1 Scope

This document specifies the default dimensional and geometrical tolerances and details of sheet metal parts.

.

Chapter 2 shows general information.

Chapter 3 shows an abbreviation list.

Chapter 4 shows a list of terms and definitions.

Chapter 5 shows general subjects and specifications of details of a part.

Chapter 6 specifies general tolerances.

Chapter 7 shows document references.

Chapter 8 shows document changes and history.

The specifications in this document are only valid when a reference to this document is placed on the drawing above the title block.

Specifications in this document are overruled by the specifications on the drawing or other technical product documentation.

A printed version of this document is considered to be outdated.

2 General

Parts must have a general specification level with respect to dimensional and geometrical tolerances, form features and details. The general level is documented in this document.

What is dsTPD:
A leading STEP file, standard tolerances (chapter 6) and an additional dimensioned PDF.

A digital sheet metal drawing of a sheet metal part is marked with a reference to this document "dsTPD sheet metal part", this reference is placed above the title block of the dimensioned PDF.

Why dsTPD is introduced:
A STEP file in combination with all necessary requirements (e.g. tolerances) is sufficient to produce the required part. The only difference of dsTPD with traditional TPD is that the tolerances will be specified in this document (chapter 6) and no longer on drawing (dimensioned PDF), except for fits and possible critical tolerances. Designers can design on the basis of the standard tolerance values and will result in serious time reduction and improving part quality.

How to use dsTPD:
To be able to manufacture a part delivered with dsTPD; digital CAD model information is required. As already mentioned in "What is dsTPD" the part specification that the supplier receives also contains the digital model information, more specific a STEP file. This STEP file specifies all nominal part dimensions and is part of the TPD.
Contact your customer when another format (NX, Parasolid, IGES or DXF) is required.

The STEP file is leading in cooperation with the Standard Tolerances (Chapter 6). When the dimensioned PDF specifies dimensions and tolerances, those will be above with respect to the STEP file and this standard tolerance document.
Other specifications such as required material and treatment can also be found on the dimensioned PDF.

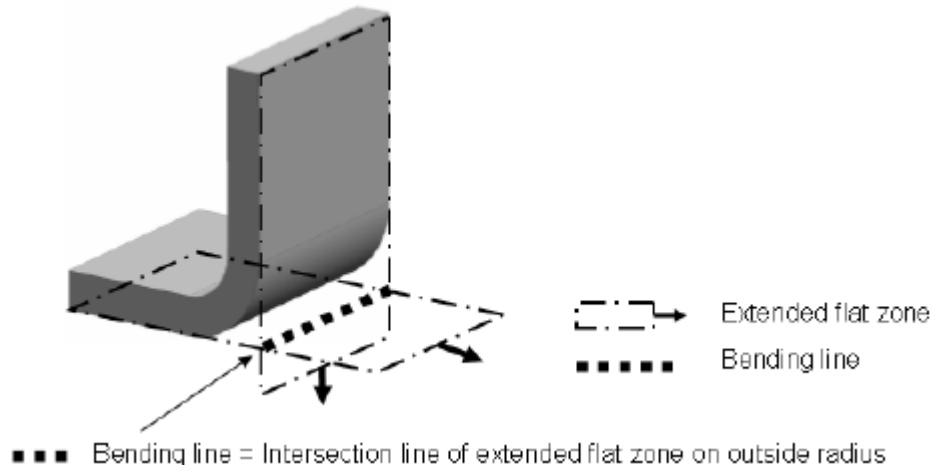
Other important general information:
Always check if the file name of the file with digital information contains the same 12NC as shown in the title block of the drawing.

With respect to geometrical part changes, the 3D model is leading.

