

KOMPAS-3D v25

Version information

Differences of version 25 from version 24

General

- Implemented the administrative management of the KOMPAS-3D system parameters located in the **System** section of the general configuration dialog. In the administrative mode of operation, you can mark the parameters that cannot be changed in the normal mode (i.e., when the user is working). Users will be able to change parameters not marked by the administrator as before, at their own discretion.
- Added the option to opt out of saving a list of recent documents and displaying this list on the Start page. Subsequently, the list can be saved and displayed again, if necessary.
- When resetting the settings, it became possible to reset the system settings, i.e. bring the settings made in the **System** section of the general configuration dialog to the original ones. Previously, this was achieved by using a profile *default.pfl* (when using a dark theme — *default_dark.pfl*). The specified profile files are now excluded from the package.
- When filling of the title block on the document sheets in the tables, vertical text can be entered into the cells. Previously, there was no corresponding command in this process.
- Added the ability to quickly navigate to the technical notes settings in the document — using the command in the menu **Layout — Technical notes** and in the context menu of the technical notes.
- The possibility of hosting KOMPAS-3D Help systems on a web server in the enterprise's local network has been implemented. This allows to get help without installing it at every workplace and at the same time does not require an Internet connection. The steps that must be performed to ensure that help is called from the local network are described in the KOMPAS-3D Help System.

3D-Modelling

- The **Midline** command has been added, designed to build a curve in the model, which is the geometric location of points equidistant from two flat curves. To execute the command, two curves and the plane on which they will be projected are set. The midline is created in the same plane.
- The command **Change the position of the component** has been added. It allows you to change the position of a component in space in various ways:
 - setting the position and orientation relative to the selected CS,
 - an offset in a given direction by a certain distance or a rotation around a given axis by a certain angle,
 - for components involved in mating **At distance** and **At angle**, moving according to the mating parameters; this method also allows you to simulate the movement of a component using animation.

To change the position, you can select several components at once.

As when using the **Move Component/Rotate Component** commands, the mating applied to the movable component can completely or partially limit the change in its position and lead to the movement of other components; it is also possible to control collisions of the movable components with the remaining components of the assembly.

- The **Relief** command has been added, which allows you to create a **relief** pattern on the edges of a body by projecting closed contours (3D curves, edges, sketch lines) onto them and then adding or removing a layer of material within the boundaries of the resulting areas.
- Added the **Segmentation of a polygonal object** command, which splits a polygonal object into segments. A segment is a connected set of triangles characterized by the relative constancy of the curvature at the vertices or the angle between adjacent triangles. The

command allows you to select sections in a polygonal object that are similar in shape to analytical surfaces (plane, cylinder, sphere, cone, torus).

- Added the **Select — By Properties** command. It allows you to select objects of various types in the model that have specified characteristics. For example, you can find components of a certain size and having specified properties in an assembly, or curves of a certain type lying on a specified layer, or operations of a certain type having variables with a specified value, etc. You can set either one condition for selecting objects or several at once. In order for conditions to be connected by the logical operator «AND», they should be set in one block of conditions. The blocks are connected using the «OR» operator.
- The **Update version of selected features** command has been added, which allows you to rebuild the specified operations using the new algorithms available in the current version of KOMPAS-3D.
- The **Rounding surface** command has been improved:
 - Added new ways to determine the width of the surface:
 - **Tangent to surface** — the rounding surface is constructed so that, in addition to the source surfaces, it also touches the surface specified by the user,
 - **By boundary curves** — the rounding surface is constructed so that its boundaries coincide with the curves specified by the user that lie on the original surfaces.
 - Added a new way to set the function type to change the parameter values — **By values at points**. In this method, the user specifies points on the line of movement of the section, sets the required parameter value in each of them, and determines the law of parameter change between the points: ruled or spline interpolation.
- It is now possible to create a special type of variant in the model — a variant of the geometry composition. The geometry composition option is designed to isolate a part of the geometry of the source model in order to load it in isolation from other objects in this model. Using specially prepared composition options in complex parts can significantly reduce RAM consumption and improve computer performance when working with large assemblies.
- The KOMPAS-3D system setting has been added, which prescribes the add-on of options to the components of the assembly when loading it and determines the priorities of these options. For example, in models of fasteners, variants of the geometry composition can be created, representing these products in a simplified form: without chamfers, rounding, holes. If you set this simplified version as a priority during configuration, you can reduce the opening time of the assembly (in proportion to the number of fasteners inserted), as well as obtain an assembly drawing with the image simplifications provided by the standard.
- The object selection processes have been improved and refined:
 - Improved work with filters:
 - It is now possible to set your own filter sets for each command, which involves specifying model objects (while the filter list is limited to the types of objects used in the command), and for the state when no command is running. All generated filter sets are saved until the end of the document session.
 - Added an indication of the object filtering mode: if at least one filter is enabled, a special icon is displayed in the upper-right corner of the graphic area. Clicking on it disables the filters of the current set (the other filter sets, if any, do not change).
 - When specifying source objects in various commands, it is now possible to automatically select groups of curves and faces that meet a certain condition. For example, you can select a chain of curves and edges by specifying one of the links in this chain; by specifying a face, you can select it and all adjacent faces or all other faces created by the same operation, etc.
The commands defining the object selection condition are available during the operation on the Quick Access Toolbar and in the context menu.
The objects selected by the condition are combined into a set. The set contains information about the initiating object and the selection condition. Due to this, when other objects that meet the condition appear in the model, they are automatically added to the set, changing the result of the operation using it accordingly. If necessary, the set can be edited, for example, to change the initiating object or condition, exclude some objects from the set, etc.

