PolTolerances

Tolerances for plastic moulded parts according to DIN ISO 20457 / DIN 16742

User Manual

General, position and profile shape tolerances

Described using PolTolerances 3.5.6.1 · As of: October 2026

Makrolar GbR Berlin

Contents

Contents 2
1 Introduction 3
2 User interface 3
3 Determining the tolerance group 4
3.1 Point evaluation 5
3.2 Assignment help(s) 5
4 Tolerancing dimensions 6
4.1 Length dimensions (size dimensions) 6
4.2 Direct position tolerance 9
4.3 Profile shape deviations 9
4.4 Determining a realizable tolerance series 10
4.5 Realizability of functional dimensions 11
5 Managing and printing projects 12
5.1 “Project management” tab 12
5.2 Printing / Print Preview 13
6 Reference 13
6.1 Tolerance values tables 13
6.2 Processing shrinkage ranges 14
7 Choosing the standard: DIN ISO 20457 or DIN 16742 15
8 Technical background 16
8.1 Tolerancing principles and recommendations 16
8.2 Scope of the standard 16
8.3 Processing shrinkage 17
8.4 Stiffness and hardness level of the materials 18
8.5 Classifying the tolerance groups 18
8.6 Mould-related part dimensions 19
8.7 Nominal dimension definitions 19
8.8 Dimensional reference levels and acceptance conditions 20
8.9 Tolerance series 21
9 Menus 22
10 Tips and frequently asked questions 22

1 Introduction

PolTolerances determines general tolerances for plastic moulded parts according to **DIN ISO 20457** or **DIN 16742**. From the manufacturing process, the moulding compound and its shrinkage behaviour, the program derives the tolerance group (TG) of the part. For every dimension you enter, it then shows the possible production deviation and whether it has to be agreed with the manufacturer of the moulded part.

Besides length (size) dimensions, PolTolerances covers position tolerances with cylindrical tolerance zones and profile shape tolerances. It also helps you choose a realizable tolerance series and checks under which conditions a required functional dimension can be achieved. You save the inputs for a moulded part as a project and print them as a report.

It is important for manufacturers and buyers of plastic moulded parts to agree on sensible dimensional tolerances. Excessive accuracy requirements cause high costs for moulds, production and quality assurance. Dimensional, form and position deviations of plastic moulded parts cannot be compared with those of metal parts, because the properties of the materials differ considerably in production and use. DIN ISO 20457 and DIN 16742 help to take the technically feasible accuracies into account as early as the part development stage. Chapter 8 summarizes the technical background.

> **Note:** The **“Commercial products plus”** tab on the left edge of the window is not described in this manual. It has a manual of its own (“Commercial Products – User manual”).
>
> **DEMO version:** The DEMO version only supports nominal dimensions in the range 5 … 55 mm. The program shows a corresponding message in the work area.

2 User interface

After start-up, PolTolerances opens the **“Edit project / moulding”** tab. On the left you describe the moulded part; on the right you enter the dimensions.

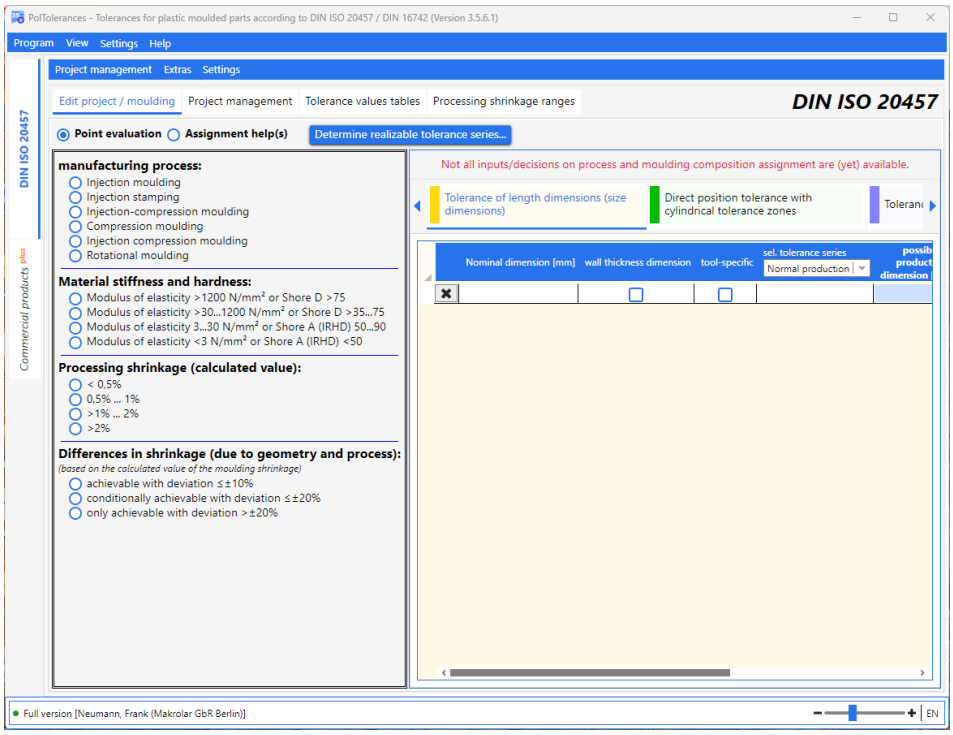


Figure 1: PolTolerances after start-up

Element	Purpose
Menu bar	**“Program”**, **“View”**, **“Settings”** and **“Help”** (chapter 9).
“DIN ISO 20457” tab (left, vertical)	Work area for determining tolerances. The label shows the selected standard (DIN ISO 20457 or DIN 16742).
“Commercial products plus” tab (left, vertical)	Database of commercial plastics, see its own manual.
Blue bar **“Project management · Extras · Settings”**	Project commands, additional tools and the choice of standard (sections 4.4, 4.5, chapters 5 and 7).
“Edit project / moulding” tab	Determine the tolerance group and tolerance dimensions (chapters 3 and 4).
“Project management” tab	Create, load, save and print projects (chapter 5).
“Tolerance values tables” tab	The tolerance tables of the standard for reference (section 6.1).
“Processing shrinkage ranges” tab	Shrinkage values of many moulding compounds (section 6.2).
Status bar	Licence status (e.g. “Full version”), zoom slider for the display size and the selected language (“EN”).

3 Determining the tolerance group

The tolerance group (TG) is the basis of all general tolerances. PolTolerances offers two ways to determine it. You choose the method at the top of the **“Edit project / moulding”** tab:

- **“Point evaluation”** – you rate manufacturing process, material stiffness, processing shrinkage and differences in shrinkage individually (section 3.1).
- **“Assignment help(s)”** – you select the moulding compound group and answer a few questions about the material (section 3.2).

As long as information is missing, the work area shows in red: **“Not all inputs/decisions on process and moulding composition assignment are (yet) available.”** Once everything is complete, this is replaced by **“General tolerances for length dimensions: Tolerance Group TG …”**.

3.1 Point evaluation

1. Select **“Point evaluation”**.
2. Under **“manufacturing process”**, select the process, e.g. **“Injection moulding”**.
3. Under **“Material stiffness and hardness”**, select the matching modulus of elasticity or hardness range.
4. Under **“Processing shrinkage (calculated value)”**, select the shrinkage range of the moulding compound. If needed, look the values up in the “Processing shrinkage ranges” tab (section 6.2).
5. Under **“Differences in shrinkage (due to geometry and process)”**, select how precisely the shrinkage can be predicted.

The tolerance group appears immediately at the top right. Hover over the **“i”** symbol next to it to see the individual point scores.