3 Abbreviations

Abbreviations	Definition
dsTPD	digital sheet metal TPD
GSA	Generic Standard of ASML
R-inside	Nominal inside bending radius (smallest radius) (see figure 3)
STEP	CAD file format, a representation of the data of a CAD-file.
S	Nominal sheet metal thickness
TED	Theoretical Exact Dimension
TPD	Technical Product Documentation

4 Terms and definitions

Term	Definition
Angular dimension	Dimension on angles, including right angles (90degrees)
Datum	A theoretically exact geometric reference (such as axes, planes, straight lines, etc.) to which toleranced features are related, see ISO 5459 for more details.
Finished part	Part as specified in the TPD, treatments included.
Flat zone	Zone, of a bended sheet metal part, without any influence of a bending. The bending zone is no part of the flat zone.
Sheet metal part	Single part (not welded or brazed) made of sheet metal (except lead), and which gets its shape by such operations as laser cutting, bending, punching and shearing. Sheet thickness ≥ 0,8mm and ≤ 6mm, the surfaces that determine the sheet thickness are not machined/milled. When nuts are permanently mounted on the part, this part can be seen as a sheet metal part.
Bending line	Imaginary intersection line of the extended surfaces of two flat zones adjacent to a bending, the line lies always on the side of the largest bending radius, see figure 1.  Figure 1 – Bending line
Bending zone	Zone at both sides of a bending radius in which material is deformed, deformations are caused by a bending operation, see dark grey zones in figure 2. The bending zone contains the bending radius and two areas each starting from the bending radius. The dimension “bz” (see figure 3) starts from the bending radius and is perpendicular to the bending line.

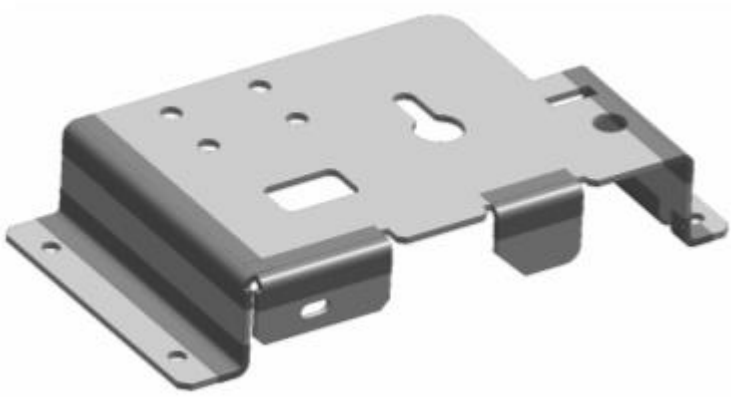
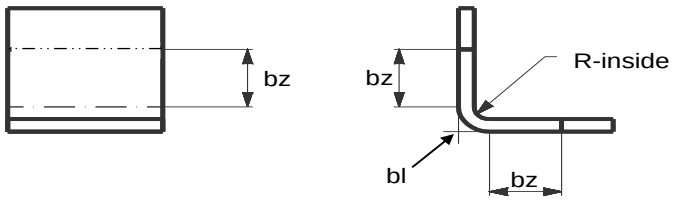


Figure 2 – Bending zones are shown in dark grey



Where:
bz= dimension related to bending zone.
bl= bending line (figure 3 shows axial view on bl, see also figure 1).

Figure 3 – Dimensions bz related to bending zones

5 General subjects and specification of details of a part

5.1 General subjects

- Specifications in the TPD are valid for the finished part.
- The envelope requirement according to ISO 8015 (edition 1985) is valid.
- The drawing could show simplified dimensioning of holes according to ISO 15786 (2008).

5.2 General part specifications

The general part specifications are really important to understand exactly how to manufacture Sheet Metal parts.

- Dimensions of the 3D model (STEP file) are leading. All dimensions of the 3D model are theoretically exact dimensions.

- The drawing shows as less as possible dimensions, for example the theoretically exact dimensions related to the general tolerance specifications are not shown on the drawing. Following dimensions can be derived from the model (STEP):
  - theoretically exact dimensions;
  - dimensions of features in the bending and flat zone;
  - dimensions of a flat zone.

- When a theoretically exact dimension, related to the general profile tolerance in combination with dimensions on the drawing causes a not unambiguous situation, the general profile tolerance on that specific theoretically exact dimension must be ignored.

- The point of measuring must be:
  1. From an edge to a feature/edge/bending.

  When measuring from an edge is not possible (e.g. only bendings):
  2. From a feature to a feature/bending.

  All examples can be found in the Appendixes.

