GENIUS TOOLS®

INNEO® That's IT.

GENIUS TOOLS Parameter

Version 11.0.0.0

Description of all functions

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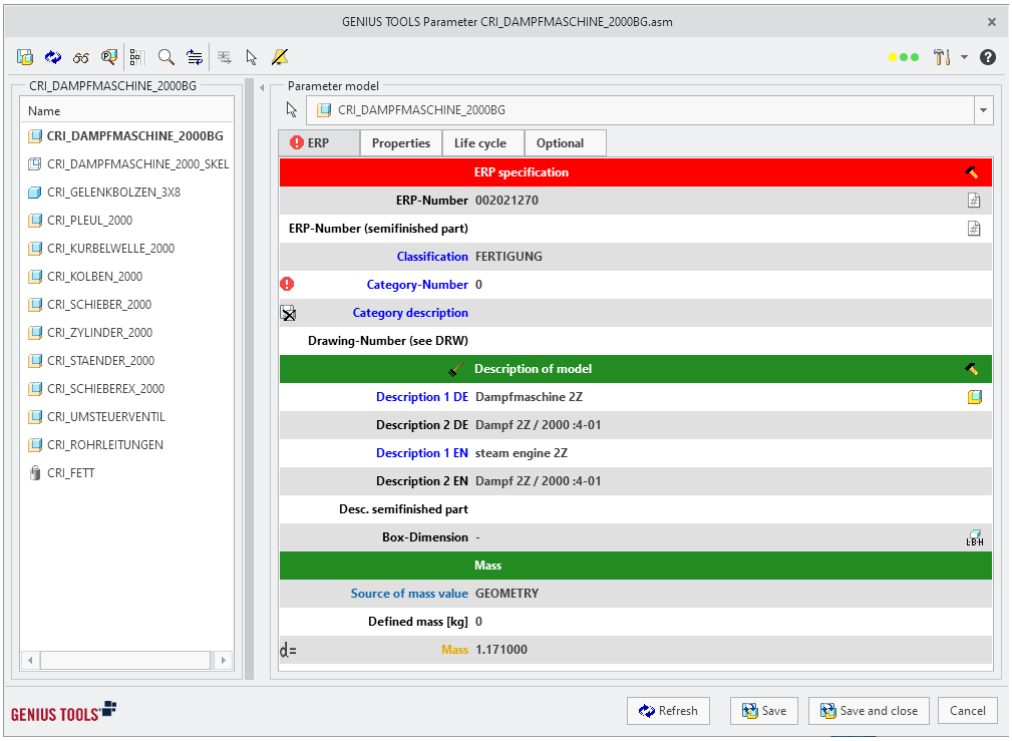
I. GENIUS TOOLS Parameter

GENIUS TOOLS Parameter contains the following components.

GENIUS TOOLS Parameter is part of the product package Startup TOOLS.

1. Parameter management („Parameter“)

The component *Parameter* is particularly suitable for creating uniform meta data and using it for the automated generation of parts lists, as well as for preliminary calculations or for a connection to commercial systems. The graphical user interface can be configured for the user, for example to define mandatory input fields as to ensure the completeness of the master data and a uniform, up-to-date database.



The following functions are available:

- classifiable meta data definition including
  - a mechanism for auto-selection of parameter definitions
  - freely configurable groupings
- easy creation of parameters
  - with default properties for all types
  - specification for effective ranges (parts, bodies, assemblies and drawings)
- easy editing of parameters by means of

- free input
- lists and tables (from files and databases)
- auto-suggestion function
- format checks
- input dependencies
- adoption of parameters from other Creo models

