PolCycleTime – Cooling and cycle time calculation, price and demand calculations, conditioning time calculation

Inhaltsverzeichnis

Content

Cooling and cycle time calculation

The subject of the program is the computer-based estimation of the cooling and cycle time for standard injection molding of non-porous thermoplastics and thermoplastic elastomers.

For special process variants of injection molding, an analogous application is possible, although the accuracy may be somewhat limited.

Typical applications include:

- Machine presetting
- Pre-costing
- Production flow planning
- Thermal balances
- Presetting data for process simulation

For cycle time minimization, systematic optimization under real production conditions is usually recommended. User input is supported by notes, which are available as helps or orientation areas.

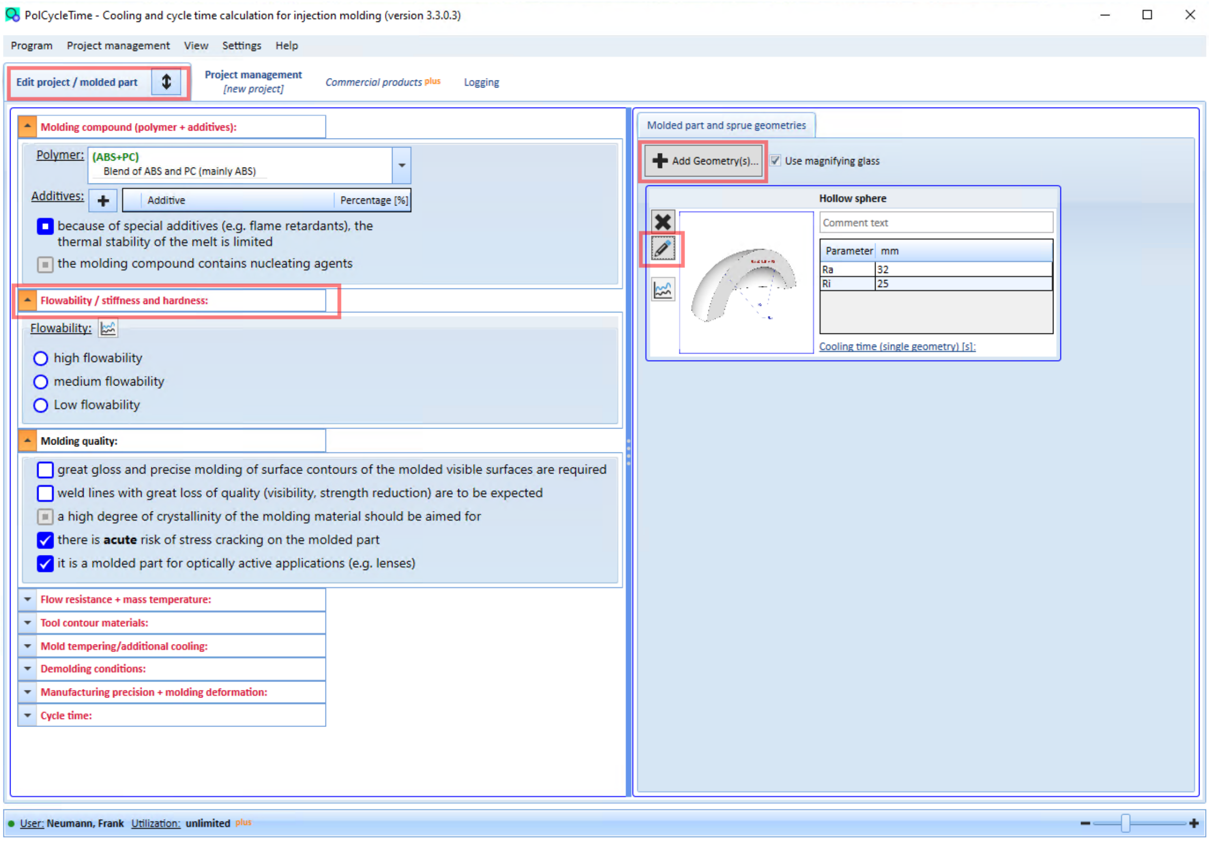
In practice, cycle time classifications are handled differently. Therefore, the program-internal cooling time and the cycle time are to be defined below.

- Cooling time: Part of the time from the start of mold filling to demolding of the molded part, i.e. the physically effective total time of the molding compound cooling in the mold.
- Cycle time: Cooling time plus demolding time of the molded part and dry running time (idle time) of the machine.

The calculation of the cooling time is based on analytical solutions of transient heat transfer, which have been considerably extended and modified with respect to the concrete conditions.

Quick start

The view for editing a project ...



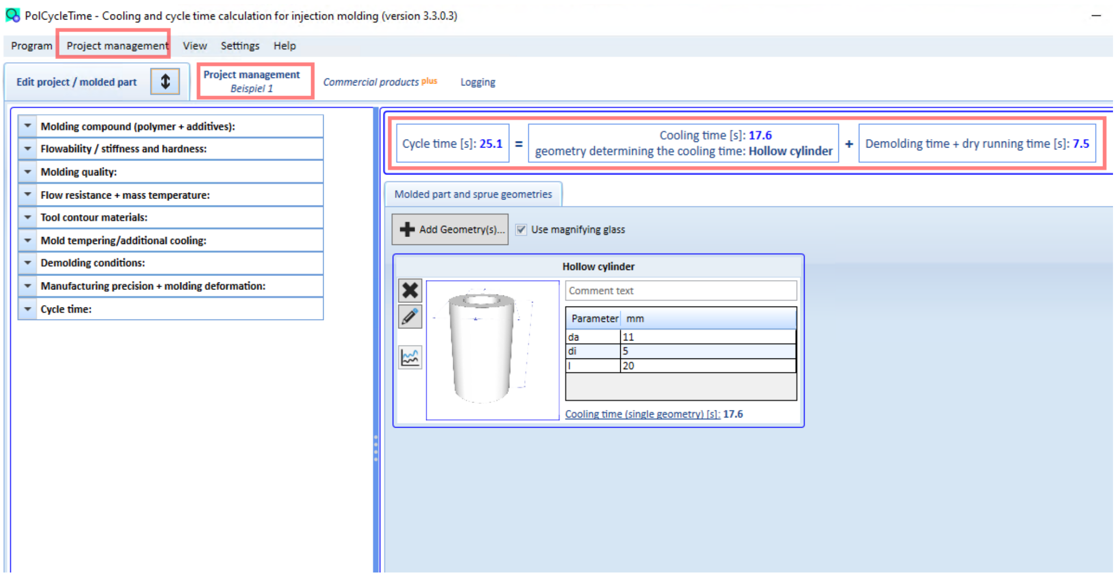
The following sequence is only a suggestion. The order of your inputs/decisions is of secondary importance only. Results are displayed or updated immediately when all required inputs and decisions are available.

1. Enter all your inputs and decisions that affect the cooling and cycle time. The inputs and decisions are grouped into individual groups. The heading of the group is **red** if there are still entries missing within the group.

2. Add the relevant molding and sprue geometries (in the right part of the view)
   a) Switch to the "*Molded part and sprue geometries*" tab
   b) Click the [*Add geometry(s)*] button. Another window will open.
   c) Select the geometry(s) by double-clicking on each one. The selected geometry is added to the corresponding list in the background.
   d) Close the geometry selection window.

3. Edit the geometry data.
   a) To do this, click on the button with the **pencil** symbol (to the left of the corresponding geometry). Another window will open.
   b) Enter the geometry data and confirm your entries with the [*Enter*] key.
   c) The window will then close.

4. As far as all inputs are available, the results (top right) are displayed. If necessary, you can now correct or specify your entries and decisions later.



5. Finally, switch to the "*Project management*" tab and save your entries and decisions as a project file. We recommend that you create a corresponding project file for each mold part or mold part-tool-machine combination.

All project information is displayed in a print preview and can be printed out (menu or "Project management" tab).

