HiCAD

3-D Training

HIC-TR-GB3D912

3-D Training

1. Text Section
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1 Quick Overview

HiCAD Basics

This part of the training book deals with the basics of the CAD program HiCAD, such as creating, saving and loading of drawings, point options, part structure etc., including related exercises.

Text Section + 3-D Training Section

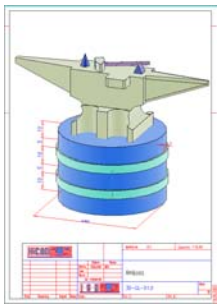
From “Chapter 30: “Beginning of 3-D Training: Solid Primitives“ onwards, this training book provides various exercises for the 3-D module. Each exercise is preceded by one or several chapters explaining all functions needed do be able to do the exercise. If you are already familiar with the basics of HiCAD, you can immediately start with this chapter.

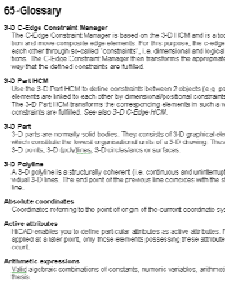
Glossary and Index

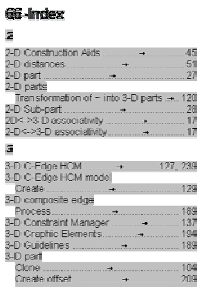
The Glossary provides explanations of the essential CAD-specific terms. In the Index you will find an alphabetical list of the training book topics and the corresponding pages.

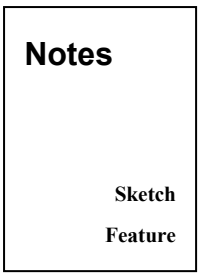
Notes

This section of the training book contains empty pages for your personal notes.









2 FAQ

Experience has shown that some questions arise particularly frequently. To spare you unnecessary browsing through the pages we will begin with 12 FAQs:

Question 1: What are the functions of the mouse buttons?

a) Left mouse button (LMB): Select function / confirm entry (e.g. a radius)

b) Middle mouse button (MMB): Exit function / cancel

c) Right mouse button (RMB): Context menu (you can right-click on an icon, a part, the drawing area, a dimensioning, a text and numerous other objects).

Question 2: How do I select a point option?

a) All point options represented by *one* letter (R, A, D, W, P, N) can be selected via the corresponding keyboard button.

b) RETURN key of the keyboard

c) Right-click just when HiCAD suggests a point option

d) (quick) double left-click

Question 3: How do I draw an uninterrupted polyline even if different point options are required?

a) left-click on the drawing area and select new point option

b) RMB > Polyline > Continue, and select new point option

c) overwrite previous point option with new point option (only possible for point options represented by one or two letters)

Question 4: Which point options do I select if a reference point is required?

Select „Relative-Return“ (REL RET). Just press the “R“ key for point option „Relative“, then confirm by pressing the RETURN key.

Question 5: Which types of re-use are available?

a) HiCAD Clipboard

b) Insert Part > Others > From drawing, Via document master

c) Insert Part > Insert part, via DB document master

d) Insert Part > Insert main part, via Standard Part catalogue

e) Insert Part > User Library > New part library

In this context please also note how you save parts to a catalogue, resp. to the database. The procedure is described in detail in Chapter 51: “Re-Use”.

Question 6: How do I close a polyline in a 3-D Sketch?

a) Sketch > Others > Sort GE in active sketch

b) Information > 3-D, Further… > Information – 3-D Sketch

c) Information > 3-D, Further… > Information – 3-D Sketch 1

After further changes select „Sort GE in active sketch“ again.

Question 7: I want to convert a 2-D part to a 3-D part. A closed contour is required for this. How do I make sure that the contour of the 2-D part is closed?

a) 2-D-Geometry > Tools > Sort GE… (Drawing or Active part)

b) 2-D-Geometry > Change > Delete overlaps

c) Information > 2-D, Further… > Polyline

After further changes select the "Sort GE…" function again.

Question 8: How do I perform a translation by a particular value with a translation vector?

Start point: Select an arbitrary point in the drawing area. End point: Relative.

Question 9: How do I change the grid in Sketch functions?

"–" key: refine the grid; "+" key: coarsen the grid; space key: for direct value input

Question 10: How do I convert a HiCAD drawing to a graphic?

a) CTRL + C > Select format and size > Select detail / Select background colour

b) CTRL + W

c) PRINT key

d) Activate the "Drawing" tab and select Others > Extras > Tools > Create Enhanced Metafile (EMF). You can also select a superordinate part with a *right-click*.

Question 11: How do I load the automatic "10 minute save" option?

Open / Explorer / Local Disk / hicad / temp / switch to „SZN" extension in the textbox „Files of type"/ select **hicad_N1**. During this process, N is the number of the drawing which with you have worked. If you want to load the drawing in the third compartment of the tab „Construction", load **hicad_31**; if you have worked in the first compartment, select the drawing **hicad_11**.

Question 12: How do I exchange the ISD logo in the drawing frame with my own company logo?

Perform the following steps in an empty drawing:

a) Drawing > Insert drawing frame > Do not complete title block > select a frame, e.g. DIN A0 > insert this frame in your drawing.

b) In the "2-D Part structure" tab of the ICN, right-click on the name of the *drawing*. Select the "Auxiliary parts On/Off" function, then select "Auxiliary parts On".

c) Delete the sub-part with the old logo and create a new sub-part (e.g. with the name "Logo") below the "SCHRIFE" entry. Draw the new logo there, e.g. with lines, texts etc. You can also create a new sub-part to the part "Logo", draw a company logo as a closed contour there and hatch it with very narrow hatching line spacing (e.g. 0.05 – an example for this would be the "HiCAD eye" symbol). Save the part "Logo" as *Part*.

d) Activate the entire drawing frame and save it as 2-D part without database

   i) with the name "DINA0" suggested by the computer, if you are creating a frame for your own company,

   ii) with a different name, if you are creating a frame for a customer, for whom you regularly create drawings.

e) Now load, one after another, frames DIN-A1 to DIN-A4. For each of them you can delete the sub-part with the old logo and load your own company logo saved in step c), as a sub-part to the part „SCHRIFE". Then continue with step d).

3 Abbreviations

In this training book the following abbreviations are used:

- ICN Information and Communication Navigator (Browser)
- LMB left mouse button
- MMB middle mouse button
- RMB right mouse button
- HCM HiCAD Constraint Manager (Parametrics)

4 Learning Targets

This training book provides a tutorial enabling you to operate the HiCAD 3-D module. The topics covered in this training book correspond to the contents of a 3-D basic training course at the ISD. Target groups of this training are, for example, design engineers, design draughtsmen or product managers. The tutorial has been structured in such a way that you can work through the chapters successively. You can however also start with a chapter of your choice.

All participants in the training should have sufficient skill in the operation of the HiCAD 2-D module. As a good knowledge of 2-D design (especially the HiCAD part structure and the point options) is essential to working in 3-D, the first exercises in this training book will deal with the HiCAD 2-D module.

In this training manual you will learn how to

- convert the product structure to the HiCAD part structure,
- create, process and correct 3-D parts,
- change 3-D parts in entire drawings,
- use standard parts,
- dimension parts,
- insert texts,
- create detail drawings and assembly drawings,
- print and plot drawings and output data via interfaces.

The target of the training consists in enabling you to create engineering documents of your product that are suitable for production.

Each of the following sections is explained in the following way:

- Icons and descriptions of the essential functions,
- References to the appropriate chapters in the Online Help, if required,
- Answering of questions and clarification of issues that may arise,
- Explanation of the processing steps,
- One exercise on the topics dealt with in the chapter.

5 How Can I Get Further Information?

5.1 Online Help

In addition to this training book, you can get more information on the 2-D and 3-D modules from the following sources of information:

- The Online Help for the 2-D and 3-D module. You can call the Online Help with the F1 key or by clicking the "**?**" symbol (at the top right of the HiCAD GUI) and selecting **Help Topics**.
- The above-mentioned Online Help also contains the "Basics" section that provides a general overview of the working techniques and technologies used in HiCAD. It explains the concept of the system, the Graphical User Interface (GUI) and various design methods, and provides definitions of HiCAD-specific terms. General functions such as the working with drawing details or the printing and plotting of drawings are described in this Online Help section.
- The "What's New in HiCAD?" section of the Online Help provides an overview of new functions and enhancements in HiCAD.

Furthermore, the Online Help provides information on the Industry Solutions (Steel Engineering, Metal Engineering, Plant Engineering etc.), or on special modules such as Parametrics, 2-D->3-D Conversion, Macro Technology, Freeform Surface Technology, or HELiOS Product Data Management.

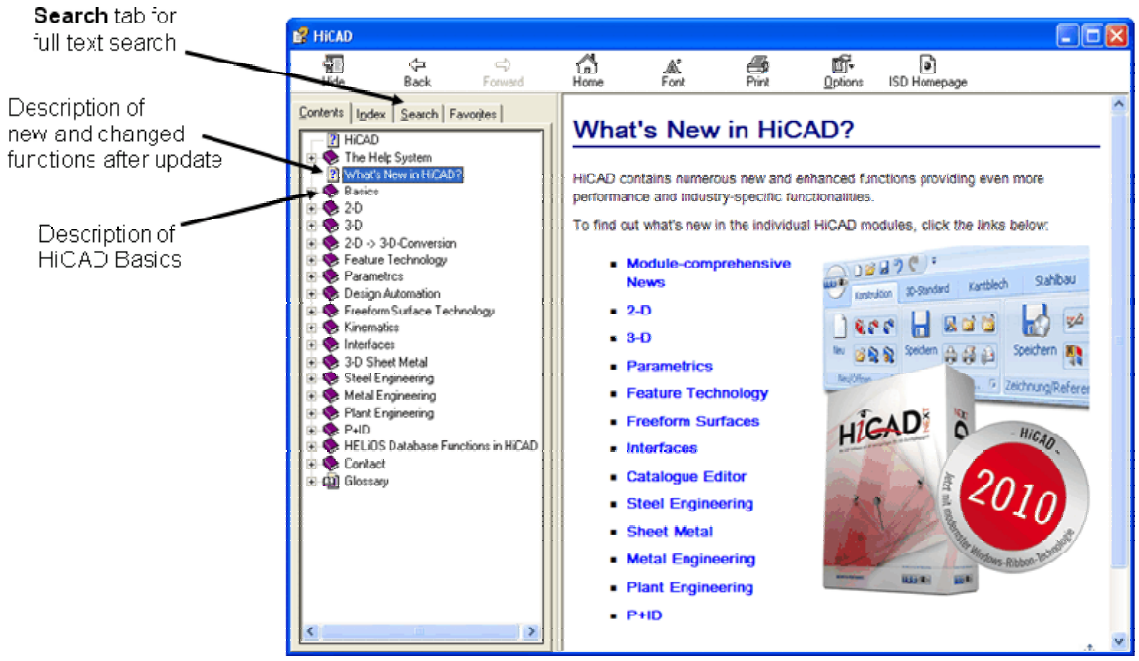


Fig. 1 The HiCAD Online Help

6 Why 3-D?

Today an increasing number of companies rely on 3-D models to reduce their development times and remain internationally competitive.

The advantages of 3-D models are however still widely underestimated although they significantly contribute to an increased competitiveness and can be used by many departments of a company in a productive and efficient manner.

6.1 Clear and Descriptive Visualisation

If you want to represent several views of a real part in a 2-D CAD system, you need to create for each view an appropriate 2-D model. There are however no dependencies between the individual 2-D views. This means that changes applied to one of the views do not automatically

affect the other views. Therefore a pure 2-D CAD system does not enable you to check the consistency of the individual views to ensure a realistic, “true-to-life” representation of the part.

In a 3-D model, by contrast, each point is described exactly, thus preventing geometrically inconsistent models and allowing a realistic modelling of parts. Thanks to optimum illumination and assignment of material properties, a 3-D model can be rendered in a photo-realistic manner and presented in a far more descriptive and promotionally effective way than a 2-D drawing. The representation options run the gamut from simple ray-tracing to tracking shots and simulated motion sequences, which comes in handy for presentations on trade fairs or at a customer’s company.

6.2 Reduce Errors

3-D models can be used to evaluate prototypes during the construction process and check their functional capabilities in early stages: The earlier errors are detected, the earlier they can be eliminated or even avoided right from the start, thus significantly improving product quality while reducing production costs.

Exploded views can be derived directly from the 3-D model. Transport, motion and assembling simulations can be performed and checked during the design process, enabling you to detect and prevent collisions during assembling and transport processes or motion sequences at early stages of a project.

In the field of conveying technique, for example, you can perform collision checks via simulation, which enables you to reliably assess the operability of a product. In the field of mould making, you can use motion or assembling simulation to analyse moulding processes.

6.3 Work Flexibly

3-D bodies can be processed and modelled much easier: For example, if you add a bore somewhere in a 2-D drawing, you are forced to adjust the different views of the model individually. But if you add the bore in 3-D, the production drawings will be adjusted automatically.

Furthermore, designing in 3-D allows a significant reduction of routine and/or repetitive tasks due to many automation technologies, e.g. Feature Technology or Referencing.

6.4 Accelerated Product Development

Another advantage of 3-D models over 2-D drawings consists in the fact that they contain much more data. This enables the user to process 3-D models including their data with subsequent applications. Analyses and simulations on a virtual model are just as possible as Rapid Prototyping, FEM calculations or the automatic generation of tool paths.

Altogether, the superior geometry data of the 3-D model allows a broad range of applications and processings throughout the entire production process. The tighter integration of all departments involved in the process enables a significant reduction of product development times, which constitutes an important factor for global competitiveness.

7 About the ISD and HiCAD

7.1 About the ISD

The **ISD Software und Systeme GmbH** is one of the worldwide leading providers of highly integrated CAD/CAM/PDM solutions for innovative product development processes.

Our software solutions for Engineering and the manufacturing industry are developed at our headquarters in the Technology Park Dortmund. These are:

- **HiCAD**, the CAD system for development and design engineering,
- **HELiOS**, the PDM solution for the organisation of product development processes,
- **HiCAM**, the CAM product group for a passing of construction data to NC-controls in production departments.

In addition, you can also use our wide range of services, including Consulting, Product Development, Installation, Training, Maintenance and Hotline, which enable short lines of communication, high flexibility and an optimum adaptation to customer requirements.

More information about the ISD and its product can be found on the Internet at: http://www.isdgroup.com

7.2 About HiCAD

HiCAD is a CAD solution for the entire product development and design engineering process. Due to its modular structure, the program is just as suitable for beginners as for complex, highly automated construction processes.

2-D/3-D functionality, object-oriented working and an associative data structure allow a very convenient operation. The hybrid data model with surface/solid modeller, Feature Technology and Freeform Surface modeller enables an extremely high performance, which can even be increased further by the following Industry Modules:

- Steel Engineering / Metal Engineering,
- Stairs construction,
- Sheet Metal,
- Plant Engineering,
- P+ID (**P**iping & **I**nstrumentation **D**iagrams),
- Freeform Surfaces,
- Macro Programming,
- 2-D and 3-D Variants,
- Assembling/Transport/Motion simulations,
- Integrated Product Data Management,
- BOM creation, ERP interface, data transfer to NC programs
- … and many other programs. For further information, please contact your sales agent.

HiCAD's intuitive user interface enables a quick familiarisation with the product and easy access to 2-D/3-D data structures as well as to the integrated Industry Solutions. The extraordinary versatility of HiCAD enables a combined, simultaneous working in 2-D and 3-D – in one file!

You can save your existing 2-D data and use the 2-D->3-D Conversion module to automatically convert your 2-D drawings to 3-D model drawings at any time!

An integral part of **HiCAD** is the *Information + Communication Navigator (ICN)*, which clearly outperforms common structure browsers. The *ICN* provides direct access to all opened models, drawings and views, part and assembly structures, attributes and the PDM properties of individual parts. You can create and process your drawing while the ICN is open. During the process, the representation in the *ICN* is constantly adjusted to the actual construction in the drawing area. All structures and properties can be modified or processed via mouse-clicks in the *ICN*. In this way, you can also activate a wide range of part processing functions. Multiple selection options (as known from Windows applications) enable a modification of several parts in one single processing step.

HiCAD's Performance Characteristics:

- Microsoft Windows Standard
- Combined working in 2-D and 3-D
- Part-oriented designing
- Information + Communication Navigator (ICN)
- Automatic object snap and context menus
- Extensive Symbol and Standard Parts Libraries
- Macro Technology and Macro Recorder
- HiCAD Constraint Manager (HCM)
- 2-D->3-D Conversion Technology
- Assembling/Transport/Motion Simulation
- Freeform Surface Technology
- Part Referencing
- Part-oriented Feature Technology
- Material Editor and lighting models
- Integrated Product Data Management
- Data transfer from other systems (CATIA, ME10, STEP, DXF, DWG etc.)
- BOM creation, ERP interface, data transfer to NC programs
- Powerful Industry Solutions
- Modular, extendable structure

7.2.1 HiCAD's Concept at a Glance

- The *Information + Communication Navigator* (*ICN*) provides a permanent overview of complex structures and data. It enables you to control and process your drawing and provides quick access to individual drawing components.
- The unique 2-D<->3-D associativity within one file is enabled through a combination of 3-D design and 2-D legacy data maintenance.
- Comprehensive Industry Solutions for Steel Engineering, Plant Engineering, Sheet Metal and many other fields of industry provide optimum functionality for all use cases.
- The combination of parametric and non-parametric modelling enabled through our self-developed ESM kernel guarantees a maximum of flexibility.
- Significant time savings can be achieved through the automation of standard component creation via Macro and Variant Technology.
- Function simulations and analyses help you avoid errors in the 3-D model at an early stage.
- Top-performance even for large assemblies consisting of tens of thousands of parts.
- The integrated PDM functionality ensures a safe referencing, thus creating the basis for Concurrent Engineering.

- The system is operated in a similar way as Windows applications, which enables a quick familiarisation and keeps training costs low.

7.2.2 2-D<->3-D Associativity

The so-called “2-D<->3-D associativity” enables a combined, simultaneous designing in 2-D and 3-D at any time.

As **HiCAD** provides such associativity, a HiCAD drawing can contain 2-D and 3-D objects from various industries in one file.

“2-D<->3-D associativity” means in particular:

- For example, views created with HiCAD 3-D can be turned into production drawings with the help of 2-D functions. As dimensionings and texts have already been entered in the 3-D model, they will already exist in the derived views.
- 2-D drawings – even from foreign systems – can be used as a basis for the creation of 3-D model drawings. The 2-D->3-D Conversion module, which has been especially developed for this purpose, enables an automatic conversion of 2-D detail drawings to 3-D models.

8 The Screen Configuration

HiCAD's user interface is based on graphical symbols. All functions can be selected via self-explanatory symbols.

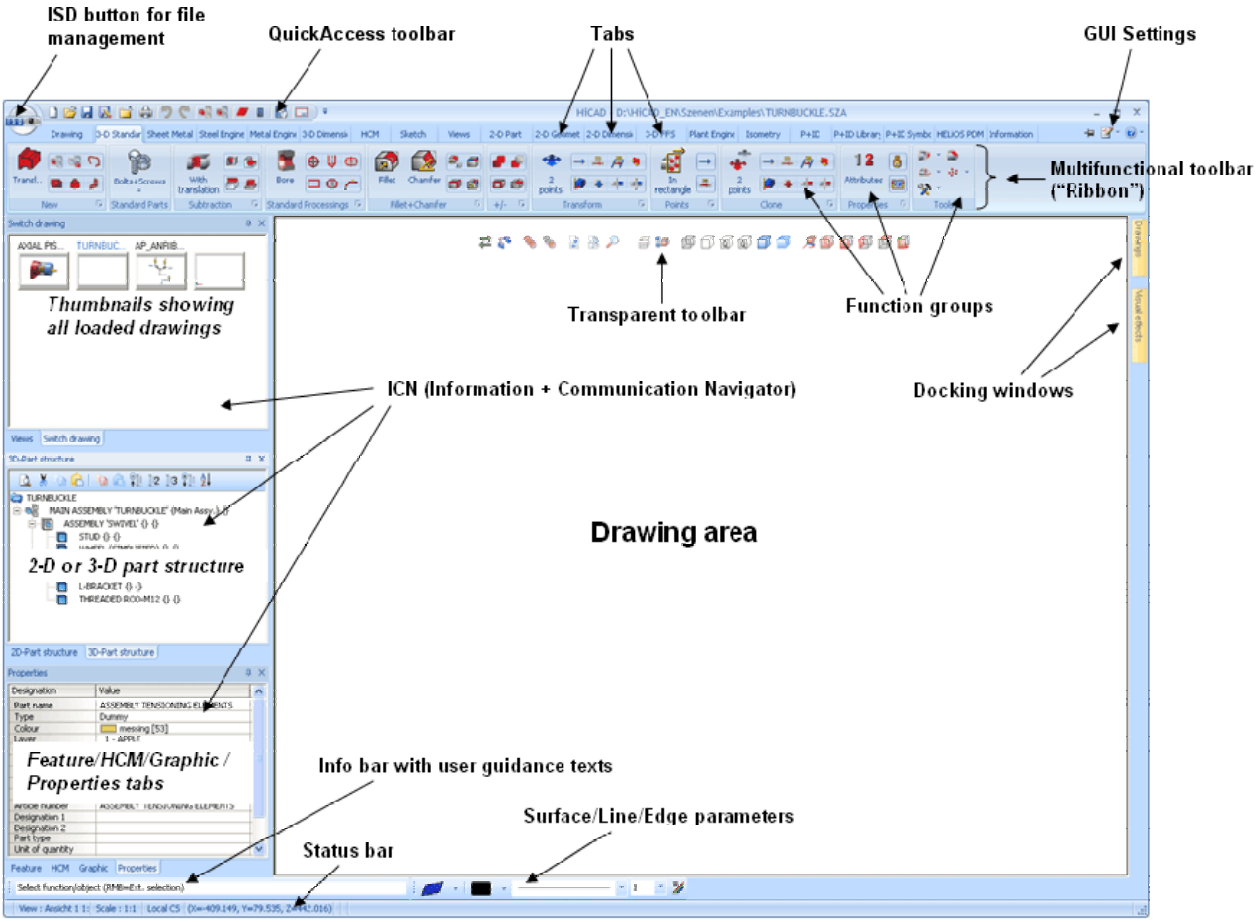


Fig. 2 The HiCAD User Interface (Default configuration)

The HiCAD user interface mainly consists of the following areas:

- the drawing area,
- the transparent toolbar,
- the multifunctional toolbar ("Ribbon"),
- the tabs of the Ribbon,
- the QuickAccess toolbar,
- the Information + Communication Navigator (ICN),
- the info bar with user guidance texts,
- the "Surface/Line/Edge Parameters" bar,
- the status bar.

With the exception of the drawing area and the Ribbon, the above elements of the user interface can be freely positioned, hidden and re-displayed again.

8.1 The Drawing Area

In this area you create and process your drawing. The drawing detail displayed in this area is the current (active) detail. Use the Zoom functions to enlarge or downsize the detail.

see Online Help

8.2 How Do I Select a Function?

You use the functions of the Ribbon, the context menus, the transparent toolbar, and the Quick-Access toolbar to tell HiCAD what tasks you want to perform.

8.2.1 Multifunctional Toolbar ("Ribbon")

The Ribbon is composed of different tabs, which in turn consist of several function groups. Most of the functions are illustrated with small images (symbols). To select a function on the Ribbon, just left-click the desired function. If you activate a different tab, you will obtain different functions.

8.2.2 Context Menus

If you right-click a part, a 3-D view (pink dotted rectangle), the drawing area or a dimensioning, a context menu for the selected object will be displayed. Examples: If you right-click a 2-D or 3-D part in the drawing area or in the ICN, you will get various part-specific functions such as "Translate part", "Subtract part" etc. If you right-click the drawing area, you are offered drawing-specific functions, such as "Change scale" or similar. If you right-click a dimensioning of a drawing object, you are offered various dimensioning functions. The selecting of functions via context menus is often the quickest way to call a function: If you select an object by right-clicking it, you automatically indicate to HiCAD which of the parts or dimensionings you want to modify or process.

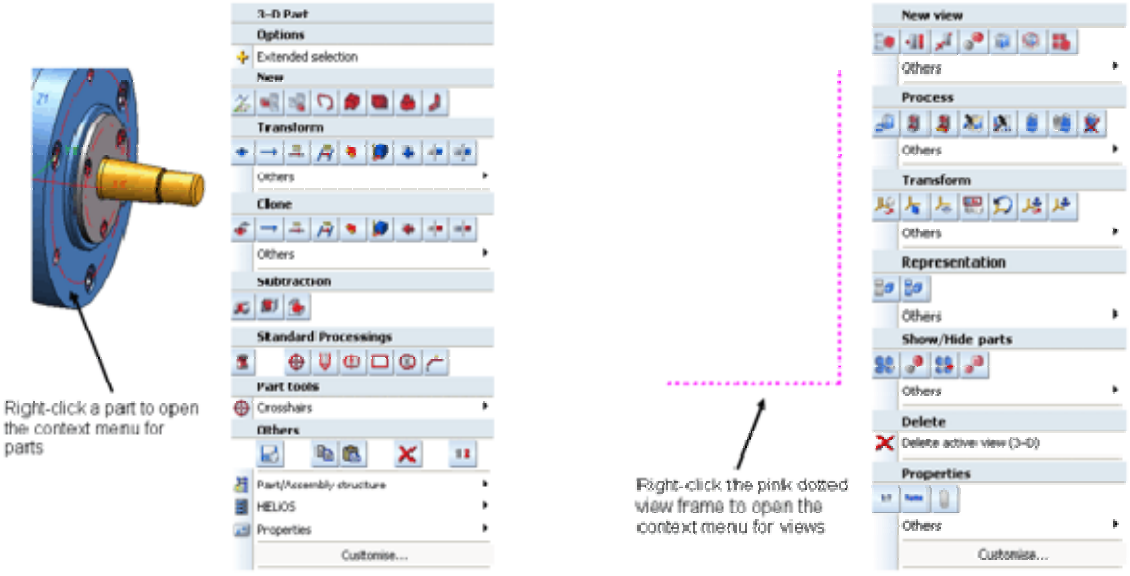


Fig. 3 Context menu of a 3-D part (left) and of a view (right)

8.2.3 Pull-Down Menus

Pull-down menus are called via the function groups of the Ribbon. You open them by clicking the arrow symbol in a function group. To select a function in the pull-down menu, left-click it.

Fig. 4 Example of a pull-down menu

A pull-down menu can contain sub-menus. Functions containing sub-menus are identified by an arrowhead pointing to the right. The sub-menu is displayed if you move the mouse pointer on an appropriately identified menu item:

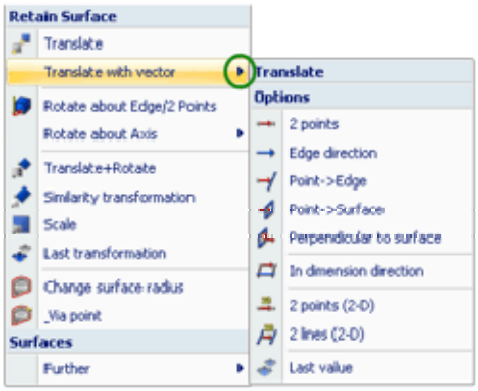


Fig. 5 Sub-menu of pull-down menu

Please note:

- If a pull-down menu is already displayed for a function group of the Ribbon, just move the mouse pointer over the other menu items. If there is a sub-menu to the menu item, it will be displayed without a further mouse click.
- To close a pull-down menu, simply left-click outside the pull-down menu.

8.2.4 Pop-Up Menus

In contrast to a pull-down menu, a pop-up menu can appear in any visible position of the screen area. You activate a pop-up menu by a right-click.

In HiCAD one distinguishes between two types of pop-up menus:

- Pop-up menus providing functions for quick access and designing alternatives
- Context-sensitive pop-up menus, also called context menus.

QuickAccess Functions and Designing Alternatives
If the cursor points on any function, right-click to open the below menu containing several QuickAccess functions. You can use this menu, for example, to take over frequently used functions into the QuickAccess toolbar.

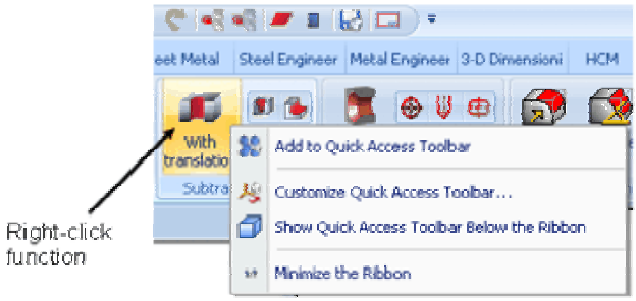


Fig. 6 Right-click on any function

Now let us assume that we have already activated a function, e.g. “Rectangle, 2 points, Sketch”. Now you can right-click to open a pop-up menu with designing alternatives (sub-menu) for this menu item or function:

Fig. 7 Sub-functions of the 2-D function **Rectangle, 2 points, Sketch**

Context Menu
If the cursor points on an object in your drawing, e.g. on a part, a view or a dimensioning, right-click to activate a context menu with appropriate functions for the selected object. Please also see chapter “8.2.2 Context Menus“.

A pop-up menu appears directly under the current cursor position. Left-click outside the menu to close the pop-up menu again.

8.3 The Info Bar with User Guidance Texts

In this field, which can normally be found at the bottom of the HiCAD screen, the user guidance texts are displayed. These texts show you what further steps need to be taken to execute a function. Especially in case of functions requiring reactions to several prompts it is very important that you take a look at the Info bar to see the next step that HiCAD prompts you to take.

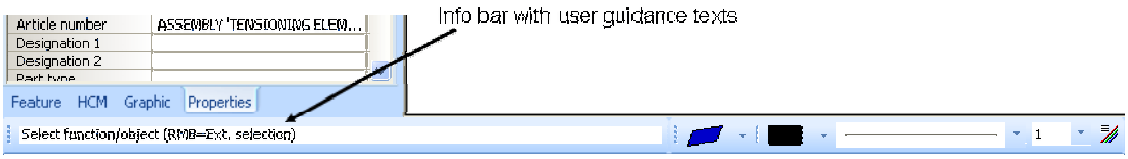


Fig. 8 The Info bar with user guidance texts

Furthermore, the user guidance text tells you whether you can activate a context menu with a right-click or whether any special functions are available.

Please note that you can display a further bar with user guidance texts via Settings > Docking windows > Info Bar, as shown in the figure below:

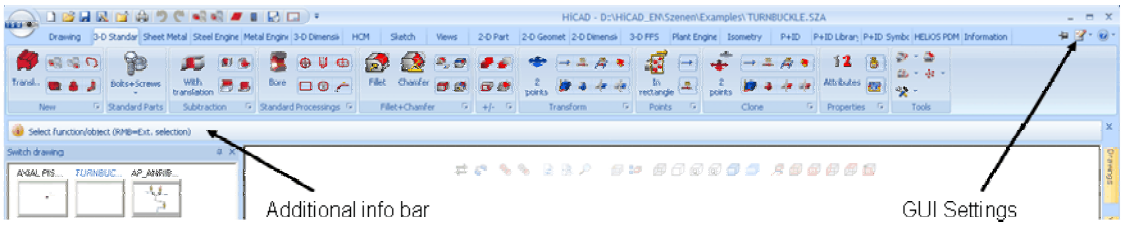


Fig. 9 Additional Info Bar with user guidance texts

see Online Help

8.4 The Status Bar

The fields in the Status Bar solely serve the purpose of information. They contain

- the name of the active part,
- the active view,
- the scale of the current drawing,
- the type of the selected coordinate system, and
- the current cursor coordinates

are displayed.

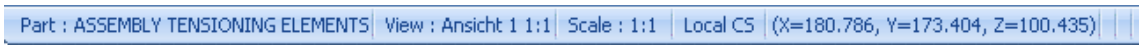


Fig. 10 The Status Bar (detail)

9 The HiCAD Drawing

Construction files in HiCAD are referred to as **Drawings**. They have the file extension .**SZA**.

A drawing consists of 2-D and/or 3-D parts. These can, for example, be individual parts, assemblies or bores.

For the sake of structuring, one distinguishes between

- **Main parts** and
- **Sub-parts**.

This structure is referred to as **Part structure** and constitutes the basic concept of all HiCAD applications.

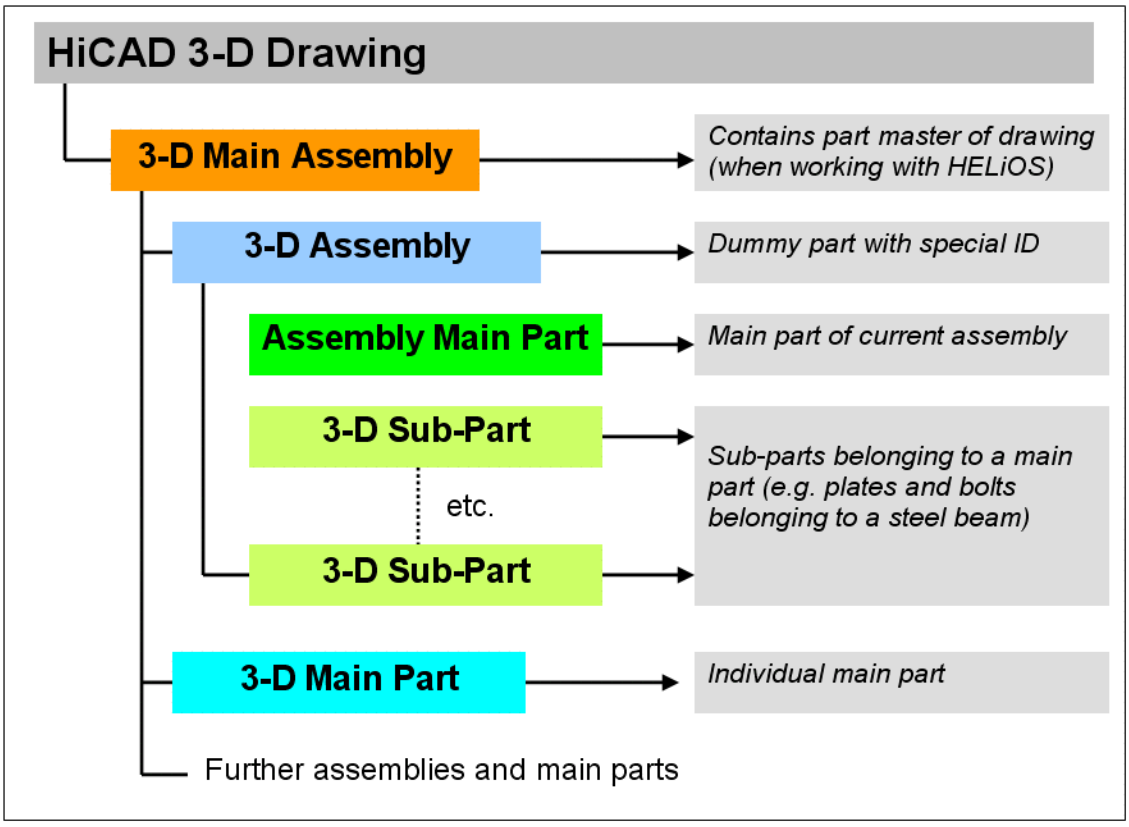


Fig. 11 Part structure in 3-D

In the 2-D module, too, there are main parts and sub-parts. As in the 3-D module, main parts are independent parts, i.e. such parts which do not belong to any other part. Sub-parts are parts which belong to another part (or assembly), to which they have been welded, bolted, glued etc.

Generally, a HiCAD drawing contains information on:

- the coordinate system and the unit of measurement,
- the 2-D and 3-D parts,
- the hatchings,
- the 2-D and 3-D texts,
- the 2-D and 3-D dimensionings,
- the non-graphical elements (object data).

9.1 What is the Purpose of Part Structures?

Especially when dealing with very complex drawings, a logical structuring of data is absolutely essential. In HiCAD such structuring is enabled by a subdivision of drawings into main parts and sub-parts. This part-oriented structure allows constructions whose logical composition corresponds to that of real products, consisting of a main assembly, (sub-)assemblies, and individual parts.

The HiCAD part structure is displayed in the form of a “tree structure” – similar to the directory structure of your hard disk on your computer.

The benefits:

- The drawing has a clear, descriptive structure.
- You can link production data and BOM information to the constructed part.
- The part structure enables the creation of structure lists and quantity lists.

- The parts where-used list promotes a re-use of parts.
- Company-specific series can be represented via part parameterisation.
- Parts can be transferred from a draft to a detail drawing for production.
- Re-used parts can be changed in one step in all drawings.
- Designing in 2-D requires overlap calculations in the assembly drawing, which are realised via the overlap levels of the parts.
- The associativity of hatchings is realised via the part.
- Scaling for individual parts, independent of the drawing scale, enables better representations.
- Processing functions usually refer to parts.

The observance of a logical and clear part structure automatically leads to a professional, meticulous and clean construction of the parts. In this way, error sources are reduced significantly.

An overview of the part structure of you current drawing is displayed in the middle window of the *Information + Communication Navigator (ICN)*.

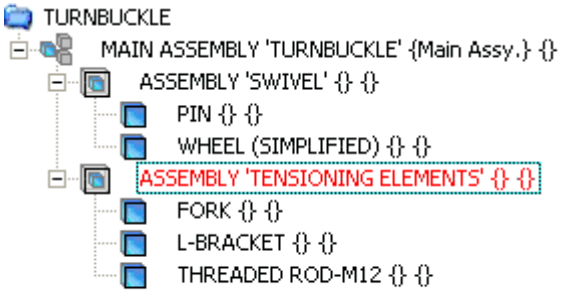


Fig. 12 Part structure in the Information + Communication Navigator

9.1.1 Main Parts and Sub-Parts

A main part is the highest organisational unit in a HiCAD and can in turn consist of several subordinated parts, which are called sub-parts. These sub-parts, too, can consist of further sub-parts. The subdivision of part into main parts and sub-parts serves the purpose of a logical structuring.

Just like a drawing, each main or sub-part has its own name enabling its retrieval or activation.

10 Basics of the 2-D Module

The operation of the 3-D module requires basic knowledge of the 2-D module. The following chapters will therefore deal with various 2-D functions and exercises, and mainly concentrate on the topics which will later also be relevant for the operation of the 3-D module (e.g. point options, part structure etc.). If you have already attended the HiCAD 2-D Training, you can skip the next chapters and begin with Chapter 30.

10.1 Drawing Creation: The First Steps

To create a new, initially empty drawing file, select the **New drawing** function.

You have the following options to activate this function:

- Activate the **Drawing** tab on the Ribbon, and then select the **New drawing** function in the **New/Open** function group.
- Select the keyboard combination **CTRL+N**.
- Select the **New drawing** function on the QuickAccess toolbar (at the top of the screen).

The further procedure depends on whether you want to create a drawing with or without database:

Procedure WITH database (additional module)

If you want to create a drawing *with* database (additional module), activate in the HiCAD Start Centre the checkboxes **Create new drawing** and **Enter drawing into database** and select a scale. After clicking **OK**, select **New drawing, without part master**. If you also want to use the Part and BOM Management of the database, select **New drawing, with part master**. This procedure is explained in detail in a separate database training.

If you select the **New drawing, without part master** option, the following input mask for documents appears:

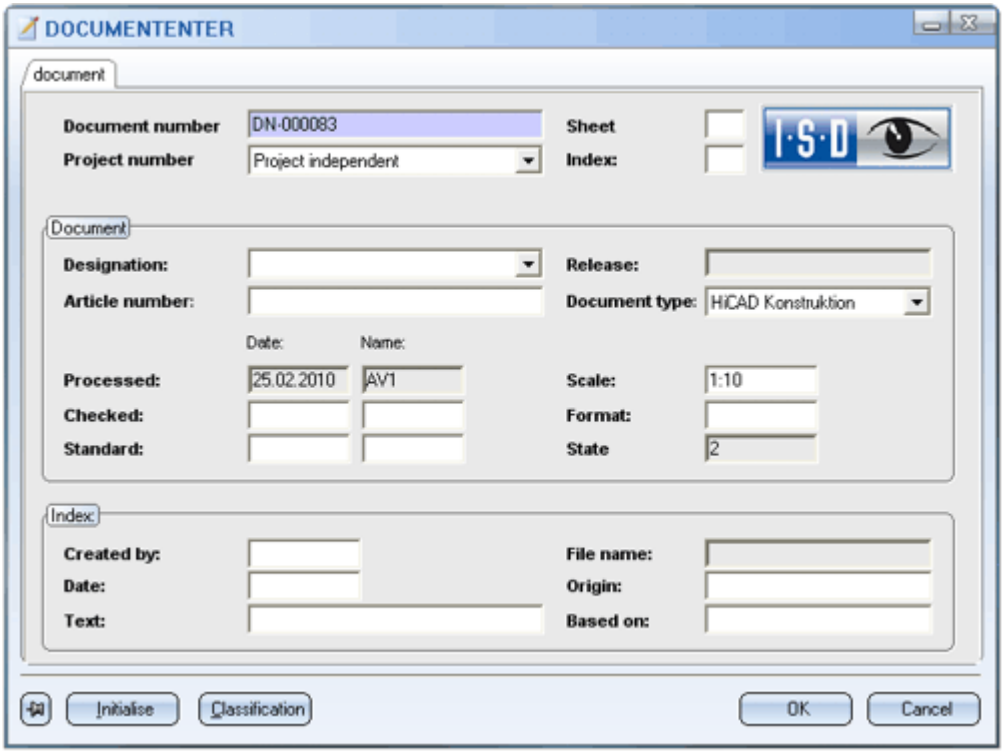


Fig. 13 New drawing, with database (additional module). Input mask for documents

The scale of the current settings in the HiCAD Start Centre is applied. If you right-click the drawing area you can change the scale with the **Main scale** function.

Procedure WITHOUT database

See Online Help

If you want to create a drawing *without* database, activate in the HiCAD Start Centre the **Create new drawing** checkbox, but *do not activate* the **Enter drawing into database** checkbox. After clicking **OK**, the new drawing automatically obtains the name “DRAWING”, followed by a consecutive number. Use the **Save as** function in the QuickAccess toolbar to replace this name by a more descriptive name.

Fig. 14 New drawing, without database. Automatically created drawing name “DRAWING1”

10.2 Creating a 2-D Part

As described above, a HiCAD drawing can contain 2-D and 3-D parts which in turn consist of 2-D graphical elements and 3-D geometry elements. Each of these parts can be a main part or sub-part, or - when working with the 3-D module - an assembly or main assembly.

A **Main assembly** (e.g. a Main assembly ‘Motor’) is the highest organisational unit in the drawing and can in turn consist of several **Assemblies**, e.g. *Assembly ‘Drive’* and *Assembly ‘Sheet covering*’. The Assemblies are made up of individual parts, such as base sheet, lateral sheet LH or lateral sheet RH. For the sake of a clear structuring, one distinguishes between **Main parts** and **Sub-parts**.

This part-oriented data structure enables the creation of constructions whose logical structure corresponds to that of real products. It is therefore recommended that you plan your product structure before you begin with your actual drawing and arrange the part structure of your product appropriately.

As for the drawing, each part, whether main part or sub-part, is assigned a name via which it can be retrieved or activated.

Please note: In contrast to the 3-D module, there are no explicit assemblies or assembly main parts in the 2-D module. Here we use only main parts and sub-parts. If desired, we can however call a 2-D main part, e.g. *Main assembly ‘Motor’* and a sub-part, e.g. *Assembly ‘Sheet covering’*.

10.2.1 2-D Main Part

To create a new 2-D main part, use the **New main part** function. You can access this function

- in the **New** function group of the **2-D Part** tab of the Ribbon, or
- by right-clicking on a free space in the drawing area.

Select the **New main part** function and enter the part name. Then switch to the construction functions in the **2-D Geometry** tab.

10.2.2 Main Assembly (2-D)

In the **New** function group of the **2-D Part** tab you will also find the **Main assembly (2-D)** function. As with the “New main part” function, you create a 2-D main part – with the difference that all existing 2-D parts are automatically moved below the newly created 2-D main assembly.

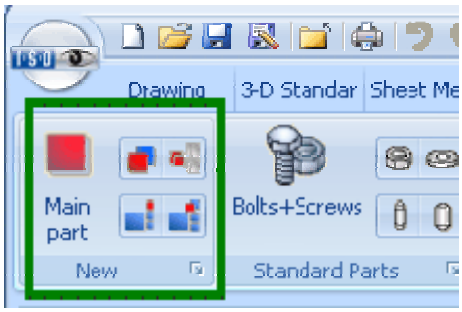


Fig. 15 Functions for the creation of a 2-D part

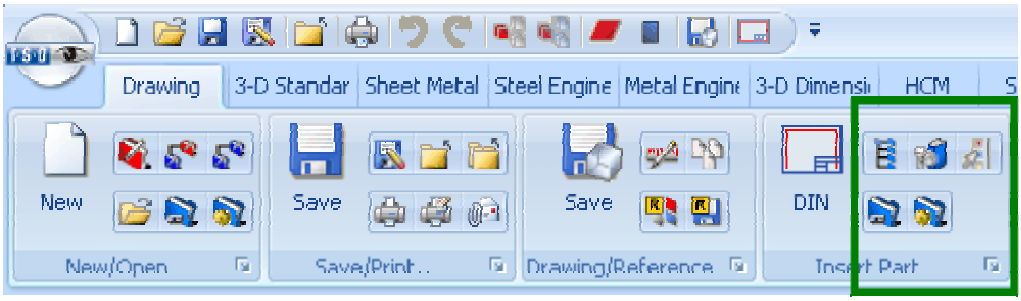


Fig. 16 Functions for the loading of parts

Select the required function (e.g. **New main part**) and enter the part name. Depending on the selected function, choose the required construction function from the Ribbon or the pop-up menus.

see Online Help

A detailed description of the individual design functions can be found in the Online Help.

You can also access the different functions by right-clicking an existing part. The context menu will only offer appropriate functions for selection, i.e. if you select a 2-D part, the context menu will only contain functions for creation and processing of 2-D parts. If you right-click a 3-D part, the context menu will only contain 3-D functions etc.

10.2.3 New 2-D Sub-Part

To assign a new sub-part to a part, right-click the part to which you want to assign a new sub-part. Select **2-D Part, New**, then select **New sub-part.**

Please note: You can right-click the part either in the drawing area or in the ICN (structure browser).

10.2.4 The Active Part

The currently selected part in the drawing is called “active part”. The HiCAD processing functions normally refer to this part. For example, when inserting dimensionings or texts in a drawing, they are always assigned to the currently active part. Active parts are marked red in the *ICN* (*Information + Communication Navigator)*.

You have the following options to activate a part in a drawing:

- Move the mouse pointer on the part and click the part. The following part-relevant functions will then be applied to this part.
- Activate the part in the *ICN.*
- Move the mouse pointer on the part and right-click the part. A context menu appears, displaying the possible processing functions for this part. A right-clicked part is automatically activated.

11 Point Options

11.1 General

In this chapter we will learn how to use so-called point options and several ways to select them.

Point options are used for line functions, transformation functions or similar, e.g. to specify the position of a line, rectangle or translation vector. Almost all point options in 2-D can also be used for 3-D parts.

First, let us take a look at the rectangle function to which we will apply various point options at a later point:

Rectangle, 2 points

The Ribbon is made up of numerous tabs, which in turn consist of several function groups (see Chapter 18 “The Screen Configuration“). The **Polygon** function group of the **2-D Geometry** tab contains functions enabling you to

- sketch rectangles or define them via specification of 2 points,
- draw or translate rectangles in a particular angle
- draw hexagons, octagons and n-gons

Line colour and line type can be set at the bottom of the screen.

11.2 How to Select Point Options

11.2.1 Selection of Point Options

Point options are used to create, identify or position parts. They can be considered subordinate functions to the other functions. To activate a point option, you first need to call another function (e.g. “Rectangle, 2 points“, “Polyline“, or “Translate part“), which requires an object identification or a point specification first.

There are 5 different ways to select a point option:

The HiCAD Autopilot:

If you want HiCAD to suggest a snappable point to you, place the cursor in the vicinity of this point. If the Autopilot has found the point, an appropriate symbol is displayed next to the cursor:

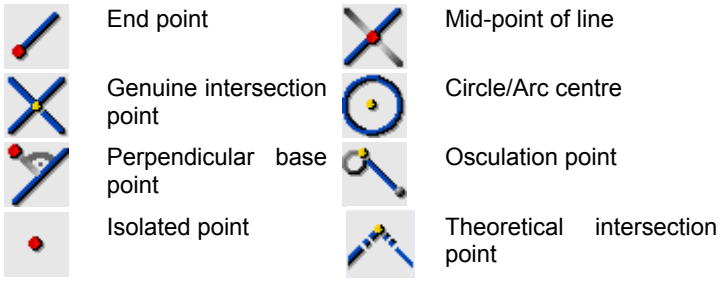


Fig. 17 Snap point symbols (Examples)

You apply a snap point by a simple left-click.

