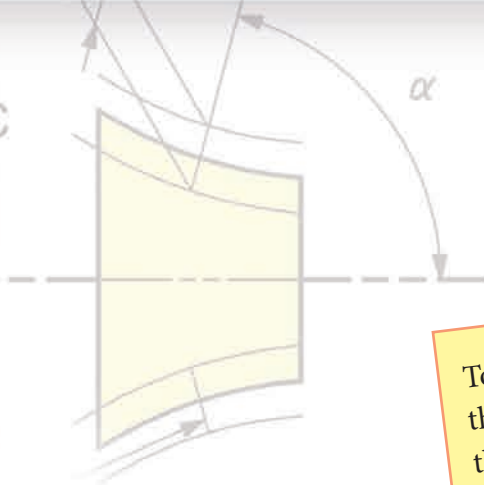
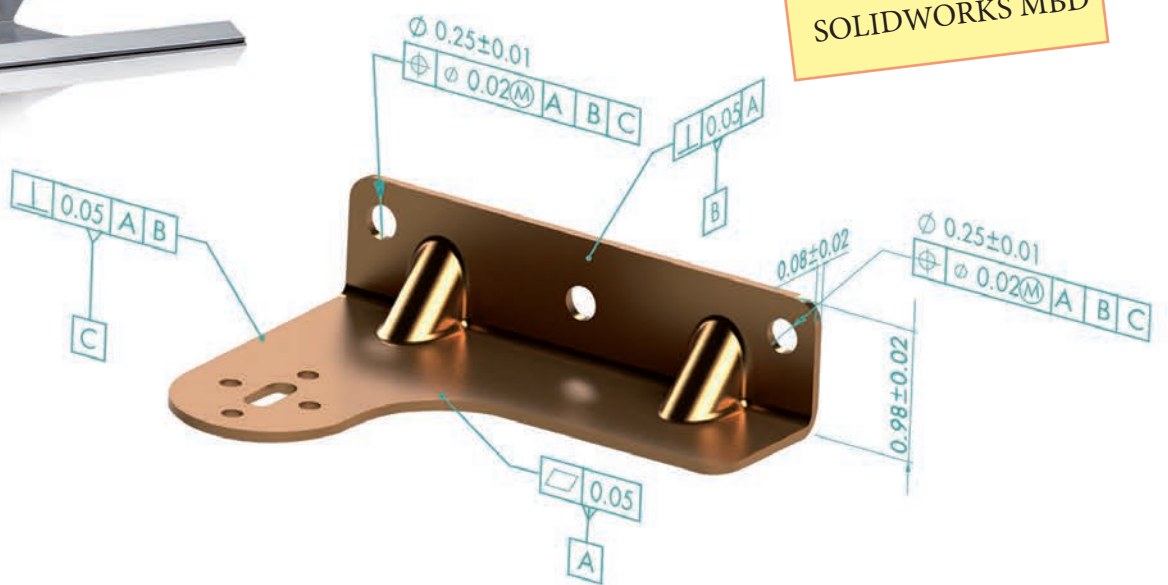


STEFANO TORNINCASA

TECHNICAL PRODUCT DOCUMENTATION USING ISO GPS - ASME GD&T STANDARDS



To help understand the 3D annotations, the book includes a complete tutorial on SOLIDWORKS MBD



EDIZIONI
il capitelletto

FOREWORD

Designers create perfect and ideal geometries through drawings or by means of Computer Aided Design systems, but unfortunately the real geometrical features of manufactured components are imperfect, in terms of form, size, orientation and location.

Therefore, technicians, designers and engineers need a symbolic language that allows them to define, in a complete, clear and unambiguous way, the admissible variations, with respect to the ideal geometries, in order to guarantee functionality and assemblability, and to turn inspection into a scientifically controllable process. The *Geometric Product Specification (GPS)* and *Geometrical Dimensioning and Tolerancing (GD&T)* languages are the most powerful tools available to link the perfect geometrical world of models and drawings to the imperfect world of manufactured parts and assemblies.

This book is intended for designers, process engineers and CMM operators, and it has the main purpose of presenting the ISO GPS rules and concepts. Moreover, the differences between ISO GPS and the American ASME Y14.5M standard are shown as a guide and reference for the drawing interpretation of the most common dimensioning and tolerancing notations.

A complete SolidWorks MBD tutorial has been added to the appendix of this book: *SOLIDWORKS Model Based Definition (MBD)* is a drawingless manufacturing solution that is embedded inside a SOLIDWORKS user interface. It helps companies define, organise and publish product and manufacturing information (PMI) in a 3D format that complies with international standards.

The author, Professor Stefano Tornincasa, has carried out research activities for over thirty years in the field of functional design and geometric tolerances. He was President of the ADM Improve Association (Innovative Methods in PROduct design and deVELOPMENT) from 2011 to 2015 and has published more than 180 national and international scientific papers.

He is co-author of the best-selling book on Industrial Technical Drawing, which is currently adopted in the design courses of most Italian universities (E. Chirone, S. Tornincasa, Industrial Engineering Design, Volumes I and II, ed. Il capitello Torino).

