

KOMPAS-3D v23

Version information

General

1. When creating a reference to an object's property, the selection of a constituent part of the product that is not visually represented, but is merely a set of properties, became available. Previously, it was impossible to select such a component and, consequently, create a link to its property.
2. The sequence of selected document templates can now be manually altered in any way.
3. In the dialog **About the program** a button **More details** has been added, allowing you to view the list of installed KOMPAS-3D modules (KompasFlow, Shafts and Mechanical Transmissions, Materials and Assortments, Standard Items for KOMPAS, Advanced Tools for Reading Proprietary Formats, APM FEM, Unit Converter, Application Development Tools) and their versions. If the modules are not installed, then the **Details** button is missing in the dialog.
4. For KOMPAS-documents saved in KOMPAS-3D v23, the version number in Windows Explorer is displayed not in two, but in four groups of digits, for example: 23.0.0.1929.
5. The Microsoft Access Database Engine component is no longer installed with KOMPAS-3D, whether in a full or custom installation. If necessary, it can be installed manually by running the corresponding file from the `\Support` folder of the installation kit.

3D Modeling

1. A new command has been added — **Rounding Surface**, which creates a transition surface between two specified surfaces. The transition surface is tangent to these two surfaces along their borders, which belong to these same surfaces. Various options are available for adjusting the shape of the transition surface and trimming its lateral sides.
2. A new command has been added — **Remove solid/surface**. It deletes selected solids and/or surfaces from the model. With this command, you can remove, for example, the results of intermediate constructions that are no longer needed. In addition, this command allows the deletion of solids/surfaces that are not required for work from the imported component.
3. Commands for editing the dimensions of object edges and surfaces have been added:
 - **Change fillet size**, allowing to edit fillets with a constant radius by increasing or decreasing the radius value. Automatic selection and corresponding modification of other fillets whose edges are tangentially connected to the edge of the edited fillet is available.
 - **Modify face size**, allowing you to edit the diameter of a cylindrical or spherical face. These commands are mainly intended for modifying imported models, i.e., without construction history, but can also be applied to models with history. Commands **Change Fillet Size** and **Modify face size** are located on the new **Direct Modeling** panel in the **Solid Modeling** set. The commands have also been transferred to this panel: **Delete Faces** and **Move Faces**.
4. Added a new command — **Replace faces**. It allows you to replace the selected facet of a solid or closed surface with a new facet. The new face represents a section of the surface of the object specified as the replacement. The replacing object can be a face or a plane; the new face can coincide with it or be offset by a specified distance in either direction. The faces adjacent to the replaced face are extended or trimmed to keep the solid/surface whole. The **Replace Faces** command is also located on the new **Direct Modeling** panel.
5. New command added — **Replace components**. It allows you to specify a new source file for one or multiple (including different) assembly components. When specifying components

subject to replacement, you can include a search in the assembly of components with the same source as those already specified. It is possible to replace considering the versions/variants of the models: for this, the new source must have versions/variants with the same numbers as in the previous one.

During the replacement of identical components, correspondences between initial and new elements can be established. Thanks to this, after replacement, the operation of existing mates in the assembly will not be disrupted, and the operations performed in the assembly will not lose the objects they were based on.

