



Advance Steel

User's Guide



This document contains a brief description of the software functions and is not a replacement for the training program. This guide includes information about all the modules, including those that are optional. For detailed information regarding the program's functions, refer to the online help provided in Advance.

In case of any discrepancy between the information given in this guide and the information given in the software, the software is the most up to date source.

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TABLE OF CONTENTS

WELCOME	7
Introduction	8
Advance	8
Specialized areas	9
Advance and AutoCAD®	9
Technology	9
Communication options	10
Individual preferences	10
Chapter 1 Installation	11
General	12
System requirements	12
Hardware	12
Software	12
Distribution	12
License	13
Obtaining a user license	13
Using USB dongles	13
Installation	13
Server and Client configuration	17
Authorizing the software	19
License management	21
Converting databases (Merging)	22
Support	25
Support by telephone	25
Chapter 2 The 3D Model	27
Elements of the 3D model	28
Beam & Plate	28
Processing	28
Bolt patterns & welds	29
Joints	30
Structural elements	31
Auxiliary objects	31
Special parts	32
3D Modeling workflow	33
Chapter 3 Advance interface	35
Starting the program / working environment	36
Advance toolbars / menus	36
Using Advance	39
Other important tools for using Advance	39
Creating Advance objects	40
Object properties	41
Layer	42
Chapter 4 Creating a 3D Model	43
Creating a building grid	44
Creating a beam	46
Straight beams	47
Compound sections	48
Curved beams	48
Poly beams	49
Folded profile	50
Construction systems (Cold rolled profiles)	51
Welded beams	52

Creating plates	53
Rectangular plate	53
Polygon plate	54
Gusset plates	55
Splitting / merging - plates	56
Shrinking/enlarging polygon plate	56
Folded plate	57
Coordinate systems	58
Object Coordinate Systems	59
Coordinate systems at curved beams	60
UCS at a bisecting line	61
Defining a coordinate system	61
Beam processing	62
Processing objects	62
Rule-based beam processing	63
Process section	63
Section contour	64
Plate processing	65
Plate processing – weld preparation	66
Cut plate at UCS	67
Cut plate at plate	67
Plate processing – independent from the UCS	67
Plate processing – dependent on the UCS	68
Corner processing	69
Polygon plate processing	69
Changing Advance objects	70
Changing objects using grips	70
AutoCAD® Manipulating tools	71
AutoCAD® command properties	71
Advance command properties	71
Representation Type – Snaps – Grips	72
Bolt and Hole Patterns / Shear Studs	74
Creating a bolt pattern	75
Creating a hole pattern	76
Creating shear studs	76
Shifting a bolt/hole pattern	76
Splitting a bolt/hole pattern	77
Recalculating a grip length	77
Bolts on gauge lines	77
Welds	77
Connections	78
Changing connections	78
Checking connections	79
Working methods I	80
Selecting objects	80
Object filter	80
Marking / Unmarking objects	81
Searching / Marking objects	81
Changing representation type	82
Views on the model	83
Structural elements	83
Portal frames	84
Flat frames	84
Single-span Bracings	84
Purlins	85
Pylons	85
Joists	85
Half trusses	86
Joints and connection objects	86
Joints	87
Stairs	107
Connection objects	108

Special parts, special sections	109
Special parts	109
User sections	110
Working methods II	112
Model Browser	112
Model views	113
Advance – copy / rotate / mirror	113
Chapter 5 Numbering	115
Numbering tool	116
Starting the numbering	117
Numbering options	118
Checking the results.....	118
Numbering with standard parts	118
Deleting part marks	119
Assigning / Changing a prefix	119
Prefix configuration	119
Creating a main part.....	120
Creating an assembly part	120
Chapter 6 Checking the model	121
Checking the model for errors	122
Collisions in the model	123
Displaying checking results.....	123
Displaying checking results again	124
Advance Audit checking	124
Advance audit checking (database).....	124
Steel construction checking	125
Defining the center of gravity	125
Updating defaults	125
Numbering validation.....	125
Displaying objects with identical part marks	126
Selecting objects with different part marks	126
Marking the main part of assembly	126
Marking loose parts.....	127
Marking parts without marks	127
Checking for duplicate part marks	127
Identifying objects with identical part marks	127
Differences between two objects	127
Chapter 7 Lists / Bills of materials	129
Creating lists.....	130
Creating a model extract	130
Creating a structured BOM.....	132
Document Management – Structured BOM	134
Chapter 8 Creation of general arrangement and shop drawings.....	135
Drawings.....	136
Model - Drawing principles.....	136
Drawing styles	136
Drawing management	136
Processes	137
Drawing creation and AutoCAD® objects	137
Creating drawings	137
Prototype files	138
General arrangement drawings.....	138
Drawing management	140
Document management.....	140
Document Manager - drawing details	141
Registering/canceling drawing from model	144
Drawing style management	144

Drawing layout.....	144
Creating/changing project information	144
Node details.....	145
Elevations	146
Single part drawings	147
Tie beam.....	148
Columns	148
Assembly drawings.....	149
Special structural parts	149
Special structural parts.....	150
Alternative drawing styles.....	151
Processes	153
Drawing processes.....	153
Cameras.....	155
Alternative drawing processes	156
Labeling and dimensioning.....	157
Drawing cleanup.....	157
Appendix.....	159
HSBasis.....	161
HSDetailing	165
HSConnection	165
HSExtended	166
HSCollision.....	166
HSDetailingBasis.....	166
HSDetailing	167
HSIFDSTVBOM	167
HSIFDSTVNC	167
HSIFPM (HSExtended)	168
HSSTAAD	168
Index.....	169

WELCOME

GRAITEC has developed a global CAD / Analysis & Design platform that allows you to verify and optimize your structures. It accelerates the creation of reinforced concrete and steel structures with tools dedicated to modeling and accelerating document creation for the construction process: general arrangement drawings, detailing, manufacturing drawings, lists, BOMs, etc. Advance includes:

- The Steel suite for creating structural steel arrangement drawings, shop drawings, lists and NC files
- The Concrete suite for creating reinforcement and structural concrete drawings
- The Design suite for the analysis and design of reinforced concrete and steel structures and automated creation of design reports

Advance is part of the GRAITEC CAD / Analysis and Design suite and therefore directly connects to other GRAITEC engineering solutions.

This user guide, dedicated to the Steel suite, is structured into 8 chapters describing a steel construction project from start to finish.

*All software tools described in this guide and all remarks related to the product pertain only to the Advance Steel suite and for reading simplification only the generic name **Advance** is used.*

In this chapter:

- *Introduction*
- *Specialized areas*
- *Advance and AutoCAD®*
- *Technology*
- *Communication options*
- *Individual preferences*



Introduction

This user guide for Advance is an introduction to working with Advance, describing the basic Advance methodology with detailed description of the most important tools.

The user guide can be used as a learning tool but is also useful as a basic reference for individual topics using the index.

Since not all Advance tools are described in detail in this guide, refer to the *online Help* for more information on all commands and parameters. The general Advance work methodology is explained in this **Introduction**, including typical industry applications for Advance, exchanging information and specific user software configuration.

The **Creating a 3D model** chapter explains the use of the most common Advance construction elements (i.e., beams, plates, connection elements, structural elements such as stairs) and joints. Using simple examples, a general overview of the basic tools and methods used to create a model is presented.

The **Numbering** chapter describes the options provided by Advance to automatically assign model elements with clear single part and assembly marks.

The various methods for **Checking the model** in Advance will insure a correctly built 3D model and accurate bills of materials. This chapter describes the tools for checking *collisions in the model* and *correctness examinations*.

The automated document creation such as bills of materials from the 3D model is presented in the **Lists / Bills of materials** chapter.

The **Creation of general arrangement and shop drawings** chapter provides an overview of the diverse options for automatically creating general arrangement drawings, fabrication, and fitting drawings. It also demonstrates detail drawing cleanup.

Advance

Advance is a leading edge steel construction application integrated into the latest AutoCAD® version under the Windows operating system.

With intelligent Advance objects, a three dimensional model is created and stored in a drawing (in DWG format).

The Advance **Model** forms the basis of the 3D construction. It contains and manages objects (e.g., beams, plates, bolts, welds) including their features and relationships to each other. Complex structures are created using Advance structural elements (e.g., a portal frame or a stairway) with all the required features, joints, and connections, within a command.

The Advance model becomes the master reference for other tools.

- Dimensioned and labeled general arrangement and shop drawings are automatically created from the model information. They are stored in separate DWGs using the user preferences for format, page setup and drawing look and feel (the Advance drawing style). It is also possible to create several details on a single sheet.
- The general arrangement and shop drawings are created from the information contained in the model and are managed by the Advance **Document Manager**. The update tool in the Document Manager makes single click drawing adjustments possible after model changes.
- Structured BOMs (bills of materials) and NC-information are also created from the model and include all model information such as part marks and quantities. These documents are also managed by the Document Manager. Bills of material/structured BOMs and NC-information are listed and previewed in the Document Manager.

Through multiple interfaces, the communication with GRAITEC and 3rd party engineering applications is guaranteed.

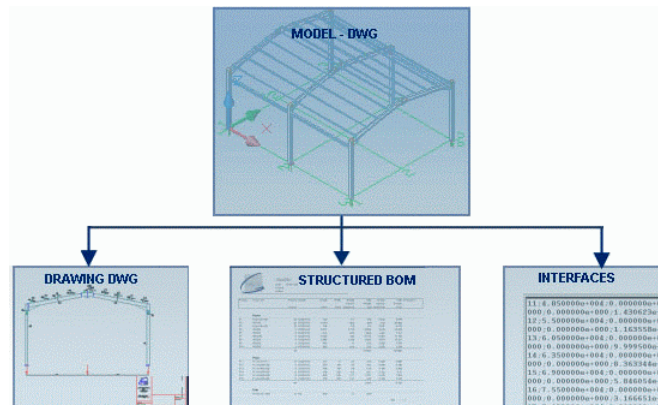


Figure 1: Document creation

Specialized areas

The Advance 3D steel construction software is adapted to both standard and specialized construction. A variety of cold rolled construction systems for purlins, side rails, accessories and cladding sections are available in Advance (i.e., Albion, Ayrshire, Canam Manac, Canfer, Fisher, HiSpan, Kingspan, Krupp, Metsec, SAB, SADEF, Staba, StructuralSections, Ward, Zeta, etc.). Individual sections and other elements are easily set as either user sections or special elements and stored in tables (libraries) for reuse.

The construction rules previously implemented are applicable for existing construction systems, Advance elements, and other elements.

Advance and AutoCAD®

The latest AutoCAD® version has been extended (through the ARX-Technology) with specific steel construction elements such as beams, plates and bolts. Advance elements are individual objects that are used like standard AutoCAD® objects.

Advance is completely integrated into AutoCAD® making it easy and intuitive to learn. Advance benefits from the latest AutoCAD® ARX Technology and makes the best use whenever possible of the existing work methodology including manipulation tools on geometric elements, snaps and grips, etc. Therefore, users are immediately familiar with the basic tools.

AutoCAD® serves both as a graphic engine and an object oriented database for Advance. The complexity of commands is reduced as Advance objects are processed within AutoCAD® tools and all information is stored in the DWG.

Advance's integration within the AutoCAD® user interface is fully optimized. All Advance tools are grouped by type in space saving toolbars.

Technology

Advance uses the latest standard industry technologies such as Windows and AutoCAD® for tight integration in the Microsoft Office environment. Advance information, construction rules and tables (libraries) are stored in MS-Access databases.

The ODBC technology links the model and drawings and provides quick communication for joint macros construction rules.

The values entered in the Advance user menus are directly converted to a graphic display on the screen (through the MFC-user interface) so that the effects of a new value are viewed immediately.

The Facet modeler enables large model manipulation with high speed since the file size can be kept small.

Advance is MDI (Multi Document Interface) capable. This means that several drawings with models and linked detail drawings can be opened simultaneously within a single AutoCAD® session. With this functionality, elements are copied from one drawing to another using **drag and drop**.

Communication options

Advance elements are saved as proxy-graphics with lines or surfaces. Therefore, Advance DWGs can also be viewed in **standard** AutoCAD®. The saved proxy-graphics can be clicked through a system variable.

From the Advance model, information transfer files are created by PCS (Program system in the steel construction) or PSS (Interface product of the steel construction).

Individual preferences

Advance has default values for immediate creation without extensive configuration. Advance is preconfigured to use common profile sizes, standard plate thicknesses, standard bolt information, etc.

The default values are changed with the Advance **Management Tool**.

The default intelligent joint settings based on section sizes are set to individual user requirements by saving the information in an easy to use library.

Various drawing styles containing rules for dimensions, labeling and presentation of objects, are used for creating drawings from the model. Advance contains a variety of preset drawing styles accessible through a menu for automatic drawing creation: General Arrangement drawings, Single part shop drawings, assemblies and many more.

Moreover individual drawing styles can be user defined. The creation and editing of user defined drawing styles is described in the **Drawing Style Manager** guide.

Chapter 1

Installation

In this chapter:

- *System requirements*
- *Distribution*
- *License*
- *Using USB dongles*
- *Installation*
- *Server and Client configuration*
- *Authorizing the software*
- *License management*
- *Converting databases (Merging)*
- *Support*



General

Read this user guide carefully. For additional information on individual topics, contact your local dealer or GRAITEC support.

System requirements

To successfully install Advance certain requirements have to be met.

Hardware

- ✓ PC with Pentium Processor min. 2 GHz (4 GHz recommended)
- ✓ Main memory min. 2 GB
- ✓ AutoCAD® compatible graphics-card (for more information, see <http://www.autodesk.com/autocad-graphicscard>)
- ✓ Min. 1 GB free disk space on the hard disk
- ✓ Network adapter
- ✓ DVD Drive

Software

- ✓ Installed operating system Windows 2000, Windows XP Professional or Windows (32-bit or 64-bit version).
- ✓ AutoCAD® 2004, ADT 2004, AutoCAD® 2005, ADT 2005, AutoCAD® 2006, ADT 2006, AutoCAD® 2007, ADT 2007, AutoCAD® 2008 or AutoCAD® Architecture 2008.
- ✓ Installed TCP/IP Protocol

Distribution

- ✓ Advance is delivered on DVD.
- ✓ The license file is delivered by e-mail.

License

An authorization code is required to use Advance commands in AutoCAD®. Without this authorization only the AutoCAD® commands will work. An authorization code is required for each workstation that uses Advance.

Obtaining a user license

The **advance.aslic** license file is provided by GRAITEC based on the specific information of the computer where Advance Steel was installed. The information required to create the **advance.aslic** license file is as follows:

- Name of the computer HOSTNAME
- Adapter Address ETHERNET ADDRESS

To get this information, browse the installation DVD, go to the Tools folder and double click **LMTOOLS.EXE**. Go to the **System Settings** tab and write down the HOSTNAME and the ETHERNET ADDRESS.

Request your license file by email to. If you have any problems contact your reseller or GRAITEC support.

If Advance was purchased request a user's license; otherwise, request an evaluation license. In both cases the **advance.aslic** file will be sent by e-mail.

Using USB dongles

To use the USB dongle, install the hasp driver on the Advance Steel DVD.

Browse the installation DVD, go to the Tools folder and double click **hdd32.exe**.

The license file is provided by GRAITEC based on the dongle ID.

To get this information, browse the installation DVD go to the Tools folder and double click **GetHasp-DongleId.exe**.

Installation

Requirements for a successful installation:

- In your Windows session you must be logged in as administrator or must have administrator rights. If you are not logged in as administrator or you do not have administrator rights, an error message appears during the installation.
- The TCP/IP protocol is usually setup properly by windows. If problems occur, verify the connection to the license management software by using the **ping** command. This command tests the host name and the IP address. If the IP address is confirmed and not the host name, there might be a problem with the name server. If this is the case verify that the queried host name is in the local HOSTS file or in the DNS database. Below is an example of how the ping command is used:

```
C:\>ping kh.meinnetz.net
PING executed for kh.meinnetz.net [ IP 192. ... ] with 32 Bytes of information:
Answer from 192. ... : Bytes=32 time<10ms TTL=128
Answer from 192. ... : Bytes=32 time<10ms TTL=128
Answer from 192. ... : Bytes=32 time<10ms TTL=128
Answer from 192. ... : Bytes=32 time<10ms TTL=128
```

Note: *Several Advance Steel versions can be installed on a computer at the same time.*

Proceed with the installation as follows:

1. Close all active WINDOWS applications.
2. Insert the installation DVD into the DVD drive.

The setup program starts automatically and the following dialog box appears.



Figure 2: Advance installation start-up screen

To install Advance, click **Setup**.

If the Setup did not start automatically, continue as follows:

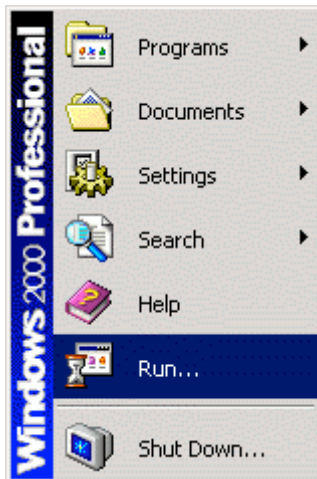


Figure 3: Run option on the Start menu

The AutoPlay tool on the computer could be switched off and thus the installation does not start automatically. Start the Advance setup program using the **Run** command.

To start a program using the **Run** command:

- Click **Start** and then **Run**.
- Enter the path and the name of the program to start in the **Open** field.

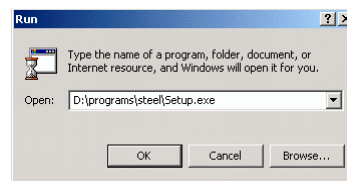


Figure 4: Run dialog box

- In the "Run" dialog box, click **Browse** to select the **Setup.exe** program on the DVD in the folder shown in Figure 4.

Start by clicking **OK**.

The installation starts.

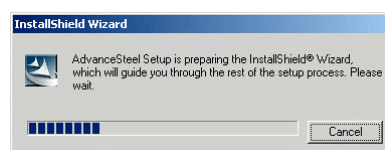


Figure 5: Installation progress

3. Select the preferred language.



Figure 6: Language selection

4. Read the license agreement. Click **I accept** to agree to the specified terms and click **Next** to continue.

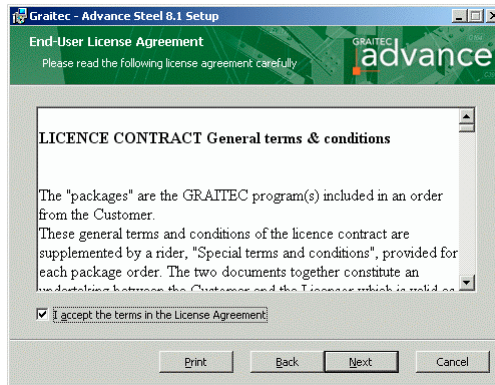


Figure 7: License agreement

5. Select the type of installation and click **Next** to accept.

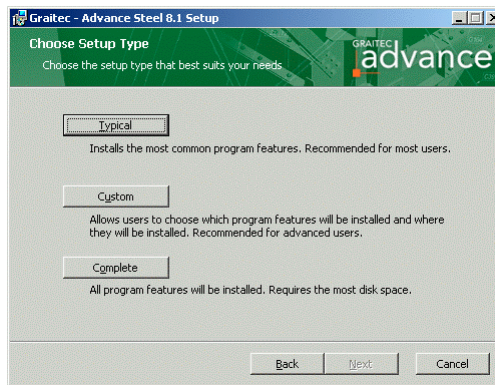


Figure 8: Installation type

- ✓ **Typical:** installs the most common application features. The software is installed on the Windows drive in **C:\Program Files\Graitec\AdvanceSteel**.
- ✓ **Custom:** installs only the selected application features.
- ✓ **Complete:** installs all application features.

6. To have a custom installation, or to install Advance in a different folder, select **Custom**.
The components that can be installed are displayed.

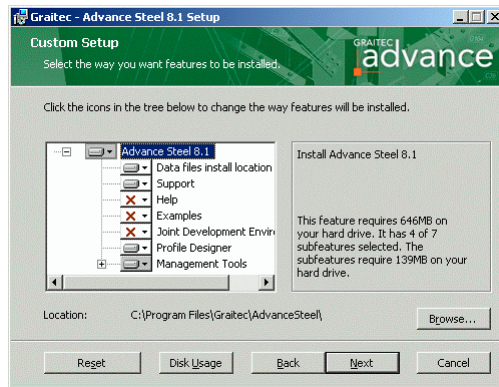


Figure 9: Component selection

*To check the available space on each hard drive, click **Disk Usage**.*

Note: *If you already have installed Advance Steel, select another installation folder.*

To change the destination path, click **Browse**. In the next dialog box, enter a path or select a different folder in which to install Advance and click **OK**, then **Next**.

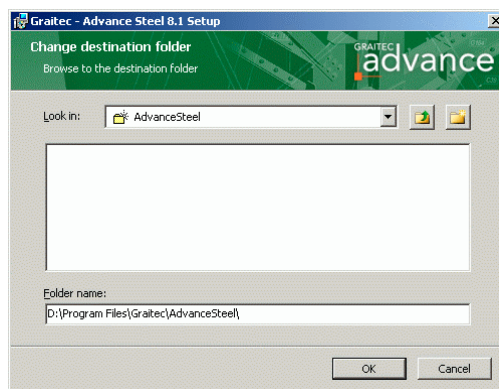


Figure 10: Installation folder selection

7. Click **Install** to start the installation.

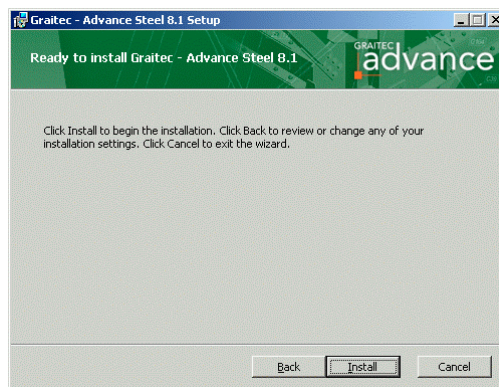


Figure 11: Installation complete

Server and Client configuration

For licensing, specify if the computer is a server or client using a small program called License Server. To install this program, proceed as follows:

1. Browse to the **Tools\License server** folder on the installation DVD.
2. Select the **Setup.exe** file.
3. Select the preferred language.

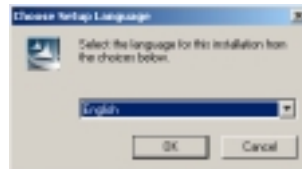


Figure 12: Language selection

4. Read the message and click **Next** to continue.

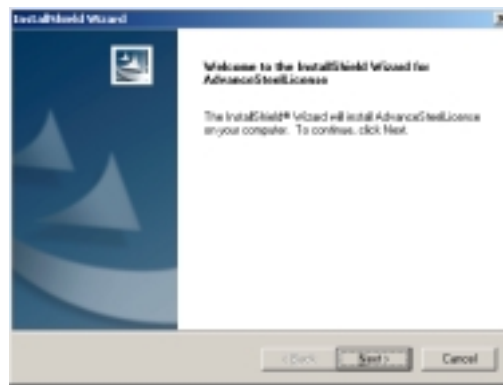


Figure 13: Welcome dialog box

Server

If you have more than one license and the current installation is for the server, install the **Server** License Management only. When installing the product as a **Client** on other workstations you will have to set the name of the server as being the name of the current computer.

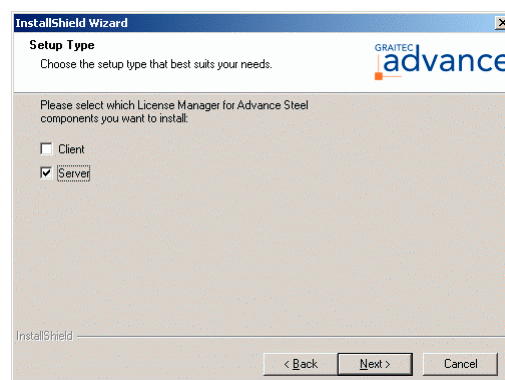


Figure 14: Selection of the Server License Manager

Client

If you have more than one license and the current installation is for a Client (not Server), select only the **Client** option.

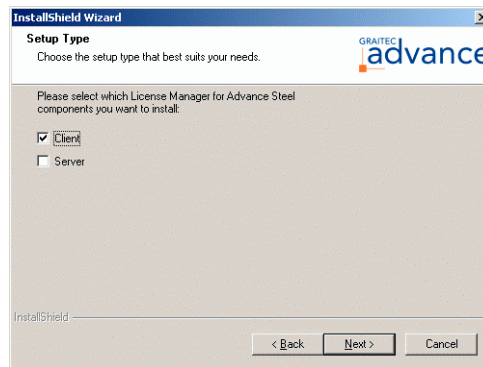


Figure 15: Selection of the Client License Manager

The name of the server is required during the next installation step.

After entering the server name and clicking the **Next** button, the installation program searches for the specified computer. This search might take some time.

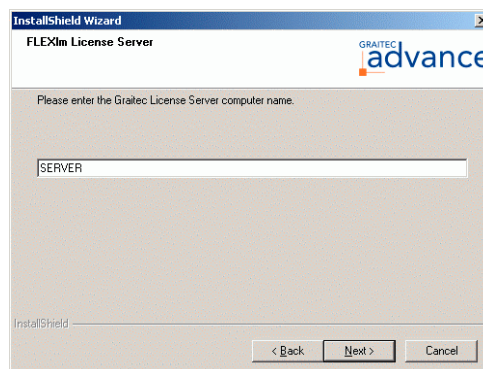


Figure 16: License Server computer name

If the server is not found, the following error message will appear:



Figure 17: Error message

Check the name of the server.

This message will also appear if the server was not found, for example, because it was not turned on. You can then continue the installation.

Server and Client at the same time

If installing multiple licenses on a network and if your workstation is a *Server* and *Client* at the same time, you have to install the License Management for both *Server* and *Client*. In the next step, you will have to provide the name of the machine on which the license manager is installed in the **Server** option.

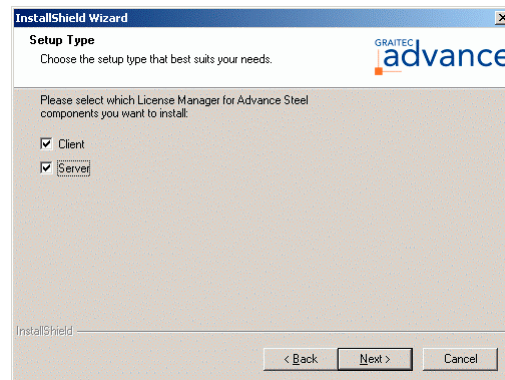


Figure 18: Selection of Client and Server License Managers

Authorizing the software

In Advance vocabulary “*Authorizing the software*” means installing a license to use the software!

A license file (**Advance.aslic**) specific to the computer from which it was requested is sent by e-mail.

To authorize Advance software, follow the procedure described below.

To request a user's license or an evaluation license, start Advance, select the “Request your user license” option and click **Next**.

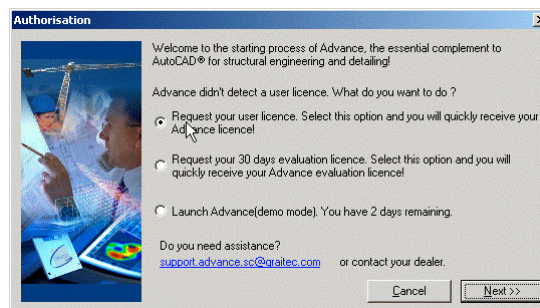


Figure 19: Advance authorization window

An information window appears. Click **Next** after reading the messages.

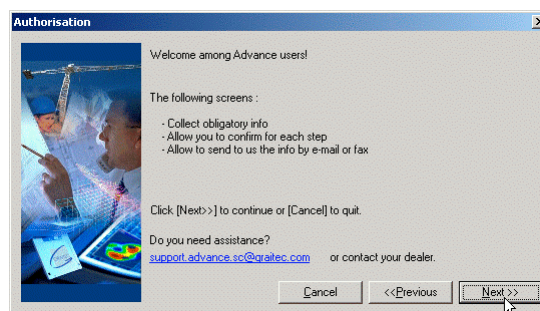


Figure 20: Authorization information

Enter all the required information for the license request and click **Next**.

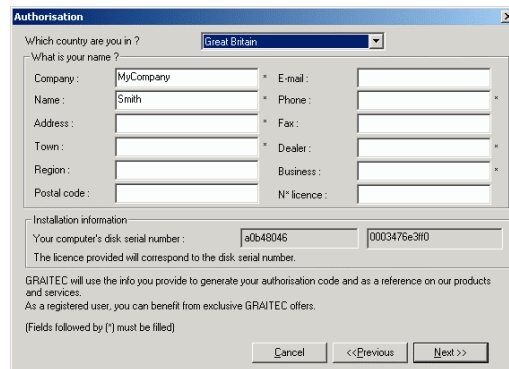


Figure 21: Authorization required information

Advance provides the option to send the information by e-mail or by fax. Select your preference and click **Next**.

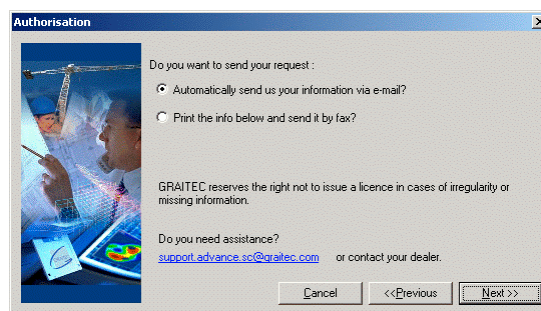


Figure 22: Selection of method for sending the request

A window with a new information message appears. Read the information and then click **Finish**.

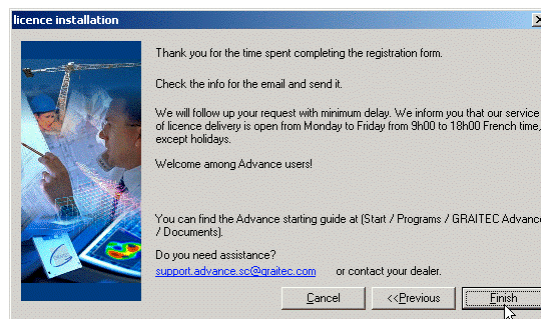


Figure 23: Authorization information

If the method for sending the license request was by e-mail, an e-mail message is automatically prepared. Send the request to authorization.advanced@graitec.com

Note: For a network license specify the number of users for this license.

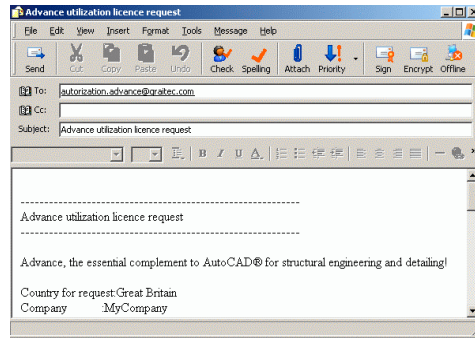


Figure 24: Email for license request

If the method for sending the license request was by fax, a fax is automatically prepared with the formatted text. Print the fax and send it to (450) 628-0400.

Within 24 hours an e-mail containing the required **advance.aslic** file (for network or standalone license) will be sent to you.

Save the file on the hard drive and double click it to authorize the software.

For more information refer to the **Readme.pdf** file.

License management

The **LMTools** application is installed with the license server software. This application verifies whether the license server is active.

1. On the Start menu, select the **License Server** icon in the **Graitec** group.
2. In the "LMTools" dialog box, enter the complete path to the license file using **Browse** on the **Service/License File** tab. This file is in the directory where the license server was installed.

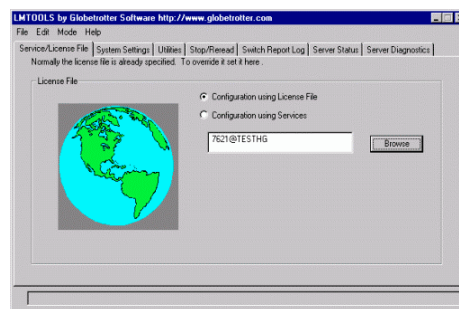


Figure 25: LMTools dialog box – Service/License tab

3. Select the **Server Diagnostics** tab and click **Perform Diagnostics**.
4. Close the application.

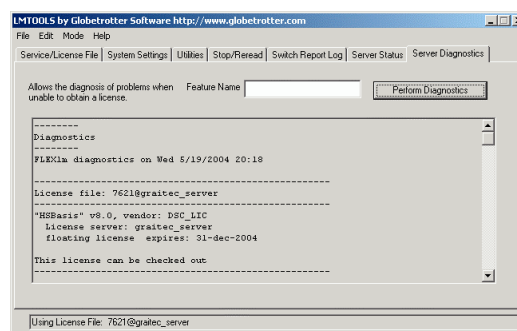


Figure 26: LMTools dialog box – Server Diagnostics tab

Converting databases (Merging)

Old databases can be converted to new databases using the Management Tool. This might be required on a new revision of Advance Steel or if users want to exchange information.

Note: *Only the database from a previous version can be converted.*

A column with author information is a prerequisite for converting databases.

Note: *The tables (libraries) cannot be converted if the structure of the tables (libraries) was changed.*



Save the databases before converting them.

Start the **Management Tool**.

Click "Convert database" . The "Convert database" window opens.

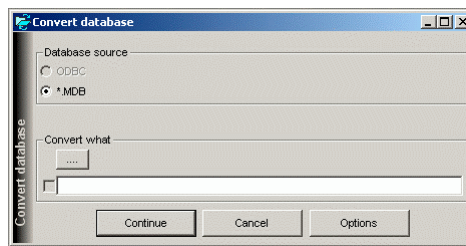


Figure 27: Convert database window – MDB database source

Select the database (*.mdb) by entering the filename in the "Merge what" field. For example, search for the database AstorBase.mdb.

In the "Convert database" window, click **Options**. In the "Options" dialog box select the preferred settings.

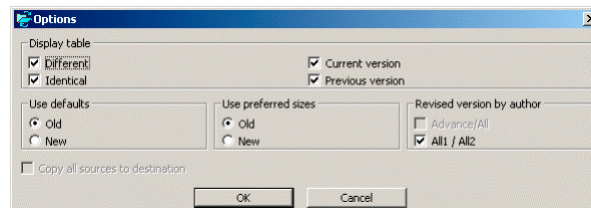


Figure 28: Options dialog box

Display tables (libraries)

This option defines what is displayed when the tables (libraries) have been analyzed (i.e., different tables (libraries), identical tables (libraries), in the current release and/or the previous release (to be converted)).

Use defaults

This option determines whether the defaults of the previous release (to be converted) or those of the new release are used.

Use preferred sizes (for objects in library)

This option determines whether the preferred sizes of the previous release (to be converted) or those of the new release are used.

Click **Continue** after selecting the database. The tables (libraries) in the databases are compared. The “Comparison result” window then appears.

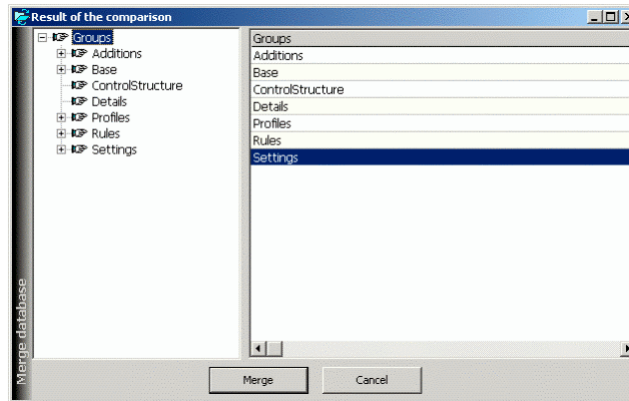


Figure 29: Comparison result window

The new Advance Steel databases are listed under “Groups” tree. For example, open the list under “Base” and a window (Figure 30) with the following content appears.

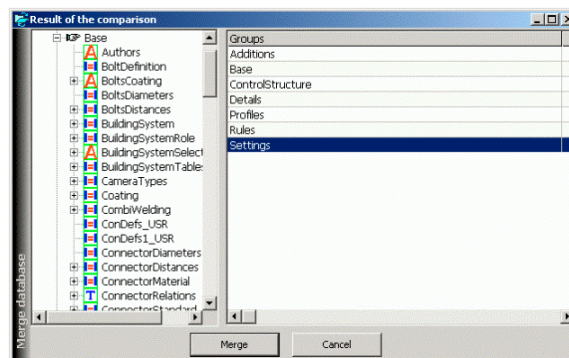


Figure 30: Comparison result window – List under "Base"

The icons in front of the table names indicate the comparison result:

-  Table is identical
-  Table can be automatically converted
-  Table cannot be converted because the table structure has been modified.
-  Tables (libraries) available only in current release.

Only tables (libraries) with status set to “Automatic” can be converted.

Tables (libraries) with a specific status are selected by a right click in either side of the window. From the context menu, select or unselect criteria (i.e., different, identical or in current / previous version).

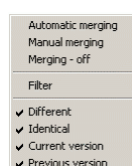


Figure 31: Context menu

If **Filter** is selected, another window will appear where tables (libraries) to be displayed are selected.

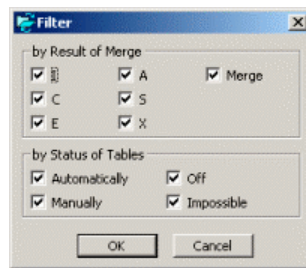


Figure 32: Filter dialog box

The code letters in the right window or in the “Filter” dialog box are defined as follows:

- I Table is in the source database but not in the destination database
- C Source database contains columns that are not available in the destination database
- E The source and destination databases have different columns with identical values in the fields
- A The entries in the source and destination database are identical but have different authors
- S Table contains fields with links
- X Tables (libraries) have different structures

If all the tables (libraries) that can be automatically converted are not to be converted in the new database, unselect conversion (**Converting – Off**).

To **convert** only some information records from a selected table, click the manual conversion option in the right panel of the window.

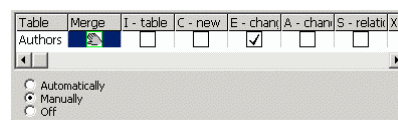


Figure 33: Conversion options

The status is changed from “Automatic” to:



Converting – Off



Converting Manually

After selecting the conversion method for all convertible tables (libraries), click **Convert** and the process will begin. Every element to be converted manually will require validation.

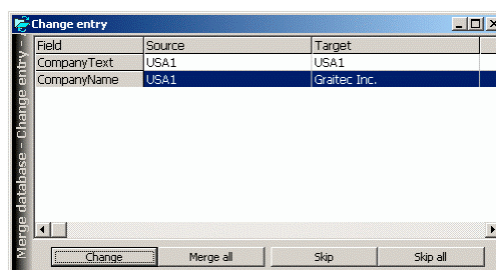


Figure 34: Change entry

When the conversion is done all windows of the **Convert database** tool must be closed.

Support

GRAITEC has hotline and e-mail support for Advance.

Support by telephone

Ask your dealer or closest GRAITEC office for the appropriate telephone number.

Chapter 2

The 3D Model

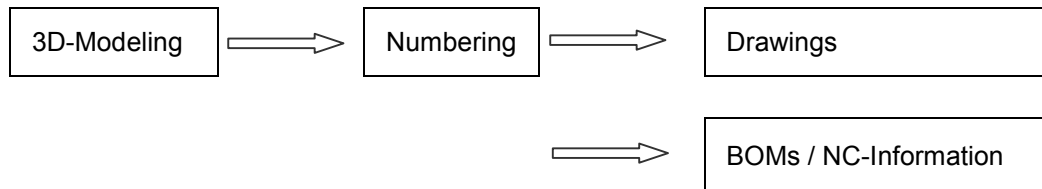
In this chapter:

- *Elements of the 3D model*
- *3D Modeling workflow*



Elements of the 3D model

The Advance 3D model is built from elements such as beams, plates, structural elements, bolts, welds, features, and joints. Once the model is finished, checked and numbered, all output like structured BOM, NC data, general arrangement and detail drawings can be completed.



Beam & Plate

The 3D model is built mainly from Advance basic objects:

- Beams created as section classes, simple sections, compound sections, or curved sections
- Plates as rectangular plates or as polygon plates with any contour

Beams and plates are created directly in the model and are displayed, by default, in the 'wireframe' mode.

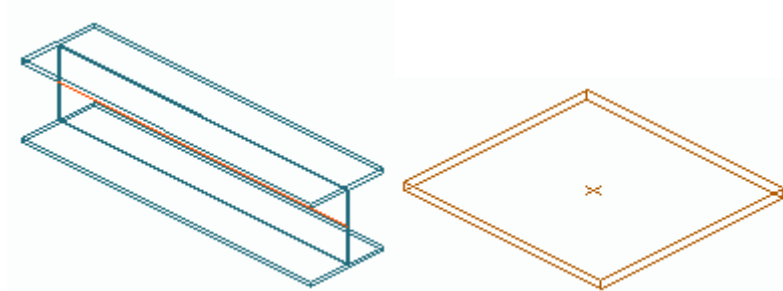


Figure 35: Advance Beam and Plate (standard representation)

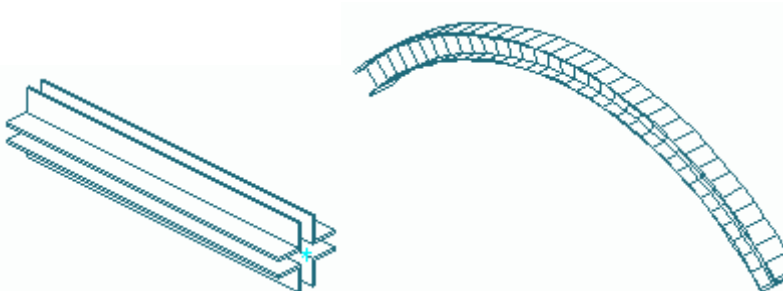


Figure 36: Compound section and curved beam shown in the AutoCAD® 'Hidden' representation

Processing

The basic objects (i.e., beams and plates) have available processing features. The most important processing types are:

- Beam processing: trimming, coping, miter cuts, rectangular and circular contour cuts, or any type of contour.
- Plate processing: corner finishes, chamfers, outer plate contours and inner contours, etc.