- The **Equidistant to Curve** command has been improved: when constructing an equidistant with an offset along the surface, you can specify not one face, but several. The possibilities of cropping equidistant edges and extending to the edges of the edges have also been added.
- Improved the **Projection Curve** command:
 - The command is now working with points and groups of points. It is also possible to project curves and points simultaneously.
 - In the **By direction** method, you can enable the creation of curve sections parallel to the projection direction. This allows you to eliminate breaks in the curve in cases where among the specified faces there are those that cannot be projected onto, for example, flat faces parallel to the direction of projection.
- Hole creation commands now have the ability to create multiple holes at the same time (previously, only one hole could be created in one command call).
 The position of the holes is determined by the points that the user sets. These can be individual points, as well as groups of points: arrays by points, sketches with points, etc.
 The direction of construction for each hole is perpendicular to the surface passing through the corresponding point. If the points already lie on the surface, then it is determined automatically, and if not, then the surface should be specified manually. In addition, you can set a single direction for all holes by specifying an object, for example, a straight line.
- The processes **Distance and angle**, **Edge length**, **Area** have been improved:
 - the «dimension» object created by these processes can now be edited by specifying other model objects to perform the measurement;
 - a standard **Create Object** button has been added to the process, completing the current measurement with the creation of a «dimension» object in the Tree (the **Create Dimension** button that formed this object earlier is excluded).
- The **Convert to a polygonal object** command has been finalized:
 - The conversion accuracy setting has been added: the user can control the maximum linear and angular deviations and the maximum edge length.
 - The polygonal objects resulting from the conversion are now associatively linked to their source bodies and surfaces: if they change, then the polygonal objects also change. If necessary, the conversion operation can be destroyed — then the polygonal objects will lose their connection with the source ones.
 - It is now possible to edit the operation **Conversion to a polygonal object** (available if the operation is not destroyed).
- The **Model Family** command has been improved:
 - It is now possible to add new columns to the existing family table by specifying properties and variables of the template model that have not yet been included in the table (previously, this resulted in the formation of a new empty instance table in the same file). The added columns are placed after the existing ones. They are filled in, as before, when editing the family table in a tabular editor.
 - The template of the family can now be a model containing variants (previously it could not). The same options will be present in the instances created using this template.
- When editing the auxiliary plane, it became possible to change the way it was constructed: the heading of the Parameters Panel displays the same buttons for creating planes as when constructing a new plane. After clicking the appropriate button, you should specify the reference objects of the plane according to the selected construction method. The connection of the plane with the objects dependent on it is not disrupted as a result of such editing. The properties of the plane that were set during creation are also saved.
 This feature — changing the construction method — has also become available when editing auxiliary axes.
- It is now possible to select the direction of the normal of the auxiliary plane during its creation or editing.
- In the **Surface trimming** command, it is now possible to set a tolerance for the accuracy of joining curves or surfaces forming a section object. This allows you to combine several specified curves or surfaces into one continuous section object, provided that the gaps between them are less than the specified tolerance value.

- In the process of copying geometry, it became possible to automatically copy threads built on the elements of the copied body.
- The **Surface by grid of curves** command has been improved: the Save direction option has appeared, which is available if the curves of the V direction are not set, but some kind of coupling condition is set.

Later, when changing the shape and/or location of the interface objects, the enabled option provides a reconstruction of the surface based on the fixed mutual direction of the normals/tangents.

