universal
mechanism

User`s manual

Data Import from CAD Programs and Formats

2020

Contents

9. Data import from CAD programs

9.1. General information

Universal Mechanism Software (UM) allows data importing from CAD programs in the following ways,

1. Data import from **STEP** and **IGES** file formats using free of charge CADToUMConverter.exe, see Sect. 9.2, page 9-6 for more details. The converter uses libraries from Open CASCADE Technology package, that is distributed under GNU Lesser General Public License (LGPL) version 2.1.

2. Conversion of data from **STEP, IGES, Parasolid, SAT** format files is implemented with the help of an intermediate external CADlook (www.cadlook.com) program that is used as the intermediate converter, see Sect. 9.3, page 9-8.

3. Direct import from some CAD programs. At this time the interface is developed for the following CAD software:

- **SolidWorks** 2012 and later, 32- and 64-bit versions are supported, see Sect. 9.5, page 9-13;

- **Autodesk Inventor** 2013 and later, 32- and 64-bit versions are supported, see Sect. 9.4, page 9-12; converter for **Autodesk Inventor** is available via the following link: universalmechanism.com/download/inventoruoumaddin.exe

- **Unigraphics NX** 9.0, 32- and 64-bit versions are supported; it is available via the following link: universalmechanism.com/download/ugtoumaddin.exe. See Sect. 9.7, page 9-20 for more details.

- **PTC Creo 1.0** and later, 32- and 64-bit versions are supported; it is available via the following link: http://www.universalmechanism.com/download/creotoumaddin.exe. See Sect. 9.8, page 9-21for more details.

- **Pro/Engineer Widefire** 4 and 5; it is available via the following link: http://www.universalmechanism.com/download/proetoumaddin.exe, see Sect. 9.9, page 9-23.

- **KOMPAS-3D** 7+ and later, see Sect. 9.6, page 9-16.

  All the converters mentioned above are free besides CADlook program.

  The following data is imported from the CAD programs listed above:

- graphical images,

- inertia parameters (is not provided by the CADToUMConverter.exe),

- mates (for SolidWorks and KOMPAS-3D 8+ and later assemblies only); there are restrictions in importing mates.

4. Reading STL and 3DS files, which allow import data, e.g., from AUTOCAD, Catia v5 and so on.

In **UM Input** program, use the **Tools | Import from CAD** menu command, see Figure 9.1. The command for direct import from CAD program (**SolidWorks**, **Autodesk Inventor**, **Unigraphics**, **KOMPAS-3D**) is available if the corresponding software is installed.

Plug-ins for data export from CAD programs in the intermediate **UM CAD** format (plug-ins for **Autodesk Inventor, SolidWorks, Creo, Pro/ENGINEER** и **Unigraphics**) are free. These

plug-ins do not require any license or **Universal mechanism** to be installed on a computer. You can install them on any computer with **Autodesk Inventor** and **Unigraphics NX**, export any assembly or part to **UM CAD** format, and then copy the resulting files to the computer with installed **Universal mechanism** and then open them in **UM Input** program via the menu **Tools | Import from CAD | UM CAD file**.

    To install plug-in administrator`s rights are required. **Run** the installer **as administrator**. Before installing export plug-ins close **UM Input** and CAD program, for which plug-in is installed.

    To import data from **3DS** and **ASC** (Autodesk ASCII Export File) files (**Import from CAD | Files *.3DS and *.ASC** menu item) a license to **UM CAD Interfaces** module is not required. Data import from **3DS** and **ASC** is included into any UM configuration and is supported by the base module **UM Base**. For everything else, namely to import data "at once" from SolidWorks, Autodesk Inventor and KOMPAS-3D, as well as loading data from files UM CAD (*.ucf format), **STEP** and **IGES** files, as well as STL-files, you will need a license for module **UM CAD Interfaces**.
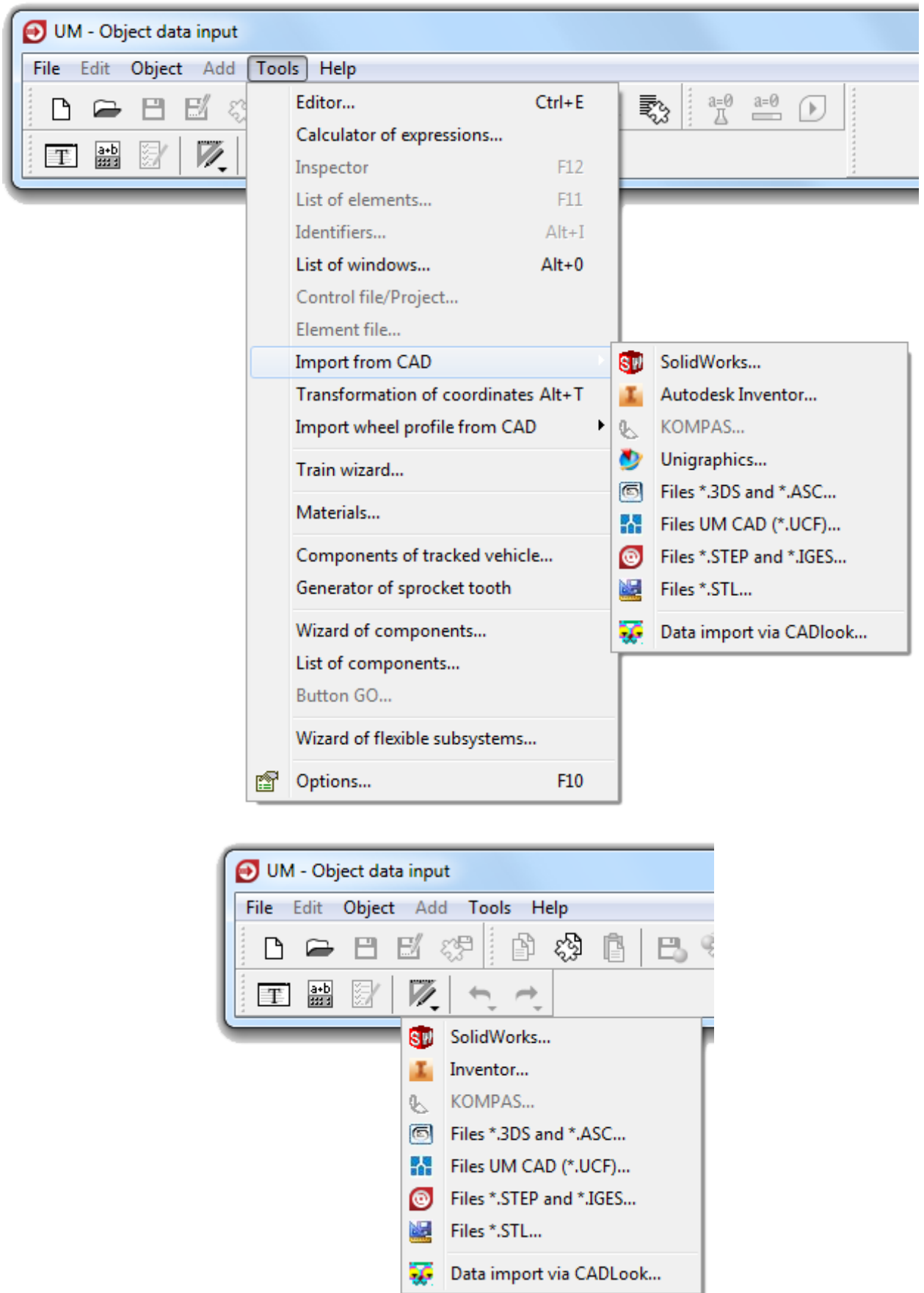
Figure 9.1. Menu commands for importing data from CAD

9.2. Conversion of STEP and IGES files using free converter

To import data from STEP (both AP203 and AP214) and IGES files one can use free converter that is distributed along with Universal Mechanism software. This converter is distributed as a stand along program and may be installed on any computer. It does not require any UM licenses including license for **UM CAD Interfaces**.

Note	A license for **UM CAD Interfaces** is required to load **UM CAD** files (*.UCF) in **UM Input** program.

The converter is included into UM starting with v. 8.4, and as a stand-alone program is available via the following link: http://universalmechanism.com/download/cadtoumconverter.exe.

License	The converter uses libraries from Open CASCADE Technology package distributed under GNU Lesser General Public License (LGPL) version 2.1.

Importing data supposes the following steps.