Professor Tornincasa has conducted training courses on GD&T in many of the main manufacturing companies in Italy, and it is from this activity that he has derived his skill and experience in functional design.

His other research topics have been focused on product development, cycle innovation through digital models and virtual prototyping methodologies (PLM)

<http://webd.polito.it/workbook/>

INDEX

FOREWORD.....	PAGE 3
1. INTRODUCTION	4
2. CLASSIFICATION AND INDICATION OF GEOMETRIC TOLERANCES	9
3. DIMENSIONING WITH GEOMETRICAL TOLERANCES	18
4. THE GD&T LANGUAGE ACCORDING TO THE ISO AND ASME STANDARDS.....	22
The fundamental ISO 8015 standard.....	25
General geometrical tolerances.....	29
The main differences between the ISO GPS and ASME GD&T standards.....	35
5. INTERDEPENDENCE BETWEEN THE SIZE AND FORM	37
Maximum material condition.....	37
Least material conditions.....	40
Virtual condition.....	40
6. DATUMS	42
Indication of the datum features.....	44
Location of a workpiece in a datum reference frame	48
Selection of the datum features.....	49
Types of datums.....	52
Datum features referenced at MMR and LMR (Size datum)	55
Customised datum reference frame.....	56
Examples of other modifiers used to indicate datums.....	59
Datum targets.....	61
Contacting feature.....	65
7. FORM TOLERANCES.....	68
Straightness tolerance.....	68
Flatness.....	72
Roundness.....	76
Cylindricity.....	77
8. ORIENTATION TOLERANCES	79
Parallelism.....	79
Perpendicularity.....	82
Angularity.....	85
9. LOCATION TOLERANCES.....	89
Position tolerances.....	89
Position tolerance applied to median surfaces.....	93
Effects of Specifying the MMR Modifier.....	105
Concentricity.....	107
Symmetry.....	108
10. PROFILE TOLERANCES.....	114
11. RUN-OUT TOLERANCES.....	120
Circular run-out.....	121
Total run-out.....	122
12. GEOMETRICAL PRODUCT SPECIFICATION FOR NON-RIGID PARTS.....	124
1. Introduction to SOLIDWORKS MBD	128
2. The first step towards MBD: making a 3D model the master by leveraging on modelling dimensions with annotation views	130
3. Using DimXpert for coordinate tolerancing.....	137
4. 3D Views	148
5. GD&T with DimXpert	154
6. Preparing the model and reading manufacturing information	161
7. Leveraging on PMI	168
INDEX.....	172

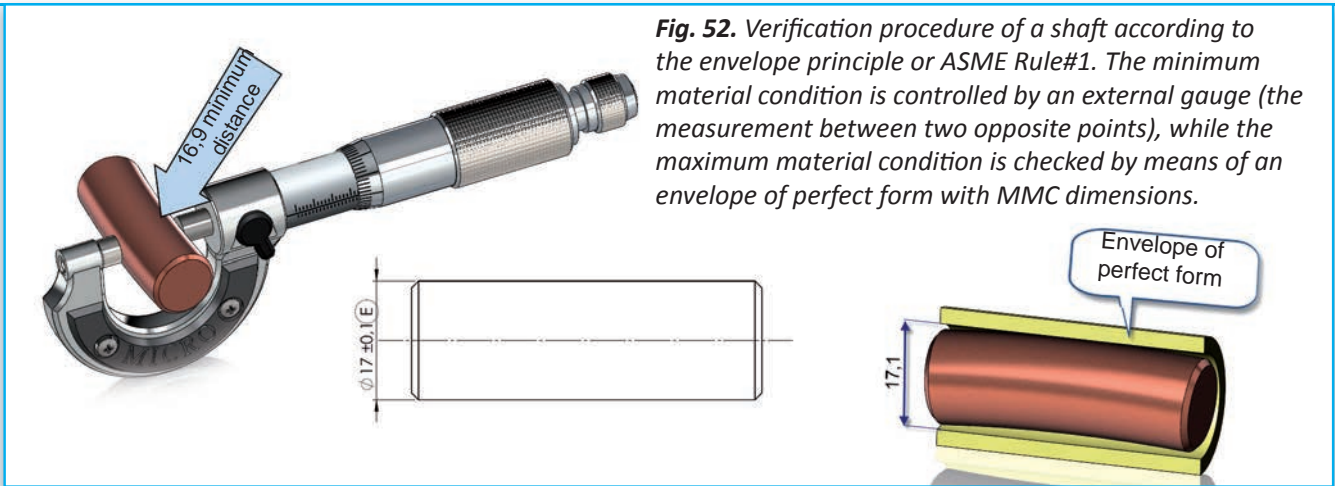


Fig. 52. Verification procedure of a shaft according to the envelope principle or ASME Rule#1. The minimum material condition is controlled by an external gauge (the measurement between two opposite points), while the maximum material condition is checked by means of an envelope of perfect form with MMC dimensions.