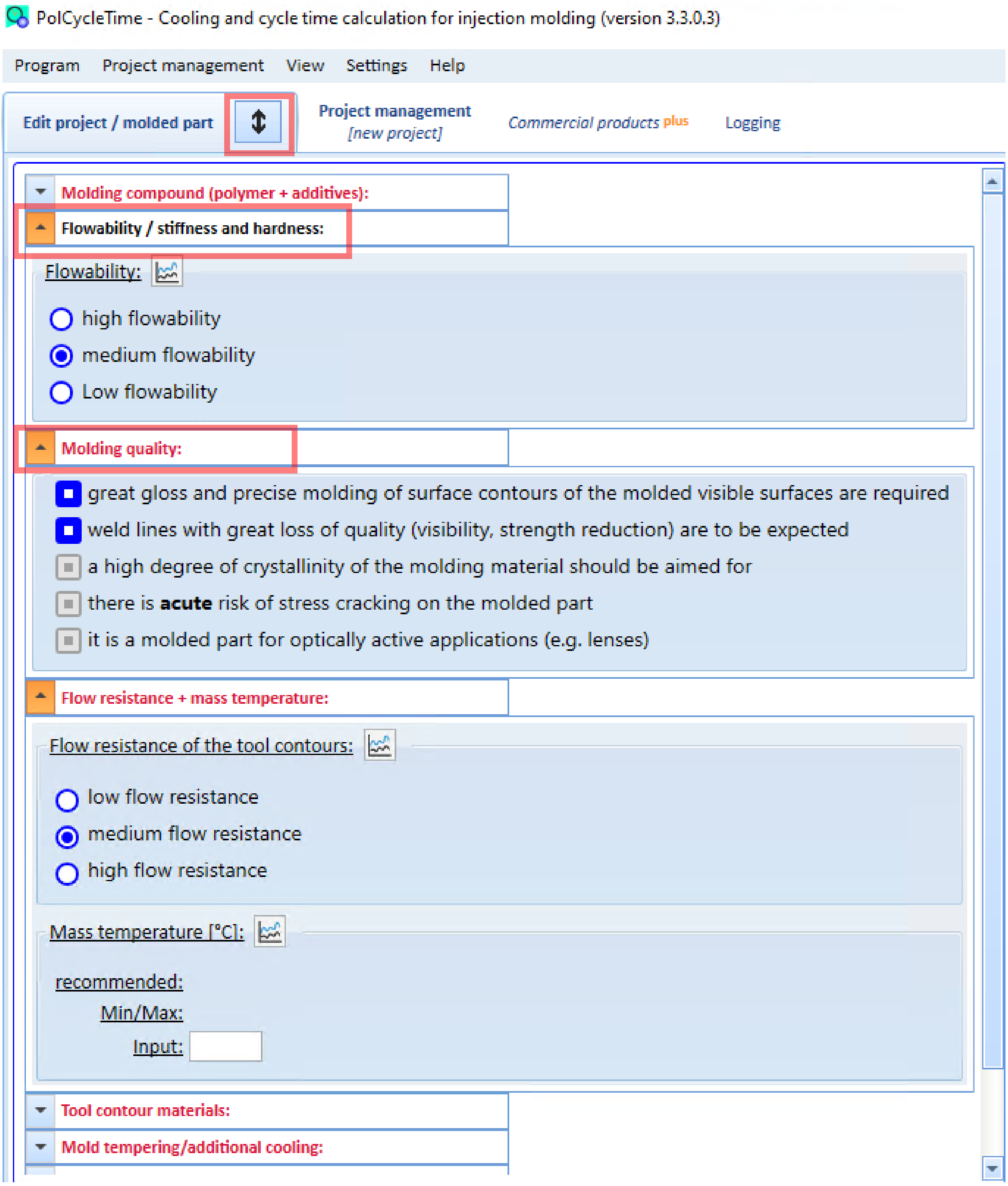
Operation

We recommend that you create a corresponding project for each molded part.

You can save all your entries/decisions as a file and print them out.

You can find more information on project management here.

Inputs / Decisions



- The required inputs/decisions are grouped logically.
- we recommend that you process the inputs from top to bottom accordingly, because information on subsequent inputs is partly based on previous inputs/decisions.
- Each group has a corresponding heading. This heading appears in **red font color** if entries are still missing or incorrect within the group. As soon as all entries within the group are complete, the heading appears in **black font color**.
- You can expand and collapse the groups individually. If you want to show or hide all details of the groups, click on the button with the double arrow in the "*Edit project/moulded part*" tab.

The technical details and operation of the individual entries/groups are described below.

[Molding compound / additives](#)

[Flowability / stiffness and hardness](#)

[Molding quality](#)

[Flow resistance and mass temperature](#)

[Mold tempering and additional cooling](#)

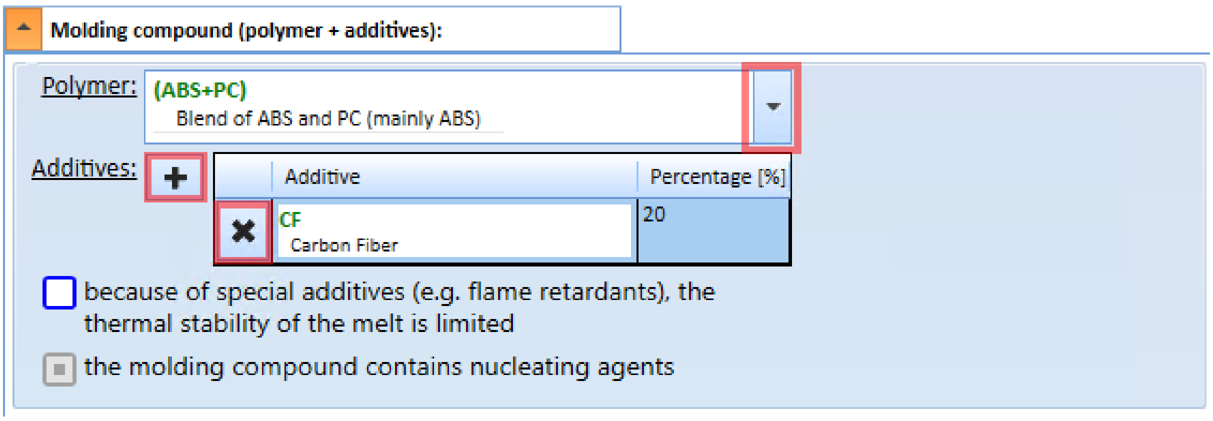
[Demolding conditions](#)

[Manufacturing precision + molding deformation](#)

[Cycletime](#)

Molding compound / additives

- The polymers without additives and the additives are labeled with abbreviations and designations according to EN ISO 1043 (1997), DIN EN ISO 18064 (2005) and ISO 1874 (1992). Density ranges of the polymers (at 23 °C) provide a further help for classification, but no flame retardant additives were considered. In some cases, characteristic properties (e.g. density, Shore hardness, heat deflection temperatures) are given in addition to the abbreviation to define the type.
- PTFE is considered as an additive because of its solid consistency and infusibility (powder, flakes). It is not a blending component.
- If necessary, the molding compound supplier must publish the composition of the molding compound with regard to the type and mass fraction of the additives.
- Biodegradable/compostable polymers as defined by DIN EN ISO 14885 and DIN EN ISO 13432 can be identified by the addition ***bioab*** at the beginning of the text to the short description/designation.



1. Click on the drop-down box to select the polymer
2. For **polymers with additives**, you must also specify the additives and the corresponding percentage.
   - To do this, click on the button with the **plus** symbol.
   - Double-click in the "*Additive*" column to display a selection box in which you can then select the additive.
   - Confirm the selection with [*Enter*] and then enter the percent in the "*Percentage %*" column.
   - You can add a maximum of 5 different additives.
   - To delete an additive again, click on the corresponding button with the **X** symbol.

Flowability / stiffness and hardness

- Relative flowability within a molding compound range is the fluidity graded by rheological test values (e.g. MFR; MVR). Thermoplastic melts are structurally viscous and viscoelastic media whose viscosity (or fluidity) depends not only on the temperature but also on the amount of mechanical stress exerted on the melt by shear and strain. A comprehensive description of the flow behavior is only possible using elaborate flow curves or viscosity functions. For a simplified characterization of the flow behavior, rheological single-point data of the melt are generated, which are currently mostly measured as volume flow rate (MVR) or mass flow rate (MFR) according to ISO 1133. Different test conditions are prescribed for each polymer type, so that a general comparison between different polymers is not possible. However, the molding compound manufacturers give an evaluation of the flow behavior within the respective type range (e.g. easy-flowing, normal-flowing, heavy-flowing), so that the program user is sufficiently informed.
- Stiffness and hardness are quantified by the original E-modulus from the rapid standard tests (tension, bending, compression) as well as by the shore hardness according to DIN EN ISO 868 (method A and D) or by the sphere indentation hardness according to DIN ISO 48 (International Rubber Hardness Degree IRHD).

Molding quality

- When evaluating the gloss of plain molding surfaces, an analogous fine processing or polishing quality of the mold contours must be assumed. For structured visible surfaces, special attention must be paid to the fine structures.
- The occurrence of weld lines can be clarified by geometric considerations (swelling flow) or by computer-aided filling pattern simulation. Possible quality reduction (visibility; strength reduction) has to be decided according to requirements and experience.
- Higher degrees of crystallization in semi-crystalline polymers result, among other things, in greater stiffness and heat resistance, improved media resistance, lower post-shrinkage with increased processing shrinkage, and significantly improved abrasion resistance.
- Abrupt cooling can cause "internal tensile stresses" in the molded part, which in the case of brittle plastics (e.g. PS) cause "hairline cracks" or, when exposed to specific media, stress cracks. Analogous effects are possible when overmolding non-preheated metal or ceramic inserts.
- Molded parts for optically active applications (mostly made of acrylate polymers) require residual stress-free slow cooling.

Corresponding qualities of the mold contour surfaces can be derived from design and functional requirements for the molded part surface quality (smoothness, gloss) and its manufacturability (demolding, post-treatment, film formation). Their gradation is given below with the arithmetic mid-roughness value Ra according to DIN EN ISO 4287:

- Technically rough (without fine machining): $R_a \geq 3$ µm.
- Smooth (with fine finishing. e.g. grinding): R_a = 0.8 to 1.6 µm
- Glossy (coarse polish): R_a = 0.2 to 0.4 µm
- High gloss (fine polish): R_a = 0.05 to 0.10 µm
- Reflective (optical fine polish): R_a = 0.012 to 0.025 µm

When deciding on the surface quality, it must be taken into account that with increasing polishing quality, the costs for manufacturing and maintaining the molds increase significantly, while the imaging precision on the molding surface decreases. The type of plastic and processing conditions can also have a significant impact on imaging reproducibility. In particular, the special optical grade (mirror-like surface) should therefore only be provided for transparent plastics for special requirements (e.g. lenses, scales, CD).

Molded parts for optical applications require very low residual stresses (density constancy) and long-term holding pressure effects to avoid sink marks in relatively thick-walled parts. Both conditions are realized, among other things, by high mold contour temperatures. For this purpose, a special orientation for high contour temperatures is specified in the software, which is only slightly below the glass temperatures of transparent thermoplastics.

The "*plate*" or "*shell*" is to be selected as the geometry element for lenses and the like, and average wall thicknesses are to be estimated from the courses of the radii of curvature.