Processing of existing basic objects (e.g., beam trimming and coping) is displayed as green processing contours within the object. The processing objects cannot exist alone and are part of a basic element (i.e., beam or plate). The object processing are edited as individual objects.

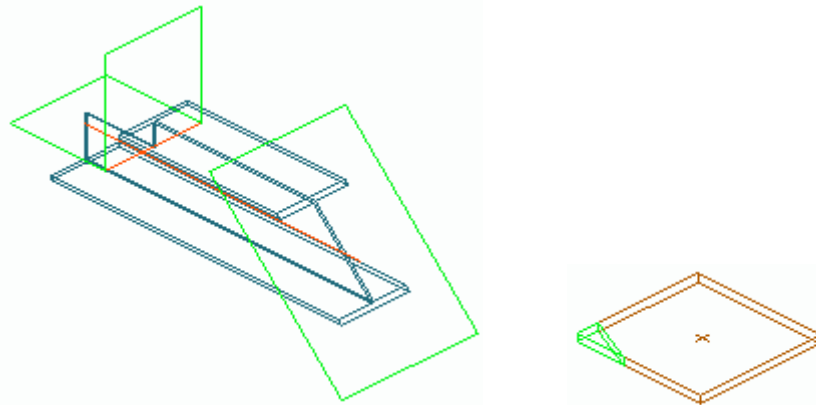


Figure 37: Contour processing – cope and corner section

The variety of processing options in Advance allows for almost any beam and plate contour.

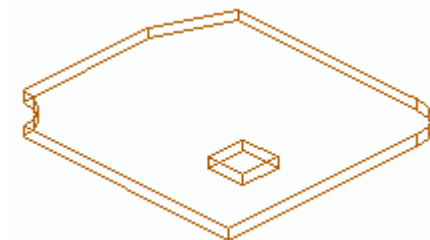


Figure 38: Plate processing

If a basic element is deleted all processing objects will also be deleted.

Bolt patterns & welds

The basic objects (i.e., beams and plates) can be connected with:

- Bolt patterns (or holes only)
- Welds

These objects establish a connection between objects (e.g., beams and plates). This information is stored on the objects (i.e., beam or plate) including any bolt pattern (with its definition) or welds (with its relevant properties). Any individual element in the connection “knows” what holes, bolts, or welds it contains or with which element it is connected.

A bolt pattern can describe one or several bolts, which are automatically created in any plane together with the appropriate holes.

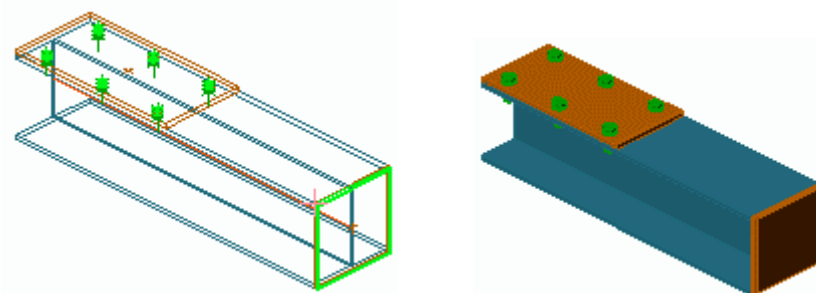


Figure 39: Bolt Pattern, Weld Point

Changes in the bolt pattern automatically update the holes.

The tools for creating bolt patterns are used for bolts in addition to:

- Holes, slotted holes, countersunk holes, blind holes, threaded holes and punch marks
- Shear studs

The above are all created with their respective properties or definitions.

It is also possible to create the various hole types as part of a **bolt** object and a separate **hole** object.

Weld points are displayed as crosses in the model.

Joints

Another option for connecting the basic elements is the use of Advance joints. Joints are complex elements that consist of basic elements and dependent elements that are controlled by a construction rule.

All individual elements in the joint, including their properties and processing objects, are held together and represented as a gray box (connection object).

All connection objects and definitions are included in the gray box.

Example: Haunch connection

A portal column and a rafter are connected with an intelligent rule-based joint. For example, this is a haunch connection, including plates, stiffeners, and bolts.

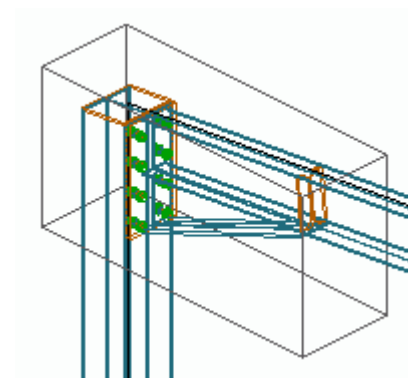


Figure 40: Connection object (gray box)

The following joints are included in Advance:

- Connections for North America: end plates, clip angle connections, base plate, flat bracings, and HSS bracings
- Frame corners
- Gable wall connections and pin-ended column connections
- Ridge connections and splices
- Platform connections, end plates, web connections, and shear plates
- Gusset plates and diagonal bracing (HSS and Flat)
- Base plates and stiffeners
- Tube connections
- Turnbuckle bracing
- Pylon construction connections

And on the alternative **UK Joints** toolbar, the following additional flyouts are available:

- UK Hot Rolled Joints
- UK Cold Rolled Joints

The rules and structure of joints are stored in MS-Access tables (libraries). Adjustment of these rules to user requirements (or creation of new rules) is possible with knowledge of Advance's macro programming language.

In addition to the joints provided in the standard Advance package, interactive (also called **manual**) joints can be created, stored and reused.

Structural elements

Structural elements are an intelligent group of basic elements and dependent elements classified as follows: symmetrical **goal post** frames and portal frames, single-span bracings, purlin positions, various stairs, pylons, joists, and half trusses.

These elements are created as a group of several basic elements with relationships to each other. Elements and their relationships are held together and stored with a structural frame. The structural frame is displayed in the model as a white continuous line (Figure 41).

Example: Portal frame

A portal frame consists of four sections: two grouped columns and two grouped rafters connected using a structural element macro.

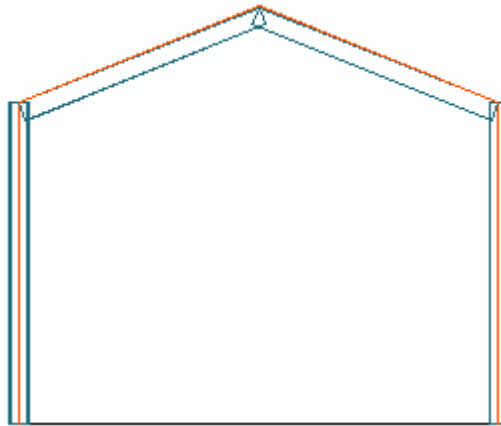


Figure 41: Portal Frame

All changes to one element affect the entire group. If a rafter section is changed then the second rafter section will also be changed. The column sections of the frame behave in the same way. A change in the total height or the column height affects the entire structural element.

Auxiliary objects

Auxiliary objects are:

- Grid axes or
- Level symbols

These objects do not belong directly to the frame but support the construction process. Nevertheless, they are important elements.

 Example: Building grid

A building grid that corresponds to the dimensions of the construction forms the basis of 3D Modeling and helps with orientation in three-dimensional space.

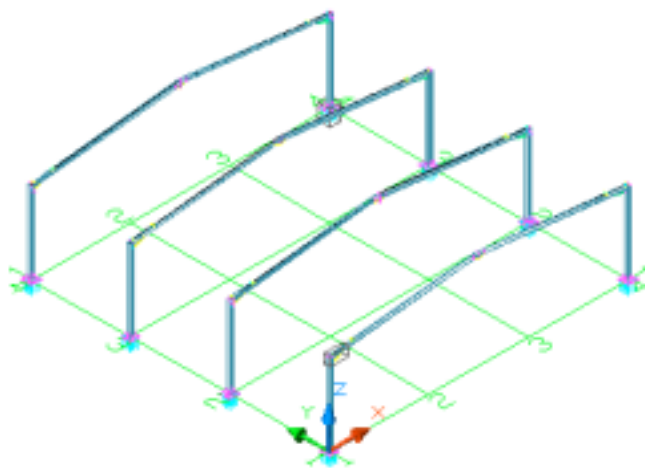


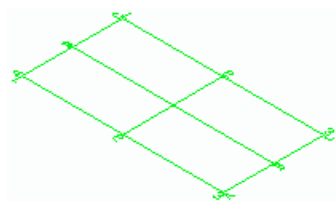
Figure 42: Building grid with portal frame

Special parts

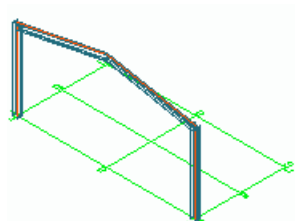
Objects that are not Advance standard objects can be created as special parts. When Advance creates drawings and bills of material with special parts they are handled like standard objects. If these objects (special parts) are to appear in the bill of material, they must be provided with Advance properties. Advance information that can be attached is:

- Weight
- Material
- Coating
- Name
- Commodity
- Lot/Phase
- Part mark (single part and assembly)
- Model Role
- Other

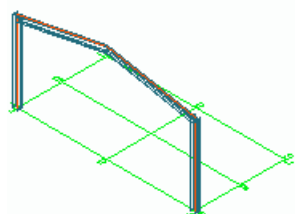
3D Modeling workflow



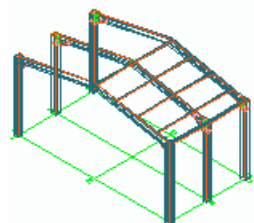
Create a building grid



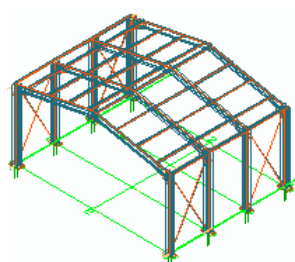
Create beams, plates, structural elements



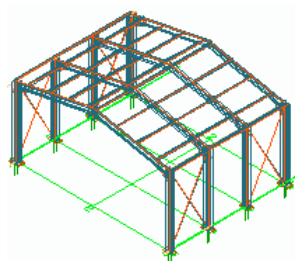
Edit beams / plates (processes)



Add Advance elements with joints / interactive connections



Add other interconnecting members



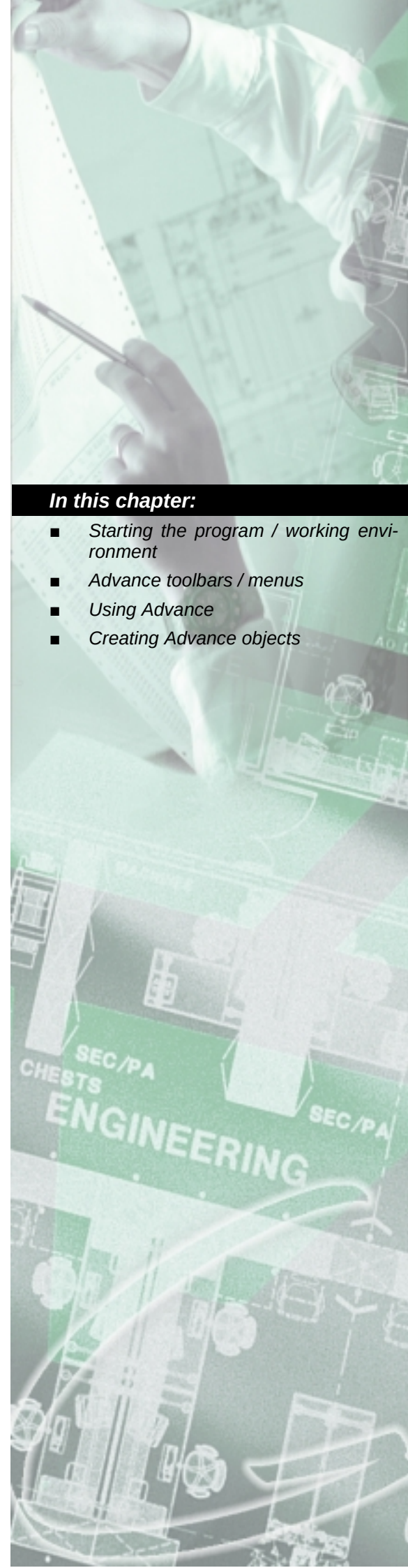
Controls / checks for collisions and measurements

Chapter 3

Advance interface

In this chapter:

- *Starting the program / working environment*
- *Advance toolbars / menus*
- *Using Advance*
- *Creating Advance objects*



Starting the program / working environment

Start AutoCAD® / Advance by double clicking on the **Advance Steel Icon** on the desktop.



Figure 43: Advance Steel icon on the desktop

AutoCAD® and Advance start and the user interface appears.

The **ASTemplate.dwt** template drawing is automatically loaded when starting AutoCAD® / Advance. It contains important defaults like the current coordinate system, the orientation, object snap settings, layer assignment and color definitions so that modeling can immediately begin.

Advance starts in a three-dimensional isometric view. The User Coordinate System (UCS) is active and set to the World Coordinate System (WCS). All coordinate entries refer to the UCS.

- Save a project with an appropriate name in an AutoCAD® **DWG** format. Use **File** in the upper left corner of the screen. Each project should be saved in its own folder for easier file management.

Among other features, template drawings and files for certain drawing formats are delivered with Advance.

Advance toolbars / menus

All Advance tools are grouped by type and are available on toolbars positioned in a space saving manner in the AutoCAD® environment.

To keep the interface clear and make each command easy to find, only two Advance toolbars, **Standard** and **Advance Steel**, appear when AutoCAD® / Advance is started.

The toolbars contain commands and sub-toolbars in the form of flyouts. Flyouts open by clicking them and contain other Advance command buttons.

The **Advance Steel** toolbar icons call the Advance main toolbars that contain logically grouped Advance tools, objects, and elements. When accessed these main toolbars replace each other (with exception of the **Design assistance** toolbar) so that the model workspace remains uncluttered with plenty of space.



Figure 44: **Advance Steel** – toolbar

If, for example, the Advance **Create objects** toolbar (first button from the left) is opened through the **Advance Steel** toolbar, it remains on the display until another main toolbar (e.g., **Joints** toolbar, second button from the left) is opened. The newly opened toolbar (e.g., **Joints** toolbar) replaces the **Create objects** toolbar and remains open until another toolbar is called.

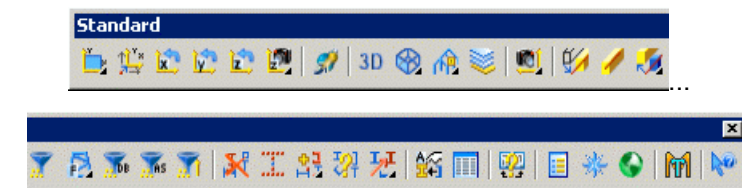


Figure 45: Advance **Standard** – toolbar

The **Standard** toolbar includes tools for building, displaying, changing, and marking Advance elements, the Advance Document Manager, drawing layout, and other help tools. It also contains frequently used AutoCAD® tools for use with Advance.

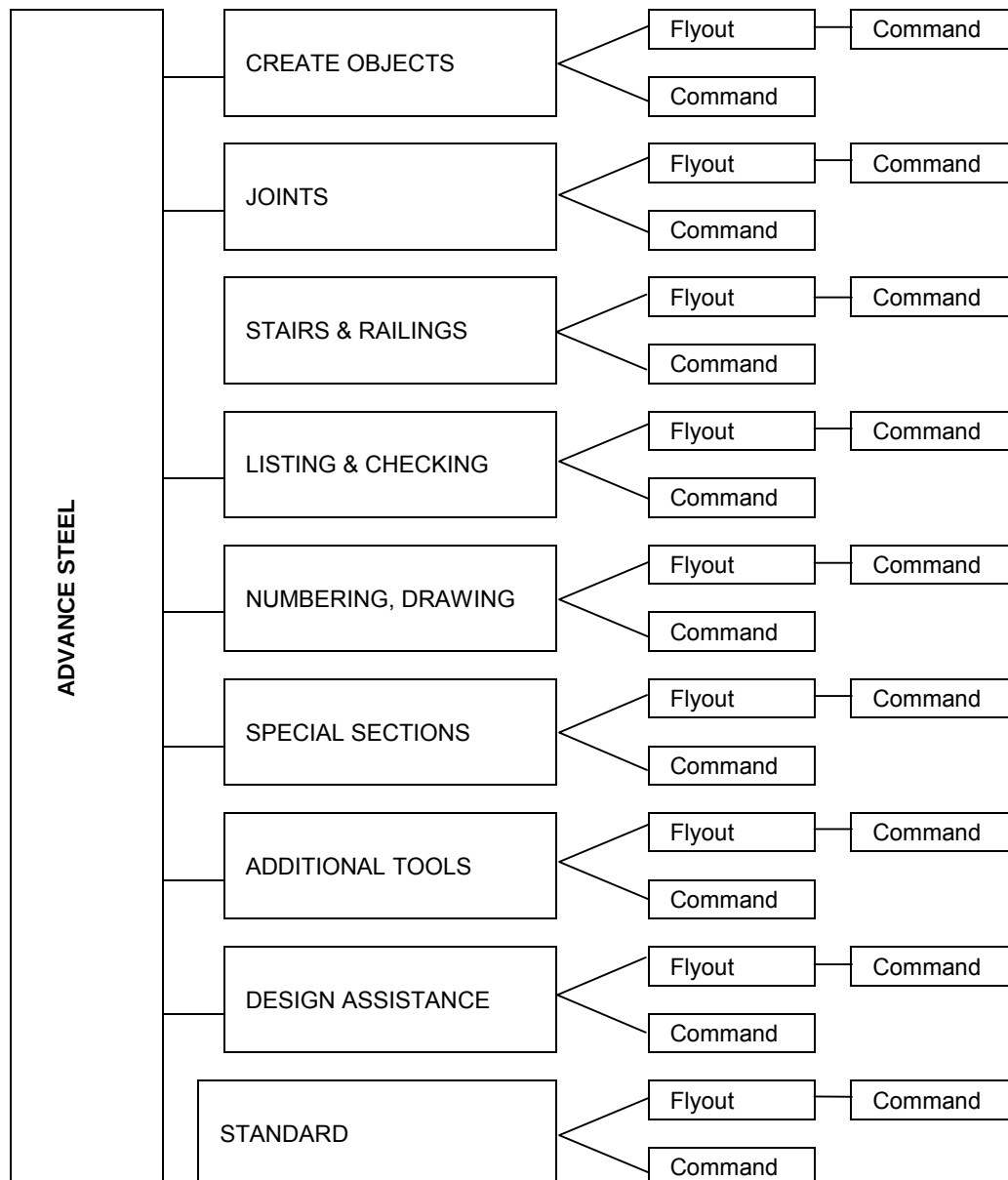


Figure 46: Arrangement of the Advance toolbars



Figure 47: Advance **Create objects** toolbar

The **Create objects** toolbar contains commands for grids, structural elements, beams, beam processing, plates, plate processing, construction systems, bolts, holes, shear studs, welds, and for the display and modification of connections.

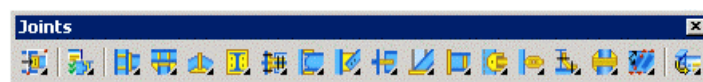


Figure 48: Advance **Joints** toolbar

The **Joints** toolbar contains all joints, arranged by individual element type, and the corresponding commands for handling these joints and connection objects.

The toolbars reviewed up until now are essential for creating a three-dimensional model. The **Design assistance** toolbar can also be used.

Figure 49: Advance **Stairs & Railings** toolbar

The **Stairs & Railings** toolbar contains commands for creating stairs, railings, ladders, and specific joints (i.e., stair footing, end plate with cope, shear plate, platform splice, angle connection, railing anchorage, and bolted end plate).

Figure 50: **Design assistance** toolbar

The **Design assistance** toolbar contains important checking tools (i.e., collisions in the model and checking databases), which are required during construction and are also available on other toolbars. Special Advance commands to copy, rotate and mirror connections are included. This toolbar accelerates the construction work by having frequently used commands in one place.

Figure 51: Advance **Listing & checking** toolbar

The **Listing & checking** toolbar includes commands for structured BOM, collision test, save with proxy-graphics, Advance **Audit** checking, steel construction technical checking, special parts, and update defaults. This toolbar does not have commands for the construction environment. It is used for managing / checking the model and for communication with other programs.

Figure 52: Advance **Numbering, drawing** toolbar

The **Numbering, drawing** toolbar contains all commands pertaining to numbering, drawing creation, automatic drawing creation with processes, and the Document Manager. The predefined drawing styles are different for each installation.

Figure 53: Advance **User sections** toolbar

The **User sections** toolbar includes all required commands for creating custom sections (can be added directly in tables (libraries)).

Figure 54: Advance **Bonus** toolbar

The **Bonus** toolbar contains applications being developed for an upcoming release and is not yet documented.

Using Advance

The main toolbar contains **buttons** that are tools or call flyout toolbars. The **Flyout Toolbar** buttons have a black triangle in the lower right corner. These flyouts automatically open when the button is pressed and held down.

Example: Opening grid commands

- Call the Advance **Create objects** toolbar on the **Advance Steel** toolbar.

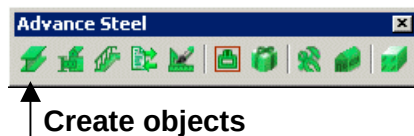


Figure 55: Create objects icon on the Advance Steel toolbar

- Press and hold down the first button of the Advance **Create objects** toolbar.
- The **Sub – Axis grid** flyout appears.

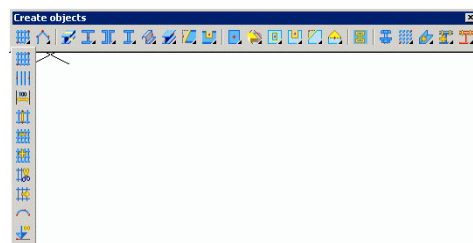


Figure 56: Accessing the **Sub – Axis grid** flyout

Instead of individually calling flyouts (that close on their own) several toolbars can be called using **View** on the AutoCAD® menu bar. If more tools from the sub-toolbar are required for the construction of a grid, select other toolbars from the **Toolbars** tab (e.g., select the AstBasic menu group and then **Sub – Axis grid**).



Figure 57: **Sub-Axis grid** toolbar

Start an Advance command by clicking a toolbar button. The command appears on the AutoCAD® command line at the bottom of the screen.

Other important tools for using Advance

- Abort commands by pressing the **Esc** key.
- The current command and prompts are displayed on the command line window at the bottom of the screen. Open and close the command line window with the **F2** key.
- The **right mouse click** behaves like the **Enter** key (depending on the AutoCAD® system settings).
- Hovering the cursor over a toolbar button displays the button **tooltip**.



Figure 58: Undo command on the AutoCAD® **Standard** toolbar

- The “Undo” command on the AutoCAD® **Standard** toolbar cancels one or several executed commands.

- The “Match properties” command on the AutoCAD® **Standard** toolbar copies properties from one object to another. The properties to transfer are selected from a list.

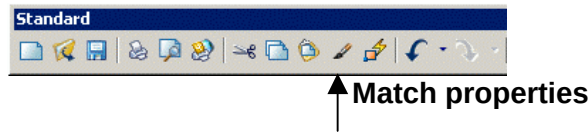


Figure 59: Match properties command on the AutoCAD® Standard toolbar

Creating Advance objects

Advance objects are created in 3D-space using the program tools. The object's orientation depends on the current UCS (User Coordinate System).

Points created by selecting or using coordinates determine the position and orientation of the object in space. A **dialog box** will appear in which different settings (e.g., geometric sizes, etc.) and drawing styles can be changed (e.g., dimension/label on the drawings).

The settings in the dialog box are sorted on different **tabs** that vary based on object type.

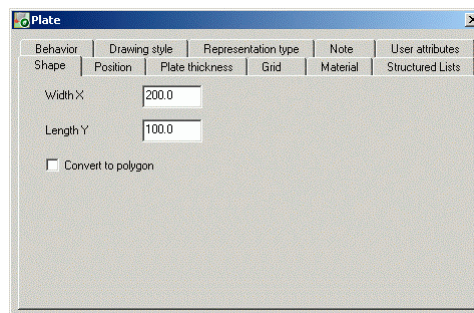


Figure 60: **Plate** dialog box

When a dialog box field (e.g., dimension, position, etc.) is changed the model updates instantly (model preview):

- By clicking the next field,
- By closing the window (click the X in the right upper corner),
- By using the tab key to move to the next field or
- By using the enter key to select the value in the current field.

Initial settings of an object (e.g., a plate) are stored and can always be recalled and updated using the same dialog box.

Advance will recall the last dialog box values and preferences until the next time the same tool is used. After restarting AutoCAD® / Advance, the dialog boxes return to the default settings.

The default settings are restored using the system menu of the dialog box (right click the header line of the dialog box). By clicking **Use defaults** on this system menu, the dialog box entries return to the default settings.

Object properties

All created objects have default **Properties** that can be changed in the Advance **Properties** dialog box, which automatically appears upon creating an object.

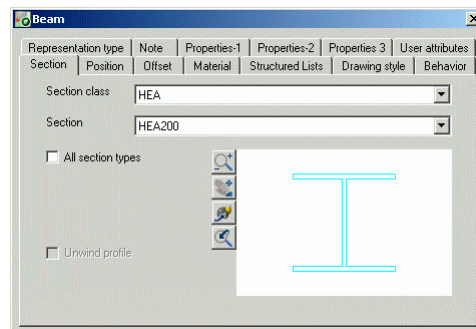


Figure 61: “Beam” – properties dialog box

The object properties are classified as follows:

- **Geometric Properties** (i.e., position in the model and shape)
They are set on creation and can be changed afterwards using standard AutoCAD® commands (e.g., move, rotate, copy, etc.) and grips.
- **AutoCAD® specific properties**
They are set in the AutoCAD® property dialog box. These properties are changed using the AutoCAD® **Properties** command.
- **Technical properties**
They are properties for the on-screen representation (e.g., representation of a beam with or without processing) and non-graphic properties (e.g., material, name, etc.). This information goes in the structured BOMs and drawings.

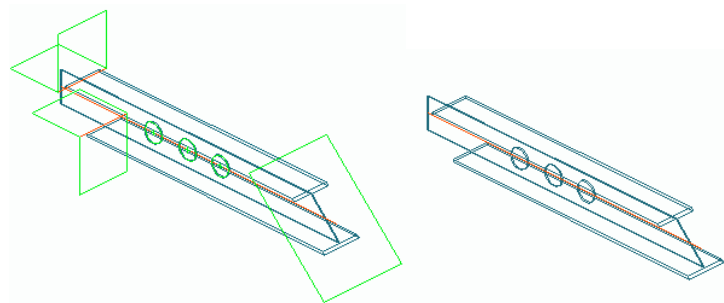


Figure 62: Beam “features” and “standard” representation

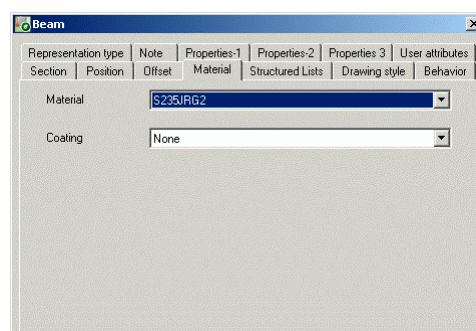


Figure 63: Non-graphic properties for the structured BOM

To open an object dialog box double click on the object.

Another way to access the properties is to select the object(s) and with a right mouse click select **Advance properties** from the context menu (depending on the AutoCAD® system settings).

Layer

Advance objects are, by default, created on the active Layer but can also be placed automatically on specific layers with different properties. This feature is set in the **ASTemplate.dwt** DWG drawing template file using the Advance **Management Tool**.

Advance objects are displayed, by default, in the following colors:

Object	Color
Building grid	Green
Beam	Yellow
Plates	Magenta (pink)
Bolts	Cyan (turquoise)

Chapter 4

Creating a 3D Model

In this chapter:

- *Creating a building grid*
- *Creating a beam*
- *Creating plates*
- *Coordinate systems*
- *Beam processing*
- *Plate processing*
- *Changing Advance objects*
- *Advance command properties*
- *Representation Type – Snaps – Grips*
- *Bolt and Hole Patterns / Shear Studs*
- *Welds*
- *Connections*
- *Working methods I*
- *Structural elements*
- *Joints and connection objects*
- *Special parts, special sections*
- *Working methods II*



Creating a building grid

The Advance auxiliary object “building grid” makes it easy to place Advance objects within a construction design.

A building grid consists of axes in the X- and Y- directions. Grids facilitate placing of the construction elements and for orientation in the 3D view. Placing a building grid is the first step of 3D modeling in Advance.

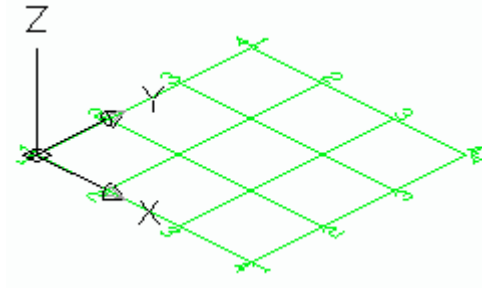


Figure 64: Advance building grid

A building grid is created in the X/Y-plane of the current coordinate system and consists of two independent axis groups.

Picking three points creates a curved single axis.

A building grid can be copied, for example upwards, several times if needed. For better recognition, the grids can have different labels (A, A', A'') on the various planes with a different color on a new AutoCAD® layer (recommended).

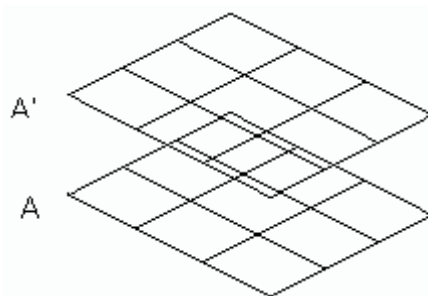


Figure 65: Building grid

 **Example:** Creating an axis group by setting the distance between axis

- Start AutoCAD® / Advance with the ASTemplate.dwt template file.
- On the **Sub – Axis grid** toolbar (flyout), click .

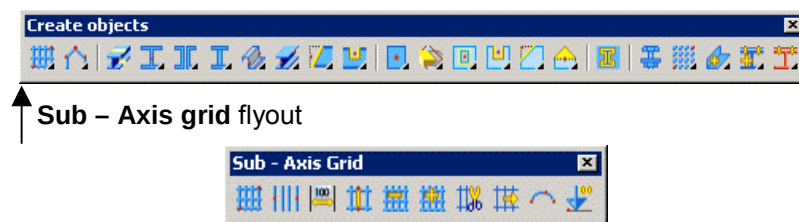


Figure 66: **Sub – Axis grid** toolbar

- Enter the starting point of the first grid line as (0,0,0) (please give end points of the grid line. starting point: **0,0,0 Enter**),
- Set the end point of the grid line by dragging the mouse pointer (with **Ortho**-mode on, F8) in the X-direction and enter 8000 (end point: **8000 Enter**).

- Determine the direction of the axis group by picking a point in the Y-direction on the screen (*orientation of the group: Enter*).
- Next, enter 2000 for each distance between axes until the AutoCAD® command line has a total value of 8000 and then press **Enter** twice. The axes group in the X-direction is created and the **Axes, parallel** dialog box appears.

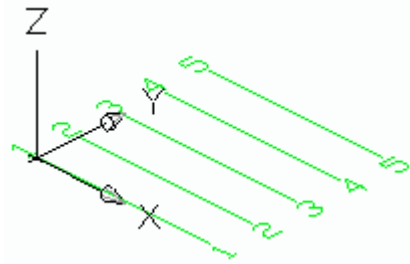


Figure 67: Building grid: axes in X-direction

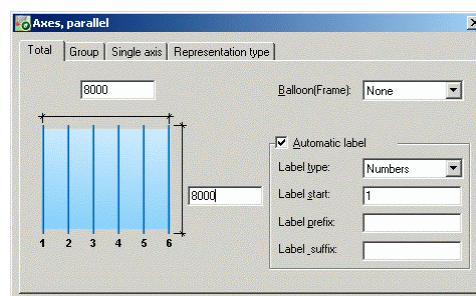


Figure 68: **Axes, parallel** dialog box

- On the **Total** tab of the dialog box, the length property (distance between the first and the last axis) and width (length of the axes) can be changed. The axes are automatically labeled with numbers or letters.
- Close the dialog box by clicking on the cross (X) in the upper right corner.
- To create a complete building grid, repeat the same steps for an axis group in the Y-direction.

To change the axis group, select a group and select **Advance Properties** from the context menu.

When the **Group** tab is selected, the color of the selected group is changed to red. The number of single axes or their distance within the total length is indicated.

The **Single axis** tab provides options for individual axis labeling (if automatic labeling is turned off on the **Total** tab). The selected axis is displayed in red. An axis can be added to the right or left of a main axis with an option to use a label containing the main axis name with a suffix and a prefix.

The grid axes can be hidden or switched to single axis representation on the **Representation type** tab.

- To create a complete grid with a group of axes in both the X- and Y-directions, use the "Building grid" tool.

This building grid is created as a **standard grid** by clicking  and right clicking twice or by giving specific coordinates for the origin point and a second diagonally opposite point.

In both cases, the grid size, division and labeling can be modified using **Advance Properties** (double click the grid). Since two axis grids are created in the course of this command a dialog box does not appear.

The **Sub – Axis grid** toolbar contains the following tools for creating and designing building grids:

Button	Function
	Create grids by individual groups with 4 axes
	Create grid groups with distances

Button	Function
	Create an axis in a group
	Create an axis from a group
	Create a group of axes
	Cut a group
	Extend a group
	Create a curved single axis
	Create a level symbol

Detailed information on individual tools, representation types, snaps, grips, and object coordinate systems is available in the *Axis grid* chapter of the *Advance online Help*.

Creating a beam

In Advance, a variety of beams are preset such as rolled I sections, channels, angles, T-sections, tubes, Zeds, flats, round bar, square bar, square/rectangular hollow sections, cold rolled sections, purlin sections, and other section classes (**Sub – Section classes** toolbar). All types of beams are available from the listed sections. **Compound** and **welded sections** can be also created. There are various types of compound and welded members available on two flyouts: **Sub – Welded sections** and **Sub – Compound sections**.

The beams can be created as **curved** or **poly beams**.

Additionally, individual sections can be set as special sections (refer to the *Special parts, special sections* chapter).

For beam creation, five flyouts are available on the **Create objects** toolbar:



Figure 69: Buttons for creating beams

- Creation of beams – section classes (flyout)
- Creation/ explode of compound sections (flyout)
- Create sections from various cold rolled systems (e.g., Canam Manac, Canfer, etc.)
- Creation of beams (flyout)
- Creation of welded beams (flyout)

The buttons contain flyouts, which are explained in the *Straight beams*, *Compound sections* and *Construction systems* sections of this chapter.

Straight beams

Straight beams are created in the Advance 3D model relative to the current user coordinate system (UCS) by entering one starting point and one end point.

The current user coordinate system (UCS) determines the position of the sections' main axes: the web of a beam runs in the Z-direction of the UCS (i.e., the 'top' of the section is in the Z-direction).

 **Example:** Creating a straight beam HEA 400 x 4000 mm long

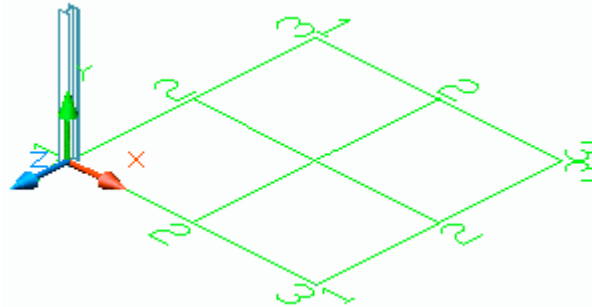


Figure 70: Beam HEA 400

- Click a suitable UCS (see Figure 70).



Figure 71: Advance **Sub – Section Classes** toolbar (flyout)

- On the **Create objects** toolbar, **Sub – Section Classes** flyout, click .
- Select a starting point at (0,0,0).
- Move the mouse pointer upwards in the Y-direction (the setting **Ortho** causes an exact orientation entry) and enter 4000. A standard I-section is created.
- The "Beam" dialog box appears. Select the section class (HEA) followed by the section (HEA 400).

For beam length and placement, three different object axes types are available:

- The **insertion axis** (system line) is an axis that has two points and forms the relation of a beam.
- Reference axes** are important specific axes for sections such as edge lines or center lines.
- The **gravity axis** is the line of the cross-section areas of the center of gravity. Not all sections (e.g., C-shaped profiles) have these lines.

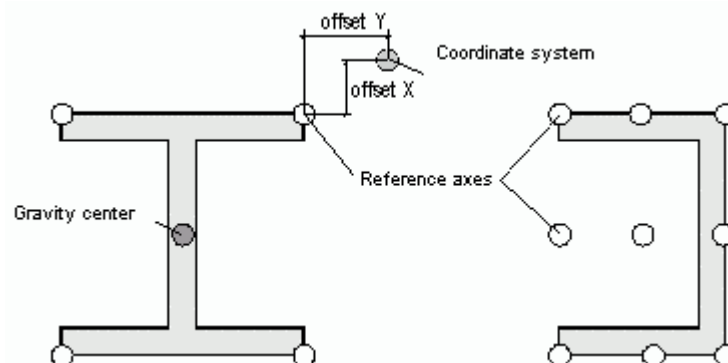


Figure 72: Axes of a beam

The standard insertion axis (system line) of a beam lies in the center of gravity. The beam is moved or rotated relative to the insertion axis or reference axis by changing the values on the **Position** and **Offset** tabs of the "Beam" dialog box.

A variety of other properties, such as entries for **drawings style** (type of dimensions and labels used in the automatic detailing by processes), material, or for **behavior**, are used for identification of identical parts, collision test, and creation of BOMs.

During drawing creation the **Model role**, set on the **Note** tab, will influence the output. An object, set as a column, is dimensioned and labeled differently by the automatic detailing process than if the object is set as a beam.

The **Properties-1** tab displays beam information such as weight and coating surface area, whereas the **Properties-2** tab displays section values such as flange thickness, web thickness and height, etc.

Compound sections

Two or four sections, as a combined section (named **Compound sections**), are created at a system line. The sections are handled as a single section so that a trim will affect all sections.

To create **Compound sections**, use the **Sub – Compound sections** flyout of the **Create objects** toolbar.



Figure 73: **Sub – Compound sections** toolbar (flyout)

Compound sections are created like simple sections by entering a start point and an end point according to the current coordinate system.

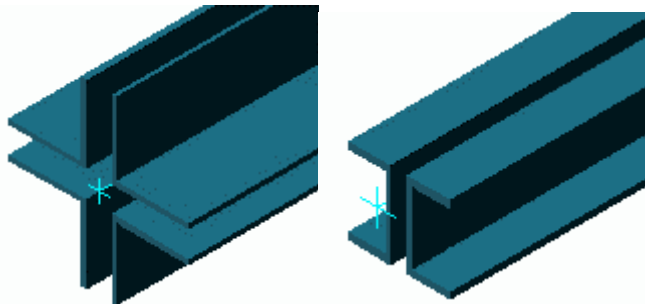


Figure 74: Compound sections

The **properties** in the dialog box such as position, material, drawing-style, or representation type correspond to those of the simple sections.

On the **Shape** tab, the distances between the single profiles in the X- and Y-directions can be changed.

The last button of the **Sub – Compound sections** toolbar separates the assembly into single beams. Each beam then gets its own system line. The single axes are overlapped and each beam has an offset value.

Before creating joints **compound sections** must be separated into **single beams** because joints can only be attached to single beams.

Before creating BOMs and drawings **compound sections** must be separated into single beams otherwise they are treated as *one* beam and this would result in an incorrect quantity of beams in the BOM.

Curved beams

The section classes previously listed in this chapter can also be created as curved beams. To create a curved beam, click the corresponding button and give a starting point and an end point for the length of the section, followed by one circle point for the radius.

As in the case of straight beams, the current coordinate system determines the position of the main axes of the beam. The curved beam web runs in the Z-direction of the current UCS (i.e., the 'top' of the section is in the Z-direction). The created curved beam can be rotated 90° about its system line.

 **Example:** Creating a curved beam HEA 240 between two columns (straight beams)

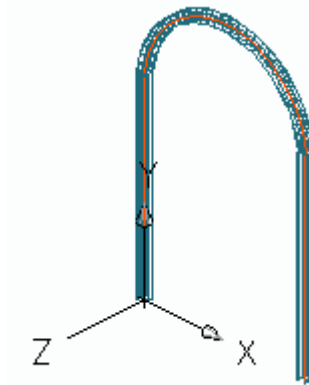


Figure 75: Curved beams

- Select a user coordinate system as shown in Figure 75.
- On the **Create objects** toolbar, **Sub – Beams** flyout, click . Use the upper system line end points of the columns as the start point and end point.
- Define the radius of the curved beam with a circle point. The circle point must be defined in the X/Y-plane of the coordinate system. Alternatively, pick a point at any radius and then specify the radius on the **Curved** tab of the dialog box. It is very important that the radius is set while the system line is at the center point (center of gravity). The system line can then be set on one edge and the radius previously entered will then correspond to that edge. Note that the value for the radius in the dialog box will then change since it displays the radius to the center of the section!
- The curved beam in this example must be rotated 90°. Rotate the beam on the **Position** tab of the **Beam** dialog box.

All other beam properties are the same as those of straight beams. On the **Curved** tab, in which the radius is set, there is a tolerance field to set the circle representation accuracy.

Poly beams

The section classes listed in the *Straight Beam* section of this chapter can also be created as poly beams.

To create poly beams, a polyline must be created. Any 2D or 3D polyline can be converted to a poly beam.