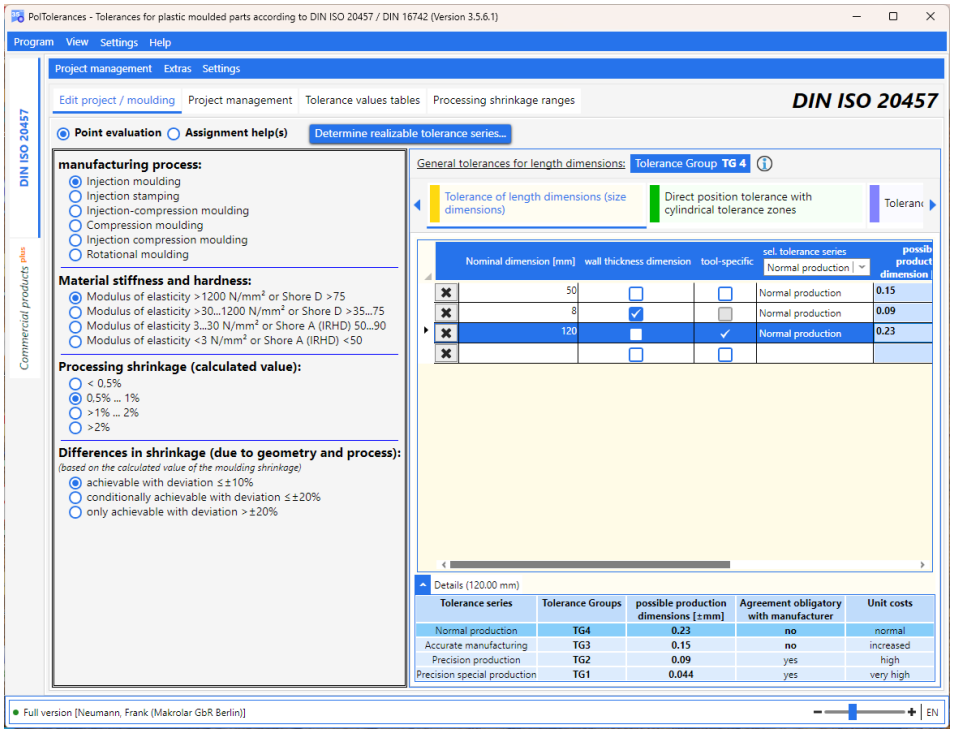


Figure 2: Point evaluation complete: tolerance group TG 4, with three length dimensions entered below

> **Note:** Guidance on rating the individual criteria – material stiffness, calculated value of the processing shrinkage and shrinkage variations (precise, conditionally precise or only imprecise) – can be found in sections 8.3 to 8.5.

3.2 Assignment help(s)

The assignment help is useful when you know the moulding compound but not the individual evaluation criteria.

1. Select **“Assignment help(s)”**.

2. Under **“moulding compound groups”**, select the group of your compound, e.g. **“Semi-crystalline thermoplastics”**. Below each group name you find examples of the plastics it contains (e.g. PE, PP, PA6, POM).
3. In the lower area PolTolerances shows the matching manufacturing process and asks about fillers and reinforcing materials, modulus of elasticity, processing shrinkage and differences in shrinkage. Answer every question.

The check boxes in the lower area have three states: a filled box means “not decided yet”. Click the box until it shows the state you want (empty = no, tick = yes). The tolerance group only appears once all questions are decided.

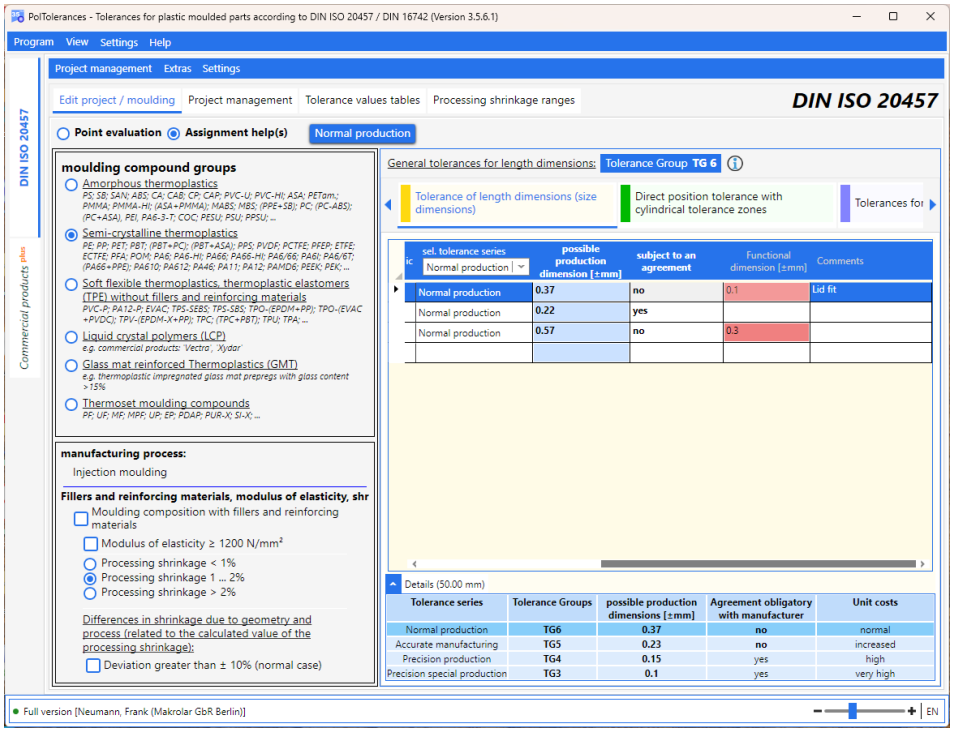


Figure 3: Assignment help for a semi-crystalline thermoplastic without fillers: tolerance group TG 6

4 Tolerancing dimensions

On the right of the **“Edit project / moulding”** tab there are three sub-tabs. If not all tab headers are visible, use the blue arrows to the left and right of the tabs.

Sub-tab	Content
“Tolerance of length dimensions (size dimensions)” (yellow)	General tolerances for length dimensions (section 4.1).
“Direct position tolerance with cylindrical tolerance zones” (green)	Position tolerances as the diameter of the tolerance zone (section 4.2).
“Tolerances for profile shape deviations” (violet)	Surface and line shape tolerances (section 4.3).

4.1 Length dimensions (size dimensions)

1. Double-click the **“Nominal dimension [mm]”** cell, enter the dimension and press Enter. A new empty row for the next dimension appears automatically below.

2. Tick **“wall thickness dimension”** if the dimension is a wall thickness, or **“tool-specific”** if the dimension is formed by one mould half only. For a wall thickness dimension, “tool-specific” cannot be selected.
3. Read the achievable tolerance in **“possible production dimension [±mm]”** and whether it has to be agreed with the manufacturer in **“subject to an agreement”**.
4. Optionally, enter a **“Functional dimension [±mm]”**, i.e. the tolerance the part needs to function, and a **“Comment”**.

Sections 8.6 and 8.7 explain what tool-specific dimensions are and which nominal dimensions are subject to agreement.

The table is wider than the work area. Use the scroll bar below the table to reach the right-hand columns. The **“✕”** button at the start of a row deletes the dimension.

Column	Meaning
Nominal dimension [mm]	The nominal dimension from the drawing.
wall thickness dimension	Marks wall thicknesses.
tool-specific	Marks tool-specific dimensions (TS); otherwise the dimension counts as non-tool-specific (NTS).
sel. tolerance series	Selected tolerance series: Normal production, Accurate manufacturing, Precision production or Precision special production.
possible production dimension [±mm]	Achievable deviation in the selected tolerance series.
subject to an agreement	“yes”: the deviation has to be agreed with the manufacturer of the moulded part.
Functional dimension [±mm]	Optional. **Green** background: achievable. **Red** background: not achievable in the selected tolerance series.
Comments	Free text, e.g. the function of the dimension.

Below the table, the **“Details (… mm)”** area shows all four tolerance series for the selected dimension, with tolerance group, possible production deviation, whether an agreement with the manufacturer is obligatory, and relative unit costs.

