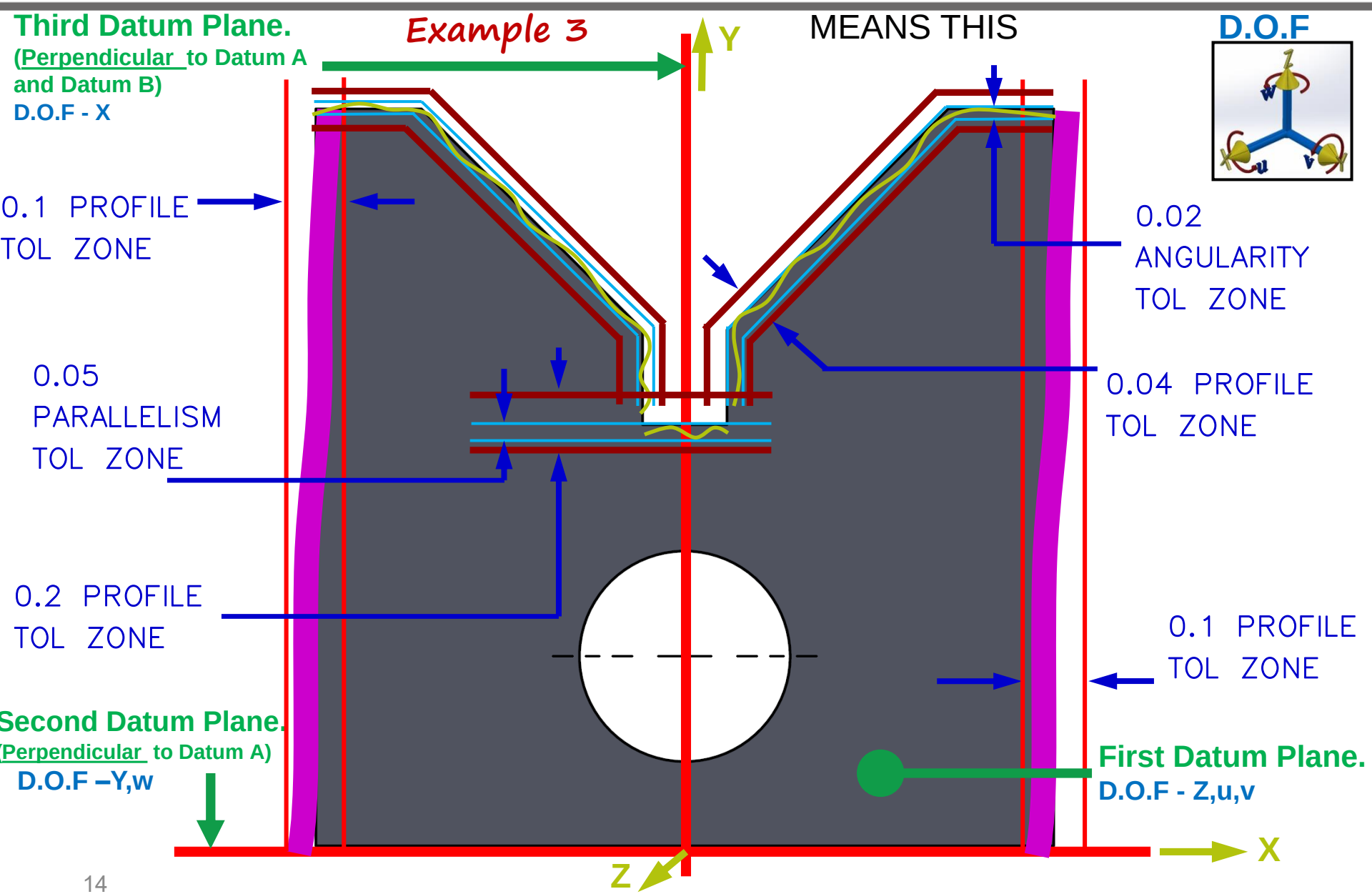
Example 3

ON DRAWING

Geometric Dimensioning & Tolerancing



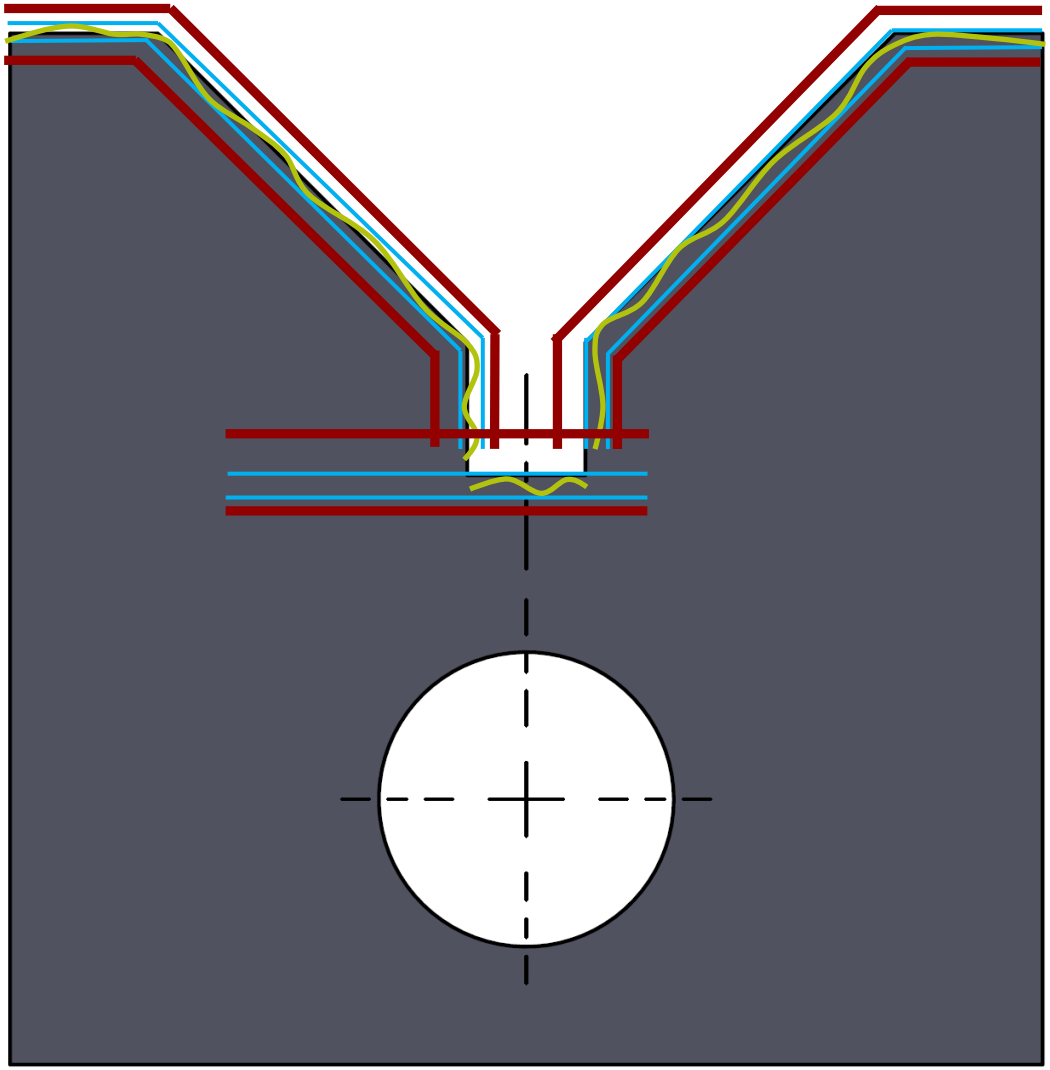
What is required to design and manufacture this component



What is required to design and manufacture this component

Example 3

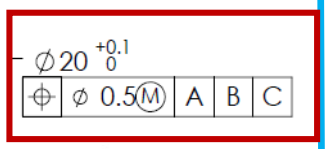
MEANS THIS

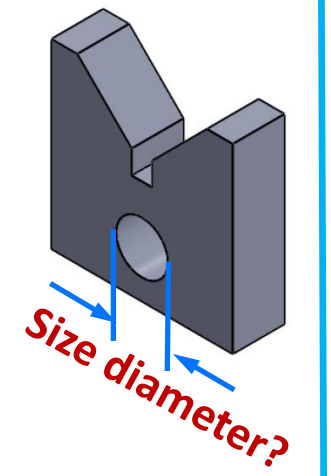


Example 3

How to measure a hole diameter

On drawing





Several methods of measurement

- caliper →

- Telescoping Gage Set →

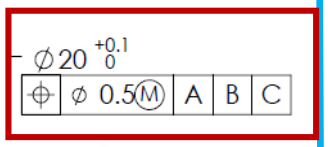
- pin gage →

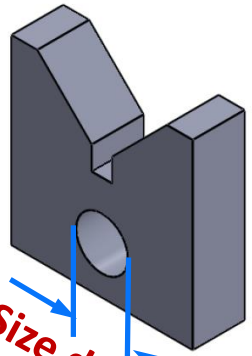