Selection via RETURN Key:

If HiCAD requires the specification of a point, you can open the point options dialogue by pressing the RETURN key. The following point options menu is displayed:

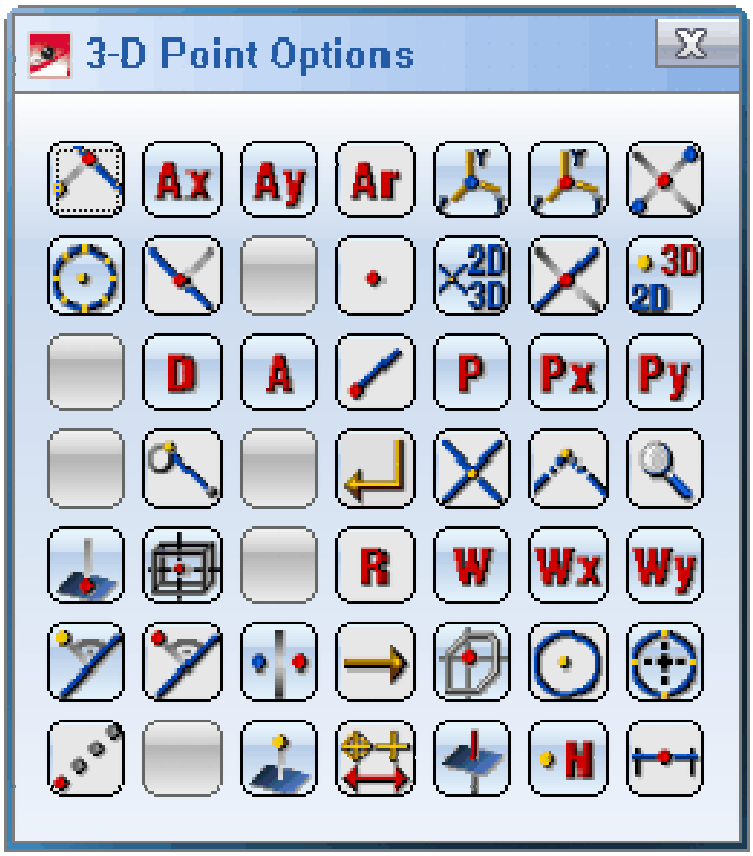


Fig. 18 Point options menu; activated, after suggestion of a point option by HiCAD, via RETURN key, quick double-click or RMB

Selection via Quick Double-Click:

If HiCAD requires the specification of a point, you can open the point options menu with a quick double-click (LMB). Please make sure that when you double-click, HiCAD does not suggest any snap points. As with the pressing of the RETURN key, the point options menu shown above is displayed.

Selection via Keyboard:

Numerical point options, i.e. point options for arbitrary coordinate entries and point numbers, as well as some angle functions can also be selected via the keyboard. These are all point options with one letter, i.e. R, A, D, P, W und N:

- Activation by simple entry of the point option letter with the keyboard (e.g. **A** for **Absolute** or **R** for **Relative**). The point option **Relative** (**R**) can also be activated with the space bar.

Please note: All point options with one or two letters, i.e. AX, AY, AR, D2, D, A, P, PX, PY, R, W, WX, WY and N, can be directly changed in the HiCAD pocket calculator, by simply overwriting the point option specification at the start of the input field (followed by a space).

Selection via RMB:

Move the cursor close to a graphical element, a dimensioning, or similar. As soon as the HiCAD Autopilot suggests a point option, press the right mouse button. The point options menu is displayed.

Complete list of point options in HiCAD:

	(E) Online to nearest line/edge, through point		(I) Snap point
	(AX) Absolute X-Position		(S) Genuine intersection point
	(AY) Absolute Y-Position		(S2) Theoretical intersection point
	(AR) Point from X- and Y-Position		(LU) Zoom
	(VI) From point I		(FF) Perpendicular to surface (only 3-D)
	(V) From point		(SP) Surface centroid
	(M2) Mid-point between 2 points		(SK) Body centroid (only 3-D)
	(ZP) Centre of polyhedral circle		(R) Relative coordinates
	(O) Online to edge through point		(W) Relative angle and distance
	(J) Isolated point		(WX) Relative angle and X-distance
	(SD) Intersection point 2-D/3-D lines		(WY) Relative angle and Y- distance
	(M) Midpoint of line/edge		(FL) Perpendicular base point on line/edge
	(ZD) 3-D point from 2-D drawing (only 3-D)		(F) Perpendicular base point on nearest line/edge
	(D) Distance from line/edge start		(G) Mirrored point
	(A) Absolute Position		Enter via keyboard
	(P) Absolute angle and distance		(Z) Centre
	(PX) Absolute angle and X-distance		(L) Last point
	(PY) Absolute angle and Y-distance		(OF) Online to surface through point (only 3-D)
	(SZ) Sketch		Switch snap radius on/off
	(T) Tangent		(KF) Intersection point Edge/Surface (only 3-D)
	(RET) Apply current cursor position		(N) Named point
	(QP) Quad point		(B) Dimensioning point

Fig. 19 Overview of 2-D and 3-D point options

11.2.2 Some Exemplary Point Options

The following paragraphs illustrate the exemplary point options “Relative coordinates”, “Online to nearest line and Distance from line start. All other point options are described in detail in the Online Help. You access the Online Help by pressing the F1 key or clicking the question mark symbol at the top right of the HiCAD user interface.

Relative Coordinates

Select these point options to set a point with X- or Y-coordinate values relative to an existing point (reference point). The reference point is the last entered point in the active function.

- Activate the 2-D point options.
- Choose point option **R**.
- Enter the values X and Y (e.g. 50 40).

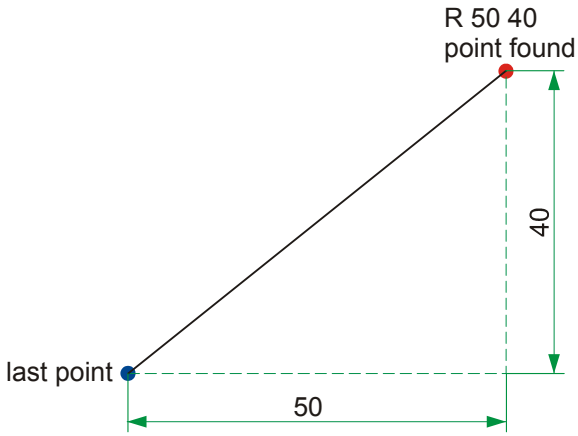


Fig. 20 Relative coordinates

You can now enter further X/Y coordinates to create the next relative point.

Values from the drawing can be taken over by using auxiliary codes:

Z Taking over of a value from the drawing,

L Taking over of the last distance

FA Specification of a scaling factor

Snap point

Use this point option to identify the point at the end of a line or the contact point of two lines (corner).

- Place the cursor in the vicinity of the line the end of which you want to identify.
- Activate the 2-D point options.
- Select point option **I**.

HiCAD identifies the line end or the line corner which is nearest to the cursor position.

Online to Nearest Line

Use point option **O** to set a point onto a line.

- Activate the 2-D point options.
- Select point option **O**.

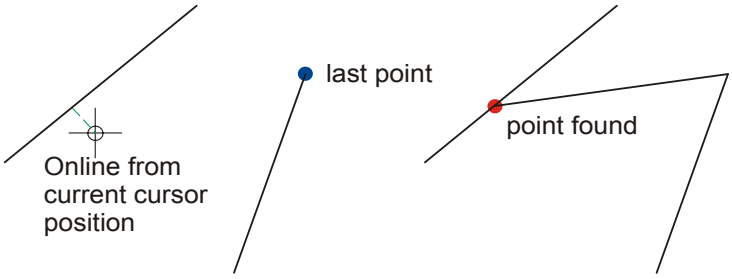


Fig. 21 Online to nearest line (shortest way from cursor position to nearest line)

With this option you can place the end point of a polyline exactly on a line. HiCAD takes the shortest way from the cursor position to the nearest line. This point option is frequently used for placing processing symbols on lines.

D

D

Distance from Line Start

Use this point option to set a point on a line element via the distance to the reference point. The reference point is the start point of a selected line.

- Activate the 2-D point options.
- Select point option **D**.
- Enter the distance. If you do not enter a distance, HiCAD offers the distance from the start point to the end point of the line.
- Identify the reference line in the vicinity of the start point on that side of the line from which you want to define the distance for the new point on the line.

The side of the start point determines the direction of the distance from the start point to the new point on the line. If you enter a distance of 40 mm, and select the reference line in the vicinity of the left start point of the line, HiCAD moves 40 mm from the left start point along the line (from left to right) and places the new point there. If you enter a negative value, HiCAD moves away from the line.

The value is saved internally and remains available for a possible re-use as “Last distance”, auxiliary code **L**.

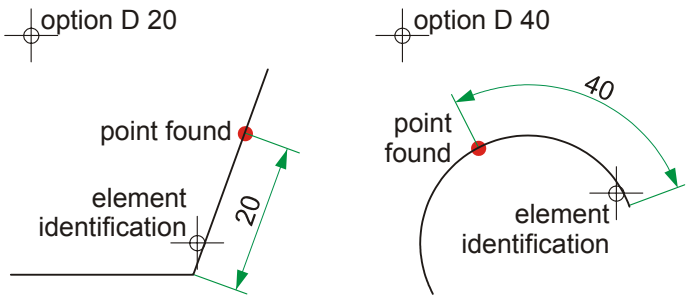


Fig. 22 Distance from line start

Instead of entering a value for the distance, you can also take the distance over from the drawing, by using, as for point option **R**, the auxiliary codes **L**, **Z** or **FA**.

12 How to Use the Mouse Buttons

HiCAD is operated via the mouse. You use the mouse buttons as follows:

Mouse Button	Function
Left (LMB, RET) 	■ Activation of part, elements, beam/profile groups or other objects (depending on the current identification mode) ■ Marking/Selection of objects ■ Calling of functions ■ Confirmation of entries (e.g. radii etc.) ■ Drag&Drop of views, dimensions, texts etc. (press and hold down mouse button) ■ Zooming of drawings (press and hold down mouse button)
Right (RMB, END) 	■ Activation of context-sensitive menus: ● If a function containing sub-functions is active, a pop-up menu with appropriate functions is displayed. ● If an object has been selected, a context menu with appropriate processing functions for this object is displayed. For example, if you right-click a dimensioning, a context menu with dimensioning function appears, if you right-click a view, a context menu with view functions is displayed etc. A right-clicked object is automatically activated. ■ Special functions (see info bar)
Middle (MMB, ESC) 	■ Cancelling of functions ■ Scrolling of drawings (mouse wheel) ■ Dynamic rotation of views (If you have only a mouse with 2 buttons, you can, instead of pressing the MMB, press the LMB and RMB simultaneously. If you have a wheel mouse, you use the wheel as MMB.)

13 Select Functions via the Keyboard

You can use the keyboard of your computer, or various keyboard shortcuts to call functions:

Key allocations for the processing of drawings

↑, ↓, →, ←	Move (translate) drawing detail
Page up, Page down	Zoom in, Zoom out
Home	View all
ESC	Cancel
End	Define detail
Del	Redraw
Ins	Rotate view
F1	Activate Online Help
F2	Dynamic zoom
F3	Hide/show 3-D coordinate system
Ctrl+C	Create graphic detail from drawing and copy to clipboard (EMF or Bitmap)
Ctrl+K	Move 3-D CS representation (see F3)
Ctrl+N	Create new drawing
Ctrl+O	Open drawing
Ctrl+P	Print
Ctrl+S	Save drawing
Ctrl+Z	UNDO
Ctrl+⇧+Z	REDO
Ctrl+6	Variables memory
Ctrl+7	Create macro
Ctrl+8	Call macro
Ctrl+W	Create graphic detail from drawing and save/copy to clipboard
Ctrl+Alt+W	Identical with Ctrl+W
0	3-D Processing plane
1	Only show active 3-D part
2	Show all parts
3	Standard axonometry
7	Glass model
8	Shaded without edges

Key allocations for the display of parts in the ICN

↑, ↓	Switch active part, 1 row up / down
Page up, Page down	Move to top/to bottom
Pos1	Jump to first item
End	Jump to last item
Del	Delete item in part structure
Ctrl+A	Select all parts
Ctrl+C	Copy part / part list to clipboard
Ctrl+F	Find
Ctrl+V	Paste part / part list from clipboard
Ctrl+X	Cut part / part list
Ctrl+Z	UNDO
Ctrl+⇧+Z	REDO

14 More Functions and First Excercise

14.1 Change Point Options without Ending the Line Function

Many point options are “infinite” functions. An example: You have selected the **Polyline (2-D)** (without sketch!) function and use the **Relative coordinates (R)** point option. As long as your polyline does not leave the currently selected detail of the drawing area, you can, after confirming the X- and Y-coordinates, use the **Relative coordinates** point option again. If you want to continue the polyline with a different point option, e.g. **Absolutes angle + distance (P)**, left-click on the drawing area, then press the RETURN key to access the point options. You can then continue with a point option of your choice, e.g. with **Absolute angle + distance (P)**.

Tip 1: Point options can be directly overwritten. You can, for instance, overwrite the letter “R” and the space with the letter “P”. (do not forget to re-enter the space!).

Tip 2: Let us assume that you have left-clicked on the drawing area and pressed the RETURN key to access the point options. You could also have right-clicked and selected the **Polyline > Continue** function in the context menu. After this, you could also have pressed the RETURN key to get to the point options.

Tip 3: After pressing the middle mouse button (MMB) you could have selected point option **(P)** by pressing the P key on your keyboard.

14.2 Insert Drawing Frames and Title Blocks

Use the **Insert drawing frame** to insert a drawing frame with title block as main part into the active drawing. You access this function either by selecting **Insert Part > Insert drawing frame** on the **Drawing** tab, or via the context menu of the drawing.

- Activate a view of the *Sheet* area.
- After selecting the **Insert drawing frame** function, the selection menu for DIN frames appears. Select an appropriate frame in DIN format and specify how you want to complete the title block.
- Specify the fitting points for the position of the frame. The title block is completed automatically (if the corresponding option was activated in the previous step).

The DIN frame is always inserted as a main part into the drawing. The part becomes not visible until you right-click on the name of the drawing in the middle window of the ICN (2-D Part structure) and select **Auxiliary parts On/Off > Auxiliary part On**.

If you have changed part master data and want to update the title block accordingly, use the **Update title block** function in the **HELiOS PDM functions for active drawing function group** of the **Helios PDM** tab. If you work without database, you also can complete the title block subsequently. On the **Drawing** tab, select **Extras > Tools > Complete title block**.

14.3 Save Drawing

To save the current drawing, select the **Save drawing** function.

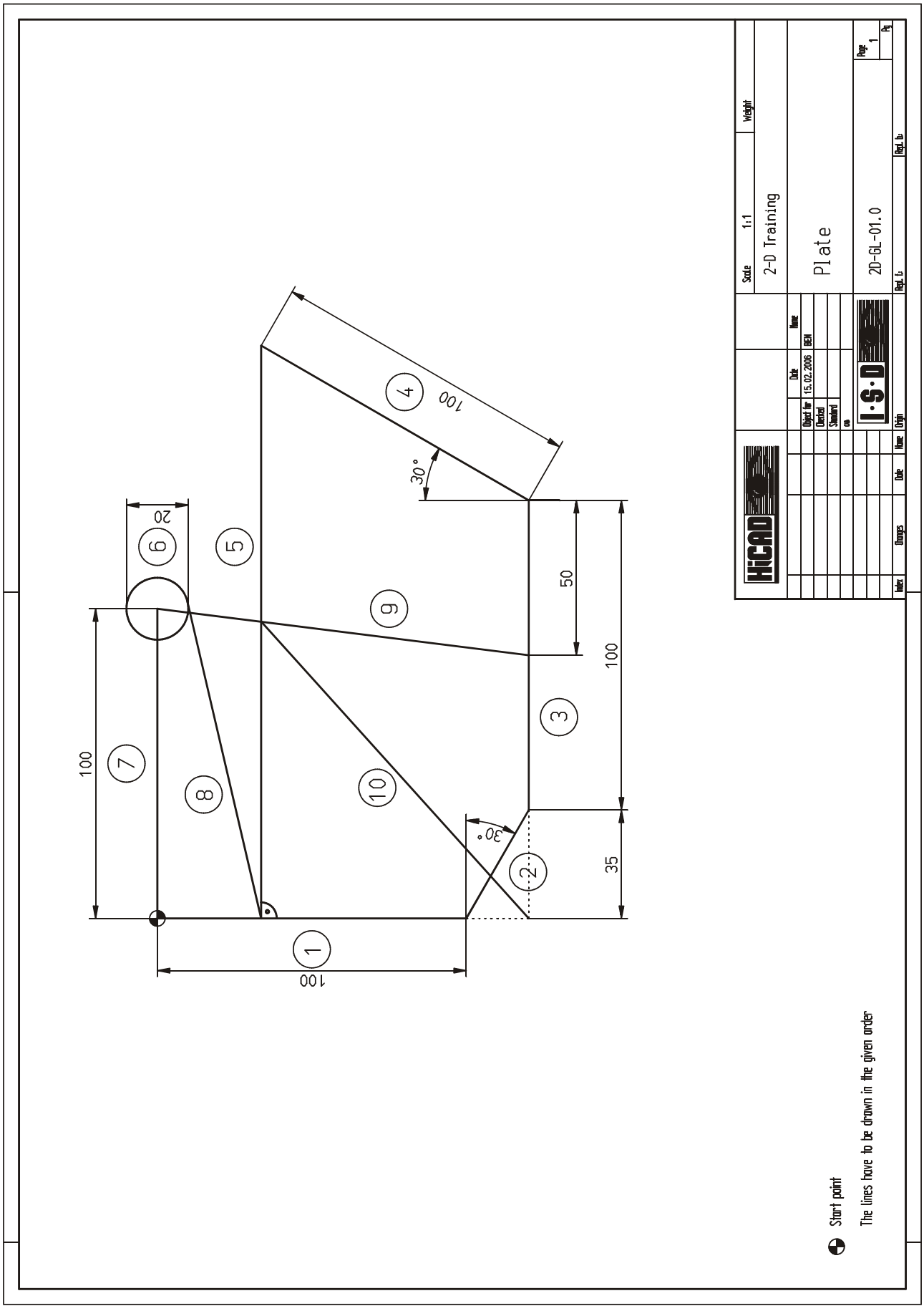
You have the following options to activate this function:

- On the QuickAccess toolbar, click the **Save drawing** symbol.
- Select the same-named function in the **Save/Print** function group of the **Drawing** tab.
- Select the keyboard shortcut **CTRL+S**.

If you want to save the drawing with a different name or via an Interface, select the **Save as …** function on the QuickAccess toolbar.

14.4 Exercise 1 (2D-GL-01.0) Plate

- Learning target: Creating a drawing, creation of main parts; purpose of the three mouse buttons; point options; different methods to select the point options (via the HiCAD Autopilot, Return key, keyboard etc.); functions (like View all, Rectangular zoom window etc.), which can be selected via the keyboard; drawing frame; save drawing.
- Create a new drawing with the database (additional module!). If you do not have a database in your company, you are enabled to switch the database function off on your training computer.
- Create the main part “Plate”.
- Create the graphic elements of the “Plate” exercise in the given order!
- Save the drawing.
- Space for notes:


Start point
The lines have to be drawn in the given order
Plate
2-D Training
2D-GL-01.0
Scale 1:1
Weight
HiCAD
I·S·D
100
100
50
100
35
20
30°
30°

15 Point Option R (RET)

R (RET)

If you do not want to refer to the last used point when using a relative point determination (i.e., for example: R, P, PX or PY), but select a different reference point, close the corresponding mask without any value entries (i.e. without making any entries, left-click on the drawing area or press the RETURN key). HiCAD first prompts you to specify the new reference point, and then to enter the coordinates.

This applies, for example, to the functions:

- R – Relative,
- P – Polar coordinates,
- PX – Polar coordinates with X-distance
- PY – Polar coordinates with Y-distance

Point Option R (RET) is by far the most versatile point option. It enables you, for instance, to replace almost all auxiliary lines.

In the following example, you want the corner B of the second rectangle to be positioned at a distance of 100 to the right and 25 to the bottom relative to the corner A of the first rectangle.

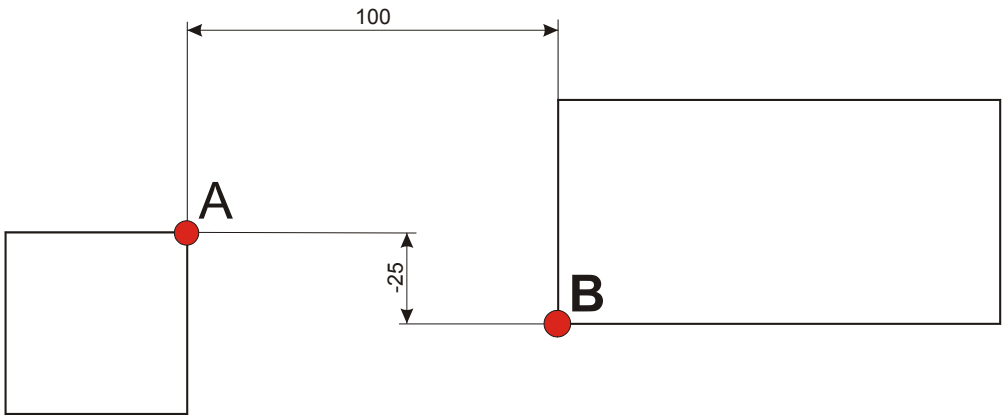


Fig. 23 Point option R(RET)

- Select point option R, e.g. by pressing the R key on the keyboard (do not enter any numbers).
- Press the RETURN key or left-click on the drawing area.
- Specify reference point: Corner A (see image above) with point option I
- Enter 100 for the X value, and -25 for the Y value.
- *End* point of rectangle with point option R (e.g. 120 60).

Instead of R(RET) you can also select point option V. If you select V, you specify a reference point with an arbitrary point option. An input field appears enabling you to enter an X- and an Y-coordinate to specify the movements from the reference point to the right, resp. to the top (for negative coordinates: to the left, resp. to the bottom).

Instead of R(RET) you can also select point option VIS. If you select VIS, HiCAD automatically uses the intersection point or the line end with is closest to the cursor, as reference point. An input field appears enabling you to enter an X- and an Y-coordinate to specify the movements from the reference point to the right, resp. to the top (for negative coordinates: to the left, resp. to the bottom).

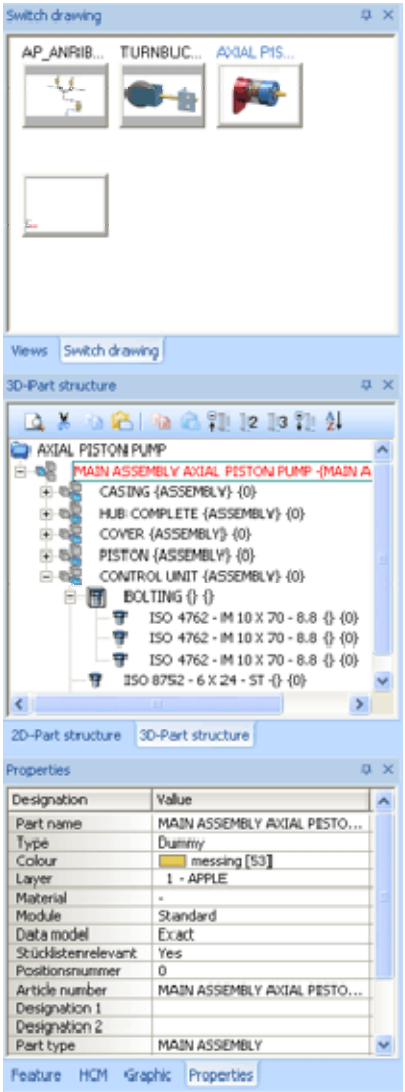
16 Notes on the Information and Communication Navigator (ICN)

An integral part of HiCAD is the so-called *Information + Communication Navigator* (short: ICN), which, in terms of versatility, surpasses common structure browsers by far.

The ICN enables permanent, direct access to all opened models, drawings and views, to assembly structures, attributes and PDM properties of the individual parts. The ICN identifies referenced parts appropriately and supplies information about parts which have changed or are currently processed.

View structures are also visualised, including all dependencies between views and 2-D/3-D part structures, as well as between 3-D models and list views. You can switch between views in the ICN, without having to activate any view function.

The design process can take place while the ICN is open; the navigator display is constantly synchronised with and adjusted to the construction object in the drawing area. All structures and most of the properties can be directly processed in the ICN via mouse click. In addition, many part processing functions can be activated by right-clicking a part in the ICN. Multiple selection options (as known from Windows) enable you to change several parts at once, with one single processing step.



Upper ICN window

Tabs: Views, Switch drawing

Middle ICN window

Tabs: 2-D Part structure, 3D-part structure

Lower ICN window

Tabs: Feature, HCM, Graphic, Properties

Fig. 24 The *Information + Communication Navigator* (Example)

HiCAD can also manage objects without part character, e.g. auxiliary lines, DIN frames, etc. In the middle window of the ICN, right-click on the name of the drawing and select the **Auxiliary parts On/Off** function in the context menu of the drawing.

For more detailed information on the ICN please read the “The Information + Communication Navigator” chapter of this training book.

17 Main Parts and Sub-Parts

The structure of main parts and sub-parts (part structure) is a basic idea of all HiCAD modules. Therefore, it is mandatory that you carefully read and understand the contents of the following two pages.

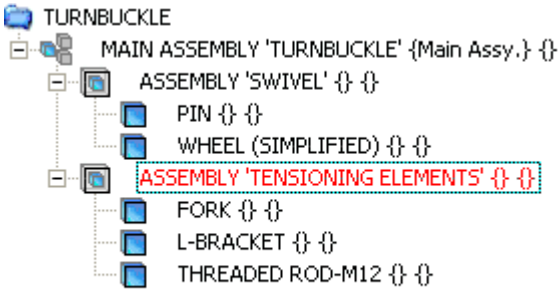


Fig. 25 Example of a 3-D part structure

Please note:
Pin and Wheel are mounted to each other, i.e. they belong to one assembly (in our above example, this is the *Assembly “Swivel”).* Via this Assembly “Swivel”, they can be moved, copied, deleted etc. together.
In the same way, Fork, L-Bracket and Threaded rod form a structural unit. Therefore, they both belong to the *Assembly “Tensioning elements”.*
Both assemblies together constitute the product “Turnbuckle”, which is represented in the part structure as the *Main assembly “Turnbuckle”.* All parts of the drawing belong to this main assembly. This means that if you move the main assembly, all subordinated assemblies and their parts are automatically moved (copied, deleted, etc.).
If, for example, you decide that the Thread rod is not mounted to Fork and L-Profile, but belongs to the Swivel, you can move it below the *Assembly “Swivel”.*
In the same way, you may want to assign further (subordinate) assemblies to the *Assembly “Swivel”.* Please note however that in the 3-D module you can create only one main assembly per drawing.

Please also note:
In the 2-D module there are no assembly / main assembly parts with an explicit assembly ID. The parts are however structured in the same way into main assemblies, (sub-)assemblies etc. You can, for example, create a normal 2-D main part and call it *Main assembly “Turnbuckle”,* or a sub-part and call it *Assembly “Swivels”.*

17.1 Interactions and Automatisms within Part Structures

- Transformations: If, for example, a part is translated or rotated, its sub-parts are automatically moved as well.
- Overlaps: You can determine for entire 2-D parts whether they should be placed in front of or behind another part. You do not need to do this for each single line of the part (in 3-D, HiCAD automatically knows the depth of the parts).
- Hatching: Part hatchings do in no case require any rework. Sub-parts (e.g. bores, cut-outs, bolts etc.) are automatically excluded from hatching. When creating assembly drawings, contours often change due to part overlaps. Even in this case, rework is not required, as the part hatching depends on the part structure, and not on the line contours.
- Re-use: If you re-use a part in another drawing, all information inherent in this part (hatching, overlap level, master data for bills of materials, part attributes etc.) are taken along with it. The automatic transfer of master data, for instance, guarantees that in the new drawing, too, no part is missing in the bill of materials.
- Bill of Materials: Can be created “at the push of a button”, as names of the parts as well as corresponding master data are already defined in the drawing.

17.2 When Do You Use Main Parts, When Sub-Parts?

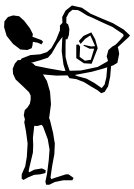
How do you structure a drawing into main parts and sub-parts?

Part rule: All parts which are, physically, one part (i.e. parts you can pick up with your hands, e.g. a flange or a bolt), are regarded as individual parts. Reason: If you want to move or re-use this part, you can do so in one step and do not need to move each line individually. The next two rules decide whether you turn this part into a main part or sub-part:

Main part rule: If a part is independent and does not belong to any other part in the drawing, it is a main part. Please note that in the 3-D module you can create only one main assembly per drawing.

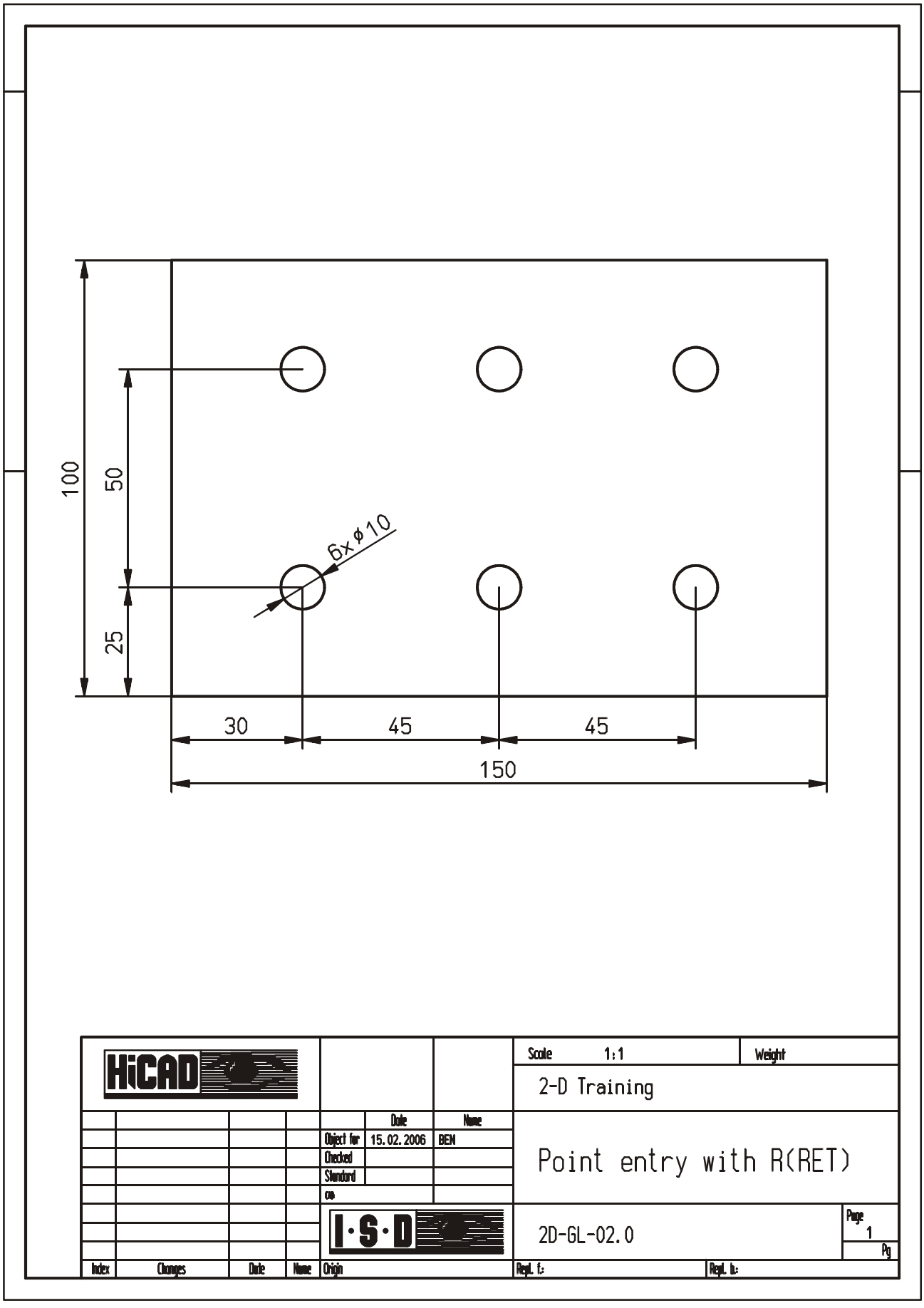
Sub-part rule: All parts that belong together are assigned to a so-called assembly. They are sub-parts to this assembly or main assembly. Bores and bolts always belong to a workpiece, which means that they are always sub-parts (these are automatically created for standard parts and construction aids).

17.3 Exercise 2 (2D-GL-02.0) Input of Point Options with R(RET)



- Learning target: Create a drawing; point option R(RET); purpose of the three mouse buttons; different methods to select the point options (via the HiCAD autopilot, Return key, keyboard etc.); functions (like View all, Rectangular zoom window etc.), which can be selected via the keyboard; activate sheet view; insert drawing frame and save.
- Create a new drawing with the database (additional module!). If you do not have a database in your company, you are enabled to switch the database function off on your training computer.
- Create a main part “Assembly Wire Bending Cantilever”.
- Create a sub part “Base plate 150x100” and draw it.
- Create a sub part “6 Stay Bolts”.
- The point option R(RET) is the most extensive point option, e.g. you can replace nearly every auxiliary line with it. Therefore, use exclusively the R(RET) option to position all 6 stay bolts!
- Save the drawing.

Hint: Please pay particular attention to the main and sub-part structure. This is the basic idea of all HiCAD modules. Main parts are parts which are “independent”, i.e. which do not belong to any other part in the drawing. In this exercise the main part is the Assembly “wire bending cantilever”. Sub parts are all parts which belong to the respective superior part of the drawing. In this exercise the sub parts are the base plate and the stay bolts.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Weight
2-D Training
Date
Name
Object for 15.02.2006 BEN
Checked
Standard
Point entry with R(RET)
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Repl. b.:

18 Point Options P(RET), PX(RET), PY(RET)

P

P (RET)

The training book chapter “Point Option R (RET)“ provided an introduction to the same-named point option: First you needed to select the point option “Relative coordinates (R)”, and close the mask without entering a value, by pressing the RETURN key. You could then specify a reference point. From this reference point, you moved to the right, resp. to the top using X- and Y-coordinates.

Px

PX (RET)

Analogously, you can use the P(RET) function for the “Absolute angle + distance (P)“ point option. As for R(RET), you close the corresponding mask without entering any values, by pressing the RETURN key or left-clicking on the drawing area. HiCAD then prompts you to specify the new reference point. After that, HiCAD will not ask for an X- or Y-coordinate, but for an angle and an absolute length.

Py

PY (RET)

If you select PX(RET) instead of PX, (or PY(RET) instead of PY), the same prompts as for point option P(RET) will appear. Only the entered length will not be interpreted as absolute length, but as X- (or Y-)length.

19 2-D Construction Aids and Standard Processings

Various construction aids are available for your drawing. These can be found in the **Construction Aids** function group of the **2-D Part** tab. These construction aids are fitted as independent parts (i.e. as sub-parts to the active part), they are however usually not shown in the part structure, as they are no “parts” in the proper meaning of the word, but rather processings of parts. The ICN displays only those parts which would also turn up in a bill of materials.

Use the functions of the **Construction Aids** function group to draw, besides creating shortenings and details,

- Bores, countersinks, threads and shaft processings
- Hole patterns, slots and punchings,
- Shortenings and details.

HiCAD will automatically create a sub-part. If it is a “genuine” part (nut, washer etc.), this sub-part is shown in the ICN. If it is not a “genuine” part, but a processing (bore, countersink, feather key etc.) this sub-part is not shown in the ICN, although it exists as a sub-part. It would, for example, be transformed (translated, rotated etc.) together with the superordinate part.

You can however visualise automatically created main parts and sub-parts: In the middle window of the ICN, right-click the name of the drawing and select the “Auxiliary parts On/Off” function.

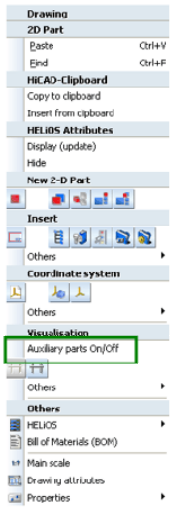


Abb. 26 Right-click name of drawing and select “Auxiliary parts On/Off”

The selected parts can be inserted into the active drawing repeatedly if required. Right-click to end the insertion and select a different construction aids.

Bore, Top View and **Bore, Side View**

Use these functions to create an individual bore for a through hole with a crosshairs. The bore can be inserted repeatedly, if required.

- Enter the diameter of the bore.
- If you select *Bore side view* HiCAD prompts you to specify the length.
- Select the insertion direction.
- Specify the fitting point in the drawing.

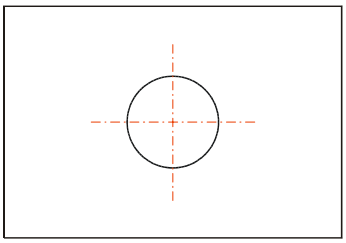


Fig. 27 Through hole

Right-click to get back to the function selection, a click on the middle mouse button ends the insertion.

Circle

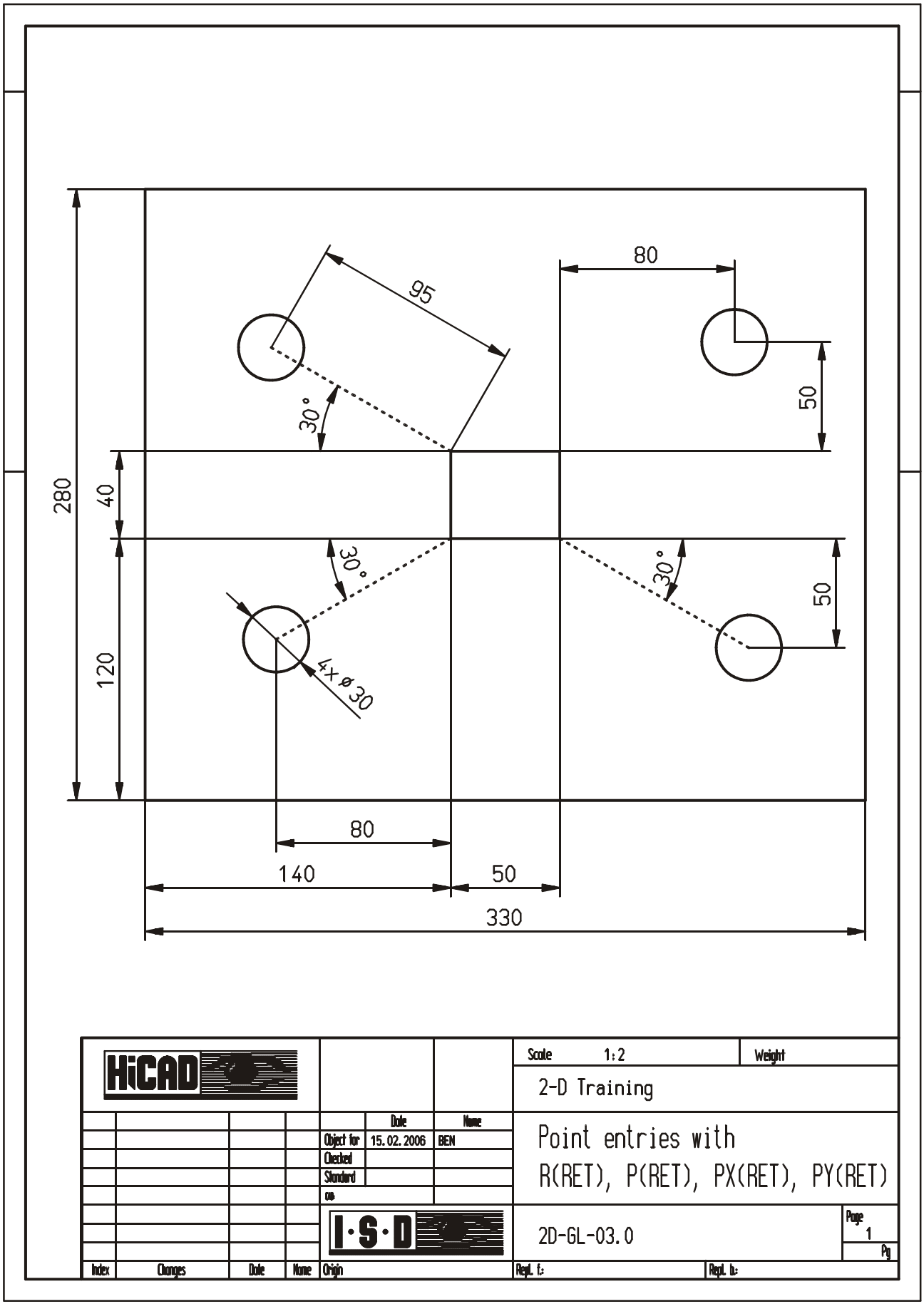
The functions **Polyline**, **Rectangle** and **Circle** can be found in the **Others** sub-menu of the **Construction Aids** function group. These functions are used in the same way as described for the bores. Please note that here as well, it is assumed that the construction aids are used for the creation of *processings*. The sub-part created by HiCAD will therefore remain invisible in the part list of the ICN. It can however be used like a part *in the drawing* (e.g. by right-clicking it and selecting one of the processing functions from the context menu, such as Translate part, Rotate part etc.).

When inserting a Construction Aids or Standard Processing (e.g. Bore, Hole pattern, Circle etc.) HiCAD automatically creates a sub-part. By this means it is ensured that if, e.g., you translate a sheet, all bores within the sheet are automatically translated as well. Besides, you can translate, rotate, and mirror the bore in the drawing as an individual part without having to move each single line. To do this, right-click the bore and select one of the processing functions from the context menu. Construction Aids and Standard processings are not regarded as “genuine” parts, but as *processings*. Therefore the sub-part created by HiCAD remains invisible in the ICN, where only “genuine” parts are displayed.

Before inserting the processing (bore etc.), the user needs to activate the part to which the processing should be applied (e.g. a sheet). Although HiCAD creates the sub-part automatically, HiCAD needs to know to which part the sub-part is applied. If you right-click the sheet, it is activated automatically. Irrespective of whether a sub-part is created by yourself or automatically by HiCAD when inserting processings: Sub-parts are always created for the currently active part.

19.1 Exercise 3 Input of Point Options with R(RET), P(RET), PX(RET), PY(RET)

- Tip: If there is not much time, you trainer should only show you this exercise.
- Learning target: Point options R(RET), P(RET), PX(RET), PY(RET); Construction Aids, create a drawing; add title block and save; purpose of the three mouse buttons; different methods to select the point options (e.g. by pressing the RETURN key on the keyboard or by using the right mouse button in the moment when HiCAD suggests a point option etc.); functions (like View all, Rectangular zoom window etc.), which can be selected via the keyboard, various methods to select functions (e.g. select the Construction Aids via the corresponding icon in the user interface or via the right mouse button at the part in the drawing or via the right mouse button at the sheet in the ICN).
- Draw the sheet.
- Use the “Rectangle“ and “Circle“ functions in the Construction Aids. Use the point options R(RET), P(RET), PX(RET) and PY(RET) to move them to the right position.
- Please remember that the Construction Aids are not only available as an icon on the user interface, but also via the context menu of the part in the drawing (right mouse button) respectively via the context menu of the part in the ICN.
- Display the auxiliary parts in the ICN (right-click on name of drawing in the middle ICN window).
- Save the drawing.
- Space for notes: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20 Dimensioning and Information

20.1 Dimensioning

The 2-D Dimensioning functions can be found on the **2-D Dimensioning** tab. If you right-click a dimensioning in the active drawing, a context menu containing further dimensioning functions is displayed.

A dimensioning is composed of the following elements:

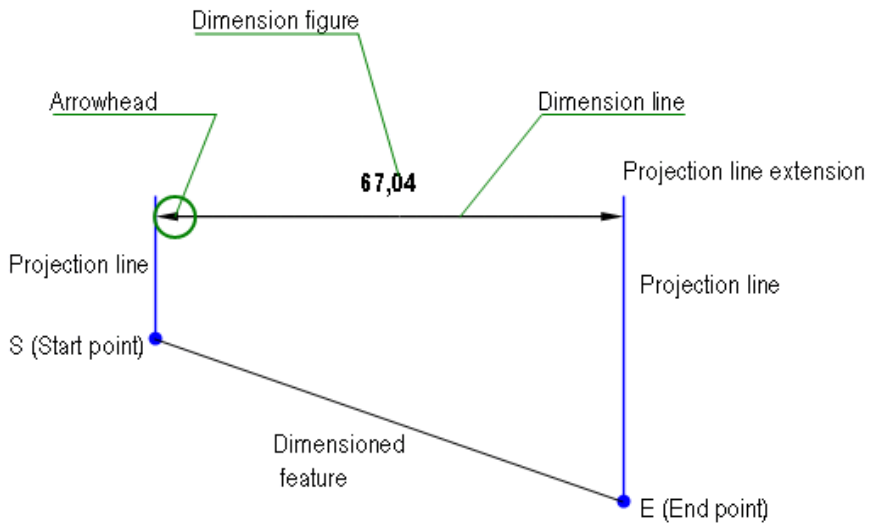


Fig. 28 Dimensioning

Each newly created dimensioning is assigned to the active part. This also applies if the dimensioned line does belong to a different (inactive) part.

Please always activate the part you want to dimension. This is the only way to guarantee the associativity of the dimensioning. “Associativity” means that dimensions are automatically updated if form or position of the dimensioned part are changed.

The following paragraphs will deal with some exemplary dimensioning functions. For a more detailed description of the dimensioning functions, please read the relevant chapter of the Online Help.

20.1.1 Linear Dimension, Axially Parallel (2-D)

Use the **Linear dimension, axially parallel** function, as well as the other functions from the **Parallel** function group to create linear dimensions which are parallel to the X- or Y-axis. Axially parallel dimensionings will retain their axially parallel position even if the part is rotated. Select the distance you want to dimension by identifying two points. Some dimensioning functions also allow the selection of a line.

For identification, all point options are available. You can identify any point in the drawing. They do not necessarily need to be end points of a distance or line, but can also belong to different parts.

20.1.1.1 Linear Dimension, Parallel Dimensioning Points

Use this function, or the other functions of the **Direct** function group to create a linear dimension which is parallel to the dimensioned points (i.e. normally not parallel to the X- or Y-axis).

- Identify the two points.
- Enter the third point for the position of the dimensioning.

Axially parallel dimensions retain their position even after rotation of the corresponding part. Linear dimensions which have been created parallel to the dimensioning points, are rotated with the part.

If you want to dimension a line, you do not need to select point option I for the start and end point. It will suffice if you left-click, *without* any point option, in the vicinity of a line end. If a dimensioning function is active, HiCAD will automatically select the nearest line end. Please note: This is *only* applies to dimensioning functions! If you draw a *line* and then click in the vicinity of an already existing line, the end of the old line will not be selected as the beginning of the new line; HiCAD would start the line at the position of the cursor. For all non-dimensioning functions, the end of a line can only be selected via point option I !

20.2 Information Tab

If you do not dimension a part, but only a length, a radius or similar, you can also use the functions provided by the **Information** tab.

All information stored in the system can be called via the functions of the **Information** tab. They enable you to detect point coordinates, distances, angles text attributes, dimensioning attributes, the contents of the variables memory, and much more.

The values calculated with these functions are passed to system variables in the variables memory. For example, if you measure a length or a radius with the functions of the **2-D Distances** function group, the result will be saved to the system variable %Z3. This may be of interest if you use the Information menu in a macro and require the measured length for another function (e.g. for a part, in order to translate a length measured with the help of the Information menu).

The following paragraphs deal with some exemplary functions of the Information menu. All other functions for measuring of distances are similar and are described in detail in the Online Help.

20.2.1 2-D Distances

Use the functions of the **2-D Distances** function group to detect distances between:

- Points,
- Line elements,
- Radii,
- Points and line elements.

The calculated distance is assigned to system variable Z3.

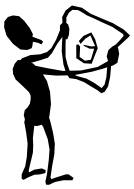
20.2.1.1 Information, Distance 2 Points

Use this function to measure the distance between arbitrary points.

- Use the point options to identify the two points.