- The 3D model is always modeled in the middle of the tolerance field, except for fits. Fits are always indicated according to ISO 286 part 1 and 2 on the drawing.
  - Example: dimension 6 with tolerance (+/- 0,05) is modeled “6”.
  - Example fit: Dimension 6 with tolerance G7 (+ 0,004; + 0,016) is modeled “6”, on the dimensioned PDF “6G7” is shown.

- Mentioned Standard Tolerances in chapter 6 need to be met.

- Dimensions that indicate the length, width, height and thickness of the part are given on the drawing between parentheses (means “for information only”).

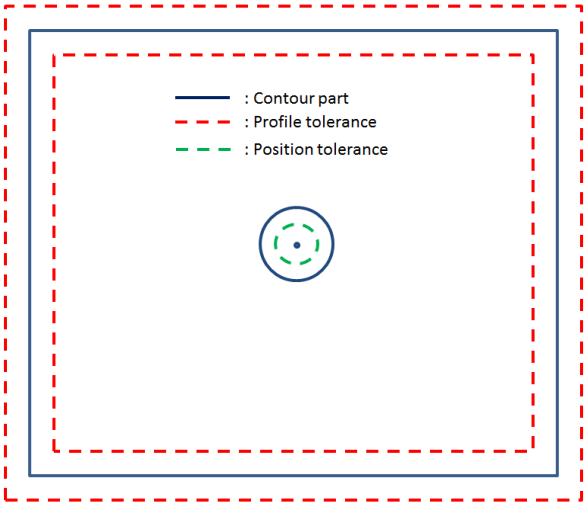
- All external edges must be free from burrs: sharp edges must be broken to avoid contamination and injuries, also marks and oxides must be removed.

- Inside bending radius must be equal to sheet thickness.

6 General specification sheet metal parts

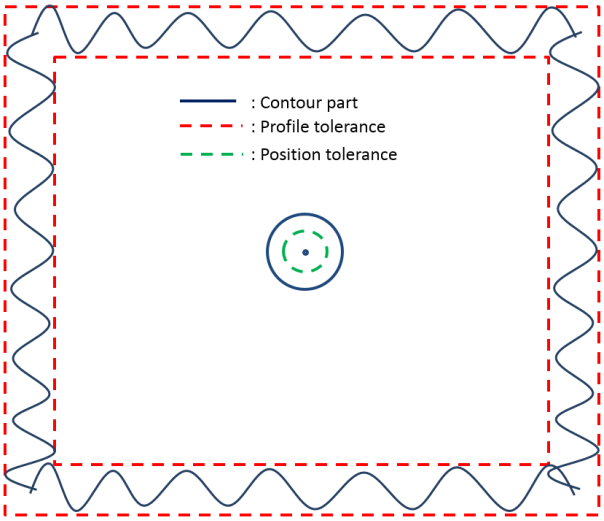
With the general profile and position tolerance a box is created around and inside every flat zone.
These tolerances are applicable for every dimension!

Customers must deliver nominal models, together with acceptable tolerances mentioned in this chapter. An example of an optimal contour can be found in next picture by the blue lines, tolerances on the contour (profile tolerance) are indicated by the red dashed line and a possible deviation of a hole (position tolerance) is indicated by the green dashed line.



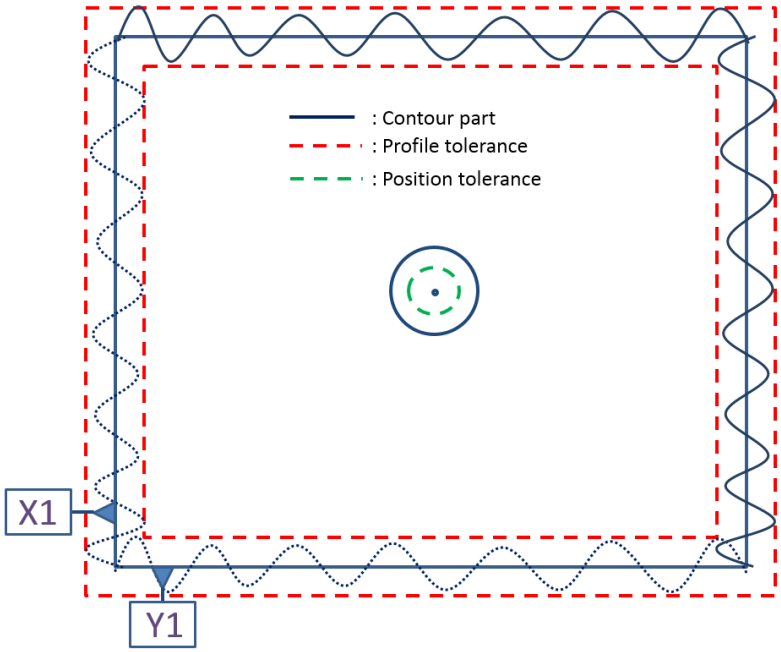
For suppliers it's a utopia to deliver a perfectly made product without any deviation, therefor tolerances are mentioned for every dimension. The perfect straight lines will deviate from their straightness, perpendicularity and parallelism, all these geometrical tolerances are prescribed and must be fulfilled. In next picture an extreme example of what can happen to the contour of the part is shown.