The position of the beam is independent of the coordinate system since the beam position is determined by the position of the polyline.

 **Example:** Creating a poly beam

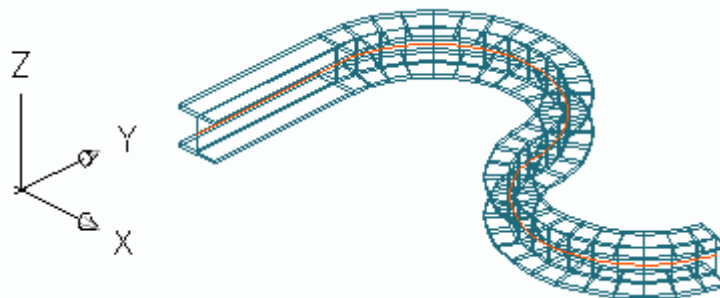


Figure 76: 2D poly beam

- Draw an AutoCAD® polyline in a suitable user coordinate system.
- On the **Create objects** toolbar, **Sub – Beams** flyout, click .
- Select the polyline.
- Confirm the selection by pressing **Enter**.
- The polyline can be kept or deleted.
- Type Y for Yes (or N for No) and finish by pressing **Enter**.

All other beam properties are the same as those of straight beams.

Any line or arc can be converted to a beam using the **Beam, polyline** tool.

Folded profile

Folded profiles are created using the **Sub – Beams** flyout of the **Create objects** toolbar. The cross section is defined directly inside the model by specifying the points (point by point) in the current UCS or by selecting an AutoCAD® polyline. The folded plate can be an open or closed section.

 **Example:** Creating a closed folded profile in the current UCS

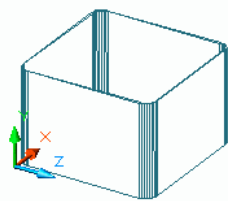


Figure 77: Closed folded profile (Z was vertical when created)

- Place a coordinate system with the X/Y-plane in the appropriate plate plane.
- On the **Create objects** toolbar, **Sub – Beams** flyout, click .
- Select the corner points one after another.
- End by pressing **Enter**.
- Click the start point of the system line.
- Click the end point of the system line.
- The folded profile is created.

A folded profile is also created by selecting an existing **polyline**.

The folded profile properties are set and changed in the dialog box. The thickness and the positioning are set on the **Section** tab of the properties dialog box. The radius of each corner can also be changed.

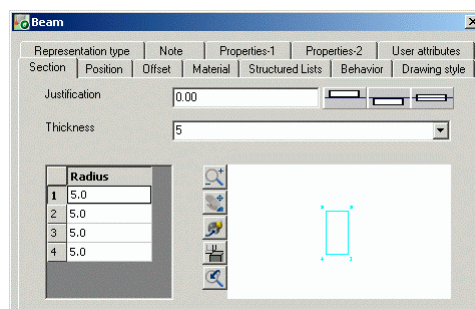


Figure 78: Section properties for the folded profile

Folded profiles have a specific representation type: "Exact (possible to modify the section)".

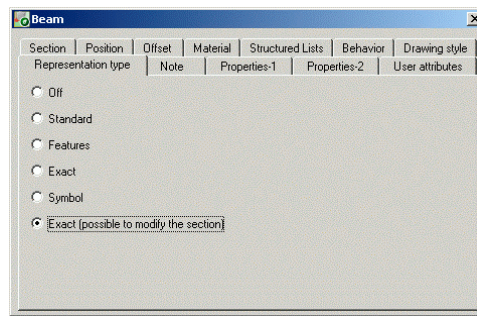


Figure 79: Folded profile – Representation type

If the representation type is set to "Exact (possible to modify the section)", the shape of the folded plate is modified by moving its grips.

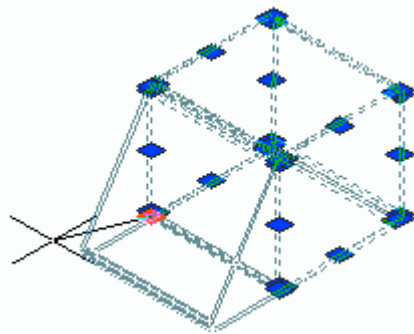


Figure 80: Folded profile – grip points

Construction systems (Cold rolled profiles)

The **Sub – Cold rolled profiles** toolbar (flyout) contains a variety of construction systems. They include cold rolled purlins, purlin accessories, side rails and curtain wall sections, etc., arranged by supplier: AGBrown, Albion, Ayrshire, BW Industries, Canam Manac, Canfer, Fischer, HiSpan, Kingspan, Krupp, Metsec, SAB, SADEF, Staba, StructuralSections, Ward, Zeta, etc. New systems are regularly added.



Figure 81: Sub – Cold Rolled Profiles toolbar (flyout)

These elements are created in the same manner as the other sections by entering a starting point and an end point. The rotation is dependent on the current user coordinate system.

Example: Creating a Krupp box channel ZR

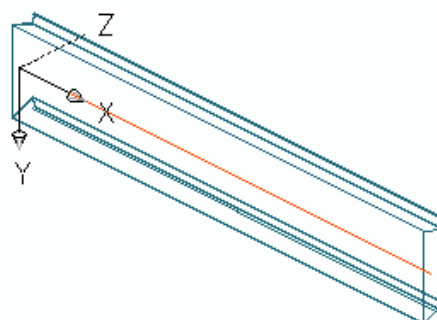


Figure 82: Krupp box section

- Select a suitable user coordinate system (see Figure 82).
- Click the Krupp tool (the **K** button) on the **Sub – Cold Rolled Profiles** toolbar.
- Select a starting point followed by an end point.
- The section is created.

Welded beams

Welded beams are created at an insertion axis. The sections are handled as a single section so that a trim affects all sections.

To create **welded beams**, use the flyout on the **Create objects** toolbar.



Figure 83: **Sub – Welded beams** toolbar (flyout)

Welded beams are created in the same manner as simple sections by entering a starting point and an end point in the current coordinate system.

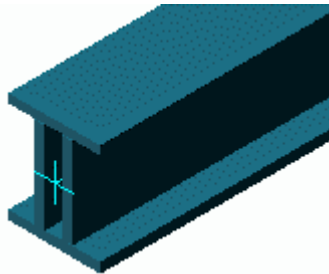


Figure 84: Welded beam - Hollow

The section class and the section of each individual element in the welded beam are controlled on the **Section** tab of the properties dialog box. The **Welds** tab configures the welds between the different elements.

Welded beams are stored in a library using **Save** and **Save as**.

Welded beams have two specific representation types:

- **Complete**: when the welded beam is cut, the entire section is cut.
- **Separated**: only the selected element of the welded beam is cut.

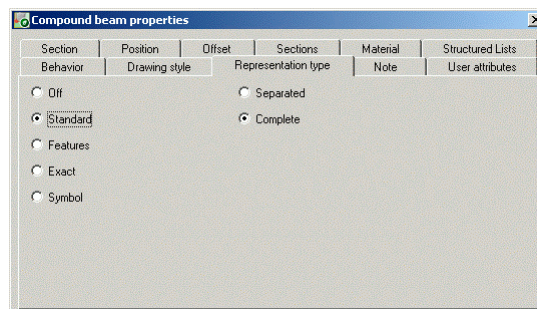


Figure 85: Compound beam properties – Representation type

Creating plates

Plates are created in Advance in almost any shapes and sizes in any plane. The Advance default plate thickness is controlled by the Management Tools.

The tools to create plates are on the **Sub – Plate** and **Sub – Folded Plates** toolbars (flyouts) of the **Create objects** toolbar.



Figure 86: **Sub – Plate** and **Sub – Folded Plates** flyouts

Tools includes:

- Rectangular plates
- Polygon plates
- Gusset plates
- Folded plates
- Conical and twisted folded plates
- Split and merge existing plates

3D folded plates of any shape are created by joining different plates together.

Rectangular plates are created using a center point, using two diagonally opposite corner points, or using three points that define a plane.

To create a polygon plate, pick the corner points directly or create a polyline that does not have to be closed. Polygon plates can be resized.

Any plate can be converted to a polyline and vice versa. The re-converted plate retains its initial properties.

Rectangular plates can be converted into polygon plates but the reverse is not possible.

Rectangular plate

Rectangular plates are placed in the X/Y-plane of the current coordinate system.

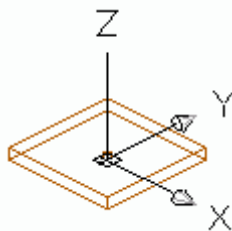


Figure 87: Rectangle plate, center

Example: Creating a rectangular plate using the center point

- Place a coordinate system with the X/Y-plane in the appropriate plate plane.
- On the **Create objects** toolbar, **Sub – Plate** flyout, click .
- Set a center point by selecting with an object snap or by entering coordinates.
- End by pressing **Enter**. The plate is created with a default length, width, and height.

In the “Plate” dialog box that appears after object creation, a variety of plate properties such as the plate size can be defined or changed on the **Shape** tab (length X, width Y).

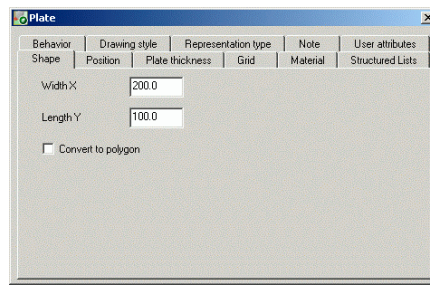


Figure 88: “Plate” dialog box - **Shape** tab

The **Plate thickness** tab is particularly useful for changing plate thickness from the bottom, center or top of the plate. For example, when increasing the thickness starting at 0.5 (the default center), half the plate thickness projects in the positive Z-direction and the other half projects into the negative Z-direction.

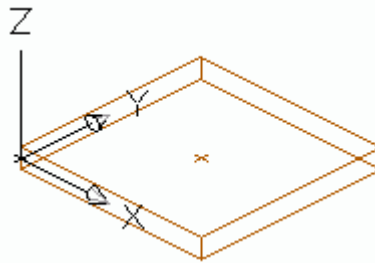


Figure 89: Plate thickness

On the **Position** tab, a reference point to the plate (e.g., center, center right, center left, bottom center, bottom right, or bottom left) can be assigned.

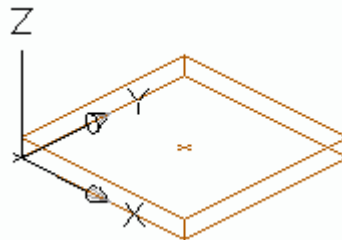


Figure 90: Plate position

The assigned reference point corresponds to the creation point. When changing the plate size on the **Shape** tab, this point remains in position.

Polygon plate

Entering the polygon corner points in the X/Y-plane of the current coordinate system will create a polygon plate.

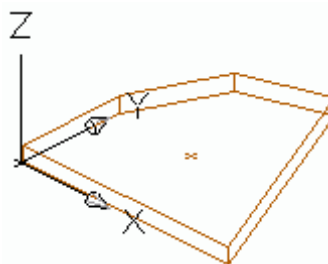


Figure 91: Polygon plate

 Example: Creating a polygon plate using corner points

- Set a suitable coordinate system.
- On the **Create objects** toolbar, **Sub – Plate** flyout, click .
- Select the corner points one after another.
- End by pressing **Enter**. The contour of the plate is closed to a polygon plate.

The properties of the plate are set and changed in the dialog box; dimensions in the X/Y-plane are not available for this plate type.

Another way to create a polygon plate is to transform an existing **polyline**, which does not have to be closed, into a plate.

In this case, the position of the plate is independent of the coordinate system since the plate position is determined by the position of the polyline.

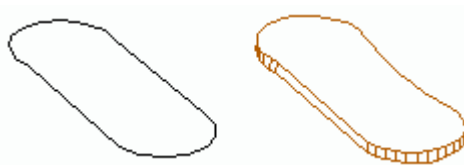


Figure 92: Polyline and created plate

- On the **Create objects** toolbar, **Sub – Plate** flyout, click “Plate on polyline” .
- Select a polyline and press **Enter**.

Any plate can be converted to a polyline.

- On the **Create objects** toolbar, **Sub – Plate** flyout, click “Plate to polyline” .
- Select the plate and press **Enter**. The plate can be kept or deleted.
- Type Y for Yes (or N for No) and press **Enter**.
- The plate is converted to a polyline and deleted.

The resulting polyline can be modified and converted back to a plate. The plate is created with its initial properties.

Gusset plates

Using the **Sub – Plate** flyout, gusset plates are placed in the current coordinate system plane on existing bracing members with predefined bolts (bolt patterns).

Two points on a straight line will define the guiding edge, which is kept when creating the gusset plate. After selecting the points, select a bracing member and its attached bolt pattern and, if required, other bracing members and bolt patterns.

 Example: Creating a gusset plate on a guiding edge

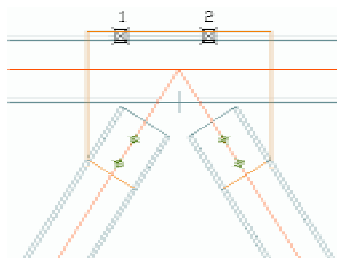


Figure 93: Gusset plate, guiding edge by point 1 and 2

- Place a UCS in the plane of the gusset plate.
- On the **Create objects** toolbar, **Sub – Plate** flyout, click .
- Determine the guiding edge with two points (1 and 2).
- Select a bracing member and its attached bolt pattern, then the second bracing member and the bolt pattern.
- End by pressing **Enter**.

All created plates are moved and/or changed using AutoCAD® and Advance commands (refer to the *Plate processing* section of this chapter).

Splitting / merging - plates

Existing Advance plates are divided into two or more plates using a polyline or two points. All plate processings (e.g., contours, etc.) are kept.

 **Example:** Rectangular plate divided on polylines

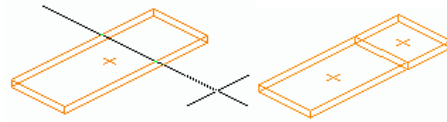


Figure 94: Plate divides on polyline(s)

- First create the polyline(s) on the existing plate at the dividing position.
- To divide the plate, click  on the **Sub – Plate** flyout of the **Create objects** toolbar.
- Select the plate / plates to divide (*identify objects...*) and press **Enter**.
- Select one or more polylines and press **Enter**.
- Enter a value for the required gap between the plates and press **Enter**.

To merge plates, click  on the **Sub – Plate** flyout, select the plates to merge, and press **Enter**.

Shrinking/enlarging polygon plate

Polygon plates can be shrunk or enlarged. One or more plates must be identified and a dimension must be entered to shrink or enlarge a plate. A negative value shrinks the plate while a positive value enlarges the plate.

 **Example:** Shrinking a plate

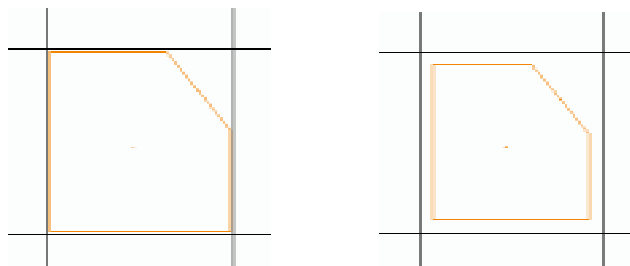


Figure 95: Polygon plate before and after shrinking

- To shrink a polygon plate, click  on the **Sub – Plate** flyout of the **Create objects** toolbar.
- Select the plate (one or more) to shrink and press **Enter**.
- Enter a dimension (e.g., 10). The plate edges move in a perpendicular direction by the entered dimension value.

Folded plate

The tools for creating folded plates are grouped on the **Sub – Folded plate** flyout of the **Create objects** toolbar.

Button	Function
	The first selected plate is the main plate and the second selected plate is moved to join the first one.
	The two selected plates are extended or cut along their bisector (intersection) and become one.
	Creates a conical folded plate starting from AutoCAD contours or from beams.
	Creates a twisted folded plate from polylines or splines.

Example 1: Two joint plates

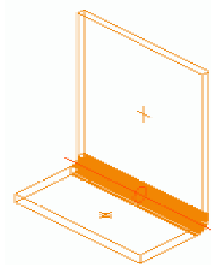


Figure 96: Folded plate

- On the **Create objects** toolbar, **Sub – Folded plates** flyout, click .
- Select the plate to connect near an edge.
- Select the second plate to connect near an edge.
- Set 90 for the angle.
- Press **Enter**.
- The two plates are joined.

The relation between the two plates can be modified in the “Folded Plate Relation” dialog box. Double click the red relation symbol.

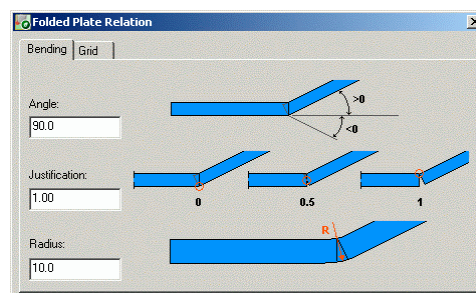


Figure 97: Bending properties for the folded plate

The angle, justification and radius values can be modified.

Any plate from the created folded plate can be set as the main plate using the “Set folded plate main object” tool.

“Check unfolding” validates if the created folded plate can be correctly unfolded in drawings and in BOMs.

Example 2: Conical folded plate using AutoCAD contours

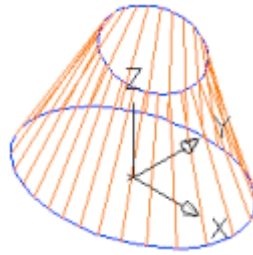


Figure 98: Conical folded plate

- Draw two AutoCAD® circles or ellipses in suitable user coordinate systems.
- On the **Create objects** toolbar, **Sub – Folded plates** flyout, click .
- Type **C** on the command line and press **Enter** to select the first contour.
- Select the first circle (or ellipse) and press **Enter**.
- Type **C** on the command line and press **Enter** to select the second contour.
- Select the second circle (or ellipse) and press **Enter**.
- In the “Conical folded plate” dialog box, set the number of facets, the plate thickness and the justification.



Figure 99: Conical Folded Plate dialog box

- The conical folded plate is created.

Coordinate systems

In AutoCAD® / Advance the **World Coordinate System (WCS)** is fixed and the **User Coordinate Systems (UCS)** can be moved/rotated. The UCS takes precedence in almost all cases except where a tool specifically refers to the WCS (e.g., Define model view with 1 point in WCS).

The Advance UCS tools are located on the Advance **Standard** toolbar.



Figure 100: Advance UCS tools

The first button opens the **Sub – UCS** flyout, which contains tools for object coordinate systems, coordinate system on curved beams, coordinate systems on a bisecting line, and coordinate system definition.

Figure 101: **Sub – UCS** flyout

The current UCS is rotated by 90° around the X-/Y-/Z-axes of the current user coordinate system using three buttons:

Button	Function
	UCS rotate around X
	UCS rotate around Y
	UCS rotate around Z

Button	Function
	A current UCS is moved to a new origin point, in which the orientation of the axes remains the same.

“View on UCS”  sets a view in the Z-axis of the UCS and “Clipped view on UCS”  sets clipping to improve the view area for the required construction.

The **Sub – Isometric view** toolbar (flyout) provides isometric views on the Advance model similar to the AutoCAD® isometric views but with a slightly different view angle.



Figure 102: Sub – Isometric view flyout

Object Coordinate Systems

Advance objects are always created relative to the current coordinate system.

Each Advance object has its own **object coordinate system**. The X/Y plane of the object coordinate system is perpendicular to the web (parallel to the flange) of a beam or to a plate surface. The Z coordinate points to the TOP of a beam or represents the default direction for a plate thickness.

If processings or connection objects are to be created at beams or plates, it is recommended to set a UCS orientation corresponding to a suitable object coordinate system. For example, an object coordinate system can be activated in a plate plane if a contour processing will be created in this plane.

- Rectangular plates have ten object coordinate systems.

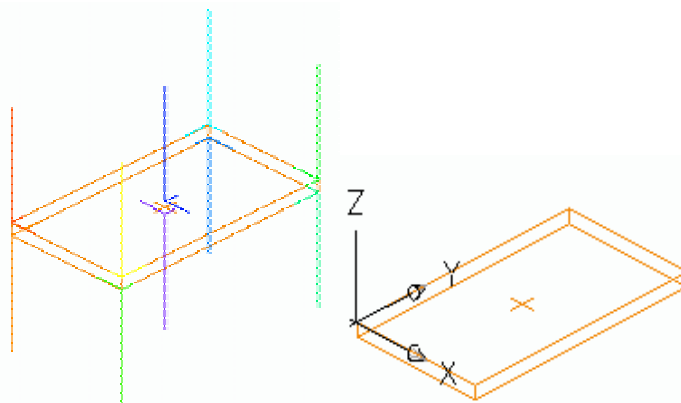


Figure 103: Plate with 10 object coordinate systems, current object coordinate system

- Beams have six basic object coordinate systems.

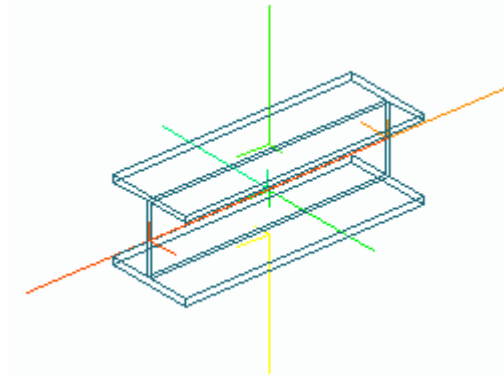


Figure 104: Beam with six object coordinate systems

Select an object coordinate system as a user coordinate system using the first tool ("UCS at object") on the **Sub – UCS** flyout.

 **Example:** Activating an object coordinate system of a polygon plate as a UCS

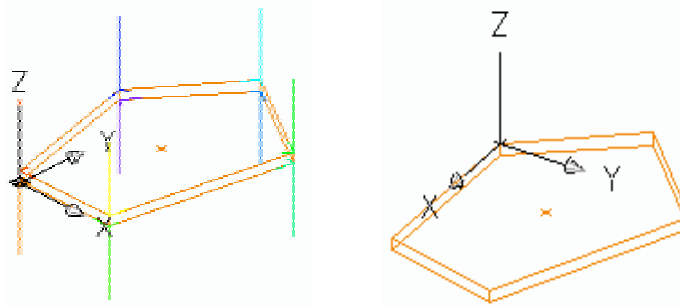


Figure 105: Object coordinate system as UCS

- On the **Standard** toolbar, **Sub – UCS** flyout, click .
- Select a polygon plate. The coordinate systems are displayed. The respective axis directions are displayed as lines with different lengths. The positive Z-axis is displayed as the longest line.
- Select one of these lines (e.g., the yellow line pointing upwards). A UCS is set corresponding to the selected coordinate system.

Coordinate systems at curved beams

A dedicated coordinate system tool is available for curved beams. Place an object coordinate system on any point of a curved beam (Z-axis follows the radial line) using the "UCS at curved beam" tool.

 **Example:** Creating a coordinate system at a curved beam

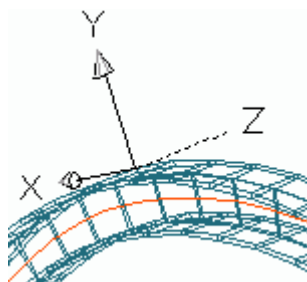


Figure 106: UCS at a curved beam (UCS has been subsequently rotated about the X-axis)

- On the **Standard** toolbar, **Sub–UCS** flyout, click .

- Select a point on the curved beam.
- Select an additional creation point for the position of the coordinate system or press **Enter**.
- The coordinate system is created.

The additional point does not need to lie on the beam. Thus, a UCS can be set on a curved beam at the exact location of an approaching beam for constructing a connection at this location.

UCS at a bisecting line

The “UCS to bisecting line” tool creates a user coordinate system at a bisecting line. This tool is suitable for creating sections at the bisecting angle of two other beams (e.g., in facade construction).

 **Example:** Creating a UCS at a bisecting angle

- On the **Standard** toolbar, **Sub – UCS** flyout, click .
- Locate a point of origin for the UCS.
- Pick another point to define the Z direction.
- Pick a point to define the first bisecting plane.
- Pick a point to define the second bisecting plane.
- The X-axis of the new UCS bisects the 2 selected planes.

Defining a coordinate system

The “Initial coordinate system” tool is useful for placement on existing objects. The position and orientation of the coordinate system corresponds to the coordinate system of the object at the time of creation. Thus, other objects with identical positions and orientations can be created.

For plates, a coordinate system is created with the same orientations as when the plate was created.

For beams, a coordinate system is created at the beginning of the beam system line. This coordinate system has an orientation that allows the creation of a new beam in the same orientation.

 **Example:** Creating a coordinate system on an object

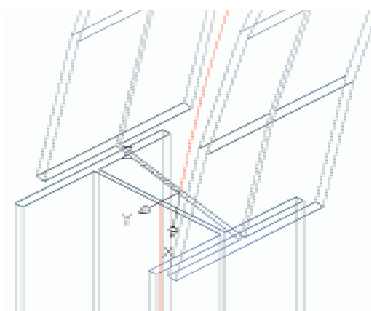


Figure 107: Initial coordinate system

- On the **Standard** toolbar, **Sub – UCS** flyout, click .
- Select an object (the curved beam in this case). At the end of the beam a red coordinate system line appears.
- Choose a red coordinate system line and with it a coordinate system.
- Press **Enter**. The coordinate system is created.

Beam processing

Advance beam processing creates almost any beam contour processing. A beam can also be processed by another element (e.g., another beam).

The beam processing is located on two flyouts of the **Create objects** toolbar.



Figure 108: Toolbars for beam processing

- **Sub – Process section** toolbar (flyout)
- **Sub – Section contour** toolbar (flyout)

Select one of the flyout icons and then click the beam end to process (e.g., trim, cope).

If two beams are individual, as for a miter cut, both beams must be identified after starting the tool. Follow the (AutoCAD®) command line prompts.

Processing is **dependent** on the UCS and is created parallel to the current coordinate system. A suitable coordinate system must first be set up. It is recommended (especially for contour processing curved beams) to place a user coordinate system at the beam before processing.

Some commands, such as creating a rectangular processing, a center point or two diagonally opposite points to determine a contour must be entered after identifying the beam end.



To create diagonal points switch off the **Ortho** mode.

Some of the processing tools work **independently** of the current coordinate system. For example, trim and cope tools are relative to the identified objects.

Upon creating a beam processing a dialog box for modifying other parameters appears.

Processing objects

Beams or plates are processed when creating a processing object. Processings are displayed as green contours in the model. These features are Advance Objects that cannot exist independently; they belong to a basic object (e.g., beam or plate).

Processing objects are created on the same layer as the associated basic object and can be deleted separately.

 **Example:** Object processing - beam trimming

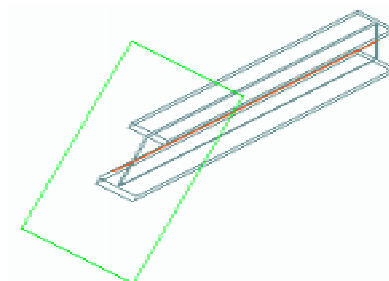


Figure 109: Beam trimming, rotation of the end plane

To trim a beam, a “shorten” processing object is created into the same layer as the beam. By setting a positive shorten value the beam is trimmed whereas setting a negative shorten value the beam is extended. The object processing is displayed in green.

If a processing object is deleted, the beam trimming including the end plane rotation is undone.

If the beam is deleted, the processing object will also be deleted.

Rule-based beam processing

Some of the beam processing (by adjusting one beam to another) is based on rules. (See the “Cope parametric – rule (contour processing)” command.) This means that when modifying a beam (e.g., a section) the processing is also modified.

As for joints, processings based on rules are controlled with a connection object, through which feature properties can be changed.

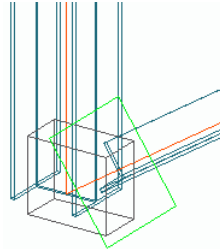


Figure 110: Connection object (dashed), processing object (rectangle)

The connection objects are represented in the model as gray objects (boxes).

Connection objects (boxes) are displayed by selecting one of the objects created by the joint and using the “Display” tool on the **Sub – Process section** toolbar (flyout) or by right clicking on the object and selecting **Joint properties** from the menu. Processings are displayed in the model using the “features” beam representation style. For joints that only create processing objects, this representation style is required to display the connection object (box).

Process section

The **Sub – Process section** flyout contains the following tools:



Figure 111: Beam processing, **Sub – Process section** flyout

Button	Function
	Shorten for straight and curved beams
	Shorten at UCS for straight and curved beams
	Miter cut
	Cope
	Cope skewed
	Split beam
	Merge beam
	Cope, parametric – Rule
	Cut at object – Rule: Beams are modified to other beams with a diagonal cut
	Cut variable – Rule: Beams can be cut on a miter and a bracing member can be modified to other beams
	Representation of connection object set by rules

 **Example 1:** Beam trimmed by 100 mm, beam end plane rotated by 30° (see Figure 109)

- On the **Create objects** toolbar, **Sub – Process section** flyout, click .
- Select a beam at a reference end.
- The “Shorten beam” dialog box appears and the beam processing is represented as a green contour using the default value.
- Select the **Angle** tab. Input the rotation angle for the beam end plane. The reference point is where the beam system line intersects the cutting plane. The beam end plane is rotated.

 **Example 2:** Splitting existing Advance beams

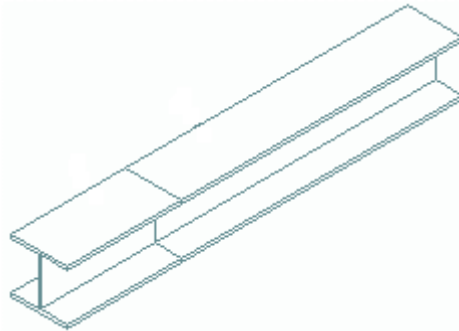


Figure 112: Split beam

- On the **Create objects** toolbar, **Sub – Process section** flyout, click .
- Select the required beam and press **Enter**.
- Select the separation point(s) and press **Enter**,
- Enter a value for the gap that should be between the single beams. Note that a gap will shorten the system lines and not apply shorten features.

Using the selected beam, two or more beams are created. Processings of the original beam are transferred onto the newly created beams. When merging beams, the two or more beams to be merged must be selected.

Section contour

The **Sub – Section contour** flyout contains the following tools:



Figure 113: Beam processing, **Sub – Section contour** flyout

Button	Function
	Rectangular contour, Center: A rectangular contour is cut from the beam. It can be an outside (cope) or inside (hole) contour. The contour is created parallel to the UCS.
	Rectangular contour, 2 Points: The processing can be assigned to a beam end to keep its position if the length changes. The feature is created parallel to the UCS.
	Circular contour, center, parallel to UCS
	Circular contour, 2 Points
	Polygon contour: Any contour (polygonal, circular shape) can be cut from the beam. For curved beams it is recommended to first set a UCS on the object.

Button	Function
	Element contour processing: A straight or curved beam is processed through the geometry of another straight beam.
	Element contour - rule

 **Example:** Creating a circular processing in curved beams.

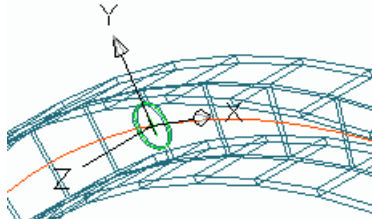


Figure 114: Circular processing in curved beams

- The processing tool is dependent on the current UCS. Therefore, set a beam coordinate system using the “UCS at curved beam”  tool (**Standard** toolbar); rotate, if needed, so that the X/Y-plane lies on the web.
- On the **Create objects** toolbar, **Sub – Section contour** flyout, click .
- Select a beam at a reference end.
- Next, set the center point of the contour processing (0,0,0).
- The contour processing is created and the “Contour processing” dialog box appears.
- Radius, position, contours (gap, width), and grid values can be changed.

Beam processings are a very powerful feature. For further details refer to the *Advance online Help*.

Plate processing

The tools for plate processing are located on five different toolbars (flyouts).



Figure 115: Toolbars for plate processing

Processing objects are classified as either independent or dependent on the current coordinate system, corner finishes, or features for polygon plates. The five different processing styles are each arranged on their own toolbars.

Polygon plates are enlarged or shrunk using the “Shrink/enlarge polygon plates” tool. This tool is frequently used in curtain wall construction to fit created elements by entering a positive or negative value (e.g., for creating glazing inside a frame).

Plate processings are created in the model as green contours. They are always perpendicular to the plate plane. Features made using a diagonally passing element are trimmed in such a way that the processing edges are perpendicular to the plate plane.

Advance plate processing objects cannot exist independently; they exist only as objects attached to a plate. The plate processings can be deleted as a separate object. If the processed plate is deleted the feature is deleted as well. Processing objects are on the same layer as the associated plate but the color is set to green for recognition. Combining individual processing objects can create any imaginable feature shape.

When creating a plate processing object a context sensitive dialog box appears. The various tabs contain additional processing options.

Plate processing – weld preparation

The weld preparation tool sets weld preparations on plate edges and changes the plate front edges (in the Z-direction) with bevels or fillets.

The **Sub – Plate cuts** toolbar (flyout) contains weld preparation tools.



Figure 116: Plate processing, **Sub – Plate cuts** flyout

There are five ways to create weld preparations:

- Cut plate at UCS
- Cut plate at another plate
- Miter plates
- Weld preparation, chamfer
- Weld preparation, fillet

To create a weld preparation, a side that has straight edges must be selected. The “Exact with weld preparation” representation type shows the resulting exact plate shape with bevels or fillets at the edges. Weld preparations are represented in green.

 **Example:** Creating a bevel weld preparation

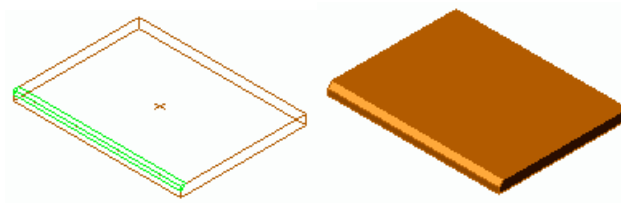


Figure 117: Plate with weld preparation

- On the **Create objects** toolbar, **Sub – Plate cuts** flyout, click .
- Select a plate at a reference edge.
- The weld preparation is created and the “Weld preparation – bevel” dialog box appears. Bevel angle, bevel height, and bevel width can be changed along with the bevel side (above/below).

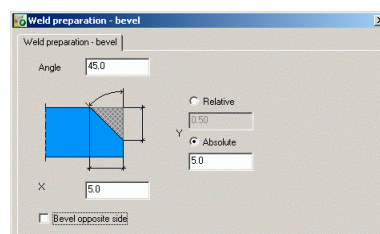


Figure 118: “Weld preparation” dialog box

Since the **bevel angle** value and **bevel width** X are related, the second value is calculated automatically when one value is entered.

Available options for bevel height are relative or absolute.

The weld preparation is related to the edges such that it is updated if the plate contour is changed.

Cut plate at UCS

The "cut plate at UCS" and "cut plate at another plate" commands extend or cut a plate to the UCS or at another plate and create a chamfer (for weld preparation).

Cutting a plate will change the plate contour. A cut can be beveled where the bevel can be set as absolute or relative.

There are 3 main options for the type of chamfer to apply to the plate:



Figure 119: Chamfer options

Cut plate at plate

This command extends or cuts one plate to another and creates a chamfer (for weld preparation) at the modified plate.

There are 3 main options for the type of chamfer to apply to the plate:



Figure 120: Chamfer options

Plate processing – independent from the UCS

The **Sub – Plate Contour** toolbar (flyout) contains plate processing tools, which are used independent of the current user coordinate system.



Figure 121: Plate processing, **Sub – Plate Contour** flyout

There are four basic processing styles.

- One or two points to determine rectangular processing.
- One centre point or two points – (centre point and radius point) creates a circular processing object.
- Any contour processing (selected through points) can be developed as inner or outer contours.
- Using the "Contour element"  feature, a plate can be processed by a beam or by another plate.

To create a plate processing, the plate is identified near its corner. Then the program requests a center or selected points that determine a processing contour.



*To create points the **Ortho** mode must first be turned off.*

A "Plate contour" dialog box appears, in which the form of the plate contour (length X, width Y), the position, contour (gap width), grid value, and corner feature (radius, boring) can be specified.

Example: Plate contour element processing

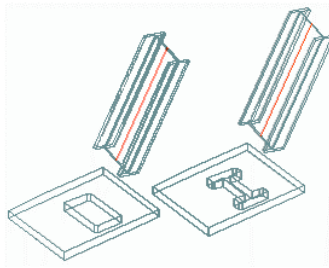


Figure 122: Plate with casing and exact processing

- With this tool, a beam processes a plate. Select the type of processing with the switch on the **Create objects** toolbar.

Button	Function
	Exact cross section
	Casing cross section

- On the **Create objects** toolbar, **Sub – Plate Contour** flyout, click .
- Click near a corner of the plate to be modified.
- Next, select the element to modify. The plate processing is performed.
- The “Contour processing” dialog box appears. The **contour** (gap width), **grid** (increment for modifying the length by the grips), and **corner finish** (fillet, boring) can be specified.

Plate processing – dependent on the UCS

The plate processings on **Plate contour UCS** the toolbar (flyout) are **dependent** on the current coordinate system.



Figure 123: Plate processings, **Sub – Plate contour UCS** flyout

The following UCS dependent processing types are available:

- Rectangular processing determined by one or two points
- Circular processing determined by a center point or two points (center and radius point)
- Any contour processing selected by points



Verify that a suitable user coordinate system (UCS) is set before starting the feature tool.

The procedure for creating a UCS dependent plate processing is identical to that of UCS-independent processing.

Some plate features, in addition to being independent, can also be dependent on the current user coordinate system (UCS). For processings dependent on the UCS, the definition points of the object processing are placed in the coordinate system plane and projected along the Z-direction onto the plate plane. The result is that the length and width of the processing in the plate plane are distorted. For an independent processing, the definition points and the object processing are placed directly in the plate plane.

 **Example:** The plate processing circular contour and circular contour UCS

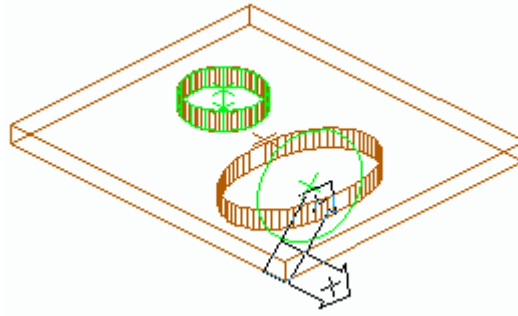


Figure 124: independent (circle) and dependent processing (ellipse) on the coordinate system

This example illustrates the effect of a UCS dependent circular processing. The circular processing lies in the coordinate system. The circle is projected onto the plate plane and presents an oval feature. The processing edges are always perpendicular to the plate plane.

Corner processing

The **Sub – Plate Chamfer** toolbar (flyout) includes commands for corner processing and fillets of plates.



Figure 125: Plate processing, **Plate chamfer** flyout

Corner processing can be attached in two ways, either by defining one or two points.

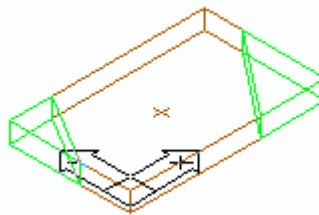


Figure 126: Corner processing on plate

 **Example:** Corner processing with two points

- On the **Create objects** toolbar, **Sub – Plate Chamfer** flyout, click .
- Specify the first point by picking the side of the plate at the start point of the cut.
- Repeat this process for the second edge.
- The corner processing is created between the two selected points and is represented with a green contour processing. The “Plate chamfer” properties dialog box appears in which the length X and the width Y of the corner processing can be changed. The first identified edge represents the X direction.

Polygon plate processing

Polygon plates are processed using commands on the **Sub – Plate corner** toolbar (flyout) of the **Create objects** toolbar. Corners can be added or deleted using commands on this toolbar.



Figure 127: Plate processing, **Plate corner** flyout

Button	Function
	To create an additional corner, select the "Create corner" tool and select a plate edge. The additional corner is created on the mid-point of the selected edge (where its position can then be modified using the grip point).
	The "Remove corner" tool deletes a corner so that a new edge is created between the adjacent corner points.

Changing Advance objects

Position, geometry, and attributes of Advance objects (e.g., beams, plates) can always be changed using:

- AutoCAD® / Advance grips
- AutoCAD® tools: Moving, Rotating, Copying, Mirroring
- AutoCAD® **Properties** command
- Advance **Properties** command

Changing objects using grips

Grips are displayed (AutoCAD® default is blue) by clicking on an element (e.g., a plate or its feature) and are activated (AutoCAD® default is red) by clicking on a grip. The elements geometry or features are changed with grips as can their size or position by dragging a grip(s). Changes using grips are limited to the X/Y-plane of the current coordinate system.

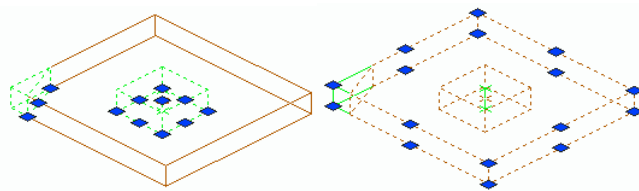


Figure 128: Grips for changing the plate processing / plate

For some manipulations the grips have no effect if modified outside the UCS X/Y-plane.

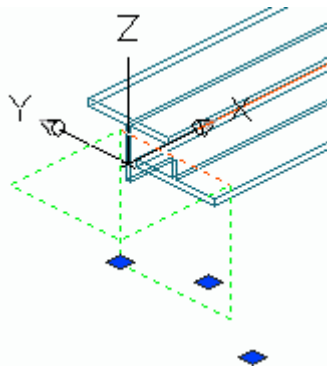


Figure 129: Advance grips for changing the beam processing

In the above example, the grip is only moveable in the X-direction.

For some objects, it is also possible to assign grid values that are used when using the grips.