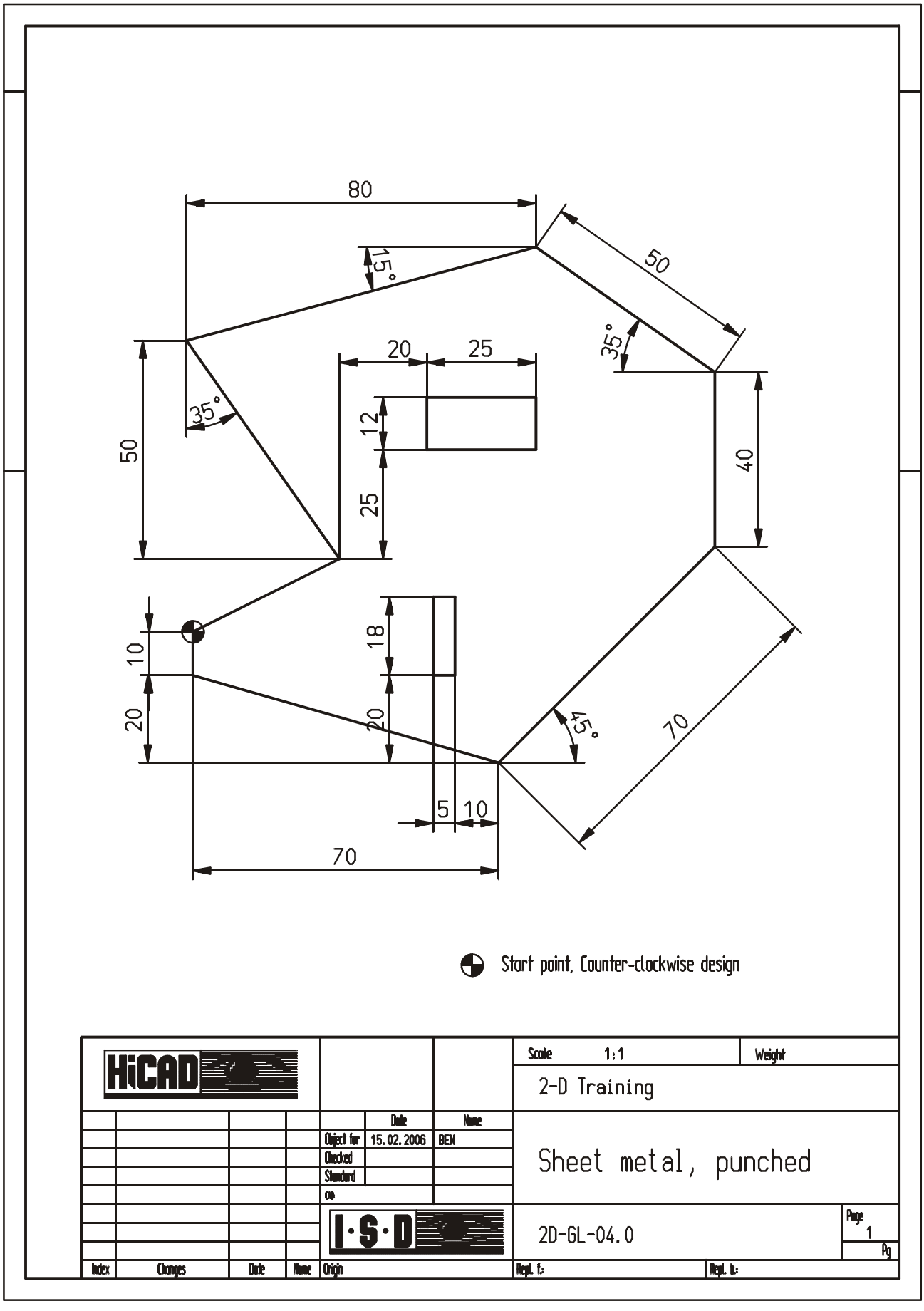
The distance between the points is shown in the text window. As with all numerical values, you can copy the value with a right-click to paste it to another input field or to a text file, if required.

As a preparation for the next exercise, please read the answer to the question “How do I draw a continuous polyline if I require different point options?” in the “FAQ” section at the beginning of this training book. Further information on this topic can also be found in chapter 14.1 “Change Point Options without Ending the Line Function“.



20.3 Exercise 4 (2D-GL-04.0) Sheet Metal, Punched

- Learning target: Creating a new drawing; add drawing frame and save; various point options, different methods to select the point options (e.g. by pressing the RETURN key on the keyboard or by using the right mouse button in the moment when HiCAD suggests a point option etc.); purpose of the three mouse buttons; Construction Aids, functions (like View all, Rectangular zoom window etc.) which can be selected via the keyboard, various methods to select functions (e.g. select the Construction Aids via the context menu (right mouse button) at the punching in the drawing area, via the context menu of the punching in the ICN or via the corresponding icon in the user interface).
- Create a main part for the punched sheet metal. Use the point options you learned.
- Try to close the polyline without interruption.
- Use the Construction Aids to add the punchings to the sheet metal part.
- Dimension the punched sheet metal. Always activate the main or sub-part you are dimensioning at the moment.
- Save the drawing.
- Space for notes: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Object for 15.02.2006 BEN
Checked
Standard
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21 Load Drawing, With or Without Database

21.1 Load Drawing

In chapters 10.1 and 14.3 you have learned how to create and save drawings. Now we want to load a drawing from the database.

To load an already existing HiCAD file from the database (additional module), select the **Load drawing, with DB** function. If you work without database, select **Load drawing**.

The *Load drawing* function can be accessed:

- via the QuickAccess toolbar,
- via the **New/Open** function group of the **Drawing** tab,
- via the keyboard shortcut **CTRL+O**.

After that, select the required file.

If you have already loaded and processed a file, but have not saved it yet, HiCAD asks you whether you want to save this drawing.

If you want an already loaded file to remain open, select an empty field in the **Switch drawing** tab of the ICN by a double-click. You can open up to 18 drawings in HiCAD.

You can also load HiCAD drawings via the Windows Explorer.

21.2 Filters in the Database Mask

If you want to display all drawings which have been saved to the server, these might well be tens of thousands of drawings. Therefore it makes sense to narrow the search by setting appropriate filters.

Procedure WITH Database (additional module):

Deactivate the two filters suggested by HiCAD for Links. As we have not linked our drawing to anything, this filter would not make any sense for us. Instead, define your own filter in one of the input fields of the database mask. This can be, e.g. 2* in the “Document number” field, or B* in the “Designation” field. You will then only be shown drawings whose document number begin with a 2, or whose designation begins with a B.

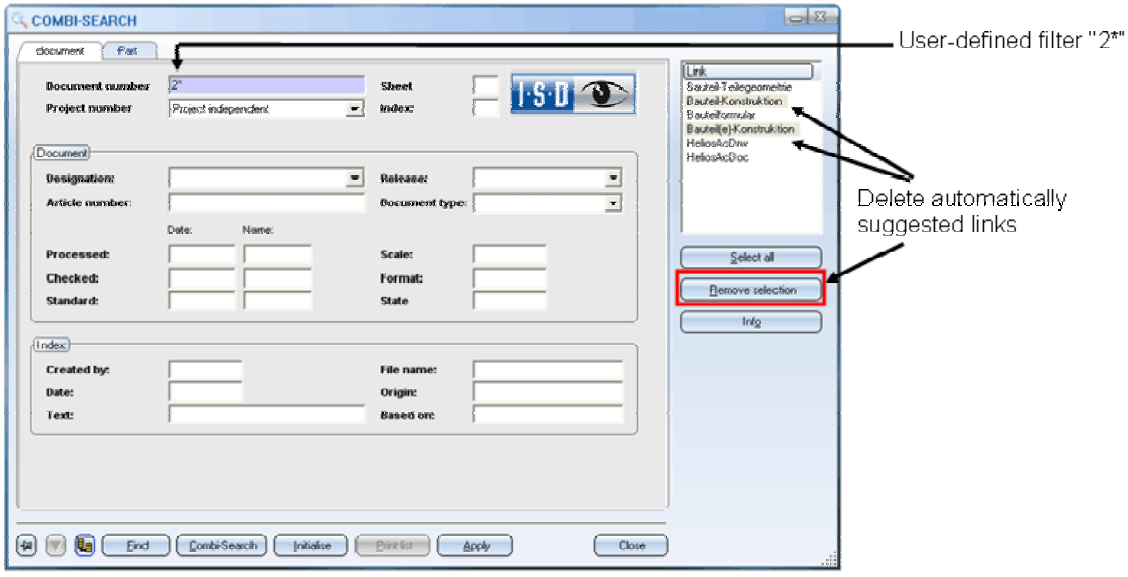


Fig. 29 Loading of a drawing via database (additional module) with filter “2*”

However, even a list of all drawings whose document number begins with a 2 can still be quite long. You can however sort this list according to all attributes, such as document number, designation, revision date etc.: Click the header of the column according to which you want to sort (i.e. Document Number, Designation etc.). If you click twice you obtain a reverse sorting of the list.

R..	Document No.	Sheet	Ind...	Designation	Document type	Release	Changed	Created on
	DN-000001	1		Entwurf	HiCAD Konstruktion	In Arbeit	24.10.2006 17:55:56	02.10.2006
	DN-000003	1		Montagezeichnung	HiCAD Konstruktion	In Arbeit	24.10.2006 17:55:59	02.10.2006
	DN-000005	1		Montagezeichnung	HiCAD Konstruktion	In Arbeit	24.10.2006 17:56:02	02.10.2006
	DN-000006	1		Montagezeichnung	HiCAD Konstruktion	In Arbeit	24.10.2006 17:56:04	02.10.2006
	DN-000007	1		Montagezeichnung	HiCAD Konstruktion	In Arbeit	24.10.2006 17:56:06	02.10.2006

Fig. 30 Result list after loading of drawing with filter 2* and sorting according to Designation

Procedure WITHOUT Database:

If you work at your company without database, you probably will have switched off the database on your training computer. After selecting the **Load drawing** function, the Windows mask with the file names of all drawings is displayed. Further selection criteria are not available when working without database. You can however select the **View menu** icon. In the following menu, select **Details**.

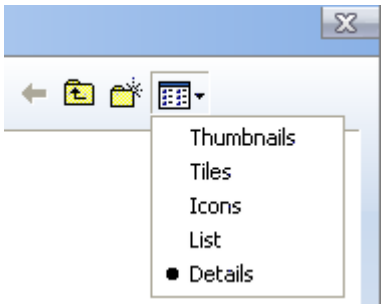


Fig. 31 Working without database: Further search criteria

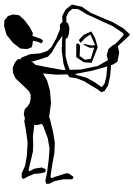
Click **Name** or **Created** in the header of the result list to sort the list according to name or date of creation. If you click **Created** twice, not the oldest, but most recent drawing will appear at the top of the list.

If you possess the additional database module, you can learn about all further functions of the document/part/BOM database in a separate database training.

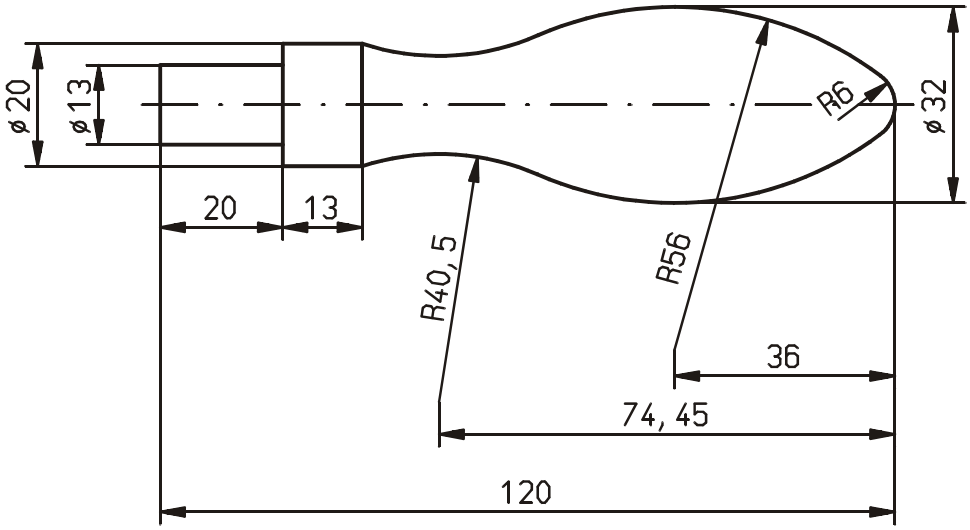
21.3 Continuation Exercise 4, Sheet Metal, Punched

- Load Exercise “Sheet Metal, Punched” from the database (if the database is switched off, please load without database). Use one (or several) search criteria and sort the result list as described in Chapter 21.

21.4 Exercise 5 (2D-GL-05.0) Handle



- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Various point options, especially R(RET); mirror lines; circles at boundary lines and points; centre line.
- All exercises so far could be dealt with by using the function R (RET). For the construction of the handle, the use of auxiliary lines is permitted.
- Familiarise yourself with the subsequent changing of the drawing scale.
- Dimension the part. Please remember that the 2-D circular dimensioning can *only* be used for circles. If you intend to use a diameter dimension for a rectangle, you will find an appropriate function in the pull-down menu of the Dimensioning Tools function (in the “Tools” function group of the “2-D Dimensioning+Text” tab). Do not only use the point option I (respectively a click close to the line without point option) for dimensioning, but also the point options Z (Centre) and QP (Quad Point).
- Space for notes:



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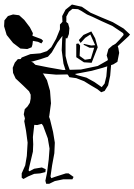
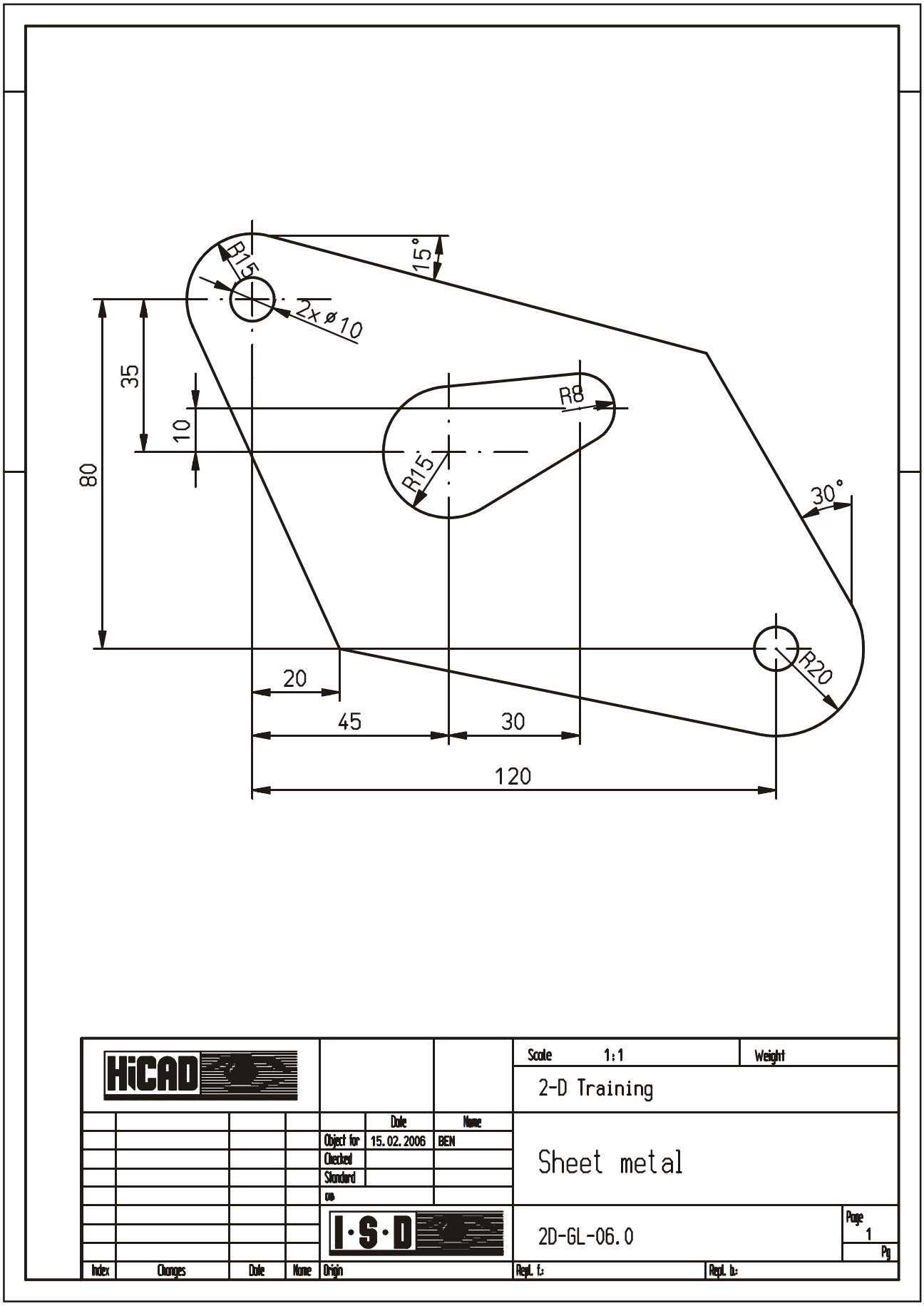
21.5 Exercise 6 (2D-GL-06.0) Sheet



- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Different types of tangents. Point options, in particular R(RET); dimensioning of the appropriate part; select various kinds of functions (e.g. Construction Aids).
- In order to design this sheet, use the following functions one after the other circle, tangent and trimming graphic elements.
- Use Construction Aids for the bores. Please remember that the Construction Aids cannot only be called via the icon in the user interface, but also via the context menu of the part in the drawing area (right mouse button) respectively via the context menu of the part in the ICN.
- Create a part for the recess too, in order to make it invisible in the ICN by default. The recess is not a part, but a processing. Please use the Construction Aids only once. Draw all further lines of the recess using the standard 2-D line menus to prevent the emergence of several sub parts for the recess. During this process, the design part needs to be active.
- Activate the appropriate main part or sub-part when you dimension it. Display the Construction Aids in the ICN if necessary.
- Space for notes:


R15
15°
2x ø10
35
10
80
R8
R15
30°
R20
20
45
30
120
HiCAD
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Weight
2-D Training
Date
Name
Object for 15.02.2006 BEN
Checked
Standard
ok
Sheet metal
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Name
Origin
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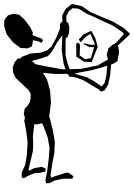
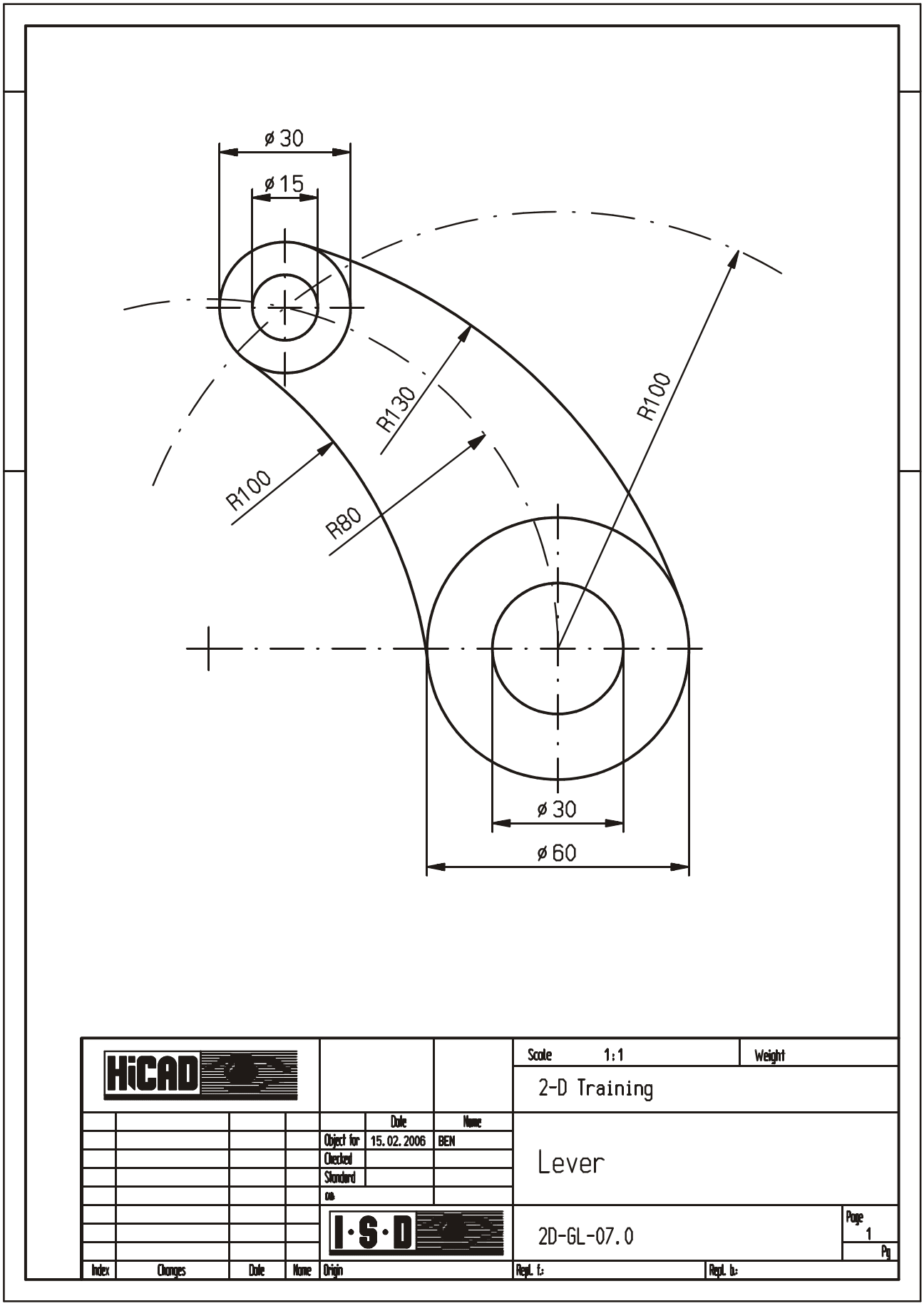
Repl. b.:



21.6 Exercise 7 (2D-GL-07.0) Lever

- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Circles, circular arcs; various methods to trim 2-D lines; centre cross; dimensioning.
- Create a new drawing and a main part “lever”.
- Use the different options to design circles and tangents.
- Correct these afterwards to create the lever.
- Dimension the part.
- Space for notes:


ø 30
ø 15
R130
R100
R100
R80
ø 30
ø 60
HiCAD
Scale 1:1
Weight
2-D Training
Date
Name
Object for 15.02.2006 BEN
Checked
Standard
as
Lever
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Repl. f.:
Repl. b.:

22 Automatic Part Dimensioning

In the **Tools** function group (between Process and Text) of the **2-D Dimensioning** tab, you will find the **Dimensioning Tools > Part** function.

This function generates dimensioning suggestions for the active part.

This function is explained in detail in the Online Help. Owing to the multitude of possible combinations, we will concentrate on three particularly important entries that will be required for the next exercise:

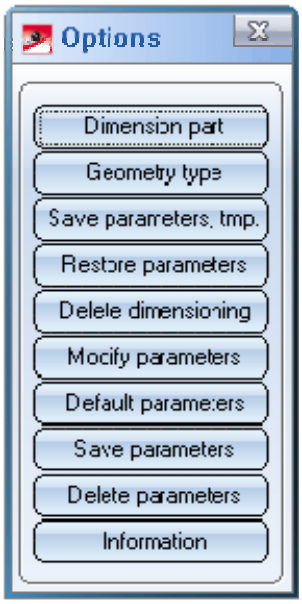


Fig. 32 Automatic part dimensioning

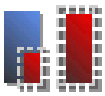
In the menu shown above, select **Geometry type**, then select **Closed contour**.

Now select **Modify parameters**. For **Linear dimensions: Type** select **Parallel dimensions** twice. (for X- and Y-direction).

Right-click to return to the first dialogue again. Select **Dimension part**.

23 Detail

Activate the **2-D Part** tab. In the **Others** pull-down menu of the **Construction Aids** function group you will find the **Detail** function. Use this function to define a detail as a part and insert it into the drawing with a scale of your choice.



- Specify the scale, then select the area for the detail. The detail is displayed enlarged.
- Select one fitting point on the detail and one on the drawing area. The selected detail is identified by a dashed circle.
- Specify the reference line start point and end point for the designation of the detail.
- Specify a name for the detail.
- Specify the position for the designation of the detail in the drawing

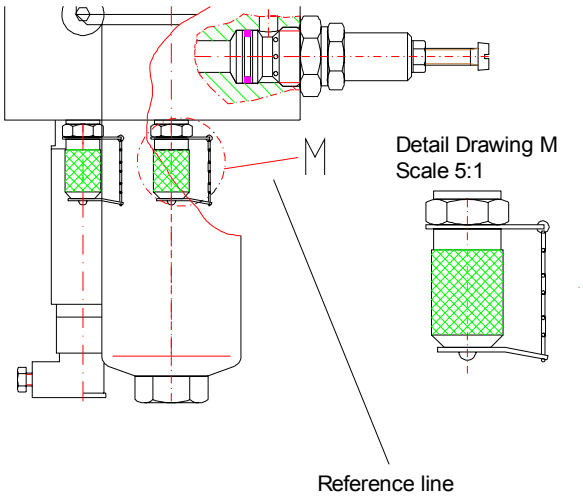


Fig. 33 Detail

24 Use Symmetries

Many parts in Mechanical Engineering are rotation-symmetrical or mirror-symmetrical. In such cases, it will suffice to draw only one half of the part. The other half is obtained by cloning (translating, rotating, mirroring) of the part. Some parts even have double symmetries. In the exercise “Sheet with hole pattern” you only need to draw one quarter, the rest of the part is obtained by mirroring it twice.

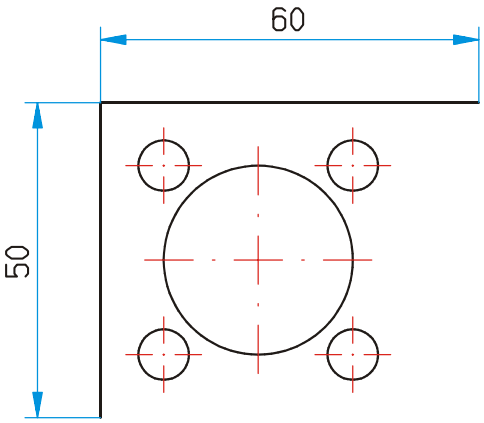
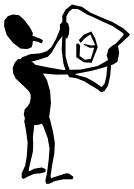


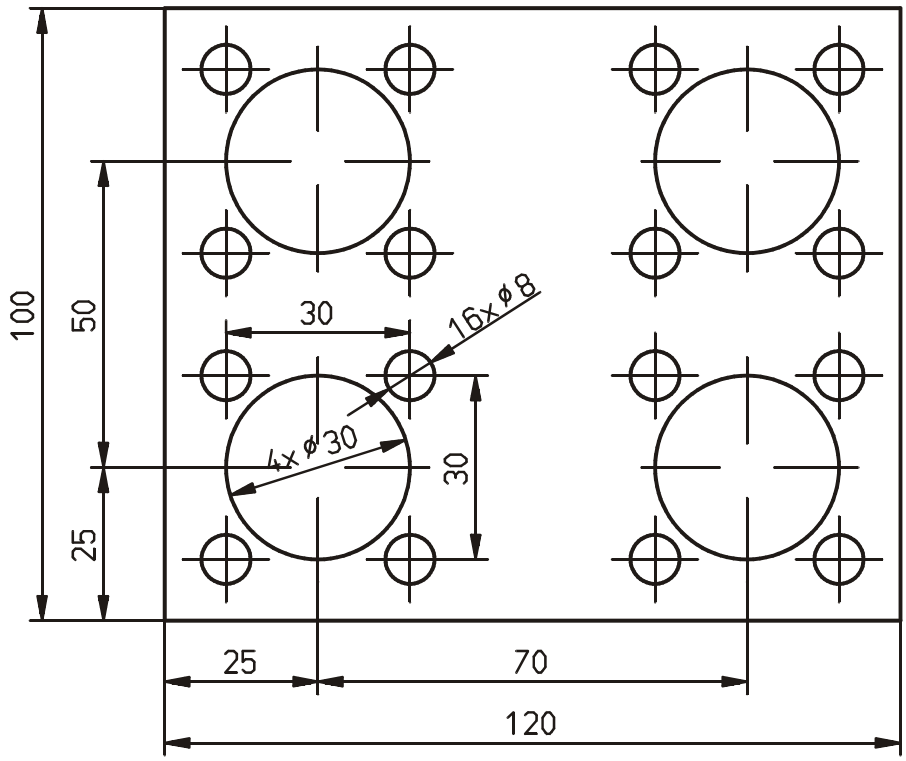
Fig. 34 Sheet with hole pattern before mirroring the part twice

Tip: Before drawing a part, you should check if the part contains any rotation or mirror symmetries. If so, it will suffice to draw only one half or even one quarter of the part. The rest can be created with the **Clone part… (2-D)** function.

24.1 Exercise 8 (2D-GL-8.0) Sheet with Bore Pattern



- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Recognise symmetries. Watch out for one or more mirror or rotation symmetries in the workpiece before drawing it. In that case you only need to draw half, or – as in this exercise – only a quarter.
- Draw a quarter of the sheet metal only.
- Use the Construction Aids for the appropriate bores.
- Create the complete sheet metal by mirroring the part “Sheet Metal” twice.
- If required, change the part structure of the bores with the “Break up” functions or by copying of lines. The required functions are described in detail in “Chapter 29: Break Up Parts“ and “Chapter 28: Move Lines and Dimensions to Another Part“ of this training book.
- Dimension the part with the automatic part dimensioning function.
- Create a detail of the part (not shown in the drawing).
- Alternative solution: If you had drawn the complete sheet metal instead of a quarter, you had been able to draw a bore with a diameter of 30 and a quad bore diagram with a diameter of 8. In that case you would have had to mirror the bore pattern and the individual bore separately twice.
- Space for notes:



HiCAD		Scale 1:1	Weight
		2-D Training	
	Date / Name		
	Object for 15.02.2006 BEN	Sheet metal with bore diagram	
	Checked		
	Standard		
	on		
I·S·D		2D-GL-08.0	Page 1 Pg
Index / Changes / Date / Name	Origin	Repl. f.:	Repl. b.:

25 Hatching

25.1 General

HiCAD offers two options for the hatching of closed contours:

- Part-based hatching (lines in one part)
- Contour-based hatchings (lines in several parts).

When using part-based hatching, a surface bounded by a closed polyline is hatched. The line elements of the closed polyline must be contained in one part.

The polyline can have any shape and complexity and consist of up to 2500 line elements. If a polyline consists of more than 2500 line elements, it needs to be decomposed into several parts, and each closed surface needs to be hatched separately.

Line elements which are distributed among several parts, do not form a proper closed polyline, even if they are optically continuous. Part-based hatchings can therefore not be applied to them.

Standard parts, in contrast, are particularly well-suited for part-based hatchings.

When using contour-based hatching, the inside of closed contours is hatched; the line elements can however belong to different parts. For this type of hatching, you need to specify a hatching reference point. On the one hand, the contour-based hatching is more flexible than the part-based hatching, but on the other hand the detection of the corresponding contour is mandatory. And in contrast to part-based hatching, sub-parts and texts are not automatically excluded here.

When using part-based hatching, all parts located below the hatched part as sub-parts, are automatically excluded from hatching (provided that they form closed polylines). The same applies to texts and dimensioning within the hatched part.

The contour-based hatching considers only those exclusions whose hatching reference point lies in the same part as the exclusion element.

You can link texts and dimensionings to hatching exclusions – they will be changed together with the exclusions.

If the part-based hatching is selected, the corresponding hatching code is entered into the active part. The line elements all belong to the same part.

If the contour-based hatching is selected, the hatching code is entered into the active part; the position of the corresponding contour elements is, however, arbitrary.

You can define arbitrary exclusions in hatchings.

The hatched contour can also be filled with an arbitrary symbol. This is not possible for part-based hatchings.

To speed up the drawing process you can also hide hatchings.

In the following example the parts have been filled with different hatching codes:

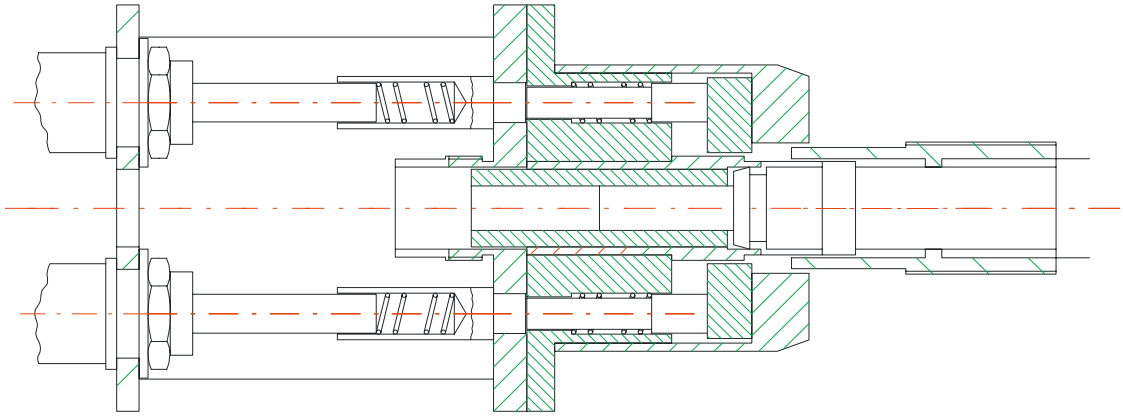


Fig. 35 Hatched part

25.2 Part Hatching

If you select Part hatching, a surface bounded by a closed polyline is hatched. The line elements of the closed polyline must be contained in one part. A polyline can consist of up to 2500 line elements.

You cannot apply part hatching to surfaces which do not form a closed polyline.

- Activate the part you want to hatch.

You can also activate a part that has already been hatched. The existing hatching will be replaced.

- Select the **Change/Define part hatching** function.

The input mask for hatching attributes is displayed.

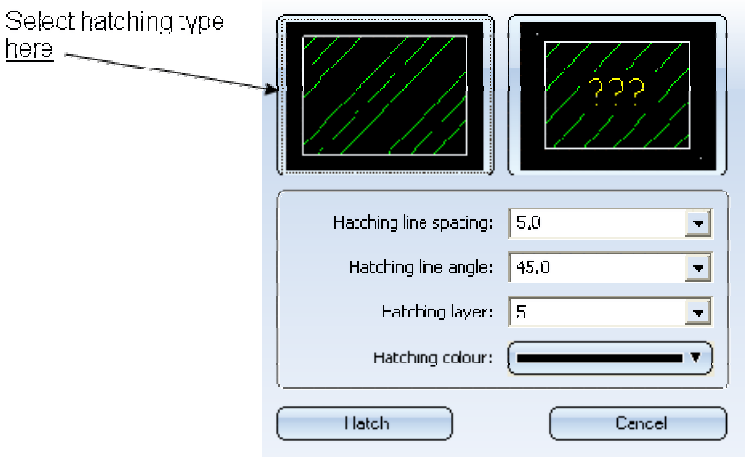


Fig. 36 Input mask for hatching attributes

Select the hatching type (see above image), the hatching line spacing, the hatching line angle and hatching colour and confirm with **Hatch**.

If the part has no own line elements, but serves only the purpose to structurally combine the sub-parts, the hatching code is automatically entered into all sub-parts which are on the same level as the part to be hatched.

Part and contour hatchings can be deleted not only via the **Delete hatching** function in the **Hatching** function group on the **2-D Part** tab, but also via the pop-up menu of the **Delete object** function on the **Drawing** tab. For both options please note that there are *different* functions for the deletion of part hatching and contour hatching.

25.3 What if Part Hatching Does Not Work?

25.3.1 Sort GE (2D)

The **Sort GE (2D)** function can be found in the function group **Tools** on the **2-D Geometry** tab. You can use this function to connect polylines which form optically a closed contour, but not according to the internal data structure. The restructuring of line elements takes place only within one part. No line element is assigned to another part.

For example, if you draw a rectangle, but select the **Polyline** function after two lines again, you will start a new polyline. This means that the contour appears optically closed, but according to the internal data structure it is not.

By applying the **Sort GE (2D)** function, the polylines will be connected, thus forming a “genuine” closed contour (i.e. they will also be closed according to the data structure).

25.3.2 Double Lines Within a Part

If there are points of contact between two parts, the contact line need to be a double line, as each of the parts has to form a closed contour if you hatch them, if they overlap or if you want to calculate their areas. However, if there are double lines within one part, they are superfluous. Sometimes they can even be the reason why hatching does not work.

Activate the **2-D Geometry** tab. In the **Others** pull-down menu of the **Change** function group, select the **Delete overlaps** function. Double lines **within one part** are deleted. Double lines at points of contact between two parts are however (for logical reasons) retained. It may happen that a hatching does not appear until after a repeated selection of the **Sort GE (2-D)** function (see training book chapter 25.3.1).

Please note: If you select the **Double lines** function in the **Delete** function group of the **2-D Geometry** tab, only double lines *of equal length* are deleted.

25.3.3 Check for Closed Contours

In the **2-D Others** function group of the **Information** tab you will find the **Polyline** function, enabling to check whether a contour is actually closed. A closed contour is, for example, a prerequisite for the creation of hatchings and overlaps.

If there are gaps in a contour, you can close them with the following functions:

25.3.3.1 Close corner, internal corner

The **Change** function group of the **2-D Geometry** tab provides numerous functions for the processing of line elements. Many of these functions are contained in the **Others** pull-down menu (small blue arrow), which looks as follows:

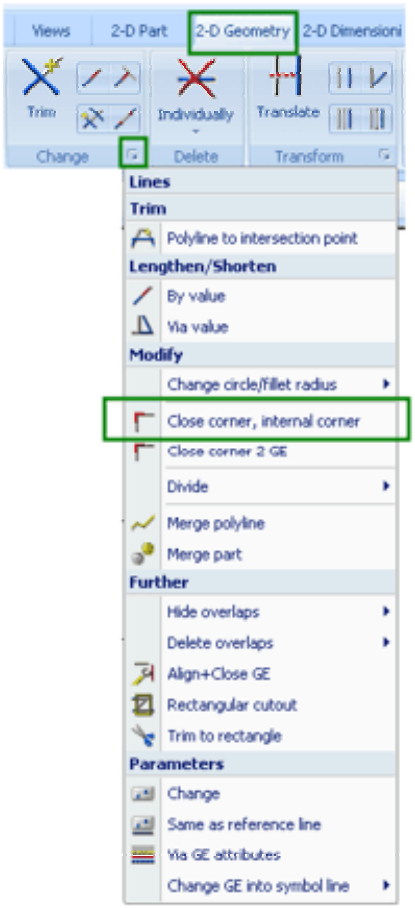
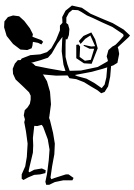


Fig. 37 **Close corner, internal corner** function

The **Others** sub-menu contains, among many other functions, the **Close corner, internal corner** function. If you click a corner from its internal side, both lines are lengthened in such a way that they form a “genuine” closed corner. If this was the only non-closed place in your contour, it is closed now and can be hatched. Besides, you can also use functions such as **Lengthen/Shorten GE, to line (2-D)** to close contours.

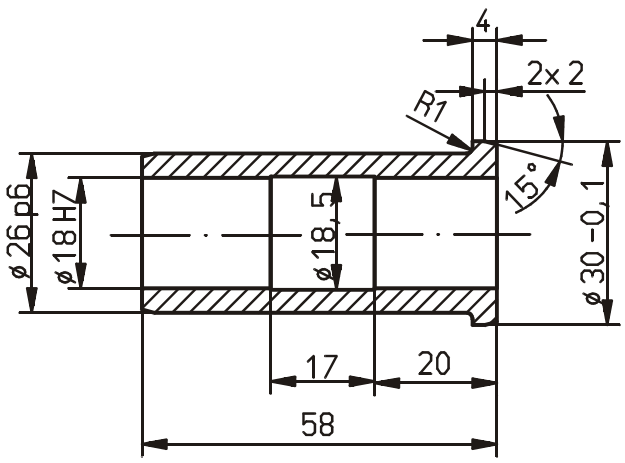
If you needed to delete another double line or close another corner, it is recommended to sort the contour once again with the **Sort GE (2-D)** function!

25.4 Exercise 9 (2D-GL-9.0) Spacer Sleeve



- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Part hatching; chamfer dimensioning; dimensioning of diameters, updating of the fits table by saving the drawing.
- Create the spacer sleeve. Please remember that you need to be closer to the line where the length of the chamfer is supposed to be with the identification point when using the “Chamfer with Length + Angle” function. If the length is to be marked off of the upper line, you need to select the upper line from the inner side. If the length is to be marked off of the left line, you need to select the left line from the inner side.
- Use the *Part* hatching function to hatch the part.
- Use the Construction Aids or Standard Processings functions for the creation of the bores.
- Dimension the workpiece. Activate the part that you are dimensioning at the moment. The pull-down menu of the Dimensioning Tools function (in the “Tools” function group of the “2-D Dimensioning+Text” tab) provides an appropriate function for diameter dimensioning of non-circular parts (e.g. revolved parts in a side view).
- Add tolerances and fits as well as the corresponding tables.
- Change one of the fits.
- Update the fits table by saving the drawing.
- Space for notes:

Normal size		allow. above	allow. below
18.000	H7	+0.018	0
26.000	p6	+0.035	+0.022



HiCAD						Scale 1:1	Weight
						2-D Training	
					Date	Name	
				Object for	15.02.2006	BEN	Spacer sleeve
				Checked			
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				as			
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26 Overlaps and Exclusions

26.1 Overlap Parts

The **Overlap** function group of the **2-D Part** tab contains various functions for the assignment and processing of overlaps. If you click the **Others** icon of the function group (small blue arrow), a pull-down menu with further functions is displayed.

Use the **Overlap part** to define parts as overlap contours.

- Identify the part that you want to be overlapped (i.e. the part you want to send to the back).
- Select the overlapping part (i.e. the part you want to bring to the front) and end the identification with the middle mouse button (or with the right mouse button if you want to stay in the context menu).

Now specify how you want to arrange the hidden edges. For this you use the functions **Hidden edges dashed**, **Hide hidden edges** or **All edges visible**. These functions are described in detail in Chapter 26.3.

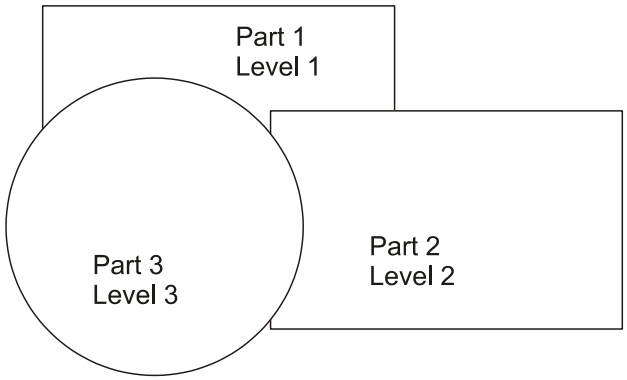


Fig. 38 Overlapped parts; Setting: “Hide hidden edges”

26.2 Define Exclusion Contour

Use the **Define exclusion contour** function in the **Overlap** function group of the **2-D- Part** tab to find define the contour of an exclusion, i.e. a bore in a sheet or similar.

Requirements for this are an overlap contour in the active part and a contour which can be defined as exclusion.

- Identify the contour start for the exclusion.
- Press the middle mouse button to end identification.
- Select the **Redraw** function to display the exclusion.

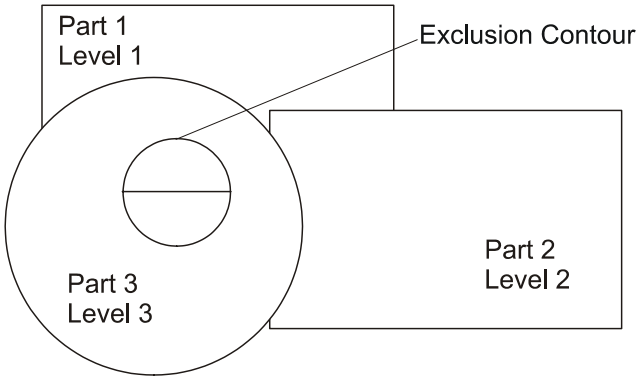


Fig. 39 Exclusion

26.3 Type of Representation for Hidden Edges

26.3.1 All edges visible

Use this function in the Visualisation menu to display all edges, including the hidden ones.

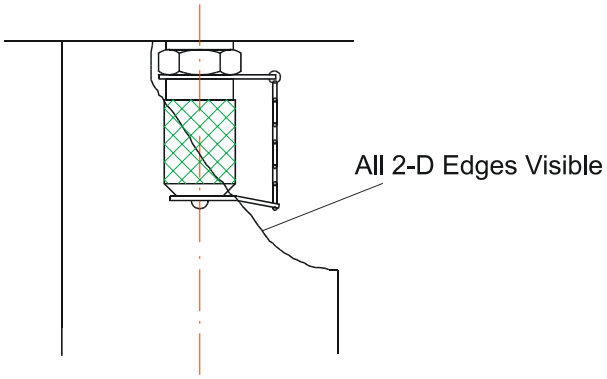


Fig. 40 All edges visible

This function applies to all overlaps of the currently processes drawing.

26.3.2 Hide hidden edges

Use this function to hide all “hidden” edges, for all overlaps.

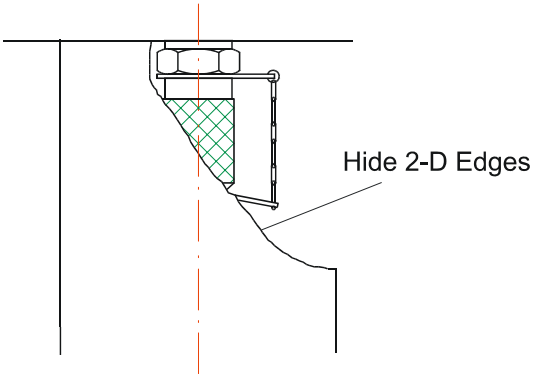


Fig. 41 Hide hidden edges

26.3.3 Hidden edges dashed

Use this function to display all hidden edges by an arbitrary line type. The line type can be chosen after selection of the function.

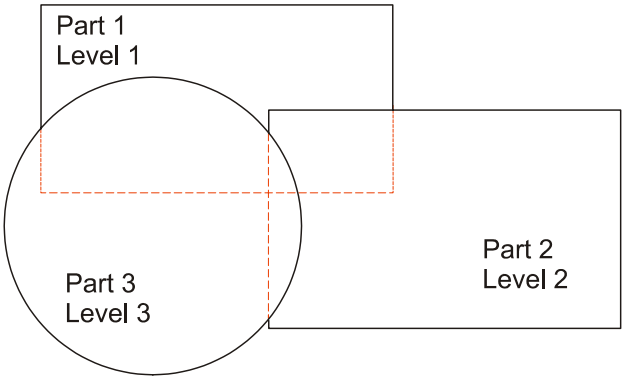
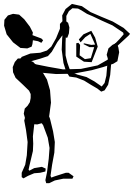


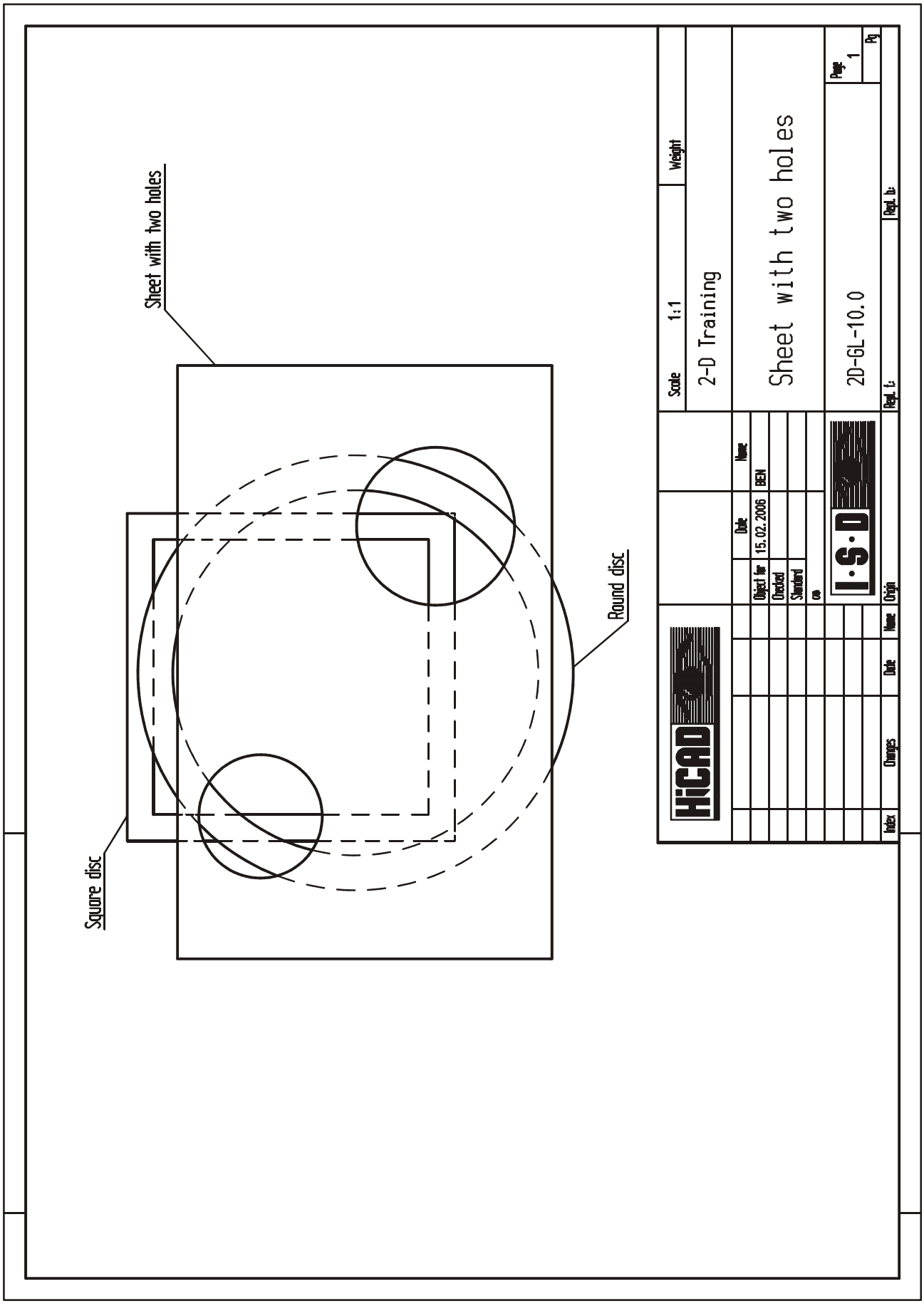
Fig. 42 Hidden edges dashed

This function applies to all overlaps in the currently processed drawing.