Flow resistance and mass temperature

- The orientation ranges of the melt temperature (melt or plasticizing temperature) are determined by rheological conditions, quality requirements and thermal stability.
- The injection pressure causes a temperature increase of the melt due to shear heating. This cooling time part is taken into account by a program-internal estimation of the injection pressure on the basis of a molding-specific minimum pressure.
- The evaluation of the flow resistance of the molding and sprue contours is based on the user's experience. The flow resistance includes the relation of flow distance to flow cross-section, whereby the impact of melt direction changes is almost insignificant.

The mold contours are filled from the nozzle (sprue) or from the gate (molded part) by a laminar source flow, provided the melt does not solidify before mold filling is complete. For a given flowability (viscosity) of the melt, the most effective flow extension or shortening of the filling time is achieved by increasing the melt temperature and/or the injection pressure. Melt redirections have only an insignificant effect on the flow resistance. Due to the thermal insulation of the solidifying surface layers, increasing the mold contour temperature only slightly reduces the flow resistance, but significantly increases the cooling time. Increasing the flow cross-section (e.g. by increasing the wall thickness) reduces the flow resistance disproportionately. With regard to the effect of wall thickness on the flow resistance of flat mold contours, the following wall thickness classification should serve as a guide:

  - below 1 mm = very thin-walled
  - 1 to 2 mm = thin-walled
  - over 2 to 4 mm = normal wall
  - over 4 mm = thick-walled

For wall thicknesses below 0.5 mm, reliable mold filling is not always guaranteed, even with small flow distance lengths.

- The software-internal orientation ranges of the melt temperature represent the most common temperature ranges in practice.
- The melt temperature ranges are determined according to the thermal stability of the melt and the flow conditions of the mold filling, as well as according to the molded part quality (surface gloss, weld line quality) that depends on them. If the melt temperatures are defined within the specified ranges, the cooling time is influenced relatively little.
- The mold contour temperature, on the other hand, has a significant impact on the cooling time. Low cooling time (high productivity) requires low contour temperatures, while special molding quality requires correspondingly high temperatures. The choice of the "correct" contour temperature is always a compromise which, with regard to the quality of the molded part, can only be decided by production trials. The practical experience of the user is very important here. Otherwise, the possible cooling time deviations must be tested by input variations, whereby the respective purpose of the cycle time calculation (e.g. cost pre-calculation, machine occupancy, machine setting) allows appropriate conclusions to be drawn.
- A further precision of the temperatures is possible by communication with experience carriers (e.g. molding manufacturers, molding mass manufacturers).
- Proponents of impulse cooling frequently claim, among other things, the advantage of a strong reduction in cooling time. In fact, this claim is neither theoretically nor practically

proven, provided verifiable investigations are evaluated.

Mold tempering and additional cooling

- When determining the contour temperature, the feasibility of the intended mold temperature control system must be checked:
    - Water temperature control for contour temperatures up to 95 °C.
    - Pressurized water temperature control for maximum contour temperatures of 140 °C to 180 °C depending on the device type.
    - Oil tempering for maximum contour temperatures of 200 °C to 350 °C depending on the device type.
- When determining the contour temperature, note that this has a particularly strong influence on the cooling time.
- The type of cooling medium circulation (constant or intermittent) has no significant influence on the cooling time.
- Additional cooling by means of a temperature control bath or cooling gauge parallel to the mold cycle time serves to reduce the production throughput time, especially for thick-walled molded parts.

Demolding conditions

- If sprues are fixed in the mold by cooling, the type of demolding determines the required temperature compensation (e.g. prevention of sticking by "post-melting").
- Demolding systems (ejection, overpressing of undercuts) may cause molded parts to deform impermissibly as a result of local pressures or other stresses.
- In the case of uneven mold tempering, so-called hot spots (hot zones) can be the cause of demolding difficulties.

Manufacturing precision + molding deformation

- Production-related dimensional variations for length dimensions as well as distortion-related form, position and angle deviations determine the dimensional accuracy immediately after the production of the molded part.
- The achievable dimensional precision is determined, among other things, by the production and quality assurance effort that can be mobilized by the molding manufacturer. The tolerance series according to DIN ISO 20457 are intended for this purpose. Precision special production is to be classified like precision manufacturing.

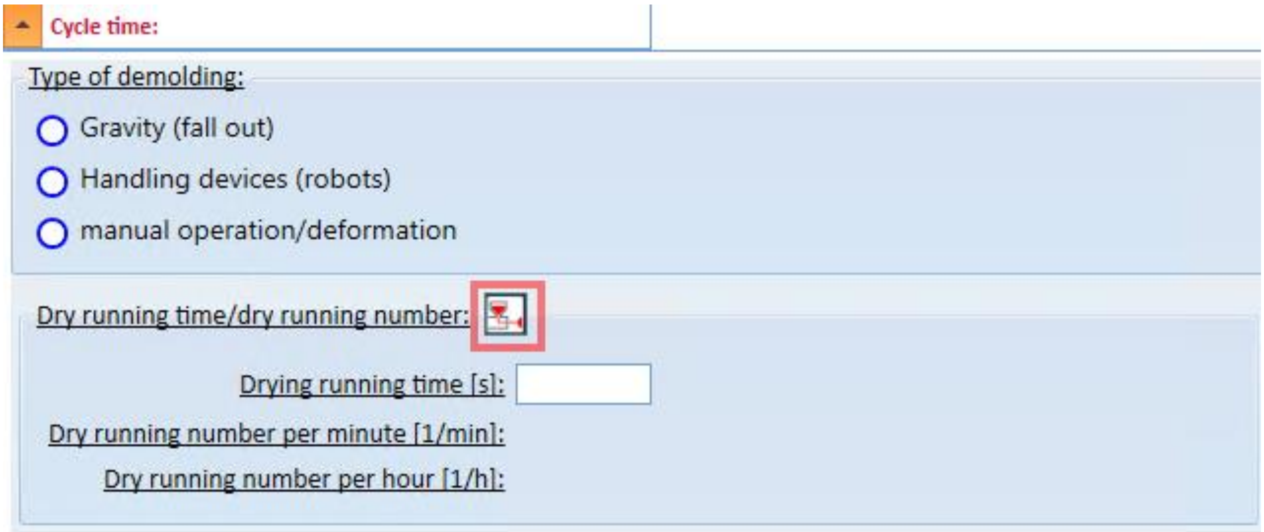
Among other things, you can use our software products **Pol*Tolerances*** or **Pol*Tolerances-AC*** to determine the tolerance series according to DIN ISO 20457. You can find the corresponding downloads on our homepage.

Cycletime

- Demolding times are specified by reference values, whereby in the cases of handling demolding and manual operator intervention, the user should make inputs according to experience. In the case of gravity demolding, the demolding time is allocated approximately in 3 stages (1; 1.5; 2 s) according to the number of cavities.
- Demolding times by handling device (robot) for molded parts of small to medium size, are usually in the range of 2 to 6 seconds.
- Dry running times or dry running numbers are determined according to EUROMAP 6 as the time for mold and, if necessary, nozzle movements. They are a measure of the basic speed of the injection molding machines. Information can be obtained from the machine documents of the SGM manufacturers or from the software database. The dry run times in the software refer to the standard stage of the respective machine type. Dry running times reduced by special power amplification must be entered directly. For the machine idle time within the cycle, 1.5 times the dry running time is specified as a flat rate value, since a more precise

determination is often not possible or is too time-consuming.

- For multi-station machines, the minimum station change time must be added to the dry running time and the total entered.

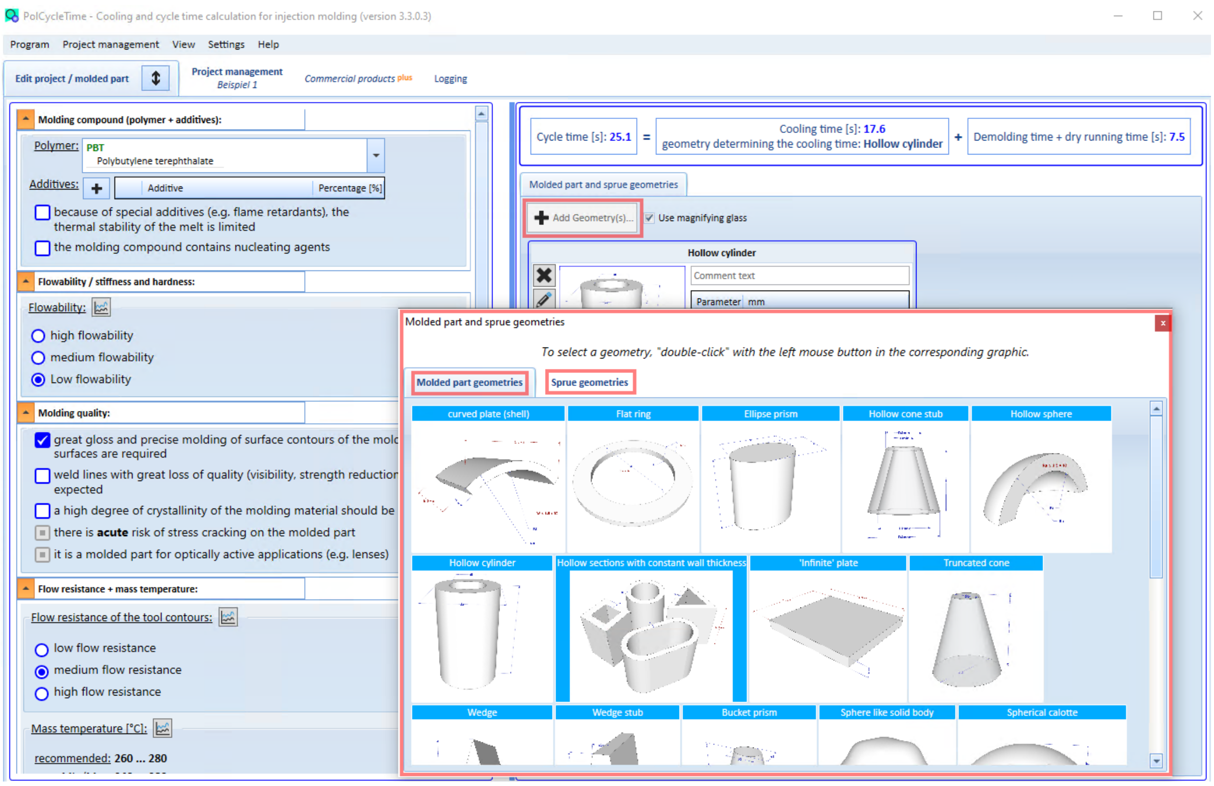


The inputs for the cycle time are **optional**. Without these entries, only the cooling time is shown in the results view.