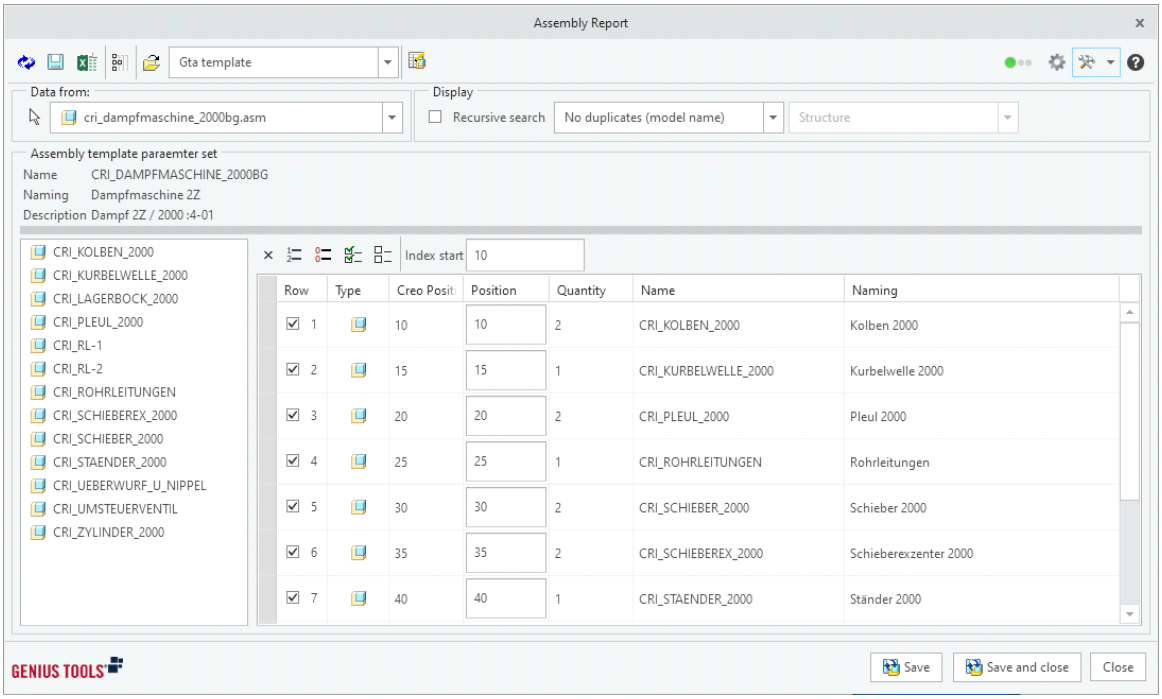
– individual Creo model tree configuration

2. Bills of materials in assembly mode („Assembly Report“)

The component *Assembly Report* allows users to create reports, such as a bills of materials (BOM), in assembly mode and generates item numbers as component parameters for further use in Windchill, Creo View and Creo drawing mode. BOM and other reports can be customized for each individual assembly using a graphical editor.

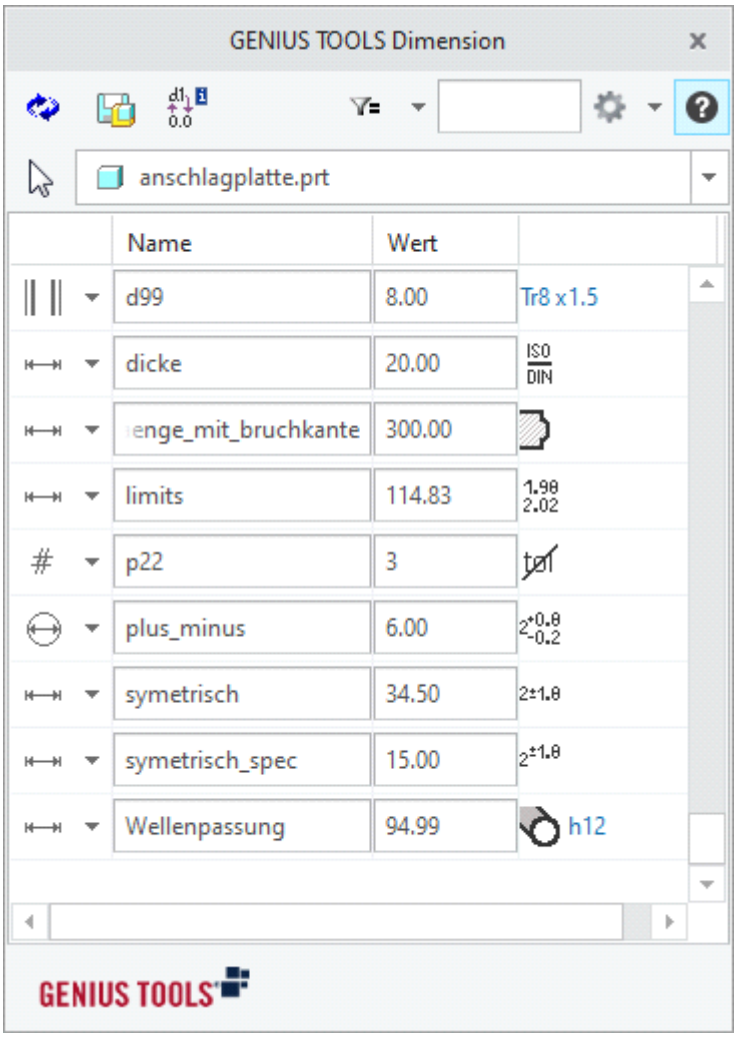
The following functions are available:

– display of tables with different display modes, e. g. single-level, multi-level and bulk bill of materials
– definition of variable columns, such as
  - assembly and part parameters
  - assembly component parameters
  - report parameters (file name, assembly level, model type, quantity, mass etc.)
  - position numbers (as assembly component parameter)
– assignment of position number (if used in BOM)
  - Start and increment value, multiple number ranges, manually editable
– multiple filtering and multiple sorting by all parameters
– export of reports to Microsoft Excel (with template) or as CSV file



3. Multi-dimensional editing („Dimension“)

The component *Dimension* allows simultaneous and fast editing of dimension values and names of a feature, a part, an assembly or the subcomponents of an assembly, as well as variable UDF dimensions.

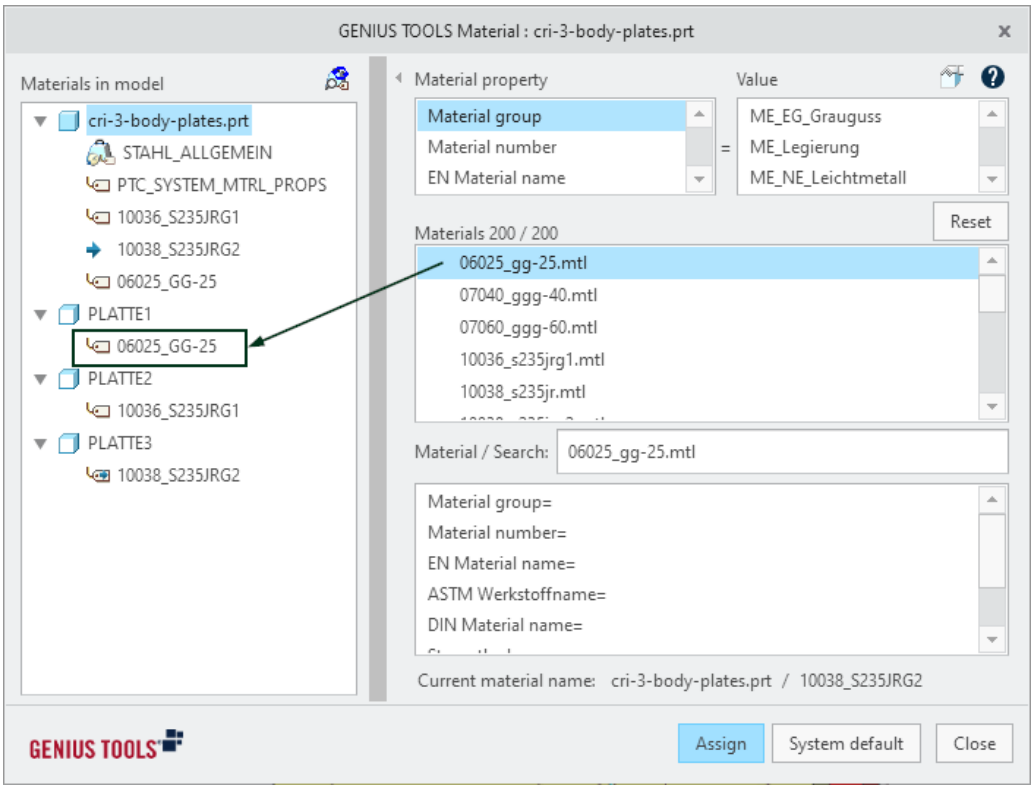


The following functions are available:

- Displaying and modifying dimensions with properties: dimension type (linear, angle, diameter, radius) dimension name and value, tolerance type, dimension status (e. g. in relations, family tables)
- Filtering the displayed dimensions by name, dimension type and tolerance type
- Free text search for dimensions including auto-suggest function
- Highlighting dimensions in the graphics window when selecting a value in the *GENIUS TOOLS Dimension* user interface
- Renaming of dimensions
- Links for quickly accessing the Creo ribbon menu *Dimension* and the Creo dialog *Relations* (for relation-driven dimensions)
- Quickly assigning dimensions to family tables
- Saving the values as a CSV-file

4. Material selection („Material“)

The component *Material* allows users to select materials based on various properties and assign them to a model or body.

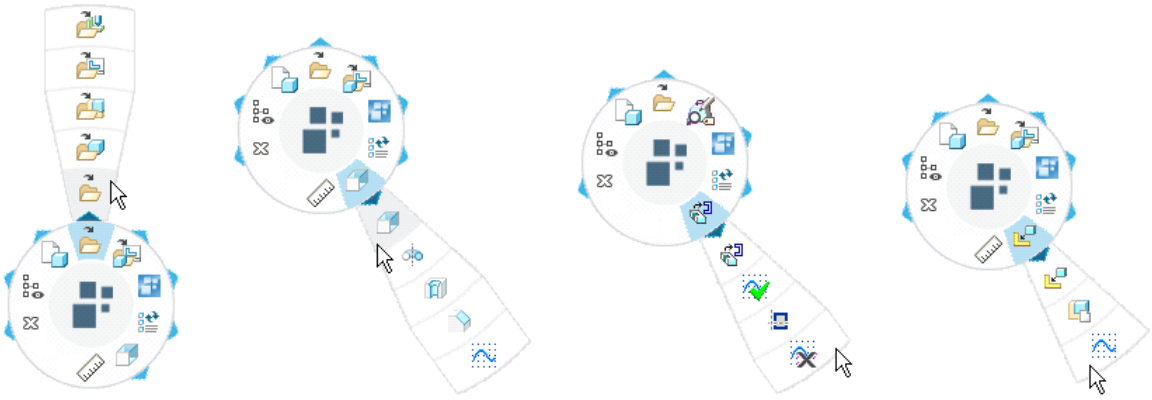


The following functions are available with the graphical editor for material selection:

- make material files from the PRO_MATERIAL_DIR directory uniformly available to all users.
- manage materials with material attributes and associated values
- adjustment and representation of the material data, e. g.
  - output in different languages
  - provision of extra information (documents, URL)

5. Ring menu and mapkey management („Quick Access")

The component *Quick Access* is a ring menu that provides quick access to suitable commands in different Creo modes and can include individually configurable mapkeys (macros).



The following functions are available:

- Using regular and intelligent mapkeys, i. e. using variables, parameters and placeholders
- Defining commands depending on mode and selection
- Different usage scenarios:
  - Central configuration
  - User-specific configuration
  - Simultaneous central and user-specific configuration
- Easy-to-use graphical editor for a homogeneous operating environment
- Exporting and importing all customized mapkeys with images and descriptions for easy data exchange

6. Transferring model properties („Value Transfer“)

This component can be used in assembly mode to change numerous values in dimensions and parameters as well as material definition files of assembly components in one step.

The following functions are available:

- search for assembly components with optional filters and view search results in a clearly laid out table format
- display of the current parameter values for each assembly component before each value change
- fast transfer of an assembly parameter (e. g. project number) to all assembly components

7. Name Generator

The component *Name Generator* assigns names with sequential numbering for file names of parts, sheet metal parts and assemblies. Name Generator can be used both individually on stand-alone workstations (local) and in a network (global).