1. Specify the STEP or IGES file clicking the **Select File** button. Then click the **Load File** button to start file loading process. It might take a few minutes, see Figure 9.2.
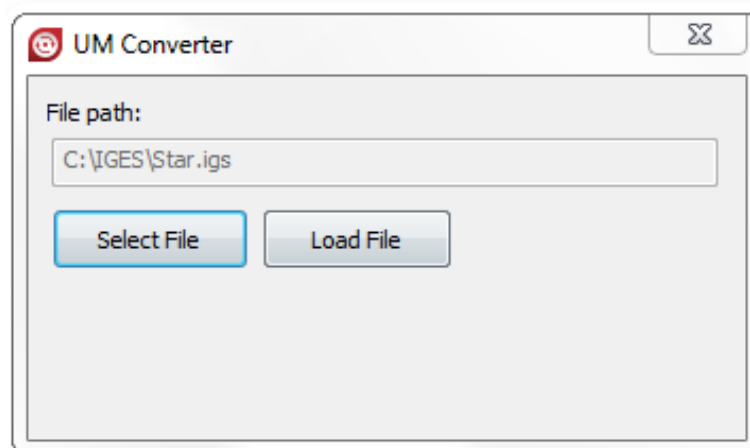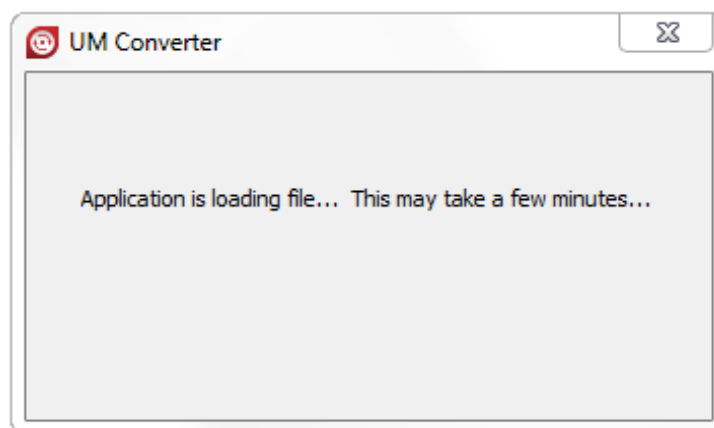

Figure 9.2. Select file to import

Figure 9.3. Loading the file may take a few minutes

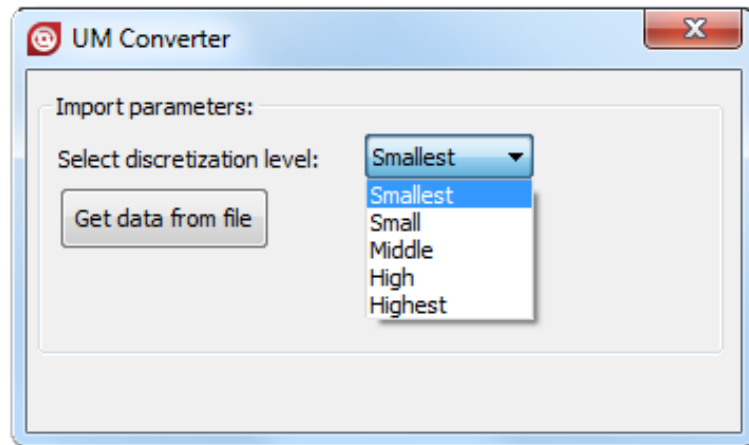2. After loading he file select the discretization level, see Figure 9.4.

Figure 9.4. Import parameters

3. Click **Get data from file** button (see Figure 9.4) to start importing geometrical data.
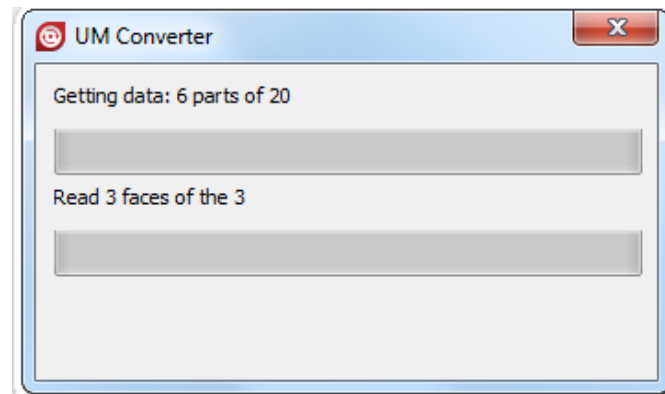
Figure 9.5. Importing progress

4. Check the gabarits of the loading model given in meters. If necessary change the **Dimensions** or directly set **Factor**, see Figure 9.6. By default all parts and components from the file will be merged within one rigid body. If your file includes several rigid bodies use **Separate object onto parts** check box, see Figure 9.6.

5. Finally click **Export to UCF file** to generate he correspondent UCF file.

When the UCF file is successfully created one can load it in **UM Input** program via **Tools | Import from CAD | Files UM CAD (*.UCF)** menu command or simply drag&drop the UCF file into the **UM Input** main window.
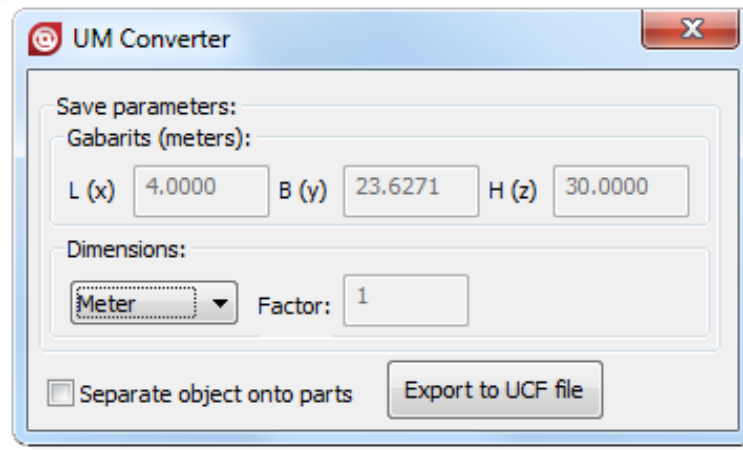
Figure 9.6. Export data to UCF file

9.3. Conversion of STEP, IGES, X_T, SAT formats via CAD-look software

An intermediate external converter is used for reading 3D CAD models in STEP (both AP203 and AP214), IGES, X_T (Parasolid), SAT formats. This program is commercial software CADlook (www.cadlook.com). The user purchasing UM with the ability data conversion from the formats listed above, must purchase CADlook software additionally. On request CADlook license can be delivered along with UM (according to the current CADlook price list) or purchased by user separately (directly or via third-party software distributors). In the case of delivering CADlook along with UM means that the price of the CADlook license must be added to the price of **UM CAD Interfaces** module.

9.3.1. 15-day CADlook trial

Figure 9.7. Steps for getting 15-day CADlook trial

The user working with UM trial might get an access to the 15-day CADlook trial. The following steps are necessary for this purpose on site www.cadlook.com, Figure 9.7.

1. Click the *Download* link (Figure 9.7-1).
2. Download the archive with the program (Figure 9.7-2).
3. Get 15-day license key by the e-mail (use the link on Figure 9.7-3 and follow the instructions).
4. Install the program following the developer instructions (Figure 9.7-4).
5. Check if the program works: read any file in step or iges formats.

9.3.2. Process of file conversion

The following steps are required for development of UM object from CAD files.

1. In **UM Input** program, use the **Tools | Import from CAD | Files *.STEP and *.IGES** menu command.
2. If UM finds automatically the program CADlook   in the computer registry, the process starts according to the steps below. The automatic search of CADlook may fail in the following cases:

- the CADlook software is not installed on your computer; install it according to Sect. 9.3.1. *"15-day CADlook trial"*, p. 9-9;
- the program is installed, but the user have no rights for reading the necessary parts of the computer registry;
- the program is installed, but the current version differs from 11.0.

In the last two cases, the direct path to the CADLook.exe should be set by the user:
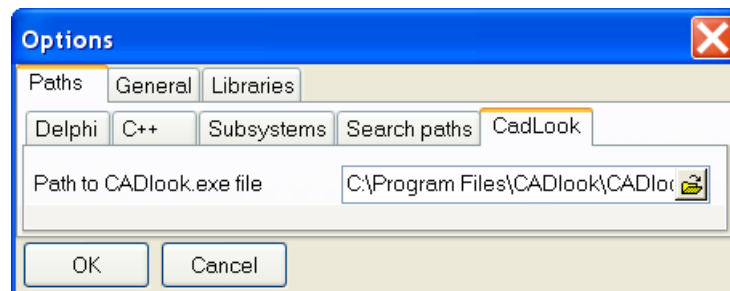
Figure 9.8. Direct setting of path to CADlook.exe

- o in UMInput, open the UM option window by the **Tools | Options** menu command;
- o open the **Paths | CADlook** tab;
- o click the ⬚ button for looking for the CADlook.exe in the install directory.