- Cmm measurement →

Example 3

How to measure a hole diameter

On drawing



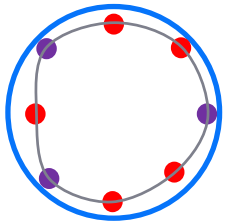


Cmm measurement

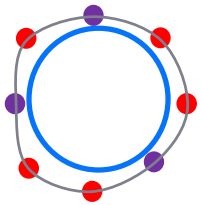
For a cylinder, a minimum of 6 points is required
3 points in 2 sections

cmm results option

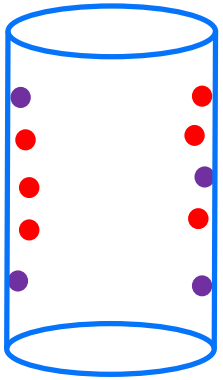
1) circumscribed cylinder



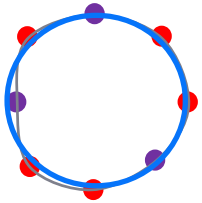
2) inscribed cylinder



3) AME Related \ Unrelated



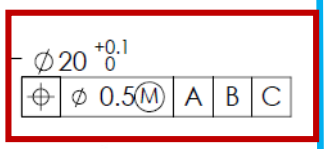
4) Average

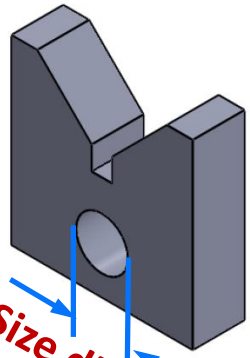


Example 3

How to measure a hole diameter

On drawing





Cmm **Average** result:

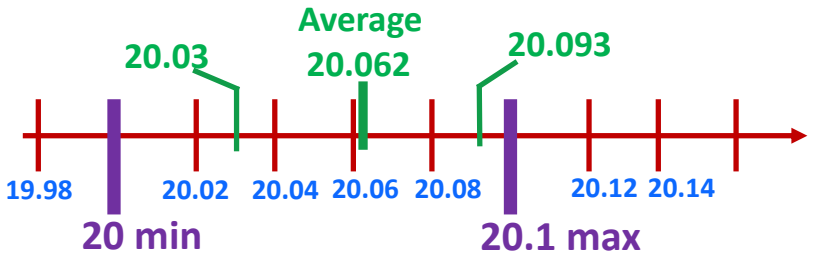
5 measurement points were taken

	Sampling number	Diameter Measurement
	1	20.07
	2	20.09
	3	20.09
	4	20.01
n=	5	20.05
	$\bar{x}$ **Average**	**20.062**
	$\hat{s}$ **standard deviation**	**0.033466401**
	D.O.F=n-1	5-1=4

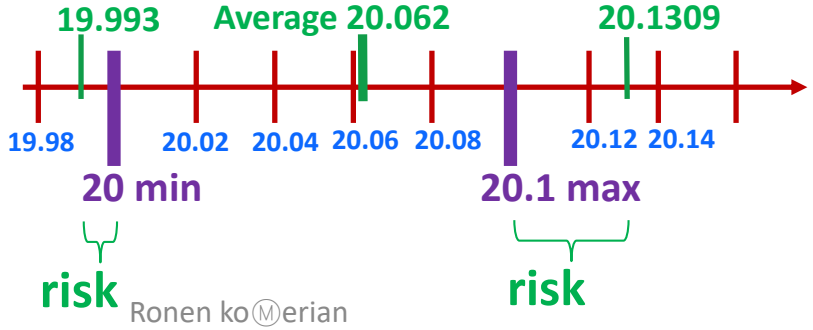
Confidence interval equation

$$p\left(\bar{x} - t_{\frac{\alpha}{2}}\frac{\hat{s}}{\sqrt{n}} < \mu < \bar{x} + t_{\frac{\alpha}{2}}\frac{\hat{s}}{\sqrt{n}}\right) = 1 - \alpha$$

Confidence interval		90%		
α		0.1		
α/2		0.05		
t(α/2)		2.132		
Minimum value	**>**	**μ**	**>**	**Maximum value**
20.03009115	**>**	**μ**	**>**	**20.09390885**



Confidence interval		99%		
α		0.01		
α/2		0.005		
t(α/2)		4.604		
Minimum value	**>**	**μ**	**>**	**Maximum value**
19.99309364	**>**	**μ**	**>**	**20.13090636**

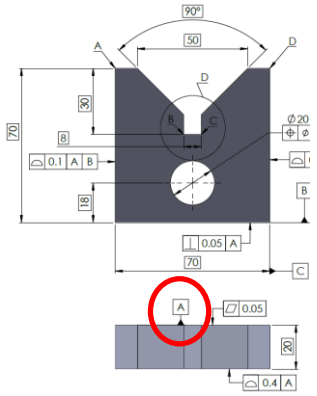


What is required to design and manufacture this component

Example 3

How to measure a Datum plane

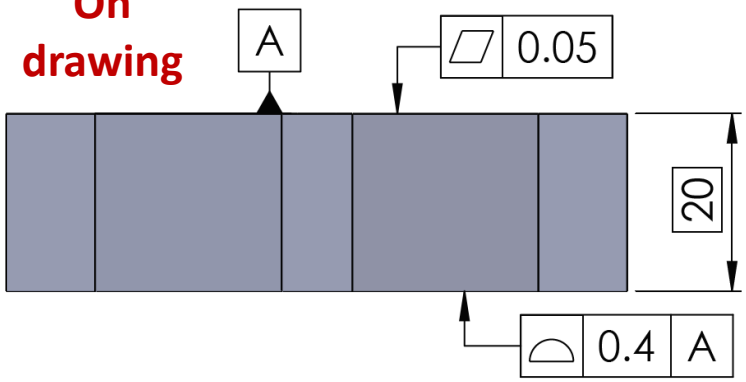
On drawing



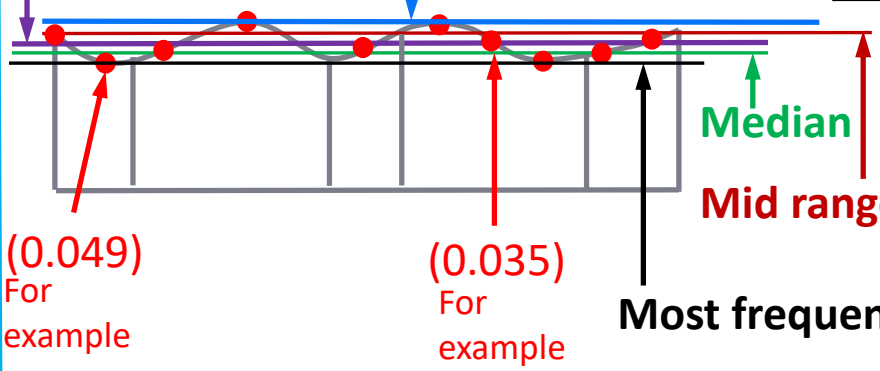
Cmm **Average** result:

10 measurement points were taken

On drawing



Means this



1	0.008	$\sqrt{\frac{\sum_{i=1}^{n}(x_i-\tilde{x})^2}{n}}$
2	0.01	
3	0.035	
4	0.012	
5	0.048	
6	0.049	
7	0.045	
8	0.049	
9	0.05	
10	0.025	