26.4 Exercise 10 (2D-GL-10.0) Sheet with 2 Holes at Specified Positions

- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Overlap 2-D parts.
- Create the three parts in front of one another.
- Define the overlaps and exclusion contours.
- Dash, respectively hide the overlapping lines.
- Space for notes:


Sheet with two holes
Square disc
Round disc
Scale 1:1
Weight
2-D Training
Sheet with two holes
2D-6L-10.0
Name
Date
15.02.2006
BEN
Object for
Checked
Standard
as
Page 1
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Index
Changes
Date
Name
Origin
Repl. to:
Repl. by:

27 Part Structure when Designing in Several Views

The basic idea of HiCAD is to create drawings with entire parts (main parts and sub-parts) instead of individual lines. This allows a fast and well-structured working, with many automatisms which would otherwise not be possible. This also has also proven helpful when designing in several views. Views are created as main parts, to which all corresponding parts are assigned as sub-part. Of course, these workpieces can in turn contain sub-parts.

Such a 2-D part structure can be seen below:

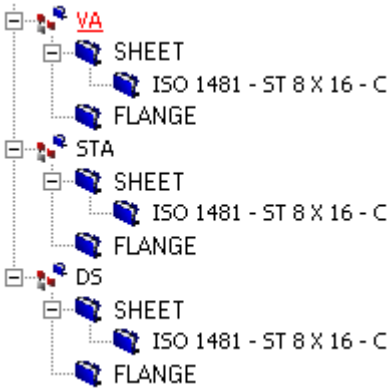


Fig. 43 Part structure when designing in several views

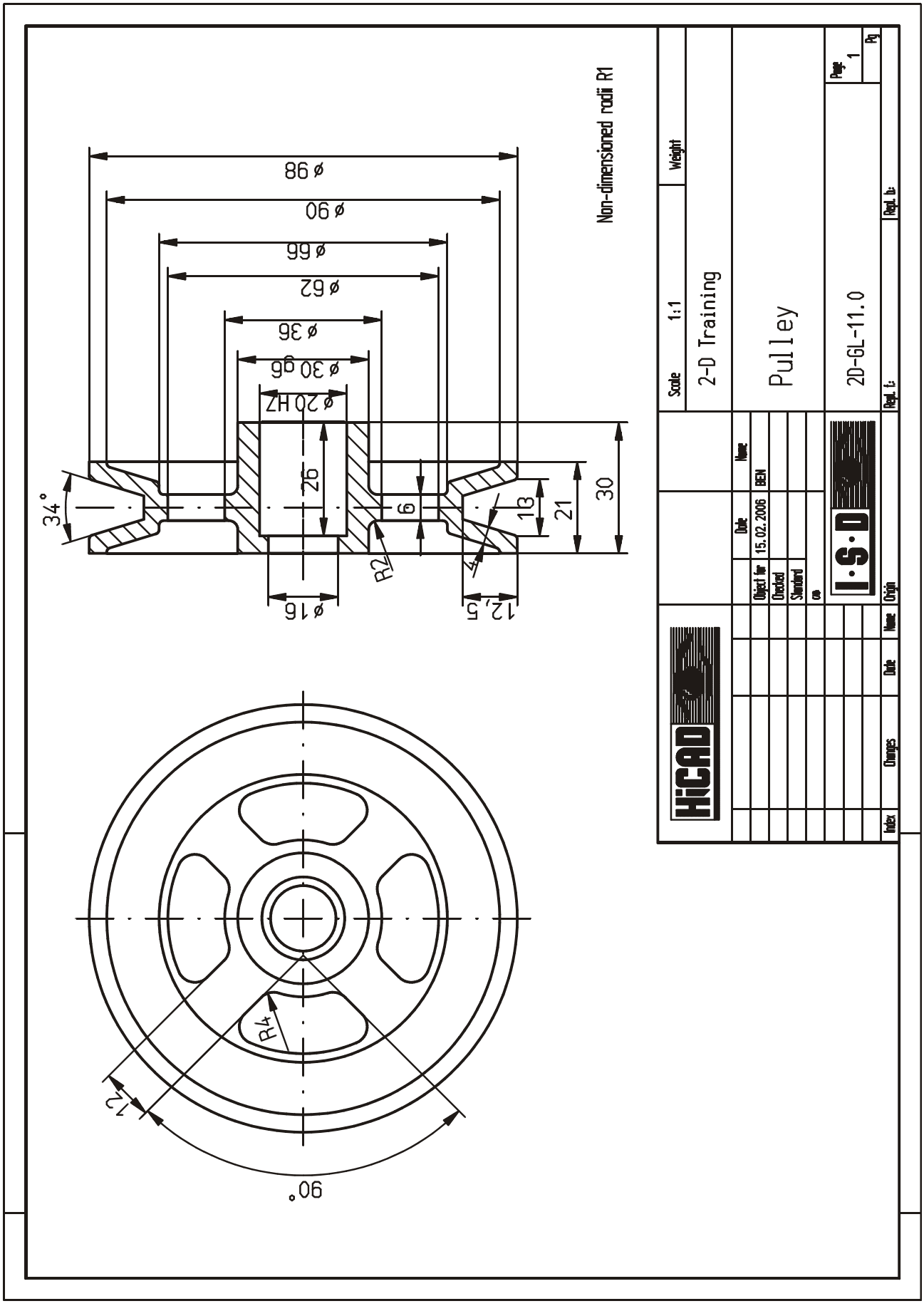
Please note that the views need to be main parts, and the workpieces sub-parts. Otherwise it would be very laborious to move a view: You would have to move, for each part, the sub-part VA (or STA and DS, respectively). In the part structure as shown above, one movement would suffice.

Fig. 44 Rendered HiCAD drawing

Fig. 45 Rendered HiCAD drawing

27.1 Exercise 11 (2D-GL-11.0) Pulley

- Please note: In order to save time, it is sufficient if the trainer shows you this exercise (in particular the part structure).
- Learning target: Part structure of 2-D drawings with several views.
- Create one main part “Front view” and one main part “Sectional view”. Create for each of the main parts one sub-part “Pulley”. To avoid confusions of the sub-parts “Pulley”, you should give them different names, such as “Pulley-FV” and “Pulley-SV”.
- Use *Part* hatching to hatch the part.
- Dimension the part.
- Space for notes:


Non-dimensioned radii R1
ø98
ø90
ø66
ø62
ø36
ø30 g6
ø20 H7
26
6
13
21
30
34°
R2
4
12,5
ø16
R4
12
90°
Scale 1:1
Weight
2-D Training
Pulley
2D-6L-11.0
Page 1
Pg
Name
Date
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as
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Index
Changes
Date
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Origin
Repl. to:
Repl. by:

28 Move Lines and Dimensions to Another Part

28.1 Take Over Elements to Active Part

In the **Others** pull-down menu of the **Tools** function group on the **2-D-Geometry** tab, you will find the **Elements > Take over to active part** function. This function retains the geometric position of the line elements, and changes only their belonging to a part, by inserting the line into the active part and deleting it from the previous part.

- In the selection menu, choose the type of identification (Individually, In rectangle, Polyline)
- Identify the line elements you want to assign to the active part.

In the image below, the line taken from part 1 is an element of part 2. You can check with the cursor if the upper line of part 1 now actually belongs to part 2:

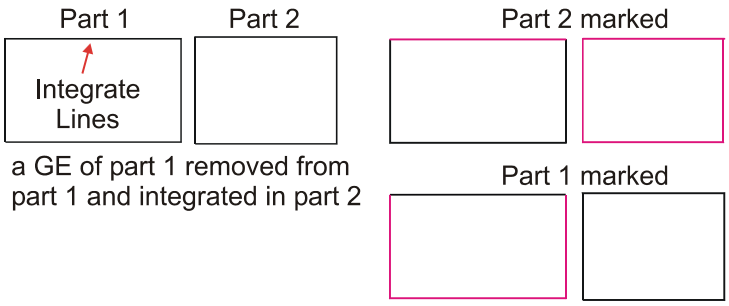


Fig. 46 Line element taken over to active part

- If you use the identification type **In rectangle**, dimensionings, texts, etc. will be taken over all well.
- You can use the **Copy to active part function** to *copy* the lines into the active part, i.e. the selected lines will also remain in the active part. This means that the lines exist twice (in the active part and in the previous part).

28.2 Dimensions to New Part

In the **Process** function group (between Tolerances and Tools) of the **2-D Dimensioning** tab you will find the **Dimensions to new part** function. Use this function to assign the selected dimensioning to the active part. The dimension is deleted from the previous part.

- First select a type of identification for the dimensioning (e.g. Individually).
- Select the part to which you want to move the dimensioning.
- Identify the dimension figure of the dimension.

You can now identify further dimensions, or end the function with the middle mouse button, or right-click to go to the dimensioning context menu.

28.3 To Which Part Do Lines or Dimensionings Belong?

If you place the cursor in the vicinity of a part in the drawing, this part is highlighted pink, including its dimensionings, processing symbols etc. Sub-parts of this part appear in orange colour. This enables you to see which lines, dimensionings, texts etc. are assigned to which part.

If you use the **Show only active part** function, you see only the active part, but **including** all corresponding sub-parts.

When using the **Only active part** function on the **Information** tab, HiCAD asks you: **Highlight sub-parts as well**? If you answer with “No”, you will only see the active part, **without** sub-parts.

28.4 Exercise “Take Over to Active Part” (without graphics)

- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Move lines and dimensions subsequently into the appropriate part.
- Create the main part “Shaft”. Draw into this part (with few lines) a shaft, a corresponding flange, a lid and a base plate. “Forget” the creation of further main parts, respectively sub parts for the flange, the lid and the base plate. After this work step all lines will belong to one main part only.

Fig. 47 Main part “Shaft”

- Add some few dimensions.
- Every workpiece should be represented by one individual part, i.e. its own main or sub-part. Reason: You are then enabled to translate, rotate, delete the part, add master data etc. This way it is easier and you do not need to translate, rotate or delete every single line. Part hatching and overlaps are based on the part structure as well. Create the parts for the flange, the lid and the base plate subsequently. Start from the assumption that the shaft and the flange belong to a (subordinate) assembly “Revolved parts”, the lid and the base plate to a (subordinate) assembly “housing”. These two assemblies belong to a superior main assembly “Drive”. Move the lines and dimensions into these parts.

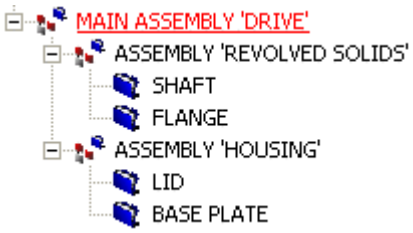


Fig. 48 One part for each workpiece

- The middle window of the ICN contains one part for each workpiece now. These are in turn arranged according to main and sub-assemblies. Check if lines and dimensions are in the right parts.

In HiCAD each workpiece is a part of its own. As soon as we have several parts it makes sense to structure them into assemblies. Each part is located as a sub-part below its superordinate assembly. This assembly can in turn be a sub-part to a superordinate main assembly. Very large drawings can have a part structure consisting of very many levels.

29 Break Up Parts

In the **Others** pull-down menu of the **New** function group of the **2-D Part** tab, you will find the functions: **Break up > 1st sub-part level**, **Break up > All sub-part levels** and **Break up > Part**. All three functions delete parts within the part structure of the drawing. Prior to that, the corresponding lines, dimensionings, texts etc. are moved to a superordinate part. This means that no lines, dimensionings etc. will be lost – the change takes only place in the part structure. If you select **Break up > 1st sub-part level**, one less level will exist in the part structure (namely the one below the active part). The lines are previously automatically copied to the active part.

If you select **Break up > All sub-part levels**, no part will exist below the active part any more. Prior to that, the lines are automatically copied to the active part.

If you select **Break up > Part** , the active part will no longer exist. Prior to that, the lines are copied to the superordinate part. If there were any sub-parts to the deleted part, they are moved level higher.

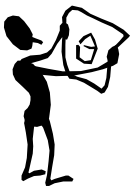
Now let us take a closer look at the **Break up > Part** function:

Use this function to break-up only the active part, and integrate its elements into the superordinate part (if there were any sub-parts to the deleted part, they will be moved one level higher).

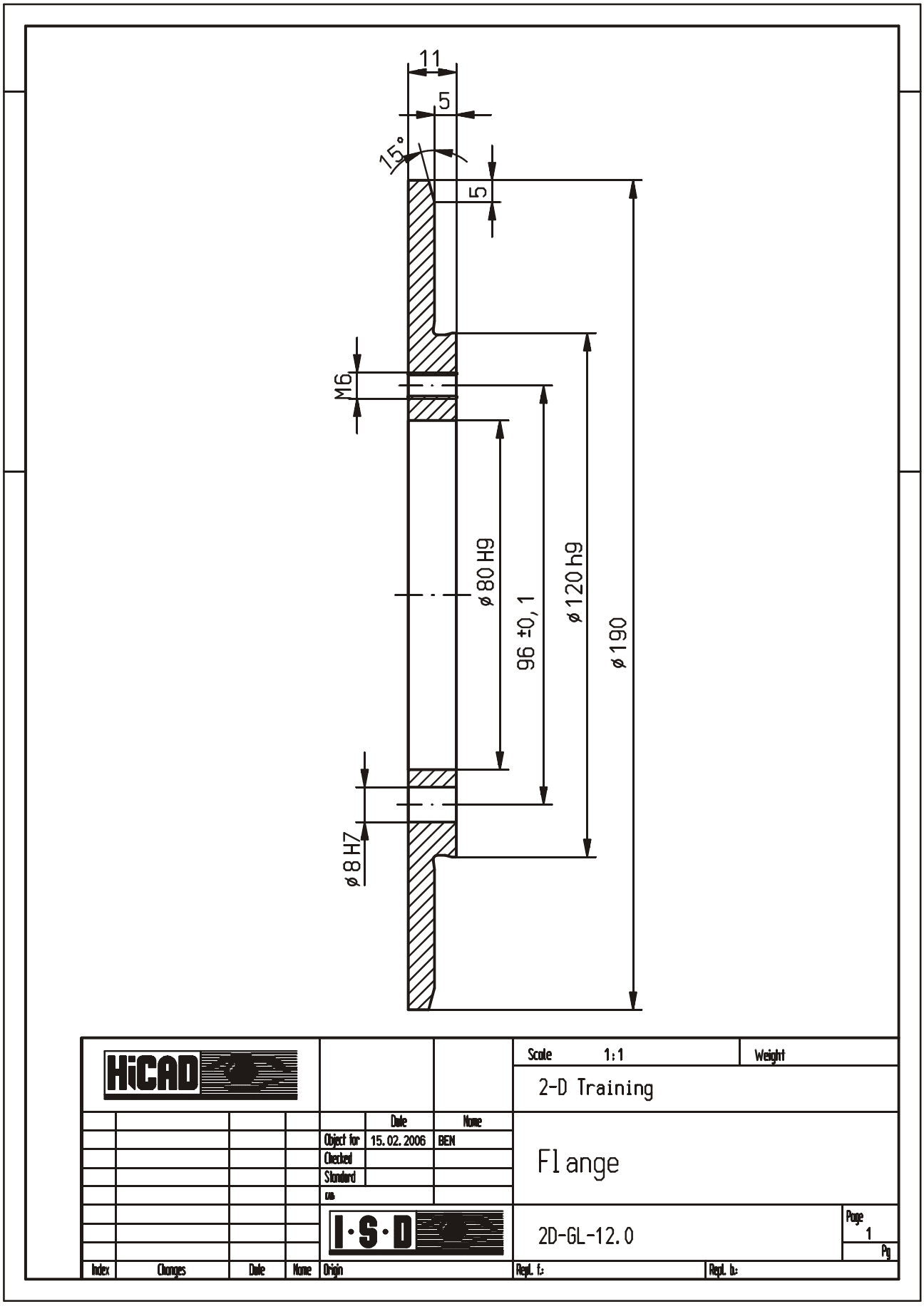
- First activate the part, then select the **Break up > Part** function.

All lines, dimensionings, texts, etc. will be moved into the superordinate part. The (now empty) active part is deleted. You would use this function, e.g. for relief grooves: These are elements of the outer contour, which means that you initially cannot use part hatching to hatch a flange, as the lines of its outer contour are distributed among the main part “flange” and its sub-part “relief groove”. If you use the **Break up > Part** function (please note that the *relief groove* needs to be active), the lines of the relief groove would be moved into the main part. Now the main part has a closed contour and can obtain a part hatching.

29.1 Exercise 12 (2D-GL-12.0) Flange



- Please note: In order to save as much time as possible for the 3-D training, you should only do this exercise if there is a lot of time left. Usually, your trainer will only show you this exercise or skip it completely.
- Learning target: Some standard parts need to be broken up (in this exercise a relief groove and in the next exercise a groove) as they are part of the *outer* contour.
- Draw one half of the flange. Create the other half by mirroring the part.
- Add the bore with the help of the Standard processings and Construction Aids.
- Add the two relief grooves.
- HiCAD created a sub-part for each relief groove. This cannot be seen in the ICN, as relief grooves, recesses, bores etc. do not have part character and therefore are not included in BOM. The lines of the relief grooves, however, are not included in the main part “Flange”. This means that the main part “Flange” does not have a closed contour and can therefore not be hatched with the part hatching. We therefore need to break up the two relief grooves. You can use the function Break up – Part” for this. Please note that in this case the relief groove needs to be active.
- Use the Part hatching function to hatch the flange.
- Alternative solution: Instead of selecting the “Break up – Part” function you could also have activated the main part “Flange” and copied the lines of the two relief grooves into the part with the “Take over GE to active part” function.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30 Beginning of 3-D Training: Solid Primitives

The previous chapters were targeted at acquiring basic 2-D knowledge, which will also be needed for the operation of the 3-D module (e.g. a good understanding of the HiCAD part structure). In the following chapters we will concentrate on the functions of the 3-D module:

Use the functions of the **Solid primitive** menu to create

- cuboids, prisms, pyramids, tetrahedrons and
- cylinders, cones, spheres and toruses.

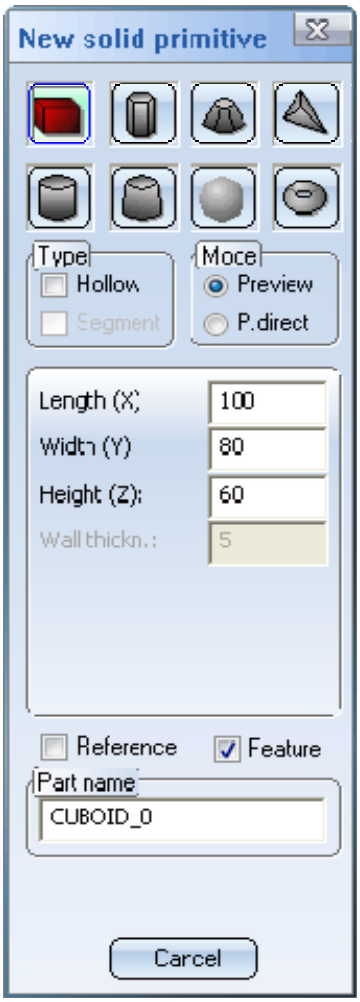


Fig. 49 Solid primitives

In the “Anvil“ exercise you will practise the working with solid primitives by creating and processing one cylinder and several hollow cylinders. Let us therefore take a closer look at the solid primitive “Cylinder”:

30.1 Cylinder

Use this function to create circular cylinders and cylinder segments. The generation of hollow cylinders and cylinder segments is also possible.

To do this, activate/deactivate the checkboxes **Hollow**, resp. **Segment** in the **Type** area.

The following table provides an overview of the value inputs that are required for various settings.

Circular cylinder as	**Type**	**Entries**
Full body	☐ Hollow ☐ Segment	Height Diameter
Hollow body	☑ Hollow ☐ Segment	Height Diameter Wall thickness (internal)
Segment	☐ Hollow ☑ Segment	Height Diameter Start angle End angle
Hollow segment	☑ Hollow ☑ Segment	Height Diameter Wall thickness (internal) End angle

Enter the required data, and place the cylinder in your drawing area. The fitting point is the centre of the surface area.

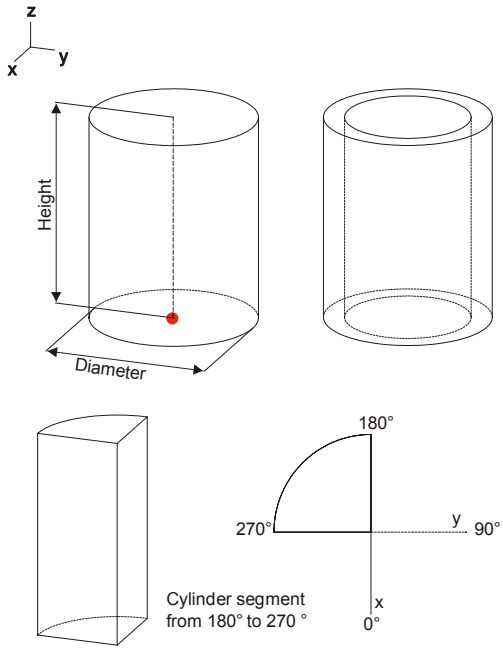


Fig. 50 Circular cylinder

Tip: In the “P. direct” mode, you can also directly specify start point and end point. The height is calculated automatically. This fitting mode makes sense if, for example, you want to fit a cylinder between 2 predefined points.

31 The Information + Communication Navigator

An integral part of **HiCAD** is the Information + Communication Navigator (ICN), which clearly outperforms common structure browsers.

The ICN provides direct access to all opened models, drawings and views, part and assembly structures, attributes and the PDM properties of individual parts. The *ICN* marks referenced parts accordingly and shows you which parts have been modified or are currently being processed.

The view structure is also visualised, including all dependencies between views and 2-D/3-D part structures, as well as between 3-D models and list views. You can switch between views in the ICN, without having to activate any view function.

You can create and process your drawing while the ICN is open. During the process, the representation in the ICN is constantly adjusted to the actual construction in the drawing area. All structures and properties can be modified or processed via mouse-clicks in the ICN. In this way, you can also activate a wide range of part processing functions. Multiple selection options (as known from Windows applications) enable a modification of several parts in one single processing step.

The *Information + Communication Navigator* is displayed on the left hand side of the screen. If required, you can also hide the ICN:

- Close the ICN by clicking the ☒ symbol.

If required, you can hide individual windows of the ICN:

- Right-click on an ICN window caption and select **Hide**.

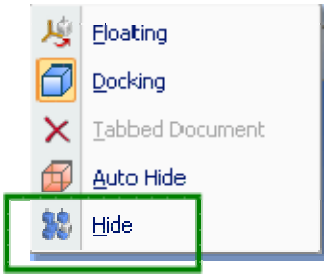


Fig. 51 Hiding of individual windows in the ICN via right-click on window caption

or

- use the **Settings > Docking window** function at the top right of the HiCAD window.

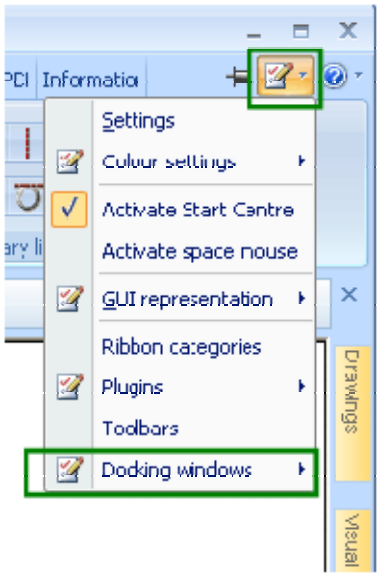
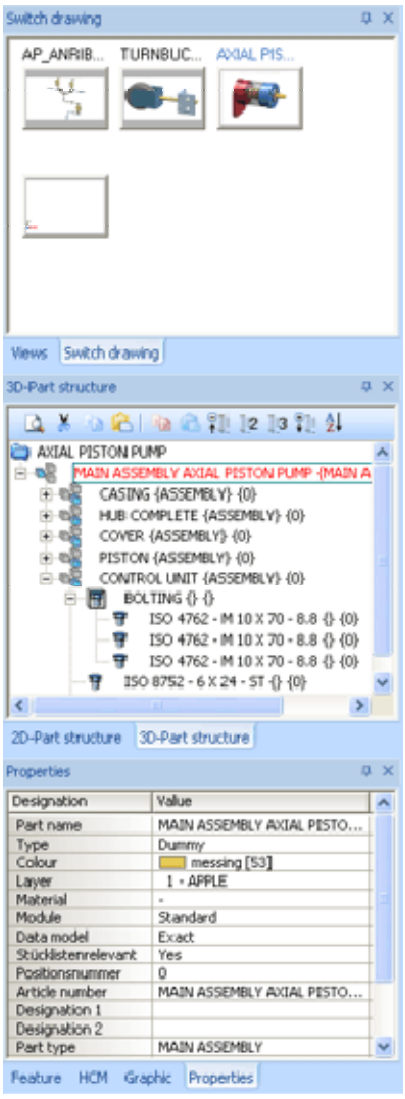


Fig. 52 Hide individual ICN windows via Settings for Docking windows



Upper ICN window

Tabs: Views, Switch drawing

Middle ICN window

Tabs: 2-D Part structure, 3D-part structure

Lower ICN window

Tabs: Feature, HCM, Graphic, Properties

Fig. 53 The *Information + Communication Navigator* (Example)

The *ICN* consists of 3 windows:

- The first window contains information about the view structure and the loaded drawings.
- The second window contains the parts structure (subdivided into 2-D and 3-D).
- The third window contains information on the individual parts, as well as the HCM and the Feature log (3-D).

You can change the size of the individual windows by dragging the frame borders with the mouse (as known from Windows applications).

Please note:

New and processed 3-D parts of the current drawing are identified in the ICN by a symbol. When saving the drawing, this identification is removed again. When exiting HiCAD, or close a drawing, please check whether there are parts marked with the symbol in the drawing. If so, you still need to save the drawing, i.e. the current state of the drawing has not been saved yet.

31.1 The “Switch drawing” Tab

You can load up to 18 drawings into HiCAD and use the **Switch drawing** tab to switch between them. The tab contains thumbnails and file names of all loaded drawings. If you work with HELiOS, the document number is displayed as well.

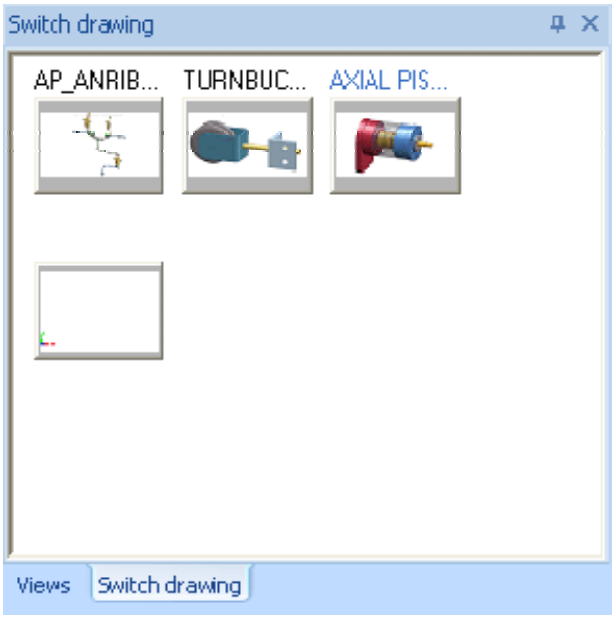


Fig. 54 The *Information + Communication Navigator* (Example)

Open several drawings
If you want to load drawings without closing the current drawing, activate the **Switch drawing** tab and click on the *empty* thumbnail. Load the required drawing, or create a new drawing.

Switch to another drawing
To switch from the current drawing to another drawing, simply click another thumbnail displayed in the window of the tab.

Remove drawing

To remove a drawing from the tab, activate the drawing with a left-click. Then select the **Close** function in the QuickAccess toolbar.

Update preview

Update the preview thumbnails in the **Switch drawing** tab by double-clicking them.

31.2 The “Views” Tab

31.2.1 Views in the “Sheet” and “Model” Area

This tab displays the view structure of the current drawing.

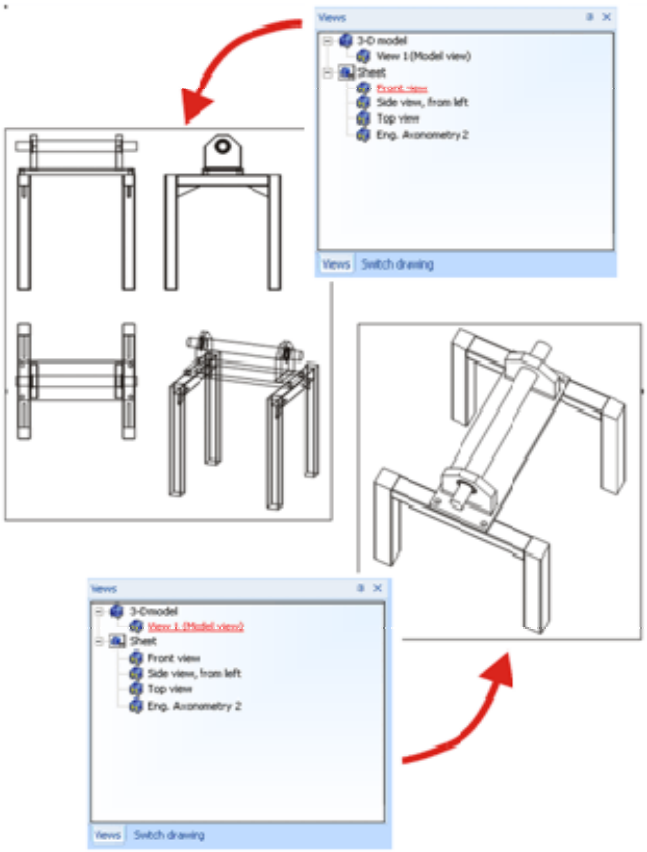


Fig. 55 Arbitrary switching between “Sheet” and “Model” views

HiCAD distinguishes between “Model” views and “Sheet” views. In the sheet views, the production drawings are displayed, which can automatically be derived from the 3-D model. In addition, you can use genuine 3-D views (so-called “model views” in HiCAD, e.g. arbitrary detail and sectional views, cut-outs with user-defined depth and direction, list views with particular parts and assemblies, and much more. This does however not affect the position of the views in the sheet area.

Please note that you will receive genuine 3-D views which you can also use for processings when representing production drawings on the basis of 3-D models. As all views in HiCAD are fully associative, changes in the production drawing are automatically applied to the 3-D model and vice versa. This enables you to freely switch between model views and production views.

Please note however that before printing you need to select the **Sectional/Detail View > Update, All** function once. You will find this function if you right-click a view in the drawing area (pink dotted rectangle) and select **Process > Others**. The sectional views and detail views are then also updated.

You can create *several* sheet areas if required. Right-click **Sheet** in the window and select **Sheet area, New**. You could now create 4 standard views from this view, and add further sectional, detail and/or list views. If you right-click “Sheet” again, you can create further sheet areas, with the method described above.

Example: If you create 4 sheet areas, these may represent one production drawing, one proposal drawing, one approval drawing, and one customer drawing.

31.2.2 Activate View

To activate a view, click on the corresponding view in the **Views** tab of the ICN, or click the pink dotted rectangle in the drawing area.

31.2.3 Properties

If you activate a view in the **Views** tab, HiCAD displays further information about the active view on the **Properties** tab of the lower ICN window, as well as a graphic preview of the view on the **Graphic** tab of the lower ICN window. As soon as you move the cursor into the drawing area again, the properties of the active part are shown on the **Properties** tab again.

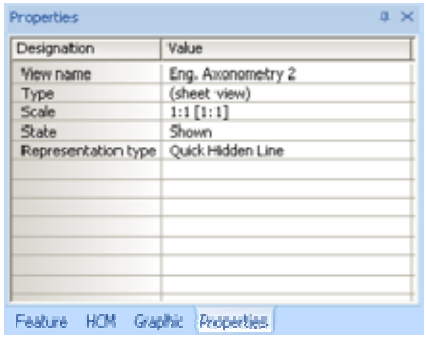


Fig. 56 Properties of a View

31.2.4 Process Views

To process a view, right-click the required view in the ICN, or the pink dotted rectangle in the drawing area. The following images show that the functions selected via the ICN are mainly “management” functions, while the functions selected via the drawing area (pink dotted frame) rather focus on “processings”.

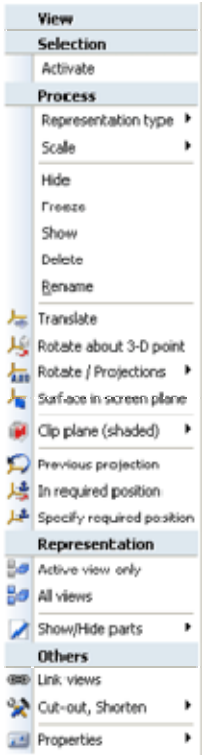


Fig. 57 Context menu for views, selected via ICN

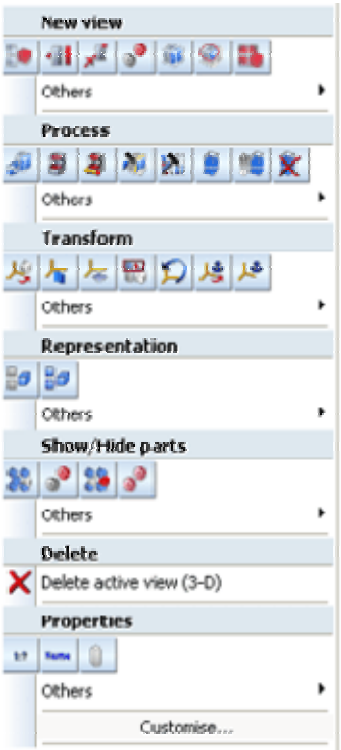


Fig. 58 Context menu for views, selected via drawing area

Please note that in the ICN you can also select several views to be moved, deleted etc. For multiple selection, press and hold down the CTRL key while making your selection.

Move views

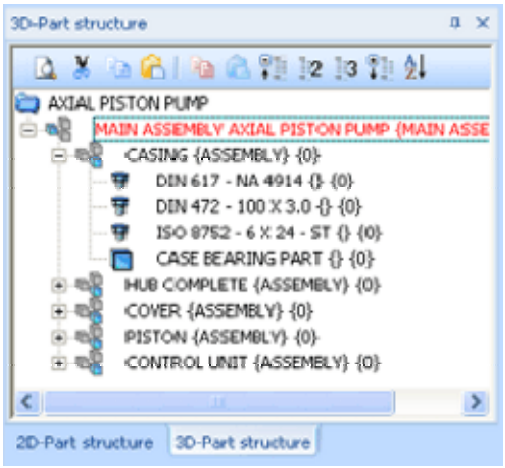
You can move views in the ICN from the model area to the sheet area and vice versa with Drag & Drop (press and hold down mouse button).

In the ICN, click the desired view, hold down the mouse button and drag the view onto the entry **3-D-Model**, resp. **Sheet**.

Multiple selection is also possible (CTRL, resp. SHIFT key, as known from the Windows applications).

31.3 Part Structure

The HiCAD part structure is displayed in the middle window of the ICN (separately for 2-D and 3-D). It provides information about the assemblies, main parts and sub-parts existing in the drawing, the part types etc.



The representation is based on the drawing and has a tree-like structure (as known from many Windows applications). You can expand the tree structure by clicking a “+” symbol, which also displays the sub-parts of the corresponding part, and collapse the structure again by clicking “ - “.

Fig. 59 HiCAD Part structure in the ICN (3D-Part structure tab)

You can use the following keys to browse in the part structure:

↑, ↓	Switch active part, up and down
Pg ↑, **Pg** ↓	To top /bottom
Pos1	Jump to first item
End	Jump to last item

For each part,

- a graphical symbol for the respective part type,
- the part name or article number (if an article number was assigned to the part via RMB > Part attributes), and,
- (in curly brackets) the item number and Designation1 (from the Part attributes)

are displayed. If required, you need to right-click the name of the drawing in the middle ICN window and select the **HELiOS Attributes, Display (update) function** to display Article number and Designation.

31.3.1 Colour Representation

In the ICN, parts are represented in different colour:

- Active part = red ,
- Selected part = dotted frame,
- Parts belonging to a 3-D part list = highlighted in dark blue

All three types of representation may occur individually, together or in arbitrary combinations.

- **Active part**

  The active part in the current drawing is shown in the ICN in red colour. The **Properties** tab displays the attributes of this part, while the **Graphic** tab provides a graphical preview of the part.

- **Selected part**

  The selected part is the part whose properties and graphic are shown in the corresponding tabs. This part is represented in the part structure with a dotted frame. Normally, this is the active part, it can however also be switched by a left-click while holding down the ALT key.

- **The current part list**

  The ICN allows multiple selection of parts. Such multiple selection is also referred to as “part list”. All parts of a multiple selection are highlighted in dark blue, their sub-parts are highlighted in light blue. The active part can (but does not necessarily need to) belong to the part list.

The following paragraphs provide an overview of the selection options for 2-D and 3-D parts and the definition of part lists.

31.3.2 Parts and Part Lists

31.3.2.1 Select Parts

You activate a part with a left-click, and open its context menu with a right-click. If you press and hold down the CTRL key you are enabled to select several parts. If you activate a part at the top of the structure and a part at the bottom while holding down the SHIFT key, all parts between these two parts are also activated.

For 3-D Drawings you can also create part lists by picking the required parts not from ICN, but directly from the drawing.

Further information on part selection filters can be found in the Online Help.

31.3.2.2 Process Part Structure with Drag & Drop

To move parts within the part structure, proceed as follows:

- In the ICN, place the mouse pointer on the part you want to move.
- Press and hold down the left mouse button and drag the part to the part or assembly to which you want to assign it (complete with sub-parts). Then release the mouse button.

If you want to turn the selected part into a main part, the cursor needs to point at the name of the drawing when releasing the mouse button!

31.3.2.3 Auxiliary Parts On/Off

HiCAD also manages parts without part character, e.g. 2-D auxiliary lines, drawing frames according to DIN etc. You have the following options to represent these objects in the structure tree of the ICN:

In the **2D-Part structure** tab of the ICN, right-click the name of the drawing. In the context menu, select **Auxiliary parts On/Off > Auxiliary parts On**. This setting will apply temporarily, i.e. for the current HiCAD session. To switch the display of auxiliary parts off again, select **Auxiliary parts Off**.

To change this setting permanently, open the ALGPAR.DAT file and set the value for the entry

Display of auxiliary parts in the part display/Browser (0/1) - Anzeige von Konstruktions-Hilfsbauteilen in Bauteilanzeige/Browser (0/1)

to 1.

31.3.2.4 Special 2-D Parts

The *2-D parts* Part

When creating a new drawing, HiCAD always inserts a main part with the name *View 1* into the 2-D structure. This main part contains a sub-part with the name *2-D parts*. Each time you select a new 3-D view, the name of the main part in the 2-D structure is adjusted to the selected view – e.g. ENG. AXONOMETRY 2 or TOP VIEW.

By the time you have moved the cursor out of the ICN into the drawing area, the display has been updated.

This part is for structuring purposes only and cannot be deleted!

When you insert 2-D geometry elements into a 3-D view without having defined a new 2-D part beforehand, these will automatically be assigned to the sub-part *2-D part*.

Example

Let us assume that you create a new drawing. The first 3-D view is created automatically. By default, a “View 1” is created below “3-D model”. This means that a part with the name “View 1” and a sub-part with the name “2-D parts” are entered into the 2-D structure. At this point, the 3-D structure is still empty.

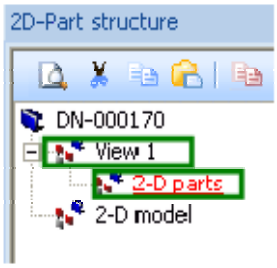


Fig. 60 2-D structure at the start of a drawing creation (middle ICN window, 2D-Part structure tab)

If you now create a new 3-D main part, e.g. with the name CUBOID, a main part with the name CUBOID is inserted in the 3-D structure. The 2-D structure is only changed if you activate a new 3-D drawing. In this case the name of the part “View 1” is replaced with the name of the new view.

The Parts *2-D model* and *2-D sheet*

All 2-D parts which are located below the part *2-D sheet* are visible if a sheet view is activated (e.g. Drawing frame).

In contrast to the 2-D parts located below an *individual view*, these parts are also visible if you select the **Only active view** option.

How can you use the part "*2-D sheet*" in practice?

Let us assume that you want 2-D parts, such as a Bill of Materials, a tolerance table or a newly created 2-D part with processing notes are only visible in the views of the Sheet area, but not in the Model area. In the Views tab, activate a view of the SHEET area. The display will be updated as soon as you move the cursor into the drawing area again. In the "2D-Part structure" tab of the ICN, move the Bill of Materials, the Tolerance table and the created 2-D part "Processing notes" below the part "2D sheet". The result looks as shown in the image below. In the view of the model area, the 2-D parts are now invisible.

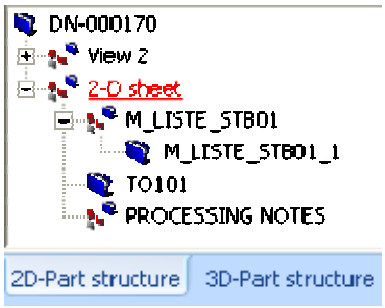


Fig. 61 Bill of Materials, Tolerance table, Processing notes below "2-D sheet"

31.3.2.5 Properties

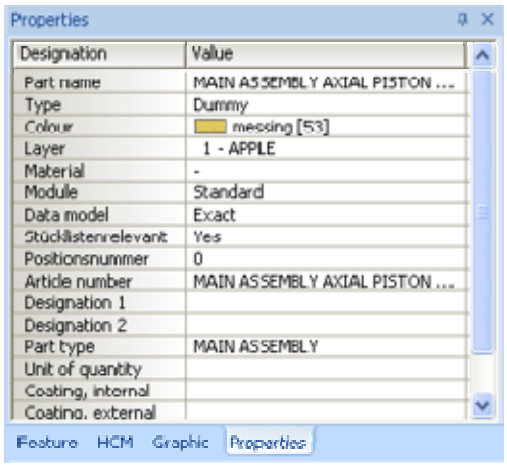


Fig. 62 Properties tab in the lower ICN window

This tab shows the Properties of the active (resp. selected) part.

31.3.2.6 Graphic

This tab shows a graphical preview of the active (resp. selected) part.

31.3.3 The Drawing Name

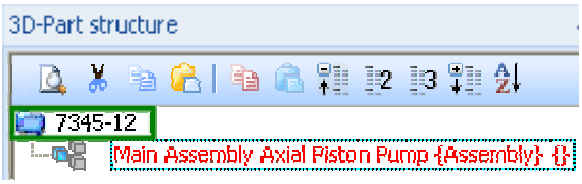


Fig. 63 Name of drawing in the middle ICN window (drawing name = article number when working with database)

The uppermost entry in a part structure (middle ICN window) is always the name of the current drawing. When working with the database it is the article number.

If you right-click on the name of the drawing, the context menus for drawings is displayed. One of many functions is the **HELiOS Attributes, Display (update)** function. Use this function to display the database attributes “Article number” and “Designation” in the ICN structure tree, or (if these are already displayed) update the display of the attributes. If the database attributes are already displayed, the **HELiOS Attributes, Hide** function is active, which enables you to switch the ICN display back to part name/part designation.

31.3.4 The “Find” Function

The **Find function** can be accessed

- via the title bar of the middle ICN window or
- via the keyboard shortcut CTRL+F.

This function provides a convenient search tool within the ICN.

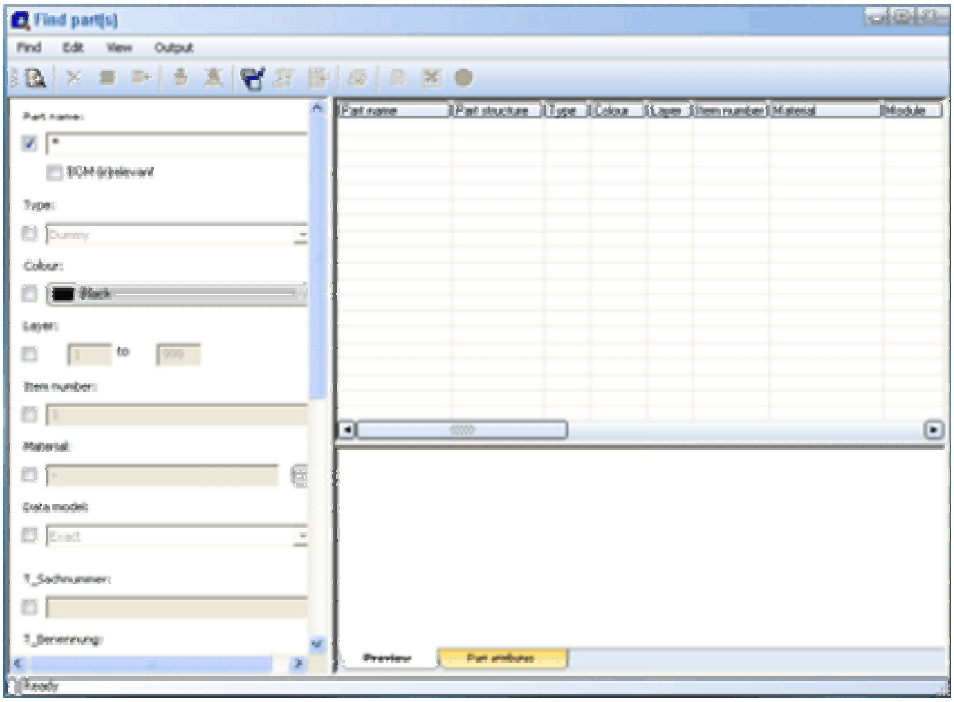


Fig. 64 The ICN search mask

You operate the search mask in the same way as most Windows applications.

31.3.4.1 Search Criteria

You specify the desired search criteria by activating/deactivating the appropriate checkboxes. Furthermore, you can enter the common wildcard characters * and ? into the input fields.

Use the **BOM-relevant** checkbox to restrict the search to parts which are BOM-relevant or not BOM-relevant. You can assign this property to parts by right-clicking them and selecting the **Part attributes** function.

☐ If the checkbox is not active, BOM-relevance is not taken into account for part search, i.e. it makes no difference whether the part is BOM-relevant or not.

☑ If the checkbox is active, HiCAD will only search for BOM-relevant parts.

☑ If the checkbox is active, but with less colour-intensity, non-BOM-relevant parts are also taken into account for part search. You achieve this activation by clicking the activated checkbox once again.