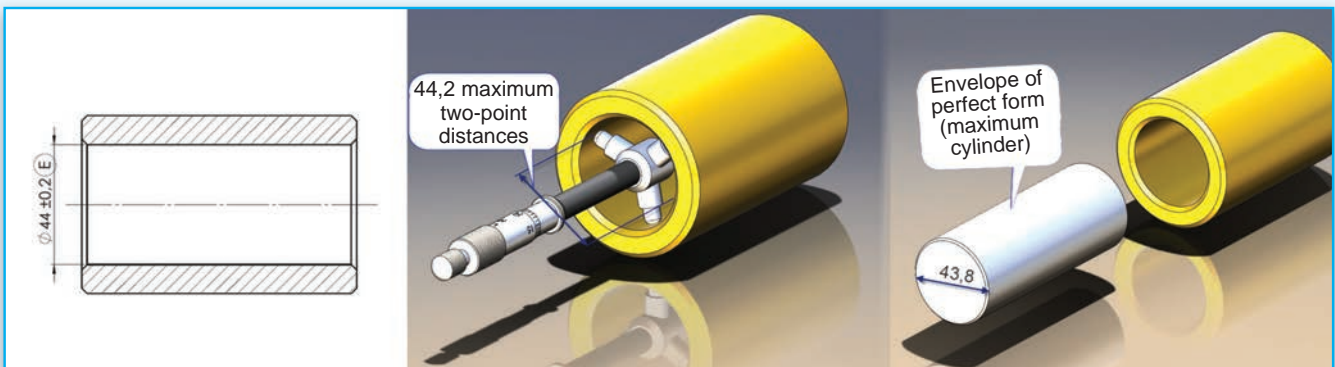


Fig. 53. Verification procedure of a hole according to the envelope principle. The minimum material condition is controlled by an internal gauge (measured between two opposite points), while the maximum material condition is checked by means of a pin with the MMC dimensions.

appropriate in the case of mating, may be restrictive for all the other geometrical features, and may make it necessary, in the latter case, to furnish an indication of exception (the ASME standards have introduced the $\text{\textcircled{I}}$ symbol, see Fig. 55), with the consequence of a source of ambiguity being created as it is not possible to be certain that the absence

of such an indication depends on the choices of the designer or rather on an oversight within a complex technical document.

Apart from this problem, the verification of the envelope principle, which requires the use of functional gauges¹ or controls carried out by means of measurement machines that have

¹Gage in ASME

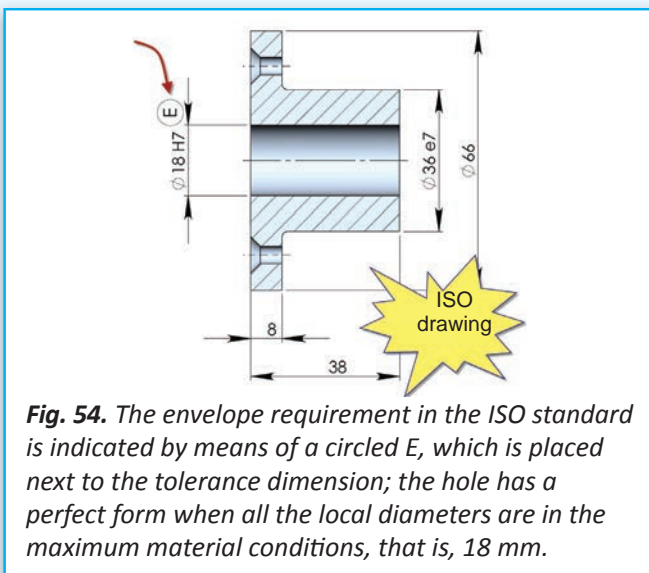


Fig. 54. The envelope requirement in the ISO standard is indicated by means of a circled E, which is placed next to the tolerance dimension; the hole has a perfect form when all the local diameters are in the maximum material conditions, that is, 18 mm.

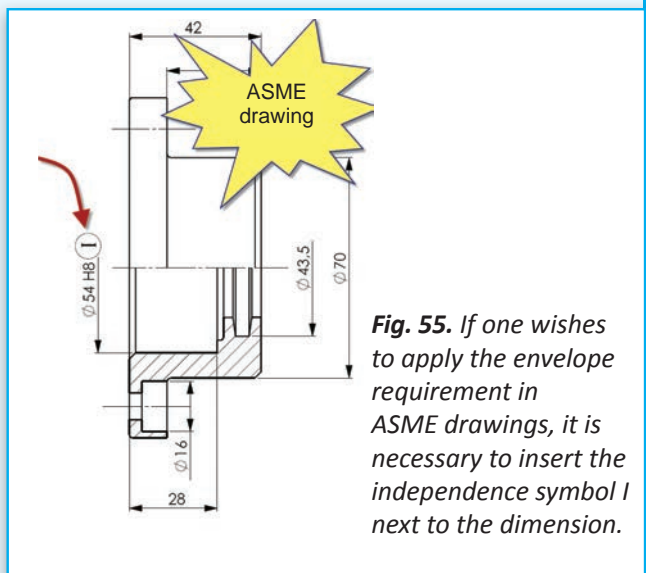


Fig. 55. If one wishes to apply the envelope requirement in ASME drawings, it is necessary to insert the independence symbol I next to the dimension.