6. The new command has been added — **Deviation Analysis**. It is used to assess the deviation of a polygonal object from a solid, surface, or face. Result The analysis is displayed as a color map. The maximum and minimum deviations are also displayed.
7. The capability of automatic creation of new models based on a template model has been implemented by changing the values of variables and properties specified in the template. Value sets can be extracted from tables *.ods or *.xls.
8. The capability to construct solids in the form of a parallelepiped, a cylinder, or a sphere according to specified parameters has been implemented, i.e., without using initial objects (sketches, axes, curves, etc.).
9. A new command for changing model orientation has been added **Normal with alignment** — an enhanced version of the command **Normal to...** The **Normal with Alignment** command sets the orientation in two steps: first, the model is brought to a position where the specified plane becomes parallel to the screen, and then it is rotated around the normal to this plane so that a straight line belonging to this plane occupies a horizontal or vertical position. A line lying in a plane can be defined by indicating a plane intersecting it or a line projected onto it.
10. Processes for constructing cylindrical and conical spirals have been revised:
 - To place the spiral, you need to specify a point in space and orientation. The orientation of the spiral essentially involves setting the orientation of the coordinate system with its origin at the specified point; the methods of setting it are the same as those used when constructing the local coordinate system (LCS). The direction of the spiral is determined by the Z axis, and the initial angle is counted from the X axis. Thus, indicating a flat object is no longer required, and a sketch with the spiral anchoring point is not formed. Additional features when placing the spiral are:
 - use the base element to set the base point and orientation of the spiral in the graphic area using your mouse,
 - specifying an edge at an arbitrary point—then the reference point of the spiral becomes the point of specification, and the Z axis is directed along the normal to the edge,
 - Indication of a circle/arc — then the center becomes the base point of the spiral, the Z axis is directed along the normal to the circle/arc plane, and the diameter of the spiral equals the diameter of the circle/arc.
 - Added the ability to build a spiral axis, provided that the spiral height is explicitly set, i.e., **By distance**. The axis is created as an object *Axial Line*; in the model's Design Tree, the spiral axis is displayed as its subordinate object.
 - Added the possibility to select a coordinate system relative to which the parameters of spiral placement are specified (coordinates of the base point and Euler angles in case of selecting this orientation method).
 - Variables controlling the coordinates of the base point of the spiral and Euler angles have been added, if the orientation is specified in this way.

When opening documents created in previous versions of KOMPAS-3D, the spirals contained in them are automatically converted to the new version format, i.e., the listed options become available when editing them.

11. The Command **Spline by Objects** now allows the creation of a spline that is associatively linked with the source objects, enabling it to be rebuilt when those objects are modified. If required, you can obtain a spline without linkage (as previously) — to do this, you need to enable the option **Scatter** during its construction or call the command **Explode** for an already constructed spline by objects.
Aside from the associative link, the command **Spline by objects** includes automatic selection of curves that are tangentially connected to the curves already specified for construction.
12. The enhanced **Curve of intersection** command: now an object in it can be a solid or a surface as a whole. Thanks to this, the intersection curve is correctly rebuilt when the number of faces of the solid/surface changes (previously, when selecting the solid/surface, the faces were added to the list of objects, which resulted in new faces not being considered, and the disappearance of faces could lead to an error in the intersection curve). In a similar way, the command **Intersection Objects**, which creates an intersection line of the sketch plane with model objects, has been refined.
13. New features when using the object copying operation:
 - During the editing of a previously performed operation of copying objects in the model, the ability to set correspondences between elements of the original and new operation results has been introduced. The initial result is the set of objects that were in the model at the start of editing, and the new one is the set that appears as a result of editing. Thanks to the establishment of correspondences, other model objects based on the original result do not lose their reference objects.
For example, a sketch was created on a copy of the auxiliary plane, but later the source file of the copy needed to be replaced. In the process of editing the copy operation, you can establish a correspondence between the plane copied from the former source and the plane copied from the new source. As a result, the sketch will not lose its flatness, but will "transition" to a new one.
In the process of setting correspondences, you can view the current result, i.e., rebuild the model with account of the correspondences already set. This allows for visually controlling the model modifications, as well as obtaining information about any remaining errors within it.
Thus, by setting correspondences, one can reduce the number of errors arising in the model after editing the copy operation (and in simple cases, avoid them altogether), and, consequently, decrease the time spent on refining the model after such editing.
Note. The ability to set correspondences is also available when editing operations of extrusion, rotation, by path, and by sections.
 - The deletion/exclusion of the source objects of the copying operation from the calculation does not lead to the deletion/exclusion of the operation itself. Instead, the copy operation is marked in the Design Tree as erroneous. The result of the copying remains in the model and is available for use by dependent objects. Prior deletion/exclusion from the calculation of copying sources resulted in the deletion/exclusion of the copying operation.
 - When copying curves, axes, surfaces, and planes, there has appeared the possibility to assign the resulting copy the reverse direction compared to the original (in the case of surfaces/planes — the direction of the normal).
 - Copying entire objects, such as control/connecting points and sketches, has become available. As the result, the copy represents an object of the same type, which can be used accordingly. For example, if you copy one or several curves of a sketch, the copies will be curves, but if you copy the entire sketch, the copy will be a sketch.
14. Enhanced command **Surface Fitting**: now, during the process of constructing a surface approximating a segment of a polygonal object, it is possible to specify a guiding object and adjust numerical parameters (if any exist). For example, when creating a cylindrical surface, you can specify a plane to which the cylinder axis should be perpendicular, and input the cylinder radius value.