Examples:

- Only the **Part name** field is activated. “2*” has been entered: HiCAD will only search for parts whose number begins with a 2.
- Only the **Item number** field is activated. „-“ (i.e. only a dash) has been entered: HiCAD will search for all parts with *no item number*.
- Only the **Material** is activated. „-“ (i.e. only a dash) has been entered: HiCAD will only search for parts to which *no material* has been assigned yet.

You start the search by selecting the **Start/Cancel search** function in the toolbar.

31.3.4.2 Process Found Parts

The parts shown in the result list can be processed directly via the result list.

Right-click the item of the result list that you want to process. Multiple selection (hold down SHIFT or CTRL key) is also possible.

All selected parts are highlighted in dark blue and can now be processed.

- via the context menu (right-click),
- with the functions of the **Edit** menu, or
- with the functions of the toolbar.

This enables you to assign a particular material to all parts or mark them as BOM-relevant.

31.3.5 The HiCAD Clipboard

You can (besides other possibilities) use the HiCAD Clipboard to copy parts from one drawing to another drawing.

Select the part or specify the part list that you want to copy from the current drawing to another drawing. Then right-click the part or part list and select the **Copy to clipboard** function.

You can then paste the current contents of the HiCAD Clipboard to another drawing:

In the ICN of the target drawing, right-click the part to which you want to assign the contents of clipboard as sub-part. If you want to paste the contents of the clipboard as main part into the drawing, right-click the drawing name in the middle ICN window. Then select the **Paste from clipboard** function.

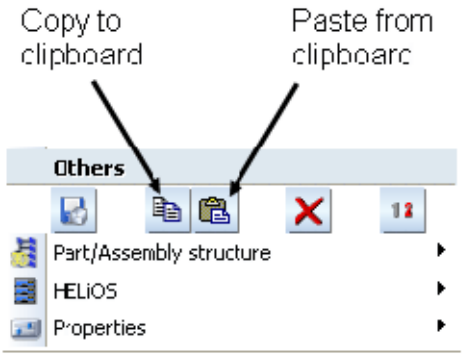


Fig. 65 Copy to / Paste from Clipboard functions (lower part of 3-D part context menu in ICN)

The copied parts are shown, and HiCAD prompts you to specify a fitting point in the drawing. This is the point via which you position the copy in the drawing.

After specification of the fitting point, HICAD prompts you to specify the position of the fitting point in the drawing.

The function group **Insert Part** on the **Drawing** tab provides further possibilities to copy parts into other drawings:

Here you can load re-usable parts, e.g.

- from the database, or the Explorer,
- from a catalogue,
- from the part library (optionally available),
- from another drawing, or
- via an interface

Fig. 66 Function group Insert Part on the Drawing tab: Further part insertion functions

31.4 The “HCM” Tab in the ICN

The “HCM“ tab of the lower ICN window contains all HCM constraints that have been assigned to your drawing.

The integrated “HiCAD Constraint Manager” enables you to effectively parameterise your drawing. Available are edge constraints as well as part constraints. Geometric relations can be assigned in 2-D and 3-D - either while creating your drawing, or to already existing assemblies. This enables a dynamic manipulation even of complex models, i.e. spatial movements are controllable and can be visualised in real-time. Three-dimensional kinematic analyses, too, can be realised in this way.

The constraints you have assigned to an active part or an assembly via part constraints can be found in the **HCM** tab of the lower ICN window.

More information on the HCM is provided in the Online Help.

The ISD offers a 2-day training for the Parametrics module. This training also covers the topics Feature Variants, Design Variants, Motion Simulations and Assembling Simulation.

32 Representations of Views

HiCAD offers a wide range of representations for views, running the gamut from glass model and Hidden Line representation to photo-realistically shaded and rendered models.

The representation functions for views refer – apart from a few exceptions – always to the active view. These functions are available in context menus which can either be activated by a right-click on the view frame in the drawing area (pink dotted frame) or on the name of a drawing in the ICN. In addition, you can find many view functions in the **Views** tab.

See Online Help

The following paragraphs deal with some exemplary types of representation. Further information on view functions can be found in the Online Help.

32.1 Glass Model

If a part is represented as glass model, all edges, including the hidden ones, are visible. This representation is called “glass model” as this setting provides a realistic representation for transparent materials only.

An advantage of this representation is that the image build-up speed is quite high, as no visualisation calculations have to be carried out.

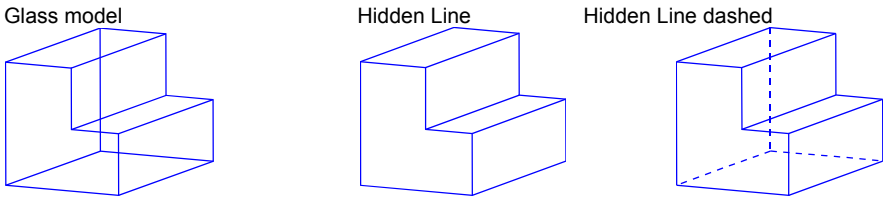


Fig. 67 Edge model functions

If you want to represent all views of your drawing as glass models, right-click the view frame (pink dotted frame) in your drawing and select **Representation, Others > All views, Glass Model**.

32.2 Hidden Line/Hidden Line Dashed

The **Hidden Line** function “hides hidden lines”. The **Hidden Line dashed** function, in contrast, shows hidden lines by representing them as dashed lines. The hidden edges of a body on layer 40 (glass) are not considered for these functions.

If you want to represent all views of your drawing as Hidden Line representations, right-click the view frame (pink dotted frame) in your drawing and select **Representation, Others > All views, Hidden Line**.

The Hidden Line calculation does not take self-intersections into account. In such cases, unexpected overlaps may occur. Use the **Collision check** function in the **Information** tab to check a drawing for collisions.

32.3 Shaded With Edges

This function enables a faster calculation, with shading and representation of visible edges. The edges are represented according to the current edge model parameters – with the exception of free edges.

We recommend that you also read the information on Drawing properties provided in the Online Help. You can access the relevant functions via **Drawing > Properties > Others > Surface approximation** and **3-D Standard > Properties > Others > Limiting angle for shading**.

These functions do also affect shaded representations. For example, a higher value for surface approximation results in many cases in a more accurate and consistent shading.

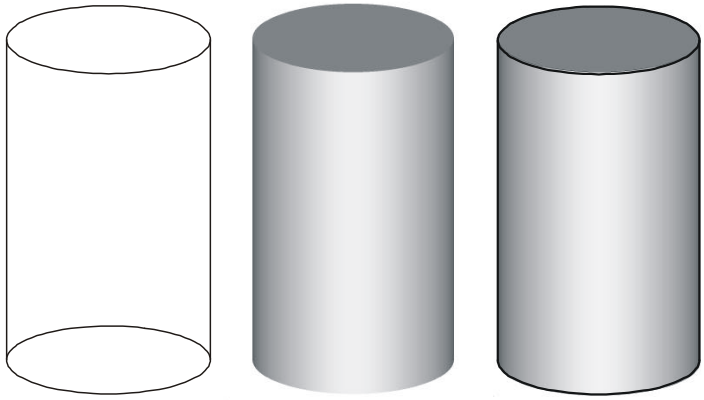


Fig. 68 Glass model, Shaded without edges, Shaded with edges

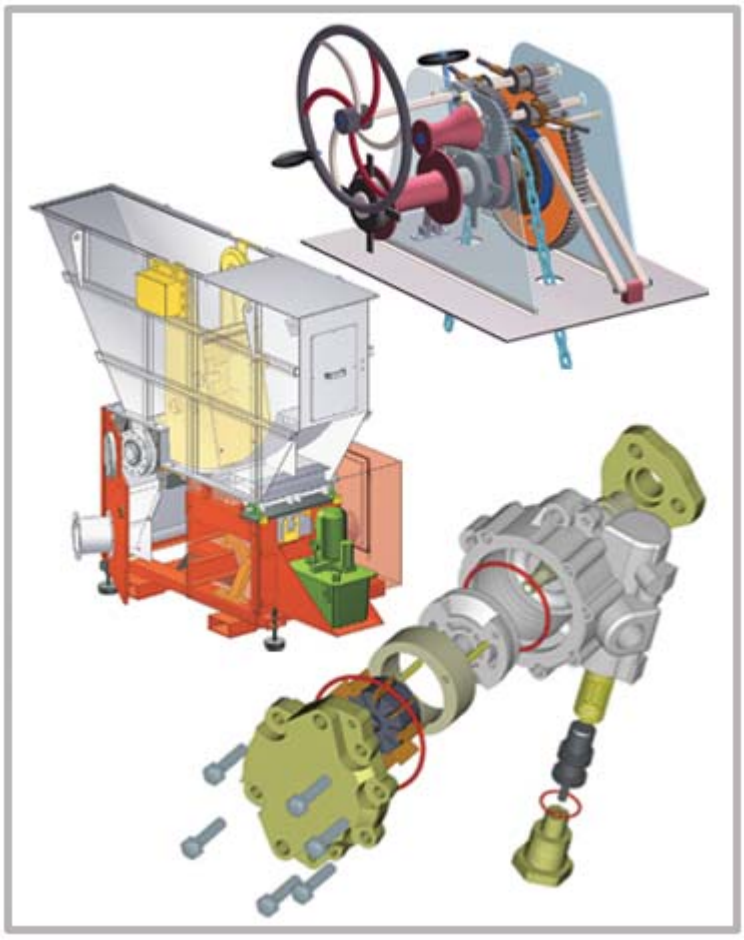
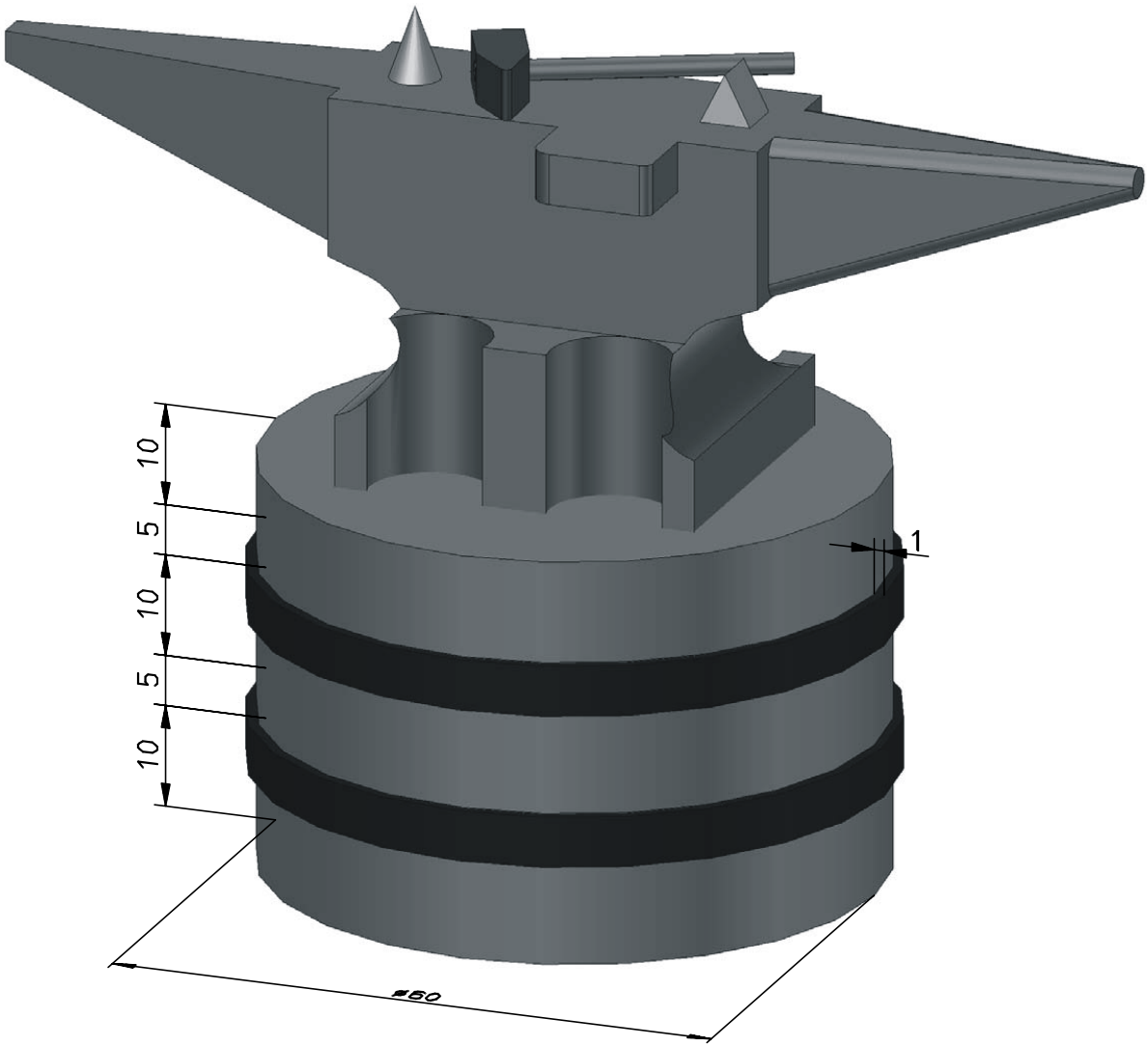


Fig. 69 Examples of shaded parts

Shaded representations as well as all types of representation beginning with “Quick…” (Quick Hidden Line etc.) are based on approximation algorithms. This enables a very fast calculation of the shading or hidden lines, even for very large and complex assemblies. In very rare cases it may occur that particular elements (e.g. the shortening of a view or similar) are not represented correctly. In such cases it is recommended to use the exact procedure, “Hidden Line”, instead of “Quick Hidden Line”.

32.4 Exercise 1 (3D-GL-01.0) Anvil

- Learning target: Solid primitives; select functions via the context menu; ICN; different types of representation.
- Load the “Main assembly Anvil + Tool” as a main part via the Explorer from the hicad / Szenen / Examples directory. Hide the sketches in the ICN (Information- and Communications-Navigator) if necessary.
- Draw the mandrel as a cone with a diameter of 5 and a height of 7. Reference the part and add it to the feature log. Save the part, with document master data, to the HELiOS database.
- Draw the block of wood and the hoops as (hollow) cylinders. Reference the part and add it to the feature log. Save the part, with an arbitrary document master, to the HELiOS database.
- You can find the “Translate part, via 2 points” function in the “Transform” function group of the “3-D Standard” tab. If you want to translate a part, you need to activate it in the drawing or in the ICN first. The most convenient procedure, however, is the following one: If you want to translate a part, right-click the part in the drawing or the part name in the ICN, and select the “Translate part, via 2 points” from the context menu. The part on which the cursor is placed is automatically translated, as you automatically activate a part by right-clicking it.
- Switch between different types of representation for the drawing object: Glass model, Shaded with edges, Shaded without edges etc.
- Familiarise yourself with the ICN:
  - Switch between the “Views” and “Switch Drawing” tabs.
  - Look especially at the sub-parts of the anvil in the middle window of the ICN.
  - Switch to the “Properties” tab in the bottom window of the ICN and assign different colours to the mandrel, block of wood and hoops. If you find the names are not meaningful, use this tab to change them.
  - Activate the part “Anvil” in particular and look at the feature of this part in the bottom window of the ICN. Activate some sketches in the feature with the “Process Sketch” functions.
- Hint: Surface colour and name can also be changed in the middle window of the ICN (by right-clicking the corresponding part).
- Space for notes:



HiCAD					Scale 1:2	Weight	
					3-D Training		
				Date	Name		
				Object for	15.02.2006	BEN	Anvil
				Checked			
				Standard			
				CAD			
				I·S·D	3D-GL-01.0		Page 1 Pg
Index	Changes	Date	Name	Origin	Repl. f.:	Repl. b.:	

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33 Transform and Clone Parts

Main parts and sub-parts can be transformed, i.e. translated, rotated, mirrored or scaled. You clone a part by additionally copying it while transforming it. The following paragraphs deal with transformations and clonings.

33.1 Transform parts

On the **3-D Standard** you will find the **Transform** function group.

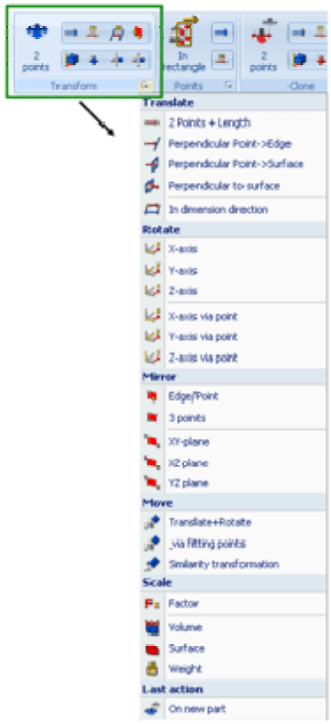


Fig. 70 The **Transform** pull-down menu

Select the **Translate part, via 2 points** function. Use this function to translate the active part via specification of 2 points which determine the translation vector.

First specify a point on the part you want to transform, then specify its new position in the drawing. If you choose the point option “Relative”, you are enabled to specify the exact length of the translation.

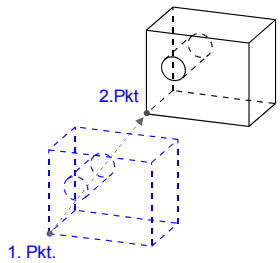


Fig. 71 Translation

When rotating a part, we specify a rotation axis instead of a translation vector. We can select the rotation axis as a line or via 2 points. Mirroring is described in the following paragraph.

33.2 Clone Parts

Parts which are needed several times in the drawing can be copied. Such copies are called “clones” in HiCAD. The corresponding functions can be found in the **Clone** function group of the **3-D Standard** tab.

The following types of transformations are available:

- Translate,
- Rotate,
- Mirror,
- Move (incl. Similarity transformation).

Let us now take a closer look at the **Mirror** function.

Select **Clone+Mirror part, via 2 edges**.

After selecting this type of transformation, the following input window is displayed:

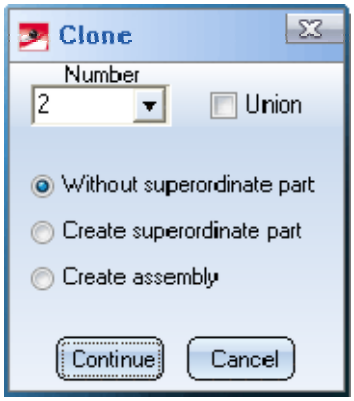


Fig. 72 The **Clone** input window

- **Number**

  Here you can enter the number of part you want to create. **Please note that the original part is already included.** If, for example, you want to have 3 parts, you need to enter a 3 (and not a 2). Please also note: When mirroring parts, the number is always 2.

- **Union**

  If you want to union the original and its clonings by a Boolean operation, so that it forms one single part, activate the Union checkbox (this only makes sense if you start with one half of a (revolved) part that you want to make a whole part. Otherwise always deactivate the Union checkbox).

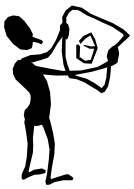
- **Create superordinate part** (only available if the Union checkbox is deactivated)

  If this checkbox is activated, all parts created by mirroring, translation, rotation etc. are combined as sub-parts under a dummy part with the name “Group”. We normally do not activate this option. Reason: The most convenient way to combine parts is the creation of *Assemblies*.

- **Create assembly** (only available if the Union checkbox is deactivated)

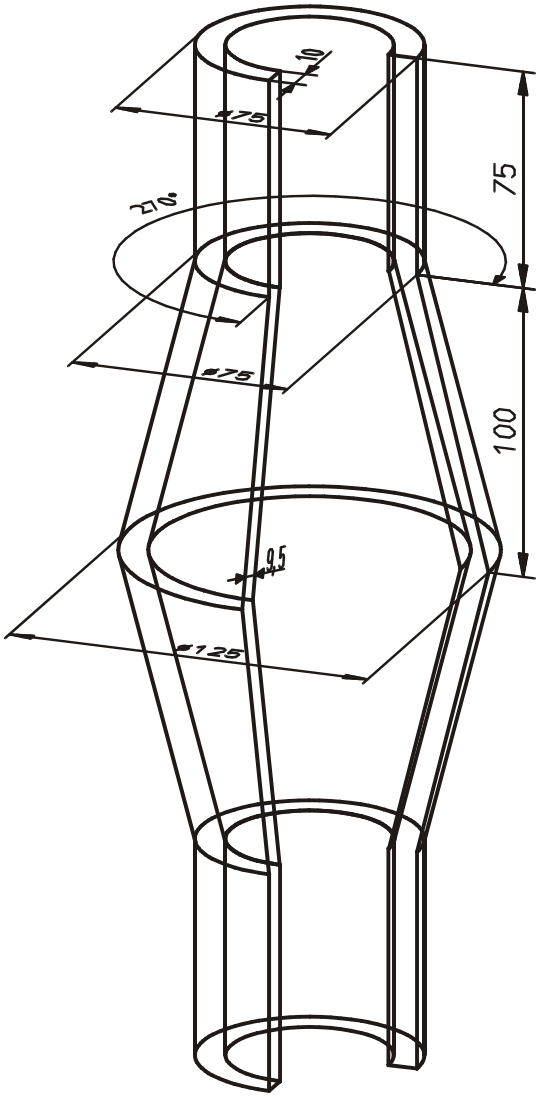
  Use this option to combine cloned parts as subordinate parts to a common assembly.

After entering the required data, exit the input window with **OK**. Then select two lines to define the mirroring plane.



33.3 Exercise 2 (3D-GL-02.0) Guide Carriage

- Learning target: Solid primitives as hollow parts and part segments; add parts; various views; select functions for the design (e.g. change scale) via context menu.
- Draw both upper parts of the guide carriage as hollow segment cylinders, respectively segment cones.
- Merge both parts using the “Add part” function. Check the result in the ICN.
- Mirror the new part in downward direction.
- Display the part in front, side and top view as well as the standard axonometry.
- Add a drawing frame (Sheet area).
- Change the scale of the drawing if required. To do this, right-click the drawing and select the appropriate function.
- Hint: Please note that Boolean operations (Add, Subtract, Intersections) have two disadvantages: They increase the size of the drawing file and the calculation time in the Feature log, especially if the added/subtracted/intersecting part contains sub-parts, such as sketches etc. Besides, the added/subtracted/intersecting part will at first be invisible in the Feature log. (even if it can be retrieved and processed in the Feature log with the “Load processing part” function). Instead, you could also have used the “Revolved solid” function to draw the Guide Carriage “as one part”. When adding very complex parts or subtracting very complex cavities, however, the attaching or excluding of simple geometries may not be sufficient, making additional Boolean operations necessary.
- Space for notes:



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							3-D Training		
					Date	Name			
				Object for	15.02.2006	BEN	Guide carriage		
				Checked					
				Standard					
				as					
				I·S·D			3D-GL-02.0	Page 1 Pg	
Index	Changes	Date	Name	Origin			Repl. f.:	Repl. b.:	

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34 Load Drawings With or Without Database

Procedure with Database (additional module)

Select the **Switch drawing** tab in the upper ICN window. Double-click the empty drawing thumbnail. In the **Drawing** tab, select the **Load drawing with DB** function.

Deactivate the two filters suggested by HiCAD for Links. As we have not linked our drawing to anything, this filter would not make any sense for us. Instead, define your own filters. This can, for instance, be *3 in the Document number field, or A* in the Designation field. If you set several filters, all search conditions need to be fulfilled ("AND" link). Therefore, you normally set only one filter.

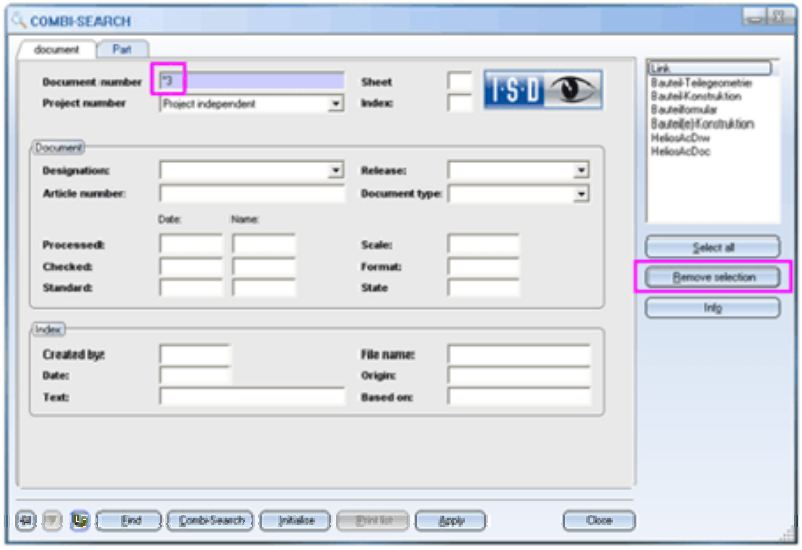


Fig. 73 Remove "Link" predefinions; set filter "*3" in the "Document number" field.

Click the **Find** icon.

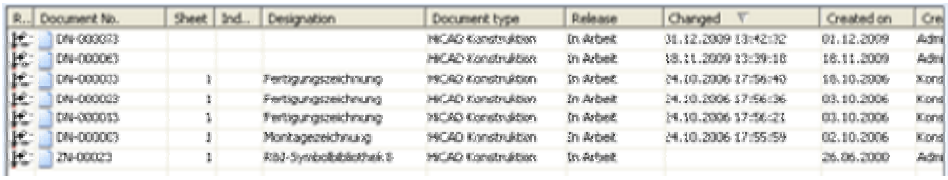


Fig. 74 Result list according to revision date ("Changed" column)

A result list is displayed which can be sorted if required. To do this, click on a column header (Document No., Designation etc.). In the result list shown below, the "Changed" header was clicked twice. Therefore, the found drawings were sorted in descending order (i.e. the newest date is at the top of the list).

Click an item in the list to get a **Preview** of the drawing. Use the arrow key to browse through the items.

Load one of the drawings with a double-click. You can also select a drawing with a single click and then choosing **Apply**.

Procedure WITHOUT database:

If you work at your company without database, you probably will have switched off the database on your training computer. After selecting the **Load drawing** function, the Windows mask with the file names of all drawings is displayed. Further selection criteria are not available when working without database. You can however select the **View menu** icon. In the following menu, select **Details**. You can now, for example, sort lists of items according to their file size, by clicking the header of the **Size** column.

35 Boolean Operations

In the function group **+/-** of the **3-D Standard** tab you will find the Boolean operations **Add part**, **Subtract part** and **Intersection**. The following example describes the subtraction of a part from the active part.

35.1 Subtract part

Use this function to subtract an arbitrary part from the active part.

Select the **Subtract part** function. Identify the part you want to subtract. If you want to retain this part after the Boolean operation, answer “Yes” to the question “Retain second part”, otherwise answer “No”.

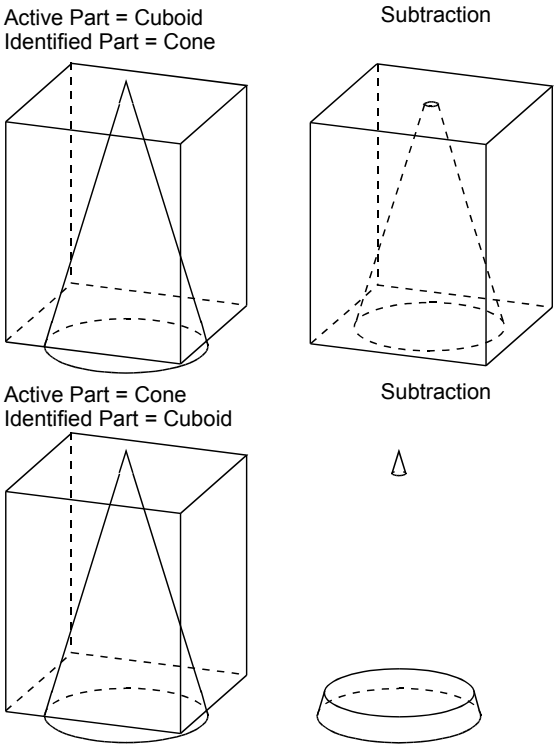
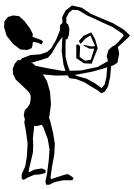


Fig. 75 Subtract parts: (Top: Cuboid minus Cone; Bottom: Cone minus Cuboid)

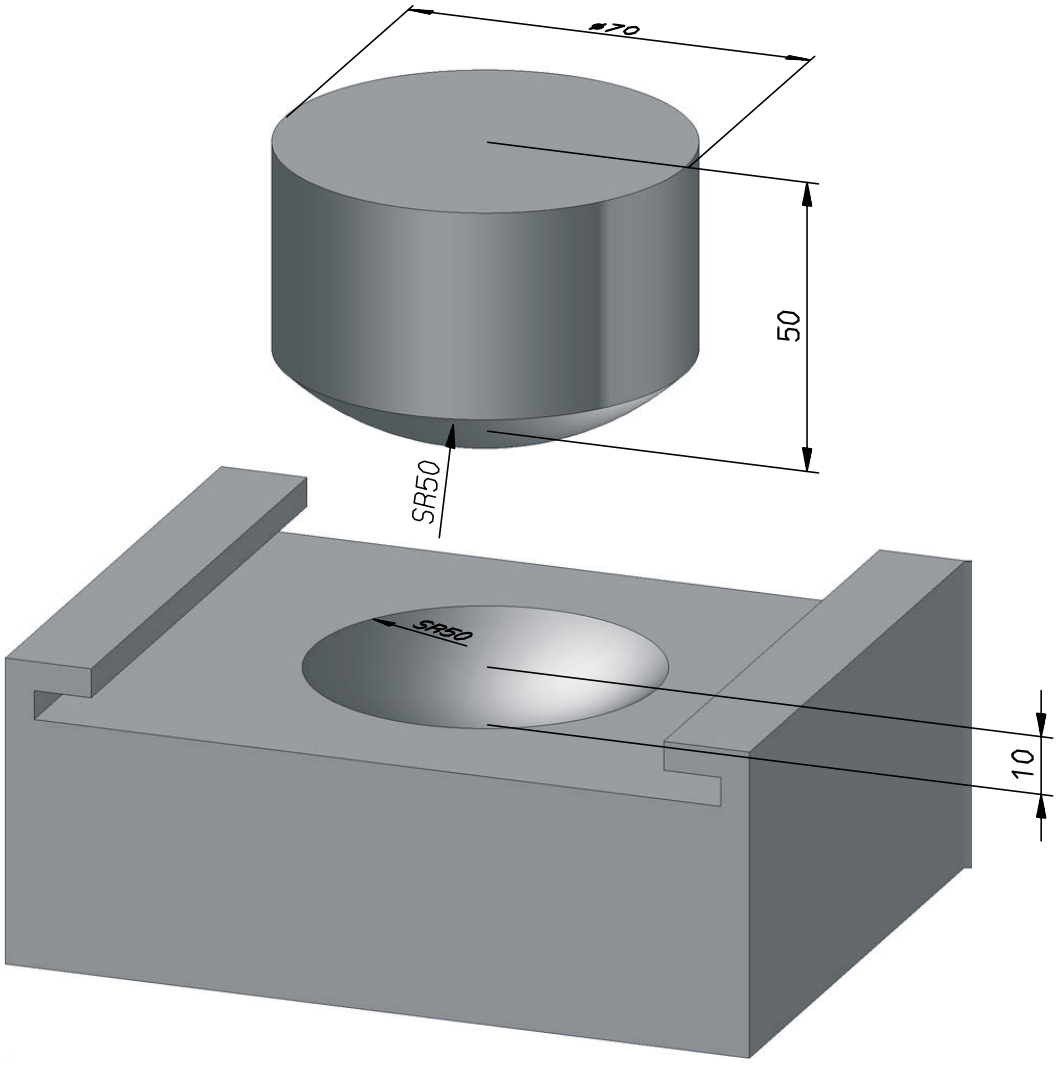
Please note that Boolean operations (Add, Subtract, Intersection) have two disadvantages: They increase the size of the drawing file and the calculation time (especially if the added/subtracted/intersected part has sub-parts, e.g. a sketch). Furthermore, the added/subtracted/intersected part is initially invisible in the Feature log (even if it can be retrieved and processed in the Feature log with the “Load processing part” function).

In a later part of the training we will learn about functions which do not have such disadvantages. These are functions such as “Subtract part, with translation/with rotation” or “Add sweep, with translation/rotation”. When adding very complex parts or subtracting very complex cavities, however, the attaching or excluding of simple geometries may not be sufficient, making additional Boolean operations necessary.



35.2 Exercise 3 (3D-GL-03.0) Female Die Part

- Learning target: Subtract; intersection; Solid primitives; Select the functions via the right mouse button if possible; load from the database (additional module!).
- Load the female die part as a main part via the Explorer from the hicad / Szenen / Examples directory.
- Create a sphere with a diameter of 100 in the centre of the female die part so that it dips into the female die part by 10 mm.
- Create a spherical indentation by subtracting the sphere of the female die part.
- Create the male die part as a cylinder with a diameter of 70 and a height of 70 (the final height of 50 is created in a later step).
- The ball-shaped rounded bottom of the male die part with a diameter of 100 and the total height of 50 are created as follows: Create a sphere with a diameter of 100 in the centre of the top surface of the cylinder. Create the intersection of sphere and cylinder.
- Hint: Please note that Boolean operations (Add, Subtract, Intersections) have two disadvantages: They increase the size of the drawing file and the calculation time in the Feature log, especially if the added/subtracted/intersecting part contains sub-parts, such as sketches etc. Besides, the added/subtracted/intersecting part will at first be invisible in the Feature log. (even if it can be retrieved and processed in the Feature log with the “Load processing part” function). Instead, you could also have used the “Subtract part, via rotation“ function to create the subtraction for the female die part. For the creation of the male die part you could have used the “Add sweep, via rotation” function. When adding very complex parts or subtracting very complex cavities, however, the attaching or excluding of simple geometries may not be sufficient, making Boolean operations necessary.
- Save the drawing.
- Load the drawing from the database (additional module!).
- Familiarise yourself with the saving via interface and with the Print menu.
- Space for notes:



HiCAD						Scale 1:1	Weight
						3-D Training	
					Date	Name	
				Object for	15.02.2006	BEN	Female Die Part
				Checked			simplified display
				Standard			
				CAD			
				I·S·D			3D-GL-03.0
Index	Changes	Date	Name	Origin		Repl. f.:	Repl. b.:

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36 Working with Sketches

36.1 Sketches

A sketch is a 3-D part with free edges which is located in a plane. You can, for example, use a sketch for derivation of extruded solids, bores, adding or of subtracting of material, as well as for c-edge sweeps or sectional views. The function can be found on the **Sketch** tab. A sketch has the part type "Part with free edges".

Please do not confuse the **Sketch** function with the **3-D Sketch** function! The latter is something completely different, namely a 3-D dummy part (e.g. for guidelines).

When you create a new sketch, HiCAD automatically assigns the part name **Sketch**, which can however be changed with the **Rename** function. As for 3-D parts, please note the following: If an article number exists in the *Part attributes*, it needs to be renamed or deleted, as in the ICN display, the article number has priority over the designation. You can access the part attributes by right-clicking the sketch (or a part).

You draw the sketch in the current sketch plane, or in the current 2-D view (front/side/top view), respectively. Use the **Plane** functions in the **New** function group of the **Sketch** tab to define or activate a different sketch plane. This is however only possible if the Sketch does not contain any lines.

Please note:

- When you create a new sketch, HiCAD automatically starts the HiCAD 3-D HCM, which is a tool enabling you to position and move the elements of a composite edge or a sketch. To this end, the elements of the c-edge/the sketch are linked with so-called constraints, i.e. dimensional and logical relations and/or restrictions. The c-edge parametrics will transform the corresponding elements in such a way that the specified constraints are respected.
- In case of 3-D objects derived from sketches, HCM constraints assigned to the sketch are taken over as parametric dimensions. This applies, for example, to extruded or revolved solids, as well as to bores, material subtractions etc. You can change the part via these parametric dimensions.

See Online Help

A detailed description of c-edge parametrics is provided in the Online Help, or in our 2-day Parametrics training.

36.1.1 New Plane

You draw your sketch in the current sketch plane. If you are working in a 3-D view, HiCAD places this sketch plane into the current processing plane, or, if there is no processing plane, into the XY-plane of the active coordinate system. If you are working in a 2-D view, the sketch plane is automatically located in this view.

See Online Help

As long as the sketch does not contain any lines, you can define a different sketch plane with the **New plane…** function.

More information about processing planes can be found in the Online Help.

36.1.2 Sketch Polyline

A polyline is a structurally coherent succession of individual lines. The start point of a following line is the end point of the previous line.

Sketch Polyline

Sketch technology enables a fast creation of polylines. It facilitates the drawing process by constantly displaying auxiliary lines along a predefined grid – starting from each last point of the drawn polyline – while also showing the corresponding angles, distances and radii at the cursor.

This grid enables you, for example, to specify the direction and the length of a line with a mouse click.

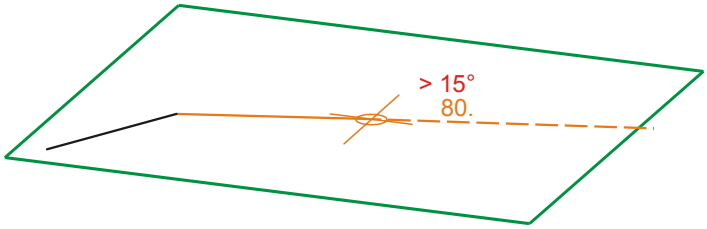


Fig. 76 Angle and Distance shown at the cursor

You can change the grid fineness, as for all sketch functions, with the following keys on your keyboard:

- Minus key: Refine grid (e.g. steps of “5” instead of steps of “10”)
- Plus key. Coarsen grid
- Space bar: The desired length or angle can directly be entered as a value via the keyboard.

You can *permanently* preset the grid fineness in the **SKIZZTEC.DAT** file. When zooming the screen, the grid fineness also changes.

If you have already drawn at least one line with the **Sketch polyline** function, and then click the right mouse button, the following context menu is displayed:

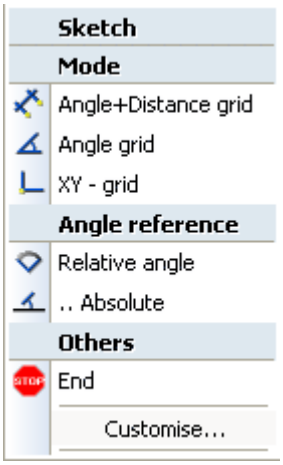


Fig. 77 Right-click after drawing a line with **Sketch polyline** function

See Online Help

The following paragraph provides an explanation of the Arc function. All other functions are described in detail in the **Online** Help.

Arc

Use this function to insert arc into the polyline. After selecting the function, an auxiliary circle is displayed tangential to the last line. Its radius is displayed at the cursor.

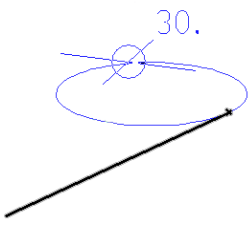


Fig. 78 Auxiliary circle; Radius shown at cursor

Accept the suggested radius with RET, or enter the value, or press the space bar and enter the value via the keyboard. Important: If you click in the 1st or 2nd quadrant of the arc, the arc is connected tangentially in the direction of the last created line element. If you click in the 3rd or 4th quadrant or the arc, the arc is connected tangentially in opposite direction of the last created line element. **Therefore, please make sure that the cursor is already positioned at the correct side of the circle when you select he radius!**

Next, the aperture angle for the arc is displayed at the cursor:

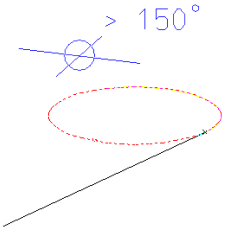


Fig. 79 Aperture angle shown at cursor

Accept the angle with RET, or press the space bar and enter the value via the keyboard.

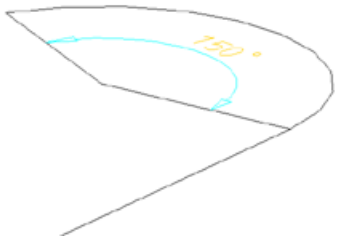


Fig. 80 Arc created with the **Arc** function

New polyline

In the **Line** function group of the **Sketch** tab you will find the **New polyline** function. This function is particularly useful if you prefer working with point options instead of using a grid.

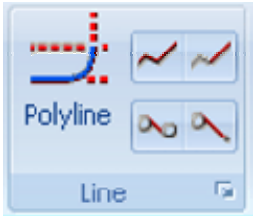


Fig. 81 The **New polyline** function

Select the point options, using one of the following options that you already know:

- RETURN key
- Point options R, A, D, P, W, N via pressing the corresponding key on the keyboard
- Right-click when HiCAD suggests any point option
- Quick double left-click

36.2 Extruded Solid

Besides creating revolved solids, adding or subtracting material, you can also create extruded solids from closed polylines in a sketch.

Use the **Extruded solid** function in the **New** function group of the **3-D Standard** tab to create general cylinders. These are derived from sketches, by translating a copy of the sketch along a normal.

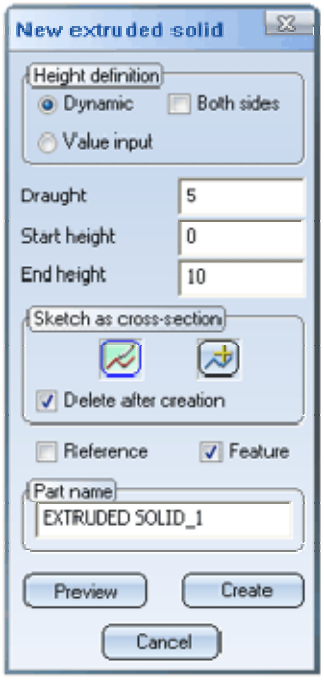


Fig. 82 Extruded solid from sketch

Proceed as follows:

Proceed as follows to create an extruded solid:

- Enter the part name.
- Activate the **Reference** checkbox if you want to reference the extruded solid.
- Activate the **Feature** checkbox to record the creation of the extruded solid creation in the feature log.
- Select the type of **Height definition** (e.g. **Value input**, if you want to enter the height as a value)
- Enter a value for End height and, if required, for the Draught and the Start height.
- Select the type of cross-section definition. In the “Sketch for cross-section” area you can either create a new sketch or select an existing sketch.
- If you first want to see a preview of the created part, click the **Preview** button. If you are satisfied with the result, click the **Create** button.

Please note:
If a sketch is active when you call the **Extruded solid** function, this sketch is used by default until you select or create a different sketch.

36.2.1 Height Definition

You can either select the height dynamically with the cursor, or enter a height via value input. Default setting is always the previous setting.

- **Dynamic**

If you activate this option, you can select the height dynamically with the cursor. Move the cursor to the required position and left-click. HiCAD then shows the height for correction. The input fields **Start height** and **End height** are locked if you select this option.

In the "Dynamic" mode, you first need to select the "Preview" or "Select" function, then left-click in the drawing area. Please also note the user guidance text in the info bar.

- **Value input**

If you select this option, you need to enter appropriate values for Start height and End height.

These values specify the distance between bottom surface and top surface of the extruded solid, starting from the sketch plane, thus defining the height of the extruded solid. Negative values cause the corresponding surface to be set in negative Z-direction.

- **Both sides**

If you activate this checkbox, you can (for both the Dynamic and the Value input option) determine that one half of the height will be set in positive Z-direction, and the other half in negative Z-direction. Alternatively, you can achieve the same effect by entering -30 for **Start height** and 30 for **End height**.

If you right-click one of the value input fields (e.g. Start height), a context menu is displayed. Here you can pick distances and angles or copy and paste variables or values from the clipboard into the input fields.

36.2.2 Draught

You can specify the inclination angle to create an extruded solid with draught. Positive and negative angles are permitted.

36.2.3 Sketch for Cross-Section

Here you can select either a new sketch or an existing sketch for the cross-section of the extruded solid.

- **Create new sketch**

Activate this symbol if you want to create a new sketch for the cross-section of the extruded solid. To create the new sketch, proceed as described in the Sketches chapter. The **New extruded solid** window will remain active during the sketch creation. After completion of the sketch you can insert the extruded solid into your drawing by selecting **Preview/Apply** or **Create**.

- **Select sketch**

Activate this symbol to use an already existing sketch as cross-section. Identify the sketch, then select **Preview/Apply** or **Create** to insert the extruded solid into your drawing.

- **Delete after creation**

Activate this checkbox if you do not want to retain the sketch after its creation.
Please note however that the sketch will no longer be available for subsequent changes if you are working *without* Feature Technology! When working *with* Feature Technology you can retrieve the sketch at any time, even after its deletion.

Closed polylines (e.g. rectangles, circles etc.) which are located within a plane of the sketch, are interpreted as bores or material subtractions.

36.2.4 The “Reference” and “Feature” Checkboxes

Reference
You can reference extruded solids already during their creation, by activating the **Reference** checkbox.

After creating the extruded solid, HiCAD prompts you to save the part. Further information on referencing is provided in the chapter “Re-Use“ and in the Online Help.

Feature
If you want to record the creation of the extruded solid in the feature log, activate the **Feature** checkbox.

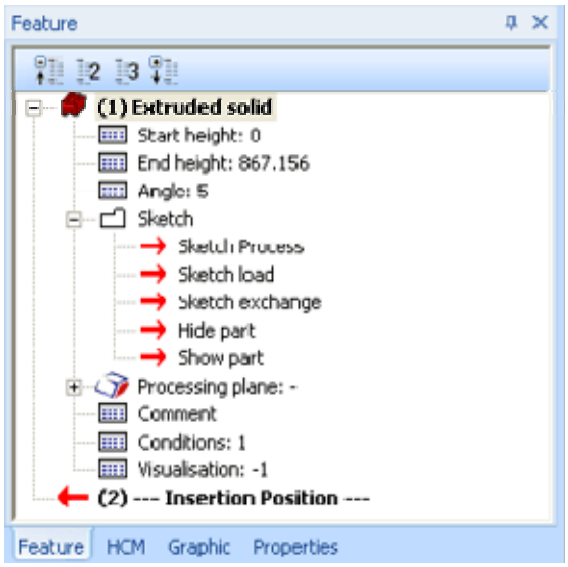


Fig. 83 Example of a feature log for extruded solid creation

If you want to apply subsequent changes to the cross-section of the extruded solid, select the **Sketch, Process** function in the feature log, change the sketch as required and confirm your changes with **Apply sketch**.

Or select the **Sketch, Exchange** function to replace the cross-section of the extruded solid with another sketch.

Use the **Sketch, Load** function to obtain a sketch which is totally independent of the underlying 3-D part. You can, for instance, use this sketch to create another extruded solid, a bore or similar.

Further information can be found in the Online Help.

see Online Help

36.2.5 The “Preview” and “Create” Buttons

If you want to see a preview of the extruded solid, click the **Preview** button. You can then correct the entered values or insert the part as displayed by clicking the **Apply** button.

If you click the **Create** button, the part is immediately inserted into your drawing.

Please note:

- If a preview is already displayed, this preview will, in case of any changes to the values in the input fields, only be updated if you confirm the changes with RETURN (ENTER).
- If you have selected dynamic height specification, you need to activate the dynamic mode after clicking the **Preview** or **Create** button. Please also note the prompt in the info bar.

36.2.6 Example

Extruded solids based on different value inputs for start height, end height and draught (inclination) have been created from the sketch shown below. Differences are shown in Fig. 85.