To find out about dry cycle times of injection molding machines, click the highlighted button shown in the picture. A further window then opens in which dry cycle times of approx. 500 injection molding machines are displayed.

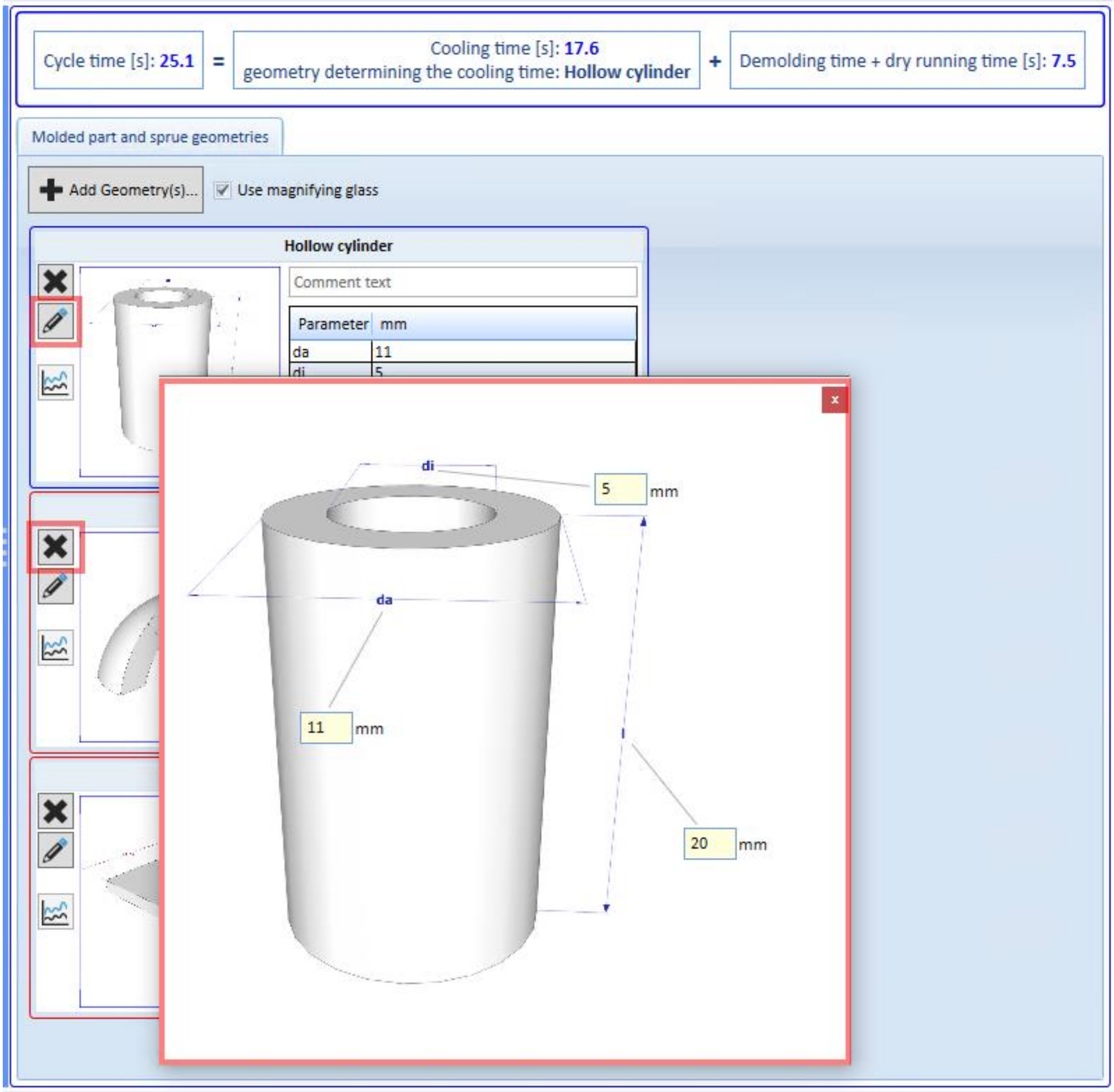
Add/edit geometries

The operation does not enforce a predefined sequence. Whether you enter all geometries first and then all other project inputs or vice versa is irrelevant. This is one of the reasons why editing is possible at any time and the results are updated immediately.



- You analyze individual parts of the molded part to determine whether they can be decisive for the cooling time of the molded part.
- To do this, mentally divide your part into sub-geometries and analyze them individually.
- Click the [*Add geometry(s)*] button in the main view. Then another window opens. You can change the size and position of this window as you like.
- Within the new window you will see two tabs "*Molded part geometries*" and "*Sprue geometries*".
- Now select one or more geometries. Simply double-click on the corresponding graphic. These geometries are then immediately transferred to the list in the main view.
- Close the window when you have added all the part geometries.

Now edit the dimensions of the added geometries



- Click on the corresponding **pencil** icon (to the left of the geometry). Another window will open.
- now enter all dimensions
- use the Enter key to complete the inputs. The window will be closed.

If all other project data are available, the cooling time per geometry and the determining cooling time of the molded part will be displayed immediately.

To remove a geometry from the list of the main view, click on the button with the **X** icon.

Notes

Injection molded parts are characterized by a wide range of geometric design variety and complexity. The software offers a wide range of options for describing the many geometric elements (molded part sections) of a molded part. Therefore, the user has the task to divide his concrete molded part into such sub-parts with sufficient precision.

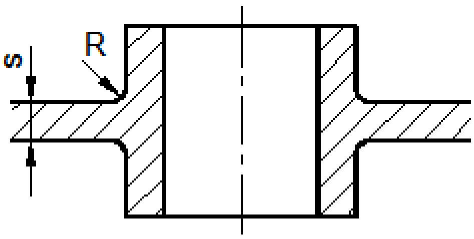
The part with the longest cooling time is then decisive for the cycle.

SPRUE GEOMETRY DETERMINING COOLING TIME

- In injection molding with a fixing sprue, in unfavorable (but usually avoidable) cases the sprue manifold cross-sections can be cooling time-determining and can be traced back to the specified manifold shapes.
- In the case of branched sprue systems, the largest cross-section should be selected in each case for the same cross-section shape.

COOLING TIME-DETERMINING MOLDING GEOMETRY

- The cooling time determining molding section shall be determined by selecting applicable geometry elements or their combination as well as by creative approximation assignments.
- All prismatic or profile-like sections can be straight or curved in the lengthwise direction. In the case of cross-sectional dimensions that vary in sections, the maximum values are usually the ones that determine the cooling time.
- Cross-section increases due to radius transitions between part sections can be taken into account by increasing the wall thickness input by approx. 25 % for adjacent flat part elements.



Example: Input: (s · 1.25) mm

Show dependency diagrams

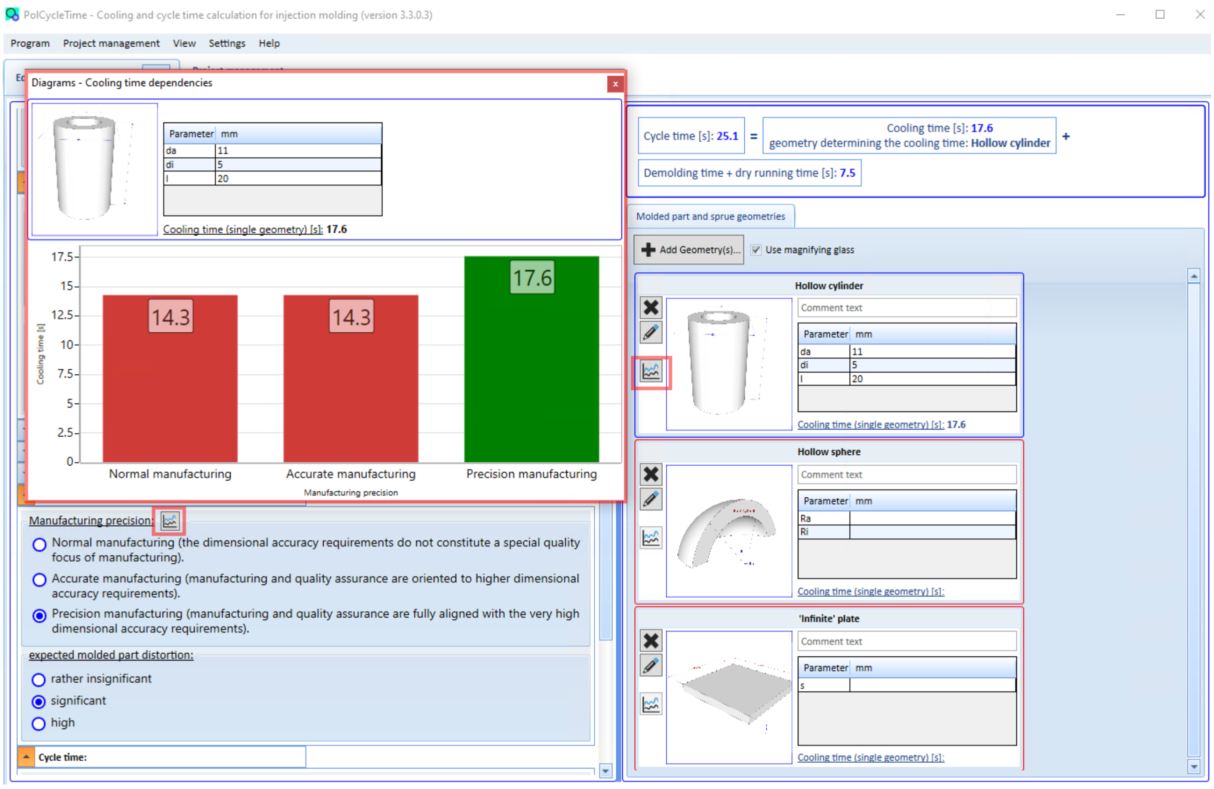
We have provided so-called dependency diagrams for certain inputs/decisions.