3. If UM have found the path to CADlook_v11.exe program, the user selects the file for conversion with the standard open file window.

a   b
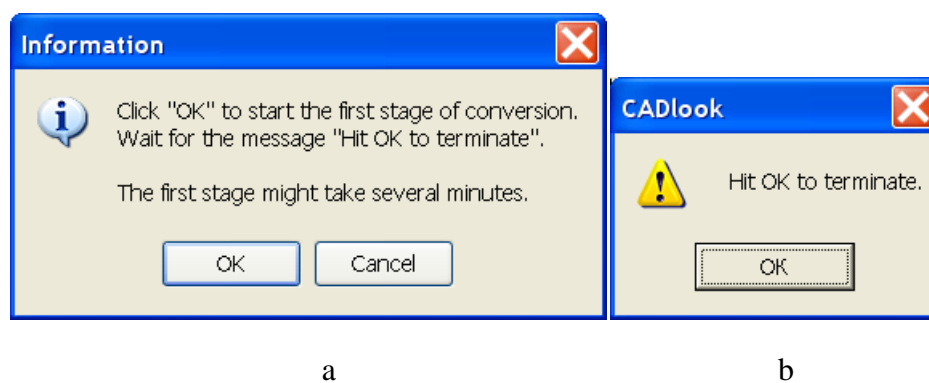
Figure 9.9. Start (a) and finish (b) of the first conversion step

4. The first step of the conversion process starts after the message in Figure 9.9a, and ends by the CADlook message about finishing its work, Figure 9.9b. At this step CADlook reads CAD file and writes a temporary file in STL or VRML formats.

Remark.　　At the first step, UM waits for finishing the CADlook.exe. Up to now, tests show highest stability of CADlook functioning. Nevertheless, if this program "hangs" the user should terminate UM by the Windows Task Manager.
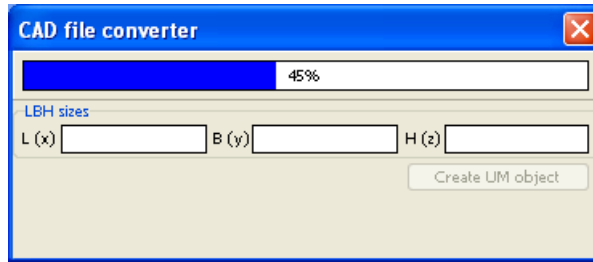
a
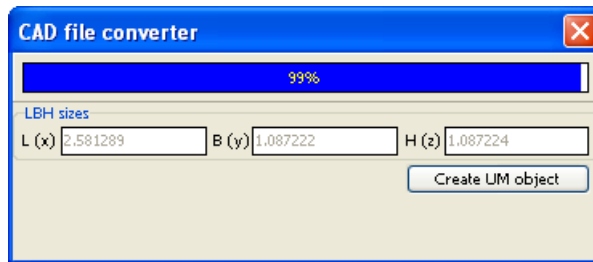
b

Figure 9.10. Running (a) and finishing (b) of the second conversion step

5. After successive finish of the first step a window for translation of data in UM format appears, Figure 9.10.
6. The final step is the creation of UM model and its visualization starting by the **Create UM object** button.

9.4. Data import from Autodesk Inventor

Data export is supported for 32- and 64-bit versions of **Inventor**, starting from version **Inventor 2013**. A free plug-in for data export from **Inventor** in the intermediate **UM CAD file (*.ucf)** is available via the following link:

http://www.universalmechanism.com/download/inventortoumaddin.exe.

For correct work of the data export from the **Inventor** environment, an installed environment **.NET Framework 4.5.1** or later is required. You can download it via the following link:

http://www.universalmechanism.com/download/fwork451.exe.

To delete a plug-in, close the **Inventor** and run the uninstaller as administrator. The uninstaller **UninstallAddIn.exe** is in the folder **C:\Program Files\Common Files\ InventorToUMAddIn**.

9.4.1. Data export from Autodesk Inventor to UM CAD file format

You can use the instruction below to export data from **Autodesk Inventor** to intermediate UM CAD file format that later can be loaded into UM. This way is more general one. It can be used even if **Universal Mechanism** and **Autodesk Inventor** are installed on different computers. Plugin, which is used for exporting data from Autodesk Inventor into UM-compatible UM CAD file format, is freely distributed and available via the following link: http://www.universalmechanism.com/download/inventortoumaddin.exe. To load exported UM CAD file into **Universal Mechanism** the license for **UM CAD Interfaces** module is required.

To export data run **Autodesk Inventor**, load a part (**.ipt**) or an assembly (**.iam**), then select **Tools / UM Converter** tool panel.
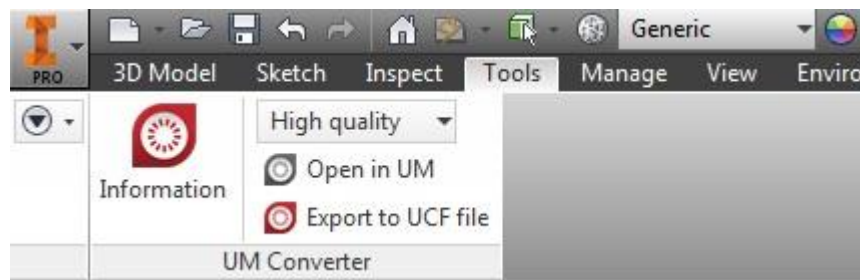
Figure 9.11. **UM Converter** panel in **Autodesk Inventor**

UM Converter tool panel includes the following controls.
- **Information** button shows **About** window.
- **Quality** drop down list is used to select required quality of the exporting data.
- **Open in UM** button runs UM and opens active 3D document in UM (requires UM 9 or later, otherwise the button will be disabled).
- **Export to UCF file** exports active document to UM CAD file format.

The installed **UM 9** or later is required to open the active 3D document in **UM Input**. If the UM version doesn't correspond the necessary one, the icon of the button will be gray, and when

you click it, the message about the requirement of the installation of newer UM version will display.

After export of the active 3D document to the **UCF** file it can be opened in the **UM Input** program and using the menu item **Tools | Import from CAD | UM CAD File** or having dragged the file into the **UM Input** working area.

9.4.2. Starting Autodesk Inventor import data from UM Input

If **Universal Mechanism** and **Inventor** are installed on one computer, the start of importing data from **Inventor** is possible directly from the **UM Input**. To do this, you should have **UM** 9 or later installed on your computer and a plugin for exporting data from **Inventor**.

To start the data import process, you must run the main menu **Tools | Import from CAD | Inventor**. If the menu command is not available, then you must reinstall the plug-in **Inventor** and export data from **Inventor**.

If all conditions are considered, the status of the **Inventor** application is defined and the further separation on the selection methods of the convertible **Inventor** document happens:

1. If **Inventor** app is not running, the **UM Input** activates the file selection dialog, where you can select the **part file (.ipt)** or **assembly file (.iam)**.
2. If **Inventor** is running, but there are no open files, the dialog of choosing files **Inventor (*.ipt** or ***.iam)** is activated.
3. If **Inventor** is running and there are one or more files opened, then a window with a list of open files appears on the screen. Select the necessary file in the appeared window. If the file is not on the list, you need to select a line **From file...**

After you select the file for conversion the opening or activation of the chosen file in **Inventor** happens. If **Inventor** is not running, then run the **Inventor** before the process of opening/activation of the file.

If the file opening in **Inventor** was successful, then the process of file conversion starts and on the screen appears a waiting form for the completion of the process.

After completing the conversion of the file, **UM Input** creates a new document and downloads in it the data from the **Inventor** file.

For correct automation operation, it is recommended to have only one running **Inventor** application.

9.5. Data import from SolidWorks

Data export is supported for 32-bit and 64-bit versions of **SolidWorks**, starting from **Solid-Works 2012** version. The free plug-in for data export from **SolidWorks** into the intermediate **UM CAD file (*.ucf)** can be downloaded via the following link:

http://www.universalmechanism.com/download/swtoumaddin.exe

An installed **.NET Framework 4.5.1** or later is required for correct work of data export from the **SolidWorks** environment. You can download **Framework 4.5.1** via the following link:

http://www.universalmechanism.com/download/fwork451.exe

To remove the plug-in you should close **SolidWorks** and run the uninstaller as administrator. An uninstaller **UninstallAddIn.exe** is located in the folder **C:\Program Files\Common Files\SWToUMAddIn**.

When you export the assembly, the color of the detail is taken from its color component in a part file **(*.sldprt)**. If you want to change the color of the conservation part in the assembly, it is necessary to open a separate part and to change its color.

