KOMPAS-3D v24

Version information

Differences between version 24 and version 23

General

1. Added library *Graphic_2023.lyt* to the *\Sys* folder, replacing library *Graphic.lyt*. All documents created in KOMPAS-3D v24 use the *Graphic_2023.lyt* library; the *Graphic.lyt* library is not used, but is present in the \Sys folder to support previously created documents.
2. Improved dialog box for setting system and current document parameters (invoked by command ***Settings — Parameters...***):
   - Settings are represented by a general list, scrolling through which it is available to view all the parameters accessible for change. In addition, it is available to select a separate section and view the settings located in it and in its subsections. Previously, to access a specific parameter, it was necessary to select a tab, and then to select a section (and subsection) on it.
   - Added the ability to search by names of sections and controls (options, lists, etc.) to the dialog box.
   - For some parameters, the controls that are used for customization have been changed, for example, the options group has been replaced by a list of options. Some parameters have been moved to other sections to group settings more logically; the set of sections is generally the same as before.
   - General interface settings have also become available in the dialog: selection of colors, language and icon size, keyboard settings, tab positions and remembering the last command. Previously, these parameters could only be changed in the menu of the **Interface settings** button in the KOMPAS-3D window header.
3. The default setting of KOMPAS-3D has been changed; the following parameters have been changed:
   - automatic obtaining of a license for KOMPAS-3D upon loading: was disabled, became enabled;
   - default name of the KOMPAS document file: it was ***Name + Designation***, it became ***Designation+ Name***;
   - default scheme of standard views: there was Z-axonometry, became Y-axonometry.
4. The font of KOMPAS-documents was changed: now the unicode font *GOST type AU* is used in creating new documents. In documents of previous versions non-unicode font *GOST type A* was applied. To ensure specified replacement the new document settings, templates, titles, the original file of special characters *GRAPHIC.SSS* and binary file of special characters *GRAPHIC.BSS*, fonts *GOST type AU* and *GOST type BU* were adjusted.
5. New features when creating and editing links in the ***Link*** dialog box:
   - rounding to the specified sign is now available for link to a property — provided that the property value is a number (previously rounding was only available for variable links);
   - if a broken link is edited, the dialog displays the text it last contained and a warning that the source of the link was not found.
6. Added the ability to include a scroll bar in the additional Design tree. The corresponding option can be found in the ***System — Screen — Background color*** section of the setting dialog box.
7. Added the possibility to delete rows in lists and tables in the Parameters panel and in the Product BOM panel with the key <*Delete*>.

8. Files *Graphic.kds* and *Graphic.kdsp*, defining the set of available document codes and the order of sorting documents by code, are corrected according to GOST R 2.102-2023. The files are located in the *\Sys* folder.
9. The size and location of the KOMPAS-3D window is now saved between working sessions. Previously, in a new session, the KOMPAS-3D window opened in full screen.

3D modeling

1. When working with an assembly, it is now possible to change the shape of components by location, i.e. without modifying source-files. Changing is performed using operation ***Deformation***, during which it is available to edit the operations performed in the component (Extrusion element, Revolution Element, Lofted element, Element by path, Pipe element, as well as operations for building curves of some types), replacing their source objects with new ones — built in the assembly or in other components. For splines and polylines, it is available to edit the coordinates of the vertices; the numerical parameters of other operations cannot be changed.
   The parameters and the result of deforming a component are stored in the assembly where the deformation was performed, i.e. the component file and its inserts in other assemblies are not changed.
2. Added commands for building a geometric body in the form of a straight circular cone:
   - ***Cone by base center, diameters and heights***,
   - ***Cone by base center, diameters and angle***,
   - ***Cone by two arcs***.
3. Added commands ***Pipe element*** and ***Cut with pipe element***. They perform operation along a trajectory with an automatically formed section in the form of a circle of a given diameter. The trajectory is specified by the user. The pipe may be solid or thin-walled. In the latter case, either the wall thickness or the second diameter value must be entered.
4. Added the ***Mid-surface*** command, which is used to build a surface that is the geometric location of points equidistant from two specified surfaces.
5. Added a new direct modelling command — ***Change face position***. It allows to move the selected faces in a given direction by a certain distance or rotate them around a given axis by a certain angle.
6. Added the ***Alignment*** command to match a polygonal object with an existing body, surface or other polygonal object in the same model. The result of the command is to change the position of the polygonal object (or one of the polygonal objects) so that the deviation between it and the second object is minimized.
7. Added the ***Geometry check*** command for diagnosing bodies, surfaces and faces for correctness of mathematical description. Executed for individual selected objects or for all objects included in an assembly or in a specified component.
   The following checks are performed:
   - ***Non-parallelism*** — detection of edges located relative to the axes of the specified coordinate system at an angle less than the specified one,
   - ***Small size*** — detection of objects with size smaller than the specified size,
   - ***Open edges*** — identification of open edges, as well as overlapping open edges of the same body/surface/face,
   - ***Juts/cuts*** — identification of faces with angles less than the specified one.
   - ***Intersections of faces*** — identification of faces of a body (or surface) that intersect other faces without forming a common edge,
   - ***Self-intersection*** — identifying self-intersecting faces and edge boundaries,
   - ***Geometric precision*** — identification of geometrically inaccurate vertices and edges (the available deviation of the position of objects from the theoretically precise one is assigned by the user),