For example, a shortening can only be moved by the given grid value (increment) using the grips in the current user coordinate system (UCS).

Grid values are always applied globally to all types of objects and are not stored per object. This means, if a value was created for a plate feature, it is also applied to other plate features that are created. This also applies for the gap between features and the objects that are to be processed.

AutoCAD® Manipulating tools

For Advance objects, the following **AutoCAD® Manipulating Tools** are implemented:

- Move
- Rotate
- Array
- Mirror.

The AutoCAD® standard tools such as “Move”, “Rotate”, etc. are available by right mouse click. First, click a grip (in **red** by default), then right click.

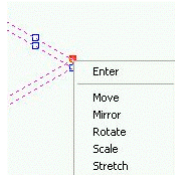


Figure 130: AutoCAD® Manipulating Tools

The right click menu is context sensitive and only the available features/tools are displayed for an object.

When changing the object geometry using AutoCAD® grips, the coordinates of the moving point are projected onto the current coordinate system.

AutoCAD® command properties

To change the AutoCAD® properties of an element such as layer, color, etc., right click on the object and select **Properties** (not Advance Properties). Alternatively on the command line type the AutoCAD® DDCHPROP command.

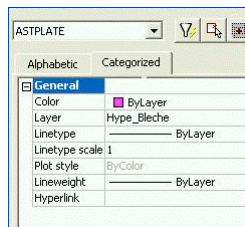


Figure 131: AutoCAD® properties of a selected plate

Note that changing layers is simpler by selecting the objects to modify, then selecting the appropriate layer on the AutoCAD® Object Properties toolbar.

Advance command properties

Each Advance object has initial default properties. The properties are viewed and changed in the Advance **Properties** dialog box.

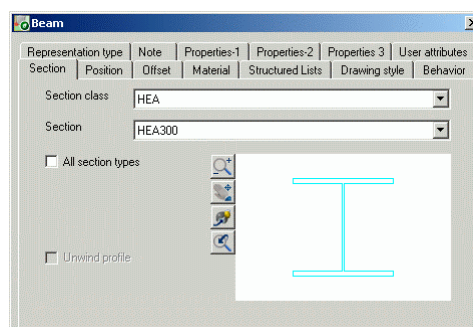


Figure 132: Advance **Properties** Beam dialog box, **Section** tab

Double clicking an object opens the **Advance Properties** dialog box. The **Advance Properties** dialog box also appears by selecting **Advance Properties** or **Advance Joint Properties** on the context menu.

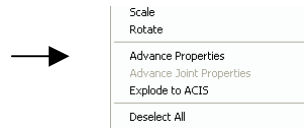


Figure 133: Context menu

Another way to call the properties dialog box is to select the required element and then click the “Properties” icon on the Advance **Standard** toolbar.

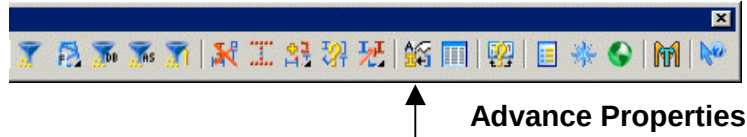


Figure 134: Advance Properties icon on the Standard toolbar

Representation Type – Snaps – Grips

To keep Advance clear and simple in the different phases of construction, Advance objects have their own different representation types (e.g., “standard”, “features”, and “exact”).

Since the different representation types of objects normally correspond with different constructive courses, the representation types in each case use different snaps and have grips.

The total construction of a model is clearest in the “standard” representation type, in which processing objects are not shown.

When processing and connecting beams or plates, the “features” representation type is very useful, especially since the required grips for changing the features are available.

- The representation types are changed for each object either in their dialog box (e.g., beams, plate, **Representation type** tab) or clicking  on the Advance **Standard** toolbar (refer to the *Changing representation type* section of this chapter).

The representation types (and available grips) are specific to each object. The examples use beam and plate objects. Refer to the Advance *online Help* for more details.

Example: Beam representation

An Advance beam is displayed in one of four representation styles:

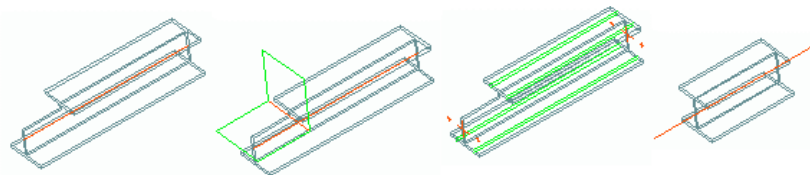


Figure 135: Beams in the representation types “standard”, “features”, “exact”, and “symbol”

Beam	Representation type	Name
Representation 1	Standard	Section edges and creation axes of the beam
Representation 2	Features	Section edges and processing objects
Representation 3	Exact	Section edges and gauge lines, crosses on end points of the creation axis and four reference points in the start and end planes
Representation 4	Symbol	Creation axis and a small part of the beam in the center

The beam has different grips in each representation type:

	Representation type	Name
Representation 1	Standard	End point creation axis, reference point in beam center
Representation 2	Features	Grips to change the processing
Representation 3	Exact	Reference and corner points of the beam
Representation 4	Symbol	See standard representation type

A beam, in all representation types, is selected at the “mid point”, “center”, “node”, or “end point” snaps. The snaps are different for each representation type.

Example: Plate

For an Advance plate, four representation types are available:

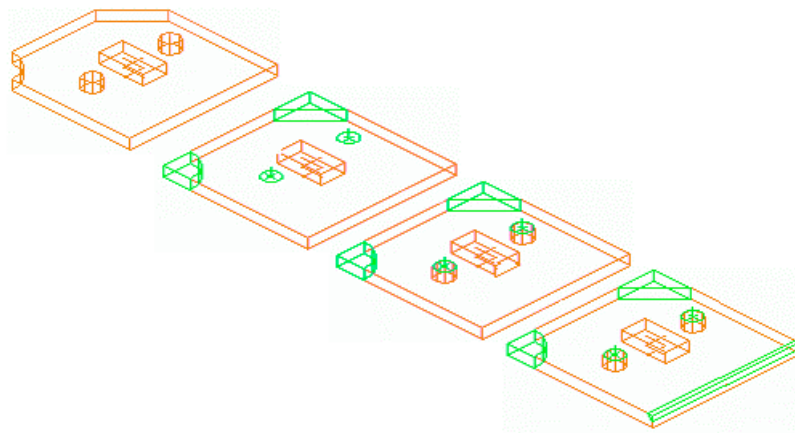


Figure 136: Plate in “standard”, “features”, “exact”, and “exact with weld preparation” representation types

Plate	Representation type	Name
Representation 1	Standard	Plate contour without processing
Representation 2	Processing	Plate contour with processing
Representation 3	Exact	Plate contour with processing and hole patterns
Representation 4	Exact with weld preparation	Same as Representation 3, with weld preparation

By changing the representation type of a plate, the following grip points are available:

Plate	Representation type	Name
Representation 1	Standard	All corner points and midpoints of the edges
Representation 2	Processing	All corner points and midpoints of the edges, grips for changing the processing
Representation 3	Exact	All corner points and midpoints of the edges
Representation 4	Exact with weld preparation	All corner points and midpoints of the edges

All corner points of the plates can be picked with the “End point” object snap.

Bolt and Hole Patterns / Shear Studs

Bolt/hole patterns and shear studs are created in any plane and are dependent on the current coordinate system.

Bolt patterns create connections between individual Advance objects (e.g., beam/plate or plate/plate). The respective connection elements and properties are stored and managed by Advance.

The structure formed by connected elements is displayed with a special Advance command. Adding and deleting elements to connections is simple. The connection is interpreted during the numbering so that it automatically distinguishes between main parts and attached parts.

Bolts are created in a **cyan** color (by default) as individual Advance elements and are changed with the **Advance Properties** command.

Holes are automatically created when bolts are created. If a bolt pattern is deleted then the hole pattern is also deleted.

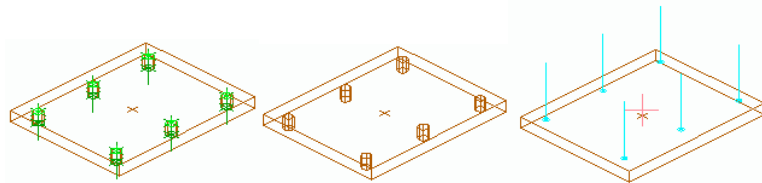


Figure 137: Bolts, holes, and shear studs, all in the standard representation

A hole pattern without bolts is created with the following options: round holes, slotted holes, countersunk holes, blind holes, threaded holes and punch marks.

Holes, like process features, are dependent objects that are displayed in green in the model. As with features, they are only visible when the object, in which they are created, is displayed using the “features” or “exact” representation types. Hole objects must be visible to change (with the **Advance Properties** command) or delete them.

Shear studs are created in the model exactly the same way (**cyan**) as bolts with the same options for displaying and editing. Holes are dependent on their reference object for their representation type.

Bolt/hole patterns and shear studs have a **Behavior** property tab that describes if and how the element numbers are part of the structured BOM.



Figure 138: Icons Bolts/Holes/Shear Studs, tool bolt/hole patterns

All three connection types (i.e., bolt patterns, hole patterns and shear studs) are created with the same commands. Click on the corresponding button to create the objects.



Figure 139: Sub – Bolt Hole patterns flyout

Bolt/hole patterns and shear studs are always created in a preset area with a certain creation point or created on gauge lines. The following toolbar buttons are available:

Button	Function
	Rectangular area, limited through diagonally opposite corner points
	Rectangular area, unlimited, with corner creation point
	Rectangular area, unlimited, with center creation point

Button	Function
	Circular area with center creation point
	Bolts on gauge line of intersecting sections

Using other tools, bolt/hole patterns can be moved or separated into single bolts/holes. The “Calculate grip length” tool recalculates the grip length.

Creating a bolt pattern

- Set a suitable user coordinate system.
- On the **Create objects** toolbar, choose the default “bolts” .
- Set the representation type if required. For example, for beams, use the exact representation type if a bolt pattern should be created on a gauge line.
- Select the appropriate tool (based on the creation area) on the **Sub – Bolt Hole Patterns** flyout of the **Create objects** toolbar.
- Select the elements to connect.
- Define a **rectangular area** using two diagonally opposite corner points (... bottom-left corner, ... top-right corner) or one **start point** or one **center point**.

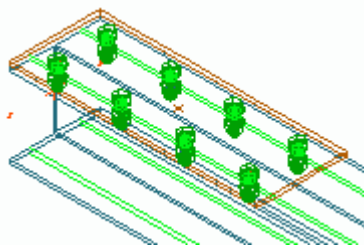


Figure 140: Bolt pattern on gauge line, “exact” beam representation, “solid” bolt representation

The bolt pattern is created and the “Bolts” dialog box appears.

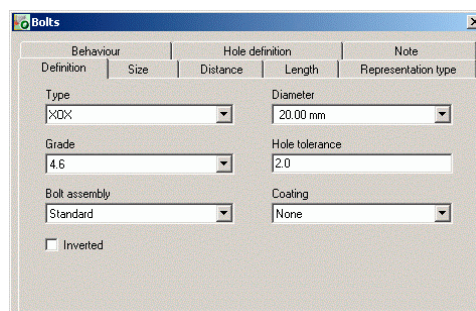


Figure 141: “Bolts” dialog box, **Definition** tab

Important information that can be modified in the dialog box:

- Definition: Type, grade, diameter, etc.
- Size: Number of bolts in the X and Y-directions
- Distances: Distances in the X/Y-direction, edge distance (bolts distance minus the plate edge or beam edge)
- Representation Type: Standard, frame, solids
- Note: Assembly type (site, site drill, shop, etc.)

Creating a hole pattern

- Set a suitable user coordinate system.
- On the **Create objects** toolbar, select the default “hole pattern” .
- Select the appropriate tool on the **Sub – Bolt hole patterns** flyout based on the creation area.
- Select one end of a plate or beam.
- Create a **rectangular area** using two diagonally opposite corner points (... bottom-left corner, ... top-right corner) or one **start point** or one **center point**.

The hole pattern is created and a dialog box appears. Important information fields that can be modified:

- Hole definition: Type (e.g., slotted holes, countersunk holes, etc.)
- Arrangement: Number of holes in the X / Y-direction
- Distance: in the X / Y-direction, edge distances.

Hole representation types are dependent on their reference object.

Creating shear studs

- Set a suitable user coordinate system.
- On the **Create objects** toolbar, select the default “Shear studs” .
- Set the representation type, if required.
- Select the appropriate tool (based on the creation area) on the **Sub – Bolt hole patterns** flyout of the **Create objects** toolbar.
- Select the element that will have the shear studs.
- Define a **rectangular area** using two diagonally opposite corner points (... bottom-left corner, ... top-right corner) or one **start point** or one **center point**.

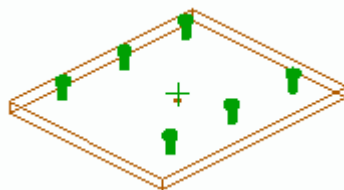


Figure 142: Shear studs in the “solid” representation type

The shear studs are created. They are displayed in standard representation and connected to a beam / plate through a weld point. A dialog box appears, in which the following information can be entered:

- Definition: Norm, diameter, coating, etc.
- Arrangement, distances and representation type (i.e., “standard” and “solids”).

Shifting a bolt/hole pattern

Bolts or holes on an incorrect surface of an object can be shifted along their axis to the correct face.

- To **shift** an existing bolt pattern, click the “Shift bolt/hole”  tool on the **Sub – Bolt hole patterns** flyout of the **Create objects** toolbar.

- Select the bolt and/or hole pattern to move and press **Enter**. Next, select the element (beam or plate) by picking it at the new reference plane.

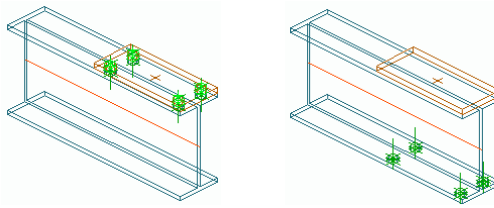


Figure 143: Shift Bolts/Holes

Splitting a bolt/hole pattern

Bolt patterns consisting of several bolts can be split into single bolts. These single bolts behave as bolt patterns created with the 'center' option.

- Split a bolt pattern (or hole pattern) into single bolts by clicking  on the **Sub – Bolt hole patterns** flyout of the **Create objects** toolbar and selecting one or more bolt patterns. The bolt patterns are split into single bolts.

Recalculating a grip length

- To recalculate the grip length of bolts, click the “Calculate grip length”  tool on the **Sub – Bolt hole patterns** flyout of the **Create objects** toolbar. Select the preferred bolt patterns and the element(s) to connect.

The grip length is recalculated.

Bolts on gauge lines

This tool creates bolts on the gauge lines of the crossing sections (they must touch each other).

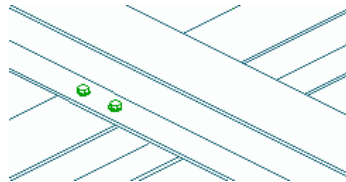


Figure 144: Bolts on gauge lines

- On the **Create objects** toolbar, **Sub – Bolt hole pattern** flyout, click .
- First select the main beam, press **Enter**, and then select the other beam. The bolt patterns are created on the gauge lines of the two sections. A dialog box appears for modifying the specifications for the bolts and positions.

Welds

Welds are created as weld points or weld lines. These objects contain not only the weld properties but also the logical connection between connected structural parts.

With the Advance **ASTemplate.dwt** standard template, weld points are created as individual elements in the form of a pink cross whereas weld lines are represented in the model as thick polylines.



Figure 145: Weld patterns (flyout)

Weld patterns are created using the “Weld point” and “Weld line” tools the **Sub – Weld patterns** flyout.



Figure 146: Weld patterns (flyout)

- To create a **weld point** with its information, click  on the **Sub – Weld patterns** flyout of the **Create objects** toolbar. Select the objects to connect, press **Enter**, and enter the center point of the weld to be created.
- To create a **weld line** with its information, click  on the **Sub – Weld patterns** flyout of the **Create objects** toolbar. Select the objects to connect, press **Enter**, and enter the start point for the first weld line and one or more end points.

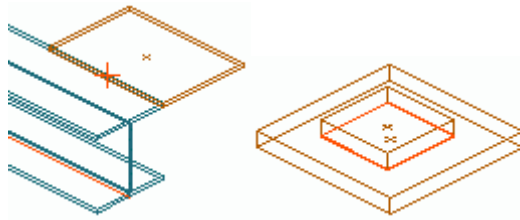


Figure 147: Weld points and weld line

Weld objects can store information separately for the **upper** and **lower** weld: basic weld, surface shape, weld design, weld thickness; **weld definition**: weld location, single length, circular (continuous); **representation type**: “Standard” or “Off” to hide the weld; “additional information” option to add weld text for additional information.

Connections

Advance objects are added or deleted from the bolt or weld connection objects with Advance tools. At the same time, the grip length (and bolt length) is modified.

The check tools highlight the connected objects in red. Checking the bolt or weld connection objects is important to insure that the correct parts are represented on shop drawings.

Checking tools for existing connection objects are located on two flyouts.



Figure 148: Tools connections / check connections

The tools on the **Sub – Connections** flyout add or delete Advance objects from a connection (change connections).

The **Sub – Check connections** flyout contains the tools for checking and highlighting (in red) the connected objects.

Changing connections

To add or remove objects from a bolt or weld connection (with adjusting of grip and bolt length), use the tools on the **Sub – Connections** flyout of the **Create objects** toolbar.



Figure 149: **Sub – Connections** flyout

Button	Function
	Connection, add objects
	Connection, disconnect objects

- Click the corresponding icon on the **Sub – Connections** flyout.
- Select the connecting elements.
- Select the object to add or delete.

Checking connections

The following tools check the connections of the connected objects and highlight them in **red**.



Figure 150: **Sub – Check Connection** flyout

The “Display connected object” and “Remove marking + display connected objects” tools highlight objects with both shop and site connections, including beams, plates, bolt patterns and welds.

Example: Checking a portal frame connection

- A portal frame consisting of two columns and two rafters was connected using various joints for the apex, eaves and base plates.
- On the **Create objects** toolbar, **Sub – Check Connection** flyout, click the "Display connected object" .
- Select an element of the portal frame.
- The complete frame is highlighted in red, provided that all connections were applied correctly.

Using the “Display objects connected in shop” and “Remove marking + display objects connected in shop” tools, the objects connected in the workshop are displayed, including beams, plates, bolt patterns and welds.

Using the “Display connecting elements” and “Remove marking + display connecting elements” tools, it is possible to examine an object with its connecting elements and identify them. For example, it can be determined that a weld point lies far away from the object to be welded.

During the command the identified connection element objects are highlighted in red and their number displayed in the text window.

- To clear the display of highlighted objects, click the “Clear highlighted objects” icon on the Advance **Standard** toolbar.



Figure 151: Advance **Standard** toolbar (Clipped), remove display

Working methods I

Advance tools for selecting, filtering, and marking objects are available on the **Standard** toolbar.



Figure 152: Advance **Standard** toolbar, (clipped)

Selecting objects

Only certain Advance elements can be selected for copying, numbering, exporting to a structured BOM, etc.

There are different options for selecting objects. The elements can be clicked (single selection) or selected using AutoCAD® tools like crossing windows or enclosing windows. Alternatively, previously highlighted objects (in red - refer to the *Check connections* section of this chapter) can be selected with the “Select highlighted objects” tool.

Another option is to select all Advance objects in an AutoCAD® drawing with the “Select Advance Steel objects”  tool.

With the “Reverse selection” tool, the selected objects are un-selected and the un-selected objects are selected.

These selection tools also apply to hidden objects or objects on frozen layers.

A very useful tool for selecting objects is the Advance **Filter tool**. This tool is particularly useful for overlapping objects (e.g., selecting a line of a structural element that lies on the axis of a building grid), or for selecting certain object types (e.g., plates) to put in standard representation, or for creating a structured BOM for beams.

Object filter

To use a command in AutoCAD® / Advance, select the elements first and the tool afterwards or vice versa.

 *The Advance element filter is a selection before a command. The filter would cancel the command if called after it.*

The Advance filter works in **two** steps:

- Click the filter tool.
- Select the object type.

Two icons are available on the Advance **Standard** toolbar for the **Filter tool**: “Filter selected objects” where the filter tool applies only to selected objects and “Filter database” which selects using the entire AutoCAD® /Advance database.

After calling the filter tool, **object types** must be selected on the **Sub – Objects** flyout.



Figure 153: Selection Objects

The following object types are available as selection objects: Structural elements, connection objects, beams, curved beams, beam – features, beam – holes, plate, plate – features, plate – holes, bolts, welds, shear studs, level symbols, special parts, cameras, and grids.

 **Example:** Filtering beams from the selected Advance elements

- Select a group of Advance objects. This is done by marking a group of objects connected with weld or bolt objects using the “Display connected objects”  tool (but may also be selected by other means).

- Next, select the highlighted objects with the  tool.
- Start the Advance filter tool by clicking  ("Filter selected objects").
- Click the second icon (**Sub – Objects** flyout) to enter the object type to filter and press **Enter**.
- The selected objects are filtered and the results are displayed in the text window.

Use the "Filter database" icon on the **Sub – Objects** toolbar to filter a certain object type from the Advance database (model).

Marking / Unmarking objects

Selected objects can be highlighted individually thus additional objects can be added to the current group of highlighted objects.

 **Example:** *Selecting / marking beams + additional marking of plates*

- Select all beams in a model using the "Filter database"  tool and choose **beam** as the **selection object (Sub – Objects** toolbar). All beams in the model are selected (shown dotted).
- Click the "Add selected objects to marking"  tool. All beams are highlighted in red.
- Repeat the above steps for plates so that all beams and plates in the model are highlighted red.

The selected objects can be deselected using the "Remove selected"  tool on the **Standard** toolbar. Markings can be deleted from a group of highlighted objects.

- Filter all plates from the model and click "Remove selected object from the marking" . All selected Advance Objects (in this case plates) are deselected.

Searching / Marking objects

The "Search filter"  tool on the **Standard** toolbar is used to select a group of elements that have the same properties.

On calling this tool a dialog box appears in which search criteria, general or object specific, can be entered. By clicking **OK**, the search will start. A listing of the found and highlighted objects is displayed in the text window.

The highlighted objects are put into a selection set (e.g., to copy them) with the "Select highlighted objects"  tool (**Standard** toolbar).

The selections will remain when other toolbars are called from the **Advance Steel** toolbar. It is of no importance how the selection was made.

Within a large model, **highlighted objects** (e.g., collision solids, connected objects) are not always easy to find. The objects can be displayed with an arrow starting from the center of the viewport for easy identification. The arrow can be deleted or hidden, just like any other object.

- On the **Standard** toolbar, **Sub – Marking** flyout, click "Search highlighted objects" . An arrow pops up from the center of the viewport to the highlighted object.
- The "Zoom to highlighted object"  tool (**Sub – Marking** flyout) zooms directly into an area at the arrow-head by selecting the arrow.



Figure 154: **Sub – Marking** flyout

Changing representation type

The representation types of the Advance elements are changed on the **Representation type** tab of the dialog boxes of the single objects.

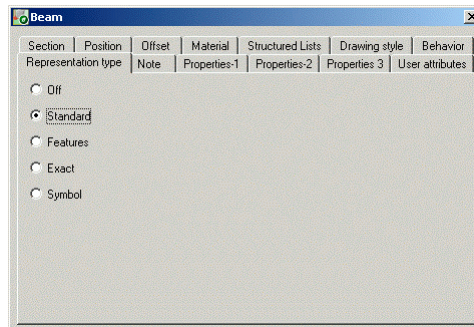


Figure 155: “Beam” dialog box – **Representation type** tab

More tools for representation of objects are located on the Advance **Standard** toolbar. These tools, in contrast to the dialog boxes, change the individual representation types faster and modify different object types (e.g., beams and plates) at the same time.



Figure 156: **Standard** toolbar - Representation type

Button	Function
	“Change representation type” displays the individual representation types. For beams, the representation types are “standard”, “features”, “exact” and “symbol” (refer to the beam example in the <i>Representation types – snaps – grips</i> section of this chapter)
	“Standard representation” directly sets the representation type to “standard” irrespective of the current representation type.

- An additional Flyout on the toolbar has two more tools.

Button	Function
	“Selected objects off” hides objects on the screen representation
	“Selected objects on” displays hidden objects

If an object was hidden on the screen representation it can only be selected using the object filter and then displayed with the “Selected objects on” command.

 **Example:** *Selecting and displaying a hidden beam*

- Filter the database with the “Filter database”  tool on the Advance **Standard** toolbar.
- Next, on the **Standard** toolbar, **Sub – Objects** flyout, select “Beam”  as the selection object. All beams including those that are hidden are selected.
- Selecting the “Representation objects on”  tool on the **Sub – Presentation** flyout displays all the beams.

Views on the model

Any part view of the model is set with any view direction using the camera. The plane view on the model is created instantly, including zoom settings, and is used as a construction aid.

The “Camera” tool is located on the **View** toolbar.



Figure 157: **View** toolbar

Define the view with the camera position and a target that provides the view direction.

Alternatively, the standard AutoCAD® view directions from the same toolbar may be used. It should be noted that these tools create views on the entire depth on the model and are not clipped to a specific plane. For more advanced view creation, refer to the **Working methods II** chapter.

Structural elements

Structural elements are grouped objects that consist of basic objects such as beams and other associated features such as shortenings. These are held together and represented by a white structural frame shown as a continuous line.

Structural elements of different shapes are on the **Sub – Structural Elements** flyout of the Advance **Create objects** toolbar.

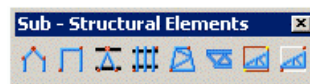


Figure 158: Advance **Sub – Structural Elements** toolbar

- Symmetrical flat frames and portal frames
- Single-span bracings
- Purlins
- Pylons
- Joists

The basic objects, from which structural elements are created, have a Model Role within the structure that determines their behavior. Basic objects with identical Model Roles are changed together (e.g., if a purlin size is changed, all purlins in the structural element will change to the same size). Exploding the structural element allows changes to individual members.

The creation dialog box is specific to the structural element type. For structural elements there is only one representation type; they differ in the type and number of the snaps and grips, which are used to change the geometry (e.g., height, width, etc.).

Structural elements are created in the X/Y-plane of the current user coordinate system hence the first step for creating a structural element is to set a correct UCS.

The gray structural frame bounding box calls the properties dialog box. If beams of a structural element are to be changed, the corresponding beams must be selected.

The “Select” tool on the **Sub – Joint utilities** flyout looks for all elements belonging to a structural frame or connection objects to copy, rotate, or mirror individual structural elements.

Some of the structural elements use the connection object technology: joists, purlins, and portal frames. The connection object is represented as a gray box (3D bounding box).

Portal frames

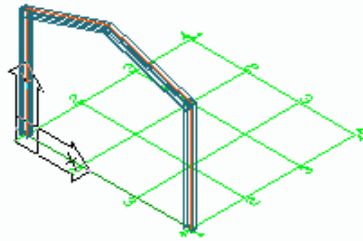


Figure 159: Portal frame, columns centered – no cuts

A portal frame consists of two columns and two rafters. The four elements are created with a single command. Of the four elements, two beams take the **model role** of column and the two beams take the **model role** of rafter. Elements with the same **model role** form a group. It is not possible to directly change the properties of individual columns or rafters.

The portal frame is created on the current UCS plane by entering two base points and one ridge point (or angle), or simply by right clicking to accept the default height.

To modify portal frame properties, select the object's (column or rafter) **Advance Joint Properties** from the context menu. The **Layout** tab in the "Portal frame" dialog box has the width, height, total height and rafter angle from horizontal. These properties are also changed in the "Structural element" dialog box.

There are two portal frame types available: frame and gable frame. The number of columns of a gable frame is modified in the portal frame properties dialog box.

Flat frames

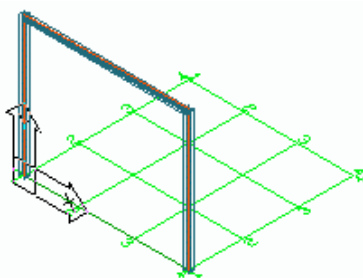


Figure 160: 2-hinged frame

This structural element is created on the current UCS plane by entering two base points and one ridge point for the height, or by right clicking to accept the default height.

Flat frames are available in 3 **types**. A frame is created with a horizontal tie beam. The columns can be shifted outwards or inwards.

Single-span Bracings

Diagonal members in the current X/Y-plane are created while entering a base point and a diagonal point.

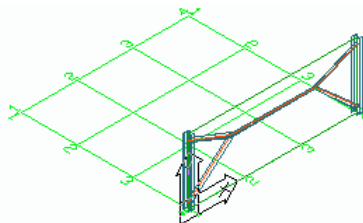


Figure 161: Created single-span bracing

There are 3 bracing types: Created bracings, crossed bracings with or without angles in the same or opposite direction, K-bracings and crossed flats. The bracings can have varying distances, above and below the selected points, in the dialog box.

Purlins

This creates a set of regularly spaced members on the selected rafters. Simply select the rafters and the purlins are automatically created. One option allows the selection of a column to position an optional eaves beam.

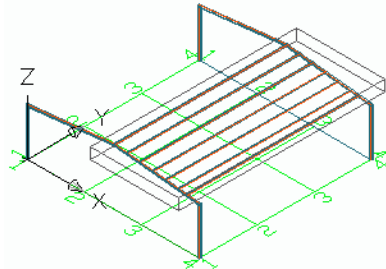


Figure 162: Purlins

The purlin properties are changed on the **Purlins** tab in the properties dialog box: purlin section, the projections at first and last rafter, the extra purlin length, and the span type. The distances are modified on the **Distances** tab. The intermediate distance can be changed and the number of purlins is modified accordingly.

An eaves beam can be added and the section type, position, offset and orientation can be selected.

Pylons

This tool creates a structural frame that contains a pylon bracing. A pylon bracing can include a symmetrical taper. To define the structural element, first enter a base point at the first strut, a point for the width, and then a third point for the height and inclination of the strut.

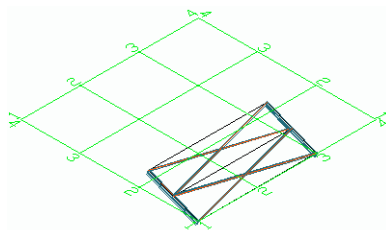


Figure 163: Pylon bracing

The fields for height and number are set in the dialog box.

Joists

To create a double angle joist, enter a start point and an end point to define the length. The third point determines the height and orientation of the joist.

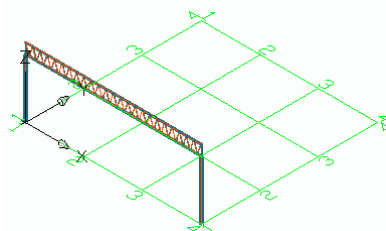


Figure 164: Joist

Once created, the length or height is modified by using the grip points of the box or by changing the properties in the properties dialog box.

Half trusses

To create a half truss, select a first base point where both sections converge. The second point determines the width and the third point is the height of the truss.

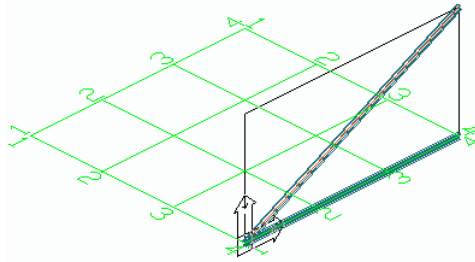


Figure 165: Half truss

Distance above and below, intermediate height and width are changed in the dialog box.

Joints and connection objects

Advance contains a variety of preset parametric joints to connect Advance elements. The joints consist of several elements such as plates, stiffeners, beams, bolts, and welds. An Advance joint is created with all its parts, connection objects, and features using a single operation. At the same time, the existing connected elements are processed (shortened, coped, etc.) and connection logic between the parts is established.

A joint is represented in the model by a gray box connection object that contains all the connection parts.

Joints and connection objects are grouped on the Advance **Joints** toolbar, which is accessible via the **Advance Steel** toolbar.

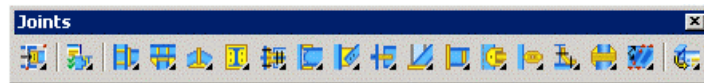


Figure 166: Joints toolbar

The tools for *connection objects* are arranged on the **Sub – Joint Utilities** flyout of the Advance **Joints** toolbar.



Figure 167: Sub – Joint utilities toolbar

The joints may be accessed using the flyouts. The joints on the whole, are classified according to the type of the individual members:

- Beam to column connections (flyout)
- Cantilever beam to column connections (flyout)
- Beam end to end connection (flyout)
- Plate connections (flyout)
- Clip angle connections (flyout)
- Moment connections (flyout)
- General bracing connection (flyout)
- HSS and flat bracing connection (flyout)
- Base plate connections (flyout)
- Tube brace connections (flyout)
- Turnbuckle bracing (flyout)

- Krupp purlin system (flyout)
- Pylon connections (flyout)
- Punch marks connections (flyout)

On the alternative **UK connections** toolbar the following additional flyouts are available:

- UK Hot Rolled connections
- UK Cold Rolled connections

The connections are intelligent: If a beam connected using a connection is changed the connection automatically adapts to the new situation according to the set parameters.

These set parameters are stored in the **connection object** (the gray box) that is hidden (by default) when the connection is created. It contains all information of the joint but not individual component properties.

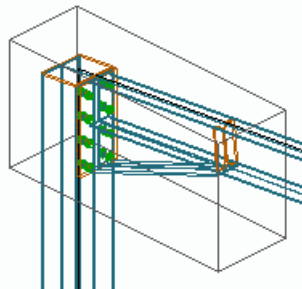


Figure 168: Connection object box

To modify a joint, double click the **connection object** (gray box) that calls the properties dialog box. Connection objects can be displayed, updated, entirely deleted, copied and changed. Their parameters can be transferred to other existing joints. Objects of the connection can be selected.

Joints

A dialog box appears when a connection is created.

After clicking on a connection, select the members to connect by pressing **Enter** after each selection.

Example:

... passing main beam ... Enter ... beam to be connected ... Enter.

Unless a special entry for existing beams exists in the connection library, a warning window will appear (Figure 169).

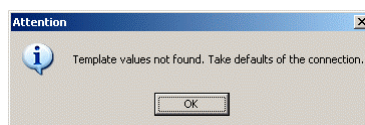
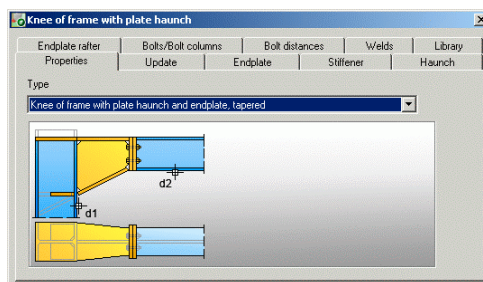


Figure 169: Warning window for joints

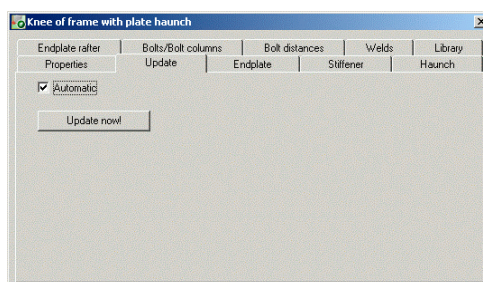
The joint is created and a dialog box with graphic explanations of the available parameters appears.

The tabs in the **Joint** dialog boxes vary according to the joint type. There are common tabs such as **Properties**, **Update**, **Table** and **Bolts** in almost all dialog boxes.

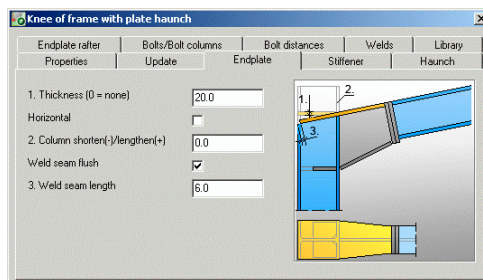
On the **Properties** tab, the **Type** pull down connections can be selected from one or more types (e.g., "Knee of frame with end plate and plate haunch tapered" or "straight"). The graphic representation changes according to the selected type.

Figure 170: "Knee of frame with plate haunch" dialog box - **Type**

On the **Update** tab, included in each joint dialog box, there is a checkbox for the automatic updating of a joint if individual beams are modified and there is a button for manual update.

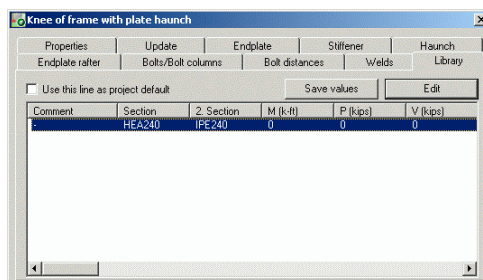
Figure 171: "Knee of frame with plate haunch" dialog box - **Update**

On the other tabs (e.g., for plates, bolts, bolt distances, etc.) the numbers in the sketches correspond to the information entry boxes.

Figure 172: "Knee of frame with plate haunch" dialog box - **Endplate**

The settings in the connection dialog box can be saved to a table. Company-specific connection settings for certain beam combinations can be saved and reused in other projects.

- To save a connection to the table, make appropriate adjustments in the dialog box tabs.
- Click the **Library** tab and click **Save values**. Next, click **Edit** to open the table. Enter a comment to identify the connection and fill in the appropriate section sizes.
- Click **OK** to confirm and return to the connection dialog box.

Figure 173: "Knee of frame with plate haunch" dialog box - **Table**

If the connected members correspond to an entry in the table, the connection is created using the entries in the table (library). If there are no default values in the tables (libraries), a message is displayed as previously described above.

Beam to column joints

Joints that connect a tie beam to a column are available on the **Sub – Beam to column** toolbar (flyout) of the **Joints** toolbar. The tie beam ends at the column. Bolted and welded joints, with or without haunches are available.



Figure 174: Sub – Beam to Column toolbar

 **Example:** Creating a knee of frame with plate haunch and end plate

- On the **Joints** toolbar, **Sub – Beam to Column** flyout, click .
- Select the column, press **Enter**, select the tie beam, and press **Enter**.
- The connection with haunch is created at the flange of the column and the beam to be connected is modified. The tie beam is connected at the flange of a column with a plate haunch. The haunch is connected at the tie beam by end plates. The **Properties** dialog box appears.
- Under “Type”, a straight or tapered haunch can be selected. The other tabs have options to set various graphically represented haunch parameters.

The “Bolted knee of frame” connection has different stiffener types. Different designs are possible with or without a cover plate. When designing a cover plate the column length is automatically modified.

The “Knee of frame bolted with haunch” tool creates a beam connection with haunch at the column flange. The haunch is created from a beam cut or from plates. Different stiffener types are available. The haunch is designed with or without a cover plate and the column is modified in length accordingly. Up to four different bolt groups can be set. Each of these bolt groups can contain several horizontal bolt lines.

The “Knee of frame at web, bolted” tool connects a tie beam with an end plate to a column web. This connection is available “with one haunch” or “with two haunches”.

A knee of frame with two haunches is created using the “Knee of frame, bolted with 2 haunches” tool on the **Sub – Beam to Column** flyout.

The “Knee of frame” or the “T-connection” can be selected as **welded** joints.

Moment joints

The moment connections are included on the **Sub – Moment joints** flyout.

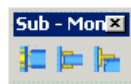


Figure 175: Sub – Moment joints toolbar

The “Moment end plate” connection is used to connect a tie beam to a column as well as two floor beams. The moment connection creates additional support (e.g., welds, plates, stiffeners) to the connection.

The “Moment flange” joint is created out of flange plates. The connection works on all possible combinations of sections. Also it works for sloped and skewed situations.

The “Moment flange T” connection is created out of T-profiles and also works on all possible combinations of sections.

Joints cantilever beam to column

Gable columns/posts are connected to rafters/beams using tools on the **Sub – Cantilever beam to column** toolbar (flyout) of the **Joints** toolbar. The orientation of the columns is parallel or perpendicular to the rafters. The connection is created using an end plate. Pin-ended columns are connected to fin plates welded to the rafter. The connection can also be made using an end plate welded to a fin plate.



Figure 176: Sub – Cantilever beam to column toolbar

Example: Pin-Ended Column, by Distance

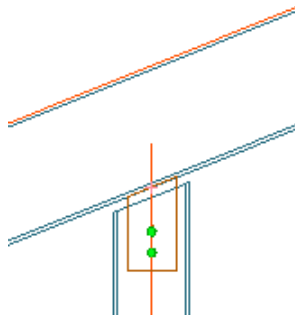


Figure 177: Pin-Ended Column, by Distance

- On the **Joints** toolbar, **Sub – Cantilever beam to column** flyout, click .
- Select the tie beam, press **Enter**, select the column, and press **Enter**.
- The pin-ended connection is created and a dialog box appears.

A pin-ended column is connected to a rafter using a simple fin plate (**Properties** tab, type: *simple joint*) welded to the tie beam or using a fin plate and end plate (**Properties** tab, type: *joint with cover plate*). Information for **plate** and **bolts** can be modified.

For a joint with an end plate, the end plate is modified in width to suit the rafter and the bolts are automatically created on the rafter gauge lines.

The **distance** between the column and the rafter (perpendicular to the rafter) can be set and the length of column is changed accordingly.

The “Pin-ended column, constant length” must be modified by its system length in advance since the fin plate is created at the end of the column system axis.

With “Gable wall, web parallel”, the joint connection is either symmetrical or asymmetrical. For an asymmetrical connection, a bolt group is located outside the column. The bolts in the rafter are automatically placed on the gauge lines.

With “Gable wall, web parallel”, stiffeners are created into the rafter running perpendicular to the rafter or parallel to the column.

Joints beam end to end

The joints on the **Sub – Beam end to end joints** toolbar (flyout) connect spliced beams for a bolted apex with or without haunches, an end plate splice, and a splice joint. Changing the parameters customizes these joints.