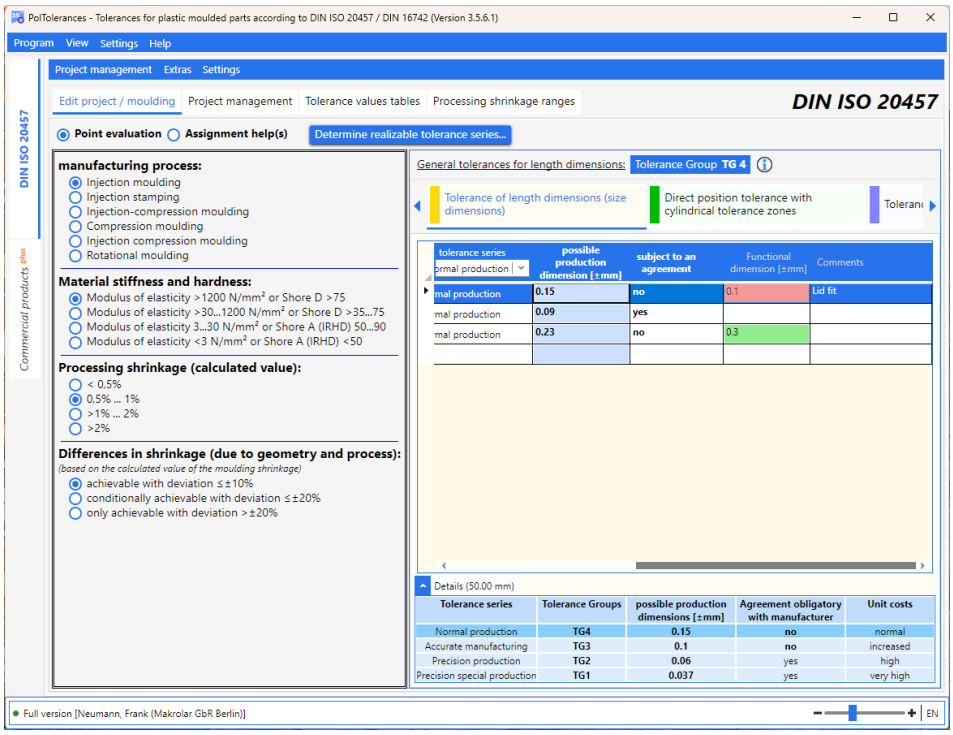


Figure 4: A functional dimension of ±0.1 mm at 50 mm is not achievable in normal production (red); the details show ±0.1 mm in accurate manufacturing

If a functional dimension is red, the details show which tolerance series achieves it. Use the drop-down list in the **“sel. tolerance series”** column to choose a different tolerance series for this dimension. Bear in mind that tighter tolerance series cause higher unit costs and may require an agreement (column “Agreement obligatory with manufacturer” in the details).

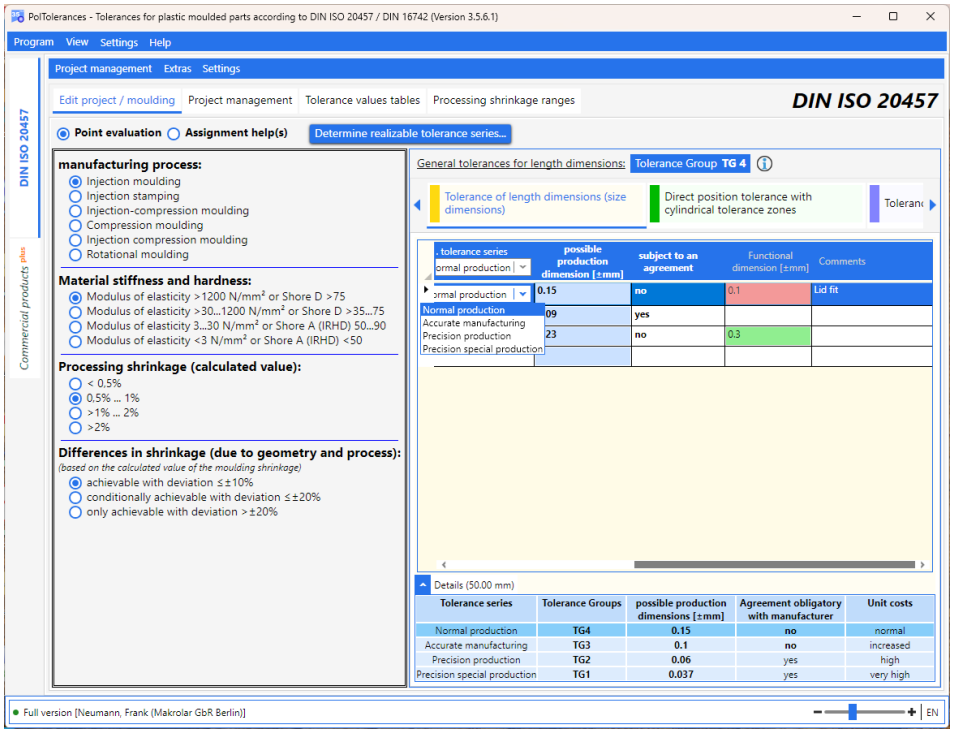


Figure 5: Selecting the tolerance series for a single dimension

4.2 Direct position tolerance

Here you determine position tolerances with a cylindrical tolerance zone, e.g. for the position of holes. The program points out that **directly tolerated position tolerances are to be agreed with the manufacturer of the moulded part**.

1. Open the **“Direct position tolerance with cylindrical tolerance zones”** sub-tab.
2. Double-click **“Nominal distance [mm]”** and enter the distance. Tick **“tool-specific”** if applicable.
3. Read the achievable diameter of the tolerance zone in **“possible tolerance Ø [mm]”**.
4. Optionally, enter the required diameter in **“necessary Tol-Ø [mm]”**. As with length dimensions, green means “achievable” and red “not achievable”.

Here too, the **“Details”** area shows all tolerance series for the selected distance.

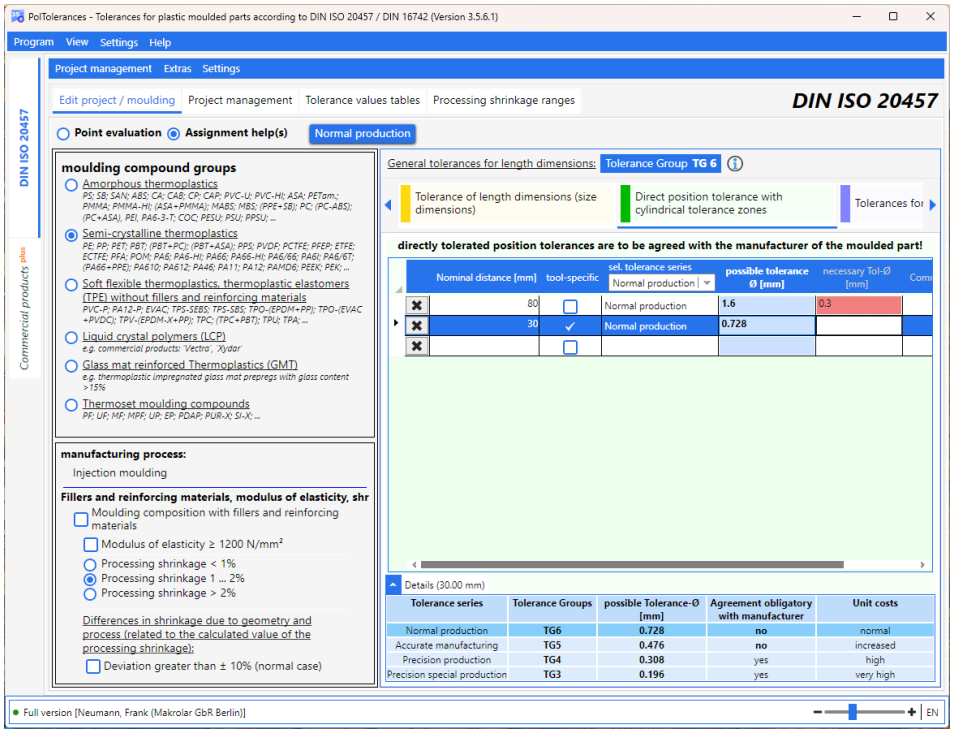


Figure 6: Position tolerances: the required Ø 0.3 mm at a distance of 80 mm is not achievable

4.3 Profile shape deviations

Here you determine tolerances for surface and line shapes. **Profile shape tolerances are also to be agreed with the manufacturer of the moulded part**.

1. Open the **“Tolerances for profile shape deviations”** sub-tab.
2. Double-click **“Nominal distance [mm]”** and enter the dimension. The general tolerance appears in **“tolerance value [mm]”**.
3. Choose whether it is a **“surface shape tolerance”** or a **“line shape tolerance”** and add a comment if needed.