A

ACS (Any Cross Section); 59; 107
 Actual Mating Envelope,
 AME; 46
 All around; 14
 All over; 14
 Altered default GPS
 specification; 25
 Angularity; 85
 ASME **BSC** (Basic); 67
 ASME Y14.5 standard; 23
 ASME-ISO comparison; 37
 Associated feature; 9
 Association methodology; 59
 Axis methodology; 94

B

Basic Dimension; 14
 Bi-directional tolerance
 of position; 105
 Bonus; 38

C

Centerplanes; 52
CF symbol, see Contacting
 feature
 Circular run-out; 120, 121
 Circularity, see Roundness; 76
 Classification of Geometrical
 Tolerances.; 11
 Coaxiality; 107
 Collection plane; 17
 Combined zone; 15
 Common datum; 53
 Common Zone, see Combined
 Zone; 15
 Complementary standard; 24
 Composite Position
 Tolerancing; 101
 Composite profile feature; 118
 Composite tolerance frame; 71
 Computer Aided Design; 5
 Concentricity; 107
 Concentricity ASME; 109

Contacting feature; 65
 Coordinate dimensioning; 6
 Coplanar Surfaces.; 119
 Customised datum reference
 frame; 56
 Cylindricity; 77
CZ (Combined Zone); 116

D

Datum; 42
 Datum axis; 44
 Datum conical surfaces; 55
 Datum feature simulator; 45
 Datum features; 43
 dDatum pattern of holes; 53
 Datum targets; 61
 Default principle; 28
 Degrees of freedom of a
 workpiece; 48
 Derived feature; 9
 Derived median line; 72
DRF (Datum Reference
 Frame); 48
 Duality principle; 28,35
DV (distance variable; 54

E

Envelope requirement; 32
 Extracted derived feature; 9

F

Feature of Size; 19
 Feature-Relating Tolerance Zone
 Framework; 109
 Filter specification; 77
 Fixed fastener formula; 98
 Flatness; 72
 Flatness (ASME); 75
 Floating fastener formula; 97
 Form tolerances; 10; 68
 Free State condition; 124
 Functional dimensioning; 4
 Functional gauge; 66. 111

Functional gauging
 techniques; 72
 Functional limits; 28
 Fundamental standard; 24

G

Gage; 33. 111
 Gauges; 33
 Gaussian (**G**) method; 59
 Gaussian circle; 70
 Gaussian dimensioning
 concept; 35
 Gaussian interpolation; 70
 GD&T; 5; 22
 General geometrical
 tolerances; 29
 General specification
 principle; 29
 General standard; 24
 Geometrical Casting Tolerance
 Grades; 31
 Geometrical tolerances
 Summary chart; 116
GPS; 9; 22
 GPS Matrix Model; 23

I

Independency principle; 25
 Indication of a derived
 feature; 13
 Indication of the
 datum features; 44
 Inspecting concentricity; 109
 Inspecting flatness; 74
 Inspection of roundness; 77
 Integral; 9
 Integral feature, indication; 13
 Intersection plane; 15
 Invocation principle; 26
 ISO 10579-NR; 124
 ISO 1101; 9; 24
 ISO 14638; 24
 ISO 2768/2; 29
 ISO 5459; 43; 58

ISO 8015; 25
ISO 8062-3; 31
ISO/TC 213; 23

L

Least Material Condition
(**LMC**); 32, 40
Least Material Requirement,
LMR; 40
Least Material Size (**LMS**); 97
Least Material Virtual
Condition, **LMVC**; 40
Least Material Virtual Size,
LMVS; 40
LMB Least Material Boundary; 55
Location tolerances; 10; 89

M

Maximum Material
Boundary (**MMB**); 61
Maximum Material Requirement,
(**MMR**); 37
Maximum Material
Size (**MMS**); 41
Maximum Material Virtual
Condition, **MMVC**; 40
Maximum Material
Virtual Size, **LMVS**; 40
Maximum Material Condition
(**MMC**); 32
Minimax Chebyshev; 18
Minimum circumscribed
association; 78
Minimum zone criterion
(Chebyshev); 77
MMB Maximum Material
Boundary; 55
MMR Applicability; 39
modifier T; 18
modifier U; 119
modifier X; 18

N

Non Rigid part; 124

O

Order of the datums; 49
Orientation plane; 16
Orientation tolerances; 10; 79
OZ symbol; 117

P

Parallelism; 79
Pattern-Locating Tolerance Zone
Framework; 109
Perpendicularity; 82
Position tolerance ASME; 109
Position tolerance median
surfaces; 93
Position tolerances; 89
Position tolerances
calculation; 96
Profile; 114, 115
Profile any line; 116
Profile tolerances; 116
Projected tolerance zone; 98, 99

Q

Qualification of the datum
features; 50

R

Reciprocity; 103
Reciprocity
requirement (**RPR**); 104
Regardless of
Feature Size, **RFS**; 37
Related actual mating
envelope; 47
Restraint Note; 126
Rigid workpiece principle; 28
RMB, Regardless of Material
Boundary; 55
Roundness; 76
Rule#1; 32
Runout tolerances; 10
Run-out tolerances; 120