When no reference systems are added by the customer, the supplier is free to define a point for qualification. Only one qualification point can be chosen in each flat zone and cannot be shifted within that flat zone (see 5.2, bullet 4).



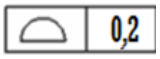
The customer has always the possibility to add reference systems on the drawing/part. In this case the supplier is obligated to use this point as qualification point!

The tolerances for that contour are still aplicable, the deviation of a laser cutting machine will not dissapear, so the accuracy will be the same as for edges without a datum. The measured value after production must be in the tolerance range discribed in this document according to the dimension in the STEP file.

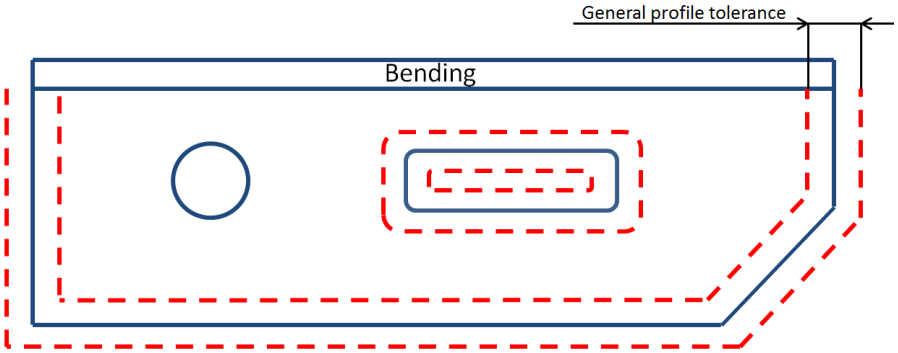


6.1 Tolerances flat zones

6.1.1 General profile tolerance

The general profile tolerance will depend on Table 1**, e.g.** → 

Unless specified otherwise the profile tolerance is related to the max dimension of the part, the total flat zone. The profile tolerance is also applicable to slots etc. and is not applicable to the position and diameter of cylindrical holes.

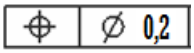


The general profile tolerance depends on the max dimension of the part:

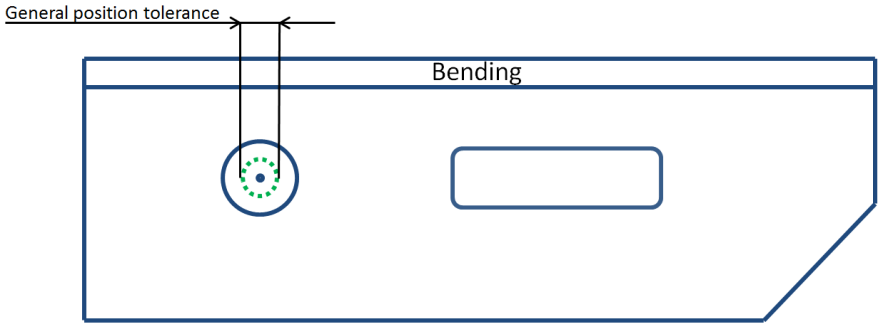
Distance (mm)	General Profile Tolerance (mm)
0 ≤ 500	0.2
500 ≤ 1000	0.4
1000 ≤ 2000	0.6

Table 1: General profile tolerances

6.1.2 General position tolerance of cylindrical holes

The general position tolerance of cylindrical holes will depend on **Table 2, e.g →** 

The general position tolerance is applicable for every circular hole and threaded hole.