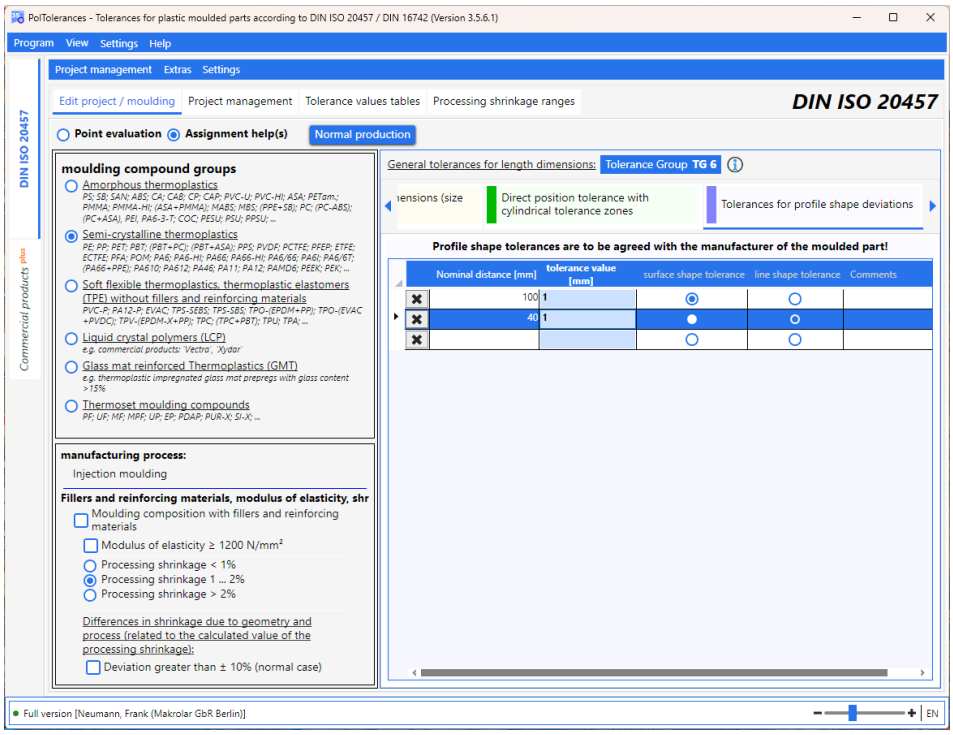


Figure 7: Profile shape tolerances for two nominal distances

4.4 Determining a realizable tolerance series

Which tolerance series a manufacturer can achieve depends on design, material, machines, personnel and quality assurance. The **“Realizable tolerance series”** checklist brings these conditions together.

1. Click **“Determine realizable tolerance series…”** (at the top of the tab) or choose **“Extras” › “Determine realizable tolerance series…”**.
2. Answer the questions in the eight sections: 1. Moulded part construction, 2. Moulding compound and recycled material usage, 3. Production conditions, 4. Personnel for machine operation, 5. Inserts and handling, 6. Tools, 7. Quality assurance, 8. Packing and transport. In each block with a red frame, select exactly one option.
3. As soon as all information is available, the dialog shows the **“Result”** at the top, e.g. “Normal production”.
4. Close the dialog. **“Reset inputs”** lets you start again.

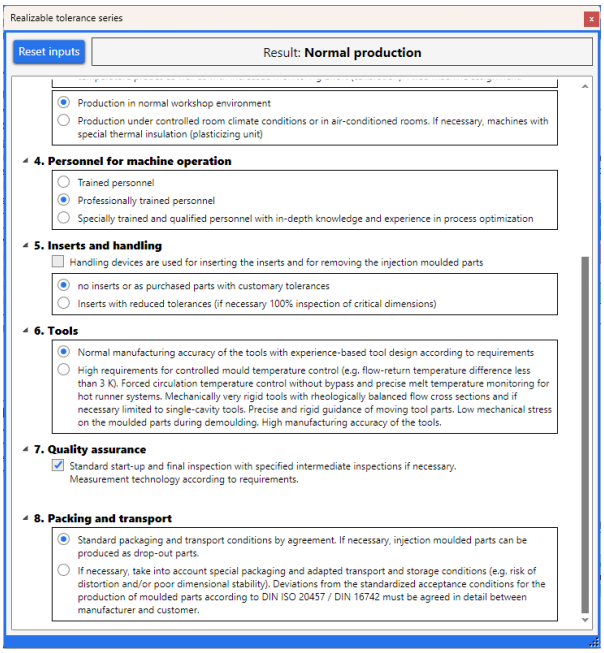


Figure 8: Checklist with the result "Normal production"

Section 8.9 describes the meaning of the four tolerance series and the prerequisites for accurate and precision production. The checklist takes the selection criteria for thermoplastic injection moulding from Annex D of the standard into account by means of an extended selection algorithm.

The result is then shown on the button at the top of the tab and is included in the project printout.

4.5 Realizability of functional dimensions

This tool answers the reverse question: **under which conditions** (manufacturing process, material stiffness, processing shrinkage, consideration of shrinkage differences and tolerance series) **can a required functional dimension be achieved?**

1. Choose **"Extras" › "Realizability of functional dimensions…"**.
2. Select the **"for linear dimensions"** or **"for position distance dimensions"** tab.
3. Enter **"Nominal dimension [mm]"** and **"necessary functional dimension [±mm]"**, and specify whether it is a **"wall thickness dimension"** or a **"Tool-specific dimension"**.
4. Click **"Find/Refresh"**. The table lists all combinations that achieve the functional dimension.
5. Narrow the list down at the top using **"manufacturing process"**, **"Material stiffness"** and **"Processing shrinkage"**. **"Reset filters"** removes the restriction.

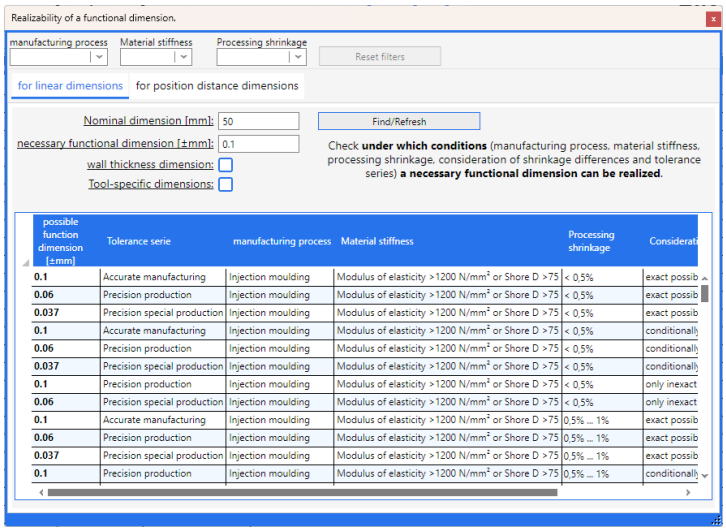


Figure 9: Conditions under which ±0.1 mm can be achieved at 50 mm

5 Managing and printing projects

5.1 “Project management” tab

A project contains all inputs for one moulded part: the tolerance group, the dimensions in all three sub-tabs and the checklist for the realizable tolerance series.

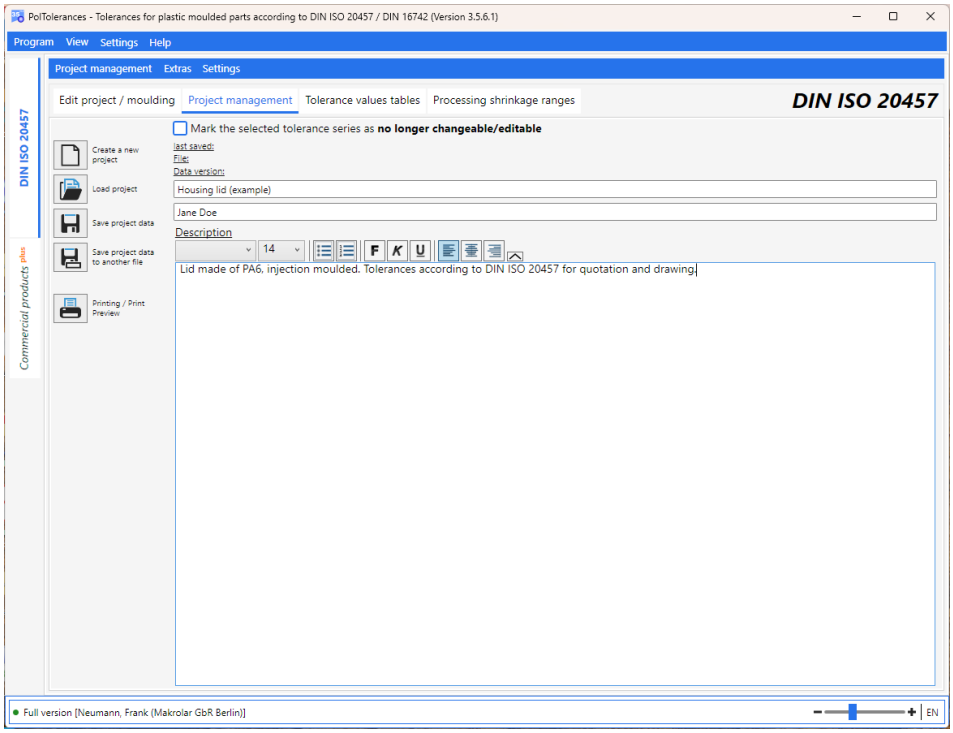


Figure 10: “Project management” tab with project name, processor and description

Button / field	Purpose
Create a new project	Starts an empty project.
Load project	Opens a saved project.
Save project data	Saves the project to the current file.
Save project data to another file	Saves the project under a new file name.