S

Selecting modifiers for position
tolerances; 95
Selection of the
datum features; 49
SF (Fixed Size); 67
Shift; 106
Simulated datums; 45
Size datum; 55
Straightness; 68
Straightness (ASME); 72
Symmetry; 108, 110
SZ (Separate Zone); 94

T

Taylor's principle Rule#1; 32
Theoretical envelope plane; 43
Theoretically Exact; 89
Theoretically Exact
Dimensions; 14
Tolerance frame; 13
Theoretically
exact feature (**TEF**); 117
Tolerance indicator; 12
Tolerance, summary chart; 127
Total run-out; 120, 122

U

UF (United Feature); 59
UZ symbol; 117

V

Virtual boundary condition
methodology; 94
Virtual condition; 40

Z

Zero tolerance; 103

Fig. 232. The tolerance zone is located by means of theoretically exact dimensions with respect to the system constituted by three mutually orthogonal planes, A, B and C.

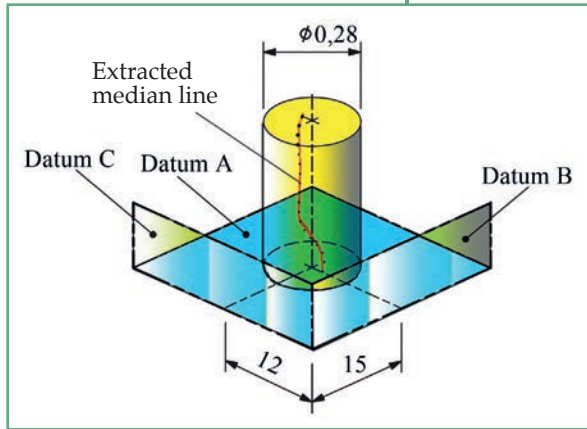
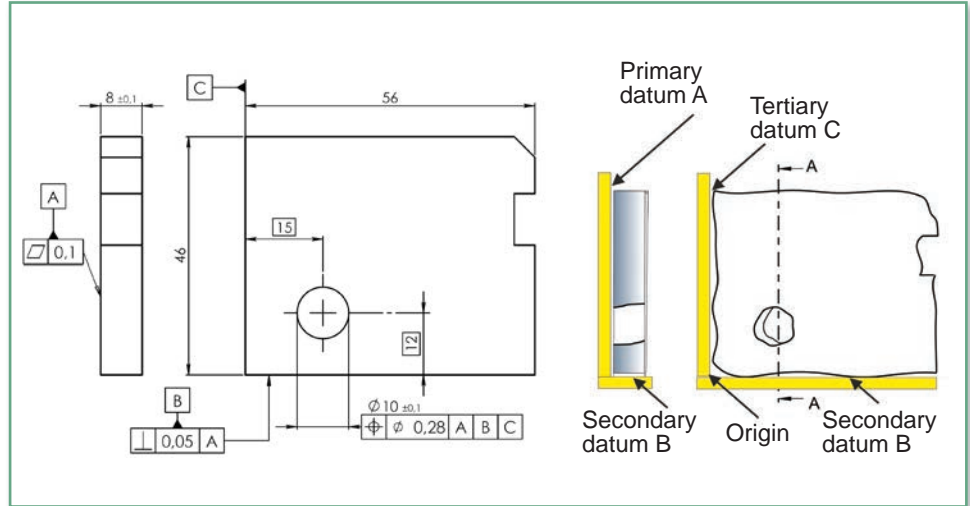


Fig. 233. Interpretation of the tolerance zone shown in Figure 232. The extracted, or derived, median line must fall within a 0,28 mm cylinder that is perpendicular to datum A and located in a theoretically exact way with respect to datums B and C.

theoretically exact dimensions with respect to three mutually orthogonal planes, A, B and C, which can be simulated by means of verification equipment, as shown in Figure 232.

The extracted, or derived, median line must remain within a 0,28 mm diameter cylinder that is perpendicular to datum A, and positioned in a theoretically exact way with respect to datums B and C (Fig. 233). The virtual dimension concept is presented in Figure 234 for the case of position tolerances; as usual, the maximum material virtual size (MMVS) of 50,2 mm of the pin is calculated by summing the maximum material size (50 mm) with the geometrical tolerance of position (0,2 mm). In the case of the hole shown in Figure 235, the most unfavourable condition is obtained by subtracting the position tolerance (0,2 mm) from the maximum material size (30 mm), thus a maximum material virtual size (MMVS) of 29,8 mm is obtained.