Figure 178: Sub - Beam end to end joints toolbar

With “Apex bolted”, two beams are spliced with bolted end plates. Up to four different bolt groups can be set.

With “Apex bolted” with **one** or **two** haunches, additional information for haunches and stiffeners can be entered.

With “Front plate splice, variable”, four separate bolt patterns are created and the plates are connected to the beams with two weld points.

With “Middle gable column”, two rafters are connected to a column using two end plates. The end plates are bolted to the rafters and welded to the double cut column.

With “Front plate splice”, two beams are spliced with end plates. It is possible to attach additional bolt lines above or beneath.

The “Front plate splice welded, flush, vertical” adds end plates flush to the beams and welds them together. The **plate thickness** is specified in the dialog box. The intersection at the beam is set to either vertical (perpendicular) or at the bisecting angle on the **Type** tab.

A “splice joint” can be created.

The “Column splice” connects two horizontal, vertical or sloped columns. It creates the moment cuts and backing bar as well as a flange gusset plate.

The “Purlin splice” connects purlins with a C-shaped profile or flat steel. Section, splice plate, side and column are specified on the **Splice plate** tab.

Clip angle joints

The tools on the **Sub – Clip angle joints** toolbar (flyout) of the **Joints** toolbar connect floor beams to beams as well as a beam to a column by clip angles.



Figure 179: **Sub – Clip angle joints** toolbar

 **Example:** Creating a clip angle connection

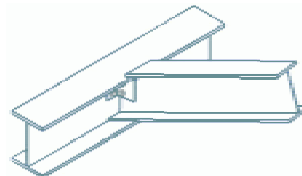


Figure 180: Clip angle connection

- On the **Joints** toolbar, **Sub – Clip angle joints** flyout, click .
- Select the passing main beam and press **Enter**.
- Select the beam to connect and press **Enter**.
- The joint is created and a dialog box appears.

The “Clip angle” connects a floor beam to another beam as well as a column to a beam. The attached beam can be sloped to the main one.

“Clip angle skewed” creates a similar connection with a folded clip angle. The attached beam can be sloped and skewed to the main one.

“Double side clip angle” connects a floor beam and two beams as well as a column and two beams. The attached beams can be sloped to the main one, as the clip angle is aligned at the main beam. The connection works also for a curved passing beam or when the beams are compound or welded.

With “Double-sided web joint, angle-end plate”, opposite beams are connected to a main beam. The first beam is connected to a double angle and the second one to an end plate.

The “Seated beam shear connection” uses welded or bolted **angles** to join two beams. The quantity and size of angles are variable.

The “Joist seat bottom” joint can be created on beams or columns of all possible shapes.

Plate joints

The tools on the **Sub – Plate joints** toolbar (flyout) of the **Joints** toolbar connect beams to beams. End plate and moment end plate connections are also included.



Figure 181: **Sub – Plate joints** toolbar

 **Example:** *Creating a double side end plate with safety bolts*

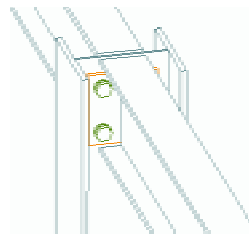


Figure 182: Double-sided end plate connection

- On the **Joints** toolbar, **Sub – Plate joints** flyout, click .
- Select the passing main beam and press **Enter**. Select the first beam to connect and press **Enter**. Select the second beam to connect.

Two opposite beams are connected through a main beam by their end plates. If the connection is used between floor beams then a cope is automatically created by default.

If the connection is used between a column (web) and two beams and the beams do not **fit** in the space between the column flanges, then a trim is created on the left and right side of the flange (secondary beam).

The cope parameters are set on four tabs of the joint properties dialog box. The plate size is calculated using the bolts vertical and horizontal distances. The cope length at the top and bottom and the cope depth can be set for both connected beams. The thickness of the weld line can be set separately for each beam.

The “Single side end plate” connects two floor beams or a column to a beam. The secondary beam can be sloped, skewed, or rotated.

With “Moment end plate connection”, projections are set at the top or bottom of the joint. Creation of a moment end plate connection is used to connect a tie beam to a column. The projections can be specified at the top or bottom in joints that support this feature.

The “Double side end plate with safety bolts” connects two beams to a floor beam or to a column. The attached beams can be sloped to the main one and a safety bolt can be added.

The “Platform splices” (toe plates) are shear connections at a beam with end plates and stiffeners. In the extended version up to four bolt groups can be set.

The “End plates, asymmetric” connects beams that do not run into the center of the main member (a column or a beam).

The “Shear plate” connects two floor beams or one column and a floor beam.

The “Through plate” joint is similar to a shear plate except it connects two secondary beams to a column by a plate. The plate goes through the column (main beam) and automatically creates a contour in the main beam. It works for horizontal or sloped/skewed beams.

The “End plate” connection welds the end plate to the tie beam and bolts it to the column or floor beam. The end plate can be created central or flush relative to the upper edge of the tie beam.

General bracing joints

The **Sub – General bracing joints** toolbar (flyout) of the **Joints** toolbar groups tools for connecting bracing members using gusset plates.



Figure 183: Sub – General bracing joints toolbar

The gusset plate connection is used on a column with a base plate or on other types of beams. The necessary bolts and welds are automatically created.

Example: Gusset plate a

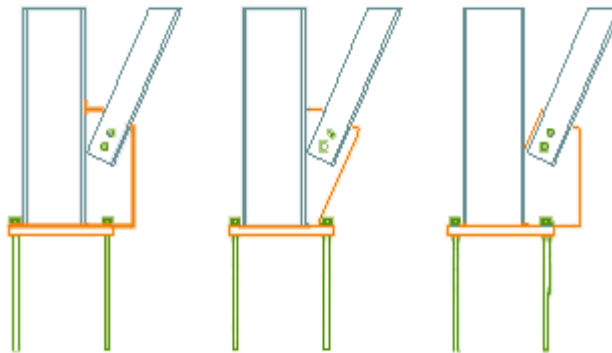


Figure 184: 3 Gusset plate types

- On the **Joints** toolbar, **Sub – General bracing joints** flyout, click .
- Select the column to connect and press **Enter**.
- Select the base plate to connect and press **Enter**.
- Select the diagonal to connect and press **Enter**.
- The gusset plate is created and a dialog box appears.

“Gusset plate to column and base plate” connects a column and a base plate to a diagonal. It can be used on the column web or flange.

“Gusset plate, one diagonal” joins a single diagonal.

Shortening the diagonals and modifying distances from the main beams can connect **two, three or four diagonals**.

A “Gusset plate” in the center can also connect two diagonals. In this case, one of the diagonals is continuous and the two others are joined at the center point.

A diagonal can be connected to a beam by **gusset plate** and **two angles**. Distances, trimming of the diagonal, and a variety of other specifications can be entered in the “Angle bracing” dialog box.

The “Diagonal bracing” connects a column, a longitudinal beam and a diagonal brace. Slotted butt straps are bolted to the diagonal and welded to a gusset plate. The gusset plate is welded to the column and to the longitudinal beam. The longitudinal beam is connected to the column using an end plate.

The “Angle bracing with plate” creates and connects four bracing L-profiles to a plate. The bracing profiles are bolted to the plate.

HSS and flat bracing joints

The **Sub – HSS and Flat bracing joints** toolbar (flyout) of the **Joints** toolbar groups tools for connecting bracing members.



Figure 185: **Sub – HSS and Flat bracing joints** toolbar

The gusset plate connection is used on a column with a base plate or on other types of beams. The necessary bolts and welds are automatically created.

 **Example:** *Creating an HSS bracing middle connection with sandwich plates*

- On the **Joints** toolbar, **Sub – HSS and Flat bracing joints** flyout, click .
- Select the main beam to connect and press **Enter**.
- Select the first diagonal and press **Enter**.
- Select the second diagonal and press **Enter**.
- The joint is created and a dialog box appears.

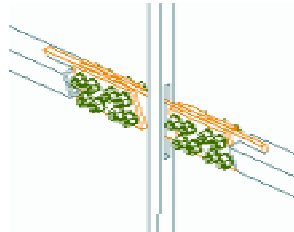


Figure 186: HSS bracing middle, with sandwich plates

“HSS bracings” connect square, rectangular, or round tubes to beams with an **angle**, a **shear plate**, or a **folded cleat**.

The “HSS bracing middle with sandwich plates” connects one main beam and two HSS diagonals.

The “HSS bracing with sandwich plates” connects one main beam and a diagonal.

The “HSS bracing with sandwich plates - additional object” connects a column, a base plate (or a beam), and a diagonal.

The “HSS bracing middle with sandwich plates” connects one main beam and two HSS diagonals.

The “Flat bracing” connects a main beam and a flat diagonal using a plate.

Stiffener joints

The **Sub – Stiffener joints** toolbar contains tools to create stiffeners and binding plates. These are created either on the current user coordinate system or independently.



Figure 187: **Sub – Stiffener joints** toolbar

“End plates” are attached to the system end point of beams. The beam can be shortened by the plate thickness and a construction gap can be created (“End plate variable” dialog box, **Plate** tab).

The “Stiffeners, perpendicular” tool creates stiffeners perpendicular to a beam at a picked point according to the **UCS**. The “Stiffener, UCS” tool places stiffeners in the plane of the coordinate system with the stiffener thickness in the positive Z-direction of the current UCS. The distance between the full stiffener and flange inner sides can be set in both cases.

Three tools are available for creating connecting plates between two beams. With the “Binding plate from point” tool, enter a start point to create the first plate and then the intermediate distance between each plate. With “Binding plate between two points”, enter a start and an end point and plates with set distances are created. With “Binding plates, end distance”, select the beams and the distance to the first plate. The distances between plates are set on the **Pattern** tab of the **Properties** dialog box.

The “Stiffener, hollow section” tool creates a stiffener plate in a square or rectangular tube. This joint also works on welded hollow beams.

 **Example:** Creating a binding plate between two beams with start and end points

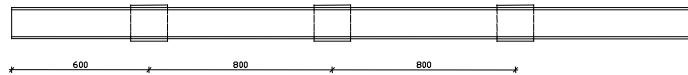


Figure 188: Binding plate between 2 points

- On the **Joints** toolbar, **Sub – Stiffener joints** flyout, click .
- Select the first beam and press **Enter**. Next, select the second beam and press **Enter**.
- Enter the start point and end point.
- The binding plates are created with standard values and the “Binding plate between 2 points” dialog box appears.
- On the **Plate** tab, specify the intermediate distance, number, and size of the binding plates. The distance between the beams will determine the thickness.
- The binding plates are welded to the beams that connect them. On the **Weld line** tab, enter the specification of the weld.

Base plate joints

The **Sub – Base plate joints** toolbar (flyout) of the **Joints** toolbar has base plate connection elements and simple plates.



Figure 189: Sub – Base plate joints flyout

 **Example:** Creating an asymmetrical base plate

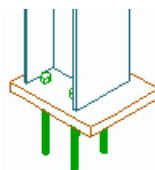


Figure 190: Joints – asymmetrical base plate

- On the **Joints** toolbar, **Sub – Base plate joints** flyout, click .
- Select the column to which the base plate should be attached and press **Enter**.
- If connection settings have not been saved a warning appears (*default values for the section cannot be found, take the standard value of the joint.*). Confirm by clicking **OK**.

A base plate with anchor bolts is created at the end of the column. The column is shortened by the plate thickness and the plate is welded to the column. Different options for bolts and plate preferences are available.

The **Type** field has selections for base plate with corresponding bolt pattern, bolt pattern in the center of the beam, and symmetrical bolt pattern.

Other graphically represented parameters are modified on the **Base plate**, **Anchor bolts**, and **Bolt distances** tabs.

“Horizontal base plates” are created at raking columns. The base plate is either horizontal to the base or perpendicular to the column. There are two kinds of horizontal base plates available: symmetric and asymmetric.

Corner column connections are created with “Corner base plate” joints. The base plate is an L shape and bolts are created at different locations. The base plate can be created on all profile shapes (e.g., I, U, C, square hollow, rectangular hollow, round pipes) and on combined sections such as star angles and double channels.

The “Base plate, twisted” tool creates a plate at the column using the position of the current user coordinate system. The underside of the base plate is positioned on the UCS plane and the column trimmed to the top of the base plate (‘top’ refers to the +Z direction of the current UCS). The base plate can be rotated around its center point.

With “Base plate with shear section and stiffener”, a variety of parameters can be specified to determine the thrust anchor, stiffeners, and welds. The graphically illustrated tabs in the dialog box assist in setting the parameters.

The “Base plate with traverse” tool creates a base plate, which can be stretched at the end of a column. Cross members (of a C-shaped profile) are bolted or welded to the flange of the column.

Tube brace joints

The tools on the **Sub – Tube brace joints** toolbar (flyout) of the **Joints** toolbar connect a compression pipe to a beam. The dimensions of the elements are set using parameters.



Figure 191: **Sub – Tube brace joints** toolbar

To create the joint, select the beam and press **Enter**. Next, select the compression pipe to connect.

 **Example:** Creating a compression pipe with end plate, double-sided

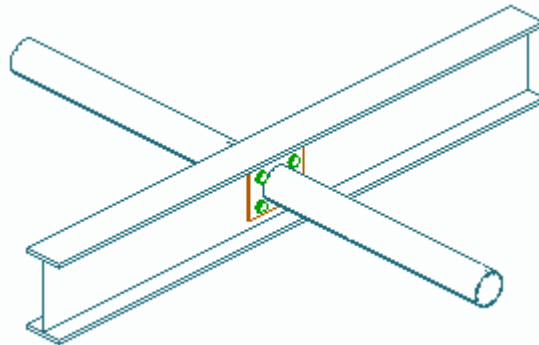


Figure 192: Compression pipe with end plate, double-sided

- On the **Joints** toolbar, **Sub – Tube brace joints** flyout, click .
- Select the beam and press **Enter**. Next select the tube and press **Enter**.
- The connection is created and a dialog box appears.

The “Compression pipe with gusset plate and butt strap” tool is available in three variants. The variants are compression pipe with slotted butt strap and gusset plate, compression pipe with slotted pipe and gusset plate, and compression pipe with slipped-in butt strap and gusset plate.

With the “Compression pipe with welded T-section” tool, a gusset plate is welded to a beam and a T-section is welded to the compression pipe. The T-section and the gusset plate are bolted to each other and the compression pipe is adjusted in length.

Other tools on the toolbar are: Compression pipe with covering plate and gusset plate, compression pipe with slotted butt strap, compression pipe with slotted pipe, compression pipe with slipped-in butt strap, compression pipe with welded T-sections, compression pipe with end plate, compression pipe with end plates, double-sided, compression pipe diagonal, and tube to plate connection. Refer to the Advance online Help for detailed instructions.

Turnbuckle bracings

The **Sub – Turnbuckle bracings** toolbar (flyout) of the **Joints** toolbar groups single bracing members and complete cross bracings of round or flat steel. The bracings are welded to beams and/or plates.

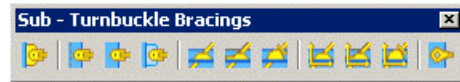


Figure 193: **Sub – Turnbuckle bracings** toolbar

 **Example:** Turnbuckle bracing with steel spherical cap

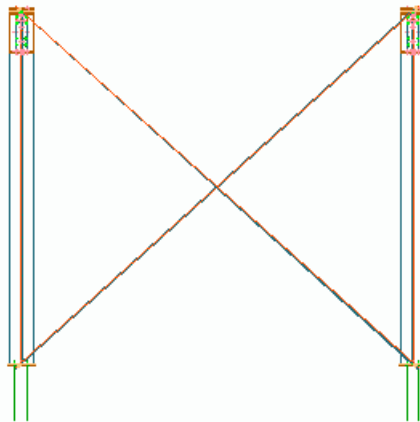


Figure 194: Turnbuckle bracing with steel spherical cap

- On the **Joints** toolbar, **Sub – Turnbuckle bracings** flyout, click .
- Select the first column and press **Enter**.
- Select the second column and press **Enter**.
- Select the beginning of the system line of the 1st column (point 1).
- Select the end of the system line of the 2nd column (point 2).
- Select the beginning of the system line of the 2nd column (point 3).
- And finally, select the end of the system line of the 1st column (point 4).

The turnbuckle bracing is created. Individual settings such as moving the system lines in the Z-direction to set the position of the bracing within the column are modified in the dialog box. These settings can be made at the top and bottom of the bracings. Variants for this bracing include “Bracing with tensioning trestle” and “Bracing with plate bracket and pad”. In a similar way, single bracings with a cast spherical cap, tensioning trestle or plate bracket and pad can be created.

“Turnbuckle bracing with gusset plate” creates a bracing member with turnbuckle, a welded butt strap, and a gusset plate welded to a beam. Both beams must be selected and then the start and end points of the bracing member must be defined.

“Turnbuckle bracing at two beams” creates a bracing member with turnbuckle and a welded butt strap bolted to a beam. The bracing is placed on the flange of the beams.

To create a “Turnbuckle bracing at two plates”, select the plate and set the start and end points of the bracing member. The bracing is placed on the flange of the beams.

"Turnbuckle bracing at beam – plate" combines both "Turnbuckle bracing at two beams" and "Turnbuckle bracing at two plates" tools. The start point of the bracing is connected to a beam and the end point to a plate. The bracing is placed on the flange of the beam.

"Bracing with bar armature" creates a bracing member with a cast steel-butt strap (armature) and a butt strap bolted to the beam. Select both beams and define the start and end points of the bracing.

Krupp purlin system

With the Z-purlins by Krupp (section class Krupp Section Purlins ZG), various joints are created at the upper flanges of beams using the **Sub – Krupp purlin system** toolbar (flyout) of the **Joints** toolbar. The toolbar includes connections for a single-span purlin at an end-span and at a splice and the attachment of a continuous purlin to an upper flange.



Figure 195: **Sub – Krupp purlin system** toolbar

To create purlin connections, first select the beam to which the purlin is attached and press **Enter**. Then select one or two purlins (according to the tool used), and press **Enter** each time.

 **Example:** Creating single-span purlins, end-span

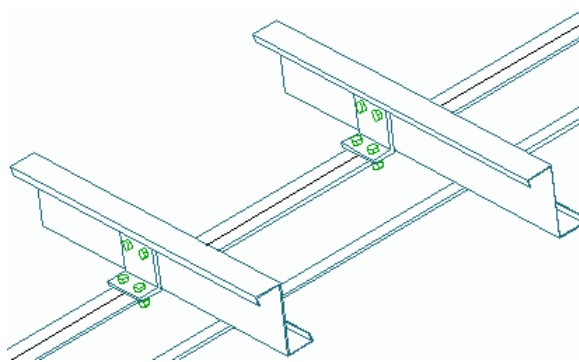


Figure 196: Single-span purlin, end-span

- On the **Joints** toolbar, **Sub – Krupp purlin system** flyout, click .
- Select the beam and press **Enter**.
- Select the purlin and press **Enter**.
- Click **OK** in the message window.

A cleat element is bolted or welded to the main member and the purlin is then bolted to it. In the "Krupp purlin splice" dialog box, enter more details about the connection (i.e., bolted or welded connection, construction offset, length of sleeve, information for bolts, etc.).

With the "Single-span purlin, splice" tool, two single-span purlins are spliced above an upper flange.

A "Continuous purlin" can be attached to an upper flange. Either a connection at an end-span or a continuous connection can be selected.

"Continuous purlin with connection element" creates a purlin splice on the upper flange. A connection with a sleeve or with a butt strap can be selected.

Joints pylon construction

The **Sub – Pylon joints** tool of the **Joints** toolbar contains joints for pylon construction.



Figure 197: **Sub – Pylon joints** toolbar

The connection between two tubes (collinear or not) is made using flange plates, the cross connection of two tubes with straps, and the connection of a tubular diagonal with a bracket.

To create a pylon construction connection, select the first tube (passing), and then select the second (connecting) tube.

 **Example:** Connecting two tube ends with flanged connection

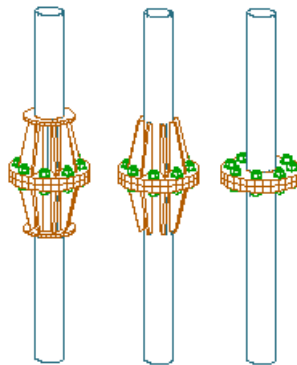


Figure 198: Tube ends with flanged connection

- On the **Joints** toolbar, **Sub – Pylon joints** flyout, click .
- Select the first pipe and press **Enter**.
- Next, select the second pipe and press **Enter**.
- In the message window, click **OK**.

The pipe connection is created and the “Pipe pylon splice with flange plates” dialog box appears.

In the dialog box, information for bolts, flange plates, stiffeners, ring plate 1, and ring plate 2 can be entered.

“Unparallel tube joint” is self-explanatory.

“Joint with straps” connects two tubes using an end plate with straps and optional stiffeners.

“Joint for base point” connects tubes using a cover plate, butt strap, and gusset plate. The incoming tube must be square to the first tube.

User connections

The tools on the **Sub – User template connections** toolbar create, store, and reuse interactive connections.



Figure 199: **Sub – User template connections** toolbar

“Create template connection” stores interactive connections in a library.

Example: Creating a user connection

- Create all joint elements: plates, welds, bolts (including bolt holes), cuts, etc.
- Save the .dwg file containing the joint elements in the **Connection templates** folder.
- On the **Joints** toolbar, **Sub – User template connection** flyout, click .
- Enter the name for the new template connection: **MyConnection**.
- Select all joint elements and press **Enter**. A blue connection object appears around the joint location.

To recall a connection, use the “Import template connection” tool. The library dialog box has a visual preview of each stored connection for quick recognition.

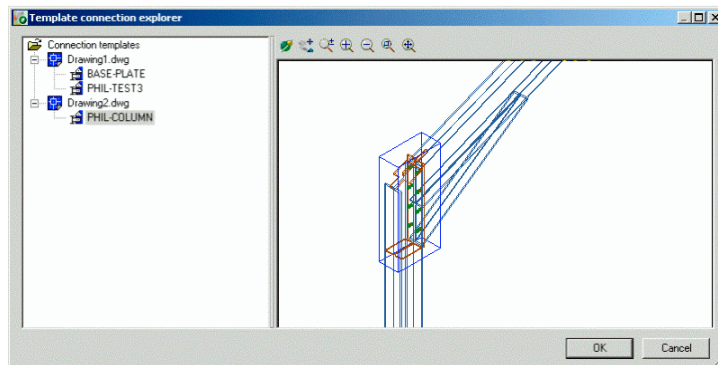


Figure 200: User connection explorer

Select a joint and connect the beams by selecting them in the proper order. Saved joints only work if the orientation and the angle between the beams is the same as when the joint was stored.

Punch mark joints

The **Sub – Punch marks** toolbar (flyout) of the **Joints** toolbar has tools for creating punch marks. The joint places punch marks on the main element (beam or column) according to the position of a secondary element (beam or column).



Figure 201: **Sub – Punch marks** toolbar

Example: Creating punch marks

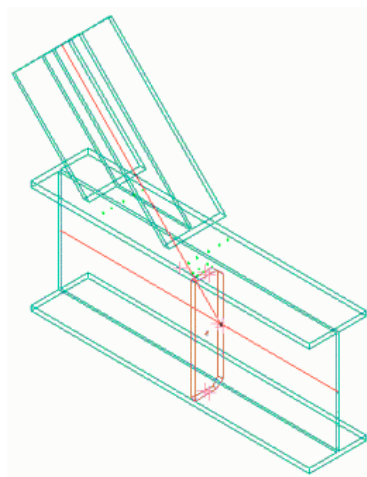


Figure 202: Punch marks

- On the **Joints** toolbar, **Sub – Punch marks** flyout, click “Punch marks” .
- Select the main element and press **Enter**.
- Next, select the second element and press **Enter**.
- The punch marks are created and the “Punch mark joint” dialog box appears.

UK Joints

The **UK joints** toolbar (optional) contains two additional flyouts: Hot Rolled Joints and Cold Rolled Joints.

The Hot Rolled Joints flyout contains the following connections:

- End Plate
 - Connects an incoming secondary member to a passing main member using an end plate. The members can be any profile shape and at any angle to one another. The parameters include options for cope chamfers and full control of the bolts and end plate positioning. There are also options for creating slotted holes. All parts are automatically bolted or welded as required.
- Double Sided End Plate
 - Same as End Plate for connecting two secondary members at each side of a main member web. Parameters are the same as those of the End Plate with additional options for different sizes and layout for the individual end plates.
- Single Beam on Post End Plate
 - Connects a beam over the top of a post using an end plate (cap plate) welded to the post and bolted to the bottom flange of the beam. The plate and bolt layout is fully controllable and the overhang of the beam can be modified.
- Double Beam on Post End Plate
 - Connects two abutting beams over the top of a post (similar to Single Beam on Post End Plate). The gap between the beams can be modified both in width and offset.
- Eaves Haunch
 - Connects a rafter / beam to a column using an end plate with a haunch. The members can be almost any profile shape and at any angle to one another in a plane. Various haunch options are available from plate and section cuts to no haunch at all. Various stiffener configurations can be added and, if required, the column top can be trimmed or beveled with various cap plate options. All bolt layouts, end plate, haunch, and stiffener sizes and positions are all fully controllable. All parts are automatically bolted or welded as required.
- Apex Haunch
 - Connects two rafters at the apex using end plates with haunches. The members can be of almost any profile shape and at any angle to one another in a plane (but not parallel – use end plate splice joint for parallel). Various haunch options are available from plate and section cuts to no haunch at all. All bolt layouts, haunch, and end plate sizes and positions are all fully controllable. There is an option to add a correction angle for presetting rafters to limit deflection. All parts are automatically bolted or welded as required.
- Fin Plate
 - Connects an incoming secondary member to a passing main member using a fin plate. The members can be any profile shape and at any angle to one another. The parameters have many options including cope chamfers, full control for the positioning of the bolts and fin plate, slotted holes, etc. The fin plate can be placed on either side of the incoming member web or even on either of its flanges. All parts are automatically bolted or welded as required.
- Angle Cleats
 - Connects an incoming secondary member to a passing main member using angle cleats. The members can be any profile shape and at any angle to one another (in a plane). The parameters have many options including cope chamfers and full control for the positioning of the bolts and angle cleats.

There are also options for creating slotted holes. The angle cleats can be placed on either side (or both sides) of the incoming member web or even on either of its flanges (or both). The angle cleats may be bolted or welded to either member.

- **Double Sided Angle Cleats**
 - Connects two secondary members at each side of a main member web (similar to Angle Cleats). Parameters are the same as those of the Angle Cleats with additional options for different sizes and layouts for the angle cleats on each side of the connection.
- **Base Plate**
 - Creates a base plate and anchors at the picked end of a member. The member can be any profile shape and at any angle (e.g., columns, beams, diagonals, etc.). The base plate can be set square to the member or horizontal or vertical to the WCS. The parameters allow for full control of the positioning and size of the bolts and base plate. All parts are automatically bolted or welded as required.
- **Single Tube Brace**
 - Connects an incoming secondary member to a passing main member using a gusset plate or bolted angle cleat on the main member and a tee (made from plate) on the secondary member. The members can be any profile shape and at any angle to one another. There are different rules for placing the bolts square to the brace member, square to the main member, or using a curved slotted hole for brace plates. The parameters allow for full control of the positioning and size of the bolts and plates. The orientation of the plates may be rotated 90 degrees. All parts are automatically bolted or welded as required.
- **Double Tube Brace**

Connects two secondary members to a common gusset plate on the main member (similar to Single Tube Brace). All bolt layouts and plate details can be set individually for each brace.
- **Triple Tube Brace**
 - Connects three secondary members to a common gusset plate on the main member (similar to Single Tube Brace). All bolt layouts and plate details can be set individually for each brace.
- **Gusset Plate – Single Member**
 - Connects an incoming secondary member to a passing main member using gusset plates. The members can be any profile shape (except tubes) and at any angle to one another (in a plane). The parameters allow for full control of the positioning and size of the bolts and plates. The gusset plates may be positioned on either or both flanges or on the web of the secondary member depending on its profile shape. The gusset plates may be bolted or welded to either member.
- **Gusset Plate – Double Member**
 - Connects two secondary members to common gusset plates on the main member (similar to Gusset Plate – Single Member).
- **Gusset Plate – Triple Member**
 - Connects three secondary members to common gusset plates on the main member (similar to Gusset Plate – Single Member).
- **Direct Bolting – End**
 - Connects an incoming secondary member to a passing main member by direct bolting. The members can be any profile shape (except tubes) and at any angle to one another (in a plane). The parameters allow for full control of the positioning of the bolts.
- **Direct Bolting – Cross**
 - Connects two crossing members (similar to Direct Bolting – End).
- **Lap Joint – Angle**
 - Connects two butted angles by forming a lap and bolts. The parameters allow for full control of the positioning of the bolts and defining the lap length.

- Lap Joint – Flat
 - Connects flats (similar to Lap Joint – Angle).
- Stiffeners – Perpendicular
 - Creates stiffeners perpendicular to the selected member at a given point. The member may be almost any shape. The parameters allow for full control of the stiffener size, shape, and position. Patterns of stiffeners along the member may also be created. All stiffeners are automatically welded as required.
- Stiffeners – Inclined
 - Similar to Stiffeners – Perpendicular except when creating the stiffeners where two points must be selected to determine the angle of the stiffeners. The angle can subsequently be changed through the parameters.
- Plain Cap Plate
 - Creates a plain cap plate at the selected end of a member, which may be of any profile shape. The cap plate may be set by plate size or by projections and can even be circular. The cap plate can be moved along the member and the member trimmed to suit. It is possible to rotate the cap plate to form a beveled end to the member.
- Fitting Joint
 - Connects an incoming secondary member to a passing main member using welds. The members can be any profile shape and at any angle to one another. The incoming member is automatically 'fitted' to the exact profile of the main member or can alternatively be trimmed to its face. A gap may be created between the fitted faces of the two members.
- Seating Cleat
 - Connects an incoming secondary member to a passing main member using a seating (supporting) cleat. The members can be almost any profile shape and, if the secondary member is passing (and not intersecting), can be at any angle to one another. The cleat may be bolted or welded to the main member but is not connected to the secondary member. The secondary member is not trimmed, which allows this connection to be used in conjunction with other connections (e.g., end plate).
- Cranked Beam
 - Connects two abutting members using a welded mitered joint and can optionally include a center plate. The members may be any profile shape and at any angle to one another. The center plate can be set by plate size or by projections. If a center plate is not required, set the thickness to zero.
- End Plate Splice
 - Connects two abutting members using end plates. The members may be any profile shape but they must be parallel to one another (for members at an angle, use the apex haunch connection). The parameters allow for full control of the size and position of the bolts, plates, and welds.
- Splice Plates
 - Connects two abutting members using splice plates. The members may be almost any profile shape but they must be parallel to one another. If the members are of different sizes (but approximately the same), the connection automatically creates packing plates but these can be switched off. The web plates, inner flange plates, or outer flange plates may be omitted by setting the thickness to zero. The splice plates can be bolted or welded to either member and there are also options for slotted holes. The parameters allow for full control of the size and position of the bolts, plates, and welds.

The Cold Rolled Joints flyout contains the following connections:

- Double Purlin
 - Connects two cold rolled purlins to a main member. The members can be at almost any angle to one another but the purlins must be parallel and must not intersect the main member. The purlins may also be rotated about their longitudinal axis relative to the main member. The purlin cleat can be from plate or tee welded to the main member or a standard supplier's cleat, which may be either bolted or welded to the main member. In either case, the purlins are bolted to the cleat and the bolt sizes and positions are according to the supplier's standard details. The purlin ends are also trimmed to the correct position. The joint includes options to add sleeves and/or stays and a 'fire wall condition' is also possible. If

a non-standard detail is set (e.g., a firewall condition) with a sleeve on a Metsec purlin, the sleeve is shown as a 10x10 bar called "unavailable" since Metsec cannot supply a firewall sleeve. For a purlin by a different supplier, this detail may be valid and, if so, the objects are created as normal. Any 'non-standard' details selected for a particular supplier result in the creation of an "unavailable" profile. All cold rolled components created using the joint will have the correct reference name for the particular supplier. The joint can be used for both purlins and side rails and for all standard section shapes for the named suppliers (i.e., Zeds, C-sections, Zeta and sigma sections, etc.).

- **Single Purlin**
 - Same as the Double Purlin except it is for a single purlin. By default, the purlin is not trimmed thus enabling the joint to be used on a continuous (multi span) purlin. For an end condition, the purlin may be trimmed or extended to give a specified overhang or a standard bolt end distance. Alternatively, a purlin stub may also be created.
- **Non-Continuous Purlin**
 - Connects a non-continuous purlin to a main member. A non-continuous purlin is set as one that intersects the main member within part or all of its depth. The purlin cleat can be either a plate welded to the main member or an angle cleat, which can be bolted or welded to the main member. In the case of a plate, it is shaped to fit the main member either into the web or around the flange. With an angle cleat, it can be placed either on the web face, the flange face, or across the flange toes of the main member. The purlin is bolted to the cleat and the bolt sizes and positions are according to the supplier's standard details. The purlin end is trimmed to the specified position and the joint also includes the option to add a stay.
- **Double Purlin Hip Joint**
 - Connects two purlins to a hip rafter using a center plate with wing plates. The purlins can be at any angle to the rafter and the macro can also be used for hip valley connections. The center plate can be either welded to the rafter or bolted using a base plate. The wing plates are welded to the center plate and bolted to the purlins using the supplier's standard details for bolt size and edge distances.
- **Single Purlin Hip Joint**
 - Same as Double Purlin Hip Joint except it connects a single purlin to a hip rafter. There is also a detail in which a single plate is shaped and welded directly to the rafter and bolted to the purlin.
- **Double Eaves Beam**
 - Connects two cold rolled eaves beams to a main member at any angle (the two eaves beams must be parallel). The eaves beam cleat can be from various combinations of plate and profiles, or a standard supplier's cleat, which may be either bolted or welded to the main member. The eaves beams are bolted to the cleat and the bolt sizes and positions are according to the supplier's standard details. The eaves beam ends are also trimmed to the correct position. All cold rolled components created by the joint will have the correct reference name for the particular supplier.
- **Single Eaves Beam**
 - Same as Double Eaves Beam except it is for a single eaves beam. By default, the eaves beam is not trimmed thus enabling the joint to be used on a continuous (multi span) eaves beam. For an end condition, the eaves beam may be trimmed or extended to give a specified overhang or a standard bolt end distance.
- **Mezzanine Floor**
 - Creates a layout of mezzanine floor beams in handed pairs as per various suppliers' standard details. Picking two diagonally opposite points sets the area of the floor. There is full control of the positioning of the beams and options for inset, over sail, and direct fix construction. Various options are also available for the inclusion of restraint ties of different profiles and positions.
- **Double Inset Floor Beam**
 - Connects two cold rolled floor beams to either side of a main beam, which may be almost any hot or cold rolled profile. The connection uses a standard supplier's cleat with the appropriate layout and bolt details. A few layout options are available within the constraints of the supplier's standard detail.

- Single Inset Floor Beam
 - Same as Double Inset Floor Beam except it connects a single cold rolled floor beam to one side of a main member. This connection also has an alternative rule for using a fin plate welded to the main member instead of a bolted supplier's standard cleat. This rule can also be used when the cold rolled floor beam is not perpendicular to the main member.
 - **IMPORTANT NOTE** – For certain profiles of some suppliers, a plate is NOT a supported detail as it can only transfer shear forces through the bolts (or friction) whereas the supplier's standard cleat may have special shaping to help transfer loads. In this case it is the user's responsibility to prove the design of the connection.
- Double Over sail Floor Beam
 - Connects two cold rolled floor beams over a main beam, which may be almost any profile. The connection uses a standard supplier's cleat with the appropriate layout and bolt details, which may be bolted or welded to the main member. Alternatively a welded plate or tee cleat can be used.
- Single Over sail Floor Beam
 - Same as Double Over sail Floor Beam except it connects a single cold rolled floor beam over a main member.

The above cold rolled joints currently support sections by the following suppliers: A G Brown, Albion, Ayrshire, Banro, B W Industries, Elwell, Hi Span, Kingspan, Kingspan Metl-Con (formerly Ward), Metsec, Phoenix, Steadmans, Structural Sections, and Tegral. The joints may be enhanced to support other cold rolled sections in the future.

In addition to these generic cold rolled macros, there are also several **supplier specific** macros:

- Ayrshire Single Swage Beam Column Joint
 - Connects a 220 or 300 deep Ayrshire Swage Beam to a column using the appropriate standard supplier's bracket. Due to the very different details for the different beam sizes, the 220 connections will trim the beam clear of the post face while the 300 connections allow the beam to **sail over** the post.
- Ayrshire Double Swage Beam Column Joint
 - Connects two 300 deep Ayrshire Swage Beams over a column using the appropriate standard supplier's bracket. Due to the very different details for the different beam sizes, this connection only applies to the 300 deep beams. For two 220 deep beams, simply apply the **single** connection twice.
- Ayrshire CAM
 - This tool is not actually a connection macro. It is used to automatically create CAM information for manufacture in the form of Ayrshire's **D3** file format. The CAM file is created for the whole model or just for selected parts. The selection is not limited to Ayrshire components, as the tool will automatically ignore any objects that are not by Ayrshire. The output is in the form of three files: the **D3** CAM file, a Warning Log, and an Error Log. The Warning Log contains any **non-standard** details, which Ayrshire may still be able to supply but possibly at extra cost. The Error Log contains any Ayrshire Components that are detailed incorrectly and cannot be supplied along with any components that are not currently supported. Any objects listed in the Error Log are **EXCLUDED** from the D3 CAM file. All three files can be viewed using the Advance Steel Document Manager.
- Metl-Con Side Rail Anti-Sag
 - This macro creates a standard anti-sag system for Kingspan Metl-Con side rails. The user selects the supporting columns at each side of the bay followed by the relevant side rails. The anti-sag system is automatically created and various options allow the strut types to be selected along with their positions and any associated diagonal ties, etc.
- Metl-Con Roof Anti-Sag
 - This macro creates a standard anti-sag system for Kingspan Metl-Con purlins/eaves beams. The user selects the supporting rafters at each side of the bay followed by the relevant purlins and (optionally) an eaves beam. The anti-sag system is automatically created and various options allow the strut types to be selected along with their positions, etc. If an eaves beam was selected then the eaves beam restraints are created.

- Metl-Con Apex Tie
 - This macro creates standard Kingspan Metl-Con apex ties between two apex purlins. The apex ties are positioned relative to an existing anti-sag system by selection or are positioned in the bay relative to selected rafters. Various tie types can be created.
- Structural Sections Side Rail Anti-Sag
 - This macro creates a standard anti-sag system for Structural Sections side rails. The user selects the supporting columns at each side of the bay followed by the relevant side rails and (optionally) an eaves beam. The anti-sag system is automatically created and various options allow the strut types to be selected along with their positions and any associated diagonal ties, etc. If an eaves beam was selected, the anti-sag system may be suspended from it and any associated stiffener cleats and eaves restraint cleats can be added.
- Structural Sections Roof Anti-Sag
 - This macro creates a standard anti-sag system for Structural Sections purlins/eaves beams. The user selects the supporting rafters at each side of the bay followed by the relevant purlins and (optionally) an eaves beam. The anti-sag system is automatically created and various options allow the strut types to be selected along with their positions and any associated diagonal ties, etc. If an eaves beam was selected then the eaves beam restraints are created and can be connected to the brackets created by the Side Rail Anti-Sag macro or can create the brackets directly.
- Structural Sections Apex Tie
 - This macro creates standard Structural Sections apex ties between two apex purlins. The apex ties are positioned relative to an existing anti-sag system by selection or are positioned in the bay relative to selected rafters. Various tie types can be created.
- Tegral Single Swage Beam Column Joint
 - Connects a 220 or 300 deep Tegral Swage Beam to a column using the appropriate standard supplier's bracket. Due to the very different details for the different beam sizes, the 220 connections will trim the beam clear of the post face while the 300 connections allow the beam to **sail over** the post.
- Tegral Double Swage Beam Column Joint
 - Connects two 300 deep Tegral Swage Beams over a column using the appropriate standard supplier's bracket. Due to the very different details for the different beam sizes, this connection only applies to the 300 deep beams. For two 220 deep beams, simply apply the **single** connection twice.
- Tegral CAM
 - This tool is not a connection macro. It is used to automatically create CAM information for manufacture in the form of Tegral's **D3** file format. The CAM file is created for the whole model or just for selected parts. The selection is not limited to Tegral components, as the tool will automatic ignore any objects that are not by Tegral. The output is in the form of three files: the **D3** CAM file, a Warning Log, and an Error Log. The Warning Log contains any **non-standard** details, which Tegral may still be able to supply but possibly at extra cost. The Error Log contains any Tegral Components that are detailed incorrectly and cannot be supplied along with any components that are not currently supported. Any objects listed in the Error Log are **Excluded** from the D3 CAM file. All three files can be viewed using the Advance Steel Document Manager.

Below the UK Hot Rolled and Cold Rolled toolbar flyouts is the **Joint macro defaults editor**. This tool is used to control the default values used by the UK joints and can be configured **Per joints** offering greater flexibility than if configured globally. This tool also allows the creation of multiple default **profiles** for each joint for use with different country or customer requirements.

Stairs

With Advance Steel, various types of stairs can be created. The tools for creating stairs, railings, ladders, and specific joints are grouped on the **Stairs & Railings** toolbar.



Figure 203: Advance **Stairs & Railings** toolbar

Spiral stair

The "Spiral stair" tool creates the central post, the handrail, the treads, the connection, and the handrail. Optionally a top cap plate can be added to the column.

Define three points and the first tread (or the last one) to create the stair. The first two points define the central post height and position. The third point defines the direction of the first tread (or the last one) and its width.

There are two ways to create a spiral stair:

- From the bottom to the top: the user defines the first step; the last step position is calculated using the first step position, the number or the height of the steps, and the rotation angle between them.
- From the top to the bottom: the user defines the last step.