8. Editing assembly parameters

This function generates component parameters in assemblies. Different component parameter values can be assigned for component models with the same name.

9. Converting multibodies into assemblies („Multibody to Assembly“)

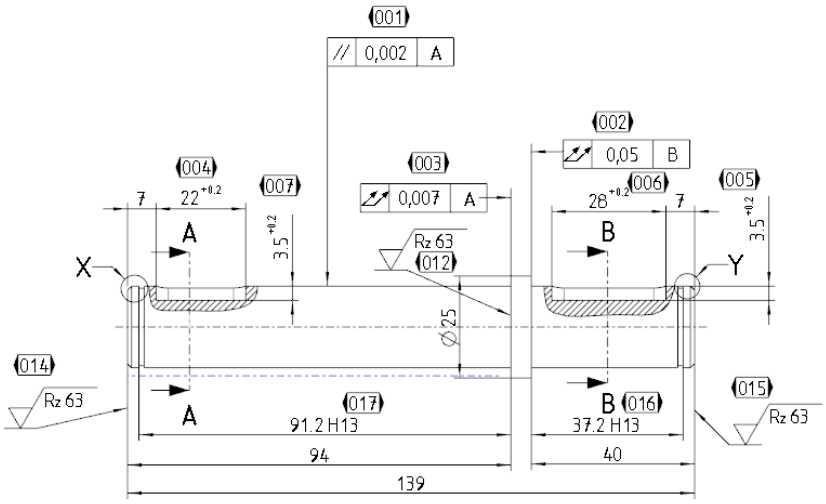
This function allows you to transfer parts that have been created with bodies into an assembly structure. Every created part includes an external copy geometry feature that references exactly one body.

10. Open / create drawing

This function opens a drawing, if a drawing with the name of the model already exists, or creates a drawing.

11. Inspection and change symbols for drawings („Inspect“)

With the function Inspect you can place, number, and manage inspection and change symbols on drawings and you can also store all versions of inspection symbols and can create a revision history of all symbols.



Automatic numbering of inspection symbols

The following functions are available:

– placing inspection characteristics freely or link them to dimensions, shape and position tolerances, notes etc.

– free placement of tables that show the properties and numbering of all inspection symbols
– clean documentation of all changes on a drawing with the Inspect Revision dialog
  · creating a snapshot of all inspection characteristics on a drawing at a specific point in time
  · defining a revision level of a drawing with a drawing revision parameter and display a history of all revisions

12. Export table to EXCEL and CSV

These function fills a file template with parameters and data from table cells of a Creo drawing. An Excel report, a CSV report or a PDF report is written.

13. Create tolerance tables on drawings

This function creates a tolerance table at a freely selectable location on a drawing using pre-defined tolerances.

14. Javascript Editor

With *Javascript Editor* you can develop and test JavaScript code in several GENIUS TOOLS components.

15. Configuration Utility

Configuration Utility is an interface for editing all configuration options and saving them to the correct locations.

The following functions are available:

– Viewing, modifying, commenting and deleting individual configuration options for each level
– Quickly checking different configuration variants using the memory function of variants

16. Further useful tools („Utilities“)

16.1 3D Note Form

Enables quick modification of dimension and parameter values in the notes on the model via editable form masks.

16.2 CS Assembler

Automates the assembly of components into an assembly using defined coordinate systems.

16.3 Export Points

Outputs reference points (single points or point fields) or dynamically generated curve points (X-Y-Z values) to a PTS or DAT file.

16.4 Extend Relations

Adds more functions to model relations that can be used to create parameters for models and bodies.

16.5 Full Backup

Quickly saves the current model with all dependent data.

16.6 Load Save Converter

Saves Creo objects from previous Creo-, Wildfire- or Pro/ENGINEER versions in the currently used version.

16.7 Open Base Model

Opens geometric base models that are the reference source for a feature.

16.8 Select Surfaces by Color

For selecting surfaces of the same color or all uncolored surfaces.

16.9 Show Information

Creates company-specific information in text form and displays it in the Creo Parametric main window.

16.10 Work Dir Manager

Automatically collects all directories used during the work process and enables the current working directory to be changed quickly.

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