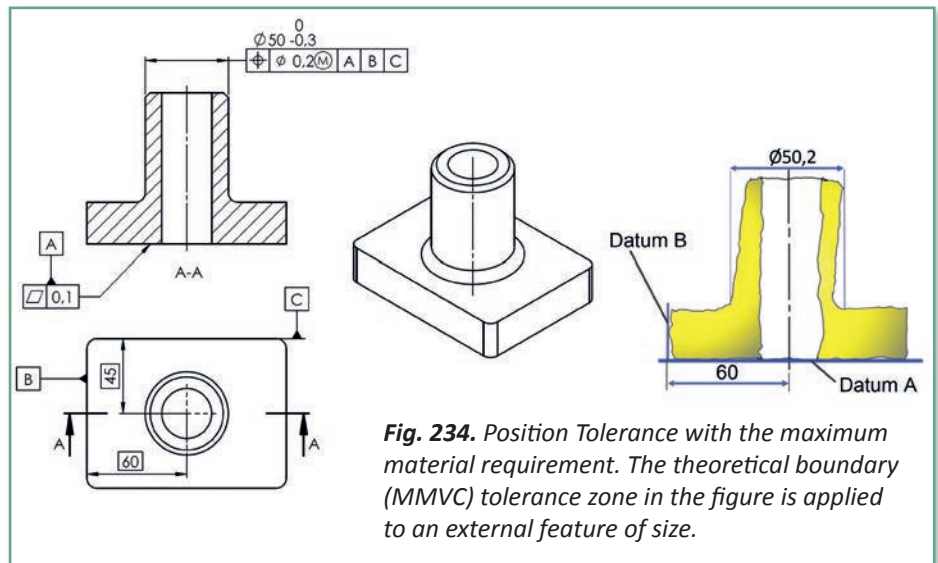


Fig. 234. Position Tolerance with the maximum material requirement. The theoretical boundary (MMVC) tolerance zone in the figure is applied to an external feature of size.

The virtual dimension defines the theoretical mating boundary, whose position is controlled by the position tolerance, and it is a constant quantity, in the sense that when the pin and the hole are produced at minimum material condition (diameters of 49,7 mm and 30,3 mm, respectively), the position

When dimensioning 2D drawings, it is left to the designer to judge whether all the manufacturing features have been completely defined or not.

DimXpert assists a designer by giving him/her a graphical display to help understand whether all the manufacturing dimensions have been defined or not. This is accomplished by using the “show tolerance status” command, which can be found in DimXpertManager (Fig. 20).

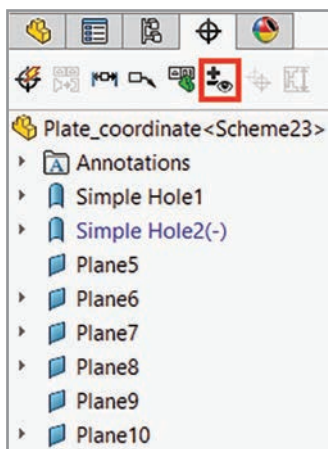


Fig. 20

By clicking on this command, the face of a part may be coloured with 3 different colours that have specific meanings:

- **Original colour of a part**

The considered manufacturing feature does not have any definition.

- **Yellow**

The manufacturing feature is not completely defined.

- **Green**

The manufacturing feature has been completely defined.

- **Red**

The manufacturing feature is over-defined.

In this case, all the faces of the part are green, with the exception of the 20 mm hole, which is shown in yellow. This indicates that it is not completely defined,