Example: Creating a spiral stair

- Set an appropriate UCS.
- On the **Stairs & Railings** toolbar, click .
- Select the first point to define the spiral center: 0,0,0.
- Select the second point to define the spiral height: 0,0,2800. The stair height is on the Z-axis.
- Type **F** to create the first step (or **L** for the last one) and press **Enter**.
- Specify the third point to define the start direction of the step and the step width: 1500,0,0.

The spiral stair is created and a dialog box appears, in which information for column, stringer, treads, post, etc. can be entered. The stair direction and the type of connection between the treads and the stringers (e.g., welds, bolts, angle welded or bolted) can be selected. The preferred settings are stored in the table.

Straight stair

Two points defining the origin and the end of the stair run creates a straight stair.

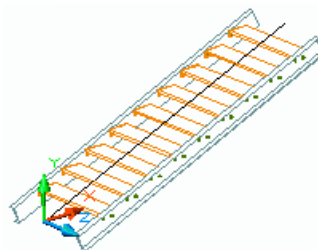


Figure 204: Straight stair

Advance creates the stringers and treads with their connections to the stringers.

The width and the offset are controlled in the properties dialog box.

The number of treads is created with the formula normally used to get the ergonomic height and width of the step.

Optionally, a lower and upper landing can be created.

Example: Creating a straight stair

- Set an appropriate UCS.
- On the **Stairs & Railings** toolbar, click .
- Select the first point to define the origin of the stair run.
- Select the second point to define the end of the stair run.
- Enter **1** to align the stair in the middle and press **Enter**.

The stair is created and a dialog box appears, in which information for **treads** and **landings** can be entered.

The dialog box to edit a **stair** element is accessed by selecting **Advance Joint Properties** on the context menu.

Connection objects

If joints require a change, right click on the connection gray box and select **Advance Properties**. Another option is to double-click the object.

Alternatively, right click only one part of the joint (e.g., a plate) and select **Advance Joint Properties** from the menu.

Tools for display, select, repeat, copy and transfer connection properties are on the **Sub – Joint Utilities** flyout of the **Joints** toolbar.



Figure 205: Advance **Sub – Joint Utilities** flyout

Button	Function
	“Create by template”: Copies the selected connection. At the new creation point, the proper elements must be selected similarly to creating a new connection. The connection is copied with all previous settings to the newly selected members.
	“Display”: Turns on invisible objects created with the connection. Click “Display” and select an element of the connection.
	“Select”: Identifies all elements in a connection by first selecting a connection object (gray box).
	“Repeat rule”: Repeats the previously executed rule for joints. The elements are selected similarly to the previously executed connection.
	“Transfer properties”: Copies parameters such as additional stiffeners or a changed haunch length to another existing connection of the same type. The workflow of this command is to first select an element from each connection to change (any number of joints may be selected) and then select the connection whose properties will be transferred. All selected joints are updated to match the properties of the last selected connection.
	“Update”: If a connection, whose automatic update has been switched off in the Update tab, has been changed then the connection object is highlighted in red. The “Update” tool manually corrects the connection object.
	“Delete all”: By selecting one or more connection objects and clicking “Delete all”, the selected connection objects are completely deleted. Objects, other than connection objects, selected at the same time will not be deleted.

The joints library contains most standard types and is continually growing. In particular cases where there is no immediate direct solution, use the connection closest to the requirement, set the best parameters, and delete the **connection object** to ‘explode’ the connection logic. Next, adjust the individual component's Advance properties and finally add or delete parts as required.

Note: *When deleting the connection object the intelligence of the joint is eliminated and individual members do not automatically update on a design change.*

Special parts, special sections

Special parts

Parts used in the Advance model and in the structured BOMs or drawings that cannot be created from other Advance objects are **special parts** (e.g., turnbuckles).

They must be designed in separate DWGs (as AutoCAD® solids) and created as Advance objects with the Advance special part tool.

The “Advance special part” tool is accessible on the **Listing & checking** toolbar (4th icon on the **Advance Steel** toolbar).



Figure 206: Advance **Listing & checking** toolbar

 **Example:** Creating a cast steel spherical cap as a special part

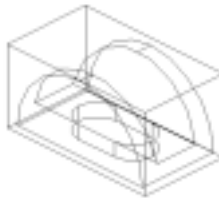


Figure 207: Cast steel spherical cap as a special part

- Create a separate DWG, draw the part using AutoCAD® solids, and then store and close the file. Verify that the AutoCAD® variable **Insunits** is set to the correct value for the units used in the models. For **mm**, set the value to **4**.
- To create a special part, on the **Listing & checking** toolbar, click .
- Indicate the creation point, which corresponds to the zero point of the DWG.
- The special part is created according to the current UCS and a dialog box appears, in which the name of the special part (i.e., name of the DWG), scale, material, information for the structured BOM, and all other preferences can be set.

As with the joints, the information for special parts is stored in the ‘bounding’ boxes. To move a special part, the box must be selected. This is automatically done upon selecting a special part object.

The box of the special part can be made visible using the  tool on the **Standard** toolbar or on the **Representation type** tab in the Advance properties dialog box for the special part.

A special part can be created at any scale. When Advance checks for identical parts, different scales for the same special part are not considered. This means that a special part can get the same position number at different scales. It is therefore recommended to draw and create special parts at full size (1:1).

The information of special parts is exported to the structured BOM and is considered during drawing creation.

The defaults can be modified to define whether or not the special part should be included in a collision check.

Special parts can be created using joint rules as in the case of bracings with turnbuckles.

Special parts can be welded and bolted to other Advance Objects. Bolt grip lengths cannot be calculated for special parts unless the default is set to do so.

User sections

Non-standard profile cross sections are called **User Sections**. They can be set in Advance and behave in the same way as standard sections in the Advance properties and functionalities.

All sections, including User Sections, are stored in MS-Access tables (libraries). The table structure for User Sections differs from the structure of standard sections. Standard section tables contain parametric values whereas User Sections include only the corner points of the cross section geometry.

User Sections are not manually entered in tables (libraries); they are drawn and transferred by means of the Advance User Section tool that identifies the profile cross section and enters the geometry into the MS-Access tables (libraries).

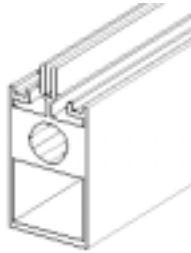


Figure 208: User Sections

Definition of User Sections

Similar to the special parts, a 'User Section' is created in two steps.

First, the section (2D cross section) is drawn with AutoCAD® elements. Then, two texts must be created with the name of the section (e.g., IPE 200) and the name of the section class (e.g., IPE).

The second step is the actual definition by a command and identification of the drawn elements. The information is proofed (e.g., the contours are not permitted to overlap) and created in the database. The section is then available in Advance.

Exact conventions and information layers must be considered when drawing and labeling special sections. Several User Section tools make this easier. These tools are available on the **User sections** toolbar with flyouts.



Figure 209: Advance **User sections** toolbar

The general workflow for drawing and defining a User Profile is as follows:

- Create a **DWG** and set the view direction to 'top view' based on the world coordinate system.
- For each User Section component set a corresponding layer. This can be done using the **Change current layer** flyout.

Note: *The special section definition is its own application (ARX). It is automatically loaded by clicking  on the **User section** toolbar.*



Figure 210: **Change current layer** flyout

- "Section outer contour": To create an exact and simple outer contour, choose the **outer contour** or **exact outer contour** layer setting. The element **MUST** be created using a closed polyline (pline), a circle, or a rectangle.

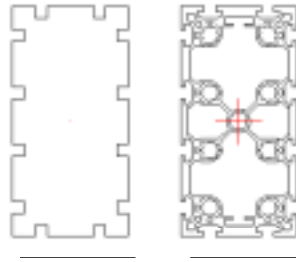


Figure 211: User Section outer contour - in standard and exact representation

- “Section inner contour”: The layer for **inner contours** (simple / exact) must be activated. Closed polylines (pline), circles, and rectangles are allowed as elements.
- Place a **section name** and assign the section to a **section class**. This must be done using text created on specific layers. The layer for the respective text must be created and activated as the current layer. Only one-line texts are permitted. They must not have blanks and in the text for the section class no negative sign.

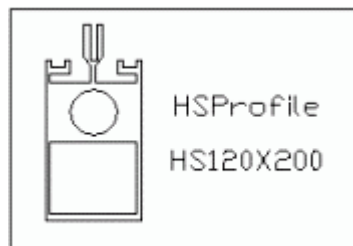


Figure 212: User Section labeling and frames

- The definition frame must contain the profile with all its information and is required to create the section. Only a rectangle drawn on the appropriate layer is permitted for this.
- “Coordinate systems”: Advance object surfaces can be selected to place the UCS. These surfaces can also be used for User Sections. The coordinate systems at the top surface of the section are automatically set. With “Add coordinates” on the **Special sections** toolbar, the surfaces can be set by selection. A symbol is created in the center of the line segments.
- Corner points of the section geometry can be set as position points of the creation axis (**Reference axis or System line**). The position of the section can be moved according to these points. When creating the section, the standard position is at the center of gravity. The position points of the reference axes are set with the tools on the **Add reference axis** flyout of the **User sections** toolbar.

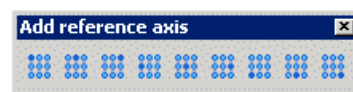


Figure 213: Add reference axis flyout

- Select the required position from the flyout, click on the screen so that the cursor becomes a crosshair, and then pick the respective corners on the outer contour (using object snap!) or from another point. A symbol is automatically created on the corresponding layer.
- “Creating Sections”: Sections are created from the geometric information and are saved in the database. Click “Create selected section” or “Create all sections” (several special sections can be designed in one drawing). Select the section to create by picking the **frame**. A message is displayed to indicate successful creation of the section(s).



Figure 214: Special sections - creation flyout

- Once the database is loaded in Advance (“Update defaults” icon on the **Listing & Checking** toolbar), the newly created section is available for use.

- If changes must be made to the geometry of existing **User Sections**, the geometry should be changed in the 2D drawing and the section recreated. Advance will replace the existing entries in the database when auditing the section names.

 *Keep the drawings where the geometry of the sections is set, as there is no option (as yet) to locate these sections if moved. All cross sections must be redrawn when databases are lost.*

User Sections Creation

After creating a User Section it is ready for use. Two options are available:

- Click “Other section” (last icon) on the **Sub – Section classes** toolbar (flyout) of the **Create objects** toolbar and select the desired special section.



Figure 215: Create special section (1)

- Click the “Beam” icon on the **Create objects** toolbar and create a section of the last created type.



Figure 216: Create special section (2)

- The “Beam” dialog box appears. Choose the user section from the list under **Section class** and **Section**.

Working methods II

Model Browser

This tool provides complete control of all the elements of a model. At any time this tool can be accessed to show up-to-date information on each object.

The model browser is accessible on the **Standard** toolbar.

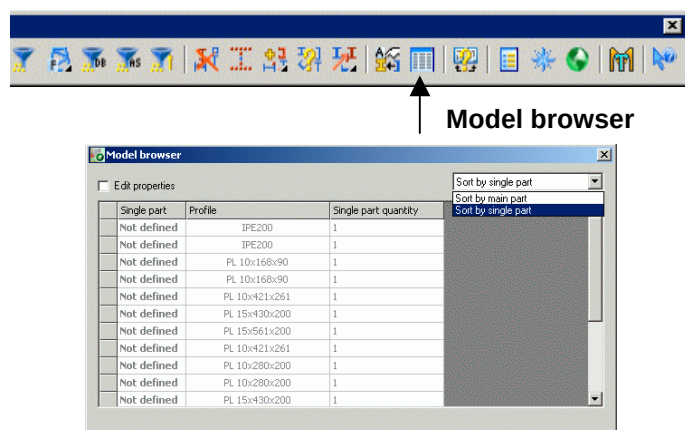


Figure 217: Model browser

Elements are sorted by main or single part mark. The element properties are displayed in the model browser by creating or removing some additional columns:

- Right click in the table.
- Select create column from the context menu. The list of properties appears.

- Select the properties to display in the model browser. The model browser displays the selected properties.

Properties are modified in the Model Browser. When **Edit properties** is checked some of the properties for the selected lines of the Model Browser can be changed.

Model views

When constructing a large model with many elements an isometric view can be confusing.

The model view feature creates local view areas that show only those elements that lie within specific boundaries. Slices are cut from the model for better visualization (e.g., to work on a single frame in a 2D-view).

The view direction can be set and nodes can also be viewed from 6 different directions at the same time.

The **Sub – Model view** toolbar (flyout) is accessible on the Advance **Standard** toolbar.



Figure 218: Advance **Model view** toolbar

First select the presentation in a single window (“Define model view” icon) or in 6 windows at the same time (“Define model view with 6 windows” icon).

To define the box and view direction, different methods are available (see example below).

After creating a local model view, the entire model can be displayed again (“All visible” icon). With the “1 Viewport” icon, 6 viewports are switched to 1 viewport. The other icons include tools to rotate the view to the right, left, upwards, and downwards.

 **Example:** Defining the first frame as a model view and representing it in 6 windows

- Choose the tool: Define model view with 6 windows on the **Sub – Model view** flyout of the **Standard** toolbar.
- In the dialog box, click **New** and select the **3 points, view direction perpendicular to surface** method.
- Pick the left back end point of the grid as the origin point. Enter the left front end point of the grid as a point in the **X**-direction and enter the end point of the system line of the column as a point in the **Y**-direction.
- Enter 100 for the extra margin and 1000 for the view depth. Describe the view with a name (e.g., “Grid 1”) and press **Enter**.
- In the dialog box, click **OK**. It will switch on the “Grid 1” view and only the first frame is visible.
- Go back to the model in the isometric view by setting the “All visible, one viewport” and “Advance view southeast” tools.

Advance – copy / rotate / mirror

The construction process workflow often proceeds much like the creation of a portal frame with certain parts of a model can be completed with all features and connections and then copied. In these cases use the AutoCAD® commands to copy, rotate, mirror, or array.



Figure 219: Advance **Design assistance** toolbar

On the Advance **Design assistance** toolbar, there are a variety of Advance tools (besides checking tools) to copy, rotate, and mirror. These are supersets of the AutoCAD® commands that add support for copying connections and linking selected objects with welds and bolts. Although the tools were designed for copying connections they can also be used for copying several sections or other objects (e.g., holes, trimmings, contours) and in most cases a multiple copy is possible.

All elements to be selected for copy must be visible. This is also true for object features like shortens, copes, etc.

When copying all elements of a joint including the connected elements, the intelligence is also copied and remains customizable. If all the elements are not copied the target copy will not have intelligence.

 Example: Copying a simple connection

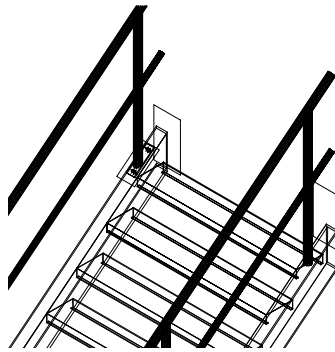


Figure 220: Advance stair

- The connection to be copied is a connection between a stair stringer and a handrail post with a base plate. The stringer profile is connected to the base plate using two bolts; the handrail post is shortened and welded to the base plate. This base detail must be copied to another handrail post. Verify that all elements are visible including the processing object at the post (if necessary, change the setting on the **Representation type** tab of the corresponding dialog box).
- On the **Design assistance** toolbar, click .
- If a selection has not been made, select the elements to copy (i.e., base plate, two bolts, welds, trimming) and press **Enter**.
- Enter the start and end points for copying. It is recommended to use the end of the system lines of the posts (visible in the "Standard" representation).
- Next, enter the coordinates of the copy vectors. Press **Enter** for all coordinates (X, Y, Z).
- At the copy destination, the objects that correspond to the objects in the initial situation are requested. The initial elements are highlighted with dotted lines. Select the object corresponding with the highlighted one at the copy destination. This correctly assigns the connection objects.
- Other destination points can be entered for a multi copy. The tool is closed by pressing **Enter** or **Esc**.

With this Advance copy tool, a complete detail with all features can be copied to another location.

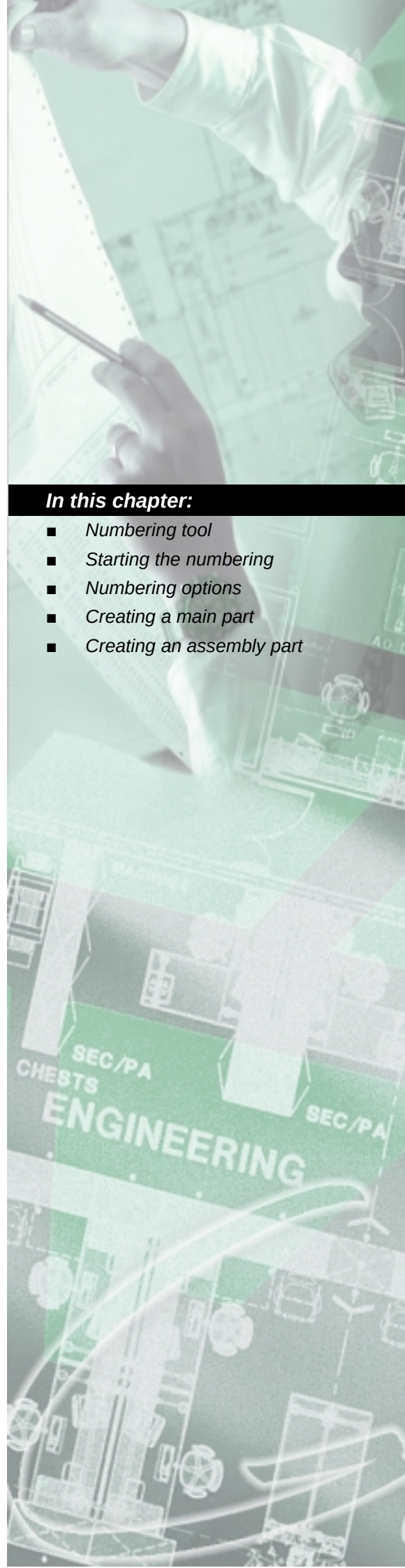
The following Advance commands for copy, rotate, and mirror are available on the **Design assistance** toolbar:

Button	Function
	Advanced copy
	Copy
	Copy and rotate, 2D
	Copy and rotate, 3D
	Copy and mirror, 2D
	Copy and mirror, 3D
	Copy with Adjustment
	Copy, rectangular array
	Copy, polar array

Chapter 5 Numbering

In this chapter:

- *Numbering tool*
- *Starting the numbering*
- *Numbering options*
- *Creating a main part*
- *Creating an assembly part*



Numbering tool

The Advance numbering tool **automatically** numbers Single Parts and Assemblies for the entire model. The basis for numbering is finding **identical parts** that should have the same mark.

The elements are compared by geometry, material properties, coating, and commodity (and behavior). The properties **name** and **lot/phase** are not considered for numbering. The **Model Role** is used by the prefix tool to assign prefixes but it is not used directly for numbering.

At first, all structural parts should be numbered so start with **single part marks**.

The program then determines the assembly marks for parts connected in the shop. The biggest part of an assembly is the main part and will get an assembly mark and all other parts are considered attached and will have a single part mark.

During the numbering process beams and then plates are the first to be numbered. In each case, the group with the most elements will get the lowest number.

If nothing is selected the entire model is numbered. When Advance objects are selected only these items are numbered. The results in either case are recorded in the AutoCAD® text window (or recorded in a text file, using a default setting, and can be saved).

All numbering tools are available on the first part of the Advance **Numbering, drawing** toolbar.



Figure 221: Advance **Numbering, drawing** toolbar

- The first toolbar button is a switch between Single Parts / Assemblies.

Button	Function
	Switch to Single Parts
	Switch to Main Parts

- Using the switch position, the following numbering tools refer to single parts or to assemblies. Unless both single part and assembly numbering are done together (recommended), then single part numbering **MUST** be done first.
- The other icons on the **Sub – Numbering** flyout are tools for assigning and deleting single part and assembly marks and defining their prefixes.

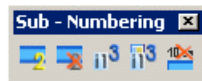


Figure 222: **Sub – Numbering** flyout

- On the **Sub – Display POS** flyout of the **Numbering, drawing** toolbar, a variety of tools are available to check the numbering correctness.
- The commands to create main and assembly parts are on the **Sub – POS check** flyout of the **Numbering, drawing** toolbar.

Starting the numbering

- Start the numbering process using the numbering  tool on the **Sub – Numbering** flyout of the Advance **Numbering, drawing** toolbar.
- A dialog box appears. The numbering for single parts and assembly parts can be performed in one step or separately. Unless both single part and assembly numbering are done together (recommended), then single part numbering **must** be done first.

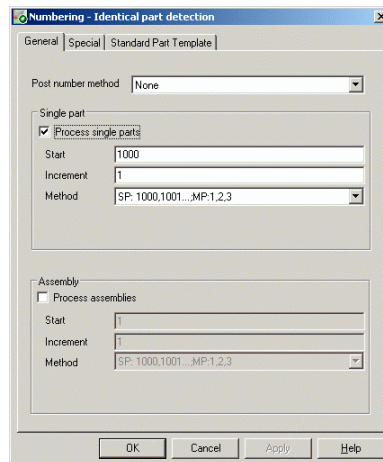


Figure 223: “Numbering” dialog box

Example 1: Single parts numbering

- Check the **Process single parts** option.
- By default, single parts marks begin with 1000 and increase by increments of 1.
- Select one of the available methods.
- Start the numbering process by clicking **OK** in the window.

The assigned marks are visible on the **Structured BOM** tab of the object dialog boxes.

Example 2: Assembly numbering

- Check the **Process assemblies** option.
- By default, assembly marks begin with 1 and increase by increments of 1.

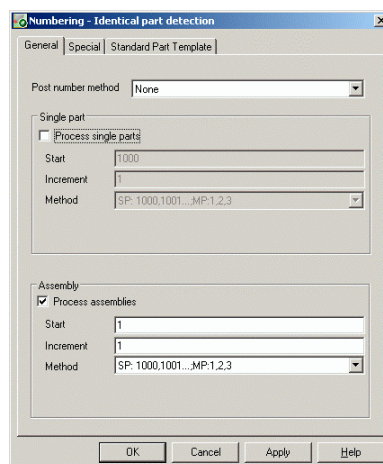


Figure 224: Assembly numbering

- Select one of the available numbering methods. For “with drawing number” numbering, define the prefix automatically given by the drawing numbering process. At drawing creation, the first part gets the prefix a001, then b001, etc.
- Start the numbering process by clicking **OK**. The result is displayed in the AutoCAD® text window.

Note that regardless of the toggle switch setting, the “Single Part” and “Assembly” numbering can be turned on/off using the checkboxes in the numbering dialog box. This can also be used to perform the recommended simultaneous “Single Part” and “Assembly” numbering.

Numbering options

Several numbering options are available in the **post number method** list:

- **Assembly group:** The numbering automatically assigns the assembly mark to the single part mark of the main part.
- **SP No (for main part) = Assembly No:** The single part mark for (usually) the biggest part of the assembly is assigned the same value as the assembly mark instead of having an individual single part mark.

With this option the single part numbering and the assembly numbering **MUST** be done at the same time and Advance forces this by disabling the other checkboxes.

Once a numbering has been performed, it is not possible to change this option...

...UNLESS the “Ungroup identical parts” tool is used, which completely removes any numbering information.

- **SP No (for standalone part) = Assembly No:** The single part mark for the standalone parts is assigned the same value as the assembly mark instead of having an individual single part mark.

Checking the results

 **Example:** Displaying the single part mark /assembly number of an object:

- Select the Advance structural part by clicking it. Open the dialog box.
- Choose the **Structured BOM** tab. The Single Part and Assembly marks are shown together with their prefixes.

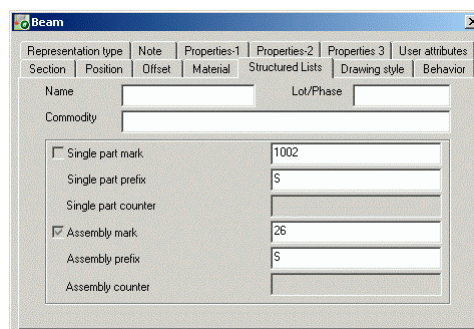


Figure 225: Marks and prefixes

Numbering with standard parts

Standard parts are created in a template model using normal Advance Steel functionality and then assigned required part marks. During a numbering process objects are compared to detect identical parts. Any part in the current model that matches a Standard Part in the template will get the same mark.

The folder

...\\Graitec\\AdvanceSteel\\Shared\\StandardPartTemplate

contains the standard part models.

The templates in this folder are listed on the **Standard part template** tab of the “Numbering - equal part detection” dialog box.

- Create a DWG model with standard parts.
- Run the numbering inside this drawing and manually set the standard part marks (and prefixes) as required.
- Save this drawing to:
`...\\Graitec\\AdvanceSteel\\Shared\\StandardPartTemplate`
- To run the numbering in another model, open the “Numbering - equal part detection” dialog box. Select the **Standard part template** tab where all drawings from the above folder are displayed. Click **New** to attach a reference model and check the option **Singlepart** or **Assembly** to use the template for single part numbering or assembly numbering.
- Several Standard Part Templates may contain the same objects but with different part marks. In this case the part mark of the template with the highest priority is used. To change the sequence of the templates use the arrow buttons on the toolbar.

Deleting part marks

This tool deletes the part marks.

- Select the structural parts whose part marks should be deleted and use the  tool on the **Sub – Numbering** flyout of the **Numbering, drawing** toolbar.

On deleting the assembly marks only the mark is deleted, not the main part identification or its prefix.

Assigning / Changing a prefix

A text prefix, separate for single part and assembly marks, can be put in front of the part mark.

- Select the desired elements, use the  tool on the **Sub – Numbering** flyout and enter an appropriate prefix without spaces or special characters.

The assigned prefix is displayed and changed on the **Structured BOM** tab of the element dialog boxes. Prefixes are also assigned in the dialog box.

Prefix configuration

The automatic prefixes for single part and assembly numbering can be configured. The prefix is based on the model role.

- On the **Numbering, drawing** toolbar, **Sub – Numbering** flyout, click .

The prefix configuration dialog box appears.

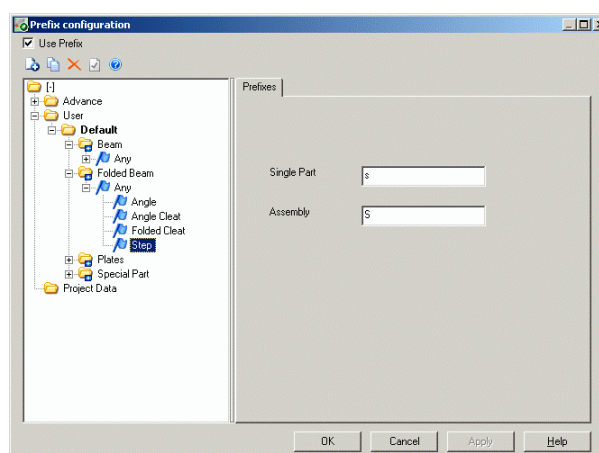


Figure 226: Prefix configuration dialog box

Different profiles for different jobs are set. The preferred profile should be set as current.

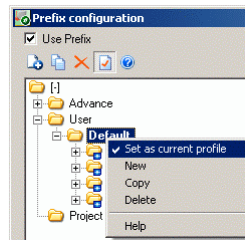


Figure 227: Prefix configuration – set current profile

Creating a main part

Main part assignment is either performed manually or by the numbering process using **Create main part of assembly** that automatically detects attached parts. Alternatively, during assembly numbering, the biggest part automatically becomes the main part.

- On the **Numbering, drawing** toolbar, **Sub – MP AP** flyout, click .
- Select the part to make the main part. The object dialog box appears with the **Structured BOM** tab selected.
- Enter the assembly mark and close the dialog box or just close the dialog box and run an automatic numbering.

Creating an assembly part

Connected parts (by shop welds or shop bolts) can be assigned to an assembly part.

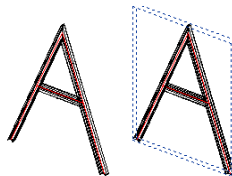


Figure 228: Assembly part

- On the **Numbering, drawing** toolbar, **Sub – MP AP** flyout, click . Select one of the connected parts and a dialog box appears.
- Enter an assembly mark and close the dialog box or just close the dialog box and run an automatic numbering.

A blue box object (the assembly part) is created around the connected parts and this becomes the main part of the assembly. All other parts automatically become attached parts.

For example, stairs normally have one stringer as the main part and all other parts are attached parts thus identical stringers get different marks (depending on the numbering method). The “Create assembly part” tool creates an assembly part box that becomes the main part title in the structured BOM. All parts within the blue box are attached parts and are listed with single part marks (i.e., both stringers will then have the same mark).

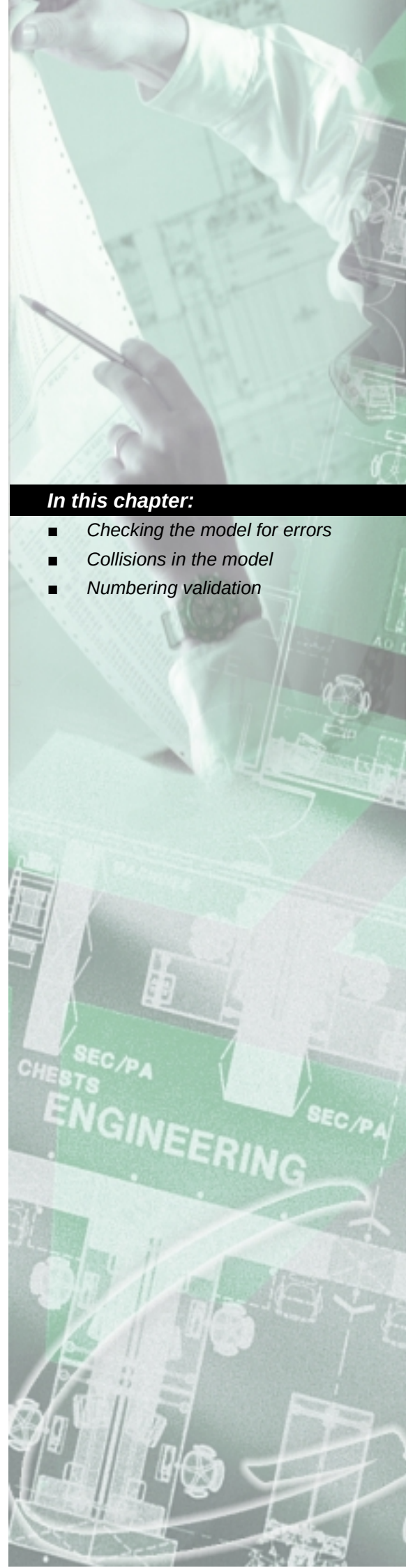
The assembly part object can be hidden using the  tool on the **Standard** toolbar.

Chapter 6

Checking the model

In this chapter:

- Checking the model for errors
- Collisions in the model
- Numbering validation



Checking the model for errors

Checks are required in various steps of the modeling process. Collision checks can help track down 'overlapping duplicate' members, members with no connections, problem connections and general collision situations.

After numbering, it is recommended to carry out a series of error checks to correct elements with duplicate part marks, un-marked elements, etc. With these tools, mistakes in the structured BOM and detail drawings (and the associated costs) are avoided.

The Advance database should also be maintained and checked from time to time with the Advance "Audit checking (database)"  and "Steel construction checking"  tools.

The checking tools are located on different toolbars, namely the Advance **Design assistance** and **Listing & checking** toolbars.



Figure 229: Advance **Listing & checking** toolbar

The **Listing & checking** toolbar contains the following checking tools:

- Collisions in the model
- Advance AUDIT checking
- Advance AUDIT checking (database)
- Steel Construction Checking

Other icons on the toolbar contains the Advance commands:

- Display Checking Results
- Display Checking Results Again



Figure 230: **Design assistance** toolbar

The **Design assistance** toolbar (which is not turned on and off with the other toolbars on the screen) contains four checking tools excluding the "Steel construction checking" and "Display checking results again" commands.

Feasibility checking is performed after numbering and the tools are located together with the numbering tools on the Advance **Numbering, drawing** toolbar.



Figure 231: **Numbering, drawing** toolbar: correctness checking

The commands for feasibility checking are available on the **Sub – Display POS** and **Sub – POS check** toolbars (flyouts).

Collisions in the model

This tool checks for collisions in beams, plates, and bolts for either the entire model or a selection set.

The program performs various checks. Bolts are checked including the creation distance at the head and nut and can be reversed in direction. Bolt patterns are checked against each other with verification for bolting tool spacing. Holes and exact beam cross sections (including root radii) are checked, identical overlapping elements (beams/plates copied on top of themselves) and incorrectly modified elements are recognized.

The tool compensates for geometric inaccuracies with a given minimum volume of the collision solids set in the defaults. ACIS-solids are considered in the collision and collisions with a small volume are ignored.



Figure 232: Advance **Listing & checking** toolbar

On the toolbar **Listing & checking**, the following tools are available:

Button	Function
	Collision test (Sub – Checking flyout)
	Display Checking Results
	Display Checking Results Again

 **Example:** Carrying out a collision check for selected elements

- Select the elements to check.
- Click the “Clash check”  tool.

The collisions are listed in the text window with continuous index numbers.

Collision solids are not created directly since there can be many. The results can be displayed with the “Display checking results” and “Display checking results again” tools.

Displaying checking results

“Display checking results” displays the collision solids for a particular collision by entering its index number. The index number is listed in the text window by the clash check.

- Start the tool by clicking  (**Listing & checking** toolbar). The index number of the desired object is requested in the text-window:

Please index of the object: _

- Enter the index number of the object on the AutoCAD® command line. The collision solid is represented in red.

In complex models collision solids are hard to find.

- The “Search highlighted objects”  command on the Advance **Standard** toolbar creates a red arrow that shows the location of the collision solids.
- Turn off collision solids and the marking arrow with the “Clear highlighted objects”  command on the Advance **Standard** toolbar.

Displaying checking results again

The results list from the Advance clash check can be recalled in the text window.

 **Example:** Listing the clash check results. Objects are listed with a continuous index number.

- On the **Listing & checking** toolbar, click . The results appear in the text window.

Advance Audit checking

Advance objects have special properties that will not be taken into consideration by the AutoCAD® audit checking. The Advance **audit** checking is specialized for Advance objects and should be performed once a day to validate coordinates of the objects (internal checking).

The following checking tools are grouped on the **Sub – Checking** flyout of the Advance **Listing & checking** toolbar:



Figure 233: Toolbar **Listing & checking**, flyout **Sub – Checking**

Button	Function
	Collision test
	Activate Advance AUDIT checking
	Advance AUDIT checking (database)
	Mark object
	Steel Construction Checking

- On the **Listing & checking** toolbar, click . The following message appears in the text window:
Audit extended by Advance
- If you click the icon again, the activation is switched off. In the text window, the following message appears:
Audit by AutoCAD

Advance audit checking (database)

This command checks the database. For example, the program can recognize and delete identical shortens of a beam. To only show the errors, run **audit** checking without fixing them.

- On the **Listing & checking** toolbar, click . The following message appears in the text window:
Audit extended by Advance
Repair detected errors? (Yes / No)

By entering *no*, a result list of the un-fixed errors is displayed.

Mark an incorrect object with the command `_AstM4CommMarkObject`, press **Enter** and enter **h** (handle). Enter the handle number from the result list.

Steel construction checking

The “Steel construction checking” command (**Listing & checking** toolbar) checks an object's attribute values in the database with particular emphasis on the distance between bolts within a pattern.

After clicking , the following request appears in the text window:

Should objects with serious errors be deleted ? (Yes No)

“Advance audit checking” checks attributes of an object in the database while “Steel construction checking” checks the values.

Defining the center of gravity

The “Center of gravity and total weight” tool on the Advance **Listing & checking** toolbar calculates the center of gravity of a selected group of Advance objects.

- On the **Listing & checking** toolbar, click . Select the elements to use in the center of gravity calculation.

The center of gravity position in relation to the world coordinate system and the total weight of the selected structural parts are displayed in the AutoCAD® text window.

An AutoCAD® point is automatically created at the center of gravity.

Updating defaults

When starting Advance the current system defaults are automatically loaded. If the default values are changed while Advance is active these changes will normally not take effect until Advance is restarted.

The  tool (the last icon on the **Listing & checking** toolbar) sets new default settings without restarting Advance. This tool imports the files again and a restart is not necessary.

Numbering validation

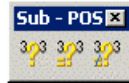
Various checks can be carried out after numbering the model. These are available on the **Sub – Display POS** and **Sub – POS check** flyouts of the Advance **Numbering, drawing** toolbar.

Depending on how the **Single Parts / Assemblies** toggle switch is set, the following tools will change to single parts or assemblies, respectively.



Figure 234: Sub – Display POS flyout

Button	Function
	Display objects with identical part marks
	Select objects with different part marks
	Mark main part of assembly
	Mark loose parts
	Mark parts without marks

Figure 235: **Numbering, drawing** toolbar, **Sub – Pos check** flyout

Button	Function
	Check for duplicate part marks
	Identify objects with identical part marks
	Differences between two objects

Displaying objects with identical part marks

All items with the same part mark, when selected, are highlighted in red. This tool can be meaningful if two objects are apparently different but show the same part mark.

- Select a structural part and click the “Display objects with identical position numbers”  tool on the **Sub – Display pos** flyout. The found items are highlighted in red.



*Objects that were previously highlighted in red will not be undone. To remove the highlight, use “Clear marked objects” on the **Standard** toolbar.*

Selecting objects with different part marks

A single object of each assembly or single part mark (depending on the “Single Parts / Assemblies” switch) is selected and highlighted in red.

 **Example:** Individual parts for each part mark should be selected for drawing creation if done manually rather than by a process.

- On the **Numbering, drawing** toolbar, **Sub – Display POS** flyout, click . An element for each part mark in the model is selected and highlighted in red.
- If the “Drawing – Single Part” tool is called, a shop drawing for each position number group is created. Without this command, each part is detailed as many times as the parts exist in the model if the system default is set to allow duplicate drawings.
- It should be noted that this command is not required if creating drawings using a drawing ‘process’ as the process automatically filters out a single instance of each part mark.

Marking the main part of assembly

The “Mark main part of assembly” command highlights the main part of an assembly in **blue** and all attached parts in **red**. A prerequisite is that the selected assembly has a main part previously identified.

- Select a part that belongs to an assembly and click the “Mark main part structure”  tool on the **Sub – Display POS** flyout of the **Numbering, drawing** toolbar.

Main parts are presented in blue, attached parts in red, and connecting elements (bolts and welds) in green.

Marking loose parts

This tool searches the complete model for loose parts (i.e., parts that leave the workshop without any connection to other parts). Any objects that are found are highlighted in red.

- After calling the “Mark loose parts”  tool on the **Sub – Display POS** flyout of the **Numbering, drawing** toolbar, a window appears displaying a message about the number of single parts found.
- Clicking **OK** marks the loose parts.

Marking parts without marks

This tool searches the entire model (depending on the **Single Parts / Assemblies** setting) for structural parts that do not have a part mark (e.g., because they were created after the numbering).

Note that, if a part is not identified as a main part and the toggle switch is set to Assemblies, this tool will not find any un-marked objects. Therefore newly added objects will typically only get detected if the switch is set to “Single Parts”.

- On the **Numbering, drawing** toolbar, **Sub – Display POS** flyout, click “Mark parts without numbers” . Any parts found are highlighted in red.

Checking for duplicate part marks

This tool is available on the **Sub – POS check** flyout of the **Numbering, drawing** toolbar.

All parts of a group with the same part mark are checked to see if they are still identical. If the parts are no longer identical, they need another part mark. This is the case if they were changed after the numbering.

- On the **Numbering, drawing** toolbar, **Sub – POS check** flyout, click .

If any errors are found, a list is displayed in the text window with the incorrect part marks.

Identifying objects with identical part marks

The model is searched for objects of a given part mark. If these objects are not identical they are highlighted in different colors.

- On the **Numbering, drawing** toolbar, **Sub – POS check** flyout, click . The text window asks for the part mark.
- Enter the part mark to check, which was displayed by the “Check for duplicate part numbers”  command. The mark must include the prefix if appropriate (which is case sensitive).

The corresponding parts are highlighted in color.

Differences between two objects

Two parts can be compared for differences.

- On the **Numbering, drawing** toolbar, **Sub – POS check** flyout, click  and select two objects.

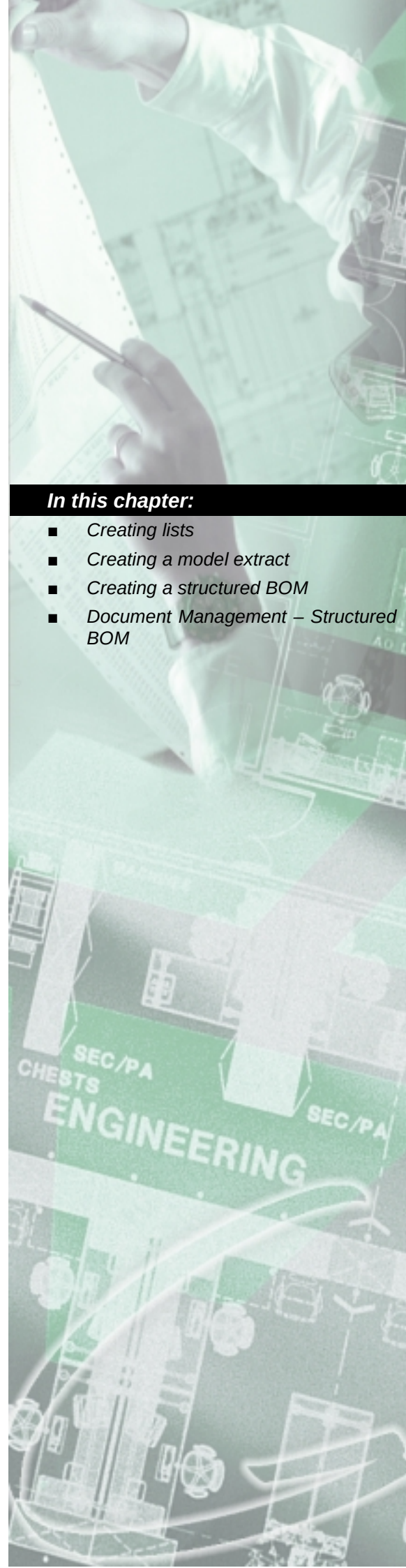
The program displays the differences in a window.