- ***Size check*** — detection of objects that are outside the cube of specified dimensions with the center at the beginning of the original coordinate system.
  It is possible to exclude layout geometry, standard parts, and hidden objects from the check.

8. Added the ***Value check*** command, which allows to assign the range of acceptable (or unacceptable) values for selected variables, as well as for the model mass. If the value goes outside the range (or enters these limits), an error message appears.
9. Added command ***Copy management***. It allows to find copy — operations in the model itself and its components up to a specified level or only in selected components. For each operation, its current status is displayed and it is possible to control the parameters. For example, the user can enable source change tracking for copy operations that have it disabled, or update all or selected copy operations. In addition, it is possible to proceed to editing any copy operation.
10. The ***Surface extension*** command has been improved:
    - For an extension by the same surface, edge simplification — this can be included to increase the length of the extension in the case where it is limited by the degeneracy of the edge equidistant of the original surface or by the fact that the equidistant has ceased to intersect with the extensions of the side edges. A simplified edge is not an precise equidistant of the source surface edge (i.e., it does not suppose a user-specified value of extension at each of its points).
    - It is now possible to select the result of the extension, i.e. to specify whether the added surface area should be a new surface or a part of the source surface. In case of bulding an extension as a new surface, it is possible to specify a non-open edge as an edge and select which of its adjacent faces should be extended.
    - It became available to extend several different surfaces in a single command call.
11. Developed command ***Rounding surface***:
    - added a building method that does not require specifying an orienting line, but only specifying the rounded surfaces (the rounding surface in this case is the result of «rolling a ball» of constant radius between the rounded surfaces; the surface boundaries are formed by the points where «ball» touches the source surfaces),
    - added the possibility of selecting the result of the operation:
      - the rounding surface and the source faces are unchanged,
      - the rounding surface and trimming of the original faces along its boundaries,
      - the trimming of the source faces and knitting them with the rounding surface into a single surface.
12. Improved ***Replace faces*** command: added possibility to replace one group of faces with another group of faces. In this case, the replacement group must necessarily be a linked set of faces, and the replaced group must not.
13. Developed commands ***Collapsed curve*** and ***Unfolded curve***:
    - both commands now work with extrusion surfaces and unfolded ruled surface: it is available to collapse curves and points onto these surfaces and unfold from them,
    - In the ***Collapsed curve*** command, it is now possible to trim the collapsed curve with edges that limit the surface, i.e. lateral edges and cut edges, if any,
    - in the ***Unfolded curve*** command it is now possible to control the position of the surface cut line for cases when it is closed (cylinder, cone, etc.) in intersection points the curve will be divided before unfolding.
14. Improved command ***Model family***:
    - The possibility of creation Family table preparations has been implemented: the user selects properties and variables to be included in the table, establishes the parameters of the table file, and then, having opened this file in the table editor, fills it in. The result is a table which column headings match the names of properties and variables, which is a

prerequisite for a workable family table. Previously, only manual creation of the Family table was available, i.e. errors in column headers were possible.