- Improved diagnostics of 3D models: it has become more accurate. However, in some cases, this may result in an increase in the number of messages issued. Error messages that do not block the construction of the model, but require the user's attention, are highlighted in a separate category — warnings. Examples of warnings: a message stating that an element does not intersect all or some objects of the add-on area, a message about the self-intersection of the body, a message about a violation of the integrity of the body. Warnings are displayed in the Tree as an exclamation mark on a yellow background. If necessary, you can disable the display of warnings.
- Improved work with properties of model objects:
 - it is now possible to set default display parameters and a default layer for model bodies and surfaces, as well as for assembly components:
 - sub-assemblies,
 - parts,
 - library components,
 - standard parts (only a default layer can be set for them).
 - For curves that may potentially include several contours (outline, projection curve, etc.) and groups of points, the default layer setting has become available.
 - During the process of inserting a component (other than a standard part) into the assembly, the configuration of the display parameters of this component became available. Previously, these parameters could only be changed after the component was inserted.
- When working in the sketch, the Layers Panel displays, in addition to the Sketch Layer Tree, the Model Layer Tree. It allows you to change some of the layer parameters, in particular, properties. For example, you can make the background layer active. Creating and deleting model layers is not available when working in a sketch.
- When making measurements in sketches, it became possible to specify 3D objects located on the background layers of the model.
- A sketch containing blocked dimensions (control dimensions which value cannot be changed due to constraints or other dimensions) receives the status «overconstrained» and the corresponding icon (!).
- The ability to view the path to the copy source document and open this document has been added to the copy management process.
- It is now possible to open a contextual link source document. The corresponding command is called for the model object associated with the assembly by a contextual link and opens this assembly itself, i.e. the document where this link occurred during contextual editing of the model.
- The default value of the *Material* property for new assembly models has been changed: now the default value of this property is not set. Previously, the material of the new assemblies was Steel 10 GOST 1050-2013. The density value for the new assemblies has not changed — it corresponds to steel (0.007856 g/mm³).
- When selecting dimensions in the Model Tree, their reference objects are highlighted in the graphic area.
- Accelerated creation of the rounding surface. The greatest effect is noticeable when both source surfaces are polyhedral and the number of faces is significant.
- The process of changing the current variant or version of the component in the assembly has been accelerated.
- Accelerated generation of family instance files.
- The data structure has been optimized in models with a complex surface shape, which also contains many other objects based on this surface, such as points or holes. As a result, the

amount of RAM required to work with such models and the amount of disk space required to store them have been reduced.

- Highlighting is implemented in the graphic area of the object highlighted in the one-line logger on the Parameter Panel during command execution. This makes it easier to control the selection of source objects in cases where all or most of the registrars in the command are single-line. Previously, the backlight worked only for multiline recorder objects.

Graphic documents

- It is now possible to limit the area of the model projected into the drawing. This area is defined in the drawing by two planes parallel to the projection plane of this type: the first passes through the point of the model closest to the observer, the second is located behind the first. If the view is a section, then the first plane coincides with the secant plane. The distance between the planes is set by the user, either by entering a numeric value or by specifying the position of the second plane on another projection of the same model.
- In the list of model objects, which is shown for an associative view in the Drawing Tree, you can leave only the visible ones, i.e. the objects actually depicted in this view. Objects that are missing from the image, such as those that are blocked by others or are located outside the local view, are excluded from the list. This makes it easier to work with drawings of assembly models containing a large number of components.
- Added the «symmetry» constraint, imposed on objects by the new **Symmetry of objects** command. To impose a constraint, you need to specify an axis of symmetry and two objects. Objects can be individual elements (segment, circle, etc.) or composite objects in their entirety (rectangle, groove, etc.). The types of specified objects must match, but there are some exceptions, for example, a segment and a centerline, circle and arc can be symmetrical. As a result of the constraint, the objects are positioned symmetrically relative to each other relative to the axis and get the same dimensions.
- The processes of setting linear dimensions with outrigger lines tangent to circles/arcs of circles have been finalized:
 - the **Linear dimension** command now allows you to select the **Vertical** and **Horizontal** types for these dimensions (previously they could only have a type **Parallel to Object** and, accordingly, show the minimum distance between circles, the maximum distance or the distance between the centers),
 - the **Linear dimension from a line to point** command now allows you to select a circle/arc as a source object, and also the buttons of setting dimension selection appeared: from the segment to the nearest point of circle, the furthest point or to the center.
- Improved commands for creating arrays and copying graphic objects:
 - To facilitate access to the array editing command, it has been added to the context panel that appears when an array or its individual objects are selected.
 - Trimming or dissecting a segment, circle, or arc of a circle that is part of an array no longer excludes this object from the array. In the case of dissection, one of the resulting parts of the object remains in the array.
 - The command **Array by grid**:
 - added the ability to set the directions of the grid axes by specifying straight-line document objects (when working in a sketch, you can also specify straight-line model objects); the array retains its connection with the specified objects and rebuilds when their position changes;
 - added options for array construction: with deletion of copies inside the grid and with placement of copies only along the axes.
 - added the ability to enter/display expressions for calculating array parameter values in the corresponding fields of the Parameter Panel.
 - In the **2D Linear pattern** command, the controls in the Parameter panel are reorganized to resemble the Parameter panel in the **Array by grid** command.
- New features for marking majority surface finish: enclosing the marking in parentheses and adding an «asterisk» after it. If necessary, you can edit the field containing the «asterisk» by entering any text. These features are also available when creating a majority surface finish marking in the model.