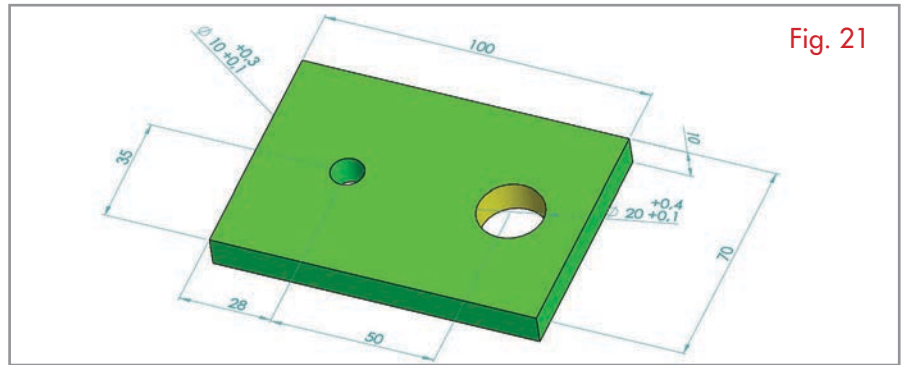


Fig. 21

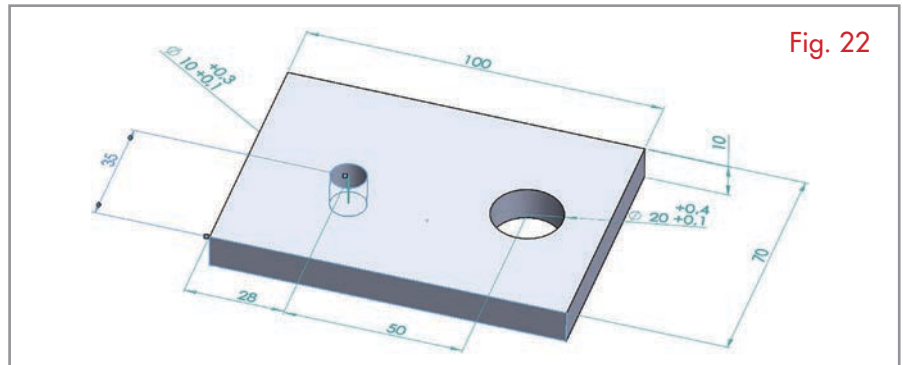


Fig. 22

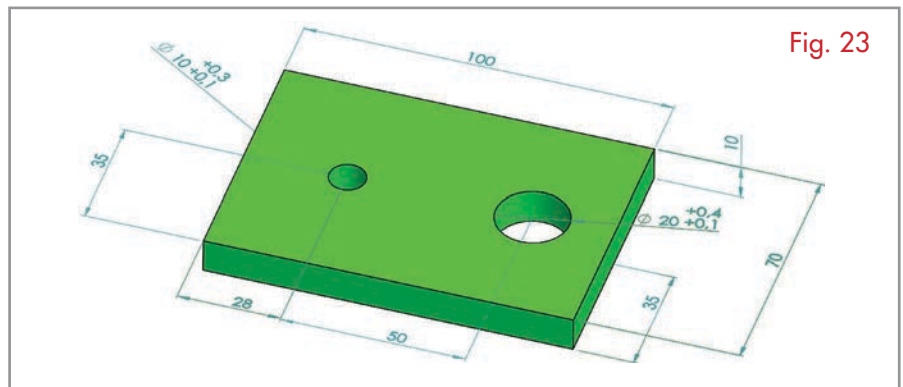


Fig. 23

as can be confirmed from the feature-based DimXpertManager tree, where a minus symbol can be seen next to the “Simple Hole2 feature”.

By putting oneself in the position of a person who is used to reading 2D drawings, it is possible to state, without doubt, that this drawing has all the necessary manufacturing dimensions. This is because a horizontal axis could be drawn across both holes on the 2D drawing, and it is hoped that the person who reads the drawing will assume that the two holes are horizontally aligned and the 35 mm vertical dimen-

sion is sufficient to define the vertical position of both of them.

DimXpert cannot make this assumption (drawing an axis here would not help). In fact, if the 35 mm dimension is clicked on, only the 10 mm hole will be highlighted, while the 20 mm hole will not (Fig. 22), which indicates that a vertical dimension would be needed to completely define the 20 mm hole location.

Adding the other 35 mm dimension and again checking the tolerance status leads to all the faces appearing in green, which indicates that all the manufacturing features are now completely defined (Fig. 23).

Fig. 20

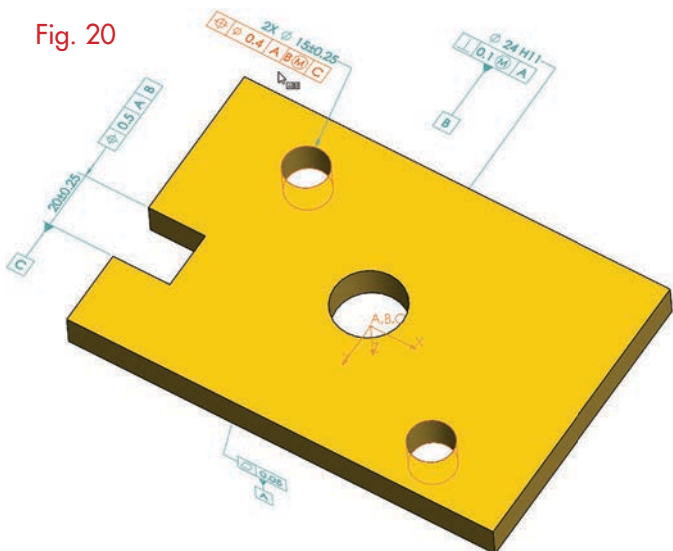
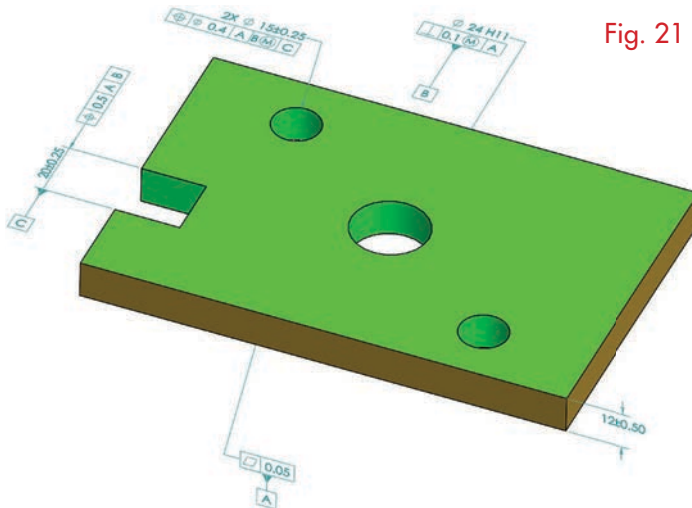


Fig. 21



It can be noticed that, after creating a surface profile tolerance, all the manufacturing features will be defined completely (since their tolerance status appears in green and, by showing DimXpertManager in the feature based display, no feature appears as underdefined).