The general position tolerance depends on the distance from a feature to the point of measurement:

Distance (mm)	General Position Tolerance (mm)
0 ≤ 500	0.2
500 ≤ 1000	0.4
1000 ≤ 2000	0.6

Table 2: General position tolerances

6.1.3 General tolerance on the diameter of cylindrical holes

The general tolerance on the diameter of cylindrical holes is excluded for threaded holes.

Diameter (mm)	Tolerance (mm)
0 ≤ 50	± 0.1
50 ≤ 500	± 0.2
> 500	± 0.4

Table 3: General tolerances on diameter

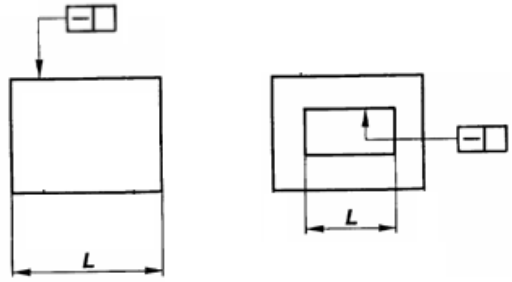
For items in paragraph 6.1.4-6.1.8 the following rule applies: If applicable and unless otherwise stated the largest length is the datum.

Attention: The flatness specification, shown in 6.1.5, is sheet thickness dependent.

6.1.4 Straightness specification

Table 4 – Straightness specification

L ≤ 200	0,1
200 < L ≤ 500	0,15
500 < L ≤ 1000	0,2
1000 < L ≤ 2000	0,4



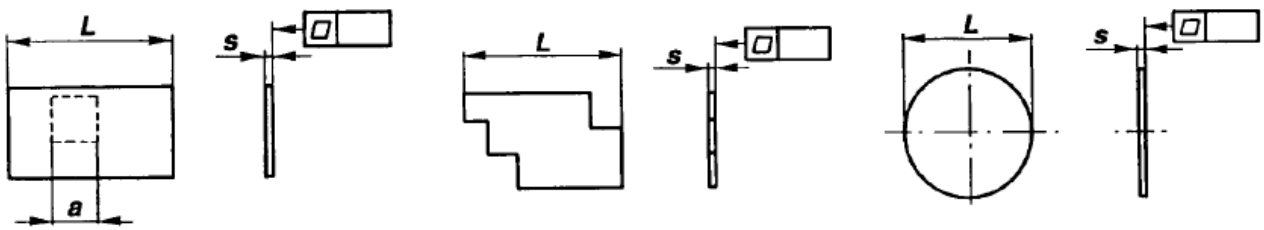
6.1.5 Flatness specification

Flatness tolerances as specified in table 5:
- Includes deformation resulting from bending and welding.
- Are specified for a certain number of s /a – combinations, where "*a*" is the side of each fictitious square, having a maximum value of "L".

Table 5 – Flatness specifications

a / *s*	≤25	>25 ≤50	>50 ≤100	>100 ≤200	>200 ≤400	>400 ≤800
0,8	0,6	0,8	1,2	1,7	2,5	3,5
1	0,5	0,7	1	1,4	2	3
1,2	0,4	0,6	0,8	1,2	1,6	2,5
1,5	0,3	0,4	0,6	1	1,3	2
2	0,25	0,35	0,5	0,7	1	1,5
2,5	0,2	0,25	0,4	0,5	0,8	1
3	0,15	0,2	0,3	0,4	0,8	1

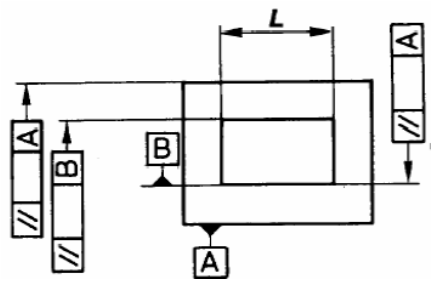
Meaning "a", "L" and "s".