Chapter 7

Lists / Bills of materials

In this chapter:

- *Creating lists*
- *Creating a model extract*
- *Creating a structured BOM*
- *Document Management – Structured BOM*



Creating lists

All model objects including their graphic and non-graphic properties, features, joints, and connections are stored and managed by Advance. Extracts are created from the numbered and saved models. The Advance List Template Wizard creates structured BOMs from the extracts. The created BOMs can be saved, printed, or exported in various formats.

Creating a model extract

The current Advance model must first be given single part and assembly marks and the DWG model must be saved in a selected location.

From the model information an extract is created which is exported to the BOM module.

Start this tool by clicking “BOM extract”  on the **Sub – BOM** flyout of the Advance **Listing & checking** toolbar.

A window appears where the model objects for the extract are selected.

In the left part of the window, different saved configurations of model objects can be selected. In the right side of the window, different selections of the **Model objects** can be made for the current configuration.

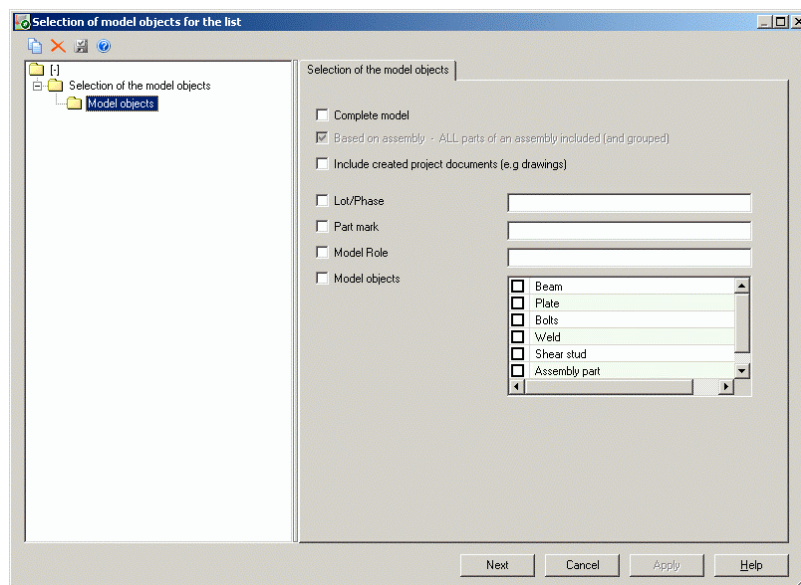


Figure 236: Model objects for the structured BOM

- To save a selection set, click “New”  on the dialog box toolbar.
- In the dialog box select a model object to start with from the list. Enter a name and click **Next**.

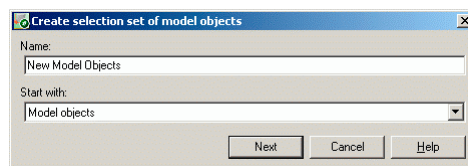


Figure 237: Set the name for the new Model object

The name is displayed in the left field.

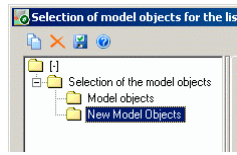


Figure 238: The new Model object

- The configuration is saved by clicking **Apply**. To continue, click **Next**.
- In the **Select destination file** dialog box, click **OK** to save the file.

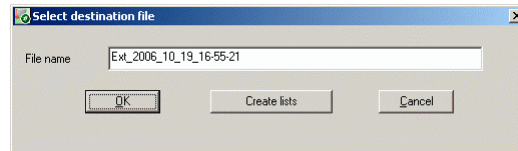


Figure 239: Select destination file

If **Create list** is clicked, the information is stored and the List Template Wizard is started to create and print the information list.

The **complete model** can be included in the extract or the **Based on assembly** option can be used to create structured reports. No other entries are needed with these options.

Other selection options include: Lot/Phase, part mark, and model role. Enter the required Lot/Phase, part mark (e.g., from 1-100), or the model role within the model (e.g., column, rafter, etc.).

 **Example:** *Structured BOM of the columns*

- After numbering and saving the model, click  on the **Sub – BOM** flyout of the Advance **Listing & checking** toolbar.
- Check **model role** and enter **column** in the text input field.
- Click **Apply** and **Next** at the bottom edge of the dialog box to create the extract.

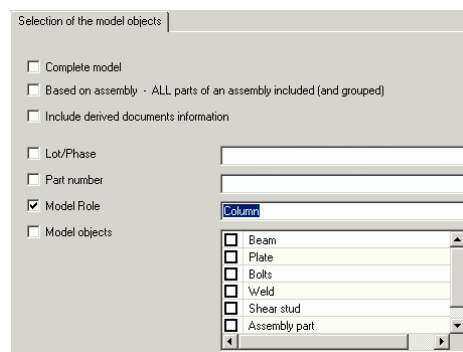


Figure 240: Selection window for structured BOM (clipped)

The **Model objects** option has other selection options (e.g., beam, plate, bolts, welds, shear studs, assembly part, special part).

After clicking **Apply** and **OK**, the model information is selected using the given criteria. A model extract is created and saved within a subfolder of the main model folder.

After each model change, it must be checked again, numbered, saved and its information re-extracted.

For each model, several model extracts for creating the structured BOM can be saved.

Creating a structured BOM

Start the Advance List Template Wizard by clicking “BOM extract”  on the **Sub – BOM** flyout of the Advance **Listing & checking** toolbar.

In the “List – Template Wizard”, select the formatting and the structured BOM (report).

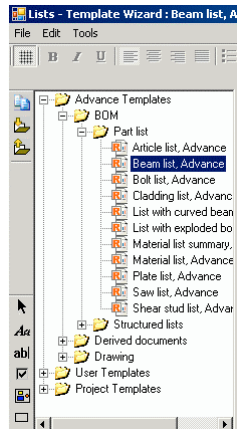


Figure 241: Template wizard – list of available BOM templates

An Advance template may be selected or users can define their own templates starting from an existing one.

- Select a BOM template.
- Click **Use**.
- A dialog box appears where the desired model extract is selected.

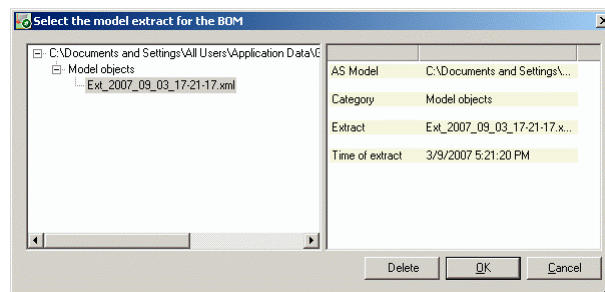


Figure 242: Model extract

- Click **OK** to finish the BOM creation.

The structured BOM appears in the window. This can be printed, saved, or exported to PDF (and other formats) using the corresponding icons on the menu bar.



Figure 243: Menu bars of the **Management Tool** and **BOM** (clipped)

To export the created BOM file, click . Select the export format from the list.

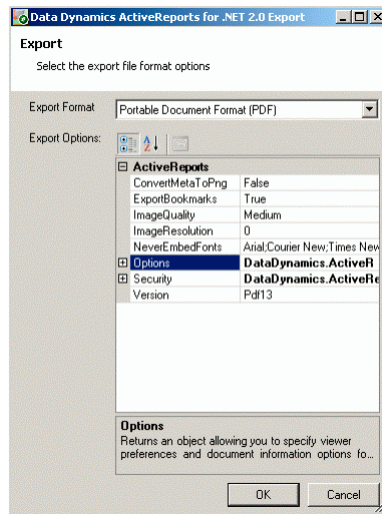


Figure 244: Export formats

- Save the created BOM file. A file name is requested in a new window. The BOM file is saved as a Report in the folder

..\[model folder]\[model name]\BOM\[BOMfilename]

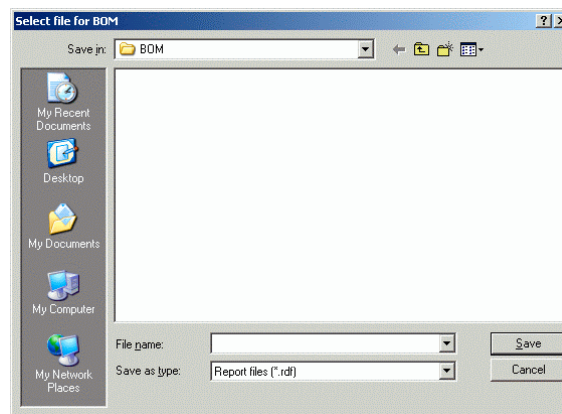


Figure 245: Report window: Save the structured BOM

GRAITEC

Date 16-Nov-2000

Project

Author

Position	Type code	Amount	Quality	Length	Width	Weight of piece	Total Weight	Surface of piece	Total Surface	Description
		(piece)		(mm)	(mm)	(kg/piece)	(kg)	(m ² /piece)	(m ²)	
Beams										
P1	PL6x120x128	20	S235JR62	120		0.7	145	0.032	0.644	
P2	IPE100	20	S235JR62	6,074		49.2	994	2.43	48.592	
P3	PL6x145x122	10	S235JR62	145		0.8	8.3	0.037	0.372	
P5	IPE300	6	S235JR62	5,011		211.5	1,268.8	5.813	34.878	
P6	IPE300	6	S235JR62	1,118		27.6	165.8	1.207	7.781	
P7	HEA240	6	S235JR62	5,006		302	1,811.8	6.961	41.163	
P8	IPE200	4	S235JR62	5,963		133.6	534.2	4.579	18.317	
P9	IPE300	3	S235JR62	463		11	32.9	0.526	1.578	
P10	IPE300	3	S235JR62	463		10.9	32.8	0.525	1.576	
		78					4,853.2		154.900	
Plates										
P11	PL15x277x71	12	S235JR62	277	71	2.3	27.4	0.040	0.596	
P12	PL15x260x180	8	S235JR62	260	180	4.9	39.2	0.066	0.766	
P13	PL15x486x190	6	S235JR62	486	190	8.6	51.5	0.165	0.989	
P14	PL15x240x225	6	S235JR62	240	225	6.3	38.1	0.122	0.73	
P15	PL15x606x240	6	S235JR62	606	240	17.1	102.7	0.316	1.937	
P16	PL20x300x300	6	S235JR62	300	300	14.1	84.6	0.204	1.224	
		44					343.8		6.193	
Bolts										
	Anchor 20 - 600	12	4.6	620		1.7	20.4			

page 1 / 2

Figure 246: Example of BOM with GRAITEC formatting (clipped)

Document Management – Structured BOM

The structured BOMs linked from the model are managed in the Document Manager as NC-data and drawing details.

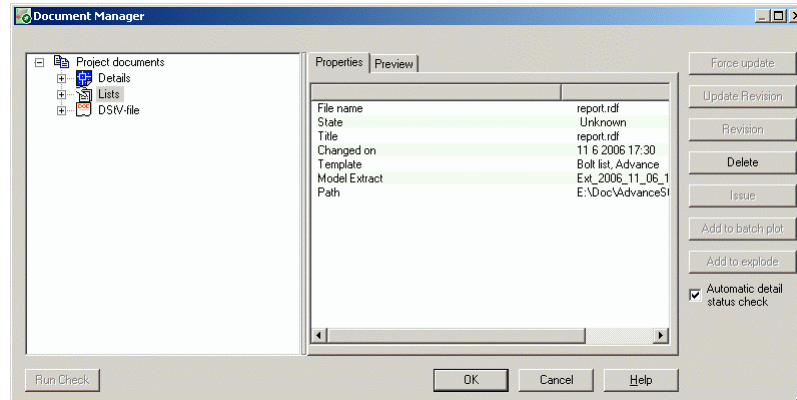


Figure 247: Document management – structured BOM, Properties

For a structured BOM selected from the **Created documents**, the information of the BOM file (e.g., file name, state, title, path) is displayed on the **Properties** tab. If modifications in the model affect the BOM, the drawing appears in the Update BOM branch.

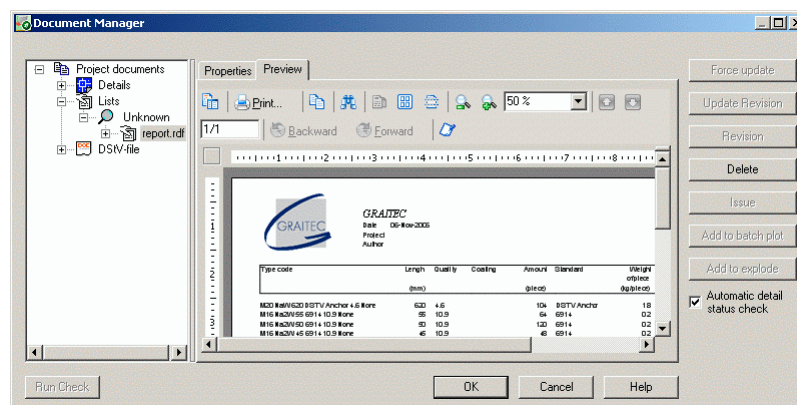


Figure 248: Document Manager – structured BOM, Preview

A preview of the structured BOM is displayed on the **Preview** tab of the Document Manager.

- The BOMs can also be printed directly from the **Document Manager**.
- Close the **Document Manager** by clicking **OK**.

Further details on the Document Manager are available in the *Creation of general arrangement and shop drawings* chapter and in the *Advance online Help*.

Chapter 8

Creation of general arrangement and shop drawings

In this chapter:

- Drawings
- General arrangement drawings
- Drawing management
- Document management
- Node details
- Elevations
- Single part drawings
- Tie beam
- Columns
- Assembly drawings
- Special structural parts
- Alternative drawing styles
- Processes
- Labeling and dimensioning
- Drawing cleanup



Drawings

Tools for managing, creating, and editing drawings are grouped on the Advance **Numbering, drawing** toolbar.

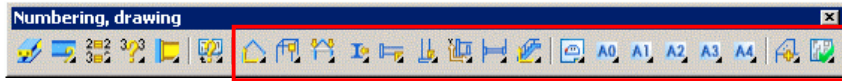


Figure 249: Advance **Numbering, drawing** toolbar

The predefined drawing styles are different for each installation and country Add-In.

Model - Drawing principles

Upon building a 3D-model, dimensioned and labeled 2D general arrangement and shop drawings can be automatically created. The drawings are created in separate DWG files from the model, however they are linked to track changes.

The model knows which drawings have been linked and checks if these drawings still correlate. Thus, the drawings can be updated after any model modifications. This link is 'one way' and changing a drawing will not modify the model.

A drawing may consist of several linked details, which are individual Advance objects with their own properties.

Drawing styles

Advance offers a variety of **drawing styles** for general arrangement drawings, sections, and shop drawings in various designs. The **drawing style** is a group of instructions used to create a detail drawing and defines the selection of the elements that are to be displayed including labeling and dimensioning preferences.

The drawing styles provide the option to automatically create drawings and to modify the layout exactly to user requirements. Drawing styles are used in a similar way to AutoCAD® dimension styles, line styles, etc.

The styles are defined with various settings (e.g., parts to be displayed, view, dimensions, labeling, representation etc.) in MS-Access tables (libraries). The drawing styles available on the **Numbering, drawing** toolbar represent a selection of the existing drawing styles available in Advance. The rules included in a drawing style can be changed.

It is also possible to define custom drawing styles using the Drawing Style Manager.

Descriptions of all Advance drawing styles are available in the *Advance online Help*.

A drawing style for a model or for selected objects can be assigned during the creation of drawings. It is also possible to assign a drawing style to an object during its creation or later using the **Style** tab in the Advance **Properties** dialog box, which distinguishes between single parts and assemblies when creating drawings.

The linked drawings (details) can be automatically dimensioned and labeled, but additional dimensions and labels can be added later using Advance's special tools (refer to the *Drawing Cleanup* section of this chapter).

Note: *Note that drawing creation depends on the Advance license. For arrangement drawings the HS Detailing Basis module is required and for shop drawings the HS Detailing module is necessary.*

Drawing management

The **Document Manager** is used to preview, manage, and erase the linked details in separate drawings (DWGs). If a DWG model or a DWG drawing is renamed, the relationship is broken but can be recreated (registered). Drawings can also be separated (unlinked) from the model.

The Document Manager interprets which detail drawings need updating after modifications to the 3D model. The updating of drawings is also controlled in the Document Manager.

Processes

In addition to drawing styles Advance has **Processes**, which automatically create drawings (using appropriate drawing styles) and arrange the linked details within a drawing (DWG) or across several drawings.

Drawing creation and AutoCAD® objects

The drawing creation in Advance is optimized for processing Advance Objects. Standard AutoCAD® objects (e.g., lines) can also be processed. This is done in two different ways:

- AutoCAD® elements in the 3D Model are used to create drawings. Like Advance Objects they are automatically included in a corresponding drawing view but as separately created **block references**. The advantage is that they are completely recognized and are modified using standard AutoCAD® commands. These blocks will not be deleted during a detail update. When creating drawings AutoCAD® objects can be included by all Advance drawing commands by pre-selecting these objects before starting the drawing tool. This selection is distinct and completely independent of special commands like selecting Advance objects. A practical example is the presentation of circular grid lines using AutoCAD® lines. When creating a plan view, these objects can be displayed in the corresponding details.
- AutoCAD® objects can also be created directly in linked drawings. This is done using standard AutoCAD® drawing commands in paper space. The geometry of the Advance drawing objects is accessed with the ENDPOINT, MIDDLE, and NODE snaps. When Advance drawings are updated the objects created are not deleted.

Complete information on drawing creation is available in the Advance online Help.

Creating drawings

Before creating any drawings, verify that in the model:

- No collisions occur
- All parts are correctly connected
- All parts have correct part marks

Create drawings with the following steps:

- Select any non-Advance elements (e.g., AutoCAD® lines, solids, etc.) or objects with certain properties (e.g., Assembly). If needed, use the search tools in Advance to mark and select the objects or use the standard AutoCAD® selection methods (e.g., picking, crossing window, etc).
- Start the appropriate tool (e.g., floor plan, section, single part, etc.).
- Other elements may be required. Press **Enter**.

The “Drawing type” dialog box appears.

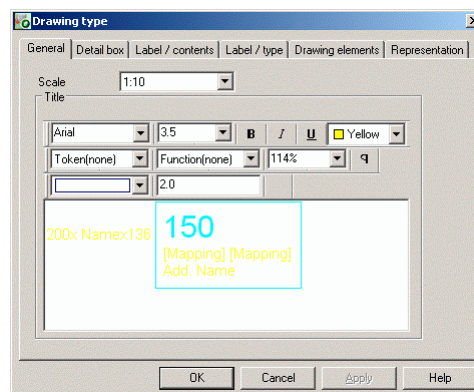


Figure 250: “Drawing type” dialog box – **General**

The various tabs provide entries for specifying drawing content.

- The **General** tab has options to specify the scale and the title of a view before its creation and to change the labels within a drawing.
- On the **Detail box** tab, values for the Z-viewport can be entered if the view depth should be limited to the front or the back (starting from origin in Z-direction of the UCS). The X/Y-viewport sets whether the entire model, selected elements, or only a local clipped view should be linked. The height (Y-direction of UCS) and the width (X-direction of UCS) can be set for the clipping.
- On the **Label / contents** tab, the various labeling types can be set for different structural parts. For example, a section label can be set to include name, length, material, and part mark.
- On the **Label / type** tab, options for using the drawing style labels are available. These specify, for example, the text orientation, the placement, etc.
- The **Drawing elements** tab includes switches for level symbols, weld symbols, and building grids. These switches are only active in drawing styles that contain these elements. For example, the building grids can be switched off for isometric views.
- The **Representation** tab contains switches for turning off hidden lines and the shorten representation (Clipping). These switches are only active in drawing styles that contain the respective representations. Clipping may be used in the **intersection** styles and in the drawing styles for main and single parts.

Prototype files

Template drawings for certain drawing formats, drawing frames, and title blocks are available in Advance and can be found in

.../Programs Files/GRAITEC/AdvanceSteel/Shared/Support/Frames

for drawing frames with various paper formats and title blocks and

/Programs Files/GRAITEC/AdvanceSteel/Shared/Support/Prototypes

for drawing frames with various paper formats, title blocks, and variable dimension styles for drawings creation.

General arrangement drawings

The **Drawing view** flyout of the Advance **Numbering, drawing** toolbar contains drawing styles that create isometric views of an entire model.

The view direction of the linked view depends on the current user coordinate system (UCS) and runs against its positive Z-axis.

The styles essentially differ by the type of labeling of the structural parts. The presentation in all styles is without hidden lines.

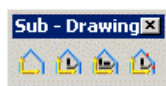


Figure 251: **Sub – Drawing view** flyout

- In “View 1”, a style for a view of a model without structural part labeling or hidden lines is available.
- In “View 2”, the Advance model is presented as in “View 1” but labeled with main part marks with a leader line.
- The drawing style for “View 3” labels the main parts of the model with profile names (e.g., HEA240).
- The tool in “View 4” is for specific items of the model (e.g., displays only columns). The displayed objects are labeled with main part marks and leader lines.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

Example: Creating an isometric view 3 at scale 1:50

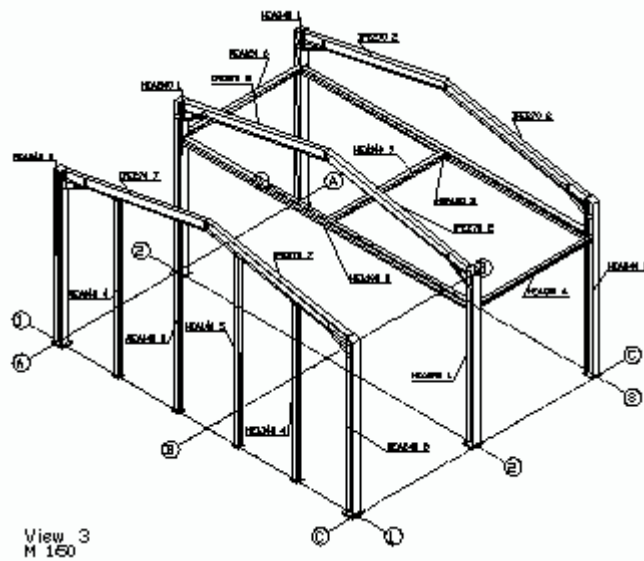


Figure 252: Isometric view

- Verify that the single parts and main parts are numbered.
- To create an isometric view, set the UCS to the screen plane with the AutoCAD® **View UCS** command on the AutoCAD® UCS toolbar (flyout) of the AutoCAD® standard toolbar.
- Call the “View 3” tool on the **Drawing view** flyout of the Advance **Numbering, drawing** toolbar. The “Drawing type” dialog box appears.
- Enter the scale 1:50 on the **General** tab. In the “Title” area, the title of the view (i.e., font type, style, size, color, etc.) is assigned and the labels inside a drawing are changed. This is entered with the scale in the lower left corner of the drawing (see description above of view 3). Other entries for **Drawing type** are available on the other tabs.

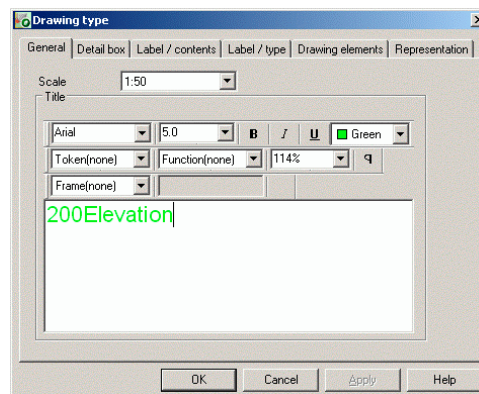


Figure 253: View detail, “Drawing type” dialog box

- Click **OK**. Another dialog box appears, in which the file name and the drawing number are entered. The drawing number field is automatically filled with the next drawing number. A drawing prototype file can be selected from the list.

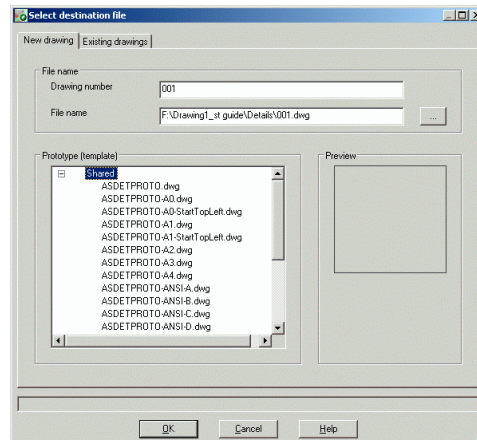


Figure 254: Select a destination file for drawing details

- Click **OK**. This button is inactive if the drawing number or the filename is not unique. The tool then starts and ACIS-solids can be selected.
- Press **Enter**. The view is created and saved in the specified DWG file.

Drawing management

During the drawing creation several details can be put on one sheet by entering the same file name. Moving these views to another DWG file is possible without losing the link to the model.

Alternatively, each detail view can be specified on a new DWG at the time of creation.

- A drawing detail or a list of the created drawings is previewed using the **Document Manager**. To open a drawing, use the AutoCAD® menu **File > Open** and enter the path name. The drawing can then be edited, printed, etc.

If a detail drawing is renamed or moved, it must be registered (or canceled) in the model so that the relationship between the drawing and model can be managed correctly.

Document management

The tools for document management are accessible on the **Sub – Drawing Management** toolbar (flyout) (Advance **Standard** toolbar and Advance **Numbering, drawing** toolbar). The following tools are available:



Figure 255: **Numbering, drawing** toolbar, **Sub – Drawing Management** flyout

- Document Manager
- Register drawing in model
- Cancel drawing from model
- Drawing Style Manager
- Drawing Process Manager
- Drawing Process Suite

- Necessary view wizard
- Define drawing layout

Document Manager - drawing details

The Document Manager lists all created drawings, bill of materials / structured BOMs, and the NC-data created from the model. The documents are displayed using the **Preview** tab.

The **Document Manager** manages and presents the connection between the model and the linked drawings. The user can display, update, add a revision, flag the issue of drawings, add to batch plot or batch explode, and delete the drawing details in the **Document Manager**.

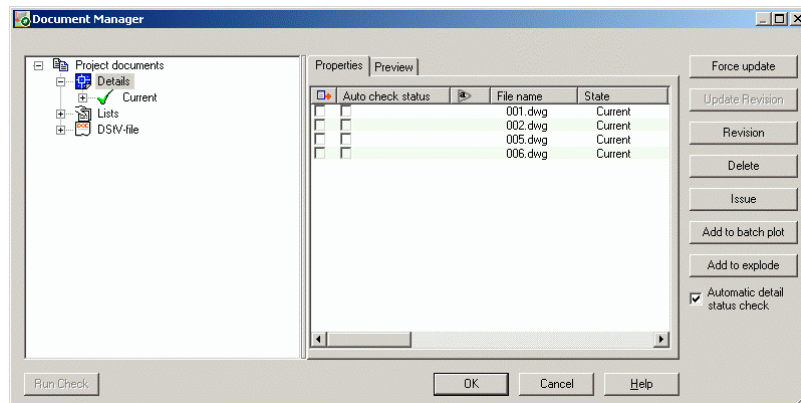


Figure 256: Document Manager – Details

The Document Manager automatically checks if drawings require an update due to model modifications or the drawings to be checked can be selected. Drawing updates are performed directly from the Document Manager. It is also possible to display a drawing on the **Preview** tab although changes cannot be made to it.

- Call the Document Manager using the first icon on the **Sub – Drawing management** toolbar.
- If **Details** is clicked under **Project documents** (in left half of the Document Manager), the created documents are listed (in right half of the Document Manager window). The current drawings are highlighted with a green checkmark with the state **Current**. A *yellow finger* marks the detail drawings that require updating due to model modifications (**Force update** in the right side of the Document Manager).
- The **Properties** and **Preview** tabs are available in the right side of the window. On the **Properties** tab, all drawings properties such as name, title, state (current, deleted or update required), date of a change, drawing style, process, etc. are displayed.
- On the **Preview** tab, a selected drawing is previewed. Changes cannot be made to the drawing. To edit a drawing, open it with the AutoCAD® **Open > File** tool and enter the path.

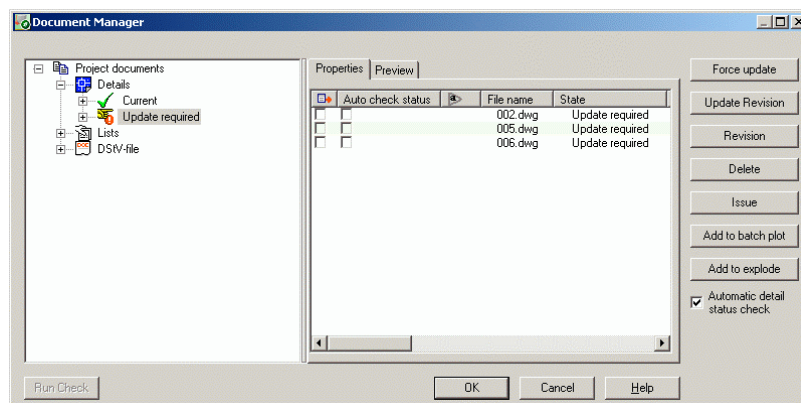


Figure 257: Document Manager, update required

Any drawings that are no longer current due to model modifications are highlighted with a yellow finger. Drawings can be selected so that the state is automatically checked.

If the **Auto check status** option is not checked in the drawings properties, clicking **Run check** will check the drawing state. The **Force update** tool is used to update the selected detail(s) with a single click. The drawings are grouped in three categories: unknown, update required, and current.

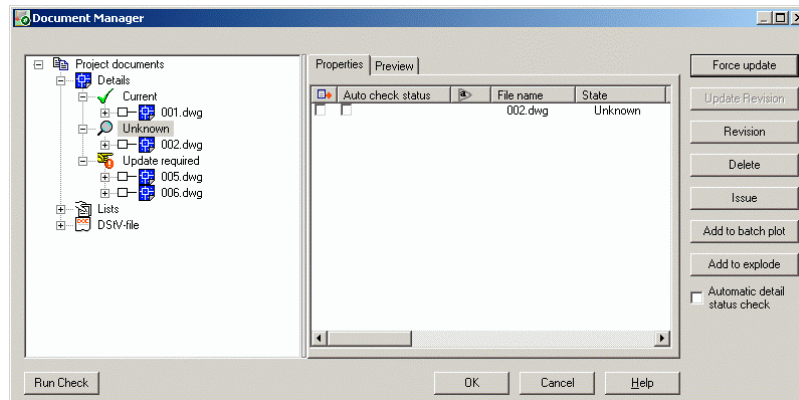


Figure 258: Document Manager, categories: Current, Unknown, Update required

- By clicking **Delete**, the selected drawings are deleted without any warning/confirmation.
- **Revision** adds a revision mark to current drawing(s) or drawings that require updating. This mark is listed on the drawing and users see the history of the drawings. The “Add revision mark” dialog box appears. Click **Add**. On the **Backup** tab specify revision control options: make a backup of the document before it is given a revision mark; specify a folder for the backup (relative to the document to be updated or absolute).

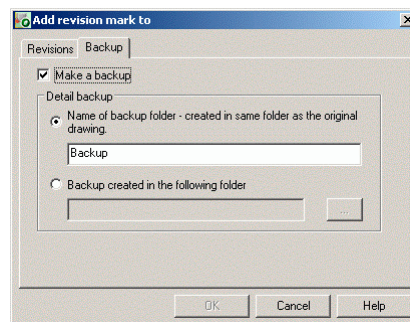


Figure 259: “Add revision mark” dialog box

If several detailed drawings must get a revision mark specify an index for all details or use the next available one.

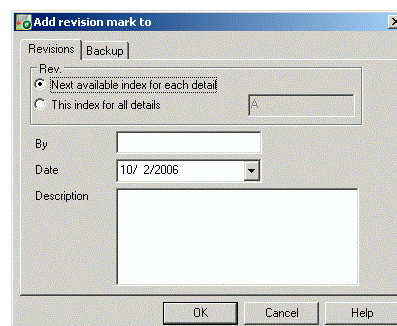


Figure 260: “Add revision mark” dialog box - multiple selection

Update revision updates a drawing to suit the model and automatically assigns a revision number or letter.

- **Issue** flags the drawing and sets the issue date and time. Checkmark boxes appear in front of the issued drawings and the date and time appear in their properties.

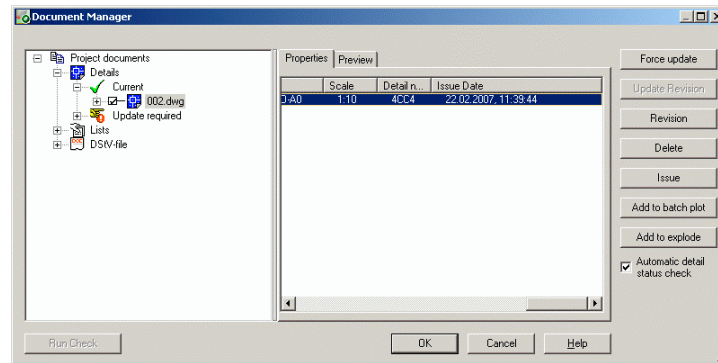


Figure 261: Document Manager - issue date

- The selected issued drawings can be un-issued by clicking the checkmark box. A confirmation message appears.

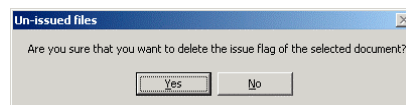


Figure 262: Confirmation message

A batch explode can be run for all selected drawings. Select the drawing(s) and click **Add to explode**. The drawings are copied to a new branch called "Batch explode".

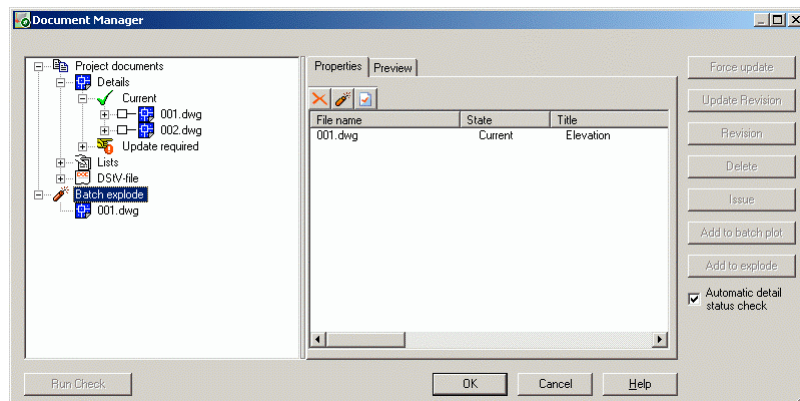


Figure 263: Document Manager - batch explode

Different profiles can be defined for detail explode settings using the Detail Explode Tool. Configure the layer name, the color, and the line type to use for beams, plates, bolt holes, etc. This user configuration is stored for reuse in future projects.

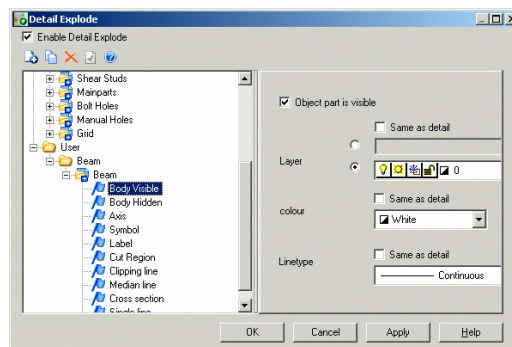


Figure 264: Detail explode

Close the Document Manager by clicking **OK**.

Registering/canceling drawing from model

- The “Register drawing in model”  and “Cancel drawing from model”  tools on the **Sub – Drawing management** flyout (Advance **Standard** toolbar and Advance **Numbering, drawing** toolbar) register or cancel drawings from the model.

 **Example:** Renaming drawings (DWGs) / moving to another folder

- Rename / move drawings with Windows Explorer.
- Re-register the renamed / moved detail drawings in the model.

Drawing style management

- The  tool on the **Sub – Drawing management** flyout (Advance **Numbering, drawing** toolbar) calls the **Drawing Style Manager** to create custom drawing styles (refer to the *Drawing Style Manager Guide*).

Drawing layout

- The “Define drawing layout” tool on the **Sub – Drawing management** flyout (Advance **Numbering, drawing** toolbar) defines settings for individual prototype files so that the arrangement of the details are determined when using drawing processes.

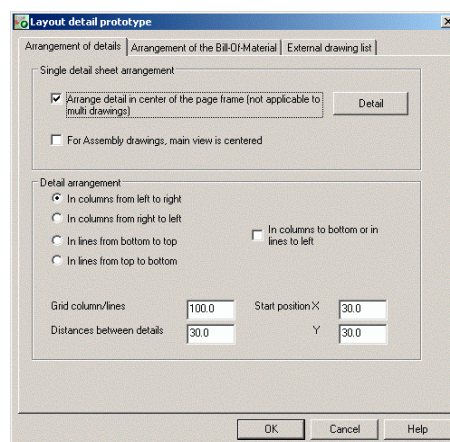


Figure 265: Drawing layout

Refer to the *Processes* section of this chapter.

Creating/changing project information

Project information can be set for the model and saved. The project information is used for the automatic filling of a title block in the drawing creation (and BOMs).

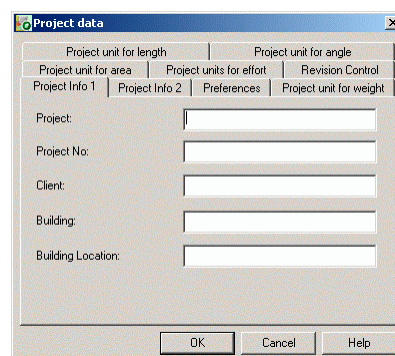


Figure 266: Project information

On the **Project info 2** tab, additional information such as contractor, designer, detailer, and date are entered.

On the **Preferences** tab, a country can be set for preferred section sizes, bolts types, etc., which are preset in the **Management Tool**.

The **Project unit** tabs contain options for all standard AutoCAD® units, including a field for the *precision*.

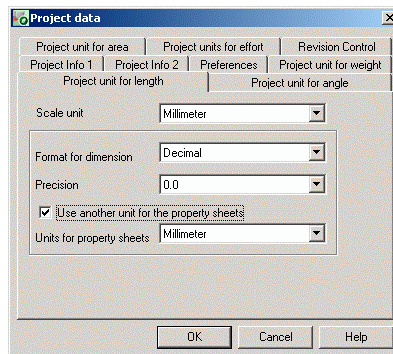


Figure 267: Project units

Filling in a title block

The title block is automatically placed and filled with project information in the linked drawing.

To automatically fill the title block it must have a particular name with set attributes. The name of the AutoCAD® block for the title block (in your prototype drawing) must be: HYPERSTEELPAGEHEADER.

Changing a title block

To modify title blocks:

- Open a file (e.g., **Advancepagehead-a3-0.dwg**) in the **AdvanceSteel\Shared\Support\Frames** folder and save it with a new name.

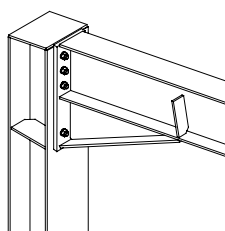
Bauherr: CLIENT			
Enterprise: BUILDING BUILDING_LOCATION			
GRAITEC		PROJECT	
91573 BIEVRES		POS_NUM	
Burospace 17	TB	None	Date
17.01.2017 10:00:00	Auftraggeber	CONTRACTOR	DATE_ORDER
17.01.2017 10:00:00	constructor	DESIGNER	DATE_CONSTR
17.01.2017 10:00:00	Drawn by	DETAILER	DATE_DRAW
17.01.2017 10:00:00	Controlle		SCALE
Plot von:	Model name: HEDEL.DWG	Run-Ne:	
PLOT_DATE	Material: MATERIAL	DRAWING_NO	

Figure 268: GRAITEC - title block

- Change the title block as desired.
- UK users may wish to start with a UK title block (e.g., UK-TITLE-GA.dwg) when creating title blocks.

Node details

With these styles, clippings can be created from the model (e.g., nodes of a frame).



Detail 2-1
M 1:10

Figure 269: Node detail

The view direction depends on the UCS and runs against the positive Z-direction. The size of a detail clipping is selected during the course of the command using two points. Node representations can be created with or without view depth.



Pay attention to the setting for **Ortho** and **Object snap** - off.

The default scale is 1:10 and dimensions are not automatically made. The clipping is switched off.

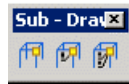


Figure 270: **Sub – Drawing node** flyout

- The **node 1** detail is not labeled.
- Alternatively, **node 2** includes labeling with main part/single part mark, number of the individual elements, and leader lines.
- With **node 3** the profile object name (e.g., IPE 270, BI 10) is also included.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

Elevations

With elevations, geometric clipping is created from the model. The view direction depends on the UCS and runs against the positive Z-direction. The views are displayed with a default scale of 1:20. The shortened representation (clipping) is activated in the X- and Y-directions. The representation has a default view depth in 2D. ACIS volume solids can also be selected.

Other types of intersection drawing styles are available and are described in the *Advance online Help*.

There are five elevation drawing styles available (i.e., **elevation 1** to **5**) on the **drawing elevation** flyout along with a drawing style for an anchor bolt plan.



Figure 271: **Sub – Drawing elevation** flyout

- The first drawing style for elevations (**elevation 1**) presents a clipping with a view depth of +/- 500 going out from the origin of the current UCS without dimensions or labels. The entire model and solids are displayed.
- In **elevation 2**, the Advance elements are labeled with their main part marks.
- **Elevation 3** labels the main parts with profile names and main part mark (e.g., section name IPE 270 2).
- **Elevation 4** creates grid and beam system line dimensioning, in addition to the main part labeling and part marks.
- **Elevation 5** includes a clipping with an unlimited depth in the front and a depth of 0 to the back.
- The **anchoring plan** has a clipping depth of 5 in the front and unlimited to the back. Grids and bolts are dimensioned while main parts and bolts are labeled.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

For the Czech Republic, the **Sub – Drawing intersection** toolbar is displayed.