- Added the following ways to assign the properties of family model-instances:
  - Determining of the density of a material by its name (available provided that the method of calculating the CMP of the Family template — ***By density***): if the Family table contains a material which name is available in the reference density file (*Graphic.dns*), then the instance is assigned a density corresponding to this material.
  - Assigning an arbitrary mass value from the Family table (available if the method of CMP calculation of the Family template — ***By mass***).

  Note: Similar possibilities for assignment of properties appeared when creating in the model of versions by reading data from the table, their availability depends on the method of calculating CMP of the main version.

15. The ***Change position*** command has been improved:

- edges, faces and vertices retain their system names after the operation, so that objects based on them (e.g. dimensions or matings) are correctly rebuilt and the sheet body does not lose its properties as a result of moving,
- added the possibility to apply the command to curves and points, not only to bodies and surfaces; moreover, it is now possible to select several objects at the same time, including different types, to change the position,
- the option ***Delete source object*** has been eliminated; the options ***Hide source curves*** and ***Save copies of bodies and surfaces*** have been added.
- using the method of moving relative to the coordinate system, the dimensions of the operation — offset distances and rotation angles — have become available; thus, these values can now be edited in the graphic area: when performing/editing the operation, or in the mode of displaying the dimensions of the specified element, or as derivative dimensions of the operation.

16. The command ***Deviation analysis*** has been improved:

- it became available to measure deviation not only between a body/surface and a polygonal object, but also between two bodies/surfaces/polygonal objects in different combinations,
- there is now a measurement precision setting that controls the number of points at which deviations between objects are measured,
- added possibility to assign an acceptable range of deviations and to highlight on the color map those areas of the analyzed objects where deviations do not exceed this range; if the range is set, the number of points falling within it is displayed in the table of results,
- the analysis process itself is started by the user after the configuration is completed and is displayed as an indicator in the lower left corner of the screen; if necessary, the process can be interrupted (previously it was started automatically after selecting objects, and interruption was impossible),
- when moving the mouse cursor over a color map, the deviation value in the point under the cursor is displayed near it,
- the setting of length units applies not only to the measurement result, but also to the parameters expressed in these units (maximum distance and range limits),
- the names of the points where the maximum deviation values are displayed have been changed: they were «L_min» and «L_max», and have become, respectively, «EI» and «ES».

17. The Standard part inserted in the part can now be converted into a blank. The blank created in this way is taken into account in the drawing of the part and in the BOM assembly containing the part with the blank. The ability to edit a standard blank part using the Standard parts reference is retained.

18. The ability to copy components via the clipboard and paste them into the same or another model has been implemented, including during context editing. Any component at any nesting level can be selected for copying; pasting is done at the first level. If necessary, during copying it is available to enable saving the matings between the copied components and between them and the coordinate system of the model containing them.
19. Improvements and new features for working with layers:
    - If a subassembly is placed in an assembly using the ***On source layers*** method, any component of that subassembly can be moved to an arbitrarily selected layer of the assembly. In this way it is possible, for example, to arrange the fasteners from all subassemblies on one assembly layer — this allows all fasteners to be quickly hidden if necessary.
    - When customizing object properties, it is now possible to select the layer on which it should be located: current layer or a layer with assigned number (if there is no layer with this number, it is automatically created when the object is created). Layer selection is available for objects that can be placed on layers: auxiliary planes and axes, points, curves, etc. The settings made for the properties of objects can be saved to a file and read from the file, for example, at another workplace.
    - If the model has more than one layer, then:
        - objects which visibility is defined ***By layer*** have a special visibility icon in the Design tree (previously they had the icon ***Visible***),
        - it is available to enable visibility determination for an object using the ***By Layer*** method in the Design tree (previously, only in the Parameters panel).
    - The Layer tree is now located on a separate panel, the display of which is enabled by the command ***Settings — Panels — Layers*** or the key combination <*Ctrl*>+<*L*>. Previously, it was necessary had to switch the Design tree to the appropriate mode to work with layers.
    - Selecting objects by specifying the layer containing them is done in the Parameters panel, and not in the dialog as before. Due to this, selecting objects in this way can be run as a subprocess of another process, such as creating an assembly report.
    - Added layer creation command that allows to create up to 999 new layers in the model at once.
20. New possibilities for selecting objects:
    - Added command ***Select by source***. It displays a list of all components of the current model that are inserts of external files, the names of these files and their paths. A filter by component type is available, for example, it is available to leave only assemblies or only standard parts in the list. Identical inserts, including those at different levels, are displayed in the list as a single line. Selecting a row selects all inserts of the corresponding component and vice versa, selecting a component selects the row and, therefore, all other insertions of the same component, if any. Selecting a component by source can be useful if it is available to replace all inserts of a particular model with another model.
    - If, when performing operation, an object that is unsuitable for the active registrar is specified, then — provided that at least one filter is enabled — a search is performed for objects that satisfy the filter within the specified one. If these objects are suitable for the registrar, then they are entered into it. For example, to select all curves in the model in the ***Continuity check*** command, it is available to enable the ***Curves*** filter and specify the root element in the Tree.
    - On the context panel that appears when selecting a primitive (vertex, edge or face), a hierarchical list of objects that include this primitive has been added: from operation to the root element of the Tree. By selecting the required line in this list, it is available to select one or another of the objects containing the selected.
    - In the additional model window, it is available to quickly specify bodies that are visible or invisible in projections onto the main plane s . This simplifies, for example, the creation