15. A setting has become available for the component to place its objects on assembly layers: either the component as a whole (i.e., all its objects regardless of the level) resides in one layer, or the component objects reside in the same layers as in their source files. At the same time, if the assembly lacks the corresponding layers, they are automatically created. Placement on source layers can be useful for managing the visibility of objects or their color using the corresponding layer parameters to which they belong.
16. Enhanced handling of layers:
 - Implemented Layers Tree. The button **Layers** on the Tree Panel is used for its display. In the Layers tree, all the functionality for working with model layers that was previously available in the Document manager is accessible. In the Sketch, the Layers Tree displays the layers of this sketch and allows working with them.
 - New features have been added:
 - A property of the layer called **Activity** has appeared, which has two values: **active** and **background**. The feature of the background layer is that objects lying on it are unavailable for selection in the modeling window. The exceptions are the operations of measurements and diagnostics. With objects on active layers, you can work in the usual manner.
 - The option to disable the display of empty layers in the list is now available.
 - The layer number can now be changed by the user.
17. Expanded capabilities of the command **Mutual deviation of two surfaces**:
 - added the possibility of measuring distances between points of two curves and between points of a curve and a surface, as a result of which the command was renamed to **Mutual Deviation**; the use of a plane as a measurement object has been excluded,
 - measurements in method **According to the normal to the object** are now performed according to the normal to the second of the specified objects, and not to the first one,
 - added measurement of the absolute minimum,
 - the ability to change the order of objects during calculation has been implemented,
 - measurement results are displayed in the table on the Parameters Panel and in the graphic area.
 - saving measurement parameters (list of objects and settings) in the document has become available; the corresponding object is placed in the Model Tree under the section **Diagnostics — Deviation Checks**,
 - The command has been removed from the group of measurement commands (**Distance and Angle, Edge Length, Area**) and placed on the **Diagnostics** toolbar separately.
18. Three methods of placement are implemented in the command **Placement of the sketch**:
 - **The method on the plane** corresponds to the primary way of allocating a sketch used previously, i.e., allocating a sketch on the designated plane or a flat face.
 - **On the trajectory** — a new method involves placing a sketch on a plane perpendicular to the specified curve at the selected point. This point is the origin of the sketch coordinates.
 - **Fixed** — corresponds to the fixed state of the sketch in previous versions, i.e. it cancels the link of the sketch with reference objects and fixes its current position in space.
19. Updated the process of building an element by path:
 - The capability to change the shape of the section during its movement along the path has been implemented. The shape of the section can be modified through the geometric links of the sketch with other curves or surfaces (via the objects of their intersections with the sketch plane), as well as by changing the sketch variables selected by the user according to specified laws.
 - In the sub-process of creating a sketch, initiated from the process of building an element along the path, the method of creation is automatically selected **On the trajectory**, a curve is selected as the path, and this same curve is used in this capacity during building an element. This approach simplifies and accelerates work since it does not require creating a separate plane for the sketch.