Button / field	Purpose
Printing / Print Preview	Opens the print preview of the project (section 5.2).
last saved / File / Data version	Information about the loaded project file.
Project name, Processor	Name of the moulded part or project and of the person working on it.
Description	Free text with simple formatting (font size, lists, bold, italic, underline, alignment).
Mark the selected tolerance series as no longer changeable/editable	Marks the selected tolerance series as no longer changeable.

The same commands are available in the blue bar under **“Project management”**.

5.2 Printing / Print Preview

Click **“Printing / Print Preview”**. The preview shows the project report: header data, the standard used, the assignment method with all inputs, the tables of length dimensions, position tolerances and profile shape deviations, and the checklist for the realizable tolerance series with its result.

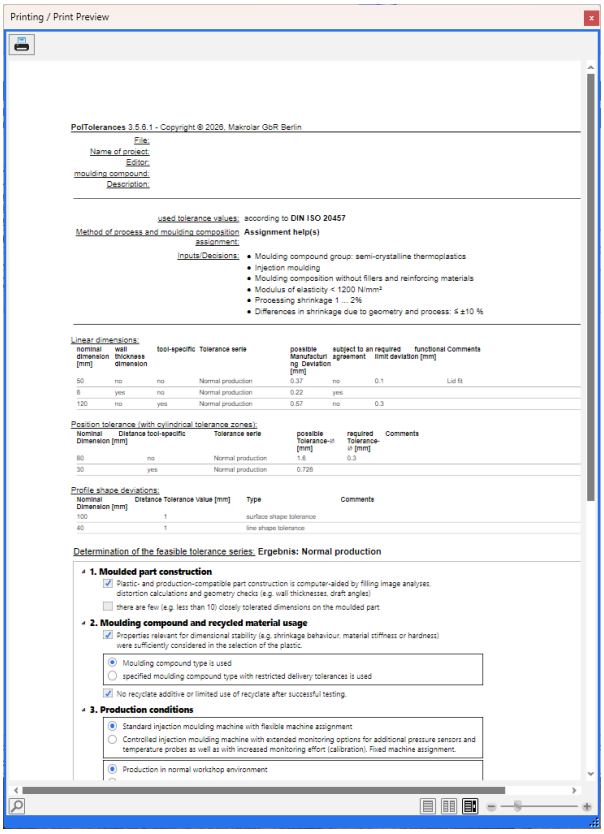


Figure 11: Print preview of a project

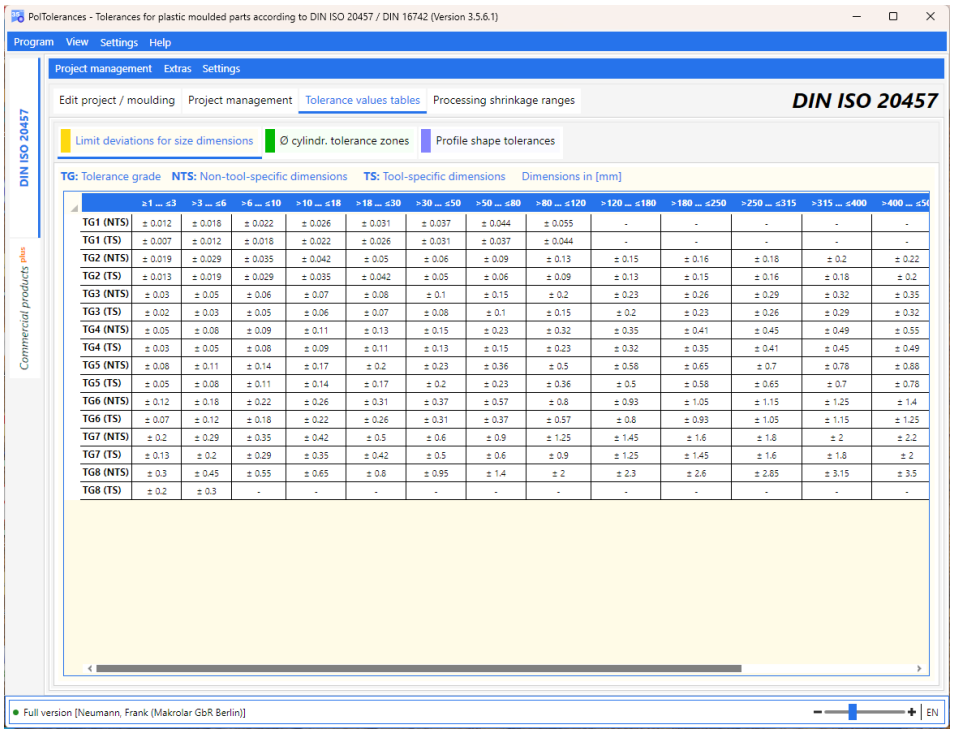
Use the printer symbol at the top left to print the report. At the bottom right you change the page view and the magnification.

6 Reference

6.1 Tolerance values tables

The **“Tolerance values tables”** tab shows the tables of the selected standard in three sub-tabs:

- **“Limit deviations for size dimensions”** – limit deviations per tolerance group, separately for non-tool-specific (NTS) and tool-specific (TS) dimensions, across the nominal size ranges.
- **“Ø cylindr. tolerance zones”** – diameters of the tolerance zones for position tolerances.
- **“Profile shape tolerances”** – general tolerances for profile shape deviations depending on the nominal dimension.



	≥1 … ≤3	>3 … ≤6	>6 … ≤10	>10 … ≤18	>18 … ≤30	>30 … ≤50	>50 … ≤80	>80 … ≤120	>120 … ≤180	>180 … ≤250	>250 … ≤315	>315 … ≤400	>400 … ≤5
TG1 (NTS)	± 0.012	± 0.018	± 0.022	± 0.026	± 0.031	± 0.037	± 0.044	± 0.055	-	-	-	-	-
TG1 (TS)	± 0.007	± 0.012	± 0.018	± 0.022	± 0.026	± 0.031	± 0.037	± 0.044	-	-	-	-	-
TG2 (NTS)	± 0.019	± 0.029	± 0.035	± 0.042	± 0.05	± 0.06	± 0.09	± 0.13	± 0.15	± 0.16	± 0.18	± 0.2	± 0.22
TG2 (TS)	± 0.013	± 0.019	± 0.029	± 0.035	± 0.042	± 0.05	± 0.06	± 0.09	± 0.13	± 0.15	± 0.16	± 0.18	± 0.2
TG3 (NTS)	± 0.03	± 0.05	± 0.06	± 0.07	± 0.08	± 0.1	± 0.15	± 0.2	± 0.23	± 0.26	± 0.29	± 0.32	± 0.35
TG3 (TS)	± 0.02	± 0.03	± 0.05	± 0.06	± 0.07	± 0.08	± 0.1	± 0.15	± 0.2	± 0.23	± 0.26	± 0.29	± 0.32
TG4 (NTS)	± 0.05	± 0.08	± 0.09	± 0.11	± 0.13	± 0.15	± 0.23	± 0.32	± 0.35	± 0.41	± 0.45	± 0.49	± 0.55
TG4 (TS)	± 0.03	± 0.05	± 0.08	± 0.09	± 0.11	± 0.13	± 0.15	± 0.23	± 0.32	± 0.35	± 0.41	± 0.45	± 0.49
TG5 (NTS)	± 0.08	± 0.11	± 0.14	± 0.17	± 0.2	± 0.23	± 0.36	± 0.5	± 0.58	± 0.65	± 0.7	± 0.78	± 0.88
TG5 (TS)	± 0.05	± 0.08	± 0.11	± 0.14	± 0.17	± 0.2	± 0.23	± 0.36	± 0.5	± 0.58	± 0.65	± 0.7	± 0.78
TG6 (NTS)	± 0.12	± 0.18	± 0.22	± 0.26	± 0.31	± 0.37	± 0.57	± 0.8	± 0.93	± 1.05	± 1.15	± 1.25	± 1.4
TG6 (TS)	± 0.07	± 0.12	± 0.18	± 0.22	± 0.26	± 0.31	± 0.37	± 0.57	± 0.8	± 0.93	± 1.05	± 1.15	± 1.25
TG7 (NTS)	± 0.2	± 0.29	± 0.35	± 0.42	± 0.5	± 0.6	± 0.9	± 1.25	± 1.45	± 1.6	± 1.8	± 2	± 2.2
TG7 (TS)	± 0.13	± 0.2	± 0.29	± 0.35	± 0.42	± 0.5	± 0.6	± 0.9	± 1.25	± 1.45	± 1.6	± 1.8	± 2
TG8 (NTS)	± 0.3	± 0.45	± 0.55	± 0.65	± 0.8	± 0.95	± 1.4	± 2	± 2.3	± 2.6	± 2.85	± 3.15	± 3.5
TG8 (TS)	± 0.2	± 0.3	-	-	-	-	-	-	-	-	-	-	-