of a model-mockup: to obtain it, it is necessary to copy only the visible bodies of the source model to a new model.

21. The Quick access panel has been updated with elements that control the process of specifying curves:

    − The ***Select parts of curves*** button is used to specify the method for selecting curves: in whole or in parts that are formed when a curve intersects with other curves. The button is present on the Quick access toolbar when working with the following objects:

        − contour and equidistant;
        − pipe element, element by path and the surface by path;
        − conic section surface and ruled surfaces, except for the ruled surface by two surfaces;
        − point on curve;
        − sketch — in the process of its placement on the trajectory.

    − List of options for selecting the curves that make up a non-branching chain: ***Single*** — selection of each curve individually and ***Connected*** — selection of the whole chain. The list is present on the Quick access toolbar when working with the ***Contour*** command and replaces the ***Non-branching node pass method*** switch that was there.

22. New features for creating copies of geometric objects and working with them:

    − if an operation is selected for copying, the list of copied operations includes not the faces of this operation, but the object which it belongs to (body or surface);

    − copying result objects have the same properties and parameters as the source objects: copies of surfaces — optical properties, copies of bodies — optical properties and CMP parameters, copies of points and curves — colors and styles;

    − phantom of copied objects is displayed as semi-transparent (previously it was wireframe);

    − if the source of the copy is an external model, it can be quickly opened — using the context menu command of the copy object in the Design tree.

23. The ***Surface adjustment*** command now automatically determines the type of surface that approximates the specified section of the polygonal object. If necessary, the result of the autodetection can be changed manually by selecting the required surface type from the list.

24. Improved the process of reading versions recorded in the **.ods* or **.xls* file: the main version from the file no longer becomes one of the versions of the current model, as before. Instead, the values of the properties of the main version from the file are sent to the properties of the main version of the current model, and all other versions from the file are placed in the Tree while maintaining the hierarchy relative to the main version. Thus, by reading versions from a file, it is available to completely copy the structure of the Version tree of another model.

25. In the array building commands, selection of faces and, accordingly, building of their copies became available. It is available to specify either one or several adjacent faces of the same body/surface.

26. In the commands ***Revolution element***, ***Revolution surface***, and ***Extrusion surface***, it is now possible to specify more than one object to use as a section.

27. In the case where a surface and an edge of another surface are specified for measuring mutual deflection, in addition to the distances between the points of the edge and the surface,also the angles between the tangent planes to the surfaces at th these points are calculated.

28. In the ***Section*** command, it is available now to use a body, a surface, a linked set of body/surface faces, and a set of unlinked but adjacent faces as a section object.