9.5.1. Data export from the SolidWorks environment

Data export from **SolidWorks** into **Universal Mechanism** is performed through the intermediate **UM CAD format (* .ucf)**. The assembly or a part is exported from **SolidWorks** to UM CAD format (*.ucf) using a special plug-in. Then the file can be opened in **UM Input** program via the menu item **Tools | Import from CAD | UM CAD File** or by dragging it into the workspace **UM Input**.

This approach assumes that if the **Universal Mechanism** and **SolidWorks** are installed on different computers, **SolidWorks** documents can be saved in the intermediate UM CAD format (*.ucf), and then copied on a computer with an installed **Universal Mechanism** and loaded into UM model.

Please note that in order to download data in UM from the UM CAD file a license for the **UM CAD Interfaces** module is required.

To start the export in **SolidWorks** environment, open or set as active document the necessary part file or assembly file. Having selected the active document, you should go to **UM Converter** tab on the menu ribbon and activate the **UM Converter** panel if it is not active.
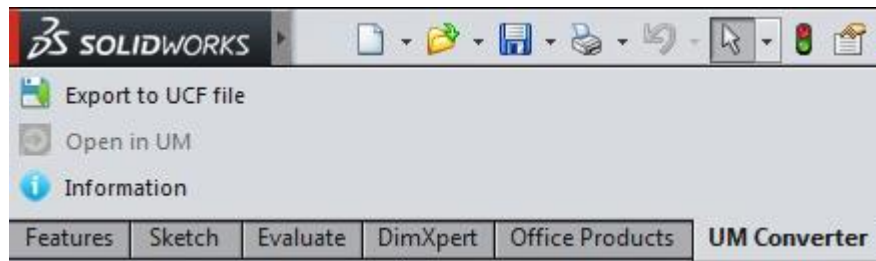

Figure 9.12. **UM Converter** panel in **SolidWorks** environment

Also, the start of the export from **SolidWorks** environment can be done by running the main menu commands. Activate the **SolidWorks** necessary document and select the needed command directly in **UM Converter** menu item.
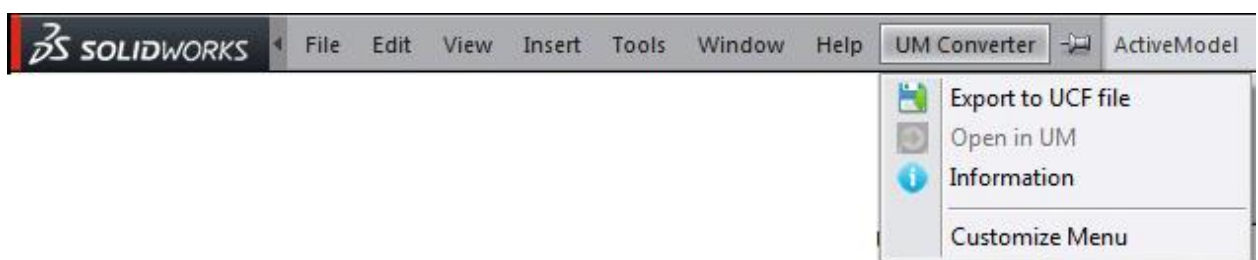

Figure 9.13. **UM Converter** menu items in **SolidWorks** environment

A menu ribbon tab and **UM Converter** main menu items of **SolidWorks** software contain the following control/command elements:

- **Information** – button/command which displays an information window about the add-in;
- **Open in UM** – button/command which cause the opening of the active 3D document in **UM Input**;
- **Export to UCF file** – button/command which causes saving of the active 3D of document in file *.ucf, defined by user.

To open the active 3D document in **UM Input**, you must have an installed **UM 9** or later. If the **UM** version does not match the required one, the message that the newer version of **UM** should be installed will be shown.

After you export the active 3D document in **UCF** file, it can be opened in the **UM Input** program via the menu **Tools | Import from CAD | File UM CAD** or by dragging the file into the workspace **UM Input**.

9.5.2. Starting the SolidWorks data import from UM Input

In case **Universal Mechanism** and **SolidWorks** are installed on the same computer, start data importing from **Inventor** is possible directly from **UM Input**. To do this, you must have **UM** 9 or later installed on your computer, and a plugin for data exporting from **SolidWorks**.

To start the import data process, you must run the main menu **Tools | Import from CAD | SolidWorks**. If the menu command is not available, then you must reinstall **SolidWorks** and export data plug-in from **SolidWorks**.

If all conditions are met to run the process, then the status determination of **SolidWorks** application and further division on the selection methods of the converted document **SolidWorks** starts:

1. If the **SolidWorks** application is not running, the **UM Input** activates the file selection dialog, where you can select the **part file (.sldprt)** or an **assembly file (.sldasm)**.
2. The file choosing dialog SolidWorks (*.sldprt or *.sldasm) activates if the application **SolidWorks** is running but there are no open files.
3. If the **SolidWorks** application is running and in it one or several files are opened, the window with a list of open files appears. In the appeared window, select the necessary file. If the file is not in the list, then you need to select in the list a line **From file...**

After you select the file for conversion, it is opened or activated in **SolidWorks**. If **Solid-Works** is not running, then before the opening/activation process of file the **SolidWorks** running is executed.

If a file opening in **SolidWorks** was successful, then the process of file conversion starts, and on the screen appears the waiting form for the completion of conversion process.

After completing the file conversion, **UM Input** creates a new document and downloads in it the received data from **SolidWorks** file.

For correct operation of the automation it is recommended to have only one running copy of the **SolidWorks** application.

9.6. Data import from KOMPAS-3D

9.6.1. Data import from KOMPAS-3D

When converting data from KOMPAS-3D documents a user is suggested to choose the necessary file: *.m3d (detail) or *.a3d (assembly).

After choosing the command, UM checks whether the KOMPAS-3D application is active, if not then starts it, downloading the selected part or assembly as the active document. The graphic images and inertial parameters of a separate part or of all the parts included in the assembly are converted.
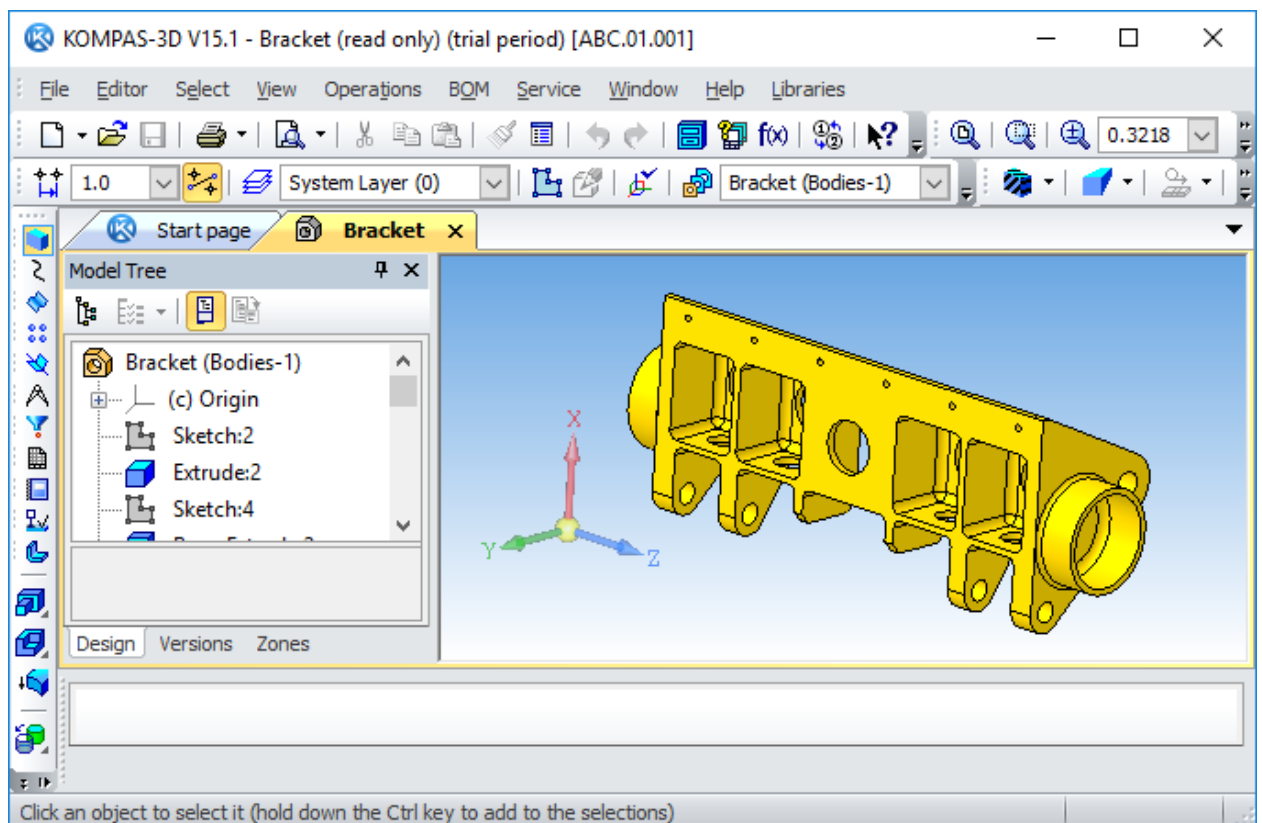
Figure 9.14. Bracket.m3d part in KOMPAS-3D

After downloading the active document is converted into UM format. In case of a part converted data is inserted into an active UM object. When it is necessary a new UM object is created. In case of assembly the converted data is inserted into the automatically generated UM object.