$\tilde{x}$-indicators		Least squares root
Average	0.0331	0.01683
mid-range	0.029	0.01732
Median	0.04	0.01819
Most frequent value	0.049	0.02315

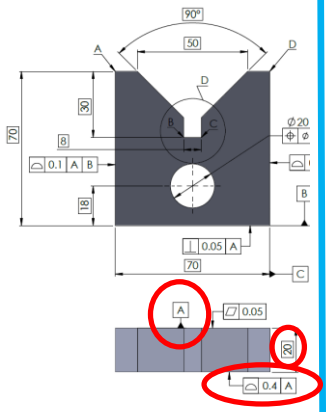
Example 3

How to measure a Datum plane

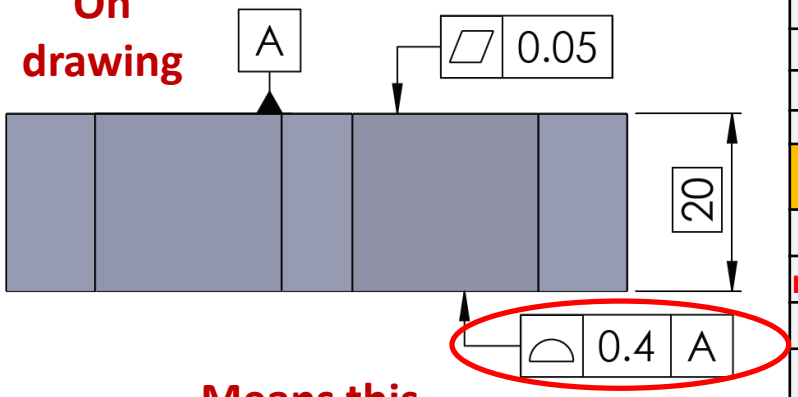
Cmm **Average** result:

10 measurement points were taken

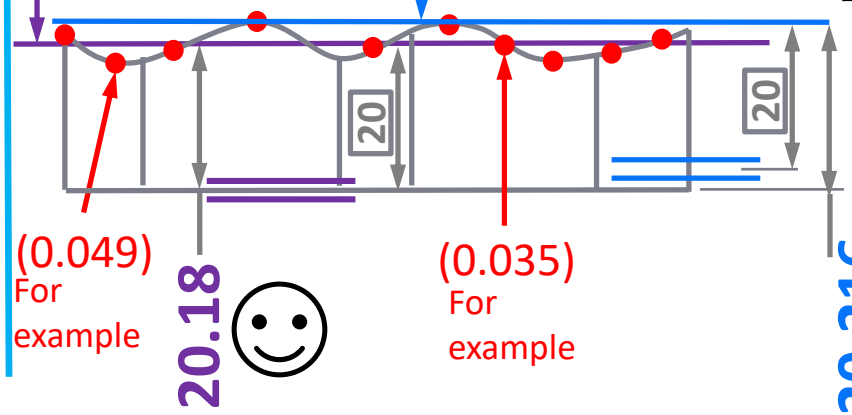
On drawing



On drawing



Means this



1	0.008	$\sqrt{\frac{\sum_{i=1}^{n}(x_i-\tilde{x})^2}{n}}$
2	0.01	
3	0.035	
4	0.012	
5	0.048	
6	0.049	
7	0.045	
8	0.049	
9	0.05	
10	0.025	

$\tilde{x}$-indicators		Least squares root
Average	0.0331	0.01683
mid-range	0.029	0.01732
Median	0.04	0.01819
Most frequent value	0.049	0.02315

What is required to design and manufacture this component

Example 3 - Position

MEANS THIS

MEANS THIS

∅0.6 Diameter tolerance zone

Hole at LMC
Ø20.1

Virtual Condition =
Hole at MMC - T.O.P
Ø20-Ø0.5=Ø19.5

Resultant Condition =
LMC + Total Pos Tol
Ø20.1+Ø0.6=Ø20.7

What is required to design and manufacture this component
Example 3

Functional gage policy

1. Absolute
2. Practical Absolute
3. Optimistic
4. Tolerant

Pin diameter and TOP?

Hole size

$\varnothing 20^{+0.1}_{0}$

⌖	⌀ 0.5 Ⓜ	A	B	C

Absolute gage policy – 10%

"Accepts no bad parts even in theory"

All gage PINS are PLUS in tolerance.
(5% ÷ 10%)