Figure 12: Limit deviations for size dimensions (dimensions in mm)

6.2 Processing shrinkage ranges

The **“Processing shrinkage ranges”** tab lists the processing shrinkage of many moulding compounds for each manufacturing process. It helps with the point evaluation (section 3.1).

1. Enter part of the abbreviation in the search field at the top left, e.g. **“PA6”**, and press Enter.
2. If needed, restrict the list using **“manufacturing process”**.

Notice: Not all combinations in the list with fillers and reinforcing materials are practically available.

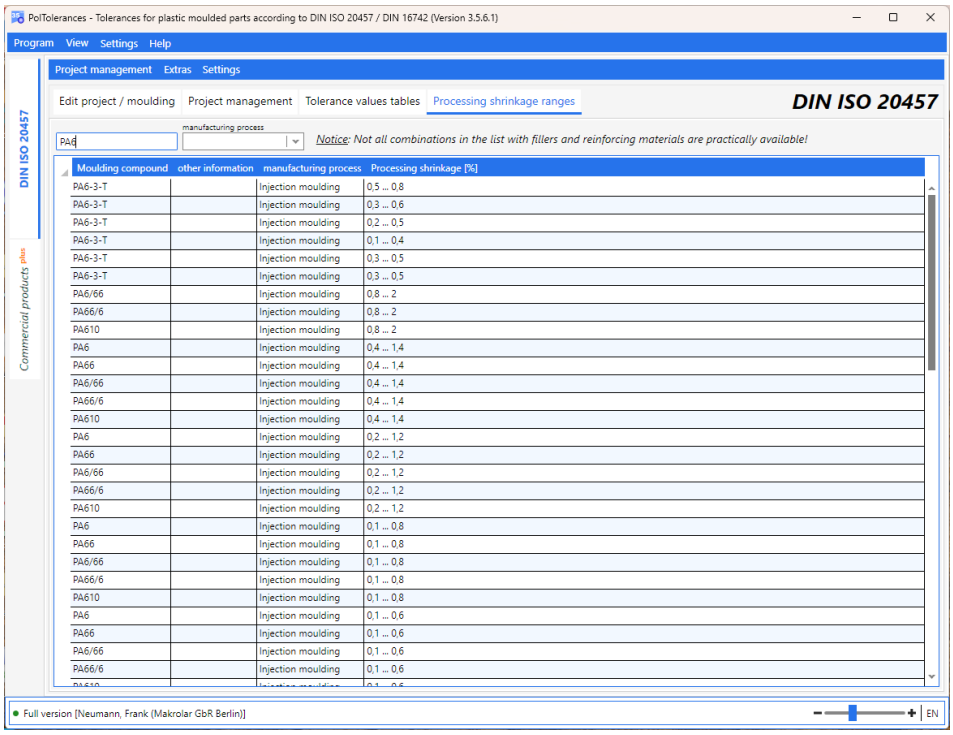


Figure 13: Processing shrinkage of moulding compounds with "PA6" in their abbreviation

7 Choosing the standard: DIN ISO 20457 or DIN 16742

PolTolerances calculates according to either DIN ISO 20457 or DIN 16742. In the blue bar, choose **"Settings"** and then **"Tolerance values according to DIN ISO 20457"** or **"Tolerance values according to DIN 16742"**. The selected standard is ticked and is shown at the top right of the work area and on the vertical tab on the left edge of the window.

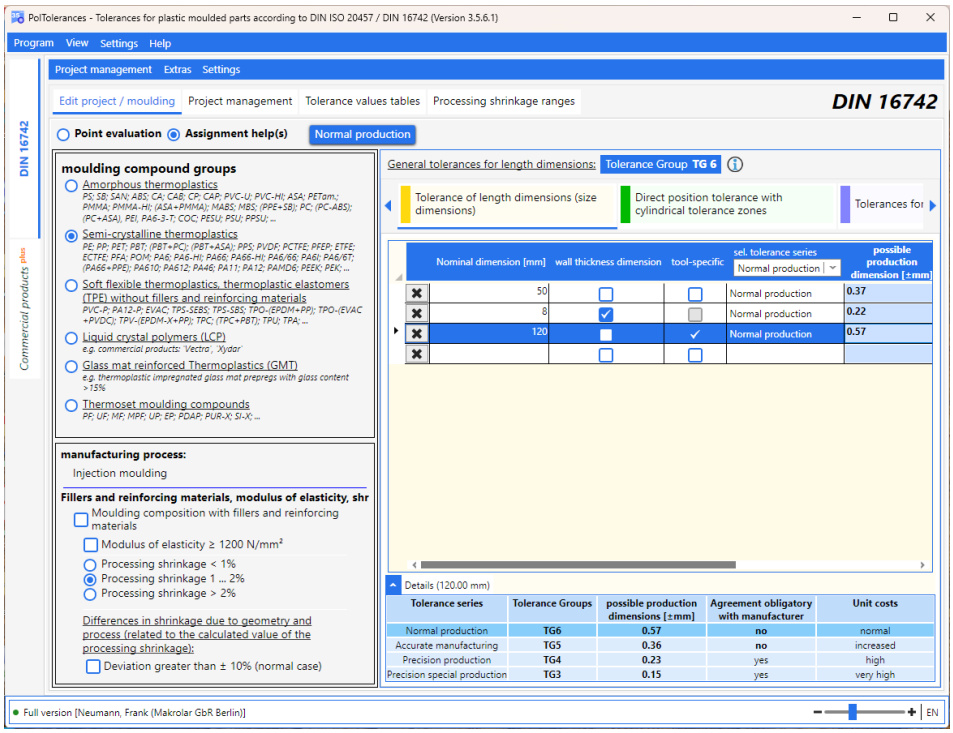


Figure 14: Work area with tolerance values according to DIN 16742

Your inputs are kept when you switch. The standard also appears in the printout under “used tolerance values”.

8 Technical background

This chapter summarizes the standards-related and technical background that is important for classifying the parts and assessing the results. The information refers to DIN ISO 20457 and DIN 16742; the text of the standard always prevails.

8.1 Tolerancing principles and recommendations

- The **independency principle** according to DIN EN ISO 8015 applies. Deviations from it (e.g. the envelope requirement) must be agreed between the contracting parties.
- Part drawings and CAD data sets correspond to the **nominal geometry**. Tolerances are specified symmetrically to the nominal geometry as limit deviations. Asymmetrical tolerances on size dimensions (e.g. fit dimensions) are converted into a symmetrical position by changing the nominal dimension to the mid-tolerance dimension, e.g. 100 −0.6 → 99.7 ± 0.3.
- The verification of the part for quality assurance must be defined unambiguously. The measuring concept is particularly important for non-rigid parts (see DIN ISO 10579).
- Unless otherwise agreed, parts must not be rejected automatically if the general tolerances are not met, provided that their function is not impaired.
- Differences in inclined dimensions caused by **draft angles** specified in the design are not part of the dimensional tolerances or of the form and position deviations. For functional dimensions on inclined surfaces, measuring points must be specified to define two-point dimensions.
- To specify **radii**, at least 90° of the circular segment must be present as a measurable contour; alternatively, radii can be toleranced with profile shape tolerances.
- **Free-form surfaces** must be specified with a profile shape tolerance; the verification must be agreed.
- Directly toleranced **angles and edges** are subject to agreement. Angles and edges that are not directly toleranced are disregarded during verification.
- **Flash** and **mould offset** must be agreed between buyer and manufacturer with regard to the position of the parting lines.
- The usual methods of **tolerance analysis of dimension chains** assume a rigid body. For plastic moulded parts, this condition is usually not sufficiently fulfilled because of the low stiffness of the material and warpage.