Note. The ability to add a text label to the roughness designation, which was previously used to enter an «asterisk» or other text after the designation, has been eliminated.

- New features for working with macroelements:
 - The command to exclude geometry from a macro element has been added. This command moves selected objects from a macro element to the document without changing their position, and it works both inside and outside the macro element editing process.
 - Objects that are inserted into a graphic document from the clipboard can optionally be combined into a macro element during insertion.
- The presence of zero-length segments in a document is no longer detected as an error. These segments may appear, for example, as a result of editing a parametric image.
- In the associative drawing, it is now possible to create links to the properties of the assembly components that are projected into this drawing.
- When working with parameterized graphic documents, it is now possible to enable/disable the display of variables in dimension captions. This can be done using the same command and key as when working with 3D models: **Names of variables in dimensions**, <F11>. (The command **Display constraints** no longer affects the display of variables in graphic documents and sketches).
- Dynamic highlighting, which highlights an object when the cursor passes over it, now also occurs when there is no running command and when the object is snapped. The dynamic search delay setting has been added, which allows you to specify the time (in milliseconds) that the cursor must remain over an object before it is highlighted.
- When you set up a line style that contains a fragment, you can enable the addition of dashes at the beginning and end. This ensures that the line with this style starts and/or ends with a stroke, and a multi-segment line, such as a broken line, does not have gaps between segments. Previously, there might have been a gap at the beginning and/or end of the line (segment), which reduced its visual length.
- Now, when you create an associative dimension, the dialog box for setting its value does not appear if the value cannot be changed, i.e., if the size is informational (previously, the dialog box always appeared).
- If you hover over the icon of a restriction applied to an object, a tooltip with the name of the restriction appears on the screen.
- Parallel processing of associative views is excluded.

Working with product BOM, properties, and reports

- It is now possible to manage versions in the Product BOM Panel: the user can edit the designation of the selected version (the base part and the version number), add or delete a version of the product. When working with assemblies (*.a3d), it is also possible to change the current version of the model in the Product BOM Panel.
- Components that are part of a subassembly that is represented in the assembly BOM by its constituent parts can now be excluded from the product and from the current BOM style (previously, all components of such a subassembly were displayed in the BOM).

Working with BOM

- When configuring the BOM style, you can now link graphs to properties (system properties or properties from libraries). In a BOM with this style, the cells of these graphs will be automatically populated with the values of the corresponding properties, if they are defined in the documents connected to the BOM objects.
- Improved management of the sending of properties from a connected document to BOM: the selection of the connected document from which BOM should receive properties has become more convenient, and the indication of this document in the list has become more visible. The control of the set of properties sent from the document has also become more convenient.

Text editor

You can now manage the process of inserting fragments and raster images into a document from the Parameters panel instead of using a dialog box.

Import and export

- The ability to save graphic documents in format SVG.
- The export of BOM command has been added to configure the BOM for export to Excel 2007-365 (*.xlsx), Excel 97-2003 (*.xls), and Calc (*.ods) formats. The configuration defines the list of exported sections, object types, and graphs. The specified parameters can be saved to a file for later use.
- When importing models in proprietary formats, you can now configure how to read the text in dimensions, captions, and other model objects. The following options are available:
 - **Import text as geometry** — converting symbols into groups of geometric objects (segments, arcs, etc.),
 - **Match texts to system ones** — replacing fonts in the source file with the font set for dimensional captions in KOMPAS-3D models,
 - **Set character correspondence** — setting the correspondence between the special characters of KOMPAS-3D and the codes of these characters in the source file.
- When reading assemblies from the STEP format, it became possible to select from where the component data should be extracted: from the attributes of the inserts or from the sources of the components.
- The possibility of exporting the specification to the format IGES (*.igs, *.iges) is excluded.