NOTE: as an alternative approach to creating a collection and surface profile tolerance, it is possible to create a general profile tolerance to completely define the model using the **general profile tolerance** in DimXpertmanager.

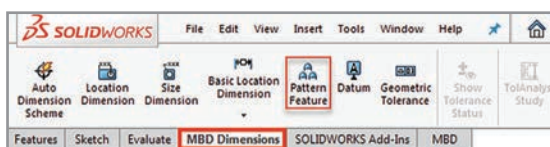


Fig. 22

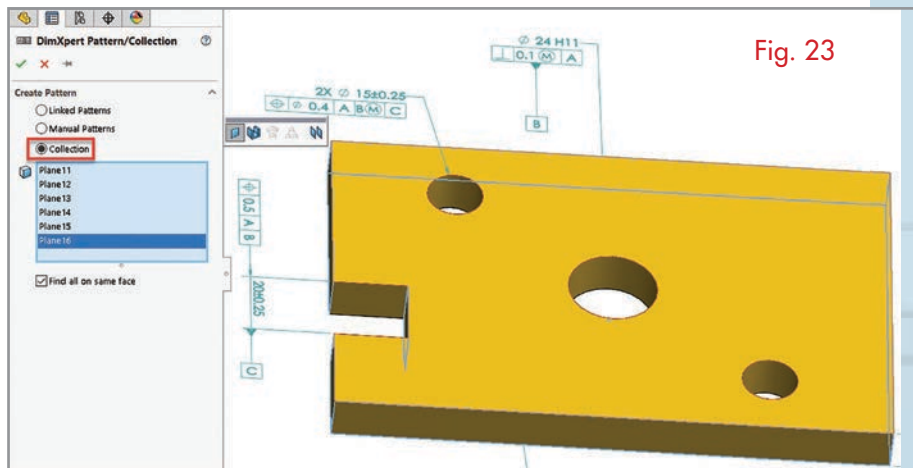


Fig. 23



Fig. 24

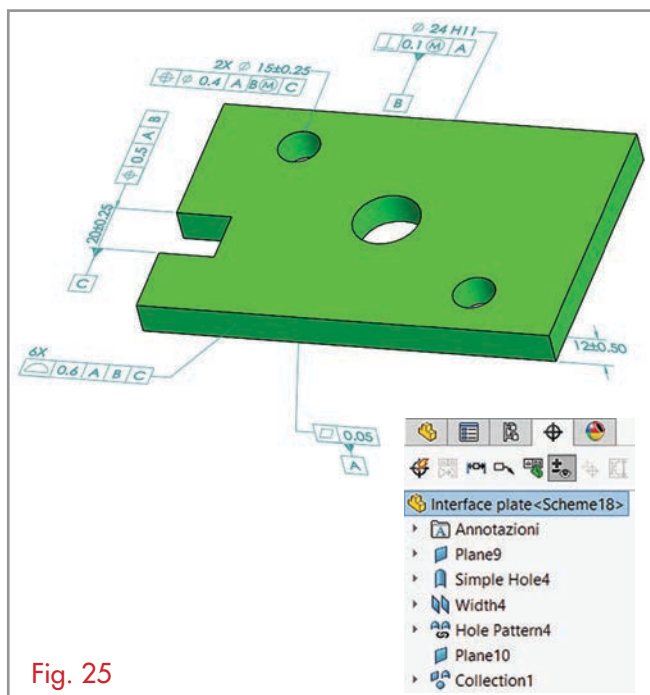
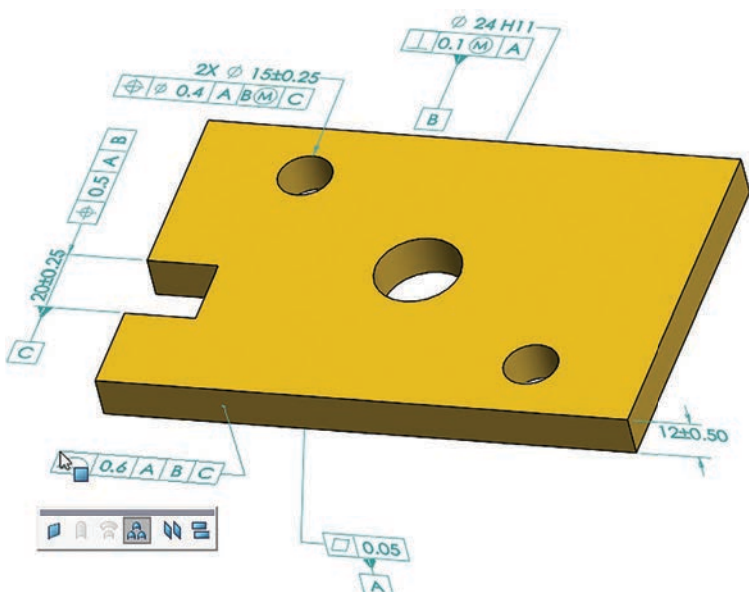


Fig. 25