8.2 Scope of the standard

DIN ISO 20457 / DIN 16742 is used to specify tolerances that are feasible in production (**production tolerances**) and to compare them with **functional tolerances**, in order to check whether a dimensional accuracy required by the design can be achieved.

- **Tolerances covered:** limit deviations for size dimensions (two-point dimensions) as indirect tolerancing (general tolerances) and as direct tolerancing (deviation stated at the nominal size); for form and position deviations, profile shape tolerances as general tolerances and position tolerances with cylindrical tolerance zones for direct tolerancing.
- **Processes:** primary shaping processes with moulds closed on all sides, such as injection moulding, injection compression moulding, transfer moulding and compression moulding of non-porous parts made of thermoplastics, thermoplastic elastomers and thermosets, as well as rotational moulding of

thermoplastics. For special variants of these processes, the standard can be applied analogously by agreement with the part manufacturer.

- **Not covered:** porous materials (e.g. foams), other processing and machining methods, and combinations of primary shaping and forming processes (e.g. injection blow moulding, injection stretch blow moulding).
- **Surface quality** such as sink marks, undesired flow structures, roughness and weld lines is not covered by the standard.

If tolerances outside the scope of the standard are required, they must be agreed with the part manufacturer and specified on the drawing.

8.3 Processing shrinkage

The processing shrinkage (PS) describes the difference between the mould contour dimension and the part dimension. It and its variations largely determine the achievable dimensional accuracy.

Defining equation	Explanation
$PS = (1 - LP / LM) \cdot 100\ \%$	PS processing shrinkage; LP part dimension; LM mould contour dimension
$\Delta PS = \lvert PS\perp - PS\parallel \rvert$	absolute PS difference due to shrinkage anisotropy; PS⊥ perpendicular, PS∥ parallel to the melt flow direction
$\Delta S = PSmax - PSmin$	range (shrinkage variation) of the PS; PSmax, PSmin extreme values due to the respective influencing conditions
$PSc = 0.5 \cdot (PSmax + PSmin)$	calculated value of the PS as the mean of PSmax and PSmin

Causes and factors influencing the processing shrinkage of non-porous plastics:

Cause	reduces PS	increases PS
Increase in density due to thermal contraction (cooling from demoulding to room temperature) and compaction by pressure	high effective pressure on the moulding compound and contour until demoulding (holding pressure); low demoulding temperature (long cooling time, low contour temperature); low coefficient of thermal expansion (rigid polymers)	low or prematurely reduced holding pressure; high demoulding temperature (short cooling time, high contour temperature); high coefficient of thermal expansion (flexible or rubber-elastic polymers)
Increase in density due to structural ordering (crystallization, gelation)	amorphous polymers; low degree of crystallinity due to fast solidification (low contour temperature, thin-walled parts); high degree of gelation of plasticized polymers	semi-crystalline polymers; high degree of crystallinity due to slow solidification (high contour temperature, thick-walled parts) or nucleating agents; low degree of gelation of plasticized polymers
Increase in density due to molecular build-up and cross-linking (curing, vulcanization, polyreaction)	high degree of cross-linking and therefore lower thermal expansion (long curing or vulcanization time, high melt temperature); largely pre-cross-linked moulding compounds (e.g. prepolymers)	low degree of cross-linking and therefore higher thermal expansion (short curing or vulcanization time, low melt temperature); uncross-linked precursors (oligomers) or monomers

Cause	reduces PS	increases PS
Change in stiffness or hardness due to additives (fillers and reinforcements, plasticizers)	additives with low thermal expansion (e.g. inorganic fillers and reinforcements); no or little plasticizer	additives with high thermal expansion (e.g. organic fillers and reinforcements); plasticizers

8.4 Stiffness and hardness level of the materials

Classifying the material stiffness and hardness for the point evaluation is straightforward, because the data are available from moulding compound manufacturers and databases. If values lie at a range limit, the user decides on the assignment. Classification for polymers without solid additives:

Hardness	E [N/mm²]	Shore hardness	IRHD	Polymers
hard	> 5000 … 20,000	–	–	LCP (self-reinforcing molecular anisotropy)
hard	> 1200 … 5000	> 75 D	–	tough-hard to brittle-hard thermoplastics, thermosets, hard rubber
semi-hard	> 300 … 1200	> 60 … 75 D	–	elastomer-like (flexible) thermoplastics, thermoplastic elastomers and rubber in different hardness grades
semi-hard	> 30 … 300	> 35 … 60 D	–	as above
soft	3 … 30	50 … 90 A	50 … 90	as above
very soft	< 3	< 50 A	< 50	as above

E: initial modulus of elasticity from the short-term tensile test according to DIN EN ISO 527; Shore hardness: indentation hardness according to DIN EN ISO 868 (methods A and D); IRHD: international rubber hardness degree according to DIN EN ISO 48. All tests at 23 °C with standard-conditioned specimens. For comparison, steel: E = 200,000 N/mm². Filled or reinforced polymers are assigned by their modulus of elasticity.

8.5 Classifying the tolerance groups

- **Process:** injection processes allow higher production accuracy than compression moulding.
- **Calculated value of the processing shrinkage:** without your own experience, the classification should be agreed with the part manufacturer. Guide values are available in the “Processing shrinkage ranges” tab (section 6.2).
- **Shrinkage variations:** they are often difficult to estimate because of the many influencing factors; the part manufacturer can give the best help. If in doubt, select “only imprecise” and then check whether a more accurate tolerance series is an option.
- **Rotational moulding** of thermoplastics is assigned to tolerance group TG 9 by the standard.
- **Multi-component parts:** a tolerance group is determined for each material and stated as a separate general tolerance (e.g. hard component TG 5, soft component TG 7). For size dimensions across materials, the less accurate material governs.

Classification aids of the standard for the shrinkage variations:

Classification	Meaning
precise	Calculated values of the PS are known, e.g. from experience, systematic measurements or simulation. Shrinkage anisotropy is irrelevant or can be taken into account with sufficient accuracy in the respective direction. Possible deviations from the calculated value are at most ± 10 %.
conditionally precise	Calculated values are known within ranges of at most ± 20 %.
only imprecise	Calculated values of the PS are only known as rough guide ranges, shrinkage anisotropy cannot be taken into account or only insufficiently, and practical experience is lacking. Possible deviations from the calculated value exceed ± 20 %.

8.6 Mould-related part dimensions

Different deformations and position deviations of mould parts caused by pressure are covered by distinguishing between **tool-specific** and **non-tool-specific** dimensions. Tool-specific dimensions are formed within the same mould part. Non-tool-specific dimensions result from the interaction of several mould parts – e.g. across the parting plane or by slides – and therefore tend to vary more. Besides the stiffness of the mould design, the processing conditions have a great influence. The following illustrations of the dimensions x are taken from DIN 16742.

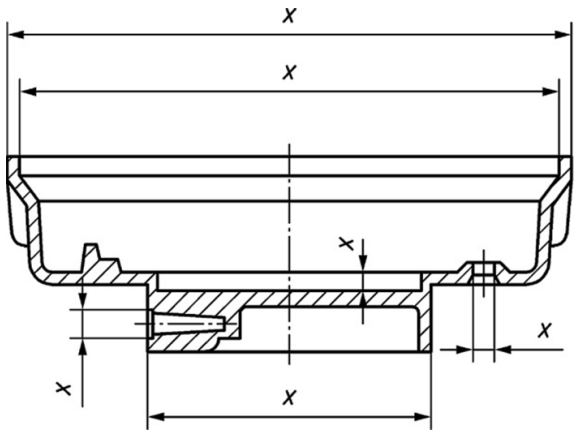


Figure 15: Tool-specific dimensions (x)

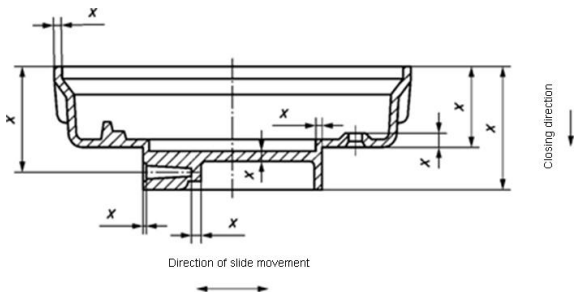


Figure 16: Non-tool-specific dimensions (x)

8.7 Nominal dimension definitions

- **Nominal size dimensions** (length dimensions, two-point dimensions) for limit deviations according to Table 2 of the standard are mid-tolerance dimensions for drawings and CAD data sets.
- **DP dimensions** (distance dimensions) are nominal dimensions for position tolerances according to Table 9 and profile shape tolerances according to Table 10. A part can have one or more datum

systems; the decisive value is the greatest distance of the toleranced feature from the origin of the datum system used. This does not have to coincide with the coordinate system of the part or assembly. Annex G of the standard contains an example.