You can display these diagrams by assigning a geometry and one of the following properties accordingly.

- Flowability
- Flow resistance of the tool contours
- Melt temperature
- Tool contour materials

- Contour temperature
- Manufacturing precision

In each of the corresponding views there is a corresponding **diagram** symbol

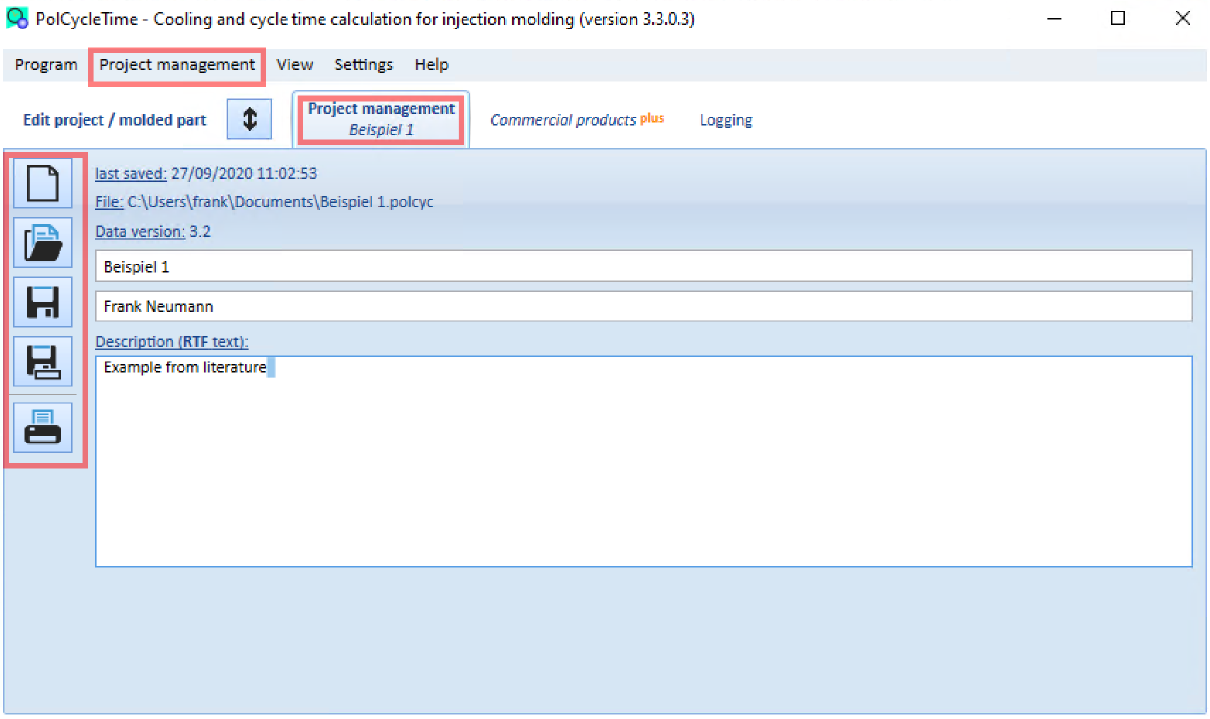


- select the geometry by clicking on the **diagram** icon of the geometry
- select the property within the input groups (also by clicking on the **diagram** icon). Now another window will be opened.
- here you can see the geometry and the possible values of the corresponding property (X-axis). Depending on this, the calculated cooling times are displayed as a bar chart or as a line chart.
- you can leave the window open and select either another property or another geometry. The display will then be updated accordingly.
- the value displayed in **green** corresponds to your current selection of the corresponding property

Here in this example you can see that with a required precision production the cooling time would increase by approx. 7 seconds.

Project management

You can manage your projects in individual files. For this purpose there is a corresponding tab as well as a corresponding menu in the main menu.



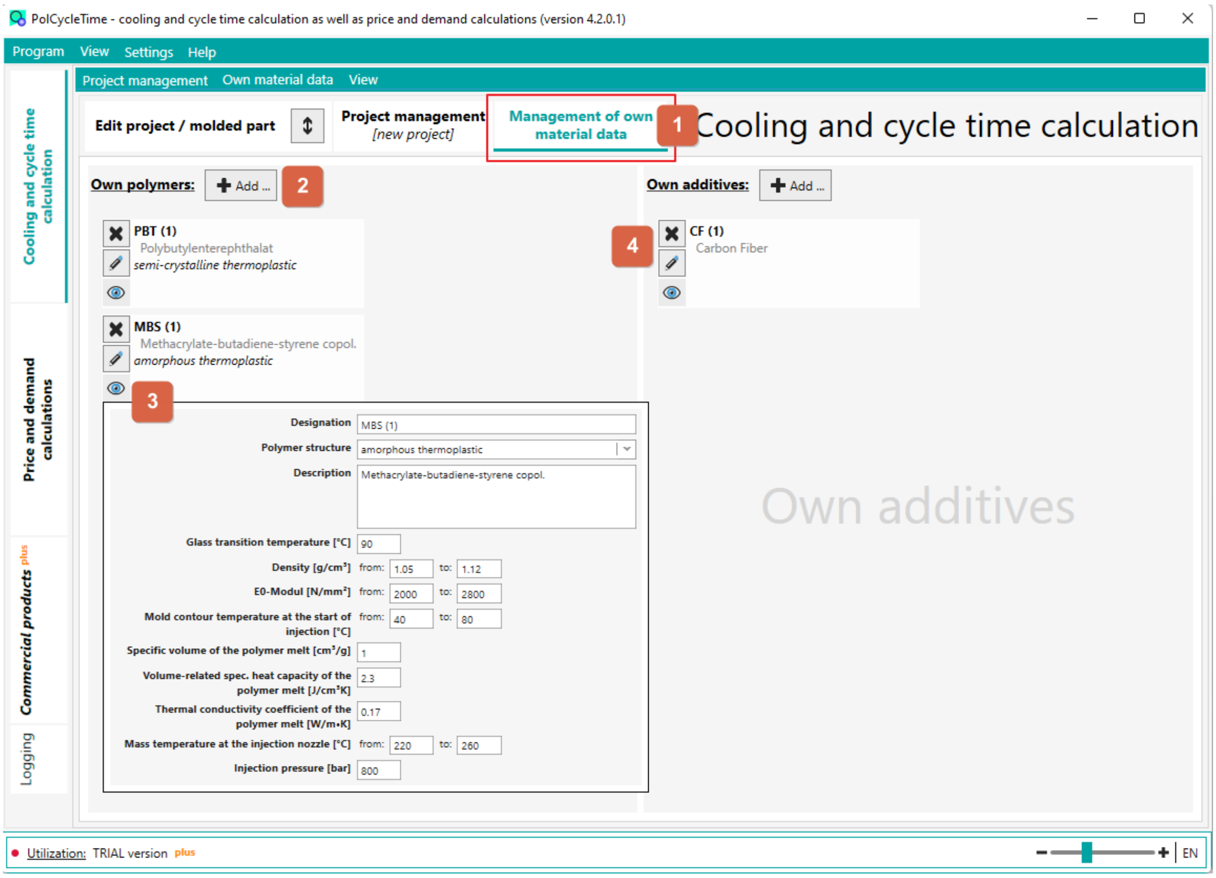
The following functions are available and can be called up via the buttons shown or the corresponding menu items:

1. Create new project. Your previous data will be lost
2. load an existing project. Your previous data will be lost.
3. save the current data. At least one project name must be entered to be able to save the project as a file.
4. save the current data under another file (file name)
5. display the entries and results in a print preview and print them out

Own material data

In addition to data from the included database, you have the option of using your own data.

In tab **1** you can see all the polymers and additives you have entered so far. Here you can manage all your own data clearly.



- via the button 2 you can add a polymer or an additive
- via button 3 you can display the current data in a popup window
- via 4 you can edit or delete the data afterwards

If the polymer or additive is used in the current project, then:

- the polymer/additive cannot be deleted
- in case of changes of data (which are important for the cooling time calculation) the results are updated immediately

further hints:

- When adding a polymer/additive, you can use the data from the database as a template
- The name of the polymer or additive must be unique. If necessary, you will be informed about this when you save the data.

FAQ's

Question:
In principle, how is the selection or assignment of the molded part geometries that determine the cooling time carried out?
Answer:
Injection molded parts are characterized by a wide range of geometric design variety and complexity. The program offers a wide range of options for describing the many geometric elements (molded part sections) of a molded part. Therefore, the user has the task to divide his concrete molded part into such sub-parts with sufficient precision. The part with the longest cooling time is then relevant for the cycle. This procedure can be quickly and reliably understood with the program, even by users with less experience.