- Added the ability to launch directly from the process of building an element along the path—the command **Broken line** for creating an element path.
 - The requirement that an open path must start in the sketch plane has been excluded—it's sufficient for the path to intersect this plane.
20. Refined command **Measure distance and angle**:
- Added display in the graphic area of the measured angle between objects.
 - Results of measurements are now displayed in the table on the Parameters Panel. The selected measurement in the table is displayed in the graphic area as a segment (or angle) and numerical value.
 - Enhanced clarity of result presentation when a direction is specified for the measurement: in the graphical area, additional objects are displayed alongside the measurement result to illustrate how this result was obtained.
21. Refined commands **Developed curve** and **Collapsed curve**:
- In the command **Unfolded Curve**, you may specify not only curves, but also points belonging to the surface — as a result of unfolding, the points will be transferred onto the plane.
 - In the **Rolled Curve** command, the possibility to specify/construct a plane tangent to a surface has been added, thanks to which segments, points, and non-planar curves can be rolled onto the surface: first, these objects are projected onto the specified plane, and then they are rolled from it onto the surface.
22. Refined Model Tree:
- If the **Tree Composition** button, managing the display of Tree sections in its structural representation, has become an indicator of the presence of disabled sections: if they exist, the button is displayed as not pressed (with a light background). Clicking the button in this state enables all disabled sections, and the button becomes pressed (with a dark background). Also, the menu button now contains the following commands:
 - **Unfolding**,
 - **Technical specifications**,
 - **Collections**,
 - **Copies**,
 - **Operations outside sections** (manages the display of operations performed in the assembly and located at the end of the Tree),
 - **Excluded from calculation**.
 - Now there's a function to delete individual subsections of the section *Diagnostics* and the whole section at once, i.e., to quickly remove from the model the measurement results of the selected type and all measurement results at once. Previously, you could only delete objects within the subsections of the section *Diagnostics*.
23. When performing operations in an assembly, it has become possible to use sketches belonging to components, and when contextually editing components, to use sketches belonging to the assembly or other components. Working with such external sketches have some limitations: it is impossible to change the dimensions of the sketch and start editing it within the process where this sketch is involved, and also it is not allowed to use separate regions of the sketch (only the entire sketch can be used). Previously, the operation sketch had to necessarily belong to the same model in which the operation is performed.
24. The possibility of a special display of "smooth" ribs — the ribs between facets that are tangent to each other — has appeared. This allows for better visual representation of models with surfaces smoothly transitioning from one to another. The settings for style and color for "smooth" edges are available, as well as the specification of the maximum angle between the tangent faces at the points of the edge at which this edge is considered "smooth". Besides, you can disable displaying of "smooth" ribs in the model. By default, "smooth" edges are not distinguished from the others.
25. In the command **Select Nearest** it is possible to enable a mode where it will work only with first level components, i.e. components inserted directly into the current assembly. In this

mode, individual parts for which the conditions specified in the command are fulfilled are not selected, but the entire first-level components containing these parts are selected.

26. The process of constructing an element by sections has been refined: from it, you can launch the construction of a spline by points to create guiding curves.
27. In a sheet part, the option to retain holes formed during circular corner treatment on the flat pattern with simplified contours has been added (previously, this flat pattern option was created without the specified holes).
28. A method has been added for determining the mass-inertia properties of a model variant, in which all these parameters are taken from the main execution of the model. The selection method is available when creating a variant, as well as when configuring its properties.
29. In operations that utilize curves, it is now possible to select a curve represented (or potentially represented) by several separate contours as a reference object, for example, *Outline Line* or *Curve of intersection*. To do this, the object should be specified in the Design Tree. If an operation can utilize multiple contours simultaneously, all the contours of the curve are engaged. Previously, in such cases, you could only select a contour (contours) but not the entire curve.
30. When specifying the properties of the model, manual entry of axial and centrifugal moments of inertia (in the central coordinate system) became available.
31. Into the command **Object Info** added the possibility of displaying thread parameters.
32. Enhanced handling of model sections:
 - The management of cross-sections is now performed in the process on the Parameter Panel, not in the dialog box.
 - The **Display Model Section** button on the Quick Access Toolbar is equipped with a menu containing a list of sections and a command to launch the process for managing sections.
33. If the result of constructing an extrusion, revolution, path, or section surface consists of several parts, then each part is now created as a separate surface.
34. Improved the drag-and-drop of objects with the mouse in the Model Tree, particularly handling situations when the dragged objects need to be placed directly above or below an existing macro-element.
35. When working in the sketch, the settings of the current model have become available (previously, only system settings were accessible).
36. Automatic creation of auxiliary projections of model objects in the sketch is excluded.
37. Refined visualization of a sketch when shifting/moving it to another plane: such objects as texts, image inserts, and fragments are now displayed in the phantom shown during this process, if present. Previously, the phantom only included geometrical objects constructed in the sketch.
38. Accelerated model rendering for display types **Shaded Wireframe** and **Hidden lines removed**. In the general case, an acceleration of 1.5 times was achieved, and on individual models — up to 4 times. The greatest effect is observed when working with models that have a large number of components entirely located inside other components (inside gearbox housings, solids, etc.).
39. Accelerated rendering (and re-rendering) of the phantom when creating and editing surfaces.