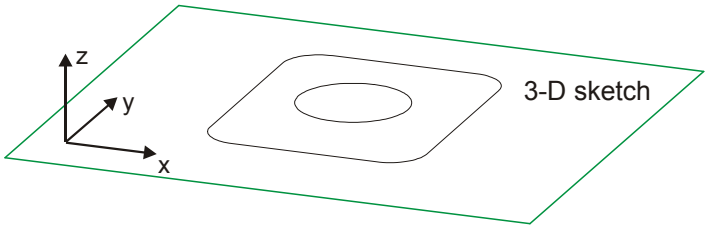


Fig. 84 Sketch

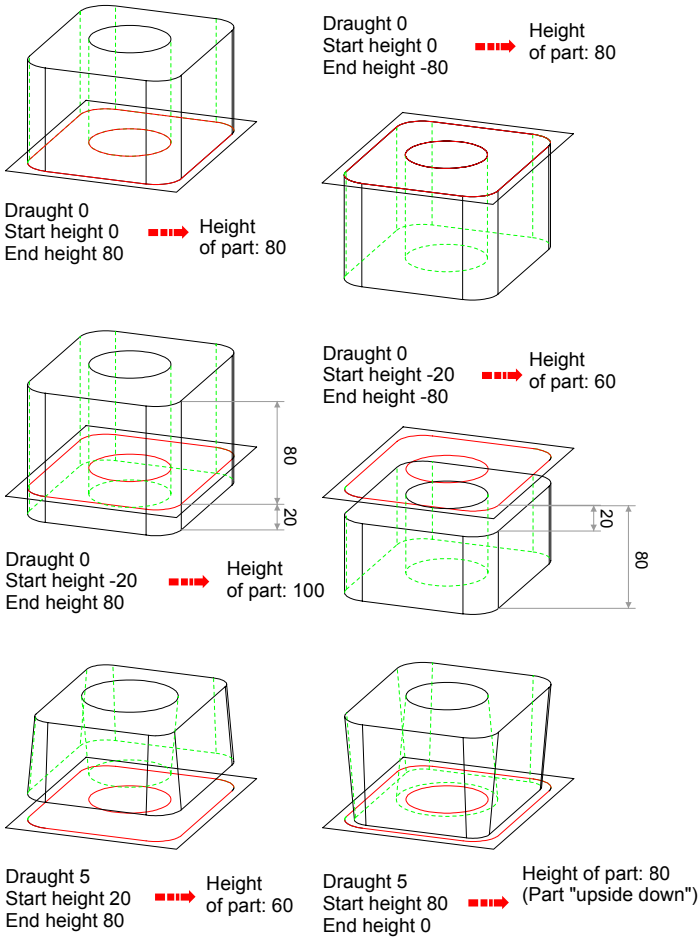


Fig. 85 Extruded solids with different start and end heights

36.2.7 Notes on Extruded Solids

- Circular arcs will produce cylindrical surfaces for analytical models.
- Inner cycles, i.e. closed polylines, circles and ellipses located in a sketch plane, are interpreted as bores/material subtractions.
- The height of the extruded solid, as well as the HCM constraints assigned to the sketch are taken over as parametric dimensions. These enable you to change the part at a later time.
- If a sketch is active when you call the **Extruded solid** function, this sketch will be used by default until you select or create a different sketch.

Before creating an extruded solid (or revolved solid, material subtractions or addings), you should make sure that the contour of your sketch is actually closed. You can use the following options for this:

- **Sketch > Sort GE in active sketch**
- **Information > Information 3-D Sketch** and
- **Information > Information 3-D-Sketch 1**

37 Parametric Dimensions

When Feature Technology is switched on, HiCAD automatically generates the parametric dimensions of a part and displays them if the part and the feature step with which the parametric dimensions have been generated is active (i.e. extruded solid).

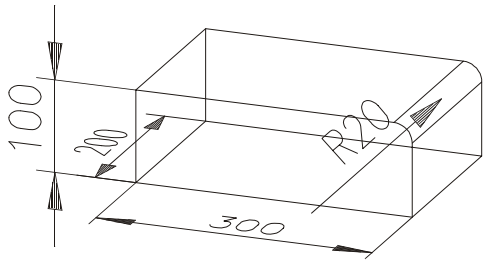


Fig. 86 Parametric dimensions

37.1 Changing Parts via Parametric Dimensions

If you right-click a parametric dimension, a context menu with various processing functions opens. The **Change feature parameters** function enables you to change the parametric dimension. Use the **Convert to drawing dimension** function to convert the parametric dimension to a “genuine” 3-D dimensioning.

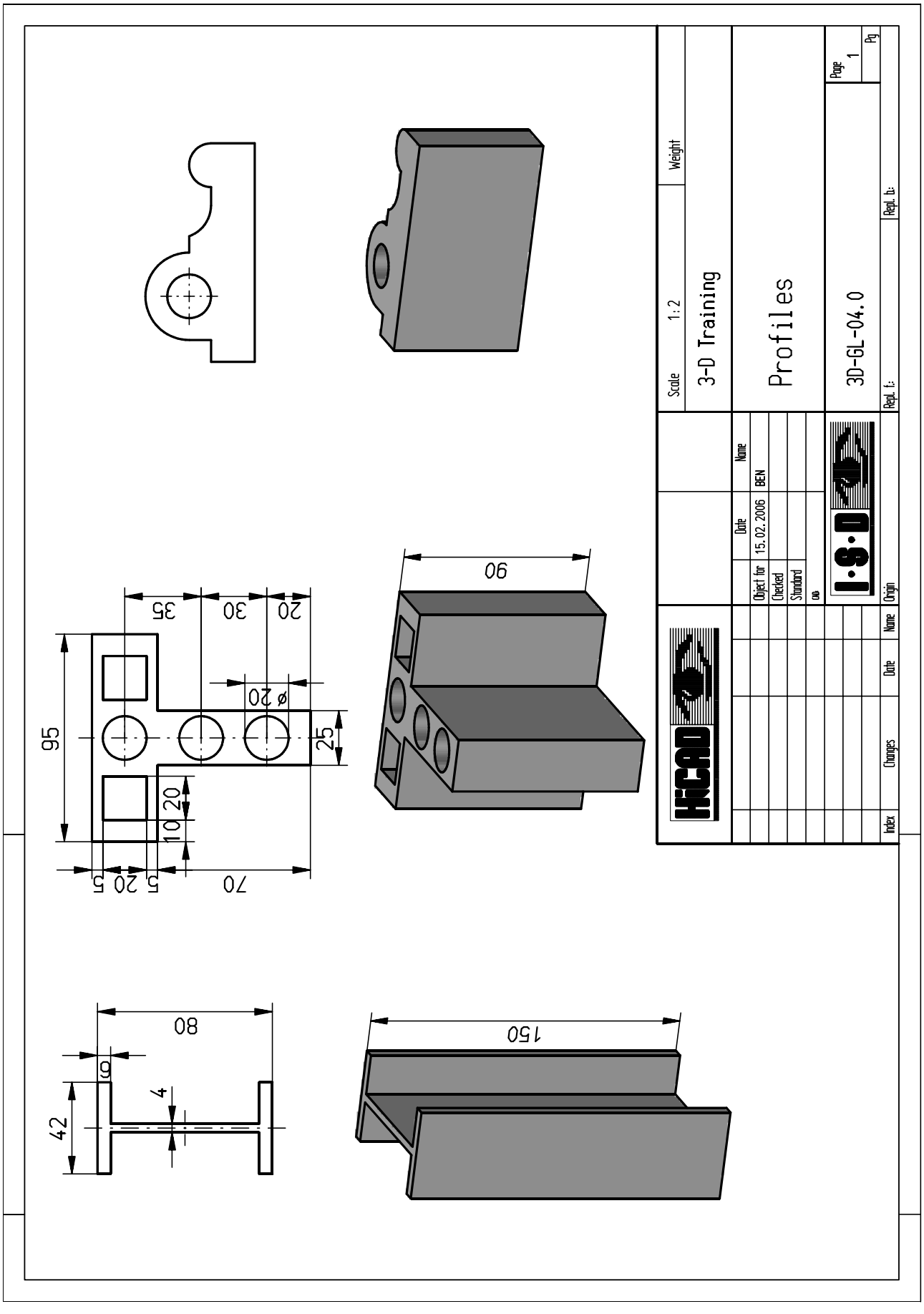
Fig. 87 Details of the **Parametric dimensions** context menu

Changing part dimensions in the feature log or via parametric dimensions is the most recommended way to model parts.

To hide parametric dimensions before printing a drawing, activate the **3-D Dimensioning+Text** tab and select **Tools > Table > Parametric dimensionings > Hide, all**.

37.2 Exercise 4 (3D-GL-04.0) Profiles

- Learning target: Sketches; extruded solids; conversion of 2-D parts to 3-D parts; modify part via parametric dimensions.
- Draw the first two profiles as extruded solids. Activate the referencing feature for this and all future parts.
- Load the part “BATTEN.FIG” from the hicad / Szenen / Examples directory. Create a sketch and copy the 2-D part into the sketch. Convert the sketch to an extruded solid (right-click on sketch).
- Select the parametric dimensions with the right mouse button and change them.
- Tip: Let us assume that you copied a 2-D part into a sketch. This sketch can now be converted to an extruded solid or a revolved solid. This is one option to transform a 2-D drawing into a 3-D drawing.
- Another option to transform 2-D parts into 3-D parts is the so called “2-D->3-D Conversion”. Ask your trainer for a demonstration of the conversion technique. Example drawings can be found in the directory local hard disc / hicad / Szenen / Beispiele (Examples).
- Space for notes:


Scale 1:2
Weight
3-D Training
Profiles
3D-GL-04.0
Page 1
Pg
Repl. f:
Repl. b:
Name
Date
Object for
Checked
Standard
BEN
15.02.2006
Origin
Name
Date
Changes
Index
90
35
30
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ø 20
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70
80
6
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42
150

38 Convert 2-D Parts to 3-D Parts

38.1 Convert via Sketch

3-D CAD models offer many advantages over 2-D drawings. For example, you can automatically derive standard views, sectional views and detail views from 3-D drawings. If you apply changes in one view, these changes are automatically applied to the other views (the so-called “associativity”). In 2-D, you would have to change all views one after another. Furthermore, you are enabled to perform motion simulations, collision checks, assembling simulations, weight calculations or statics calculations. Besides that, images derived from 3-D drawings look much more realistic than images from a 2-D drawing.

Therefore it makes sense to convert frequently used 2-D drawings to 3-D drawings. Very often you use sketches for the conversion:

- Go to the required view (front view, side view, top view) or define a processing plane.
- Define a sketch (not 3-D Sketch!).

In the **Others** function group of the **Sketch** tab you will find the **Project GE, Individual** function enabling you to copy **2-D Polyline**s and **2-D Part**s into a sketch with the same named menu items:

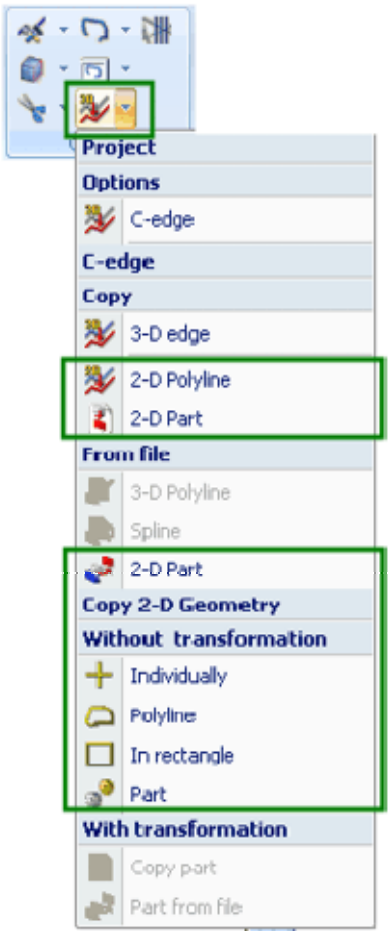


Fig. 88 Functions for the copying of 2-D lines into sketches (to convert 2-D to 3-D)

You can use the sketch to create extruded solids or revolved solids, add or subtract material.

38.2 2-D->3-D Conversion

If you have created *several views* in 2-D, the conversion of 2-D to 3-D can be performed even faster: In such cases, it is recommended to use 2-D->3-D- Conversion. To access this function, activate the **Drawing** tab and select Others > Modules > 2-D->3-D Conversion. The procedure will be explained in the following exercise by your trainer.

Further information on this function can be found in the Online Help.

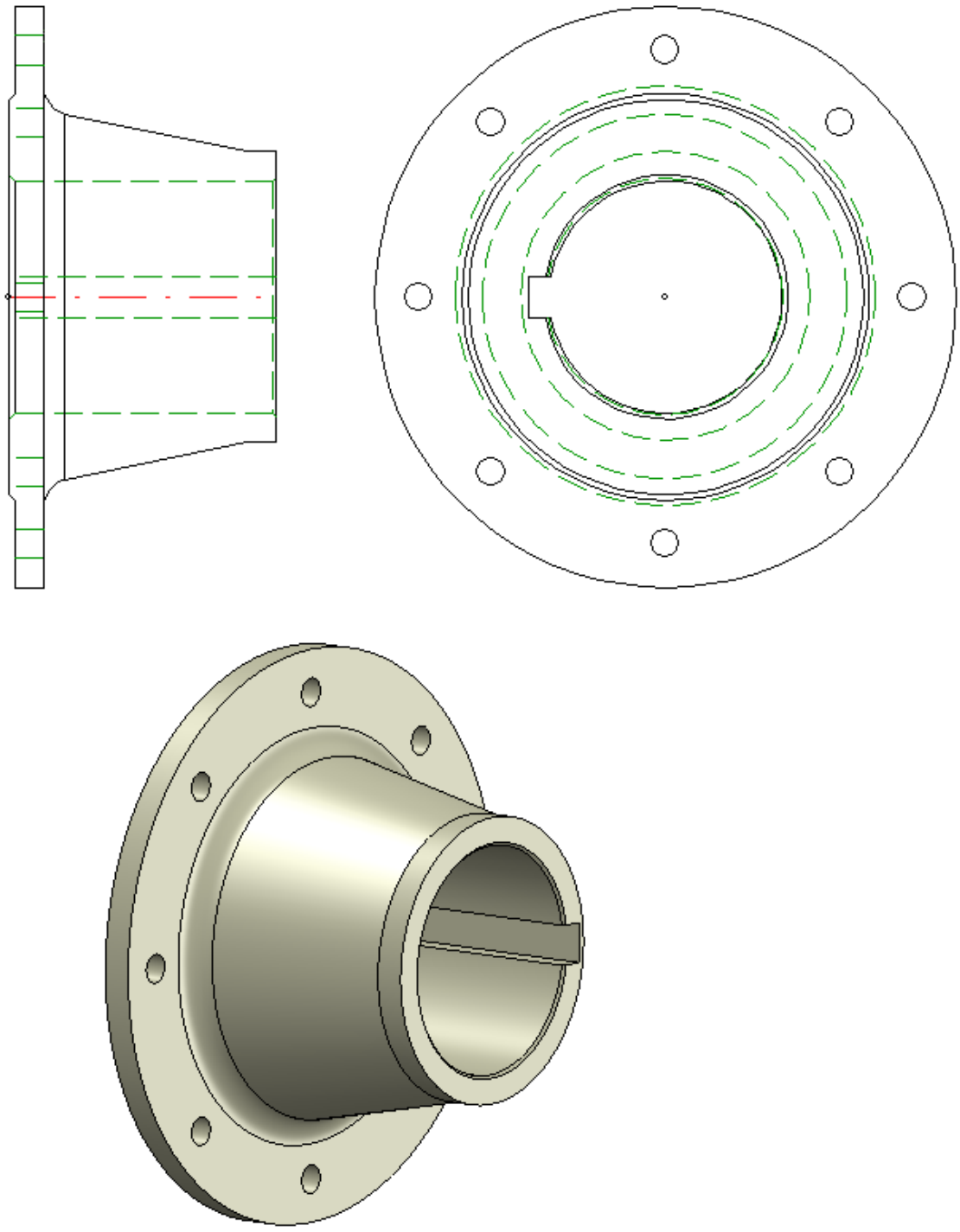
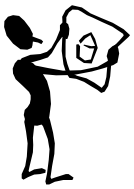
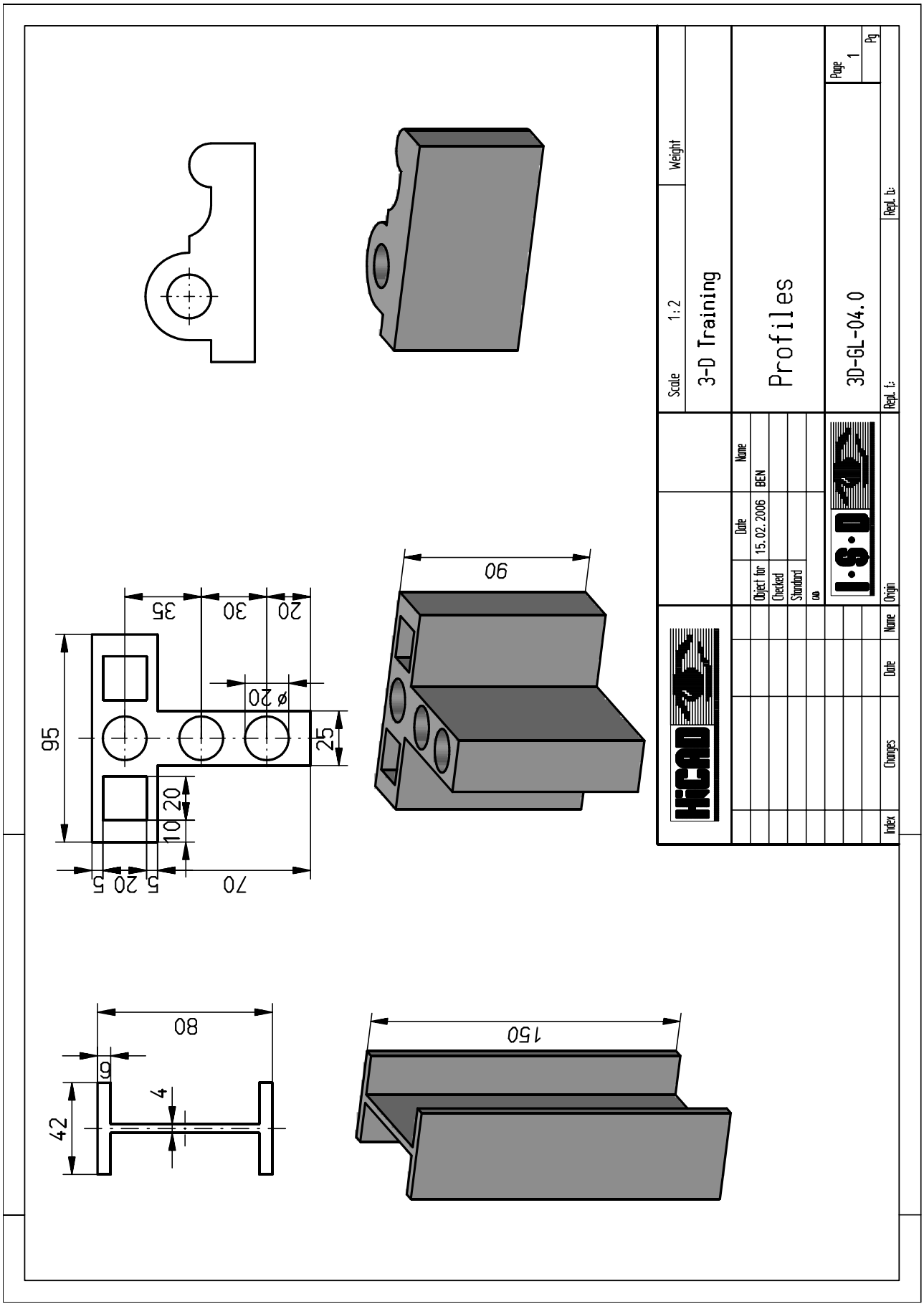


Fig. 89 3-D part created with 2-D->3-D Conversion



38.3 Continuation Exercise 4 (3D-GL-04.0) Profile

- Learning target: Conversion of 2-D parts to 3-D parts.
- Load the part LEISTE.FIG from the directory hicad / Szenen / Examples. Create your own sketch and copy the 2-D part into the sketch. On the “Sketch” tab, select the “Others” function group. Here you will find the function “Project GE, Individual” > Without transformation, Part”.
- The menu of the “Project GE, Individual” function provides a wide range of functions for 2-D lines or 2-D parts. You can use them to convert arbitrary 2-D parts to 3-D parts. These can, for instance, be parts from old 2-D drawings, but also contours from 2-D Standard Part CDs, which you require as 3-D parts in a 3-D drawing.
- Convert the sketch to an extruded solid (right-click sketch).
- Tip: Let us assume that you copied a 2-D part into a sketch. This sketch can now be converted to an extruded solid or a revolved solid. This is one option to transform a 2-D drawing into a 3-D drawing.
- Another option to transform 2-D parts into 3-D parts is the so called “2-D->3-D Conversion”. Ask your trainer for a demonstration of the conversion technique. Example drawings can be found in the directory local hard disc / hicad / Szenen / Examples.
- Space for notes:


Scale 1:2
Weight
3-D Training
Profiles
3D-GL-04.0
Page 1
Pg
Repl. by:
Repl. f.:
Name
Date
Object for
Checked
Standard
BEN
15.02.2006
HiCAD
I·S·D
Index
Changes
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39 Automatic 10-Minutes Save

In HiCAD you can open and process up to 18 drawings simultaneously. The currently processed drawing of the first 9 drawing files is automatically saved every 10 minutes. The automatic backups can be found on your local hard disk in the **temp** sub-folder of the **hicad** directory. In the lower area, select the extension .SZ**N** (instead of .SZA) in the File type field. HiCAD_11 means the automatic saving of the drawing in the *first* of the 18 thumbnails. HiCAD_91 is the automatic saving of the drawing in the 9th thumbnail. HiCAD_92 is the last but one saving of the drawing in the 9th thumbnail (i.e. 20 minutes old). If you are working with two screens, this will be the automatic saving of the drawing on the second screen.

Example: You have loaded 10 drawings into HiCAD and are now working on the sixth drawing. After 10 minutes, the drawing will be saved under the name HiCAD_61. The previous automatic saving of the drawing is copied and renamed to HiCAD_62 (Special case: When using two screens, it will be the drawing on the second screen). Later, you switch to the third drawing. From now on, it will be this drawing that will be saved under the name HiCAD_31 every 10 minutes. The already existing previous copy "HiCAD_31" will be copied and renamed to HiCAD_32. The tenth drawing will not be saved – even if you open and process it, as the automatic 10 minutes save exclusively applies to the first 9 drawings.

If you are working *without* database you need to copy the loaded 10-minutes save to another directory with the **Save as** function. (Reason: If the drawing remained in the temp directory it would be overwritten every 10 minutes.) If you are working *with* database, this will not be necessary. When loading the 10 minutes save, HiCAD will ask you: *Link to HELiOS document found. Take over file name (Y) or Delete link (N)?* Answer this question with YES. HiCAD will automatically copy the drawing back to the default path C: of the system file FILEGRUP.DAT.

40 Change Parts via Sketches

In the previous exercise, we have changed the dimensions of parts via their automatically created parametric dimensions. If we have created an extruded solid or revolved solid, material subtractions or addings via a sketch, we can also change these parts or material subtractions via the sketch. When we change the sketch, the extruded/revolved solid or material subtraction/adding derived from the sketch changes as well.

If you have switched Feature technology *off*, you must not delete the sketch after creation of the extruded solid, material subtraction etc. If Feature Technology is switched *on*, you can delete the sketch if desired, as you can retrieve it at any time with the **Sketch, Process** function in the feature log.

40.1 Change Line in Sketch

Activate a part (extruded solid, revolved solid, subtraction etc.) you have created with a sketch. In the **Feature** tab of the lower ICN window you can see the feature steps for the creation of this part. If the corresponding sketch has been deleted after creating the part, select the feature step for this part in the feature log and retrieve the sketch with the **Sketch, Process** function. The following graphic shows the sketch contour of a screw plug after selection of the **Sketch, Process** function.

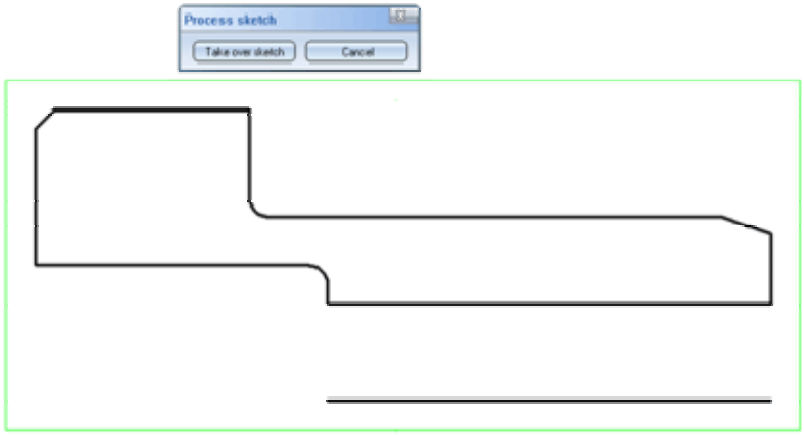


Fig. 90 Process sketch

Use the functions **Show only active part** and **Front view** to obtain a clearer representation of the sketch.

Apply some changes to the sketch, e.g. delete a chamfer change the width of the nut. Make sure during the process that you do not interrupt the closed contour of the part. Then select **Apply sketch**. The change is also applied to the revolved solid belonging to the sketch.

40.2 Apply Changes via 3-D C-Edge HCM (Edge Constraints)

In the last paragraph we have changed a sketch without parametrics. This paragraph explains how to change a sketch *with* parametrics. The 3-D C-Edge HCM (HiCAD Constraints Manager) enables the setting of constraints for edges (edge constraints). The 3-D C-Edge HCM is part of the 3-D HCM and is a tool allowing the positioning and moving of composite edge elements. In most cases, these are polylines in sketches, but can also be 3-D lines within a 3-D part. The corresponding elements are linked to each other by constraints, e.g. by

- Dimensional constraints
- Positional constraints and
- Dependencies between variables (Example: The width is defined via the variable “Width” and the height with the formula “Width/2+10”).

The 3-D C-Edge HCM transforms the corresponding elements in such a way that the specified constraints are fulfilled.

The image below shows the three function groups on the **HCM** tab used for sketches:

Fig. 91 Edge constraints (if sketch is active)

The image below shows the three function groups on the **HCM** tab used for constraints between parts:

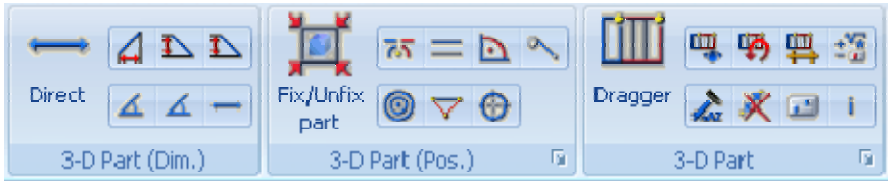


Fig. 92 Edge constraints, for lines between various parts

Constraints are assigned to individual elements of composite edges. These elements can be points, straight lines, circles or arcs.

Besides assigning constraints in order to position elements, you can use the **Dragger** to move individual elements while keeping to assigned constraints. You can also use the Dragger to detect remaining degrees of freedom in the drawing (i.e. “elements that still can be moved”).

Proceed as follows:

The definition of a parametric c-edge model consists of the following steps:

- Sketching of the required composite edges – either in a sketch or in a 3-D part
- Assigning of dimensional and positional constraints (edge constraints)
- You can enter values as well as variables for dimensional constraints
- If you assign variables (e.g. for the height of a plate or sheet in a sketch), you can also refer to already existing variables instead of entering a new variable (e.g. define the height with the formula “Width/2+10”). Please note that no spaces and arithmetic operators are allowed in names of variables! For example, “X-distance” would mean that you would subtract the “distance” variable from the “X” variable. Use the name “X_distance” instead.
- Testing of the model with the dragger or via the variables table (if variables have been defined); if required, correction of constraints or adding of missing constraints

Please note:

- Elements of a composite edge part (e.g. a sketch or a 3-D part with 3-D lines) can also be positioned relative to elements of another composite edge part. The defined HCM constraint is however always assigned to the active c-edge part.
- Only “properly drawn” composite edges can be processed properly: Please make sure that identical points, for example, are actually identical, or that lines do not consist of many segments etc.
- Do not confuse “normal” HiCAD dimensionings and dimensional edge constraints!
- A parametric c-edge model can only be changed via the constraints assigned with the *3-D C-Edge HCM*.
- The elements of a HCM model are highlighted in colour, according to the corresponding degrees of freedom:
  - Fixed elements orange
  - underdefined blue
  - overdefined red
  - fully defined green

40.2.1 Start 3-D Edge HCM

Start the *3-D C-Edge Constraints* as follows:

- Create a sketch or a 3-D dummy part.
- Activate the **HCM** tab.

40.2.2 The HCM Model

40.2.2.1 Create a HCM Model

A HCM model is automatically created if you

- assign the first HCM constraint manually, or
- selected the AutoConstraints in the Settings and then call the **Update HCM system** function, or
- **Assign dimensional constraints automatically** with the same-named function, or
- during sketching, if you selected **Specify constraints** via **3-D C-Edge > Tools/Settings**.

40.2.2.2 Display of the HCM Model in the ICN

In the **HCM** tab of the lower ICN window, all HCM-constraints assigned to a composite edge part are shown. The constraints are linked to the part and are saved together with the part. In this way, you are enabled to change assigned constraints at a later time if required.
The **HCM** tab of the lower ICN window provides information about the constraints of the individual elements of the active composite edge. It is also shown whether any elements are underdefined.

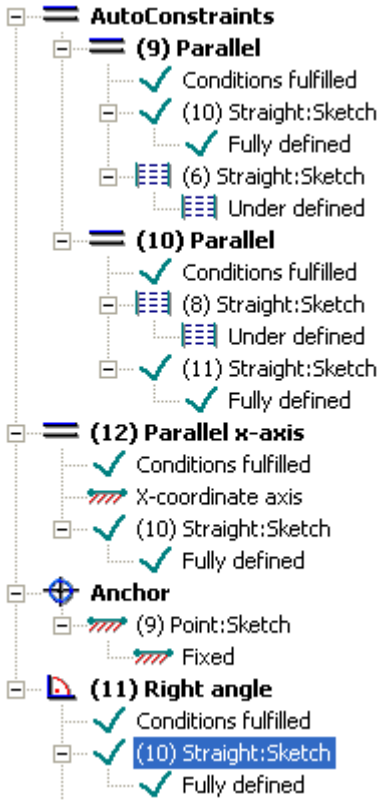


Fig. 93 HCM constraints in the ICN

You can select individual constraints in the HCM tab with a right-click. Here you will find the functions *Sort according to geometry* and *Sort according to constraints*. You are also enabled to **Change**, **Delete** etc. individual constraints.

40.2.2.3 Highlight Elements

If you select a HCM constraint in the ICN with the left mouse button, the corresponding elements in the drawing are highlighted. This highlighting is retained even after part rotations. Surfaces to which the constraint has been assigned are marked green. This default setting (mark_faces=1) is saved in the KZUGPAR.DAT file in the SYS directory.

40.2.2.4 Change HCM Model

To change the HCM model of a composite edge, you need to activate the corresponding c-edge part (e.g. the sketch). The assigned HCM constraints are displayed in the HCM tab of the ICN.

Proceed as follows:

- Right-click the required constraint in the ICN and activate in the desired function in the context menu. If you left-click a constraint in the ICN, the corresponding geometry is highlighted in the drawing.
- Right-click the required constraint in the drawing and activate the desired function in the context menu.

- Select the desired function in the **HCM** tab (e.g. **Change dimensional constraint**)

40.2.2.5 Use AutoConstraints

Besides entering dimensional and positional constraints manually, you can also automatically search for relations and constraints between the elements of a composite edge by selecting the **Use AutoConstraint** option in the **Settings** dialogue. To access this option, activate the **Tools/Settings** function in the **3-D C-Edge** function group. When starting HiCAD for the first time, the AutoConstraint option is activated.

To detect the constraints automatically, activate the **Update HCM system** or **Assign dimensional constraints automatically** option. All automatically detected relations/constraints are displayed in the HCM tab of the lower ICN window and also represented in the drawing. For further information on AutoConstraints please also see the **Tools/Settings** paragraph.

If you want to assign AutoConstraints already during drawing of lines, circles etc., select **Tools/Settings > Specify constraints**.

40.2.3 Dimensional Constraints

3D-Kantenzug(Maß...)

Use the functions of the **3-D C-Edge (Dim.)** function group to assign distance, angle and radius constraints.

You can also use formulas or variables for all constraints requiring a value input. Formulas are a common arithmetic expression, within which variables may occur. If the start value of the variable has not been defined in the sketch/part or in one of the superordinate parts yet, HiCAD prompts you to specify the start value.

Dimensional constraints are, depending on the selected function, represented as distance, radius or angle dimension.

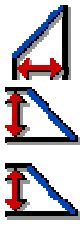
Please note:

- The dimension points depend on the dimensioning options specified in **Tools/Settings**.
- The dimensions created by the HCM are no “normal” HiCAD dimensions. Conversely, HiCAD dimensions are no dimensional constraints of the 3-D C-Edge HCM.
- Dimensional constraints assigned to a sketch are taken over as parametric dimensions.

see Online Help

The following paragraph explains the **Distance parallel to X-axis**, **Distance parallel to Y-axis** and **Distance parallel to Z-axis** functions of the **3-D C-Edge (Dim.)** function group. All other functions are described in detail in the Online Help. We also offer a 2-day training on Parametrics.

40.2.3.1 Distance Parallel to X-, Y- Z-Axis



Use these functions to assign a distance constraint between:

- 2 points,
- 1 edge and 1 point,
- 2 edges.

In contrast to the **Direct distance** function, the distances are defined in X-, Y- or Z-direction.

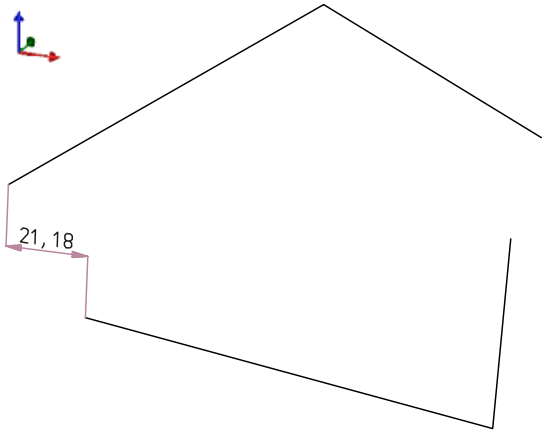


Fig. 94 Distance constraint, parallel to X-axis

40.2.4 Define Positional Constraints

Use the functions of the **3-D C-Edge (Pos.)** function group to assign positional constraints, such as:

- Fix/Unfix
- Parallel X-axis
- Coincidence,
- Symmetry,
- Tangential,
- … and many others

Positional constraints are represented in the drawing by the following characters:

T	tangential	-	coincidence
L	perpendicular	=	parallel
K	concentric	M	mid point
S	symmetric	E	equal distance
F	Fix		

See Online Help

The following paragraph explains the functions **Fix/Unfix, Concentric,** and **Parallel X-,Y-,Z-axis** of the **3-D C-Edge (Pos.)** function group. All other functions are described in detail in the Online Help.

40.2.4.1 Fix/Unfix

A special type of constraint is the fixing of individual elements. If an element is fixed, it will not be moved in case of any necessary transformations.

You can fix

- Edges
- Circles/arcs and

- All points found with point option I.

Identify the element or the point. If you have already fixed an identified element, the fixing is undone (removed).

40.2.4.2 Parallel X-, Y-, Z-Axis

If HiCAD recalculates the HCM model, it may happen that the sketch is rotated. Although HiCAD keeps to all HCM constraints, a rotation of the sketch is often unwanted. Use the **Parallel X-axis** and **Parallel Y-axis** functions (if a 3-D instead of a sketch is used, also **Parallel to Z-axis**) to prevent the rotation of the sketch (or the 3-D polyline), as these functions allow an alignment of lines parallel to X-, Y- or Z-direction of the active coordinate system. The other lines, e.g. those of a rectangle, are automatically protected against unwanted rotations via constraints such as **Parallel** and **Perpendicular**.

Select the desired type of alignment and identify one of the elements of the HCM model.

The other elements are aligned to the identified element, i.e. the HCM model is adjusted.

Very frequently, the first constraints assigned in a sketch are **Fix/Unfix** and **Parallel X-axis.** For example, by fixing the corner of a rectangular sheet in a sketch, you will also fix the direction of a possible lengthening of the sheet. Without fixing of the corner, the lengthening direction would be completely arbitrary.

40.2.4.3 Concentric

You can assign this constraint between

- a circle/arc and a point
- a circle/arc an a straight line
- two circles/arcs
- a circle/arc and a cylinder
- a circle and a sphere

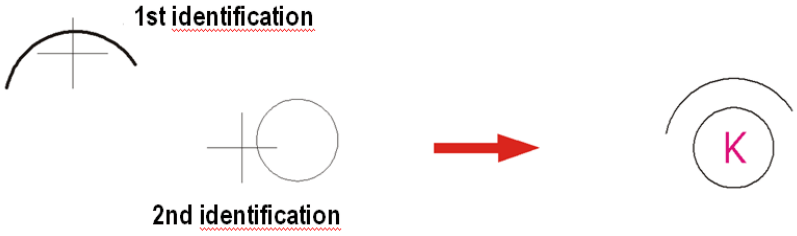


Fig. 95 Specify concentricity constraints

40.2.5 Drag Dynamically

Use this function to change a parametric model by dynamic dragging of lines and points while taking already specified constraints into account. In this way, parameter changes of the elements are visualised.

Use the crosshairs to select a point of an element of the composite edge. If the position of the selected element has not yet been specified, it can be dragged dynamically with the crosshairs. Apply the modified model with a left-click. Use the middle mouse button to restore the original situation.

The **Dragger** function is often used to detect remaining degrees of freedom in the HCM model. You can then assign dimensional or positional constraints to elements that can still be moved.

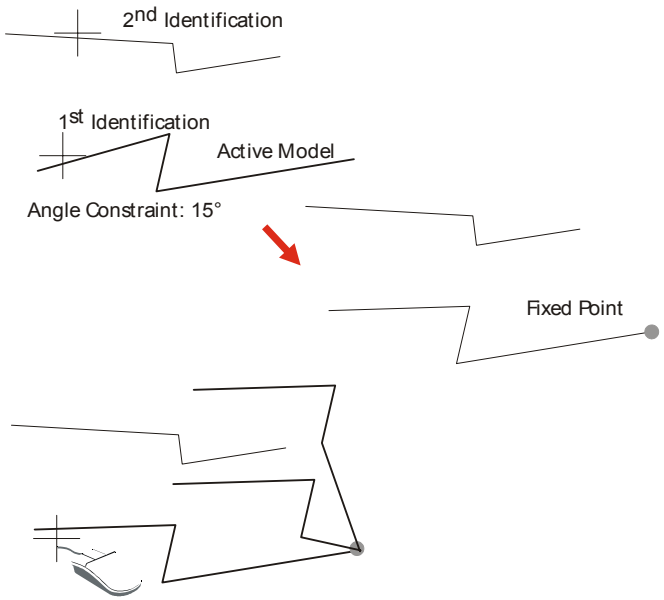


Fig. 96 Drag

You rotate the c-edge about the fixed point by moving the identified line. The length of the identified line and its angle to the second c-edge remain unchanged. Position and length of the third edge will be changed.

Please note that during dragging the dragger mode settings specified in **Tools/Settings** are taken into account.

40.2.6 Change Dimensional Constraint

Use this function to change a constraint that has already been assigned to a composite edge.

Identify the dimensional constraint, and enter the new value.

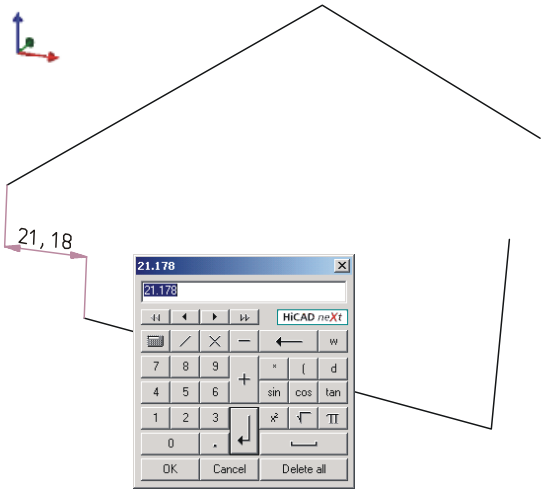


Fig. 97 Change dimensional constraint

Alternatively, you can also change dimensional constraints via the ICN.

40.2.7 Move Dimension Line

Use this function to move the dimension line of a dimensional constraint. Identify the constraint, and drag the dimension line to the required position.

Please note: You can also directly move the dimension line with Drag & Drop.

40.2.8 Update HCM System

Use this function to update the active HCM system.

This function can be helpful if, for example, not all constraints are displayed after clicking Redraw. In addition, you can use this function to search for AutoConstraints.

40.2.9 Change Variables

Instead of value inputs for distances, angles, etc. you can also use variables to assign dimensional constraints. This function enables you to display a list of the variables used in a model.

To change the values assigned to the variables in the dialogue window, double-click the required value and overwrite it.

40.2.10 Tools/Settings

Use this function to specify various settings for the C-Edge HCM. In addition, the context menu of the function allows a selection of further constraints, in addition to already assigned constraints.

40.2.10.1 Settings (3-D Part HCM)

You can find this function in the **3-D Part** function group of the **HCM** tab. Use this function to set the parameters for the auto search of constraints/relations between the elements of a composite edge. Furthermore, you can select the dragger and calculation mode.

40.2.11 Delete constraints, individually

Use this function to delete individual constraints of the active system. Identify one of the geometries to which you have assigned a constraint. HiCAD displays the found constraint and asks you whether you want to delete it.

Alternatively, you can also delete constraints directly in the ICN, or by right-clicking a constraint in the drawing.

40.3 Link Dimensions with 3-D C-Edge HCM

You can use the functions in the **3-D C-Edge (Dim.)** function group to define distances, angles, diameters, radiuses etc. and assign values to them. We can however also assign names of variables to them. The first character of the name needs to be a letter. Besides that, we can select already existing constraints via the **Change dimensional constraints** and enter a variable name instead of a value. The following function enables you to edit a list of all assigned names of variables and conveniently change them by simply double-clicking a value.

40.3.1 Change Variables

Instead of entering values for distances, angles, etc. you can also use variables to assign constraints. Use this function to display a list of the variables used in a model.

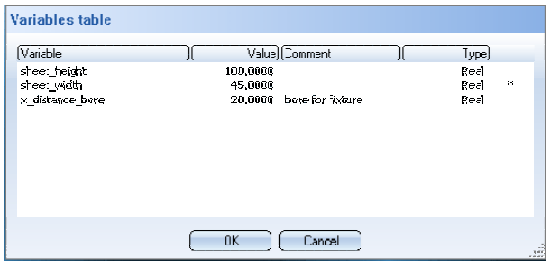


Fig. 98 Change variables

The values assigned to the variables can be changed by double-clicking and overwriting them.

40.3.2 Cross-Referential Variables Names

The assignment of variables names is particularly interesting if you refer variables to each other. In the following exercise, the Depth shall receive the variables name H1, and the Inner thread diameter the variables name D0.

Use the **Change variables** function to display the result:

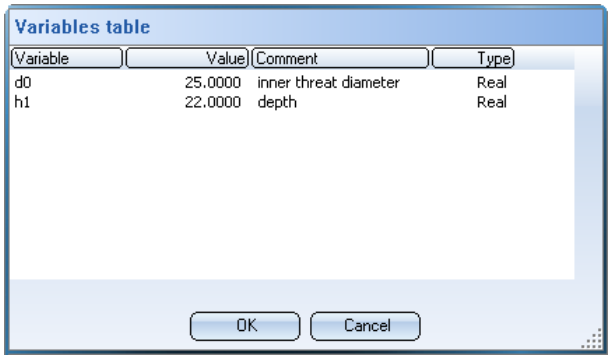


Fig. 99 Depth and Inner thread diameter as variable

You can use the **Distance parallel to Y-axis** function and enter the formula D0*1.25 for the inner diameter. This means that the inner diameter will always be 1.25 times greater than the inner *thread* diameter. As you did not assign a new variables name (D0 existed already), the variables table contains only the variables D0 and H1.

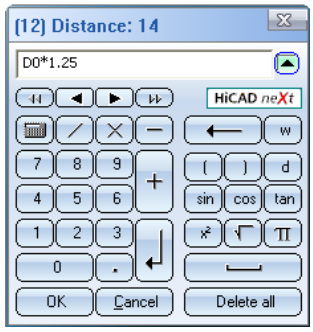
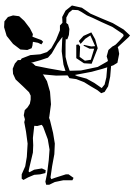


Fig. 100 Inner diameter 1,25 times larger than inner *thread* diameter

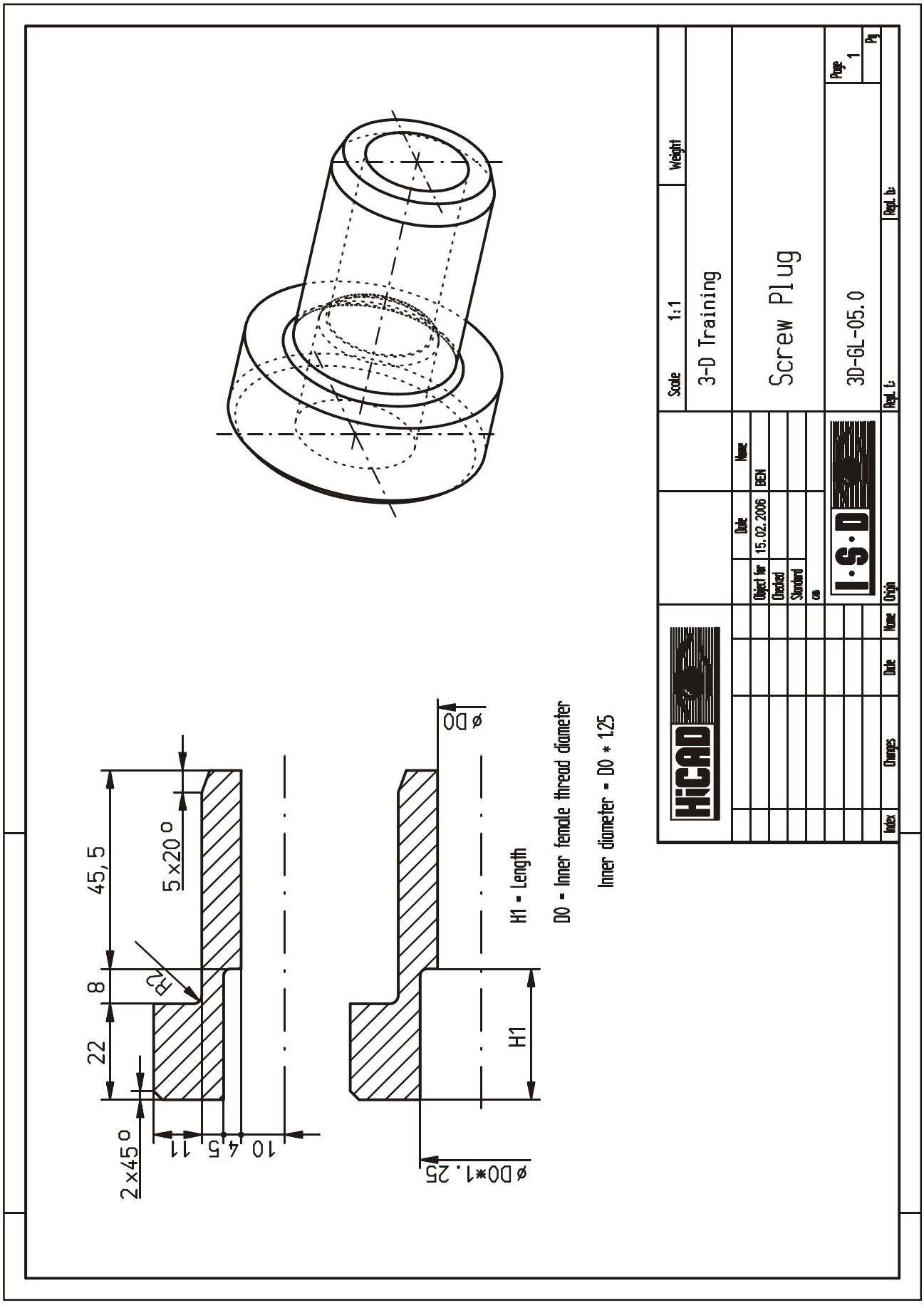
If you have used variables names in a sketch and have already created a 3-D part with the sketch (e.g. an extruded solid or a revolved solid), you can also access the variables table via the part: Right-click the part and select the **Properties / Part variables** function. This is even possible for solid primitives to which material subtractions have been applied via a sketch with variables names.

Due to reasons of time we could show you only a few Parametrics functions in our 3-D training. Extensive information on the Parametrics module is provided in our 2-day Parametrics training, such as the creation of local and global variables, constraints between *different* parts, motion and function simulations, dynamic collision checks, creation of Feature and Design Variants etc.