Hole VC = Hole at MMC - T.O.P Ø20-Ø0.5=Ø19.5

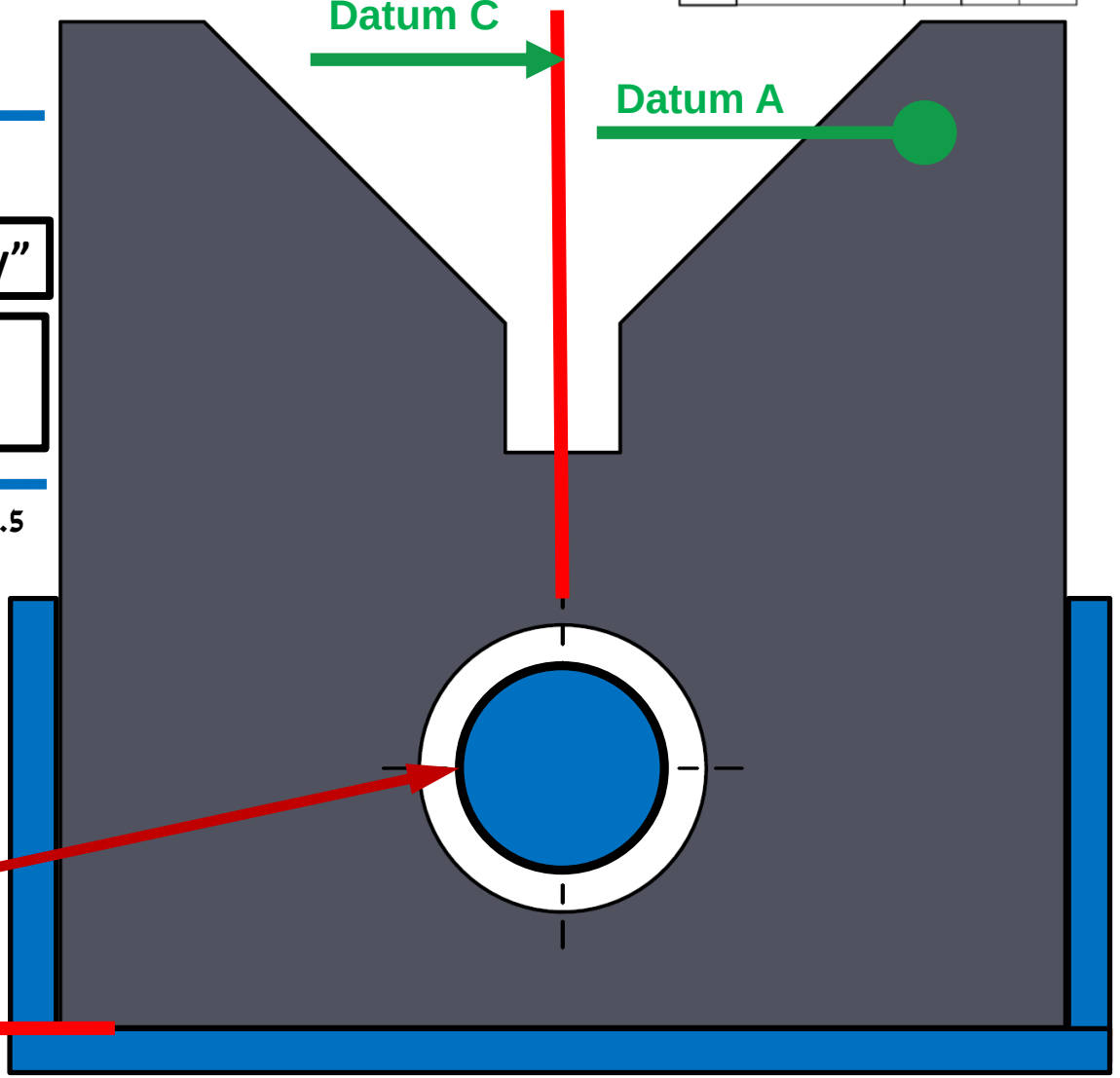
Hole tolerance = Hole at LMC - Hole VC
Ø20.1-Ø19.5=Ø0.6