Graphic documents

1. Added commands for creating grooves:
 - **Groove** — constructs a shape from two parallel segments of the same length, connected by arcs with a radius equal to half the distance between the segments,

- **Arc Groove** — constructs a shape from two concentric (large) arcs, connected by small arcs with a radius equal to half the distance between the large arcs.

Various methods are available to create a slot, as well as automatically place its centerlines and dimensions.

Commands **Groove** and **Arc Groove** are available both when working with graphic documents and in sketch mode during 3D modeling.

2. Such objects of a graphic document as the origin of the absolute coordinate system and the origin of view coordinates in applying constraints are used as ordinary points. For example, the coordinate origin can be associated with a segment constraint *Point at the middle of the curve*. The result of snapping the characteristic point of an object to the coordinate origin (when working in parametric mode) is the automatic application of a constraint on the object *Point coincidence*. Previously, the constraint *Fixed point* was applied.
3. The "behavior" of the parametrized image after editing is now retained to a significantly greater extent. For example, the result of segment splitting consists of new segments connected to each other by constraints *Collinearity* and *Point Coincidence* (previously, segments without constraints were obtained); the result of trimming by two points of a circle with an associated diameter
Arcs with associative diametrical dimension (previously the dimension was removed) etc. Additionally, the constraints applied to the endpoints of objects are retained, provided these points are not deleted as a result of editing (previously the constraints would disappear).

Working with Properties and Reports

1. Refined Product BOM panel:
 - There is now the possibility to transfer property values *Designation*, *Name* and *Note* from the product to its constituent parts. The selected property may be assigned to an individual component or to all components at once. For this, you need to invoke the command **Transfer to source** from the context menu of the individual cell or column header on the Product Structure Panel. The command is present in the menu if the property in the cell (or cells of the column) is not linked with the source.
 - Now you can configure position numbering in the assembly model. To generate/update position numbers, use the **Place positions** button on the Composition panel. It is present if the Column display *Position* is enabled on the Panel, and there are no associated BOM assemblies.
 - If a BOM (file *.spw) is attached to an item or a constituent part, the format of this document (usually A4) is automatically transmitted to the item/component. If the format transfer from another attached document, when it was enabled, is disabled, except in the case where the item/component receives the format from another BOM that was previously attached.
 - Enhanced layout of the KOMPAS-3D window when placing the product Structure Panel **At the bottom across the full width**: The panel no longer covers the lower part of the window where the orientation control element (on the left) and pop-up messages (on the right) are displayed. Now these interface elements are displayed above the Product Composition Panel. Moreover, the Panel no longer overlaps the additional Document Tree (if it is enabled), as the additional Tree now ends above the Panel.
2. The product composition panel became available when working with graphic documents.
 - In the drawing, the Product Composition Panel allows:
 - view and edit the properties of the document itself, as well as the properties of macro-elements, fragment insertions, and views,
 - create constituent parts without visual representation and set their properties,
 - View the properties of the model projected into the drawing and its components.
 - In the fragment, the Product Composition Panel allows you to view and edit the properties of the document itself.
3. A new system property *Item Name* has been added. It is intended for entering the name of the product itself and is used if the property *Name* should, in addition to this name, contain

something else. For example, the name of a part without a drawing also includes the designation of the material and other data necessary for its manufacturing.

4. The value of the property *Name* can now include a link and plain text or several links (previously — either plain text or a single link). Creating multiple references in the name may be required, for example, when entering information about a part without a drawing: its name is usually supplemented with the material designation and the dimensions needed for production (for example, length). These data can be included in the part name as references to the corresponding properties of the same part.
5. The value of an object's property can now be a reference to another property of the same object.