40.4 Exercise 5 (3D-GL-05.0) Screw Plug

- Learning target: Sketches; revolved solids; modifications of the revolved solid by deleting and redrawing lines in the corresponding sketch; modifications of the revolved solid via the 3-D Constraint Manager; modifications of the revolved solid via the 3-D Constraint Manager using variables.
- Draw the revolved solid with the given dimensions.
- Save the drawing!
- Select the “Process Sketch” function in the feature log. Delete the 2x45 chamfer and the R2 fillet. Sort lines if necessary. Take over sketch.
- Select the “Process Sketch” function in the feature log. Change some dimensions with the help of the 3-D c-edge Constraint Manager. Fix – if necessary – some points or lines. Use the dynamic drag function to control if dimensions change into the direction required and if the characteristics of the other lines (e.g. parallelism etc.) are maintained. Take over sketch.
- Select the “Process Sketch” function in the feature log again. The dimensions you defined in the previous section with the help of the 3-D c-edge Constraint Managers are still there. Change some of the data. Take over sketch.
- Again, select the “Process Sketch” function in the feature log. Delete all constraints of the active part. If you have made too many modifications already, load the drawing you saved in the 3rd step. Allocate the variables H1 and D0 with the help of the 3-D c-edge Constraint Managers. The inner diameter is always 1.25-times the internal thread diameter D0. Do not use any new names for variables for the inner diameter, but D0*1.25.
- Load the automatic data back-up of your drawing. The file is named hicad_N1 and can be found on the local hard disc in the hicad / temp directory. N represents the number of the drawing in which you are working. If you work in the third drawing, its data back-up is named hicad_31. Important: In the current version you need to switch the file extension to „.SZN“ in the “File type“ field type at the bottom.
- Tip 1: We learned how to change parts via parameter dimensions in the previous exercise. We modified the workpiece via the corresponding sketch in this exercise. We changed the lines directly in the sketch for the first modification. In the second step, we changed the sketch with the help of the 3-D C-edge Constraint Manager. In the third step, we allocated variable names in the 3-D C-edge Constraint Manager, to create dependencies between different dimensions.
- Tip 2: The function “Load sketch” in the “Feature” tab of the ICN allows the loading of the sketch belonging to the active part. You will get a copy of the sketch that has nothing to do with the original processing any more. Thus you can apply this sketch to other parts as well to create extruded solids, revolved solids, recesses or butt-joints. Example: You need the same recess for two different parts, and a butt-joint with identical contour for a third part.
- Space for notes:


Scale 1:1
Weight
3-D Training
Screw Plug
3D-GL-05.0
Page 1
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Name
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BEN
Object for
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Standard
as
I·S·D
HiCAD
Index
Changes
Date
Name
Origin
Repl. to:
Repl. by:
ø D0
ø D0*1.25
H1
H1 = Length
D0 = Inner female thread diameter
Inner diameter = D0 * 1.25
45,5
5x20°
R2
8
22
2x45°
11
4,5
10

41 Processing Planes and Subtractions

41.1 3-D Processing Plane

Sketches; also see Online help

When deriving 3-D parts from sketches (e.g. creating extruded solids or subtractions), you often create your sketch in the screen plane: You choose front, side or top view and then select, e.g. the **Extruded solid** function. The processing plane will then automatically be located in the front, side or top view etc. You can however also create and process sketches directly in the axonometric view by defining processing planes. The drawing plane will be placed into the selected processing plane.

Processing planes; also see Online Help

You can define several processing planes, only one of which can however be active. The active processing plane is highlighted by a green frame, all other processing planes have a dark red frame.

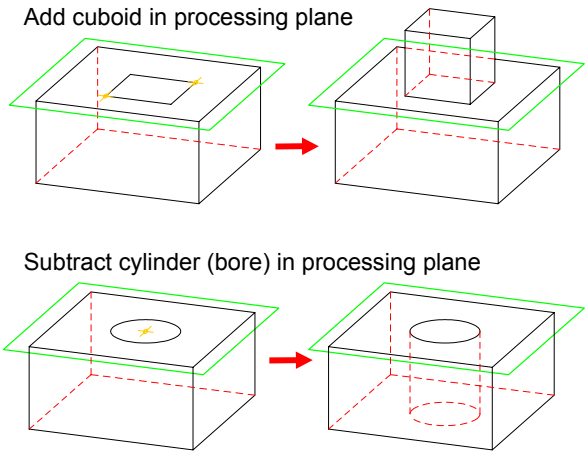


Fig. 101 Working with processing planes: Examples

The function for processing planes can be found on the transparent toolbar. Left-click to activate the **Surface-Edge-Point** function, right-click to activate the context menu for processing planes. The context menu is also activated if you right-click an already existing processing plane.

In most cases you define a processing plane via 2 edges. Please note that the first selected line is the future X-axis, the second line the future Y-axis. You normally want the X-axis in the processing plane to be horizontal and running from left to right. Therefore look vertically at the surface and select from this viewing direction a horizontal line in the left half. The future X-axis will then run horizontally from left to right. Again, look vertically at the surface and select a vertical line in the lower half. The future Y-axis will then run from bottom to top.

Almost all 3-D functions refer to the currently active coordinate system (e.g. direction of a material subtraction, direction of a text etc.). For all of the approximately 30 functions prompting you to specify surfaces, please therefore note the following: First select for the X-direction a horizontal line on the left, then select for the Y-direction a vertical line in the lower half.

41.1.1 Plane Functions

The previous paragraph explained how to define a processing plane *prior to* calling a function such as Extruded solid, or Add/Subtract part. But even if you have already called one of these functions and HiCAD has already created a sketch, you can still change the plane in which the sketch is located (please note however that the sketch must not contain any line elements). To do this, activate the **Sketch** tab and select **New > New plane…**.

41.2 Subtraction

In the Subtraction function group of the 3-D Standard tab, you will find the functions **Subtract part, via translation**, **Subtract part, with translation+ depth** and **Subtract part, via rotation**.

Fig. 102 3-D Standard tab, **Subtraction** function group

The **Subtract part, via translation** function (black rectangle) creates a material subtraction running through the entire part.

The **Subtract part, with translation+ depth** (dashed rectangle) creates a material subtraction with a limited depth (e.g. blind holes).

Subtractions are based on existing or newly created sketches, the polyline of which runs completely through the part or penetrates it with a specified depth.

The part created by the subtraction is the difference between the active 3-D part and the extruded solid derived from the sketch. The name of the 3-D part is retained as part name.

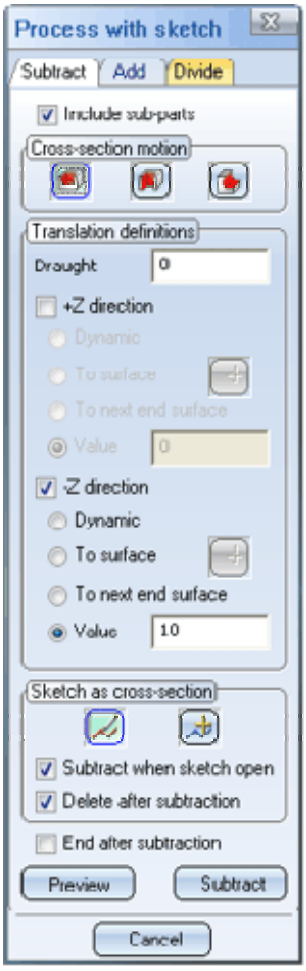


Fig. 103 Input window for the **Subtract part, with translation** function

To create a subtraction with translation and limited depth, proceed as follows:

- Select the **Subtract part, with translation+ depth** function (dashed line in the upper image on page 135).
- If you want to create the subtraction with draught (inclination), enter the inclination angle in the Draught field. The position of the draught is influenced by the mathematical sign (positive or negative value).
- Select the type of height definition, e.g. in the –Z-direction field. Example: If you select **Value**, you can enter a value for the depth of the translation. If you select **To surface**, you can specify a surface until which the subtraction will extend. If you select the **Dynamic** depth specification, left-click on the drawing area, drag the cursor to the required position, and left-click again (RET). HiCAD shows you the depth specified by the cursor position. If required, you can correct this value.
- If you selected **Value**, enter the subtraction depth.
- If you want to use an already existing sketch, click the **Select sketch** button and identify the sketch.
- To create a new sketch, click the **Create new sketch** button. HiCAD then activates the **Sketch** tab. Create the required sketch.
- Click **Preview**, then click **Apply** to execute the subtraction.

The position of the subtraction is determined by the position of the sketch – see the graphics below.

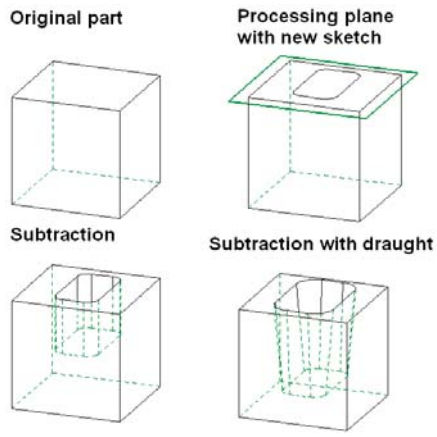


Fig. 104 Insert subtraction

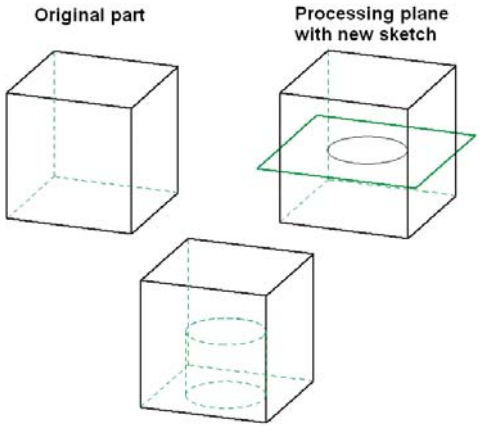


Fig. 105 Position of subtraction

42 Dimensionings

42.1 Sheet Views

The view structure of the current drawing is shown in the **Views** tab of the upper ICN window. HiCAD distinguishes between

- Model views and
- Sheet view.

In the views of the Sheet area you can create and name your production drawings and dimension the parts.

To activate an area or a view, simply move the mouse pointer onto the corresponding item in the ICN and left-click. The name of the view is displayed red.

To *process* a view, move the mouse pointer on the corresponding item in the ICN and right-click. Now you can select the required processing function in the context menu.

If the **Views** tab is open and a view is still active, information on this view is displayed on the **Properties** tab of the lower ICN window.

When dimensioning a part, you always activate (besides the part) the sheet view. Further details on model views and sheet views are provided in Chapter 43 of this training book.

42.2 3-D Dimensionings

Functions for 3-D dimensioning and texts can be found in the **3-D Dimensioning** tab.

You can also activate the tab by double-clicking already existing 3-D dimensions or texts.

42.2.1 General Notes on Dimensionings

To insert 3-D dimensionings, activate (besides the sheet view) the part you want to dimension and use the appropriate dimensioning, symbol or text functions. **Dimensions, texts, symbols etc. are always assigned to the active part.**

To process a dimensioning, move the cursor on the required dimensioning and press the right mouse button.

The 3-D dimensioning is applied to the XY-plane of the currently active coordinate system.

If you want to choose a different plane than the XY-plane, right-click after specifying the 2nd dimensioning point. You can then select a different dimensioning plane.

It is also possible to define a processing plane (= new XY-plane) *before* setting dimensionings or texts.

The next paragraph provides a description of the functions for axially parallel and angular dimensions:

42.2.2 Linear Dimensions, Axially Parallel

Use the functions of the **Parallel** function group to create axially parallel dimensionings. These will retain their position in case of transformations of the part (i.e. a rotation). This means that they will still be axially parallel even if a 30° rotation is performed on the part (which will not be the case if you use the **Linear dimensions, free** function instead).

Further functions are contained in the **Others** pull-down menu (small blue arrow):

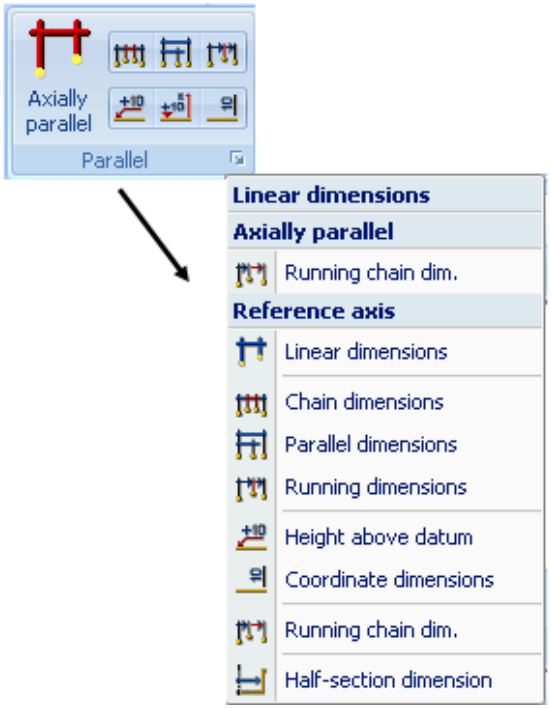


Fig. 106 The **Parallel** function group

All types of dimensioning require the identification of two edges or points. These can be arbitrary edges or points in the drawing. For instance, these points need not necessarily be end points of an edge and can also belong to different parts.

Dimension figure position

A third point must be specified for the position of the dimension figure. In this mode you have the following input options:

- **Switch dimensioning plane**

Before specifying the position of the dimension figure, you can switch the dimensioning plane if required. If you do not want to use the dimensioning plane suggested by HiCAD, press the right mouse button (if the suggested dimensioning plane is correct, you can directly move on to the next paragraph **Position the dimension line or dimension figure**). You can then, for example, select the following option:

Switch to ZX-plane

After selecting the ZX-plane, the dialogue for dimension line positioning is continued.

Position the dimension line or (RMB) dimension figure

Left-click the point where you want to place the dimension line. If you also want to position the dimension figure, right-click to open a context menu offering various options for this:

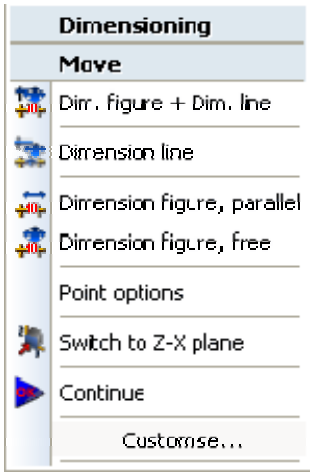


Fig. 107 Move dimension FIGURE

Selection of a reference line

Instead of using the point selection option described above, you can also select the dimension line of another dimensioning with a left-click. This reference dimension is then highlighted in pink. Your new dimensioning will then appear at the same height as the reference dimension.

The following image shows dimensionings in three different planes:

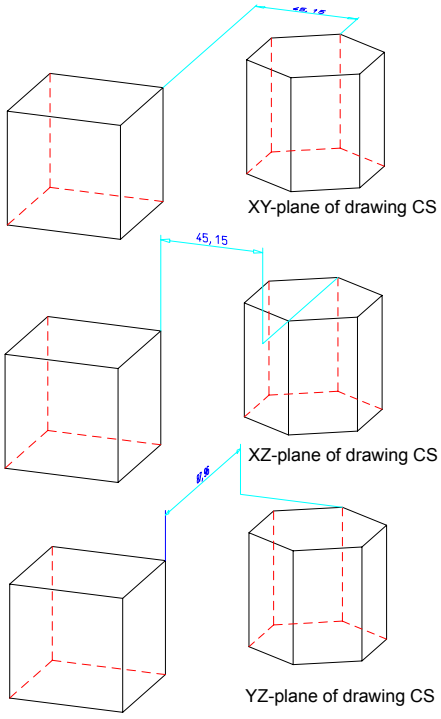


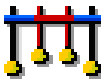
Fig. 108 Dimensioning in different planes

See Online Help

Now let us take a look at the **Create new chain dimension** and **Create new parallel dimension** from the **Parallel** function group.

Detailed descriptions of all further functions can be found in the Online Help.

42.2.2.1 Create New Chain Dimension



Use this function to create a chain dimension, i.e. a group of dimensions in which two consecutive sub-dimensions have one common dimension point.

Identify the object you want to dimension, and the position of the dimension line. Then specify the following points.

You can refine chain dimension by inserting intermediate points, or you can delete individual sub-dimensions. The length of the individual dimension lines is automatically corrected.

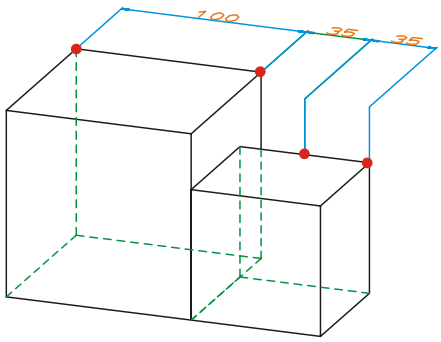


Fig. 109 Chain dimensions

42.2.2.2 Create New Parallel Dimension

Use this function to create a parallel dimension, i.e. a group of dimensions have the same start point and the dimension lines of the sub-dimensions a fixed distance from each other.

Identify the object you want to dimension, and the position of the dimension line. Then specify the following points.

You can refine chain dimension by inserting intermediate points, or you can delete individual sub-dimensions. The length of the individual dimension lines is automatically corrected.

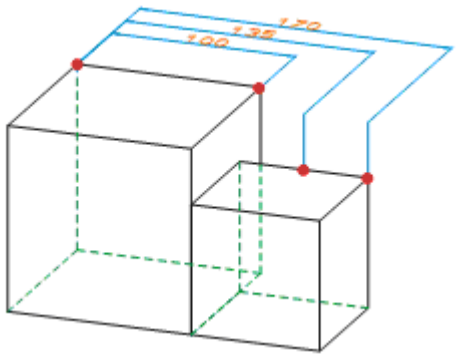


Fig. 110 Parallel dimensions

42.2.3 Angular Dimensions

Use the functions of the **Angle** function group to create angular dimensions.

Further functions ate contained in the **Others** pull-down menu of the function group (small blue arrow).

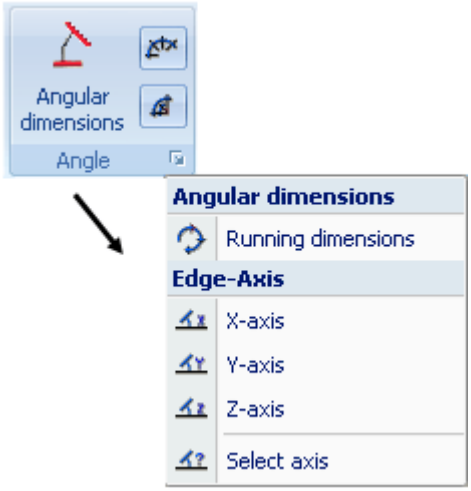


Fig. 111 The **Angle** function group

The legs of the angle you want to dimension can be specified by identifying two edges, specifying start and end point of the two lines, or selecting 2 surfaces.

For dimension line positioning, HiCAD prompts you to specify another point. As for the previously described linear dimensions, you have the option to right-click and open a context menu. Here you will find various options to move dimension lines and dimension figures, or allow angles greater than 180 degrees.

42.2.3.1 Create New Angular Dimension

Use this function to dimension angle between 2 edges.

Identify the two legs and define the position of the dimension line. If you also want to change the position of the dimension figure or allow angles greater than 180°, right-click to open a context menu after selecting the two legs.

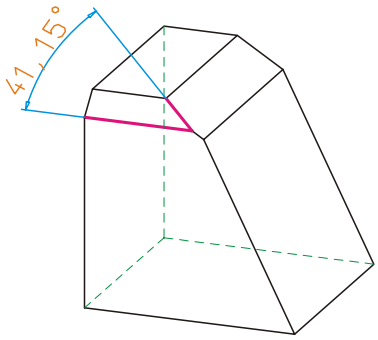


Fig. 112 2 edges/individual dimension

42.2.3.2 X-Axis, Y-Axis, Z-Axis

Use these functions to dimension angles between an edge and the X-, Y- or Z-axis.

Identify the leg. The second leg is automatically the X-, Y- or Z-axis. Then specify the position of the dimension figure.

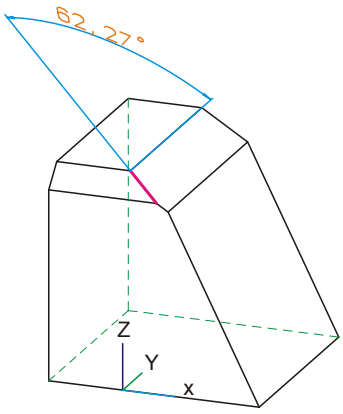


Fig. 113 3-D Angular dimension, line to Y-axis

42.2.4 Permanent Saving of Dimensioning Settings

The functions shown in the *closed* black rectangles enable you to change the settings for 3-D dimensions, parametric dimensions, HCM dimensions and Steel Engineering dimensions. These settings will then apply until you shut down HiCAD. If you want to save these settings permanently, additionally select the **Save parameters** function in the dashed rectangle. This function enables you to save your changes to the DIMENSIONING_SETTINGS.XML file. The settings will then be automatically preset when starting HiCAD.

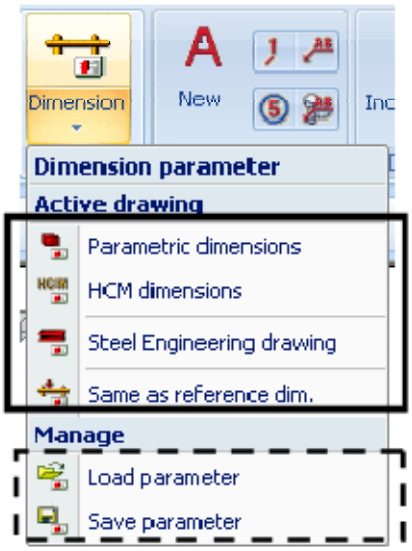


Fig. 114 3-D Dimensioning parameters

If you need to create commissioned drawings for different customers requiring different dimensioning settings, you can use the **Save parameters** to save the changes to files with different names (e.g. DIMENSIONING_SETTINGS1.XML, DIMENSIONING_SETTINGS2.XML etc.).

Before creating a drawing, you can then use the **Load parameters** function to select the dimensioning settings that you need.

Another option to permanently change dimensioning settings is provided by the **Settings** function (icon at the top right corner). Click the icon and select **Settings > Basic Settings > Dimensioning**.

Further detailed information on all dimensioning functions can be found in the Online Help.

Before dimensioning, please activate the part you want to dimension and a view of the sheet area.

42.3 Further Permanent Changes

The previous paragraph explained how to permanently save changes to dimensions. Now let us see how to change other settings permanently.

42.3.1 Permanent Changes from HiCAD

Many settings can be changed permanently via the **Settings > Settings** (function at the top right corner of the screen).

Fig. 115 Permanent changes

42.3.2 Permanent Settings in System Files

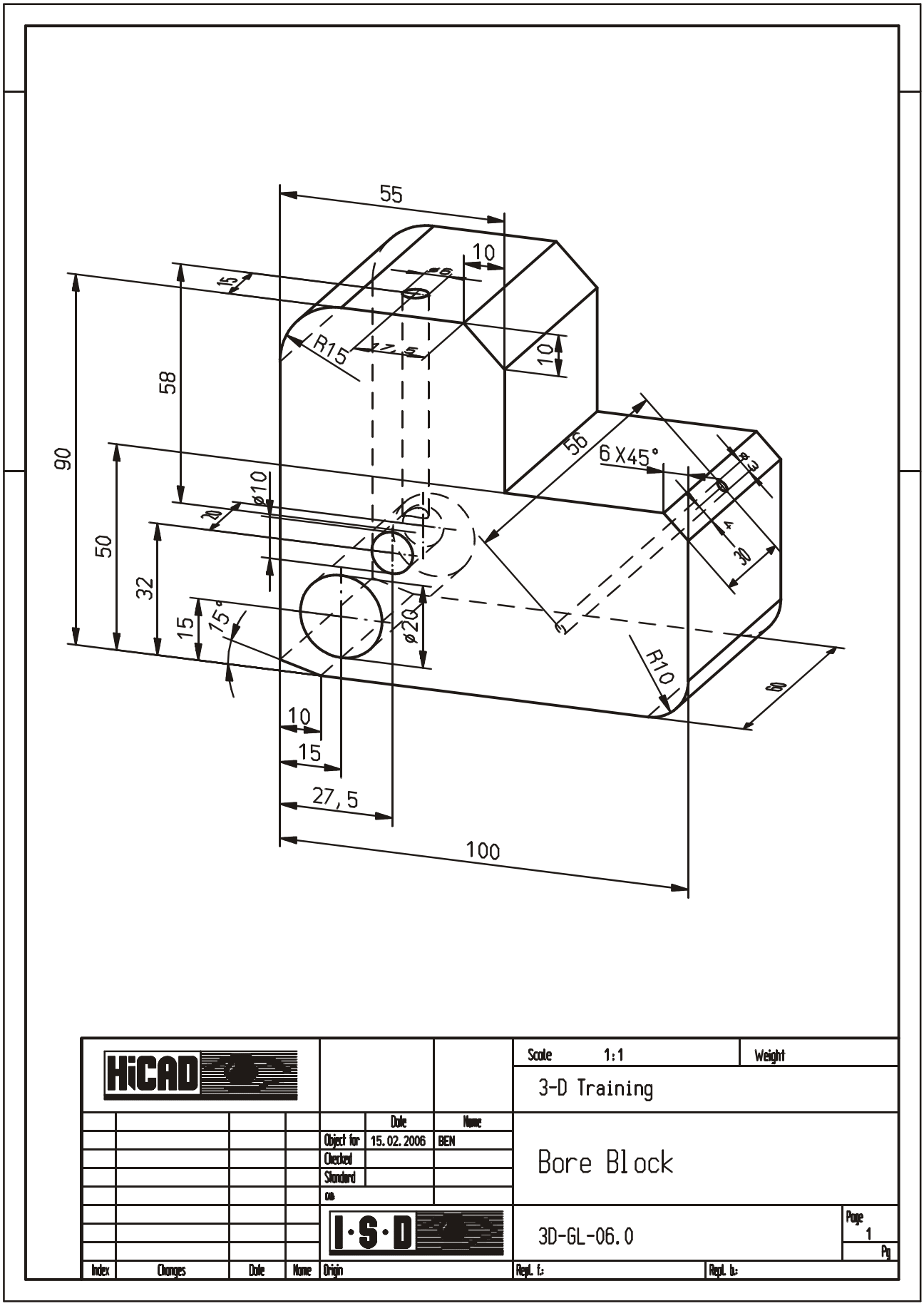
Most system files (with the exception of HICAD/EXE/FILEGRUP.DAT) can be found in the HICAD/SYS directory. Although the majority of the system files are only of interest for software developers, there are also some files which can also be changed by CAD users, e.g. the following files:

System file	Some exemplary applications
ALGPAR.DAT	Which point options do you want to be displayed when working on a 2-D or 3-D part? At what intervals (minutes) do you want HiCAD to automatically save your drawing?
FILEGRUP.DAT	Definition of the logical paths , e.g. of the default path “C:“, to which all drawings, parts, part libraries etc. are saved.

Changes applied in system files or via **Settings > Settings** are always permanent changes. This means that they cannot be revoked with any default function or a re-start of HiCAD. Therefore, please make sure that the changes you want to apply are actually useful and **keep a record of these changes** in case you want to revoke them.

42.4 Exercise 6 (3D-GL-06.0) Bore Block

- Learning target: Subtractions; 3-D dimensioning.
- Switch to the front view. Draw a sketch. As you are in front view, the processing plane is the XZ-plane. You can change into any other view while drawing the contour of the bore block. Draw the fillets and chamfers as well as the bore, but not the subtraction with a depth of 20mm. If you want to parameterise the sketch, it makes sense to add fillets or chamfers after creating the extruded solid. This procedure has the advantage that you can fully parameterise the drawing with a fairly moderate number of constraints. If the sketch already contains fillets, chamfers etc., very many constraints may be required until the sketch is fully parameterised. Create the bore block as an extruded solid from the sketch.
- Create the subtractions. A processing plane is defined by two lines. Please remember that the first line is the future X-axis and the second line the future Y-axis. You should therefore choose a horizontal line in the left half and then a vertical line in the lower half.
- Dimension the part. Activate the Sheet area (for more information see next exercise). Default dimensioning is in the active XY-plane. Use the right mouse button if you want to dimension in the XZ-plane.
- Space for notes: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Name
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Repl. b.:

43 Sheet Views and Model Views

The view structure of the current drawing is displayed on the **Views** tab of the Information + Communication Navigator (ICN), with HiCAD distinguishing between

- Model views and
- Sheet views.

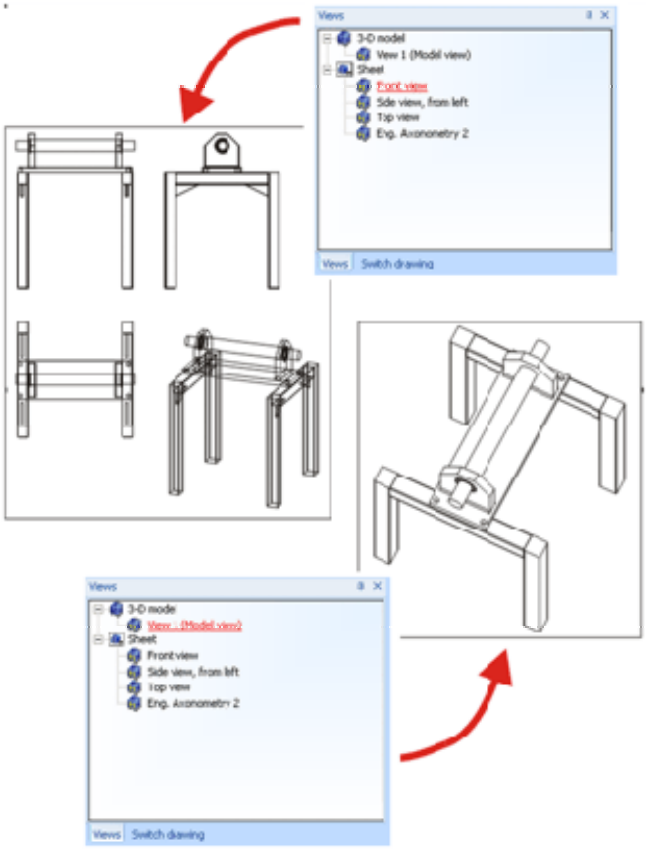


Fig. 116 Arbitrary switching between model views and sheet views

In the **Sheet** area you can create and name your production drawings. You can then switch to the **Model** area to process your 3-D model. The position of the views in the Sheet area will not be altered by adding views in the Model area. Changes of the geometry are automatically adjusted in the sheet area, thus enabling you to arbitrarily switch between model and production view.

To activate a view, move the mouse pointer on an item in the ICN or on the pink dotted rectangle around a drawing and press the left mouse button.

If you activate a view in the **Views** tab, the **Properties** tab of the lower ICN window provides information on the selected view. If you have selected the view with a single mouse-click, the information is displayed only temporarily, if you have double-clicked the view, the information is displayed permanently.

It is possible to create several sheet areas (right-click **Sheet**, then select **Sheet area, New**). This enables you to distinguish between proposal drawings, production drawings, approval drawings, customer drawings etc. by creating each of them in a separate sheet area. When processing a part in the model area (or in any of the sheet areas) the part is automatically adjusted in all sheet areas.

- When dimensioning a part, always activate the Sheet area (and, of course, also the part).
- Please note that before printing you need to select the **Sectional/Detail View > Update, All** function once. You will find this function if you right-click a view in the drawing area (pink dotted rectangle) and select **Process > Others**. The sectional views and detail views are then also updated.

44 Copy Dimensions to Other Views

In HiCAD you can create dimension-comprehensive dimensionings. Especially for complex drawings it makes sense to initially set the dimensionings in a particularly clearly structured and easy-to-understand view (e.g. a list view), and the copy these dimensioning to another view. Or you can create your dimensionings in the (also very clearly structured) axonometric view and copy them automatically to top, side or front view.

44.1 Show/Hide Dimensions

Activate the **3-D Dimensioning** and select **Tools > Table**. Click the arrow symbol below table and select the **Visualisation, Show dimensions/Hide dimensions** function.

Visualisation – Show Dimensions

First select which dimensions you want to be displayed:

 **Individually,** i.e. you identify individual dimensions.

 **Structure**, i.e. you select a structured dimension (chain/parallel/running dimension etc.)

 **Rectangle**, i.e. you select the dimensions with a drawn rectangle

 **All in part**, i.e. all parts of the active part are considered

 **All**, i.e. all parts of the active drawing are considered.

Show in ...
After specifying the dimensions to be shown you are prompted to specify the view in which the selected dimensions are to be shown:

 **Active view**
The selected dimensionings are only displayed in the active view.

 **Identified view**
Identify the view in which you want to display the dimensions.

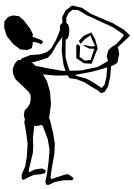
 **All views**
The dimensionings are displayed in all views.

 **All but active view**
The dimensionings are displayed in all views, except the active view.

 **All but identified view**
Identify the view in which the dimensioning are not to be displayed.

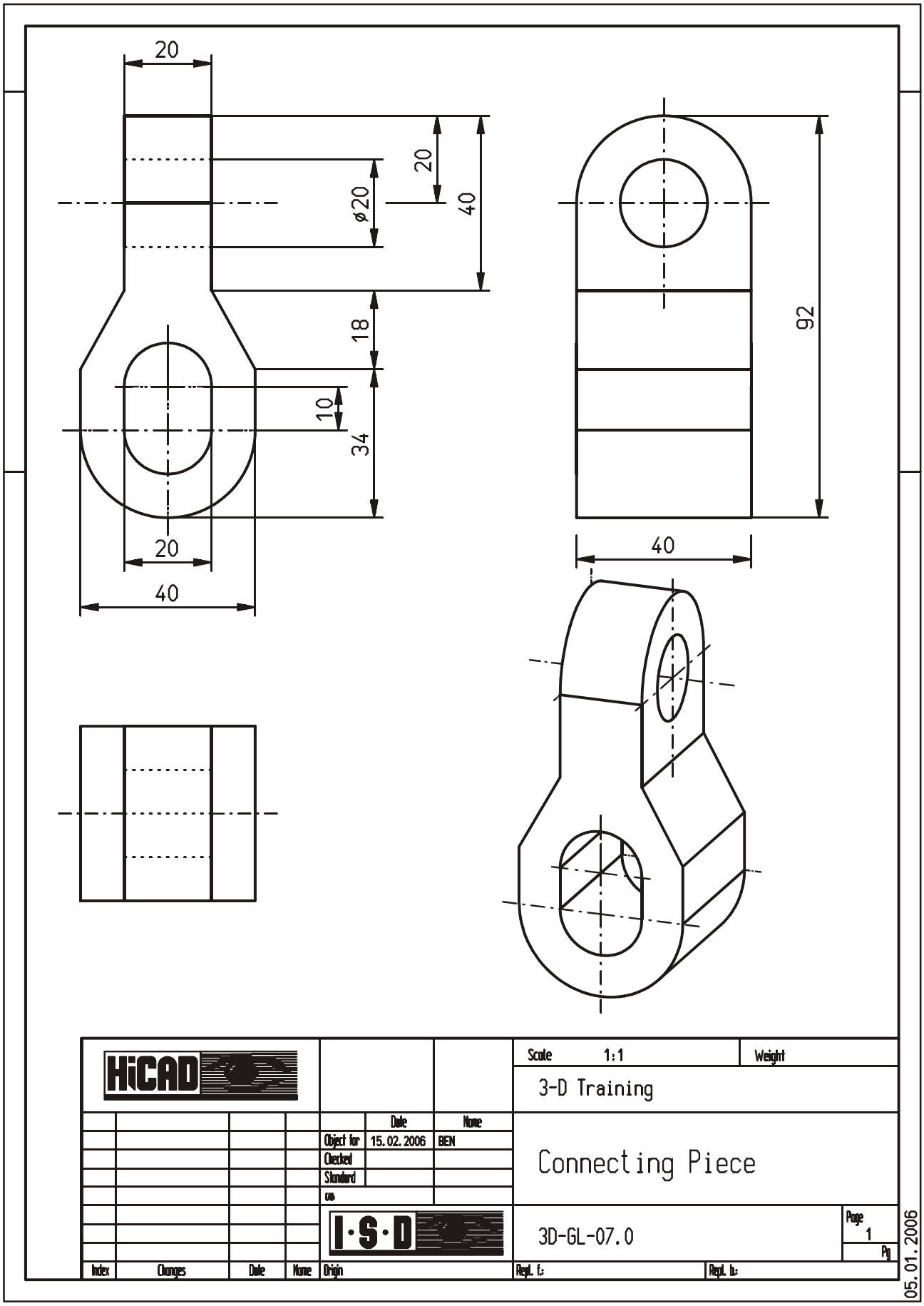
Visualisation – Hide Dimensions

Use this function to hide dimensions. The procedure is the same as for the **Visualisation, Show dimensions** function.



44.2 Exercise 7 (3D-GL-07.0) Connecting Piece

- Learning target: Extruded solid; standard processings; select views and not only parts and drawings with the right mouse button; transfer dimensions automatically to other views; model and sheet views.
- Switch to the front view. Draw the connecting piece as an extruded solid. As you are in front view, the processing plane is the XZ plane. You can change into any other view while drawing the contour of the connecting piece.
- Use the Standard Processings functions for the through hole and the slot.
- Call the “Views” tab in the ICN. Activate the Sheet area first! Create 4 views and insert the drawing frame.
- Select different visualisations for the four views. Activate the view with the right mouse button (context menu). In this case you do not need to activate the view first.
- Dimension the standard axonometry only. Transfer the dimensionings to the other three views automatically using the “Show dimensioning” function. Remove the dimensioning from the Sheet area and the standard axonometry with the “Hide dimensioning” function. When you select Hide you remove the dimensioning from the active view only.
- Activate a view in the Model area. Change the views there and execute an optional process (e.g. a chamfer). Then activate the Sheet area. You will see that the process has been taken over into all four views, but that the type and position of the views has been maintained.
- Hint: You are enabled to create and name production views in the Sheet area. Switch to the Model area to process and model the 3-D model. The advantage of the Model area is that all views, cuttings and drawing frames are hidden. Processing is thus faster and more efficient. You are also enabled to rotate the views of the part in the model without any repercussions on the production views in the Sheet area. In addition, the creation of *several* Sheet areas, e.g. for quotation drawings or production drawings, would be possible as well.
- Space for notes: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Object for 15.02.2006 BEN
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Standard
as
Connecting Piece
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Date
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Origin
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Repl. b.:
05.01.2006

45 Sectional Views

45.1 Sectional View

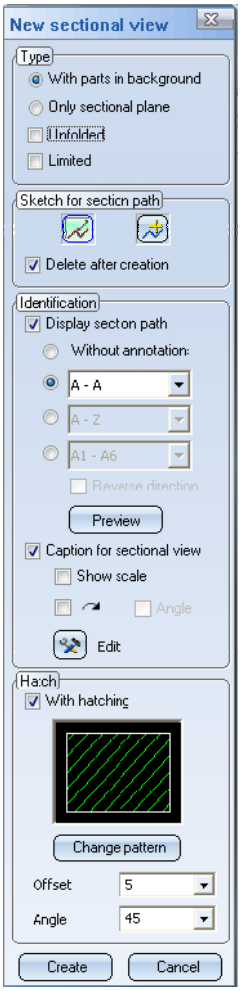


Fig. 117 Sectional view

The **New sectional view** function can be found in the **New** function group of the **Views** tab. You use this function to create sectional views, either with or without identification of the section path, while the parts themselves remain unchanged. You define the section path by a sketch that either already exists in the drawing or is newly created when defining the sectional view.

Besides creating “normal” sectional views you can also create “unfolded” sectional views, i.e. all sectional views are unfolded into a common plane. This makes sense if, for example, you want to represent the internal sides of revolved solids, pipes etc. Non-unfolded sectional views can also be laterally limited, so that the intersection body will not entirely penetrate the drawing.

The following combinations are possible:

⦿ With parts in background ○ Only section plane ☐ Unfolded	The view is displayed as a solid.
○ With parts in background ⦿ Only section plane ☐ Unfolded	A sectional view is created in which only the section surface is displayed.
⦿ With parts in background ○ Only section plane ☑ Unfolded	All section planes are expanded into a common plane. The view is displayed as a solid.
○ With parts in background ⦿ Only section plane ☑ Unfolded	All section planes are expanded into a common plane, but only the section surface is displayed.

In addition, you can restrict non-expanded sectional views laterally.

To do this, activate the **Limited** ☑ option.

Please note:

- In contrast to virtually all other HiCAD functions, we draw the section path as an open polyline, and not as a closed polyline.
- Before creating a section you should activate the sheet views. In this way the clear structure of the model area is retained.
- Sectional views are not suitable as processing and modelling views.
- For sectional views, only parts are considered whose sectional view ID is set to "Cut, In sectional view". To access this function, right-click a part and select **Properties > Cut…**.
- If you want to assign a material with a corresponding hatching to a part before creating the sectional view, HiCAD will use this hatching for sectional and detail views. If you change the hatching at a later time, you need to update the sectional/detail view.

45.2 Examples

The left image shows the original view with the sketch for the section path. The right image shows different variants of the sectional view (here: not unfolded).

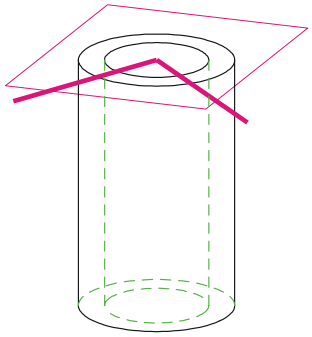


Fig. 118 Original situation

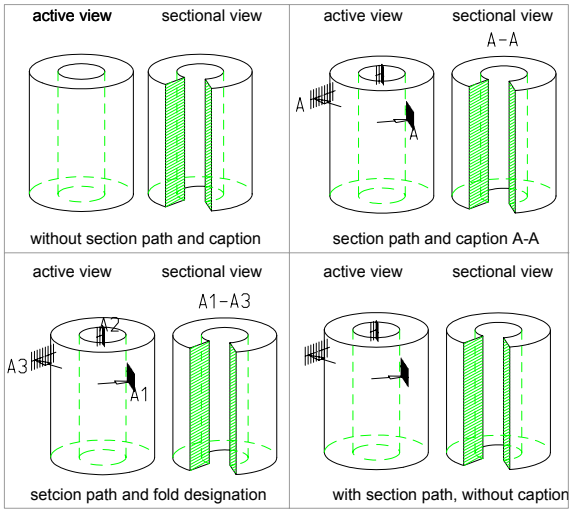


Fig. 119 Sectional views (not unfolded)

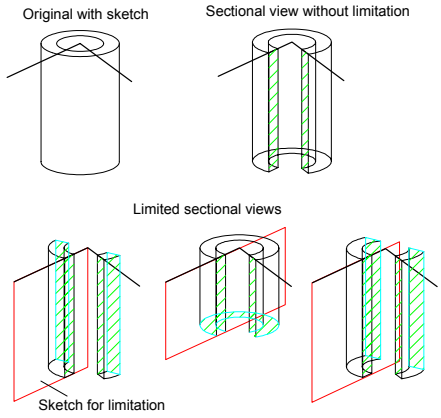


Fig. 120 Limited sectional views

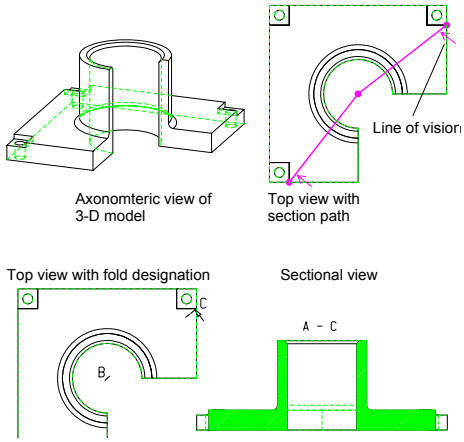


Fig. 121 Unfolded sectional view

- Further information on sectional views is provided in the Online Help. Besides sectional views, there are also detail views and cut-outs. The detail views are explained in the "**Create Detail View**" chapter of this training book.
- Before printing you need to select the "Sectional/Detail view > Update, All" function. In contrast to standard views, sectional and detail views are not updated automatically.

46 Assign 2-D Parts to 3-D Views

3-D drawings can also contain 2-D parts. These can, for example, be a bill of materials, a tolerance table or a part containing processing notes as 2-D text etc.

If you do not want to see this part in the model area, you can assign the part in the middle ICN window (2D-Part structure) to the sheet area. Please note that in middle window you can only see the views of the area which is active in the upper ICN window. This means that if a view of the sheet area is active in the upper ICN window (see next image), you will see the part “2-D sheet” (and not the part “3-D model”) in the middle ICN window (2D-Part structure tab) as well.

Example: You have created a 2-D part “Processing notes” that you want to see in the sheet area only. Proceed as follows:

- In the top window of the ICN, activate a view of the *Sheet* area.

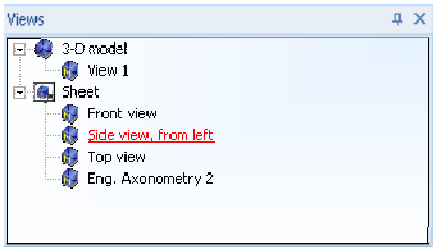


Fig. 122 Active view in SHEET area

- Move the cursor into the drawing area to enable the display in the ICN to be updated!
- In the 2D-Part structure tab, the part “2-D sheet” is displayed. Move the part “Processing notes” below this sub-part. The result is shown below:

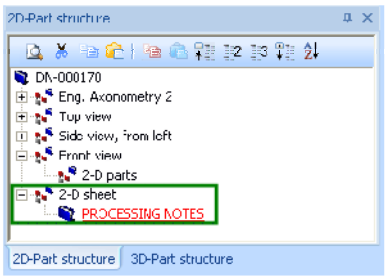


Fig. 123 2-D part “Processing notes” below the part “2-D sheet”

In the same way, you can assign other 2-D parts to the views of the Sheet area.

For many parts such as fits tables, dimensioning tables, coordinate tables etc. HiCAD automatically ensures that these parts are only visible in the Sheet area (and not in the Model area).

Please note: If you do not want a 2-D part to be visible in all views, but only in one particular view, e.g. in Eng. Axonometry 2, you can also move the 2-D part under “2-D part” below “Eng. Axonometry 2”. The result looks as follows:

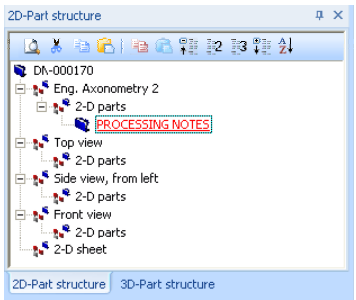


Fig. 124 The 2-D part “Processing notes” is moved below “Eng. Axonometry 2”.

47 Referencing

Referencing parts is the most efficient method for re-using parts. Referenced parts can be used and processed company-wide and project-comprehensive. The 3-D body that represents this part is not closely integrated into the model, but only added when the model is created in the current state.

Referencing of parts has the following advantages:

- Frequently required company-specific parts can be managed and changed much more effectively from a constructive perspective if they are referenced. The major advantage of this technique is that parts or assemblies can be processed and updated by a team.
- If a drawing contains referenced parts, the system checks automatically to see whether they exist in changed form. Changed parts are automatically updated, if required. Even when a drawing is saved, only the parts that have been changed are saved again. This ensures high speed even for complex drawings.
- If the HELiOS database is being used, referenced parts are blocked for other users while they are being processed. Only when the part has been released again are automatic update and other changes possible.
- Application possibilities for referenced parts include changing and managing company-specific standard parts as well as construction unit systems.

47.1 Reference 3-D Part

Use the **Reference part, Save, Detail drawing** function to reference 3-D parts. Select the settings shown below:

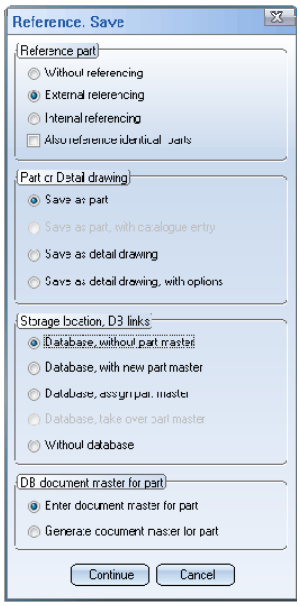


Fig. 125 Reference 3-D part

After confirming with **Continue** you complete the document master data mask for the part. HiCAD saves the part to the database and converts it to a referenced part. Of course, you can also reference a part when creating it.