Pin gage diameter Hole VC =Ø 19.5

Pin gage tol diameter
10% from Hole tol diameter
0.1*0.6 = 0.06

Pin T.O.P
Ø0 LMC
Ø19.5+0.06

⌖	Ø0Ⓛ	A	B	C

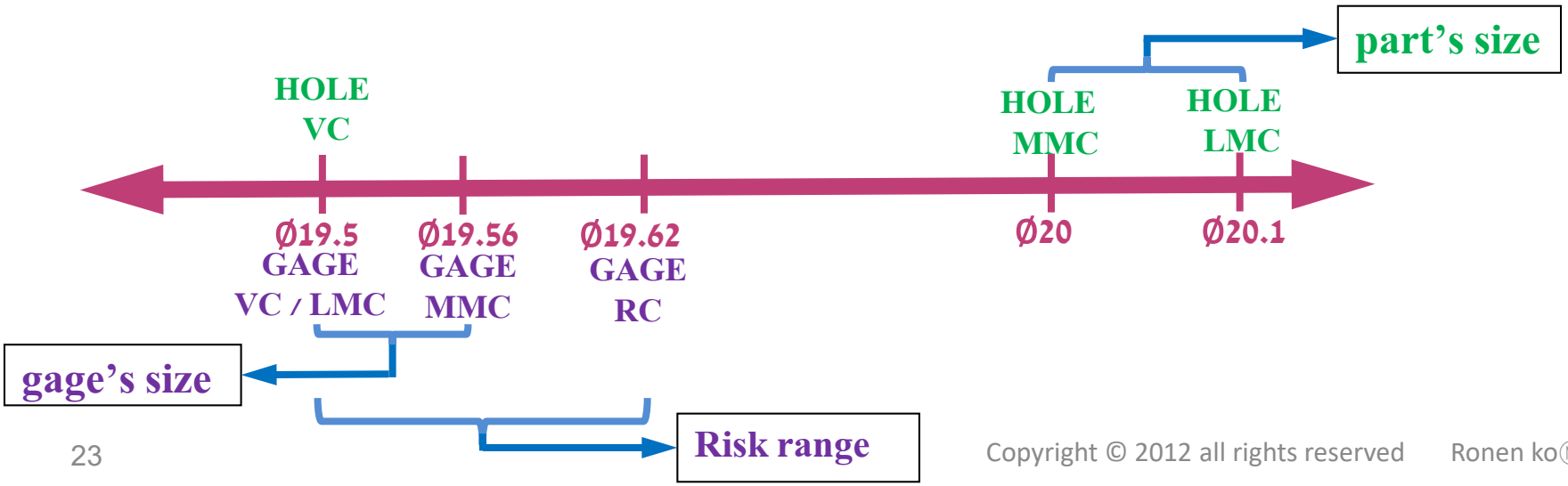


What is required to design and manufacture this component

Example 3 – Absolute gage policy - 10%

Gage Calculate

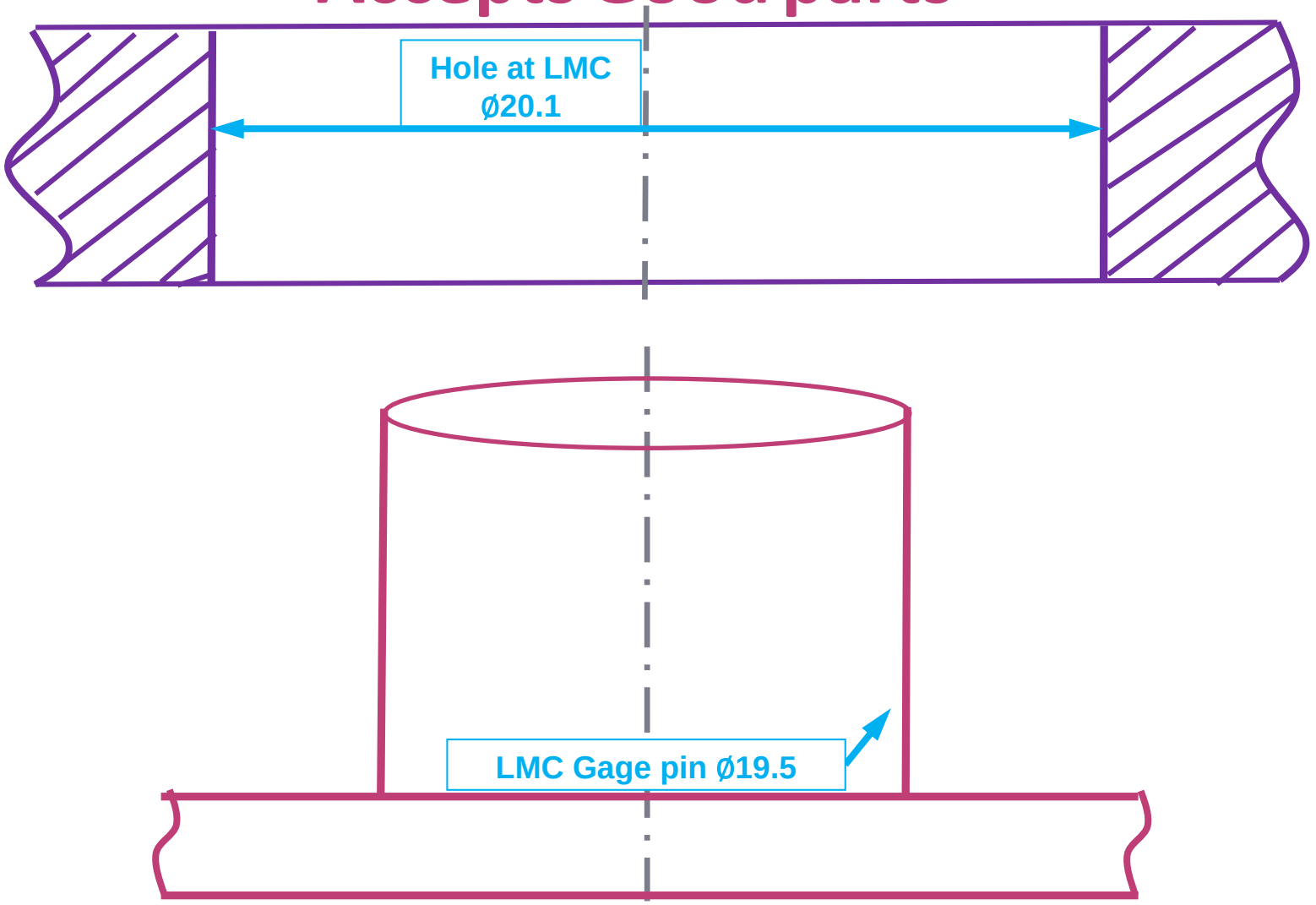
	Pin gage diameter	T.O.P (0 LMC)	Virtual Condition	Resultant Condition
LMC	19.5	0	19.5	19.5
	19.51	0.01		19.52
	19.52	0.02		19.54
	19.53	0.03		19.56
	19.54	0.04		19.58
	19.55	0.05		19.6
MMC	19.56	0.06		19.62