Working with BOMs

1. The option to simultaneously disable/enable grouping for all objects in the BOM is now available. The commands **Ungroup All** and **Group All** from the **Management — Grouping** menu are used for this purpose. After calling the **Ungroup All** command, the **Group** option is disabled for all objects, resulting in all previously grouped "duplicate" items in the BOM no longer being combined and each being displayed on its own line. Command **Group all** executes the reverse action.
2. If a BOM document (file *.spw) is attached to a BOM item, then the format of this document (usually A4) is automatically transferred to the BOM item. Transfer of format from another attached document, if it was included, is disabled, except in cases where an object receives the format from another BOM attached earlier.
3. The management of item numbers in the BOM has been reworked: a section for managing new item numbers has been added to the current BOM settings, similar to the one used when setting up an assembly. Moreover, the BOM settings now include a control for obtaining the version numbers of the product from the connected document, while the possibility of adding leading zeros to the numbers has been excluded (adding zeros is now done during the setup of the current BOM).
The section for managing the numbers of new versions of the item is also introduced in the drawing settings.
The result of these changes is more stable work of the set of linked documents (assembly, assembly drawing, and BOM), the possibility...
Settings for the digit capacity of version numbers of the product in the drawing and BOM, as well as the correct transfer of the digit capacity of the numbers specified in the assembly to the BOM (previously, in separate cases, manual adjustment of the version numbers in the BOM was required).
4. Revised the BOM Style Settings dialogs and Current BOM Settings dialogs. Changing the style of the current BOM is now available in the BOM settings dialog. Previously, the style selection was possible in the Document Tree and in the style parameters dialog.
5. 5. The dialog for managing specification descriptions has been revised, making it more informative and user-friendly.

Import and export

1. Refined and enhanced export of models
 - For setting up the export parameters, the process is now initiated by the command **File — Export** instead of using dialogs.
 - The ability to select model objects to be exported has been implemented (previously, it was possible to choose the type of objects to be exported, but not the objects themselves). Objects can be specified in the Tree or in the graphic area, including by using a frame.
 - Export using the command **File — Save As...** is still available, but configuring export parameters is not possible (the command **Save with Parameters** is excluded).

2. When saving the model in C3D, JT, and STEP formats, you can store tables.
3. When saving the model in C3D, STEP, and VRML formats, the option to enable/disable thread recording has been added.
4. The possibility of reading NX and SolidWorks CAD system models without installing the component ***Extended Reading Tools for proprietary formats has appeared.***

Add-Ons

1. Detachable joints

- New type of detachable connection added – a studed connection, created by the ***Studded connection*** command. The command allows inserting studs with a set of fastening elements (washers and nuts) into the assembly (*.a3d). Fastening elements can be added from both sides of the stud or only from one side if the stud has a threaded end. Positioning of the stud is performed in the same way as for elements of other detachable connections—using placement objects.
- The list of placement objects processed by application commands has been expanded. Auxiliary planes, axes, conical and curved surfaces are now available for use with steel. Added the ability to set displacement along three axes and rotation of the detachable connection both through the dialog and using the manipulator.
- Automatic thread selection for the specified hole is added to all connections.
- Dialogs for selecting instances of fastening elements have been refined: the properties of the elements are now presented in a table format, the set of displayed properties and corresponding filters have been expanded.
- The command ***Insert from template connection*** has appeared. The template source can be recently inserted detachable connections in the document (grouped by connection types), or selected detachable connections. Adding to the favorite connections occurs from the recent ones.

2. Permanent connections

- The capability to insert segments of structural elements of welded seams into the Welds Table has been added in the command ***Welds Table***.
- In the commands ***Weld Symbol*** and ***Weld Elements***, a weld type filter has been added, allowing you to filter weld types as others (Et.). This is necessary for standards where the weld type does not correspond to the first letter of the weld symbol (B, A, T, L). For example, weld symbols can be the following: «C», «K» and so on.
- Known bugs have been corrected.

3. eCAD — KOMPAS Converter

- The ability to select three-dimensional component models from Polinom:MDM has been added. In the process of accessing POLINOM:MDM, it is necessary to specify the storage name once, which is then remembered.
- Certain bugs found during the operation have been fixed.
- The Help system has been enhanced.