After the data conversion, UM-joints and force elements should be entered into the object.
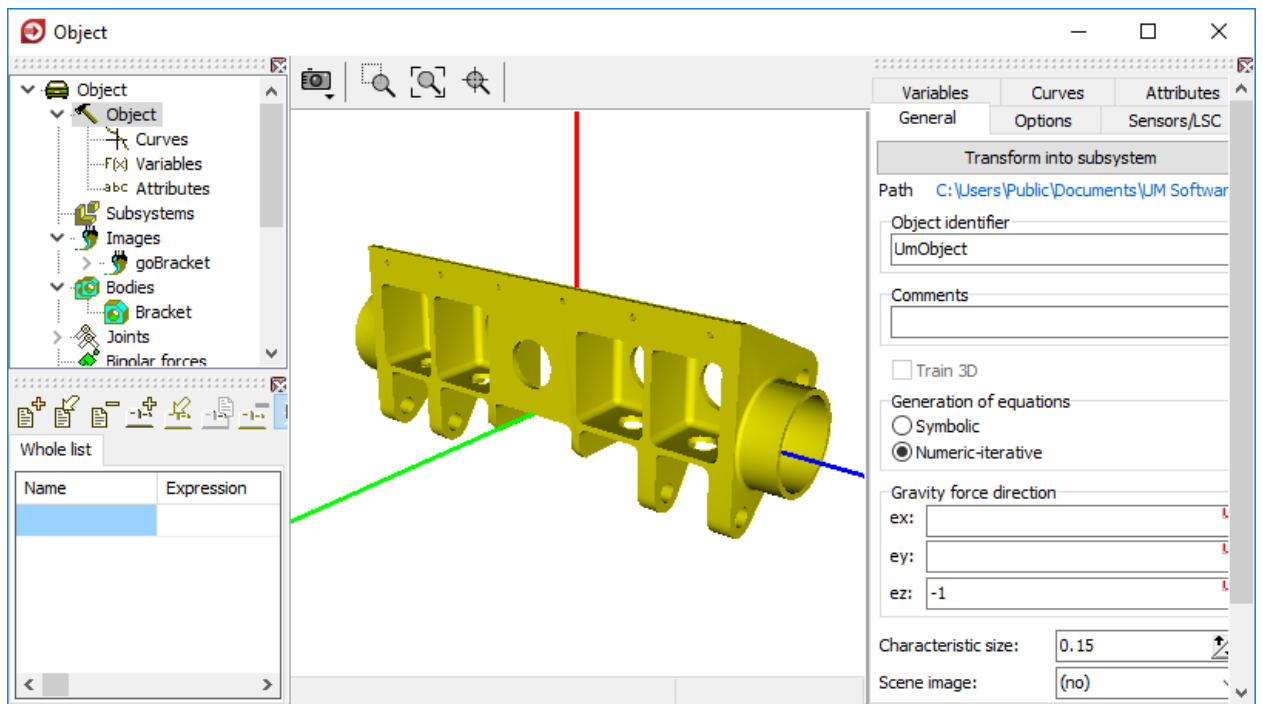
Figure 9.15. Valve body detail in UM
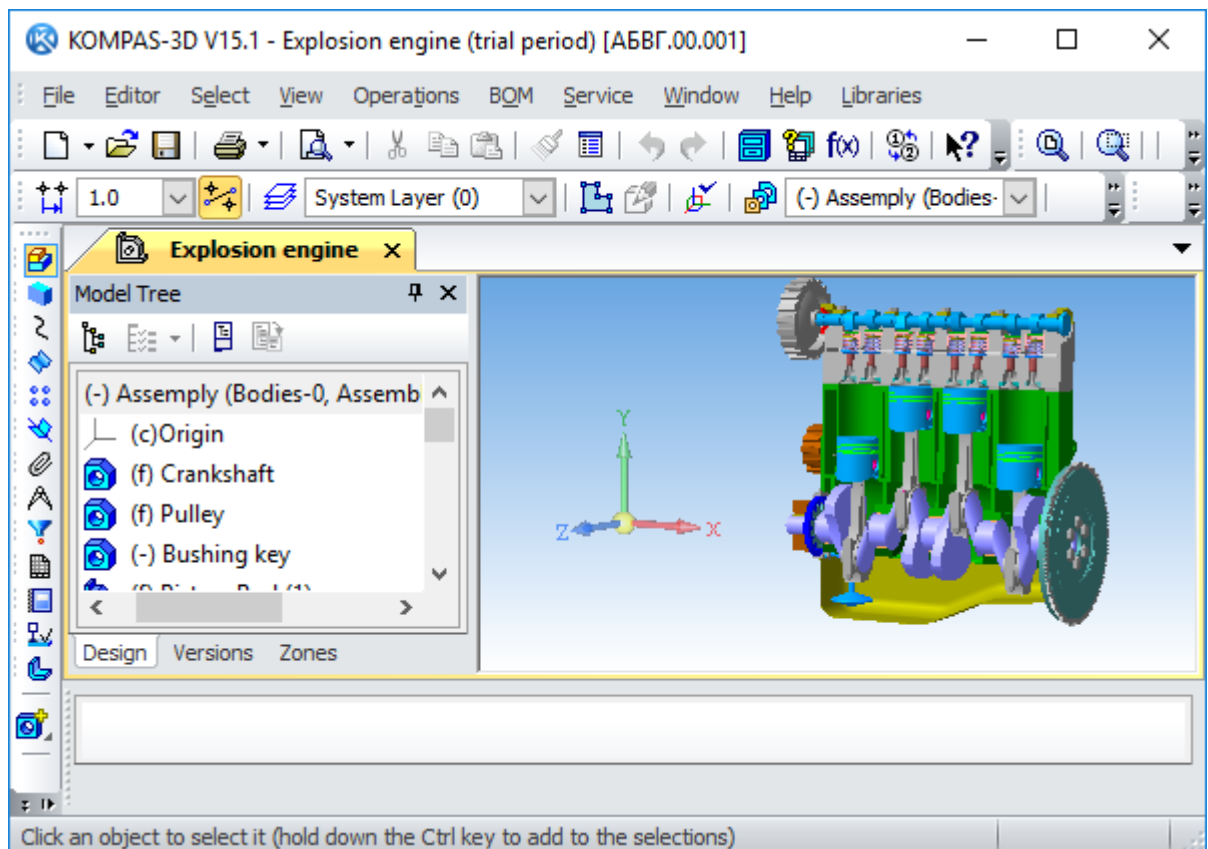
Figure 9.16. "Explosion engine.a3d" assembly in KOMPAS-3D
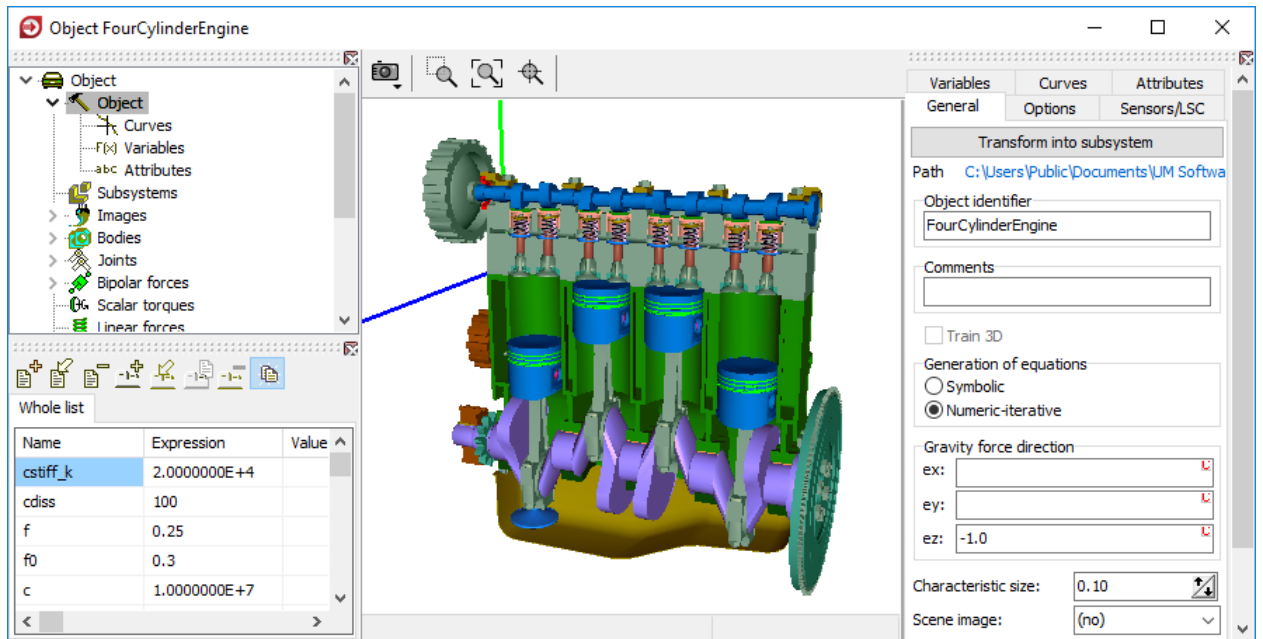
Figure 9.17. "Explosion engine" corrected and improved assembly in UM, input program
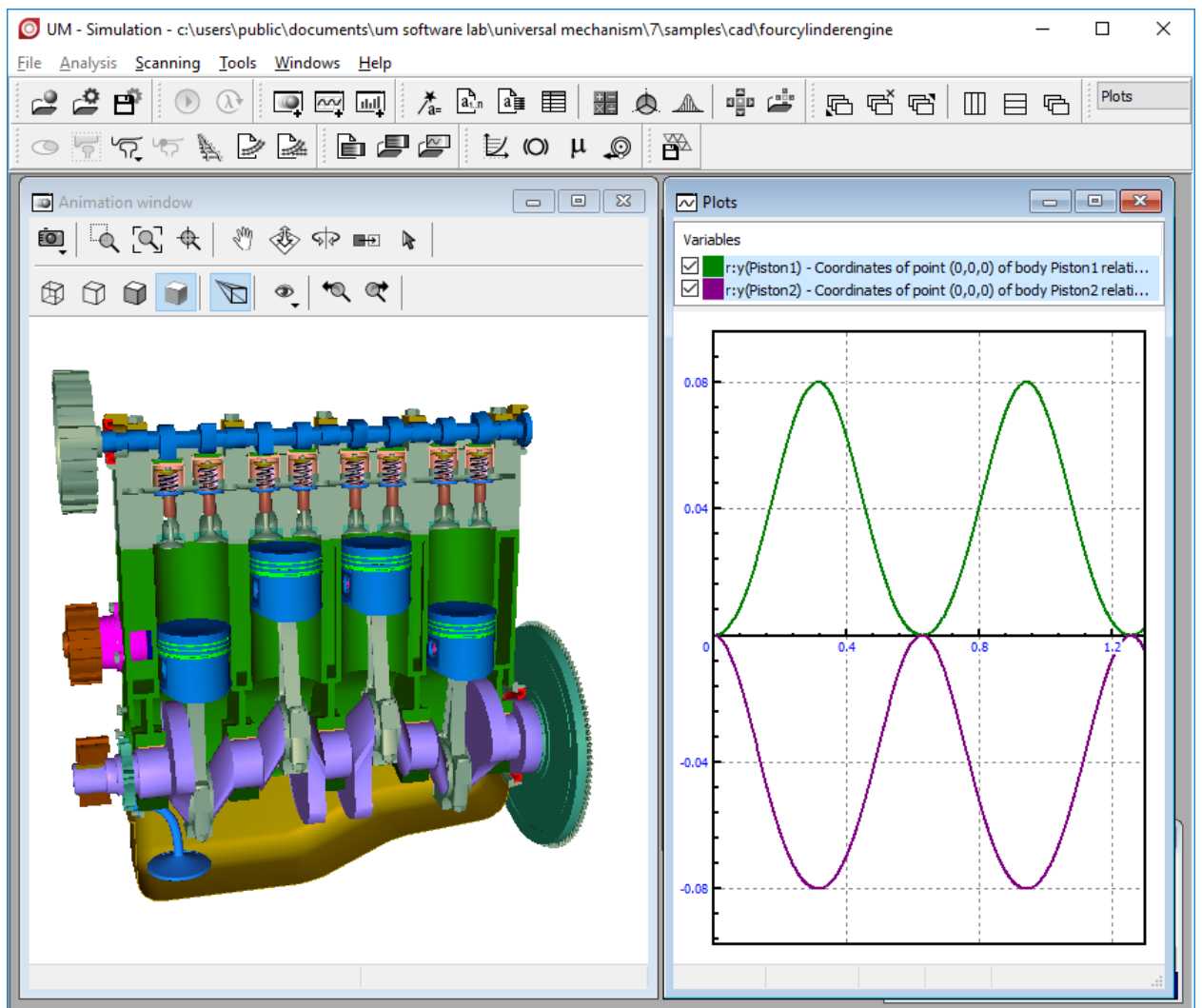
Figure 9.18. Dynamic study of the "Explosion engine" assembly in UM