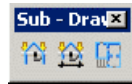


Figure 272: **Sub – Drawing intersection** flyout

Other types of elevation drawing styles are available and are described in *Advance online Help*.

Single part drawings

The **Drawing single parts** flyout contains drawing styles to create shop drawings of single parts. The view direction does not depend on the UCS but corresponds to the internal coordinate system of the selected element. The scale is 1:10. The details are automatically dimensioned and labeled. The clipping is only activated for the X-direction.

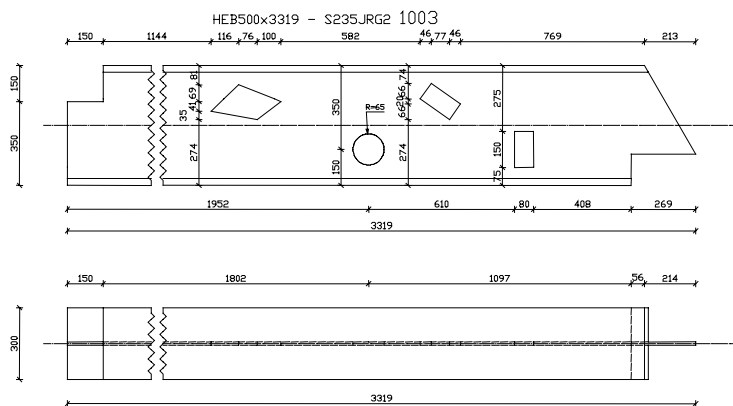


Figure 273: Single part – beam front, top

The drawing styles for single parts differ, on the whole, by the view orientation. The following view orientations are available:



Figure 274: **Sub – Drawing Single Part** flyout

- Single part – beam front (available for Czech Republic)
- Single part – beam front, top
- Single part – beam front, top, bottom (available for Czech Republic)
- Single part – beam front, bottom
- Single part - beam selection of views. Up to six views of an element can be selected by clicking the contours displayed in green in the model.
- A drawing style for single part representation of curved beams
- A drawing style for single part representation of plates (available for Czech Republic)

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

For USA, the following drawing style for single part representation of folded plates is available:

Button	Function
	Single part – Folded plate

Tie beam

The drawing style on the **Drawing tie beams** flyout creates shop drawings for non-vertical main parts. The view direction does not depend on the UCS but corresponds to the internal coordinate system of the selected main part. The labeled and dimensioned views are displayed with a default scale of 1:10. The clipping is activated in the X-direction.

The drawing styles for tie beams differ, on the whole, by the assigned view orientation.



Figure 275: **Sub – Drawing Tie Beam** flyout

- Tie beam – front
- Tie beam – front, top (available for Czech Republic)
- Tie beam – front, top, bottom (available for Czech Republic)
- Tie beam – front, bottom
- In the “Tie beam – selection of views” drawing style, up to 6 views can be selected. Click the contours displayed in green in the model.
- In the “Tie beam – with environment” drawing style, click one or more connected elements to display. Main parts are displayed from the front and top, end plates from inside, and additional intersections from the right.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

Columns

With the drawing style for columns, shop drawings for vertical elements can be created. The view direction does not depend on the UCS but corresponds to the internal coordinate system of the selected main part. Columns can be detailed similarly to tie beams but turned 90 degrees (i.e., vertically).

The default scale is 1:10 and automatic dimensioning and labeling is performed. The clipping is activated in the X-direction.

The drawing styles for columns differ, on the whole, by the assigned view orientation.



Figure 276: **Sub – Drawing Column** flyout

- Column – front
- Column – front, left
- Column – front, left, right (available for Czech Republic)
- Column – front, right (available for Czech Republic)
- In the “Column – selection of views” drawing style, up to 6 views can be selected. Click the contours displayed in green in the model.
- In the “Column – with environment” drawing style, select one or more connected elements to display. Main parts are displayed from the front and top, end plates from inside, and additional intersections from the right.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

Assembly drawings

These styles create assembly shop drawings that depend on the current UCS orientation. The parts to be detailed should be connected with shop connections. The scale is 1:10 and the details are automatically dimensioned and labeled. The clipping is activated for the X-direction. Intersections at given places can be created by selecting the green contours that are always perpendicular to the main part.

On the **Sub – Drawing main part** flyout, three styles are implemented.



Figure 277: **Sub – Drawing Main Part** flyout

- “Main part – UCS” displays a main part so that it is orientated in the drawing as in the model (see example above). Intersections can be selected with the green contours in the model; the view orientation runs against the positive Z-axis of the UCS.
- The “Intersection at UCS with selected parts” style creates any cross section of the selected part at the current UCS labeled with part marks and leader lines.
- The “Main part – curved beam” style creates a view for a curved beam with dimensioning and labeling. The representation does not depend on the current UCS.
- One more drawing style is available for Czech Republic: “Main part - weldment section UCS single”.

This toolbar is only available for Canada, Benelux, Germany, Poland, Russia, USA, and International.

*To obtain an automatic internal view of the end plate or cover plates these must have the Model Role cover, base, or end plate (Advance **Properties**, **Note** tab).*

Special structural parts

The following styles are available on the **Sub – Drawing Special Structural Parts** flyout:



Figure 278: **Sub – Drawing Special Structural Parts** flyout

- Compression pipe
- Cross bracing
- Tube development – template
- Tube development – template with tube
- Railing (*not available for USA*)
- Picket Railing (*not available for USA*)
- Stringer (*not available for USA*)
- ZED purlin
- Static plan, complete (*not available for USA and Canada*)
- Static plan, clipping (*not available for USA and Canada*).

This toolbar is only available for Benelux, Germany, Poland, Russia, and International.

These styles are for elements that need a special representation or work with several Model Roles. The **Model Role** property is created for each element on the **Note** tab of its dialog box and is interpreted by the program for functions such as drawing creation.

The “Static plan complete” and “Static plan clipping” drawing styles create model overviews with system lines.

For France, the **Sub – Drawing Special Structural Parts** flyout contains three more drawing styles:



Figure 279: **Sub – Drawing Special Structural Parts** flyout

- Single part – beam development
- Single part – curved beam
- Main part – curved beam

For USA, the **Sub – Drawing Stairs Railings** flyout contains drawing styles for stairs and railings.



Figure 280: **Sub – Drawing Stairs Railings** flyout

Special structural parts

The view direction does not depend on the UCS but corresponds to the internal coordinate system of the selected main part. The default scale is 1:10 and automatic dimensioning and labeling is performed. The clipping is activated in the X-direction.

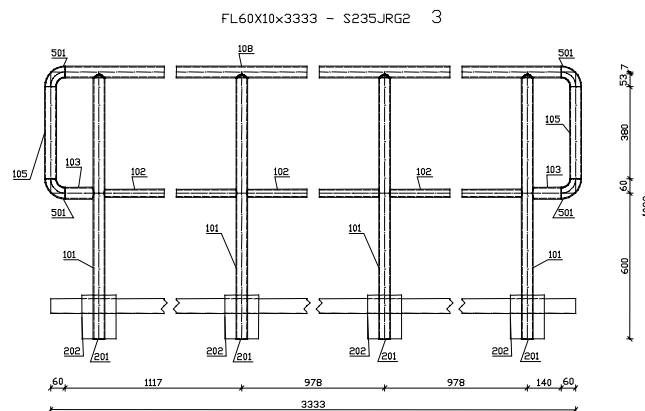


Figure 281: Railing straight

This result can only be achieved if the following Model Roles are used in the model (on the **Note** tab): Handrail, newel, post, railing strip, railing end wing, pipe bend, and base plate.

Alternative drawing styles

Alternative drawing toolbars, specific for UK and France, contain many other general arrangement drawing styles.

All styles depend on the current UCS and the view is against the positive Z-direction. The differences between the styles are the presentation, labeling, and dimensioning of the various objects and are controlled by the assigned **model role** in the model.

The flyouts contain various arrangement drawing styles and include:

- Foundation drawings - The details are automatically labeled and the clipping is switched off.
 - On the **Sub – Foundation drawing** flyout, the following styles are available:



Figure 282: Sub – Foundation drawings flyout

Button	Function
	Stanchion Layout - All
	Stanchion Layout - Pickbox
	Base detail - Pickbox
	Base detail - Box600 - This drawing style is available only for the UK

- Floor plan drawings contain drawing styles for plan views from the model. The views are displayed in a default scale of 1:50 and they are automatically dimensioned and labeled.



Figure 283: Sub – Floor plan drawings flyout

There are two floor plan drawing styles available on the **Floor plan drawings** flyout:

Button	Function
	Floor plan - hot rolled - all
	Floor plan - Hot rolled - Pickbox: the X/Y-viewport is selected

For the UK, two more drawing styles are available:

Button	Function
	Floor plan - cold rolled - all
	Floor plan - cold rolled - PickBox

- Roof plan drawings - The views are displayed in a default scale of 1:50 and the details are automatically dimensioned and labeled. The clipping is switched off.



Figure 284: Sub – Roof Plan Drawings flyout

The following drawing styles are available on the **Roof plan drawings** flyout:

Button	Function
	Roof plan - hot rolled - all
	Roof plan - hot rolled – Pickbox
	Roof plan - Cold rolled - All
	Roof plan - Cold rolled - Pickbox

- Elevation drawings contain drawing styles for elevation drawings. The views are displayed in a default scale of 1:50 and the details are automatically dimensioned and labeled.



Figure 285: Sub – Elevation drawings flyout

The following drawing styles are available on the **Elevation drawings** flyout:

Button	Function
	Elevation - Side - Hot Rolled - All
	Elevation - Side - Hot Rolled - Pickbox
	Elevation - Gable - Hot Rolled - All
	Elevation - Gable - Hot Rolled - Pickbox

For the UK, two more drawing styles are available:

Button	Function
	Elevation - Cold rolled - All
	Elevation - Cold rolled - Pickbox

- Section drawings contain drawing styles for sections from the model. The views are automatically dimensioned and labeled.



Figure 286: Sub – Section drawings flyout

There are three section drawing styles available on the **Section drawings** flyout

Button	Function
	Section - all - X/Y viewport - automatic
	Section - Pickbox: the X/Y - viewport is selected
	Section - select: only selected elements are detailed
	Section - box1000 - This drawing style is available only for the UK country Add-In

- 3D drawings contain drawing styles for 3D drawings of the entire model or of the selected part. The clipping is switched off and the default scale is 1:50.

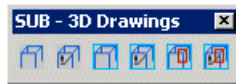


Figure 287: Sub – 3D Drawings

The following drawing styles are available:

Button	Function
	3D view - All - no labels
	3D view - All - with labels
	3D view - Select - no labels
	3D view - Select - with labels
	3D - View - Select+Pickbox - no labels
	3D - View - Select+Pickbox - with labels

Processes

Advance drawing processes are used to accelerate drawing creation by automatically assigning drawing styles and layouts to the selected parts.

A process includes a number of sub processes: selection of parts, sorting, drawing style selection, rules for arranging details on the sheet, and rules to attach new sheets.

The default processes are located on the **Numbering, drawing** toolbar according to the formats A0, A1, A2, A3, and A4 and are accessible on the flyouts.



Figure 288: Numbering, drawing toolbar, processes

For each format (e.g., A0) the same series of selection groups are available. The selection groups shown for the format A0 recur for the formats A1, A2, A3, and A4.

Drawing processes

The following flyout/tools are available for each drawing size format:



Figure 289: Processes - selection per format

- All single parts
- All single parts plate
- All single parts beam
- All main parts
- All main parts plate
- All main parts beam
- All main parts with attached parts

- All cameras
- Selected single parts
- Selected single parts plate
- Selected single parts beam
- Selected main parts
- Selected main parts plate
- Selected main parts beam
- Selected main parts with attached parts
- Selected cameras.

For each different part mark, a detail is created so that duplicate drawings are **not** created.

Within a process, a specific sorting can be entered in the dialog box.

Advance offers specific drawing styles that support fully automated drawing creation. These styles differ mainly from other styles in that the title text includes the number of pieces in the model, or in the case of single parts, the number of pieces on the main part are also displayed. Intersections are created automatically.

The drawing creation is managed by the Document Manager, which lists, displays, and, if required, updates, and deletes drawings.

 **Example:** Automatic detailing of all main parts in a model with attached parts (A0)

- After the model has been checked and numbered, start the automated detailing process for all main parts (including their attached parts) with the “Main parts with attached parts” tool on the **A0** toolbar (flyout) of the Advance **Numbering, drawing** toolbar. The “Process properties” dialog box appears.

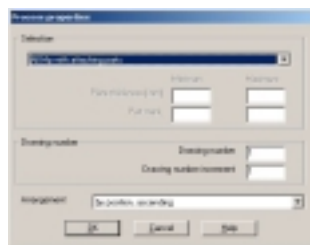


Figure 290: **Processes:** “Process properties” dialog box

- Set the drawing number options for the drawings created by the process: the first number and the increment value.
- Besides the **selection**, information for **arrangement** can be entered. Choose a selection and press **OK**.

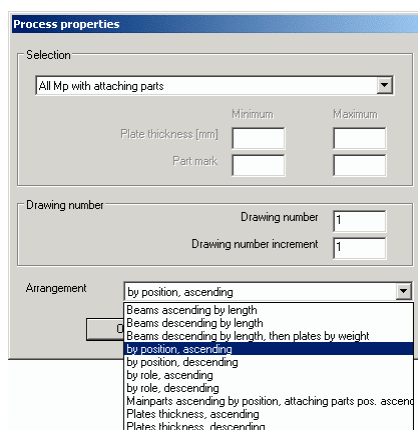


Figure 291: Drawing processes, sorting

The details are created and placed in the drawing.

- The detail drawings for all beams are created automatically. In the **Document Manager** each linked beam detail is listed and attached to the same DWG (e.g., ...\\Program\\GRAITEC\\AdvanceSteel\\Work\\Projectname\\details\\A0-Detail00001.dwg). When the drawing is full a new drawing A0 detail 00002 is attached.
- In the **Preview** field of the Document Manager detail drawings are previewed.
- Open a drawing (e.g., **A0-Details 00001.dwg**) using **File > Open** in the top menu bar of AutoCAD®. Individual details will be seen together on the drawing and the drawings can be plotted or edited.
- Close the DWG and go back to the model.

With the A0, A1, A2 formats, many details are placed on the sheet until it is full. Then a new drawing is started. With the formats A3 and A4 formats, the details are placed on individual sheets.

The details are automatically placed on the sheet by the process. Before starting the process the arrangement direction can be changed on the sheet. This must be done in the prototype drawing using the "Define drawing layout" tool on the **Sub – Drawing management** flyout.

Cameras

The creation of drawing views and elevations has considerably improved and made faster by the Advance camera design.

The camera is an object representing the location, angle, and depth of a view, allowing individual focus for automated detail drawing creation. With the camera's model views, elevations or overview settings and node details can be presented, named, and saved.

The cameras can also be used in processes for automatic detail creation. Two of the automated detailing processes in Advance refer to cameras:

- Create detail drawings from all cameras
- Create detail drawings of selected cameras

Tools for defining different cameras are accessible on the **Sub – Create cameras** flyout of the Advance **Standard** toolbar.



Figure 292: **Sub – Create cameras** flyout

- Create camera (UCS): With this tool, elevations / views are set in the model for use in detailing.
- Create camera(s) at node: With the camera at node details, six cameras are set at joints. These correspond to the six view directions of the details and can be set for use in detailing.

Creating a camera (UCS)

A camera is created using the current user coordinate system. The view direction runs against the Z-axis and the size of the clipping is determined with a detail box with X/Y/Z-depth.

- To define a camera of this type, determine the view direction on the object by setting a corresponding UCS.
- Next, enter an origin for the camera.
- The "Camera" dialog box appears in which a **camera type** (i.e., no, node, overview) can be entered. This information is important so that the camera process can distinguish between overview and node.

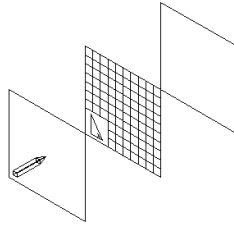


Figure 293: Create camera (UCS)

The grid is stretched around the set origin, which is at the center, and the size of the grid is determined by the values in the detail box. The positive and negative Z-values refer to the clipping distance in front and behind the grid on the UCS plane. The pencil-top shows the view direction against the positive Z-axis.

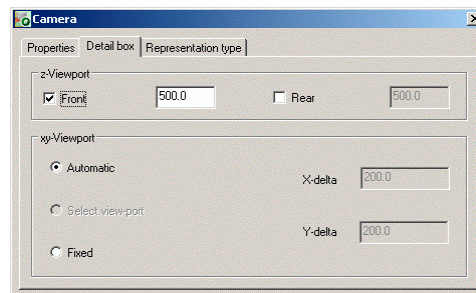


Figure 294: "Camera" dialog box

Creating a camera (s) at a node

The node camera tool is specifically modified for the Advance joints and determines the orientation and the size of node views.

For the detail presentation of a node, one or more cameras with different view directions are created at a connection object. The user has a choice between six cameras with different colors.

Note: The connection object (gray box) must be visible so that it can be identified in the command course.

- To define one or more node cameras, display the connection object(s) and then start the "Create camera(s) at node" tool.
- Six different colored cameras appear. They are created with a standard detail box size. The size of the box and other properties can be changed later in the **Properties** dialog box of the created camera.
- Select one or more cameras on the screen and press **Enter**. The selected cameras are displayed in blue and the others disappear.

The corresponding drawing processes can interpret the created cameras.

Alternative drawing processes

Alternative "Drawings" toolbars contain many additional drawing processes for each country.

 **Examples:**

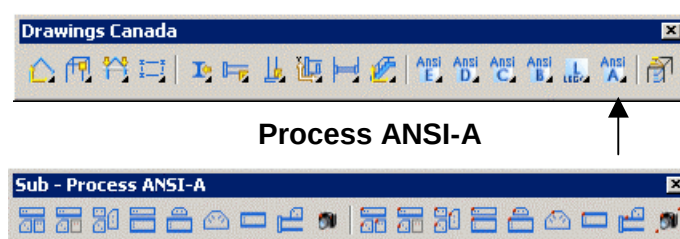


Figure 295: Sub – Process ANSI-A flyout

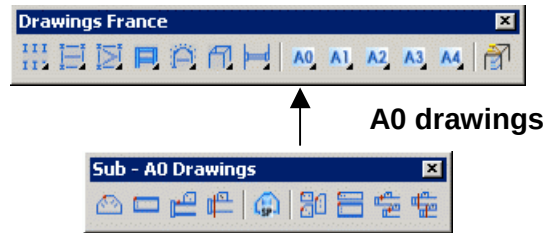


Figure 296: **Sub – A0 Drawings** flyout

All processes are arranged by sheet size and a flyout for automatic sheet size includes various single part (fitting) and assembly drawings. There are options to create one detail per sheet or to fill a larger sheet with many details. The processes use the drawing styles and prototypes from the corresponding country.

Labeling and dimensioning

In Advance, the 3D model and the associated drawings are linked to each other. It is therefore possible to create additional dimensions, labels, and level symbols in the linked drawings using the model information.

The following tools are available on the **Sub – Labels & Dimensions** flyout of the **Numbering, drawing** toolbar.



Figure 297: **Sub – Label & Dimensions** toolbar (flyout)

- Create a weld symbol, label, level symbol
- Create a slope symbol
- Create a dimension point, delete a dimension point
- Dimension chain (flyout)
- Change a dimension style, change a text style
- Hold menu

Drawing cleanup

First, open the drawing for editing. This can be done on the upper menu bar of AutoCAD® using **File > Open** and entering the file name (e.g., ...\\Program\\GRAITEC\\AdvanceSteel\\Work\\Details\\Hall1**det1.dwg**). In the preview of the **Document Manager**, changes cannot be made.

The following tools are available on the **Finishing** flyout of the **Numbering, drawing** toolbar:



Figure 298: Toolbar (flyout) **Sub – Finishing**

- Update a detail
- Rearrange all details
- Rearrange all views in a detail
- Change a detail
- Change a prototype file
- Update the page header
- Revision mark, revision table
- Move a view

-
- Delete a view
 - Create a view / section
 - Create a BOM, update a BOM
 - Explode a detail
 - Detail explode settings
 - Toggle the display of green frames around details
 - Hold menu.

Appendix



HSBasis

Command	Description
AstModelBrowser	Access the Model Browser
AstM4CommProjectData	Definition of the project information, units, and preferences
AstM5PrefixConfigurationExplorer	Prefix configuration
AstDocCheck	Start the Document Manager
AstM4About	Displays the Advance Steel About dialog box
AstM4BeamContourNotchCircle1Dig	Beam processing - Circular contour by center point
AstM4BeamContourNotchCircle2Dig	Beam processing - Circular contour by two points
AstM4BeamContourNotchPolygon	Beam processing - Polygonal contour
AstM4BeamContourNotchRect1Dig	Beam processing - Rectangular contour cut
AstM4BeamContourNotchRect2Dig	Beam processing - Rectangular contour by two points
AstM4Cecs	UCS at object
AstM4ChangeBindingLength	Calculate grip length
AstM4changerep	Change the representation type
AstM4CommAstorToCIS2	Export CIS/2
AstM4CommAstorToIFC2x	Export IFC
AstM5CommAstorToSDNF	Export SDNF
AstM8CommAstorToPSS	Export PSS
AstM4CommAudit	Extend the AutoCAD® audit checking to include Advance Steel objects
AstM4CommBatchErrorsMarkObject	Display the checking results
AstM4CommBatchErrorsPrint	Display the checking results again
AstM4CommCIS2ToAstor	Import CIS/2
AstM4CommClipCrossSection	Create a beam cross section
AstM4CommCrBeamBent	Create a curved beam
AstM4CommCrBeamPoly	Convert a polyline to a polybeam
AstM4CommCreateAssemblyMainPart	Create an assembly part
AstM4CommCreateConnection	
_AstM4CommCreateConnection INCL	Connection, add objects
_AstM4CommCreateConnection EXCL	Connections, disconnect objects
AstM4CommCreateMainPart	Create main part of assembly
AstM4CommCrConicalFoldedPlate	Create a conical folded plate
AstM4CommCrTwistedFoldedPlate	Create a twisted folded plate
AstM4CommCutPlateToPlate	Miter plates
AstM4CommDSTVStaticIn	
_AstM4CommDSTVStaticIn NORMAL	Statics read
_AstM4CommDSTVStaticIn INVERT	Statics read, WCS inverted
AstM4CommExplodeCombiProfile	Disassemble compound sections
AstM4CommGotoMark	Zoom to highlighted object
AstM4CommGridAxes	Create a group of four axes
AstM4CommGridAxesSequences	Create a group of axes specifying the distances

Command	Description
AstM4CommGridDeleteAxis	Delete the selected axis from the grid
AstM4CommGridExtend	Extend grid axis
AstM4CommGridInsertSequence	Create a group of axes after the selected axis
AstM4CommGridRectangle	Create a building grid
AstM4CommGridSingleAxis	Create a single axis
AstM4CommGridSingleCircleAxis	Create a circular grid with single axis
AstM4CommGridTrimm	Trims the building grid after a boundary object
AstM4GRTCLiveUpdate	Check for updates
AstM4CommHoleAssignData	Change holes
AstM4CommHSVersion	Display version and build number
AstM4CommIFC2xToAstor	Import IFC
AstM4CommInsertPlateVertex	Add a corner to a polygonal plate
AstM4CommLine2Beam	Convert a line to a beam
AstM4CommMarkConnMeans	Display the connection means
AstM4CommMarkMainPart	Mark the main part of an assembly
AstM4CommMarkObject	Mark an object
AstM4CommMarkSelAdd	Add selected objects to marking
AstM4CommMarkSelSub	Remove selected objects from marking
AstM4CommModelSelection	Search filter
AstM4CommMove2Plane	Move bolt or hole pattern
AstM4CommPlatesShrink	Shrink or expand a polyplate
AstM4CommPlate2Pline	Convert a plate to a polyline
AstM4CommPLine2Plate	Create a plate at polyline
AstM4CommRemovePlateVertex	Remove a corner of a polygonal plate
AstM5CommRepPSS2Astor	Import PSS
AstM4CommSaveWithFaceProxy	Save with area Proxy-Graphics
AstM4CommSaveWithLineProxy	Save with line Proxy-Graphics
AstM4CommSaveWithoutProxy	Save without Proxy-Graphics
AstM5CommSDNFToAstor	Import SDNF
AstM4CommSelectAllDscObjects	Select all Advance Steel objects
AstM4CommSelectMarkedObjects	Select highlighted objects
AstM4CommSeparateScrews	Split a bolt group
AstM4CommShowLocalCS	Define a coordinate system
AstM4CommShowMarkings	Search highlighted objects
AstM4CommSpecialPartBlock	Advance Steel special part
AstM4CommStandaloneParts	Mark loose parts
AstM4CommStartMT	Start the Management Tools
AstM4CommStructAsJoint	Generic command for creating a structural element
AstM4CommStructAsJointVB	Generic command for creating a structural element
AstM4CommUnmarkObjects	Clear highlighted objects
AstM4CommWrite2DXF	Save as DXF
AstM4ConnectAxes	Connect single axes into one grid

Command	Description
AstM4CrAxis	Create single axis
AstM4CrBeamByClass	Generic command for creating straight beams with specific section class
<i>_astm4crbeambyclass I</i>	Create a beam with a rolled I section
<i>_astm4crbeambyclass U</i>	Create a beam with channel section
<i>_astm4crbeambyclass W</i>	Create a beam with angle section
<i>_astm4crbeambyclass T</i>	Create a beam with T-section
<i>_astm4crbeambyclass O</i>	Create a beam with Circular hollow section
<i>_astm4crbeambyclass Z</i>	Create a beam with Z section
<i>_astm4crbeambyclass F</i>	Create a beam with flat section
<i>_astm4crbeambyclass D</i>	Create a round bar
<i>_astm4crbeambyclass Q</i>	Create a square bar
<i>_astm4crbeambyclass H</i>	Create a square hollow section
<i>_astm4crbeambyclass C</i>	Create a cold rolled section
<i>_astm4crbeambyclass A</i>	Create other section
AstM4CrBeamContourNotch	Create a contour cope on a beam
AstM4CrBeamMFC	Create a beam
AstM4CrBeamMiterCut	Miter cut for two beams
AstM4CrBeamNotch	Cope at a beam end
AstM4CrBeamShort	Create a beam shorten
AstM4CrBeamShortUCS	Create a beam shorten at the UCS
AstM4CrCompBeam	Old compound section
AstM4CrCompoundBeam	
<i>_AstM4CrCompoundBeam WeldedISymmetric Default</i>	Create a welded beam I symmetric
<i>_AstM4CrCompoundBeam WeldedIASymmetric Default</i>	Create a welded beam I asymmetric
<i>_AstM4CrCompoundBeam WeldedHollow Default</i>	Create a welded beam hollow
<i>_AstM4CrCompoundBeam WeldedT Default</i>	Create a welded beam T
<i>_AstM4CrCompoundBeam WeldedColumn Default</i>	Create a welded beam, column
<i>_AstM4CrConByVb TaperedBeam</i>	Create a tapered welded beam
AstM4Create1DigChamfer	Create a chamfer with one point
AstM4Create2DigChamfer	Create a chamfer with two points
AstM4CreateConcavFillet	Create a concave corner finish
AstM4CreateConvexFillet	Create a convex corner finish
AstM4CreatePolyPlate	Create a polygon plate by specifying the corner points
AstM4CreateRect1DigPlate	Create a rectangular plate by specifying the center point
AstM4CreateRect2DigPlate	Create a rectangular plate by specifying two points
AstM4CrRaster	Create a building grid
AstM4CrSequPar	Create a parallel axis group
AstM4CrStruct3GFrameSymm	Create a 3-hinged frame, symmetric
AstM4CrStructMultiFrame	Create purlins
<i>_AstM4CrStructMultiFrame DEF</i>	

Command	Description
AstM4CrStructMultiTowerFrame _AstM4CrStructMultiTowerFrame DEF	Create pylons
AstM4CrStructSingleFrameBracing _AstM4CrStructSingleFrameBracing DEF	Create a single-span bracing
AstM4CrStructSlopeFrame _AstM4CrStructSlopeFrame DEF	Create a half truss
AstM4Database	Statics write total
AstM4FoldedPlateCheckUnwinding	Check unfolding
AstM4FoldedPlateExtend	Create a folded plate
AstM4FoldedPlateSetMain	Set folded plate main object
AstM4HeightKote	Create a level symbol
AstM4HelpCommand	Access help
AstM4NecessaryViewsRules	Necessary views configuration
AstM4PlateContourNotch	
_AstM4PlateContourNotch RECTANGLE	Plate processing UCS - rectangular contour by center
_AstM4PlateContourNotch RECTANGLE2DIG	Plate processing UCS - rectangular contour by two points
_AstM4PlateContourNotch CIRCLE	Plate processing UCS - circular contour by center
_AstM4PlateContourNotch CIRCLE2DIG	Plate processing UCS - circular contour by two points
_AstM4PlateContourNotch POLYGONE	Plate processing UCS - polygonal contour
AstM4PlateInnerContour	
_AstM4PlateInnerContour RECTANGLE	Plate processing - rectangular contour by center
_AstM4PlateInnerContour RECTANGLE2DIG	Plate processing - rectangular contour by two points
_AstM4PlateInnerContour CIRCLE	Plate processing - circular contour by center
_AstM4PlateInnerContour CIRCLE2DIG	Plate processing - circular contour by two points
_AstM4PlateInnerContour POLYGONE	Plate processing - polygonal contour
AstM4PSSDirect	Export Steelfab
AstM4RemoveHyperSteel	Delete all AS objects from external dwg
AstM4ReopenDatabase	Update defaults
AstM4SelectObjectByName	Select Advance Steel objects
AstM4SsetDatabase	Statics write selection
AstM4SwitchReprMode	Hide or show objects
AstM4UndoSwitch	Disable or enable UNDO
AstM4WeldSeam	Generic command for the creation of welds
_AstM4WeldSeam POINT	Create a weld point
_AstM4WeldSeam STRAIGHT	Create a weld line
AstM5BoltsHolesSwitchConnector	Switch to shear studs
AstM5BoltsHolesSwitchHoles	Switch to bolts / holes / shear studs
AstM5CommBoltsHolesAllSides	Create bolt or hole rectangular pattern set by 2 points
AstM5CommBoltsHolesCircle	Create bolt or hole circular pattern set by the center point
AstM5CommBoltsHolesLLSides	Create a rectangular pattern set by the corner point
AstM5CommBoltsHolesRect	Create a rectangular pattern set by the center point

Command	Description
AstM5CommCamera	Create a camera, UCS
AstM5CommCameraNode	Create camera(s) at node
AstM5CommEcsAtPickPoint	UCS at curved beam
AstM5CommEqualPartsOnePartOutOf	Select objects with different part marks
AstM5CommEqualPartsWithoutNum	Mark parts without a mark
AstM5CommSemanticCheck	Steel construction technical checking
AstM5DetSetArranger	Define a drawing layout
AstM5EqualPartsChangePrefix	Assign or change the prefix
AstM5EqualPartsCheck	Identify the objects with duplicate part marks
AstM5EqualPartsCheckGroups	Check for duplicate part marks
AstM5EqualPartsDelNumber	Delete part marks
AstM5EqualPartsDetect	Numbering
AstM5EqualPartsMarkSame	Display objects with identical part marks
AstM5EqualPartsReset	Ungroup identical parts
AstM5EqualPartsShowDiff	Differences between two objects (pos check)
AstM5EqualPartsSwitchMain	Switch to main part numbering
AstM5EqualPartsSwitchSingle	Switch to single part numbering
AstM4LoadSTEP	Write SDDP
AstM4CrBisectorCS	Set the UCS to the bisecting line
AstM4CommExplodeToAcis	Explode object to ACIS
AstM4CommBevelWeldingPreparation	Weld seam preparation, chamfer
AstM4CommFilletWeldingPreparation	Weld seam preparation, fillet
AstM4CutPlateAtUcs	Cut plate at UCS
AstM4CutPlateAtPlate	Cut plate at an other plate
AstM4CommUserSectionBeam	Create a folded beam
AstM4CommBillOfMaterial	Create lists
AstM4CommStartConcrete	Start Advance Concrete
AstM4UCSZFilter	Switch for working 2D / 3D (for snap points)
AstProcessesSuite	Drawing process suite
ASTransform	Select the transformation type
ASTransformAndConnect	Advance Copy tools

HSDetailing

Command	Description
AstM4DetRegist	Register a detail to the model
AstM4DetDeregist	Deregister a detail from a model

HSConnection

Command	Description
AstM4CrConByRule	Generic command for creating a joint
<i>AstM4CrConByRule 1 ParamFit</i>	Cope parametric rule
<i>AstM4CrConByRule 1 Saw_1</i>	Cut at object – rule
<i>AstM4CrConByRule 1 SawAngle_Asym</i>	Cut variable – rule

Command	Description
AstM4CrConByVB	General command for creating a joint
AstM4CreateGussetPlate	Create a gusset plate
AstM4TransferCons	Create using template (intelligent connections)
AstM4TransferParams	Transfer properties (intelligent connections)
AstM4UpdateConOb	
_AstM4UpdateConOb UPDATE	Update a joint
_AstM4UpdateConOb DELETE	Delete a joint
AstM4UserTemplateConn	
_AstM4UserTemplateConn _c	Create a template connection
_AstM4UserTemplateConn _i	Import a template connection

HSExtended

Command	Description
AstM4CrBeamElemContNotch	Element contour - exact cross section
AstM4PlateElementContourNotch	Plate processing by element contour
AstM4CommMergeBeams	CONVERT sections
AstM4CommMergePlates	CONVERT plates
AstM4CommSeparateBeams	Split beams
AstM4CommSeparatePlates	Split plates at lines
AstM4CommSeparatePlate	Split plates using two points
AstM4CommDeepExplode	Explode a joint and all sub-joints

HSCollision

Command	Description
AstM4Balancepoint	Center of gravity and total weight calculation
AstM4CheckCollision	Clash check between selected objects

HSDetailingBasis

Command	Description
AstM4CommDetRevisionTable	Revision table
AstM4DetChange	Change detail
AstM4CommDetAddHorLinearDim	Linear horizontal dimension
AstM4CommDetAddSlopeLinearDim	Linear sloped dimension
AstM4CommDetAddVertLinearDim	Linear vertical dimension
AstM4CommDetArrangeDetails	Rearrange all details
AstM4CommDetArrangeViews	Rearrange all views in detail
AstM4CommDetUpdateDetail	Update detail
AstM4DetManualCut	Create view / section
AstM4CommDetailRevision	Revision mark
AstM4CommExplodeDetail	Detail explode settings
AstDetCheck	Check details status
AstM5DetailStylesExplorer	Drawing Style Manager

Command	Description
AstM5DetailStylesManagerNew	Start the new Drawing Style Manager
AstM4CommDetAddDim	General dimension
AstM4CommDetAddDimSlope	Create slope symbol
AstM4CommDetAddDimAngle	Angle dimension
AstM4CommDetAddDimArc	Arc dimension
AstM4CommDetAddDimRadial	Radial dimension
AstM4CommDetByStyle	3D view
AstM4CommDetailing	Generic command for calling detail styles
AstM4CommDetailingProc	Generic command for calling processes
AstM4CommDetCrHeightKote	Create a level symbol
AstM4CommDetCrWeldSeam	Create a weld symbol
AstM4CommDetAddAnnotation	User label
AstM4CommDetInsertAnno	Create a label
AstM4CommDetModDimAdd	Create a dimension point
AstM4CommDetModDimRemove	Delete a dimension point
AstM4CommDetModDimStyle	Change a dimension style
AstM4CommDetModTextStyle	Change a text style
AstM4CommDetUpdateBOM	Update lists
AstM4CommDetSwitchHighlightState	Toggle the display of green frames around details
AstM4DetViewDelete	Delete a view
AstM4DetViewTransform	Move a view
AstM5DetailProcessExplorer	Start the Drawing Process Manager
AstM4CommDetAddHorLinearDim	Linear horizontal dimension
AstM4CommDetAddSlopeLinearDim	Linear sloped dimension
AstM4CommDetAddVertLinearDim	Linear vertical dimension
AstM4CommDetInsertBom	Create a list

HSDetailing

Command	Description
AstM4CommDetAddDimArc	Arc dimension
AstM4DetProc	Processes
AstM4CommDetInsertBom	Create a list
AstM4CommChangePrototype	Change prototype file

HSIFDSTVBOM

Command	Description
AstM4CommPSSOut	Write PSS

HSIFDSTVNC

Command	Description
AstM4CommPSSOut	Write PSS
AstM4CommCreateNCByStyle	HSIFDSTVNC NC, DXF (plates), DXF (all objects)

Command	Description
AstM4NCSettings	NC Settings

HSIFPM (HSExtended)

Command	Description
AstM4CommPSSIn	Read PSS
AstM4CommPSSOut	Write PSS
AstM4Astor2RepPSS	Export PSS

HSSTAAD

Command	Description
AstM4CommAstorToSTD	Export STD
AstM4CommSTDToAstor	Import STD

Index



- 3D model, 28
- Access databases, 9
- Access tables, 31
- Advance Management Tool, 10
- Advance properties, 71
- Advance Steel** toolbar, 36
- Angle connections, 30
- Any contour, 64
- Any contour processing, 67, 68
- Apex, 90
- ARX-Technology, 9
- Assembly marks, 116
- Assembly part, 120
- ASemplate.dwt, 36
- Audit checking, 124
- AutoCAD® Manipulating Tools, 71
- AutoCAD® properties, 71
- AutoCAD® specific properties, 41
- Auxiliary objects, 31
- Axis groups, 44
- Base plate, 30
- Basic objects, 28
- Beam processing, 28, 62
- Beam to column, 89
- Beams, 28
- Behavior, 48, 74
- Bolt pattern, 29
- Bolt patterns, 74
- Bolts, 75
- Building grid, 44
- Buttons, 39
- Camera, 155
- Checks, 122, 125
- Circular contour, 64
- Circular processing, 68
- Circular processing object, 67
- Collision check, 123
- Collision solids, 123
- Colors, 42
- Compound sections, 46, 48
- Compression pipe, 96
- Connection, 79
- Connection object, 63, 86, 108
- Construction systems, 9, 51
- Contour element, 67
- Curved beams, 48
- Cut at object, 63
- Cut variable, 63
- Default values, 10
- Diagonal bracing, 30
- Dialog box, 40
- Display connected object, 79
- Display connecting elements, 79
- Display objects connected in shop, 79
- Document Manager, 8, 136, 141
- Double-sided web joint, angle-end plate, 91
- Drawing formats, 138
- Drawing frames, 138
- Drawing style, 136
- Drawing styles, 10
- Drawings, 136
- DWG, 8
- Element contour, 65
- Elevations, 146
- End plates, 30
- End plates, asymmetrical, 92
- Esc** key, 39
- Extract, 130
- F2** key, 39
- Features, 28
- Flat bracings, 30
- Flat frames, 83
- Flyouts, 36
- Frame corners, 30
- Gable columns/posts, 90
- Gable wall connections, 30
- Geometric Properties, 41
- Gravity axis, 47
- Grid axes, 31
- Grid values, 70
- Grips, 70, 72
- Gusset plate connection, 93, 94
- Gusset plates, 30, 55
- Hide objects, 82
- Hole pattern, 74, 76
- Holes, 30
- HSS bracings, 30
- Insertion axis, 47
- Joints, 30, 86
- Joists, 83
- Layer, 42
- Level symbols, 31
- List Template Wizard, 130, 132
- Main part, 120
- MDI, 10
- MFC, 9
- Model, 8
- Model changes, 8
- Model objects, 130, 131
- Model role, 84
- Moment end plate, 89
- Moment end plate connection, 92
- Node, 145
- Non-vertical main parts, 148
- Numbering, 116
- Object coordinate system, 59
- Objects, 8
- ODBC, 9
- PCS (Program system in the steel construction), 10
- Pin-Ended Column, 90
- Pin-ended column connections, 30
- Plate processing, 28, 65
- Plate splice, 90
- Plates, 28, 53
- Platform connections, 30
- Platform splices, 92
- Poly beams, 46, 49
- Polygon plate, 54

-
- PPS (Interface product of the steel construction), 10
 - Processes, 137
 - Processing based on rules, 63
 - Processing contours, 29
 - Processing object, 62
 - Project information, 144
 - Properties, 41
 - Proxy-graphics, 10
 - Purlin, 83
 - Pylons, 83
 - Rectangular contour, 64
 - Rectangular plates, 53
 - Rectangular processing, 67, 68
 - Reference axes, 47
 - Report, 133
 - Representation type, 82
 - Representation types, 72
 - Ridge connections, 30
 - Search filter, 81
 - Section classes, 46
 - Selected objects on, 82
 - Selecting objects, 80
 - Shape, 48
 - Shear plates, 30
 - Shear studs, 74, 76
 - Shop drawings, 8, 148
 - Shorten at UCS, 63
 - Single part mark, 116
 - Single parts, 147
 - Single-span bracings, 83
 - Snaps, 72
 - Special parts, 32, 109
 - Splice joint, 90
 - Splices, 30
 - Standard** toolbar, 36
 - Steel construction checking, 125
 - Stiffeners, 30
 - Structural elements, 83
 - Structural frame, 31
 - Structured BOM, 8, 132
 - System line, 47
 - Technical properties, 41
 - Title block, 145
 - Toolbars, 36
 - Tube connections, 30
 - Turnbuckle bracing, 30
 - User Coordinate System (UCS), 58
 - User sections, 110
 - View, 138
 - Web connections, 30
 - Weld line, 78
 - Weld point, 78
 - Welded beams, 52
 - Welds, 29
 - World Coordinate System (WCS), 58

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