4. Equipment: Cables and Harnesses

- In the window ***Positional indicators. Contacts and chains*** it is now possible to filter the entered records by values of Name or Symbol of components.
- In this same window, the ability to quickly search in the component list by its alphanumeric code has appeared.
- When importing an interchange XML file from KOMPAS-Electric and CADMAX design systems, the ability to select individual connections from their general list for import appeared. This function allows, for example, to import into the open assembly only those connections that exist in this assembly, and not the full list of connections of the entire electrical project.
- When importing an exchange XML-file from the design systems KOMPAS-Electric and CADMAX, the capability has emerged to interactively create harness sub-assemblies in the main product with the assignment of their Designation and Name. This is possible in cases when the exchange XML file contains information about these harnesses (their

Alphanumeric Codes, Designations, and Names), as well as the distribution of connections by harnesses.

- In the window **The connections table** now provides the possibility to filter by wire/cable names.
- In this window, the option for quick search of connection addresses by components alphanumeric code and by harnesses alphanumeric code/cables has appeared.
- It became possible to model conditional conductors in the absence of wire/cable markings in connections. In some cases, at the stage of creating the electrical schematic, the circuit designer cannot immediately specify the exact types of wires and cables for the connections. Previously, the application did not create a harness/cable model in the absence of assigned conductors, but tracers need such a model, even if it is conditional, to determine the routes for future harnesses. Now the application can create a conditional model, for which you can set conditional wire diameters and their color in the settings. The conditional diameter and color of the wire will be the same for all conductors that do not have a specified real wire. In the future, after specifying the actual conductors, it will be necessary to rerun the command **create a harness/cable model**.
- Certain bugs found during the operation have been fixed.
- The Help system has been refined.

5. Shafts and mechanical transmissions

- It is now possible to build a 3D model of a part through automatic generation based on a 2D model, which is stored in the file of this 3D model. The procedure for constructing a 2D model is the same as when working with the application in graphical documents of the KOMPAS-3D system (drawing or fragment).
- Calculations and plotting of sprockets with a toothed chain for the chain drive have been implemented.
- When designing worm cutters for cylindrical gears and involute splines, based on user requests, the construction of the tooth profile in the axial section with an additional parameter table has been added.
- When creating a place for installing a round slotted nut and a locking multi-tang washer, the option to select a groove type for thread runout according to GOST 10549-80 has been added.
- Construction of the keyway on the external conical step has been refined.
- For cylindrical gears with a clock profile, upon user request, the calculation of the tooth profile of the worm cutter designed for manufacturing gears has been implemented. Additionally, the possibility of constructing an auxiliary view with the tooth profile of this cutter has been added.
- For cylindrical gears with an involute profile, the output of profile coefficients for non-standard values has been added to the parameter table.
- For involute splines, when selecting measuring roller diameter, the ability to add non-standard roller sizes is implemented.
- Modifications have been made regarding the construction and calculation of tolerances for involute splines according to ANSI B92.1-1996.
- Calculation of the bevel gear with circular teeth has been refined. When calculating based on the external circle module, a selection-calculation of the initial data for the given cutter spread is implemented.

6. Equipment: Steel structures

The interface for selecting a material profile from the Materials and Assortments Application has been reworked. All material assortments that can be used to build metal structures are grouped in one window. The selection of material type and size has become more convenient. The list of materials the user works with is configured.

7. Equipment: Cable ducts

It became possible to update settings for already created cable channels. You can update settings for both all cable channels included in the assembly simultaneously, and for selected ones.

8. Equipment: Pipelines

- It became possible to update settings for already created pipelines. You can update settings for all pipelines included in the assembly at once, as well as for selected ones.
- The interface for selecting pipe assortment instances and pipeline elements from the Materials and Assortments application, as well as Standard Products for KOMPAS, has been revamped. Selection of pipe size has become more convenient. Pipe elements types that the user works with are grouped in one window. The list of available elements for application may be configured.

9. Equipment: Ventilation systems

It is now possible to update the settings for already created ventilation systems. You can update the settings of all included ventilation systems at once, or selected ones.

10. Equipment: Unfolds

New elements have been added:

- Convergent reducer, type 6,
- Tee type 1,
- Tee type 2,
- tee — type 3,
- Elbow — type 4.

11. Mechanics. Springs

New hooks for extension springs have been added.