6.1.6 Parallelism specification

Table 6 – Parallelism specification

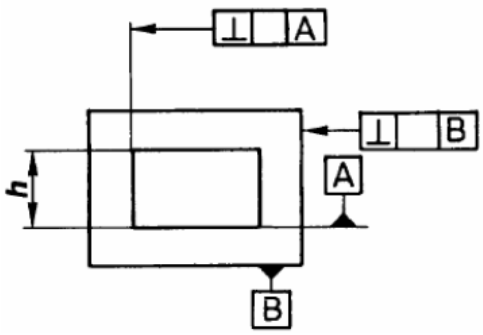
L ≤ 200	0,15
200 < L ≤ 500	0,2
500 < L ≤ 1000	0,3
1000 < L ≤ 2000	0,5



6.1.7 Perpendicularity specification

Table 7 – Perpendicularity specification

h ≤ 200	0,15
200 < h ≤ 500	0,2
500 < h ≤ 1000	0,3
1000 < h ≤ 2000	0,5



6.2 Bending specifications

6.2.1 General tolerance on a bending

Bending dimensions are always indicated on a dimensioned PDF without tolerances. Bending tolerances (multiple bendings) as specified in Table. When the bending tolerance in Table is not sufficient a tolerance will be indicated on the dimensioned PDF.

Very important:
Tolerances mentioned on the dimensioned PDF are always leading!

In table 3 the bending tolerances are specified, this is specified for L in combination with S, where L is the outside bending length (mm) and S the sheet thickness (mm).
When S > 2 → tolerances (Table) must be multiplied with correct Factor (Table) depending on S

	1 bending	2 bendings
L \ S	S ≤ 2	S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 1
1000 < L ≤ 4300	± 1	± 2

Table 8: Tolerances multiple bends

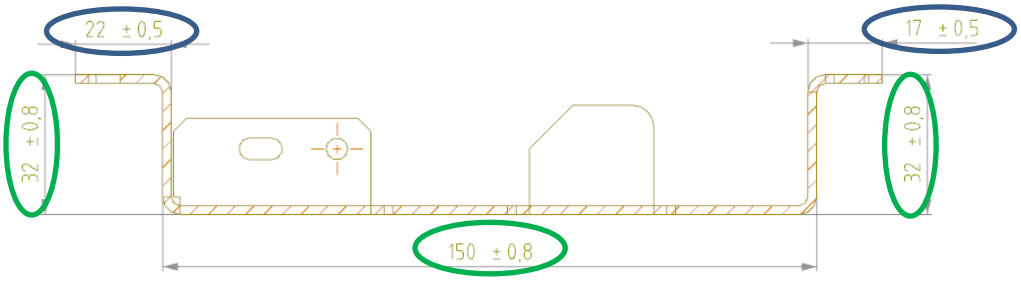
Sheetmetal Thickness S (mm)	Factor
2 < S ≤ 3	1,5
3 < S ≤ 5	2
5 < S ≤ 6	2,5

Table 9: Factor multiply tolerances w.r.t. thickness

6.2.1.1 Example bending specifications (2mm)

L \ S	1 bending S ≤ 2	2 bendings S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 1
1000 < L ≤ 4300	± 1	± 2

6.2.1.2 Example bending specifications with factor (3mm)



L \ S	1 bending S ≤ 2	2 bendings S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 1
1000 < L ≤ 4300	± 1	± 2

Sheetmetal Thickness S (mm)	Factor
2 < S ≤ 3	1.5
3 < S ≤ 5	2
5 < S ≤ 6	2.5

± 0.3 | ± 0.5 **X** 1.5 **=** ± 0.5 | ± 0.8

0.45 ≈ 0.5mm
0.75 ≈ 0.8mm

6.2.2 General tolerance on a bending angle

Unless otherwise stated the general tolerance on bent angle is +/- 1°.
This is not applicable for a hem flange.

6.2.3 Lateral protrusions on a bend

The protrusion, see figure 10, must not exceed 10% of the sheet thickness.