Question:

When graphing the cooling time as a function of the mass temperature, a minimum of the cooling time was found at certain mass temperatures in individual cases. Is this physically correct?

Answer:

The body heat to be dissipated (enthalpy) plus heat of fusion is greater the higher the mass temperature. Therefore, a continuous increase in cooling time with increasing mass temperature is to be expected under otherwise constant conditions.

Within the software, however, the shear heat component, which extends the cooling time, is taken into account. This heat component is mainly generated by shear flow in the molded part edge zones during volumetric mold filling and is dissipated more quickly at higher melt temperature under otherwise constant conditions.

In individual cases, these opposing influences can lead to a mostly very flat cooling time minimum in the practical melt temperature range. The greater the mold contour temperature, the injection pressure and the surface/volume ratio of the molded part contour determining the cooling time, the greater the mass temperature of the cooling time minimum.

Question:

How reliably can thermodynamic molding data be calculated internally in the software for any combination of polymers and additives?

Answer:

The melt state is considered to be decisive for the cooling time calculation. Physically precisely defined addition rules exist for the basic data spec. volume, spec. heat capacity and heat of fusion. For the calculation of the thermal conductivity, many models of different complexity and complexity are offered in the technical literature. The calculation model selected internally in the software (not the Maxwell model) was successfully correlated with measurement results. The sufficient calculation accuracy for the thermal conductivity model is limited to approx. 60 vol. % additive content. A noticeable error margin can occur especially for crystalline additives with strongly direction-dependent material properties. Overall, a maximum deviation of ± 10 % is expected in relation to the cooling time calculation value.

Question:

According to which aspects is the demolding temperature determined internally in the software?

Answer:

The demolding temperature is one of the parameters with a major influence on the cooling time. Depending on the requirements, it describes an average temperature level of the molded part or its maximum temperature in the center of the cross section. These requirements are regulated differently for the sprue and the molded part. The molding-specific reference temperatures (glass transition temperature, melting temperature, flow temperature) are recorded in 3 different molding groups. For the molded part, the demolding temperatures are differentiated in 5-fold increments. The assignment is made taking into account the tempering conditions (additional cooling), the required manufacturing accuracy, the cross-section thickness, the molding material stiffness or hardness, the demolding conditions and the occurrence of sink marks. For sprue cooling, there is only a 3-stage allocation according to the demolding conditions and, for semi-crystalline polymers, additionally according to the cross-sectional thickness. It can be seen from the explanations that the demolding temperature is not a characteristic value that can be simply specified. Many users would be overburdened with the query of a fixed value.

Question:

Is a polymer type-independent quantification possible for the classification of the relative flowability of thermoplastic melts?

Answer:

Thermoplastic melts are structurally viscous and viscoelastic media whose viscosity (or fluidity) depends not only on the temperature but also on the magnitude of the mechanical stress on the melt due to shear and strain. A comprehensive description of the flow behavior is only possible using elaborate flow curves or viscosity functions. For a simplified characterization of the flow behavior, rheological single-point data of the melt are generated, which are currently mostly measured as volume flow rate (MVR) or mass flow rate(MFR) according to ISO 1133. Different test conditions are prescribed for each polymer type, so that a general comparison between different polymers is not possible. However, the molding compound manufacturers give an evaluation of the flow behavior within the respective type range (e.g. easy-flowing, normal-flowing, heavy-flowing), so that the software user is sufficiently informed.

Question:

Is there any additional guidance to the software help for estimating the flow resistance of the mold contours?

Answer:

The mold contours are filled from the nozzle (sprue) or gate (molded part) by a laminar swelling flow, provided that the melt does not solidify before the mold filling is completed. For a given flowability (viscosity) of the melt, the most effective flow path extension or shortening of the filling time is achieved by increasing the melt temperature and / or the injection pressure. Melt deflections have only an insignificant effect on flow resistance. Due to the thermal insulation of the solidifying surface layers, increasing the mold contour temperature only slightly reduces the flow resistance, but significantly increases the cooling time. Increasing the flow cross-section (e.g. by increasing the wall thickness) reduces the flow resistance disproportionately. With regard to the influence of wall thickness on the flow resistance of flat mold contours, the following wall thickness classification should serve as a guide:

- below 1 mm = very thin-walled
- 1 to 2 mm = thin-walled
- over 2 to 4 mm = normal-walled
- over 4 mm = thick-walled

With wall thicknesses below 0.5 mm, reliable mold filling is not always guaranteed, even with small flow path lengths.

Question:

Are further specifications possible for the selection of melt and mold contour temperature?

Answer:

The software's internal orientation ranges represent the most common temperature ranges in practice.

The melt temperature ranges are determined according to the thermal stability of the melt and the flow conditions of the mold filling, as well as according to the dependent part quality (surface gloss, weld line quality). If the melt temperatures are defined within the specified ranges, the cooling time is influenced relatively little.

The mold contour temperature, on the other hand, has a decisive influence on the cooling time. Low cooling time (high productivity) requires low contour temperatures, while special molding quality requires correspondingly high temperatures. The choice of the "correct" contour temperature is always a compromise which, with regard to the quality of the molded part, can only be decided by production trials. The practical experience of the user is very important here. Otherwise, the possible cooling time deviations must be tested by input variations, whereby the respective purpose of the cycle time calculation (e.g. cost pre-calculation, machine occupancy, machine setting) allows appropriate conclusions to be drawn.

A further specification of the temperatures is possible by communication with experience carriers (e.g. molding manufacturers, molding mass manufacturers).

Proponents of impulse cooling frequently claim, among other things, the advantage of a strong reduction in cooling time. In fact, this claim is neither theoretically nor practically proven, provided verifiable investigations are evaluated.

Question:
Can a quantifiable grade scale be specified for gloss evaluation of smooth molded visible surfaces?

Answer:
Corresponding qualities of the mold contour surfaces can be derived from design and functional requirements for the molded part surface quality (smoothness, gloss) as well as its manufacturability (demolding, post-treatment, coating formation). Their gradation is given below with the arithmetic average roughness Ra according to DIN EN ISO 4287:

- Technically rough (without fine machining): Ra ≥ 3 µm.
- Smooth (with fine finishing. e.g. grinding): Ra = 0.8 to 1.6 µm
- Glossy (coarse polish): Ra = 0.2 to 0.4 µm
- High gloss (fine polish): Ra = 0.05 to 0.10 µm
- Reflective (optical fine polish): Ra = 0.012 to 0.025 µm

When deciding on the surface quality, it should be borne in mind that the higher the polish quality, the higher the costs for manufacturing and maintaining the tools, while the accuracy of reproduction on the molded part surface decreases. The type of plastic and processing conditions can also have a significant influence on imaging reproducibility. In particular, the special optical grade (mirror-like surface) should therefore only be provided for transparent plastics in the case of special requirements (e.g. lenses, scales, CD).

Question:
Can the risk of stress cracking in thermoplastics be characterized in more detail for better assessment?

Answer:
Stress cracking (micro to macro cracking) occurs in all thermoplastics after simultaneous exposure to special media and correspondingly large tensile stresses or strains. This time-dependent physical process requires media with special interaction with the respective plastic (wettability, diffusion, solubility) and a critical tensile stress or strain state.

Amorphous thermoplastics of greater stiffness or hardness are particularly at risk, since their critical crack initiation strain can be considerably less than 1%, depending on the media influence. Polystyrene is an extreme example of this, since microcracking ("silver shimmer") is already possible at elongations below 0.3 % when exposed to air. In the injection molding process of these plastics, the lowest possible residual stresses (e.g. through higher mold contour temperatures) should be aimed for, since cracking is already possible after demolding without additional tensile stresses.

Which media are particularly susceptible to stress cracking for the respective plastic must be explored by those with experience (molding compound manufacturers, molding part developers, technical literature).

Question:

What conditions must be particularly observed when injection molding optically active molded parts (e.g. lenses)?

Answer:

Molded parts for optical applications require very low residual stresses (density constancy) and long-term holding pressure effects to avoid sink marks in relatively thick-walled parts. Both conditions are achieved, among other things, by high mold contour temperatures. For this purpose, a special orientation for high contour temperatures is specified in the program, which is only slightly below the glass temperatures of transparent thermoplastics.

The "plate" or "shell" is to be selected as the geometry element for lenses and the like, and average wall thicknesses are to be estimated from the courses of the radii of curvature.