9.6.2. Saving KOMPAS-3D documents in UM CAD format

3D KOMPAS-3D documents can be converted into dynamic models with usage of their conversion into *.ucf files (UM Cad File). This function greatly broadens the user`s opportunities, because it allows to save the documents in the format which is compatible with UM if UM is not installed on the local computer.

To save the KOMPAS-3D document in UCF format take the following steps:

- copy the library KompasToUCFv8p.rtw (KOMPAS-3D v8+ and later) or KompasToUCFv7p8.rtw (KOMPAS-3D v7 Plus, v8) on the local computer with an installed KOMPAS-3D;
- add the library to the list of KOMPAS-3D application libraries (name of the library is "Graphics export in UM");
- download the necessary document and execute the command of the library "Save in UCF format";
- after the conversion completion, transfer the ucf file on the computer on which the UM is installed;
- run **UM Input** program and select the menu item **Tools | Import from CAD | UM CAD file**.

KompasToUCFv8p.rtw, KompasToUCFv7p8.rtw libraries are supplied with UM (**{UM}**\bin directory).

Remark 1.	When converting data from the KOMPAS-3D v7+ and later, saving cam profiles in the UCF-format is unavailable.
Remark 2.	Please note that KOMPAS-3D has no English graphic user interface since v. 16 and later.

9.7. Data import from Unigraphics

Export data is supported for 32-bit and 64-bit versions of **Unigraphics**, starting from version **Unigraphics NX 9.0**. The free plug-in for data export from **Unigraphics** in the interim **UM CAD file (*.ucf)** can be downloaded via the following link: http://www.universalmechanism.com/download/ugtoumaddin.exe

For correct work of data export from **Unigraphics** environment, the installed **.NET Framework 4.5.1** or later environment is necessary. You can download **Framework 4.5.1** from the **UM** site via the following link:
http://www.universalmechanism.com/download/fwork451.exe

To remove the plug-in you should close **Unigrapics** and **run** the uninstaller **as administrator**. Uninstaller **UninstallAddIn.exe** is located in the folder **C:\Program Files\Common Files\UGToUMAddIN.**

9.7.1. Data export from Unigraphics

Data export from **Unigraphics** in **Universal Mechanism** is fulfilled via an interim **UM CAD format (*.ucf)**. The assembly or a detail is exported from **Unigraphics** into UM CAD format (*.ucf) by using a special plug-in. Then the file can be opened in **UM Input** program via the menu item **Tools | Import from CAD | UM CAD File**.

This approach assumes that if the **Universal Mechanism** software and **Unigraphics** are installed on different computers, **Unigraphics** documents can be saved in the interim UM CAD format (*.ucf), and then copied on the computer with installed **Universal mechanism** and loaded into UM model.

Note that a license for **UM CAD Interfaces** module is required to load data in UM from the file in UM CAD format.

To start the export in **Unigraphics** environment, you should open the document or make active the necessary file of a part or assembly. When you selected the active document, you should go to **UM Converter** tab on the menu and activate the panel **UM Converter** in case it was not activated.