Add-ons

1. Equipment: Pipelines

- There was a command to build rolling rolls on pipes according to GOST 13954-74, as well as rolling rolls of non-standard sizes.
- Added ways to build a pipeline:
 - **Orthogonal** — along the coordinate axes,
 - **By points** — continuous construction of a pipeline along an arbitrary trajectory.
- The possibility of constructing a pipeline at a given distance from the surface has been implemented.

2. Equipment: Metal structures

- A single- solid switch appeared in the **Plate** command. It affects the result when the sketch contains several contours: if the **Construct as one solid** is enabled, then the plate is solid of merged parts, and if disabled, then several separate solids. By default, building with one solid is disabled.
- The command **Trim/Extend profiles** and finalized: the **Surfaces** logger in the parameters has become a group logger, which allows you to trim/extend a group of profiles with several surfaces. The **Trim/Extend profile** command has been deleted.
- The **Profile** command has been improved: now profile building can be performed in two opposite directions. For each direction, you need to select a method for determining the length and parameters, as in the **Extrusion Element** command, which is part of the basic functionality of KOMPAS-3D.
- New features for working with arrays:
- For ease of operation, the **Array, copying** panel has been added to the toolbox of the **Metal Structures** add-on, copying from the standard set of KOMPAS-3D panels.
- The objects of metal structures obtained as a result of creating an array can now be edited using the add-on.
 - Added the ability to create arrays of solids.

3. Nesting

The **Nesting** add-on for solving material cutting tasks is included. The add-on provides the following features:

- Import of assemblies in KOMPAS-3D format with automatic generation of part contours for nesting.
- Loading into the add-on the contours of parts sorted by material and thickness in FRW format.
- Taking into account the direction of the fibers (structure) of the material.
- Automatic creation and editing of a cutting map in accordance with the selected parameters.
- Assignment of waste material in manual and automatic modes.
- Calculation of material nesting efficiency.
- Formation of reports.

To use the Nesting add-on, a separately paid license is required.

4. 3D shafts & mechanical gears

- A mechanism has been implemented for the automatic creation and automatic editing of points, lines, and outline surfaces (hereinafter referred to as outlines) of 3D model steps. This design approach allows you to edit 3D models directly in KOMPAS-3D, based on essays. At the same time, rebuilding the model by applying **3D shafts & mechanical gears** does not lead to a loss of links in the 3D model.

When working with an assembly, essays can be used to create matings.

- Added the ability to design and calculate a new type of mechanical gear— **External helical cylindrical gear pair with crowned teeth**.
- A new element has been implemented on the cylindrical stage — **Long slot**. It can be made in 4 variants: prismatic, trapezoidal, triangular and semicircular.
- In the calculations of the cylindrical transmission of internal engagement and planetary gear transmission, the formation of blocking contours is implemented when selecting the offset coefficients.
- The *Shaft3D.Rtw* add-on has been reinstated, allowing editing of models previously created in this add-on. Creating new elements using this add-on is no longer possible.
- The *Thread3D.rtw* add-on has been added to the delivery package in order to restore the ability to create cylindrical elements with metric threads directly in the 3D part.
- In the calculation of flat-belt transmission, more stringent restrictions have been introduced on the arbitrary assignment of pulley geometry data.

5. Split connections

- Added **Save Templates** and **Open Templates** commands. They allow you to save templates (configurations) of various connections generated during assembly work to an XML file and then load them into another assembly. This allows the add-on to adopt split connection sets from previously designed products in new products.
- Links to the corresponding instances or groups of the POLYNOM:MDM reference book have been added to all dialogs for selecting connection elements (bolts, screws, studs, washers, and nuts)

Changes in the package

The package includes the **Ergonomics: Manikins** add-on. The add-on allows, using mannequins (models in the shape of the human body), to analyze human interaction with designed products or workstations in order to improve their usability and safety.

Purpose of add-on commands:

- Panel commands **Manikins**
 - **Manikin** — allows you to create a polygonal **Manikin** object in the model with specified anthropometric characteristics. The pose of the mannequin is selected from **Pose Library**.
 - **Placement** — allows you to edit the position and orientation of a previously constructed manikin in the model.
- Panel commands **Pose change**
 - **Segment control** — allows you to change the position of individual segments of the manikin corresponding to body parts.
 - **Manipulation** — allows simultaneous repositioning of multiple segments while respecting anatomical constraints and kinematic dependencies. For example, by

moving a segment of the hand, the position of the forearm can be changed, as for a bent arm.

- **Change pose** — allows you to replace one mannequin pose with another, including a custom one from **Pose Library**.
- Panel commands **Service**
 - **Pose Library** — Allows you to manage the mannequin pose library and view its options. The library can contain both system and custom poses.

A separately purchased license is required to use the **Ergonomics: Manikins** add-on.