Typical injection molding defects

Defect type	Defect appearance	Typical cause
Sink marks	Depressions on the part surface	Uneven shrinkage
Burn marks	Brownish discoloration of the part surface	Excessive melt temperature or too long residence time
Air streaks (air marks, air traps)	Dull/brownish/silvery/white streaks in wall thickness transitions and gates	Air in cavity is overrun and trapped by the melt
Moisture streaks	Large dull, streaky areas	Moisture in the cavity or in the granulate
Color streaks	Uneven coloration of the part	Insufficient mixing, incompatible masterbatch
Glass fiber streaks	Dull, rough part surface	Flow-induced orientation, too rapid cooling/shrinkage
Air inclusions (bubbles)	Voids on or in the molded part, near-surface bulging	Air not vented / injected, too low holding pressure, too high/fast decompression
Voids / vacuum cavities	Airless cavities in the molded part	Insufficient shrinkage compensation
Gas inclusions	Bubbles on/in the part surface	Material decomposition, excessive residence time or shear
Weld lines	Optical/mechanical weak spot (notch or color change visible)	Flow fronts meet with insufficient pressure/temperature
Gloss differences	Excessive, insufficient, or uneven gloss on the part surface	Different cooling or shrinkage conditions
Jetting	Snakelike, rough/dull line on surface – from the gate	No fountain flow, too high pressure, sudden cross-section

		enlargement, vertical mold
Records effect	Finely ribbed surface – concentric, parallel to flow path end	Melt temperature, injection speed, or mold temperature too low
Dull gate area	Dull halo (concentric rings) around gate	Gates too small, injection speed too high
Incomplete filling	Outer contour not completely formed (usually far from gate)	Premature freezing, too low injection pressure/speed, poor venting
Diesel effect / burn spots	Burn marks and unfilled areas	Insufficient venting
Flash	Film-like edge beyond intended contour	Inaccurate cavity, low viscosity melt, low clamping force, high cavity pressure
Stress whitening / cracks	Whitish discoloration, oriented cracks	Exceeding deformation/stress limit
Ejector marks	Bulging, gloss differences, markings	Processing, geometry, thermal or mechanical causes
Demolding deformation	Cracks, breaks, overstretching (typical for undercut parts)	Part cannot withstand demolding forces
Drag marks	Damaged (often textured) surface	Low mold rigidity, excessive shrinkage, insufficient draft angle
Surface flaking	Material layers peel off	Additive incompatibility, contamination, moisture, inhomogeneity, high injection speed
Cold slug	Comet-shaped mark near gate	Cooled melt from nozzle or hot runner enters cavity
Stringing	—	Insufficient cooling in gate area
Black specks / dark spots	Dark spots on surface	Mold wear, thermal degradation, contamination
Deposit formation	Gloss and surface topography defects	Low-molecular deposits on mold surface
Plating defects	Blisters, pimples, poor adhesion	Often caused by molding process/handling
Defective film hinge	Stress whitening, torn or cracked hinge	Material overstressed during demolding, frozen-in mass

Warpage	Deviation from intended shape	Shrinkage differences or demolding deformation

Price and demand calculation

Functions of this module include, among others:

- Calculation of unit costs
- Determination of tool requirements
- Analysis of machine utilization
- Calculation of machine hourly rates
- Determination of economical lot sizes
- Definition of the optimal number of cavities

... for the ***production of injection-molded and compression-molded plastic parts***.

The help for this program module is described in an additional help file, which is also part of the software installation. The help can also be accessed via the program's main menu and is also available as a PDF download on our homepage.

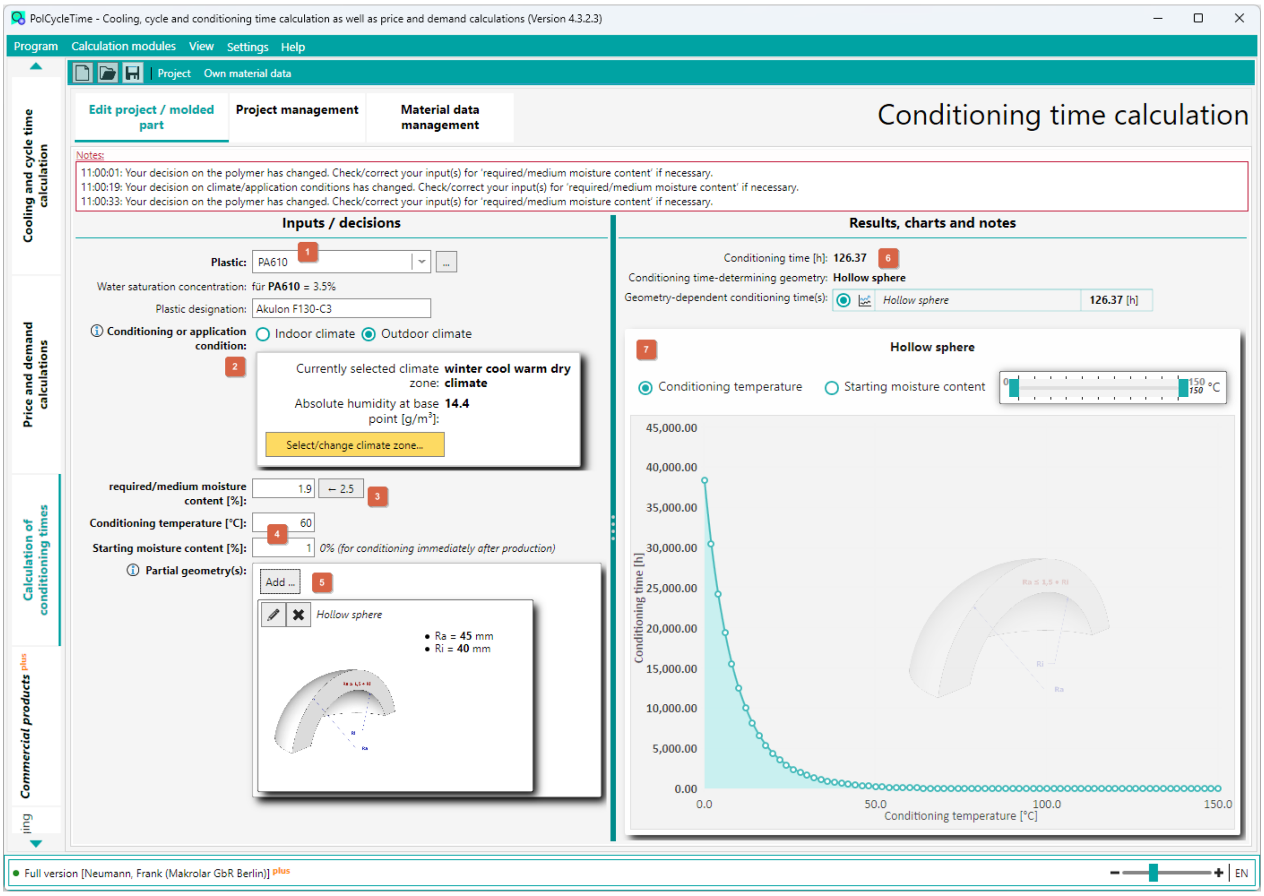
Conditioning time calculation

- Introduction
- Calculation concept
- Operation

Quick start

1. Select a plastic (1)
2. Define conditioning or application conditions (2)
3. The reference value for the desired/average water content is displayed and can, if necessary, be adopted or refined (3)
4. Define conditioning temperature and initial water content (4)
5. Mentally "divide" your molded part into relevant partial geometries. Add the corresponding geometry and enter the geometry parameters (5)

- As soon as a calculation is possible, the result is displayed (6)
- In addition to the results, interactive dependency diagrams for conditioning temperature and initial water content are shown (7)



Introduction

The water absorption of plastics influences application-determining properties with increasing water content as follows:

- Increase of dielectric losses.
- Decrease of stiffness and hardness.
- Increase of toughness (impact strength; elongation at break).
- Increased dimensional change due to swelling or drying.

The maximum water absorption of plastics up to saturation varies greatly. For non-polar plastics (e.g. polyolefins) it is often negligible, whereas for hydrophilic (polar) plastics (e.g. polyamides) water absorption must be taken into account for application-related purposes. In such cases, in order to achieve an application-specific required water concentration, post-treatment is carried out by conditioning of the plastic parts. Diffusion processes in solid materials proceed much more slowly compared to heat conduction processes.

Conditioning times also increase disproportionately (quadratically) with the cross-sectional thickness of the parts. In order to keep the post-treatment times as short as possible, conditioning is carried out in temperature-controlled hot water baths (40 °C to 90 °C) or in saturated steam systems (above 100 °C).

The conditioning temperatures must be matched to the thermal stability of the plastics. Depending on the type, amount and distribution of additives as well as on processing-affected material structures (e.g. degree of crystallinity), short-term conditioning processes may cause negative effects (discoloration, deposits, warpage, cracking). A countermeasure would be the less aggressive conditioning with warm humid climate in climate chambers.

Calculation concept

Unsteady mass transfer by water vapor diffusion is described with the same mathematical formalism as unsteady heat transfer by thermal conduction. In this context, concentration fields are calculated in the same way as temperature fields, with the material parameter *thermal diffusivity* being replaced by the *diffusion coefficient*. At the surface of the part, a saturated water vapor layer is always assumed, so that the mass transfer resistance can be neglected.

The calculation approaches of the software assume a diffusion-controlled mass transport. Capillary transport processes due to material porosity are not taken into account. Plastics with high proportions of solid additives (e.g. above 50 vol%) may, due to insufficient bonding between matrix and additive, possibly not satisfy the condition of pure diffusion.

The solution for water diffusion in plastics was already published in 1959: *Jouwersma, C.: Die Diffusion von Wasser in Kunststoffe. Chemie-Ing.-Techn. (31), 1959/10, pp. 652–658*. In this article, the author also critically discusses the approximation formulas commonly used up to that time (Eulitz; Blank).

Operation

The most important operating concepts are described below. The software is largely self-explanatory and, where necessary, help and hints are displayed at the appropriate points.

- Project management
- Edit project
- Material data management

Project management

- Projects are managed as individual file(s).
- Within project management you can:
    - Create new projects
    - Load existing projects
    - Save projects under a different file name
    - Display the data in a print preview and print it

In order to save a project, at least a project name must be provided. You can specify this in the Project Management tab.

Otherwise, the functions of project management are also available via the menu and the toolbar.

Edit project

Switch to the Project/Part tab (**1**) to edit.