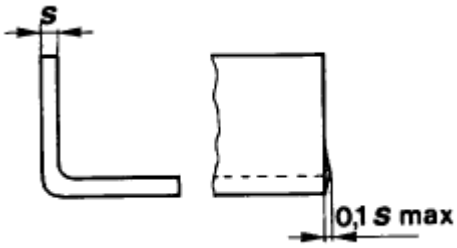


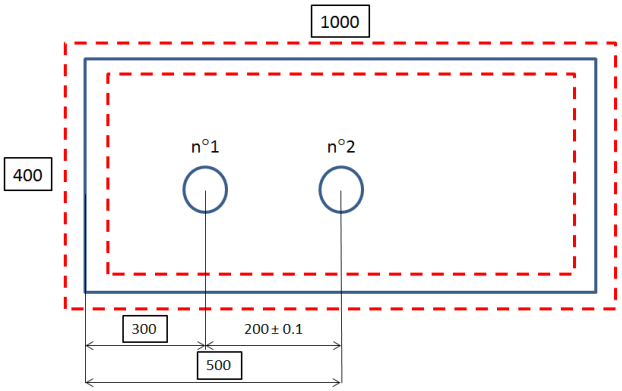
Figure 10 – Specification lateral protrusion

6.2.4 Features in the bending zone

When a feature is designed in the bending zone, the general tolerances are not applicable. The tolerances on these features must be specified by designer on the dimensioned PDF. If tolerances are not specified and it can't be solved by tooling (other knife in bending process to make bending zone shorter), please contact the point of contact of the respective part.

Appendix A: Example flat sheet metal plate

Chapter 6 is necessary to know which tolerances are applicable.



Profile tolerance:
In the STEP-file a length of 1000mm is measured for the part → a Profile tolerance of 0.4.

Distance (mm)	General Profile Tolerance (mm)
≤ 500	0.2
≤ 1000	0.4
≤ 2000	0.6

Position tolerance:
In the STEP-file a length of 300mm is measured to the feature → a Position tolerance of 0.2.
In the STEP-file a length of 500mm is measured to the feature → a Position tolerance of 0.2.

Distance (mm)	General Position Tolerance (mm)
≤ 500	0.2
≤ 1000	0.4
≤ 2000	0.6

Maximum dimension of the part → 1000.4mm
Minimum dimension of the part → 999.6mm

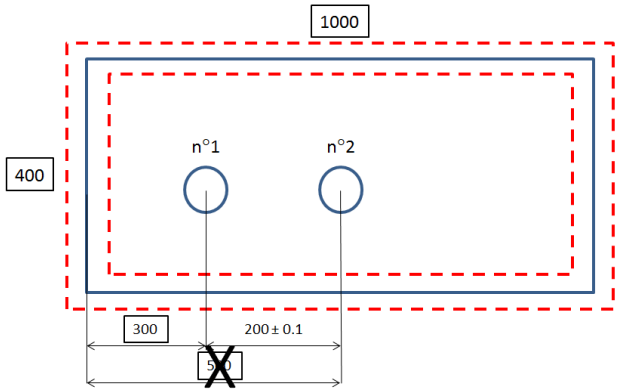
Every hole can be measured from every possible edge, not from a bending. Once an edge is chosen, measure all feature from that specific edge.
Hole n°1:

- TED = 300mm
- Profile tolerance — Max +0.2mm / Min -0.2mm
- Position tolerance — Max +0.1mm / Min -0.1mm

This result in a dimension range: Max 300.3mm / Min 299.7mm

For hole n°2 the same calculation needs to be done, when nothing is specified on the dimensioned PDF. In this case, the distance between hole n°1 and n°2 is specified on the dimensioned PDF by designer, with 200 ± 0.1.

Specifications on the dimensioned PDF always overrule TED dimensions in the STEP-file. Because a pitch is specified on the dimensioned PDF the TED of 500mm must be ignored, otherwise the part will be ambiguous.

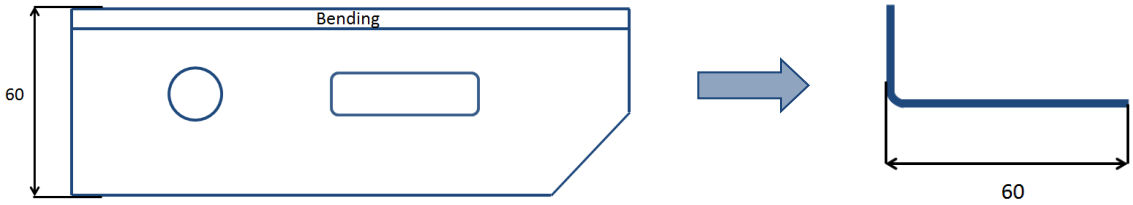


Every tolerance of a dimension needs to be respected, the supplier is free to work within the tolerance range but every dimension has a tolerance that can be controlled.