acceptance or rejection Parts according ABSOLUTE GAGE policy

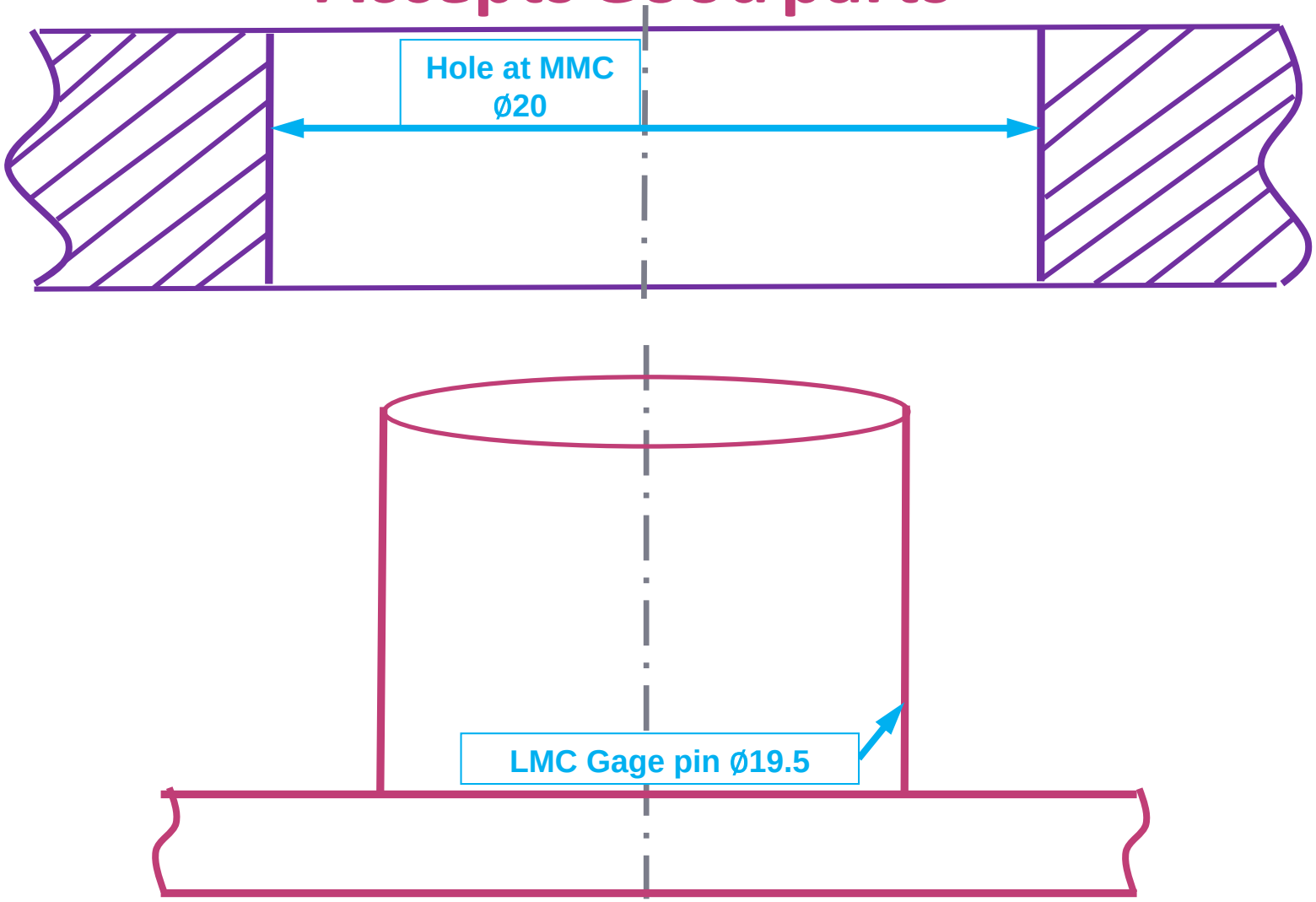
Example 3 – Absolute gage policy - 10%

Accepts Good parts



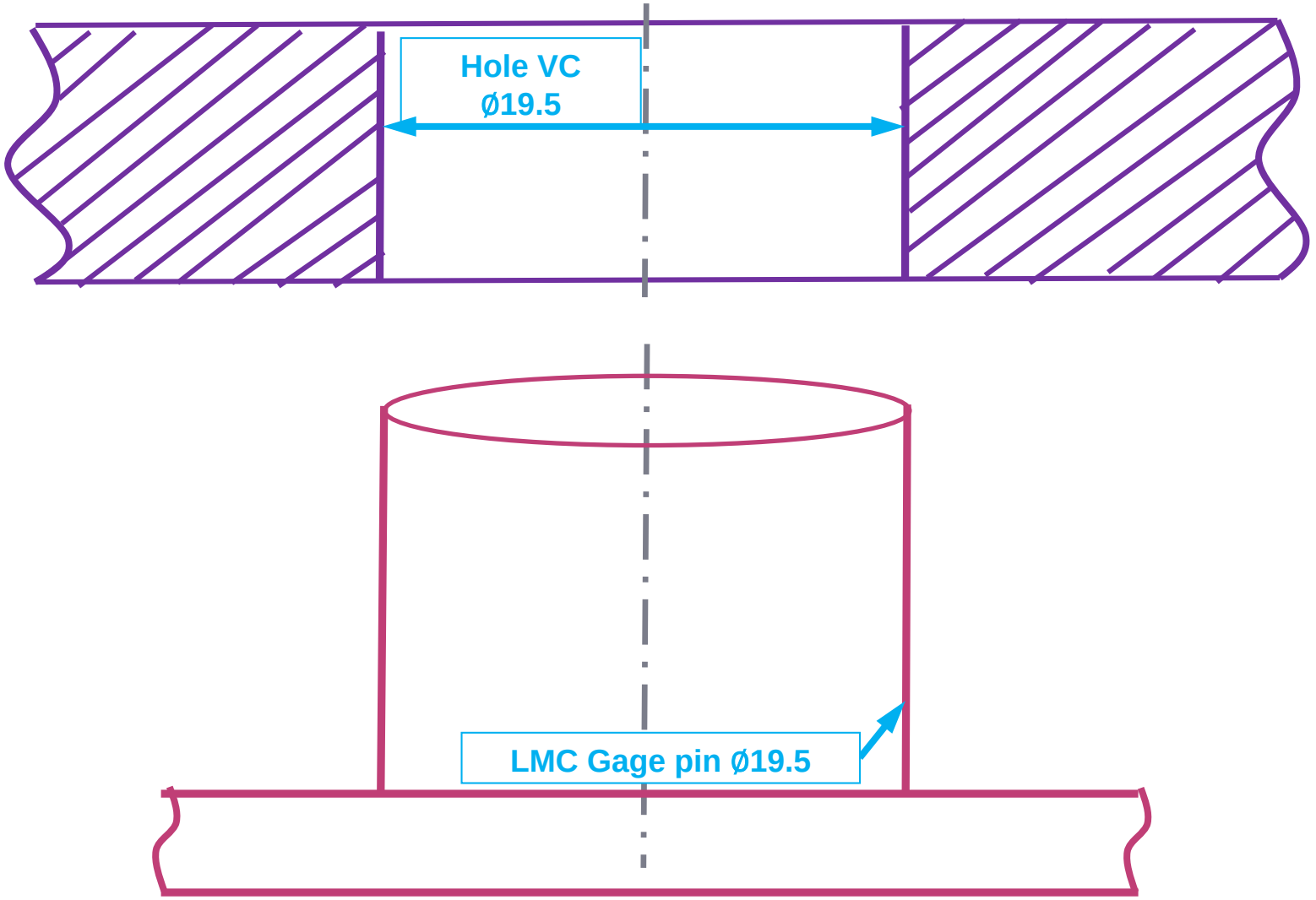
Example 3 – Absolute gage policy – 10%