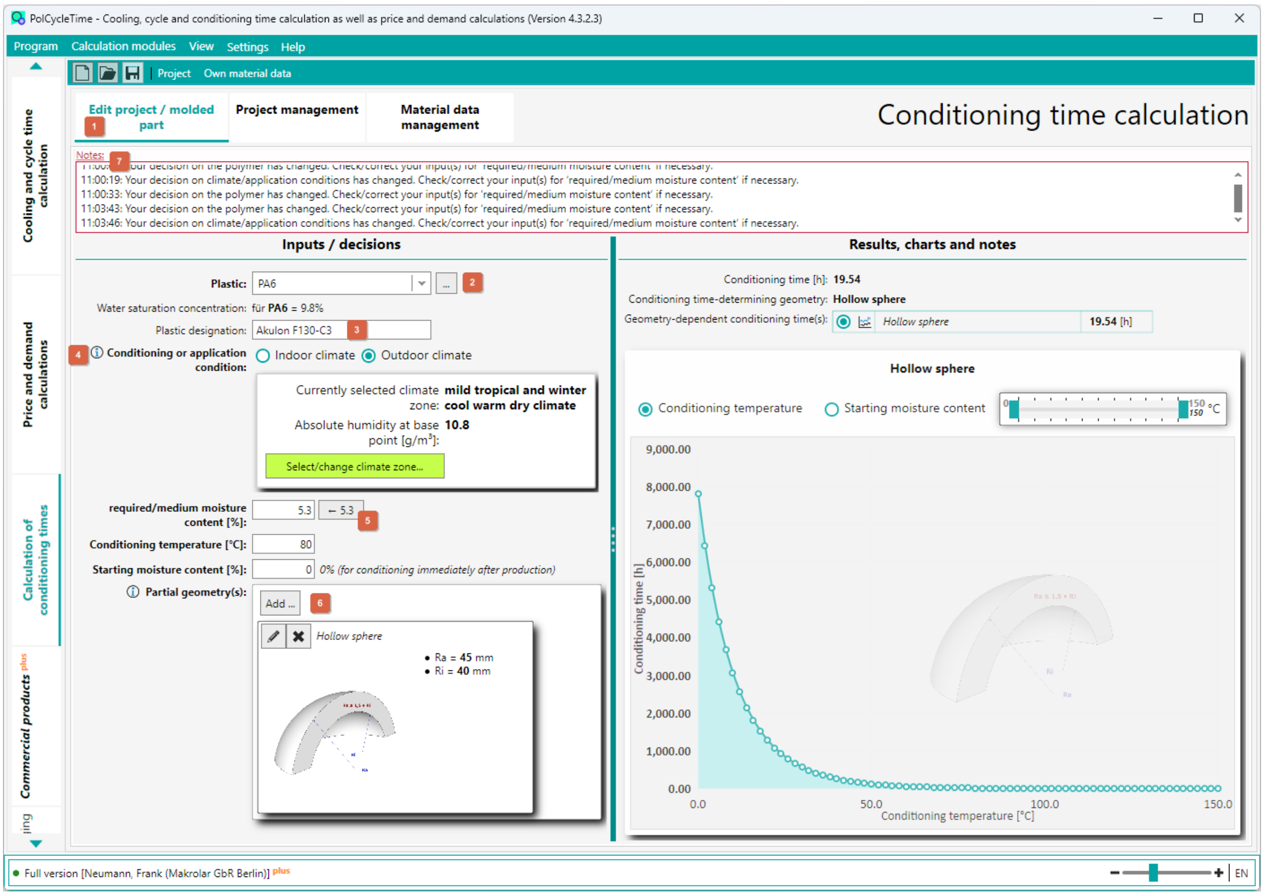
Please first read the quick start information.

- A plastic can be selected either via the selection box or through a material selection window (**2**).
- The plastic designation is optional and serves only for better documentation (**3**).
- For conditioning or application conditions you can choose between indoor climate and outdoor climate (**4**). In the case of outdoor climate, an additional window appears where you must select the climate zone by mouse click.
- Once you have decided on conditioning or application conditions, a calculated reference value (**5**) for the desired/average water content is displayed. You can adopt this by clicking the button or use your own value.
- Enter the values for conditioning temperature and initial water content.

- Add the relevant partial geometries (6).

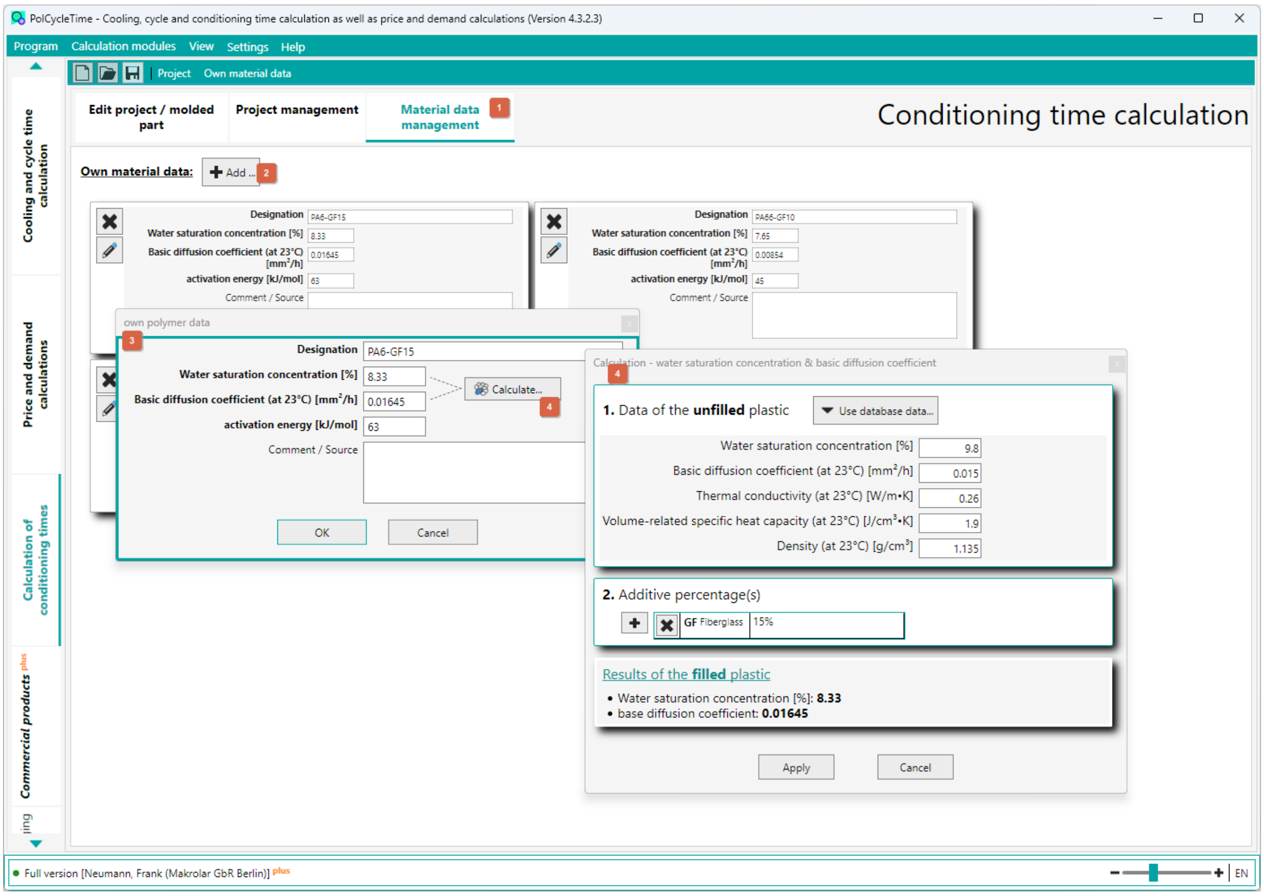
On the right side of the tab, the results (as far as possible) or notes on missing or incorrect inputs are displayed.

In the (higher-level) register, additional notes (7) are displayed if you subsequently change inputs.



Material data management

- For managing your own material data there is a corresponding tab (1).
- Click Add (2) to enter the data for a new polymer.
- Another window opens (3).
- Define the designation for the material and all other required data. For water saturation concentration and base diffusion coefficient there is an additional calculation module (4).



Calculation modules

In addition to the main modules of the software, the following calculation modules are currently available.

- Residence time calculation
- Melt density calculation

If project-specific data are available, they are automatically used as inputs.

Residence time calculation

- Three-zone screws for plasticizing units in thermoplastic injection molding are assumed.
- As the discharge volume per injection stroke, 1.5 times the calculated screw displacement volume is used. This displacement volume can be taken from the machine data for the injection unit currently in use (ISO).
- The numerical values of the cycle time and the specific volume of the melt are required for the calculation.
- For molding compounds with additives, the melt density can be calculated.
- For the calculation of the required shot weight, the number of mold cavities and the part mass are necessary. It should be noted that, for family molds (multi-cavity molds with different parts), each associated part group is considered as one part.
- Production-related material losses and runner system mass fractions are included approximately.
- For the evaluation of unacceptable thermal material degradation, the ISO recommendation of a limit of approx. 6 minutes must be observed.
- Reference values for the permissible residence time can be obtained from the injection molding machine manufacturer and/or the material supplier.

<u>Examples for permissible residence time</u>:

Plastic	**Permissible residence time [min]**
PP	5...8
PE-HD, PE-LD	5...10
PS, SB	6...8
ABS, SAN	5...8
PC, (PC+ABS)	4...7
PA6, PA66, PA11, PA12	4...6
POM	3...5
PMMA	4...6
PVC, PVC-HI, PVC-P	1...3
PPS	5...10
PEEK	5...10
PBT, PET	5...6
TPE, TPU	4...8

If the module is used in the context of cooling and cycle time calculation, existing project data are automatically adopted as inputs.

Melt density calculation

- With this module, the melt density for molding compounds with additives can be calculated.
- For this purpose, data from the internal database are used, among others:
  - Specific volume of the polymer melt
  - Density of the additive

You select the polymer and add one or more additives (specifying the respective volume fractions).