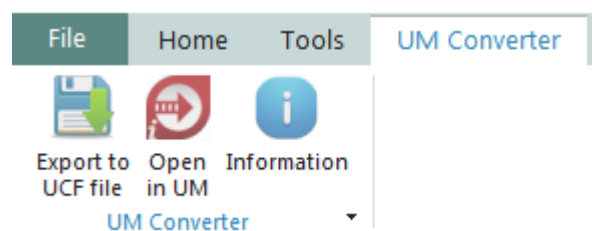Figure 9.19. UM Converter panel in **Unigraphics NX** environment

UM Converter tab on the **Unigraphics** menu contains **UM Converter** panel with the following controls:

- **Information** button shows the form with brief information about the plugin;
- **Open in UM** starts the opening of the active 3D document in **UM Input**;
- **Export to UCF file** saves your active 3D document into a user-specified *.ucf file.

To open the active 3D document in **UM Input**, you must have **UM 9** or a later version of **UM** installed on your computer. If the **UM** version does not match the required one, when you click on it a message indicating that the installation of a newer **UM** version is required will appear.

After you export the active 3D document to **UCF** file, this file can be opened in **UM Input** program and via the menu item **Tools | Import from CAD | UM CAD file.**

For correct work of data export from the **Unigraphics NX** environment you need **.NET Framework 4.5.1** environment or a later version to be installed. To download **Framework 4.5.1** use the following link: http://www.universalmechanism.com/download/fwork451.exe.

9.7.2. Direct data transfer from Unigraphics to UM

If **Unigraphics NX** and **Universal Mechanism** are installed on the same computer it is possible to start importing data from **Unigraphics** right from **UM Input** program. You need the **UM** 9 or later installed on a computer and plug-in for data export from **Unigraphics**.

Use menu command **Tools | Import from CAD | Unigraphics** from **UM Input** program to run data transfer. If the menu command is inactive, reinstall **Unigraphics** and data export plug-in from **Unigraphics**.

If all conditions to run the process are executed, then the status determination of **Unigraphics** application and further division in the selection methods of the **Unigraphics** converted document happens:

The subsequent behavior of the program depends on state of **Unigraphics** application.

6. If **Unigraphics** is not run, **UM Input** shows the file open dialog to select the target **Unigraphics** file (*.prt) to import.

7. If **Unigraphics** is already running, but has no opened documents (parts and assemblies) then Unigraphics file open dialog is activated.

8. If **Unigraphics** is already running and has one or more loaded parts or assemblies, the dialog with the list of all loaded documents appears and a user may select one of the loaded files to be imported into UM. If you want to import the file that is not in the list use **From file** option instead.

After that **Unigraphics** is activated, the selected document is loaded into Unigraphics environment and importing is started. At the end of data transfer new UM document with just imported data is created.

It is recommended to have the only running instance of **Unigraphics** on your computer for **correct data import**.

9.8. Data import from PTC Creo

Data export is supported for 32- and 64-bit versions of **PTC Creo**, starting from version **Creo 1.0**. The free plug-in to export data from **PTC Creo** in the intermediate **UM CAD file (*.ucf)** can be downloaded via the following link:

http://www.universalmechanism.com/download/creotoumaddin.exe

For correct work of the data export from the **Creo** environment an installed **.NET Framework** version **4.5.1** or later environment is required. You can download **Framework 4.5.1** via the following link:

http://www.universalmechanism.com/download/fwork451.exe

To remove the plug-in you must close **Creo** and **run** the uninstaller **as administrator**. The uninstaller **UninstallAddIn.exe** is located in the folder **C:\Program Files\Common Files\CreoToUMAddIN**.

Data export from **PTC Creo** in the **Universal Mechanism** is performed through the intermediate UM CAD format (*.ucf). The assembly or a part by using a special plug-in is exported from **PTC Creo** in UM CAD format (*.ucf). Then the file can be opened in **UM Input** program via the menu item **Tools | Import from CAD | UM CAD File** or by dragging it into the workspace **UM Input**.

This approach assumes that if the **Universal Mechanism** and **Creo** are installed on different computers, **Pro/ENGINEER** files can be saved in the intermediate UM CAD format (*.ucf), and then copied on the computer with an installed **Universal Mechanism** and loaded into UM model.

Please note that in order to download data in UM from UM CAD file a license on module **UM CAD Interfaces** is required.

To start the export in **Creo** environment you should open or make an active document the necessary part of the detail or an assembly file. Having selected the active document, you should go to the **Tools** tab on the ribbon menu and on the **TOOLKIT** panel click on the **Tools** dropdown list. Activate **UM Converter** menu item and select the necessary command in the dropdown list, wait for the completion of the command.
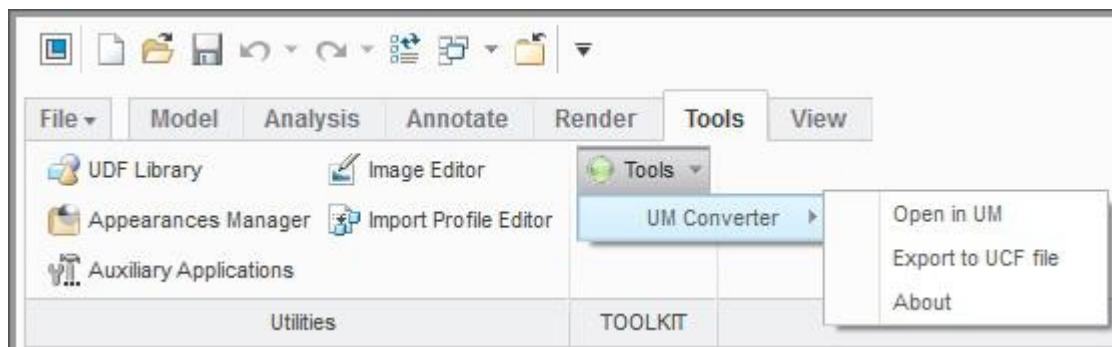
Figure 9.20. UM Converter menu items in **Creo** environment

UM Converter menu item contains the following commands:

- **About** – a command which causes the information form demonstration about the add-in;
- **Open in UM** – the command which causes the opening of the active 3D document in **UM Input**;
- **Export to UCF file** – the command which causes the saving of the active 3D document to a file *.ucf, defined by user.

To open the active 3D document in **UM Input**, you should have the **UM 9** or later installed on your computer. If the **UM** version does not match the required one, by clicking on it will indicate that the newer version of **UM** should be installed.

After you export the active 3D document in **UCF** file, it can be opened in the **UM Input** program via the menu item **Tools | Import from CAD | UM CAD file** or by dragging the file into the **UM Input** workspace.

When you export the assembly, the part color is taken from its current color component. If the part color has not been set in the mode of assembly editing, then when you export the assembly the color of the part will be grey.

9.9. Data import from Pro/ENGINEER

Data export is supported for 32-bit and 64-bit versions of **Pro/ENGINEER Wild-Fire 4** and **5**. The free plug-in for data export from **Pro/ENGINEER** in the intermediate **UM CAD file (*.ucf)** can be downloaded via the following link:

http://www.universalmechanism.com/download/proetoumaddin.exe.

For correct work of data export from **Pro/ENGINEER** environment an installed **.NET Framework 4.5.1** or later is required. You can download **Framework 4.5.1** via the following link:

http://www.universalmechanism.com/download/fwork451.exe

To remove the plug-in you should close the **Pro/ENGINEER** and run the uninstaller **as administrator**. **UninstallAddIn.exe** uninstaller is located in the folder **C:\Program Files\Common Files\ProEToUMAddIN**.

Data export from **Pro/ENGINEER** in the **Universal Mechanism** is executed via the intermediate **UM CAD format (*.ucf)**. The assembly or part by using a special plug-in is exported from **Pro/ENGINEER** in the UM CAD format (*.ucf). Then the file can be opened in **UM Input** program via the menu item **Tools|Import from CAD|UM CAD File** or drag it into the **UM Input** workspace.

This approach assumes that if the **Universal Mechanism** and **Pro/ENGINEER** are installed on different computers, **Pro/ENGINEER** files can be saved in the interim UM CAD format (*.ucf), and then copied on a computer with an installed **Universal Mechanism** and loaded into UM model.

Please note that in order to download data in UM from UM CAD file a license for the module **UM CAD Interfaces** is required.

To start exporting in **Pro/ENGINEER** environment open or make an active document the necessary part file or assembly file. Having selected the active document, select **UM Converter** menu item in the main program menu, wait for the completion of the command.