29. The ***Mirror array*** command now has the ability to build a plane of symmetry without exiting the process.

30. When creating/editing a surface finish designation, it is now possible to select multiple objects (previously, only one could be selected). If a surface finish designation is placed on a leader-landing, it is possible to create leader-landing branches pointing to these objects — if they intersect with the designation plane.
31. If copy saving is enabled when scaling a body or surface, the scaling operation has two subordinate objects in the Tree: a copy of the source object and the scaled body or surface (previously — only a copy of the source object).
32. Added the ability to display the ordinal numbers of operations in the Design tree. Operation numbers are also shown when viewing object relations, are taken into account when searching in the Design tree, and are included in the information generated by the ***Object info*** command.

    Operations are numbered in the order of their sequence in the Tree, which has a design history view, so after adding/deleting/moving operations, the underlying operations are renumbered.
33. New features for reading and writing data files (used when working with versions, families, curves, points, surfaces, arrays, etc).:

    − reading of Excel 2007-365 format (**.xlsx*) and writing to this format has become available,

    − to read and write to the Calc format (**.ods*) it is no longer necessary to have a tool for working with files of this format installed on your workplace (for example, OpenOffice or LibreOffice).
34. The ***Curvature graph*** command now has the ability to calculate the stress of a curve. The calculated value is displayed next to the curve and is changed when the curve is edited.
35. Added a new method for calculating CMP of an assembly component: ***By source mass***. In this method, the mass of the component is always taken from its source file, and the center of mass and moments of inertia are calculated from the actual geometry (or can be set manually). This method is automatically applied to a component modified using the ***Deformation*** operation.
36. The ***Object selection*** command has been improved: now it is available to select entire compound objects, such as sketches, multi-segment curves, surfaces, bodies, components. In addition, object enumeration is now available in an additional model window.
37. New features when working with load types:

    – Selecting load type before insertion into assembly. It allows to speed the process of adding «heavy» subassemblies: when selecting inserting model it is possible to set load type for it, for example, ***Empty***, and then place, assigning coordinates.

    – View the composition, i.e. the list of incoming components, for subassemblies with load types ***Empty*** and ***Size*** at all levels of nesting. Information about the composition is loaded at the moment when the branch of the subassembly is opened in the Tree when the mouse clicks on the «triangle» icon.
38. Editing in the window has become available for the component, which is a local part. This expands the possibilities of configuring the component's display: this editing method allows you to control, for example, the optical properties of individual objects.
39. It is now possible to insert a component into an assembly with the **Component parts** option enabled. Previously, its inclusion was only available after insertion.
40. When editing the contour, matching creation became available. That is, by selecting new source curves for the contour, the user can establish a correspondence between the original and new contour segments. Due to this, the objects based on the replaced contour segments will not lose them, but will use the new ones. For example, if some contour segments were projected into the sketch, then editing the contour with matching will allow you to get projections of new segments in the sketch. Editing without matching will result in errors in the sketch.

41. Commands have been added to improve navigation through the model:
— **Collapse** — in the context menu of the root element of the Model tree, as well as the branches of the Tree. The result of the team's work:
— for the root element, collapse all expanded branches at all levels, while the root element itself remains expanded.;
— for a branch, the collapse of all its subordinate branches and the branch itself.
— **Show in the tree** — in the context menu of the component selected in the graphic area. The command expands the Model tree and highlights the selected component in it. Previously, the command was only present in the context menu of the selected primitive or geometric object.
42. Added the ability to display the composition of a standard product in the Model tree.
43. An information variable of the trajectory length has been added to the set of element and surface variables along the trajectory.
44. The following sketch objects have become visible and available for use outside of sketch editing mode:
— curves with the *Normal* and *Centerline* styles contained in the inserts of macronutrients,
— texts contained in the inserts of fragments.
45. When measuring the area, it is available to specify bodies and surfaces in addition to faces — the areas of all their faces will be calculated.
46. Versions and variants of a model that depend on the same version and are located on the same level can be replaced with each other in the Tree by dragging them with the mouse.
47. A majority surface finish designation is now displayed as an object in the Design tree. This makes it easier to manage the visibility of the designation and to switch to editing it.
48. Icons for which display in the Tree is enabled (visibility, load type, projection, ordinal numbers, etc.)are also shown in the Relationship tree at the bottom of the Design tree.
49. Performance of the KOMPAS-3D system is significantly increased when displaying phantoms during work with large assemblies.
50. Improved performance of KOMPAS-3D when using load types: reduced time, as well as RAM consumption when opening an assembly with a custom load type, in which components of the second and subsequent nesting levels have the ***Empty*** load type.
51. Optimized data storage in models that contain operations based on the same object, such as many points on the same surface. Due to optimization, the speed of rebuilding has been increased and the file size of such models has been reduced.
52. Experimental functionality «Metaspline» is excluded from KOMPAS-3D. Metasplines existing in models that were created in previous versions of KOMPAS-3D will be correctly processed, but it is no longer possible to create new metasplines.