Accepts Good parts



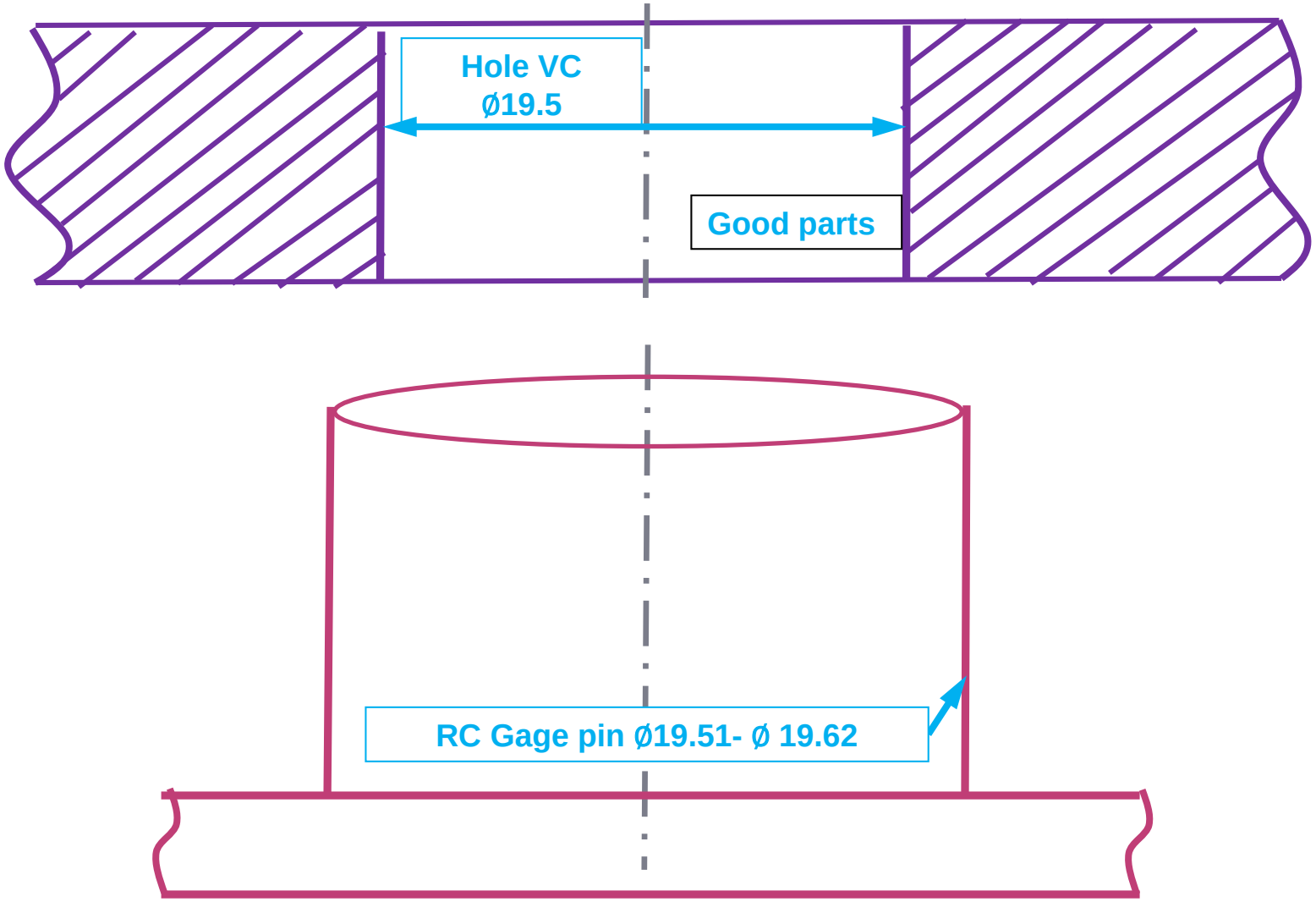
Example 3 – Absolute gage policy – 10%

Accepts Good parts



Example 3 – Absolute gage policy – 10%

Reject a Good parts



THANK YOU, QUESTIONS?

RONEN KOMERIAN