➔ Sum of distance between features and features to the edge must fit in the total possible length of the part.

Appendix B: Example bended sheet metal

Chapter 6 is necessary to know which tolerances are applicable.



The length of the bending will be always specified on TPD by the designer.
The tolerance for the bending can be found in chapter 6.2.

On TPD a length of 60mm and a thickness of 1.25mm is indicated → a tolerance of ±0.3.

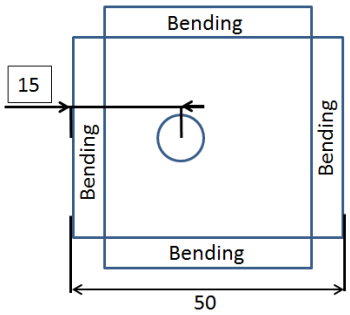
L \ S	1 bending S ≤ 2	2 bendings S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 0.5
1000 < L ≤ 4300	± 1	± 2

Because the dimension is indicated on TPD by designer, the profile tolerance may not be included. Fom the edge to the bending, indicated with a dimension, only the tolerance in chapter 6.2 is valid.

The features within the sheet metal plate must be always measured to an edge (see Appendix A). When this isn't possible, the feature must be measured to a bending (see Appendix C).

Appendix C: Example feature measured to bending

Chapter 6 is necessary to know which tolerances are applicable.



The bendings:
The length of the bending will be always specified on TPD by the designer.
The tolerance for the bending can be found in chapter 6.2.

On TPD a length of 50mm and a thickness of 1.25mm is indicated → a tolerance of ±0.5.

L \ S	1 bending S ≤ 2	2 bendings S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 0.5
1000 < L ≤ 4300	± 1	± 2

The feature:
Position tolerance:
In the STEP-file a length of 15mm is measured to the feature → a Position tolerance of 0.2.

Distance (mm)	General Position Tolerance (mm)
≤ 500	0.2
≤ 1000	0.4
≤ 2000	0.6

In the STEP-file a length of 15mm is measured from a bending → a tolerance of ±0.3.

L \ S	1 bending S ≤ 2	2 bendings S ≤ 2
L ≤ 300	± 0.3	± 0.5
300 < L ≤ 1000	± 0.5	± 0.5
1000 < L ≤ 4300	± 1	± 2

Feature:

- TED = 15mm
- Tolerance bending — Max +0.3mm / Min -0.3mm
- Position tolerance — Max +0.1mm / Min -0.1mm

This result in a dimension range: Max 15.4mm / Min 14.6mm

Every tolerance of a dimension needs to be respected, the supplier is free to work within the tolerance range but every dimension has a tolerance that can be controlled.

7 References to documents

Title	ASML standard	(Inter)national standards
Geometrical product specification (GPS) – ISO coding system for tolerances of linear sizes – Part 1:Bases for tolerances and fits	-	ISO 286 part 1 (1989)
Geometrical product specification (GPS) – ISO coding system for tolerances of linear sizes – Part 2:Tables of standard tolerances grades and limit deviations for holes and shafts.	-	ISO 286 part 2 (1989)
Fundamental tolerancing principle	-	ISO 8015 (1985)
ISO general purpose metric screw threads - Tolerances	-	ISO 965 part 1,2 and3 (1980)
Form of orientation, location, definitions, symbols on drawing.	-	ISO 1101 (2004)
Technical drawings - Simplified representation and dimensioning of holes	-	ISO 15786 (first edition, 2008-10-01)
Technical drawings – Geometrical tolerancing – Datums and datum systems for geometrical tolerances.	-	ISO 5459 (1981)

8 Document change & history record

Name	Last update	Status	Version	Comment
JJPG	2016-02-09	For review	01	Ready for review