Graphic documents

1. The ability to create arrays of graphic objects has been implemented. An array is a group of ordered objects — array instances. When creating/editing an array, it is available to control the number and position of instances, as well as exclude specified instances from the array. In addition, it is available to enable the creation of control dimensions in the array - using them it is possible to modify an already created array. If necessary, the array can be exploded, i.e. the links of the instances to each other can be removed, turning them into independent objects.
Arrays of the following types are available:
   - Array by grid,
   - Array by concentric grid.

   Arrays are displayed in the Tree of the graphic document, forming a separate group ***Arrays***.
   Arrays of graphic objects can also be created in sketches of 3D models.

2. The designation of the form and shape tolerance can now include additional signs to indicate the intersecting and orienting planes.
   The option is also available when you set the shape tolerance in the model.
3. Adding branches to designations that include a leader-line has been sped up: the process of adding a branch is started, as before, by clicking on the landing start point, but now it is looped, i.e., to create the next branch, it is not necessary to restart the process
   it is enough to specify the point to which this branch points. To exit the process, press the key <*Esc*>.
4. The process of selecting curves for measurement using the ***Curve length*** command has been improved:
   - added possibility to specify curves before invoking the command,
   - after invoking the command, curves can be selected with an encompassing or section marquee by holding down <*Ctrl*> to invert the selection or <*Shift*> to add to the already specified ones; clicking on a free graphic area deselects all curves.
5. It is now possible to link cells of the drawing title block with arbitrarily selectable properties from property libraries. This setting is made when editing the title block included in the drawing layout and makes it possible to display the values of custom properties in the drawing, including reflecting in the associative drawing the values of custom properties defined in the model that is projected into this drawing.
6. It is now possible to add a sheet to a drawing based on a sample of an existing sheet. In this case, the new sheet is placed after the sample sheet and has the same format, layout, multiplicity and orientation.

Working with properties and reports

In the Product BOM panel, it is now possible to select several arbitrarily selected parts and change their properties in a uniform way at the same time.

Working with BOM

1. Implemented automatic creation and placement in the *Documents* section of objects corresponding to documents that are associated with the document connected to BOM. This speeds up filling out the *Documents* section of the BOM. For example, as a result of connecting an assembly to BOM that has assembly and dimension drawings associated with it, the BOM will also create a *Documents* section that includes the assembly and drawings with overall dimension, in addition to the sections displaying the content of that assembly.
2. It is now possible to automatically move text in a cell from row to row. Autotransfer can be enabled/disabled for any column when customizing BOM style or when editing the BOM, while editing the BOM it is also available to also enable/disable autotransfer in an arbitrarily selected cell.
3. It is now possible to select BOM column by clicking on the heading of this column. For a highlighted column, it is available to view the name that is given to it in the style, as well as enable or disable row auto-transfer.
4. Added the ability to save BOMs in Excel 2007-365(**.xlsx*) and Calc(**.ods*) formats. Saving to the xlsx format does not require (and when saving to the xls -stopped requiring) the presence of an installed Microsoft Excel program on the workplace.
5. It is now possible to manually edit headers for additional and nested section blocks. The specified text is substituted in the header in place of a construct of the form #XXX# entered when setting the BOM style. Previously, there was only one way to edit the title - connecting a document which designation replaced the specified construction.
6. Styles of simple and group BOMs:
   - added section *Software products and databases*;
   - in the styles of group BOMs, in the section *Kits*, blocks of nested sections have been added;

- a block of nested sections *Packaging* has been added to the *Kits* section;
- the content of the block of nested sections *Tool and accessories kit* has been changed: now it contains only the sections *Tool*, *Accessories*, *Devices*, *Materials*.