- For the distance between parallel surfaces that are offset from each other, the DP dimension for limit deviations according to Table 9 is used as the nominal size.
- Nominal dimensions **below 1 mm and above 1000 mm** as well as limit deviations for **wall thicknesses** are subject to agreement.
- For **general tolerances**, only the limit deviations for non-tool-specific dimensions apply.
- General tolerances are stated in the design documentation, for example, as **DIN 16742 – TG6**.

8.8 Dimensional reference levels and acceptance conditions

The dimensional variations in production and use require three closely linked **dimensional reference levels**: part application, part production and mould making. Different physical relationships apply to each level; the design is worked out in the opposite direction to the way the dimensions are created – from the application via part production to the mould contour. For each level, position (mid-tolerance dimension C), shift (dimensional shift Δl, with sign) and variation (tolerance T) are considered: part application tolerance TA (total tolerance), part production tolerance TF and contour production tolerance TW. The principle is explained in Annex A of DIN 16742.

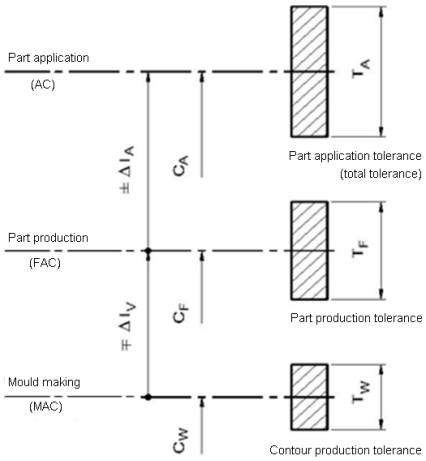


Figure 17: Dimensional reference levels for application and production of plastic parts

In the figure, AC stands for the application conditions, FAC for the fabrication acceptance conditions of part production and MAC for the acceptance conditions of mould making (German: AWB, ABF, ABW).

Application conditions (AC) are all conditions of use and storage after production that affect dimensional accuracy and function. If assembly takes place long after production, storage and assembly conditions can be treated as a special case of the AC. For soft or rubber-like materials and for assemblies made of identical materials, their influence can often be neglected; in principle, however, the AC must be determined depending on the situation and function. Causes of application-related dimensional changes are:

- climatic influences from ambient temperature, humidity, precipitation and solar radiation
- use-related energy input from heat sources and high-energy radiation
- diffusion contact with vapours and liquids and migration contact with solids
- material removal (wear) by friction, cavitation, erosion and biological influences
- mechanical deformation by external forces and moments and by relaxation of internal stresses
- molecular and micromorphological structural changes

Causes of production-related length deviations:

- variation of the processing shrinkage depending on the moulding compound and production
- uncertainty of the calculated shrinkage values used for the mould contour, especially with high shrinkage and shrinkage anisotropy
- different recovery of the parts after demoulding, depending on the material stiffness
- production-related dimensional variation of the mould contours including distortion from hardening and surface coating
- deformation and position deviations of mould parts caused by pressure
- wear of the mould contour

Warpage (bowing, twisting, distortion) is the physical cause of form, position and angular deviations. It results from local and direction-dependent differences in shrinkage (shrinkage anisotropy) and, with extreme holding or compression pressure, from the recovery of elastic residual stresses. Warpage cannot be avoided in principle, but it can be minimized.

Fabrication acceptance conditions (FAC): according to DIN 16742 and DIN EN ISO 291, the inspection dimensions are the acceptance values if the parts are stored after production at 23 °C ± 2 K and 50 % ± 10 % relative humidity and inspected no earlier than 16 h and no later than 72 h after production. If these conditions are not met, the dimensional position and deviations, inspection method, inspection period and storage and inspection conditions must be agreed and documented separately between manufacturer and buyer. Deviations can result from:

- subsequent operations with material application (painting, coating) or material removal (machining, grinding, polishing)
- annealing (anticipating post-shrinkage, relieving internal stresses, post-curing) or subsequent operations with considerable thermal load (painting, solder bath)
- conditioning, e.g. water storage (anticipating swelling, increasing toughness)
- low dimensional stability of the material, e.g. transformation of the crystalline phase (PB) or swelling and plasticization of thin-walled parts made of hydrophilic polymers (PA6, PA66, PA46, biopolymers)

Acceptance conditions of mould making (MAC): the inspected control dimensions of the mould contours are the acceptance values at a reference temperature of 23 °C ± 2 K and include distortion from hardening.

8.9 Tolerance series

Tolerance series	Meaning
Series 1 – normal production	Production with general tolerances may be feasible. Dimensional accuracy requirements are not a particular quality focus.
Series 2 – accurate production	Production and quality assurance are oriented towards higher dimensional accuracy requirements.
Series 3 – precision production	Production and quality assurance are fully aligned with very high dimensional accuracy requirements.
Series 4 – special precision production	As series 3, but with intensified process monitoring.

Important: Tolerance series 3 and 4 are always subject to agreement.

Only small functional tolerances that are absolutely necessary justify the effort of series 3 and 4, which must be compensated by a considerable price premium. Buyer and manufacturer should therefore first clarify whether the tolerance requirements can be met. The following should be checked:

- design and dimensioning of the part suitable for dimensional accuracy
- sufficiently stiff and thermally and rheologically balanced moulds
- operation of the machines and equipment by sufficiently qualified personnel including quality assurance
- delivery conditions of the moulding compounds with regard to properties relevant to dimensional accuracy, especially shrinkage variations

The “Realizable tolerance series” checklist (section 4.4) supports this check.

9 Menus

Menu	Commands
Program	**Exit**; **Info about…**
View	Switch between **“Tolerances for plastic moulded parts according to DIN ISO 20457 / DIN 16742”** and **“Commercial products”**; **Original size (100%)**.
Settings	**Server connection…**; **Activate the license…**; **Reset the license…**; **Language/Sprache…** (user interface language).
Help	**Check for software update…**; **To our homepage…**; **Register for the software…**; **View online help**; **View local help…** (F1 key); **View local Help (Commercial products database)…**
Project management (blue bar)	**Create a new project…**; **Load project…**; **Save project**; **Save Project As…**; **Print / print preview**.
Extras (blue bar)	**Determine realizable tolerance series…** (section 4.4); **Realizability of functional dimensions…** (section 4.5).
Settings (blue bar)	**Tolerance values according to DIN ISO 20457**; **Tolerance values according to DIN 16742** (chapter 7).

10 Tips and frequently asked questions

Question	Answer
No tolerance group appears, only a red message.	Information is missing. Select one option in every group of the point evaluation, or decide all questions in the assignment help (no filled check box left).
“tool-specific” cannot be ticked.	The dimension is marked as a wall thickness dimension. For wall thickness dimensions, “tool-specific” cannot be selected.
The functional dimension has a red background.	The deviation cannot be achieved in the selected tolerance series. Check in the details which tolerance series achieves it, or use “Realizability of functional dimensions” (section 4.5).
I cannot see all columns of the table.	Scroll to the right using the scroll bar below the table.
The part is not rigid or has free-form surfaces. What do I need to consider?	Agree the measuring concept and verification with the part manufacturer; free-form surfaces are specified with profile shape tolerances (section 8.1).

Question	Answer
A part is made of two materials.	Determine a separate tolerance group for each material; for dimensions across materials, the less accurate material applies (section 8.5).
Larger dimensions are not calculated.	The DEMO version only supports nominal dimensions from 5 … 55 mm.
What shrinkage does my moulding compound have?	See the “Processing shrinkage ranges” tab (section 6.2) or the datasheet in the “Commercial products plus” module.