Figure 9.21. UM Converter menu items in **Pro/ENGINEER** environment

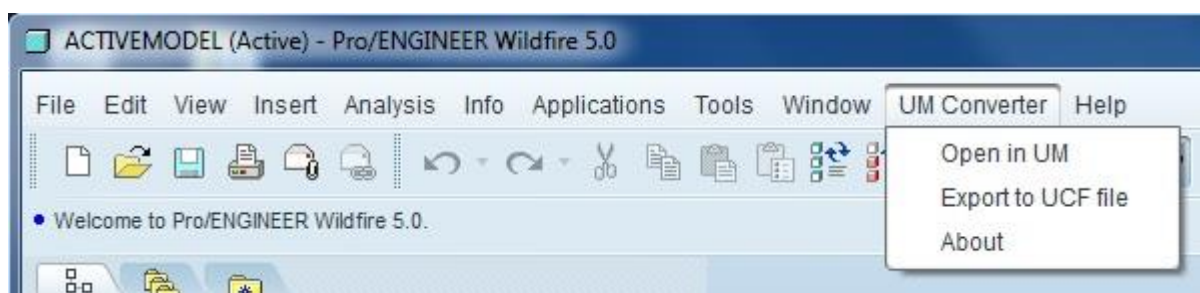**UM Converter** menu item in the **Pro/ENGINEER** program main menu contains the following commands:

- **About** – a command which causes the information window demonstration about the add-in;
- **Open in UM** – a command which causes the opening of the active 3D document in **UM Input**;
- **Export to UCF file** – a command which causes the saving of the active 3D document to a file *.ucf, defined by user.

To open the active 3D document in **UM Input**, you should have **UM 9** or later installed on your computer. If the **UM** version does not match the required one, by clicking on it will indicate that a newer UM version is required.

After you export the active 3D document in **UCF** file, this file can be opened in **UM Input** program and via the menu item **Tools | Import from CAD | File UM CAD** or by dragging the file into the **UM Input** workspace.

When you export the assembly, the part color is taken from its current color component. If the part color has not been set in the mode of assembly editing, then when you export the assembly the color of the part will be grey.

9.10. Dynamic object as a result of data conversion

Right after the conversion, the object contains one body for each of the parts of the assembly. The name of the body corresponds to the long name of the part including assembly names in the model tree. UM requires unique names of bodies. If necessary, digits 1,2,... are added to the name to make it unique.

A local system of coordinates (LSC) is assigned to each body. Position of LSC for zero value of coordinates coincides with the SC0. Motion of the LSC by simulation corresponds to the motion of the corresponding body.

The following inertia parameters of bodies are imported: mass, moments of inertia, coordinates of center of mass/gravity.

Two types of graphic objects (GO) are created for each part. GOs of the first type, the **parent GO of the parts**, are part images in their local system of coordinates without translations and rotations made in assemblies. If a set of parts in assembly is generated by one part file, only one parent GO is created with the name of the first part of the set.

The second type GOs are the assigned images with the name <go> + <Part Name>. These GOs are referred to the parent GOs, and their positions correspond to location of the corresponding part in the assembly. SC of the assigned GO defines the LSC of the body.

If the assembly contains mates, the converter tries to build joints (kinematic pair). If no mates found or if the conversion of mates fails, a six degrees of freedom joints with the name <j> + <Part name> are assigned to bodies, and the corresponding body may freely moves relative to SC0.

Use the **File | Save as** menu command to store the imported data. Two files are created simultaneously by this command in the model directory:

- *input.dat* − data in the standard UM format of model description;
- *input.umd* – data in an extended UM format, which contains additional information about the assembly structure, and allows processing the model after the conversion, in particular, joining several parts in one rigid body, (Sect. 9.11. *"Model processing after conversion"*, p. 9-26).

9.11. Model processing after conversion

As a rule, the dynamic model should be revised after the conversion due to the following causes:

- CAD programs do not support interaction of bodies and force elements;
- very often an assembly contains hundreds and thousands of parts, which can be united into several rigid bodies;
- mates are often used for positioning of parts, and not for creating joints (revolute, prismatic, universal joints etc.); an additional development of kinematic scheme of mechanism directly in UM is necessary.

It is recommended to use the extended UM format of assembly data (input.umd) while processing the imported data. Use the **File | Open *.umd** or **File | Reopen** menu command to open the *input.umd* file. Each time the user saves the data in extended format, the renewed standard data file *input.dat* is created in the same directory. This file is used for simulation of the model dynamics with the *UMSimul.exe* program. It is not recommended to modify the *input.dat* directly until the model is fully developed, because in this case the *input.umd* contains old information, and its further modification causes loss of the *input.dat* modification data.

9.11.1. Joining parts to bodies

A rigid body may include any set of parts, which are fixed relative each other. Each body immediately after the assembly conversion contains one part. To edit the list of parts for a body:

- select a body in the model list of elements;
- open the Parts tab of the inspector; the tab contains the full list of parts, and parts included in the body are checked;
- include/exclude any number of parts in the list by checking/unchecking the parts; use the special utilities for joining parts when the number of parts is large;
- accept or reject the modification by the corresponding buttons.

The body, which part belongs to, is called the **owner body**.

Including/excluding a part in/from a body involves the following actions:
- inertia parameters of new owner body are automatic recomputed (mass, moments of inertia, coordinates of center of mass/gravity);
- graphic element corresponding to the part is added to / removed from the graphic object of the body
- in case of adding, the part is removed from the old owner body;
- empty bodies (without parts) are removed, their image are removed too;
- a joint connecting the removing body is removed if another connecting body is Base0 or the new owner body, otherwise the body in joint pair is replaced by the new owner body.
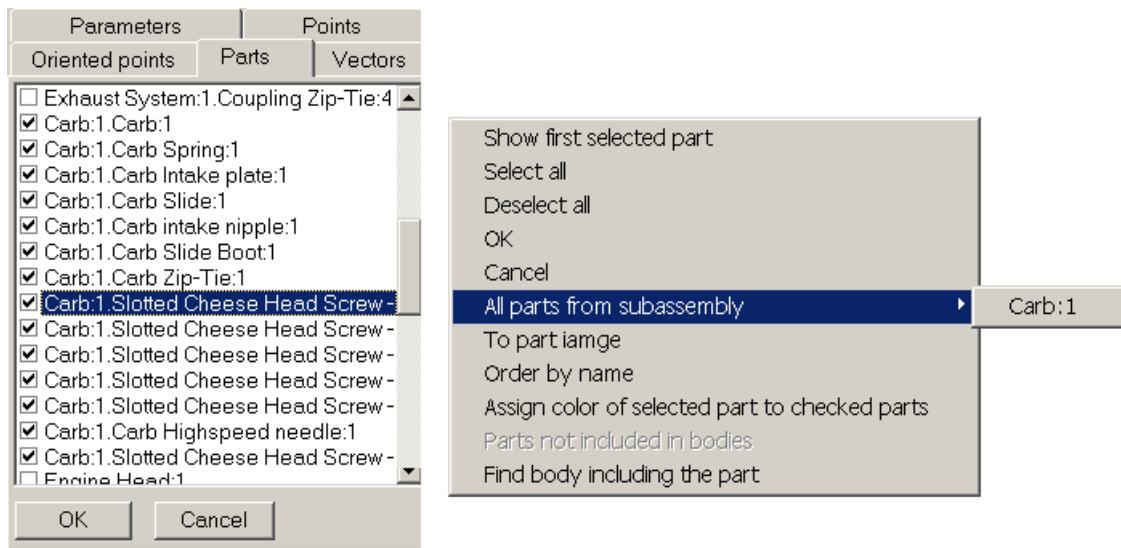
Figure 9.22. List of parts; commands of popup menu

Group selection of parts is a useful operation while unification of dozens of parts in a body (the left figure). The checked/unchecked status of selected parts is changed to opposite ones by clicking the Enter key.

Windows standards are used for group selection of elements in the list:
- **Shift** + ↑(↓) – selection of a successive located elements of the list;
- **Shift + Mouse click on the group begin element + Mouse click on the group end element** – selection of a successive located elements of the list;
- **Ctrl+Mouse click on an element** – selection/deselection of a single element.

The most effective unification of parts is possible if a whole subassembly corresponds to a body. Click by the right mouse button on one of the parts of the subassembly and select the name of the subassembly in the popup menu (a submenu of the **All parts from subassembly** menu command, the right figure)

Other useful commands and operations
- Switch the animation window to the whole object mode by the 🔲 button. Click by the mouse on the image of the necessary part/body. The corresponding part or the part list included in the body are selected in the list of part.
- The **To part image** command activates the parent GO of the part (Sect. 9.7. *"Data import from Unigraphics"*, p. 9-20), where the user can change its color or make its invisible.
- The **Assign color of selected part to checked parts** command assigns the same color to all checked parts.
- The **Parts not included in bodies** selects all corresponding parts, e.g. to find lost parts.