7. The ***Edit BOM object*** command in the context menu of the specified graphic object included in several BOM objects now has a submenu for selecting the required BOM object (previously the command called a dialog with a list of objects).
8. The command ***Sum of values in columns...*** has been renamed to ***Sum values...*** The result of its work is displayed in the information window (previously — in the dialog box).
9. The possibility of creating an external BOM object in documents has been eliminated.

Import and export

1. Added command ***Import to current model***. It is possible to import three-dimensional objects from interchange file formats (STEP, JT, IGES, ACIS, etc.) into the current model.and proprietary (UGS/NX, ProE/Creo, SolidWorks, etc.)formats. Import from formats that provide customization is done with the current setting; it is available to change it if necessary.
2. When writing models to and reading from JT format, it is now possible to pass the *BOM section* property. The transfer is performed correctly for objects belonging to standard BOM sections (*Documents, Complexes, Assembly units, Parts*, etc.)..
3. When importing an assembly from STEP and JT formats, the component file names are formed according to the default file names setting in KOMPAS-3D.
4. When importing a STEP format file, coordinate systems as well as object names are sent.
5. It is now possible to read models from Creo and Inventor CAD systems without installing the ***Extended reading tools for proprietary format*** component.

Add-ons

1. ***ECAD - KOMPAS converter***
   - Added the ability to select a local or remote POLYNOM server.
   - In a number of cases, when identical components with different Reference numbers had different three-dimensional models depending on the variant of the footprint, the Converter remembered the first established correspondence between the model and the component name, as a result of which it added the same model to the assembly regardless of its Reference number. Now a check for the uniqueness of the insert has been added, and the user can independently specify in the Converter the name of the field in the BOM file that is responsible for the Positional number.
   - If the user chose to build a realistic model from a BOM-file containing component IDs in POLYNOM:MDM, but those components were not present in the POLYNOM database, the system would hang. Now in this situation the Converter moves on to building a layout model.
   - Fixed some bugs detected during operation.
   - The Help system has been improved.

2. ***KompasFlow, fluid dynamics for Kompas-3D***
   - The view of the project tree and properties window has been changed.
   - Added a new type of layer display — Fill. Now the display of this layer can be set not only as gradients with smooth color transitions, but also as a fill with gradation of each color with clear contours of borders, and the colors of the ranges will correspond exactly to the colors set in their palette.
   - Added automatic check for self-intersections of the model mesh geometry when creating a new KompasFlow project on it. If self-intersection areas are detected as a

result of the check, the surface to which such areas belong will be marked as erroneous in the Project tree.

- The possibility of creating projects on geometric models containing the same body names has been implemented.
- The ability to modify existing and create new layers and objects between solver calculation runs has been implemented. Now when you press the ***Open solution*** screen button, the project settings and the data calculated on the solver are synchronized, which make it available to continue the calculation with changes in the layers and objects of the project.
- Added a relativity parameter in the layer properties window, which can be used to set one of three options for expressing the variables *Temperature* and *Pressure*: absolute value, relative value, and over value. The specified variant of the variable expression is also displayed in the layer legend.

3. ***Service tools***

Export of BOM is excluded from the set of BOM operations.

4. ***Equipment: Flat patterns***
   - New elements have been added:
     - Tee type 4,
     - Tee type 5.
   - Added new type of Pipe transitional from circle to rectangle — type 2. The former Pipe transitional type 2 has been renamed "Pipe transitional type 3".
5. ***Mechanics: Springs***

The building of the tension spring types of hooks views has been finalized.

6. **Equipment: Pipelines**
   - It is now possible to build a point-to-point pipeline.
   - In the interface of material/element selection from the reference, it is now possible to work with favorite objects.
7. **Equipment: Steel structures**

In the interface of material selection from the reference it is now possible to work with favorite objects.