47.2 Break Up Referencing

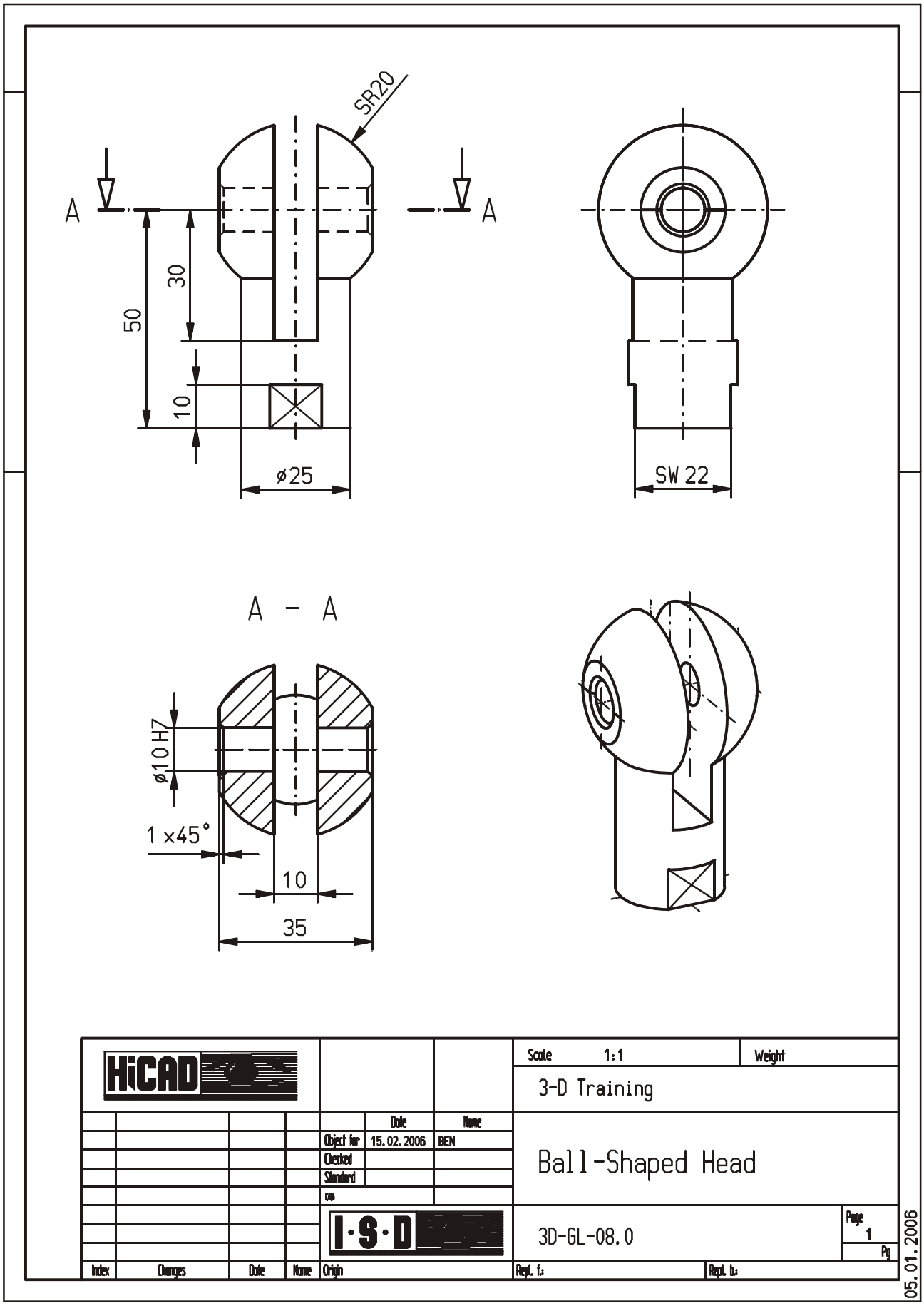
On the **Drawing** tab, select **Drawing/Reference > Others > Break up ref., individual** to break up the referencing of the part (i.e. if the part is meant to be a custom part).

See Online Help

Extensive information on referencing is provided in the Online Help.

47.3 Exercise 8 (3D-GL-08.0) Ball-Shaped Head

- Learning target: Referencing, sectional views, allocate 2-D parts to a 3-D view.
- Draw a cylinder and reference the part.
- Specify 3 points to define a processing plane in the MIDDLE of the cylinder. Make sure that the first 2 points define the future X-direction and the third point the future y-direction of the coordinate system.
- Create the ball-shaped head with the “Add sweep, via *Rotation*“ function.
- Create the 10mm gap as well as the two flattenings at the sides with “Subtract part, via Translation” in the front view.
- Create the width across flat in the side view with “Subtract part, via Translation”.
- Use the standard subtraction function and not the Standard Processings for the 10mm bore. In this case we do not need to delete the centre cross of the bore when adding the centre cross of the ball-shaped head.
- Add all necessary crosshairs and centre crosses at the ball-shaped head.
- Assign a Material to the part and create the sectional view (activate Sheet area first) and translate it if necessary.
- Hint: If you assign a material to the part before creating the sectional view, HiCAD will use this material hatching for the sectional view (as well as for the detail views). If you assign the material to the part *afterwards*, you are enabled to update the sectional and detail views.
- If you want to omit the “Gland follower” exercise to save time: Create (also in the Sheet area) a detail view and a cut-out view (not shown in the exercise on the right page).
- Add the drawing frame to the Sheet area as well.
- Copy the ball-shaped head into another drawing. Change the part in one of the two drawings. Then switch to the other drawing to see how the ball-shaped head has, due to the referencing, changed in the other drawing as well.
- Create a 2-D part and enter into this part some notes for processing of the ball-shaped head. Then translate the 2-D part in such a way that the text will only be visible if a *Sheet area* and not a Model area will be active. Please note: The part “2-D Sheet” is only displayed in the middle window of the ICN if (in the upper window of the ICN) a view of the Sheet area and not the Model area is active. Please check by activating the individual views that the processing notes are only shown in the views of the Sheet area.
- Hint: In the same way you can make sure that a fits table, a Bill of Materials or any other 2-D part is only visible in the Sheet area, but not in the Model area.
- Space for notes: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Standard
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Ball-Shaped Head
I·S·D
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Page 1
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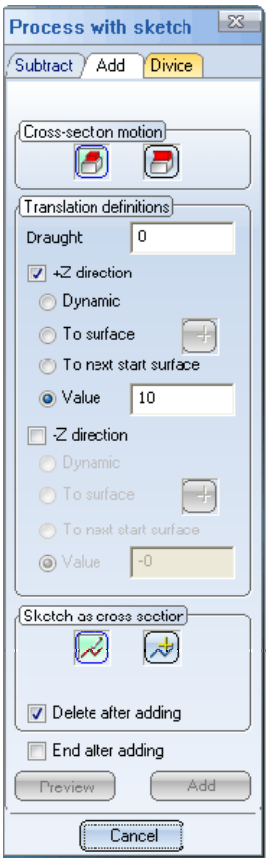
Index
Changes
Date
Name
Origin
Repl. f.:
Repl. b.:
05.01.2006

48 Add Sweeps / Standard Processings

48.1 Add Sweep, via Translation



Use this function to attach an extruded solid (sweep) to the active part, i.e. no second part is created that we would have to merge with Boolean operations.

The extruded solid is created with the help of a sketch. This sketch can either already exist in the drawing or needs to be created.

Fig. 126 The input window for the **Add sweep, with translation** function

Proceed as follows:

- Select the **Add sweep, via translation** function.
- If you want to create the part with a **Draught** (inclination), enter the inclination angle. The position of the draught is influenced by the mathematical sign (positive or negative value).
- Specify the settings for the extrusion, which is done in the same way as for the previously described **Subtract part, via translation** function.
- If you want to use an existing sketch, click the **Select sketch** button and identify the sketch.

  To create a new sketch, select the **Create new sketch** button. HiCAD switches to the **Sketch** tab, enabling you to create the required sketch.
- Click the **Preview** button to display the result. If you activated the **Value** option instead of the **Dynamic** option, select **Preview** or **Add**, left-click into the drawing area, drag the cursor to the required position and left-click again (RET). The height specified by the cursor position is displayed for your information. You can correct this value if desired.
- Click **Apply** to add the part.

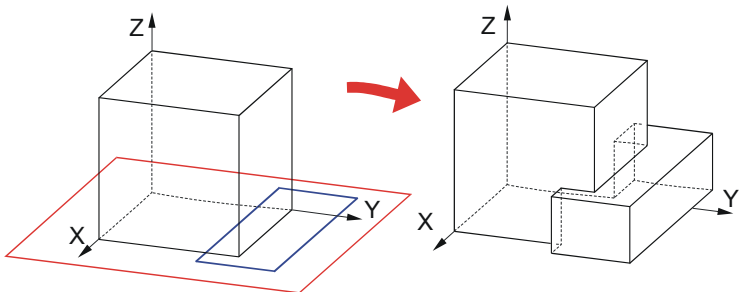


Fig. 127 Add sweep, via translation

Proceed likewise for the **Add sweep, via rotation** function.

48.2 Standard Processings

The **Standard Processings** function group of the **3-D Standard** tab offers many different types of standard processings, e.g. Through holes, Slots and **Blind holes.**

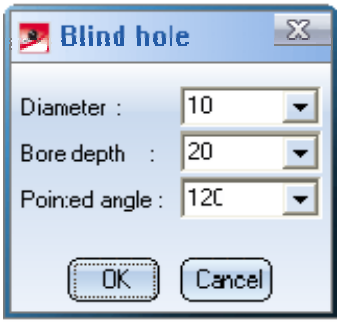
48.2.1 Variable Blind Hole

Use this function to insert blind holes into parts.

The current standard settings are considered. If required, the **Settings** window is displayed.

Identify the surface to which you want to apply the bore, by selecting 2 edges. Please note the hints given at the bottom of this page.

Enter diameter, bore depth and point angle.



Enter the point where you want the blind hole to begin. The point option R(RET) is frequently used for this.

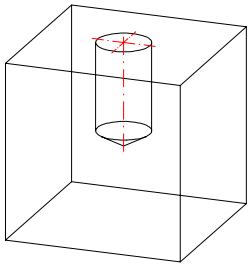
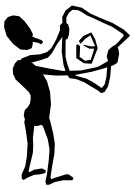


Fig. 128 Blind hole

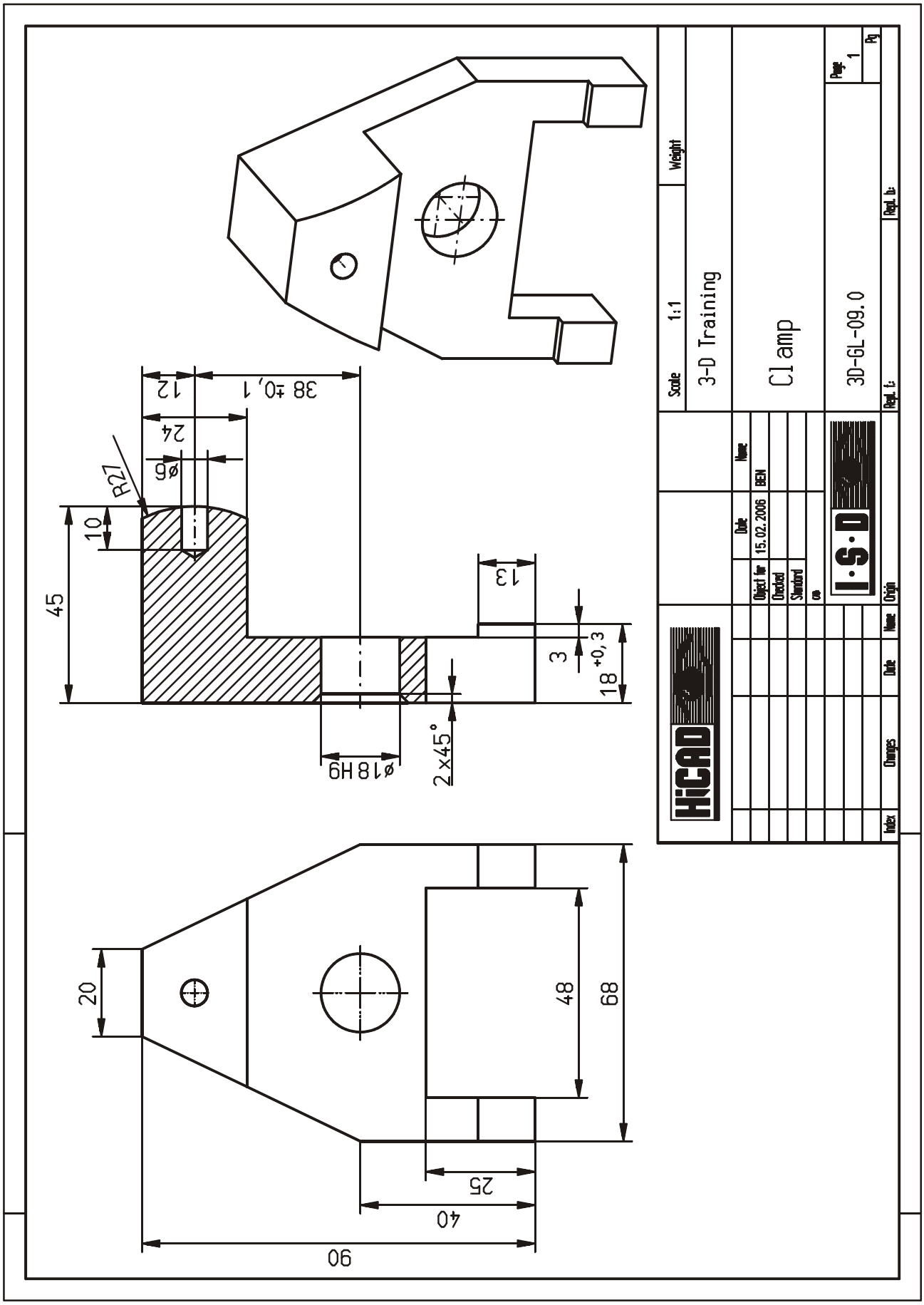
This function, too, requires the selection of a surface. Please note that when selecting the first line, you also set the direction of the future X-axis, and when selecting the second line, you set the direction of the future Y-axis. Therefore first select (view direction: vertical at surface) a horizontal line in the left half, then a vertical line in the lower half. Alternative: Select the bottom left corner to select a surface. Please note that the cursor needs to be placed in the vicinity of the future X-axis.



48.3 Exercise 9 (3D-GL-09.0) Clamp

- Omit this exercise if there is not enough time.
- Learning target: Add; subtract.
- Change to the front view and draw the 15 mm thick part of the clamp as an extruded solid.
- Use the “Add sweep, via translation” function for the feet with a depth of 3 mm and the head with a depth of 30 mm,
- Use the Standard Processings for both bores.
- Create the R27 fillet with “Subtract part, via Translation” in the side view (draw closed contour with R27 circular arc).
- Create the sectional view. Activate the Sheet area first!
- When dimensioning, do not only activate the part, but also the view you want to dimension. Activate a view in the Sheet area, in order that the view of the Model area will remain free from dimensionings!

  Hint: If you activate the drawing in the view in which you want to dimension the part, you automatically also activate the view as well.
- Space for notes:


Scale 1:1
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Clamp
3D-6L-09.0
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10
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13
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18 +0,3
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2x45°
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68
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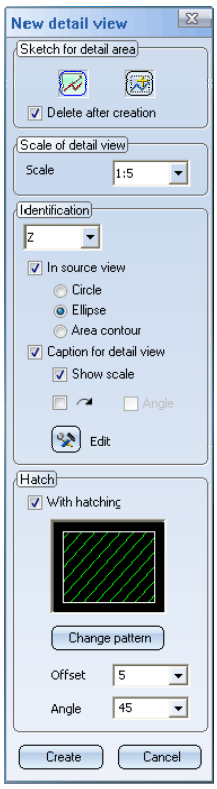
90

49 Views

49.1 Create Detail View



The **New detail view** function can be found in the **New** function group of the **Views** tab. Use this function to display enlarged details of your drawing. The detail is created with the help of a sketch. This sketch can either already exist in the drawing or needs to be created.

Proceed as follows:

- Activate a view of the sheet area.
- Select the **New detail view function**.
- Select **Create new sketch** if you want to draw a new contour for the detail.
- Draw a closed contour.
- An enlarging scale, an ID letter, the marking of the detail location and the hatching type are already suggested by HiCAD. You can change these values if required.
- After selecting the **Create** function you can position the detail view in your drawing.

If you right-click the marking of the detail location (e.g. ellipse), you are enabled to change all parameters of the detail view, and also the path of the detail view contour at any time.

Fig. 129 Detail view

Detail views are not suitable as processing and modelling views. A detailed description of the **New detail view** function is provided in the Online Help.

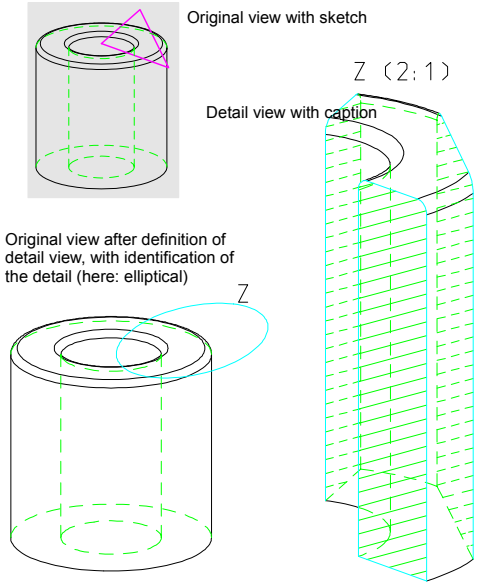


Fig. 130 Example of a detail view

49.2 Shorten View

The **Shortened view** function can be found in the **Process** function group (left of Transform) of the **Views** tab. You can also access this function by right-clicking the view (pink dotted frame) in the drawing area and selecting the appropriate function from the context menu. The latter way is the better one, as it automatically also activates the view. In the **Shortened view** dialogue, click the **Add** button and specify the start and end point of the axis, and set the division points of the shortening(s). Click **Create** to end the function.

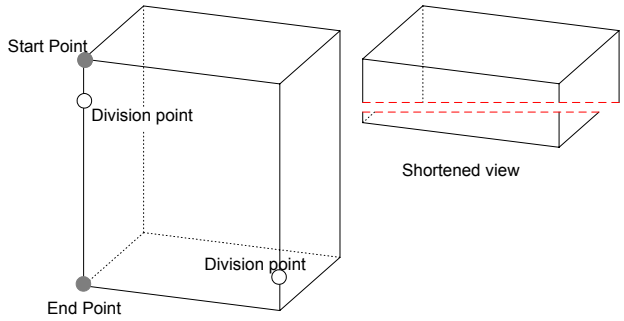


Fig. 131 Definition of a shortened view

Please note that all view representations beginning with “Quick“, as well as all shaded representations are time-saving approximation procedures, which are, for instance, unsuitable for the representation shortened views. Use “Hidden Line” instead of “Quick Hidden Line” if you want to create a shortened view.

49.2.1 Delete Shortened View

The **Delete shortened view** can also be found in the **Process** function group. It enables you to revoke the shortening.

49.3 Surface in Screen Plane

In the context menu of the view you will find the **Surface in screen plane** function. Use this function to rotate arbitrarily located planar surfaces into the screen plane. You will then look at this surface in vertical direction. You have now the following options to identify the surface:

 Identification of 2 edges  Specification of 3 points

In the figure shown below, a blind hole is inserted into the surface F1:

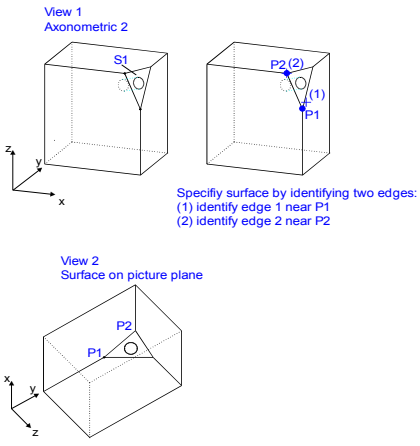
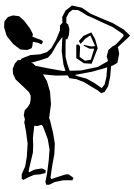
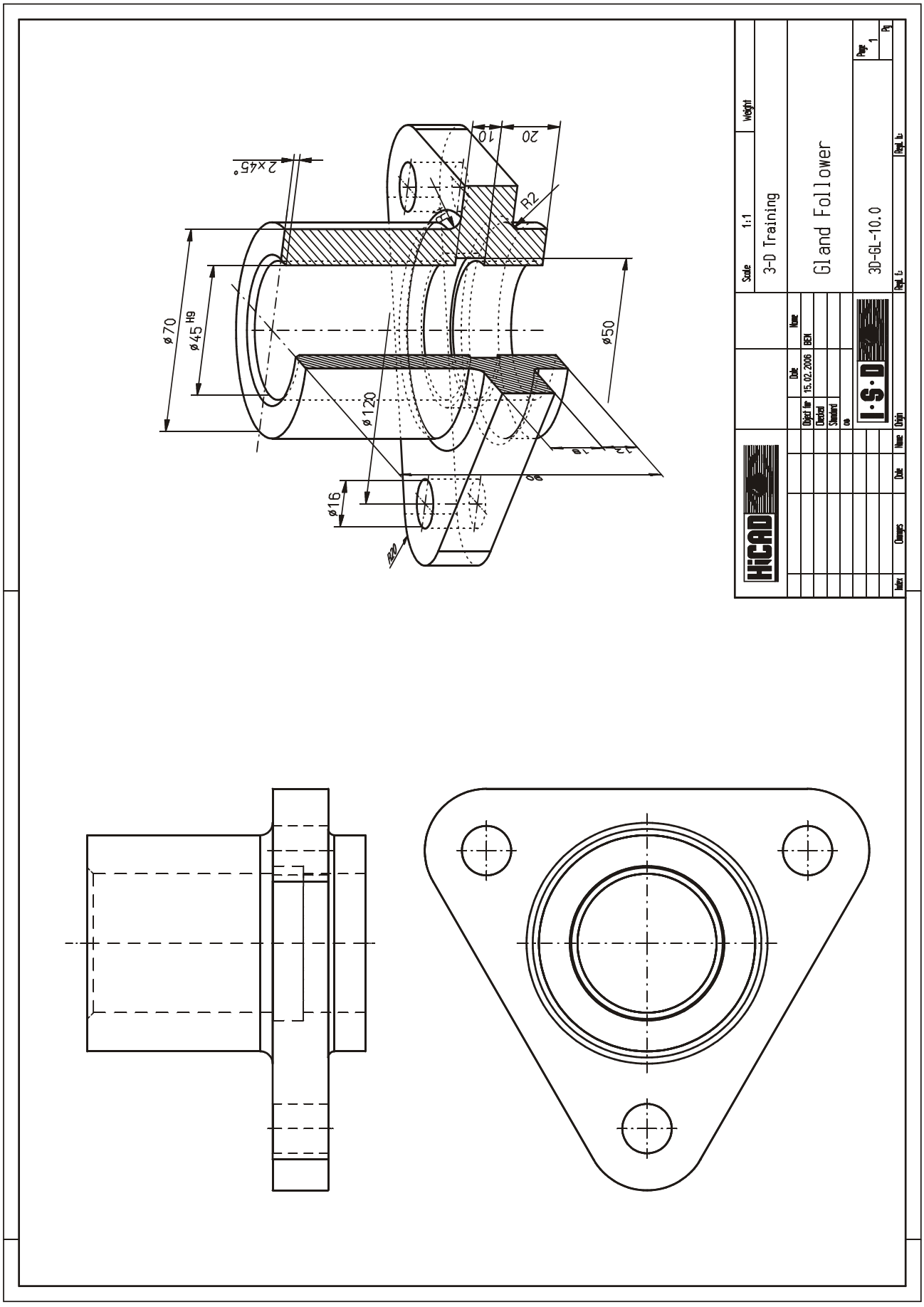


Fig. 132 Surface in screen plane



49.4 Exercise 10 (3D-GL-10.0) Gland Follower

- Learning target: Chamfer; fillet; axis representation; detail view; clip plane (shaded); shorten view; accuracy of representation of filleted surfaces.
- Create the 18mm thick plate of the gland follower as an extruded solid.
- Draw the Ø70 cylinder with “Add sweep, via translation” or (simpler) as a hollow cylinder. Merge the hollow cylinder with the extruded solid.
- The groove can be created with the “Subtract part, via Translation” or “Subtract part, via Rotation” function.
- Add the standard processings, chamfers, fillets etc.
- Axis representations can be found in the “Tools” function group of the “3-D Standard” tab.
- Create the sectional view in the Sheet area!
- Translate the views by right-clicking them and selecting the appropriate function (automatically activates the view you want to translate).
- Activate one of the views in the Sheet area. Create detail views (not given in the drawing).
- Familiarise yourself with the functions “Clip plane (shaded)“, “Shorten View” and “Surface on screen plane”.
- Tip: The accuracy of the display of the filleted surface can be set via “Drawing” / “Properties” / “Others” / “Surface approximation”.
- Space for notes:


2×45°
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ø45 H9
ø120
ø50
ø16
Scale 1:1
Weight
3-D Training
Gland Follower
3D-GL-10.0
Page 1
HiCAD
I·S·D

50 Shafts, Shaft Processings and Cut-Outs

50.1 Shaft Processing

Activate the **3-D Standard** tab and select **Standard Processings > Others > Feather keyways.** The feather keyway is one of the many shaft processings offered by HiCAD.

Set a processing plane on the shaft shoulder:

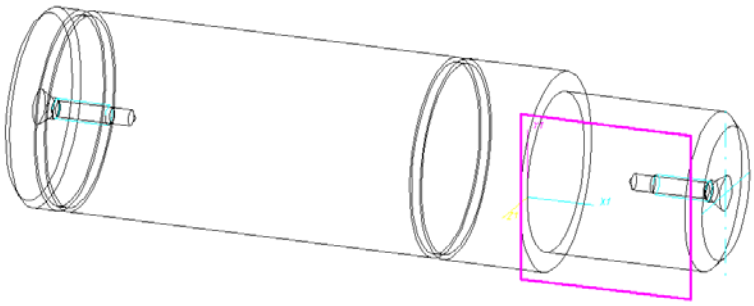


Fig. 133 Processing plane on barrel surface of shaft shoulder

Select the **Feather keyways** function.

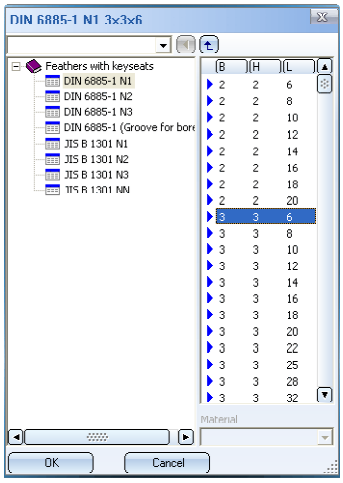


Fig. 134 Feather keyway DIN 6885-1 N1

Use the Relative-Return option to insert the feather keyway, then move a distance equivalent to half of the feather keyway width to the right (the fitting point is located in the centre of the feather keyway).

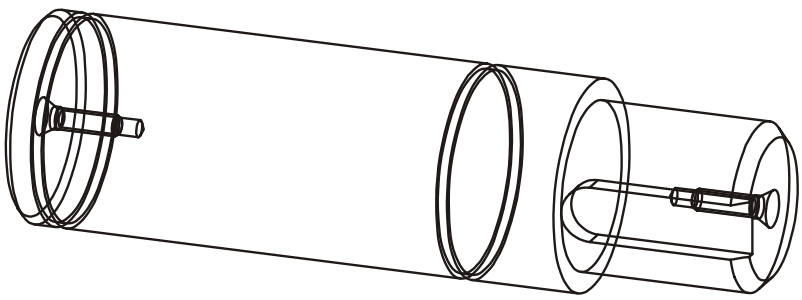
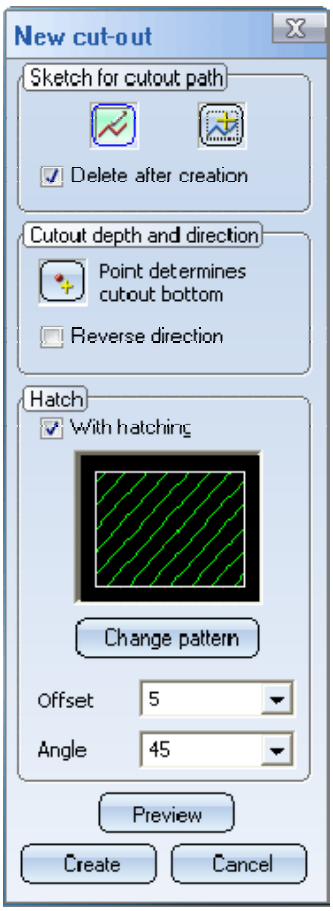


Fig. 135 Shaft with feather keyway

50.2 Cut-outs

Select the **Create new cut-out** function in the **Process** function group (left of View functions) on the **Views** tab.



Proceed as follows:

- Identify or draw (as for the sectional views) the sketch for the cut-out contour. Please make sure that the sketch is closed.
- Specify the point for the cut-out bottom. Click **Select point** and determine the point in the drawing.
- If you want to hatch the section planes, activate the **With hatching** checkbox and select the hatching data.
- Check the cut-out with the **Preview** button. If required, you can switch the direction of the cut-out by activating/deactivating the **Reverse direction** checkbox.
- Click **Create** to generate the cut-out with the current settings.

Fig. 136 Cut-out

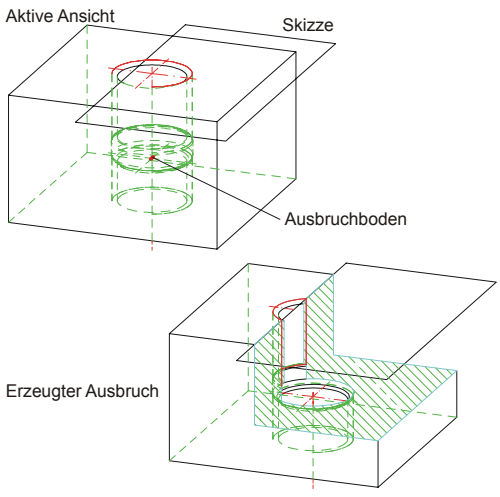


Fig. 137 Create cut-out

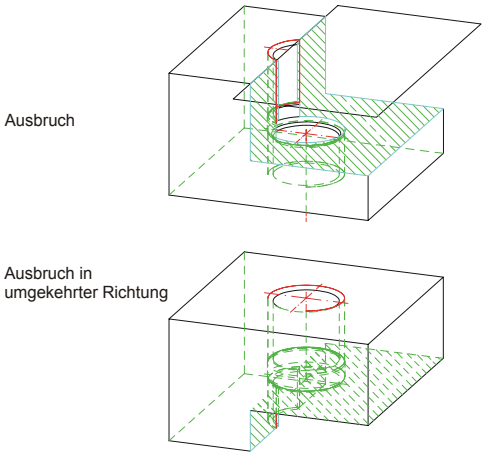
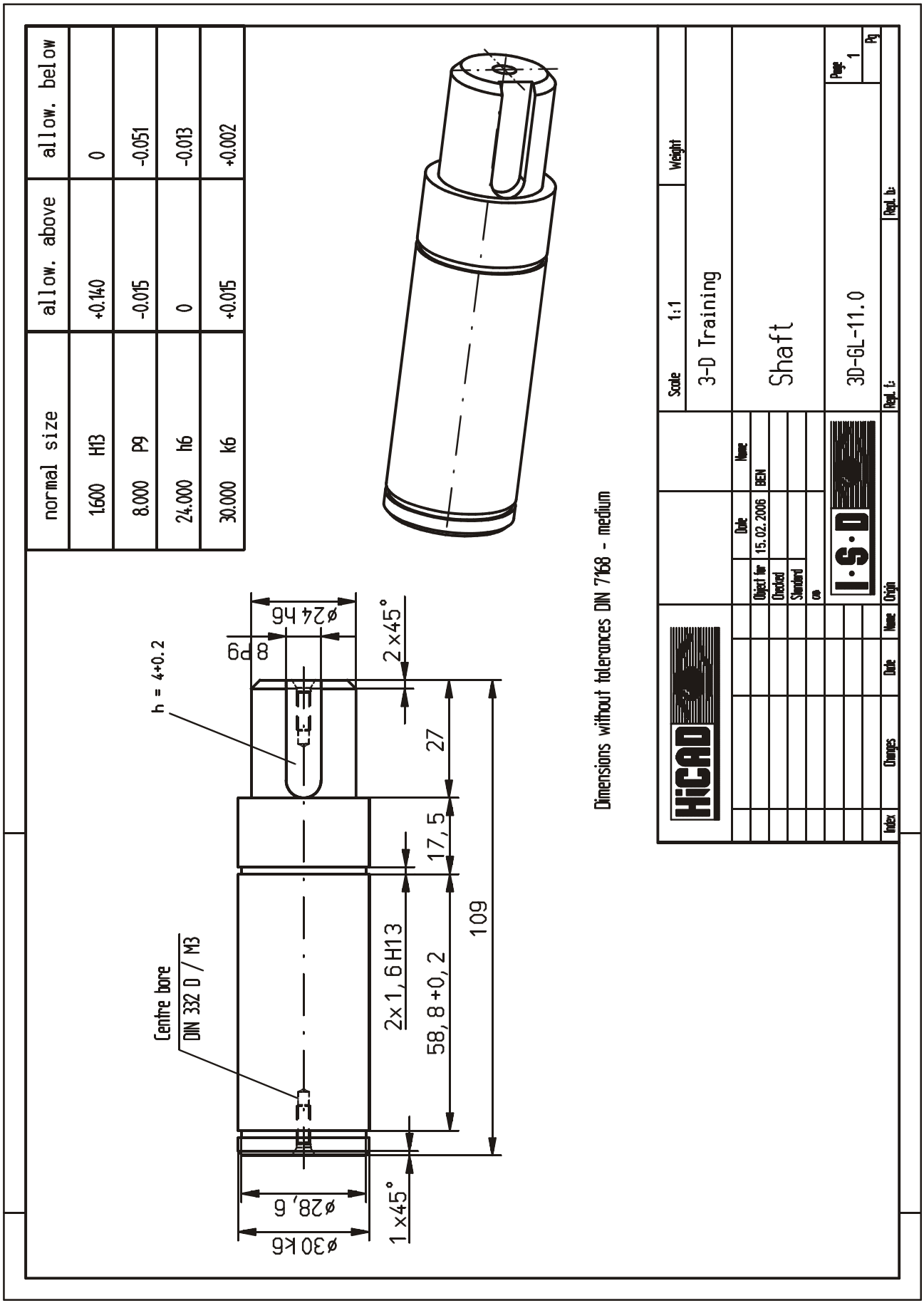


Fig. 138 Reverse cut-out direction

50.3 Exercise 11 (3D-GL-11.0) Shaft

- Learning target: Revolved solid; shaft standard parts; subsequent processing options in a sketch.
- Change to the front view and draw the shaft as a revolved solid.
- Use the centre bores and feather keyways.
- Activate, in the Sheet area, the view you want to dimension.
- Insert (in the Sheet area) a tolerance table and a drawing frame.
- Create a cut-out that makes the centre bore visible (not shown in the drawing).
- Get familiar with the options to alter a sketch subsequently: Call the “Process Sketch” function in the feature of the revolved solid. Change the revolved contour arbitrarily in the sketch. Execute another modification using the 3-D HiCAD Constraint Manager.
- Tip: We drew the shaft as a revolved solid and not as an addition of two cylinders. This provides us with more modification options via “Process Sketch” in the Feature, as well as more parameterisation options. Furthermore, we are enabled to see all creation and processing feature steps at one glance.
- Space for notes:


normal size
allow. above
allow. below
1.600 H13
+0.140
0
8.000 P9
-0.015
-0.051
24.000 h6
0
-0.013
30.000 k6
+0.015
+0.002
Dimensions without tolerances DIN 7168 - medium
Scale 1:1
Weight
3-D Training
Shaft
3D-GL-11.0
HiCAD
I·S·D
Name
Date
BEN
15.02.2006
Object for
Checked
Standard
as
Page 1
Pg
Repl. to:
Repl. by:
Origin
Name
Date
Changes
Index
ø24 h6
8 P9
2×45°
h = 4+0.2
27
17,5
109
2x 1,6 H13
58,8 +0,2
Centre bore
DIN 332 D / M3
ø28,6
ø30 K6
1×45°

51 Re-Use

Re-use speeds up the design process considerably. In almost every company there are parts which are largely similar, which would be drawn only once. When you want to use the part again, you will simply load it from the first drawing or a different location, such as a catalogue or a part library, or a database.

The loading of parts from another drawing (e.g. via the clipboard) is always possible. The loading from the database, a catalogue or a part library requires a previous saving of the part (or assembly) to these locations.

51.1 HiCAD Clipboard

You can use, for instance, the HiCAD clipboard to copy parts from one drawing to another:

In the ICN, activate the part, the assembly or the part list (multiple selection with CTRL and SHIFT, as known from Windows applications) that you want to copy from the current drawing to another drawing.

Right-click the part(s) and select the **Copy to clipboard** function.

You can then paste the current content of the Clipboard to other drawings. In the ICN of the target drawing, right-click the assembly to which you want to assign the content of the clipboard as sub-parts, and select the **Paste from clipboard** function. Select a fitting point on the part and specify the position of the fitting point in the drawing.

If you want to paste the content of the clipboard as *main part* into the drawing, select in the middle ICN window the name of the drawing. Please make sure that you selected a 3-D part in the 3D-Part structure tab (in case of a 2-D part: 2D-Part structure tab). Then choose the **Paste from clipboard.** function.

51.2 Load Part from Drawing

The functions described in the previous paragraph **Clipboard** require that you have loaded the drawing from you want to re-use any part(s). Now let us take a look at a function for which this will not be necessary:

Activate the **Drawing** tab and select **Insert Part > Others > From drawing, Via document master** (if you do not have a database, select **From drawing, Via Explorer**). This function enables you to copy a part from a drawing of your choice to the active drawing.

First select, via the document master data, the drawing containing the part that you want to copy. Then identify the required part in the drawing. Place the part in the active drawing by selecting a fitting point on the part and specifying its position in the drawing.

You can also access the above function by right-clicking the drawing area.

51.3 Save Part to Database (or Explorer)

Re-used parts can be saved to the database.

Activate the **Drawing** tab and select **Drawing/Reference > Reference part, Save, Detail drawing**.

We start from the assumption that we require neither referencing not part masters. Select the settings shown below:

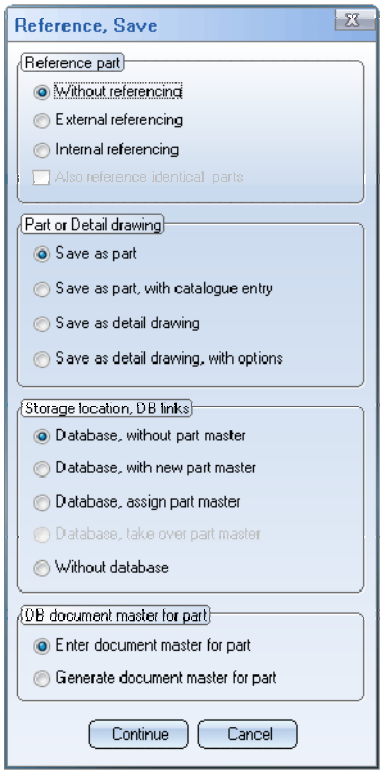


Fig. 139 Save parts for re-use to database

Enter the document master data of the part and confirm with OK.

If you do not have a database, you can also save re-used parts to the Explorer. In the above dialogue you would need to select “Without database” instead of “Database, without part master”.

51.4 Load Part from Database

In the **Insert part** function group of the **Drawing** tab, select the **Insert part, via DB document master** function. This function enables you to load a part from the part database into the active drawing. Select, via the document master data, the part that you want to load (if required, remove suggested links). As for drawings, you can set your own filters, sort according to column headers etc. Then place the part in your drawing by specifying a fitting point on the part and specifying its position in the drawing.

If you do not have a database, you probably have saved your parts for re-use to the Explorer. You can then re-load these parts with the **Insert part, via Explorer** function.

Parts can only be directly loaded as parts from the database, if they were previously saved to the database. The saving of parts to the database is explained in Chapter 51.3 “Save Part to Database (or Explorer)” .

51.5 Insert From Catalogue

Use the **Insert main part, via Standard Part catalogue** in the **Insert Part** function group of the **Drawing** tab to load parts from a catalogue. Select the required catalogue, then select the required table. Click the table and select a part. Confirm with **OK** and position the part in the current drawing, by selecting a fitting point on the part and specifying its position in the drawing.

If you want to load a part for re-use from a catalogue, this requires that you have previously saved this part to the catalogue. For this purpose you can use the program CATTEDITOR.EXE to create your own tables (and also catalogues if required). Please note that user-defined tables and catalogues can only be created in the catalogue “Factory standards > Parts and Processings”. As HiCAD provides a vector graphic as preview, parts selected for a preview graphic must not be displayed as a surface-representing approximation (i.e. no shaded or Quick* representation). For further information on how to save parts to a catalogue please see the relevant chapter in the Online Help.

See Online Help

51.6 User-Defined Part Library (User Library)

HiCAD allows the combination of frequently used parts into part libraries. The parts contained in that library can then be quickly and conveniently fitted via a pop-up menu.

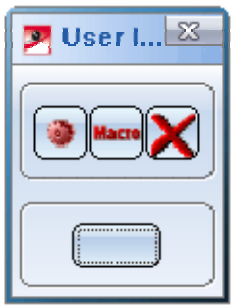


Fig. 140 The **User Library** pop-up menu

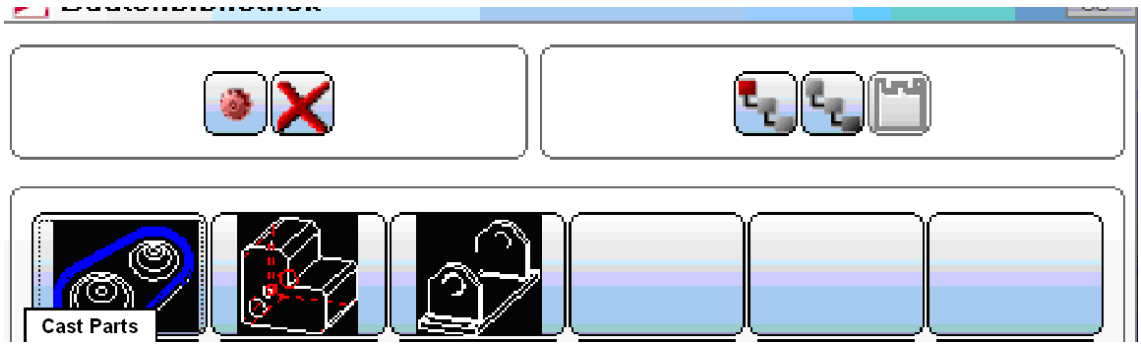


Fig. 141 Example of a part library

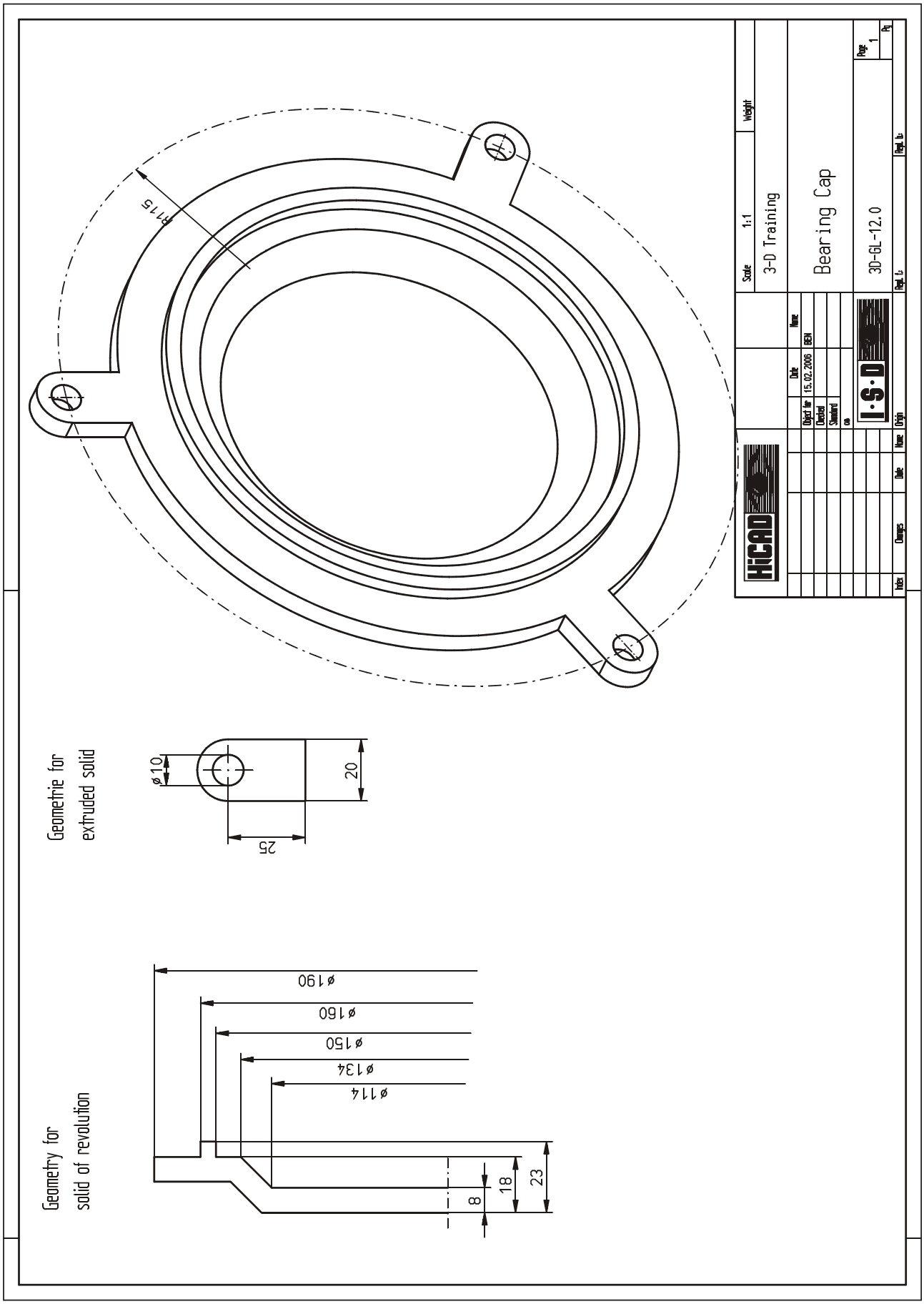
This function for the creation of user libraries is optionally available as an extension module. Existing user libraries can however also be loaded without a licence for this module.

51.7 All Re-Use Options at a Glance

Function	Application
1) HiCAD Clipboard	Copies one or several parts into the current drawing or into a different drawing.
2) Insert Part > Others > From drawing, Via document master	Loads parts from any drawing. In contrast to the clipboard function, this option requires that the source drawing (i.e. the drawing from which you want to copy the part) must be loaded in HiCAD.
3) Insert Part > Insert part, via DB document master	Enables direct access to the part you want to load. In contrast to the “From drawing, Via document master” function, this option does not require that you know the name of the drawing in which the desired part is located. The “Insert part, via DB document master” function allows only the loading of parts which have been previously saved as parts with the “Reference, Save, Detail drawing” function.
4) Insert Part > Insert main part, via Standard Part catalogue	Loads a part from a catalogue. This part needs to have been saved with the function “Reference part, Save, Detail drawing > Save as part, with catalogue entry”. Please note that catalogues provide numerous service functions (arrange parts into main series, subseries, lock parts, and much more).
5) Insert Part > User Library	In addition to the simple insertion option, you are provided with preview images, help texts and thematical subdivisions into various part libraries.

51.8 Exercise 12 (3D-GL-12.0) Bearing Cap

- Tip: Omit this exercise if there is not enough time. (you can practise the re-use functions in one of the following exercises).
- Learning target: Re-use of parts; Revolved solid; extruded solid; merge parts.
- Change to the side view and draw the bearing cap as a revolved solid.
- Create the flange as an extruded solid and create the other two flanges by cloning the part. Merge bearing cap and flanges.
- You can also access the crosshairs for hole patterns via the “Tools” function group of the “3-D Standard” tab.
- For re-use of the part please proceed as follows:
  - Load the bearing cap with the HiCAD clipboard into a new drawing.
  - Load the bearing cap into a new drawing using the function “Insert Part” / “Others” / “From drawing, with document master”.
  - Ask your trainer to show you how to create catalogues and tables. Save the bearing cap in a catalogue you created, then reload the part from the catalogue.
  - Save the bearing cap as a part in the database, then reload it as a part.
  - Create a parts library file “bearing cap” and enter the bearing cap you have just created into this file.
- Space for notes: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solid of revolution
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ø160
ø150
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3-D Training
Bearing Cap
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Date
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52 Align Part to X-, Y- and Z-Axis (Or to Another Part)

The **Move, via fitting points** function enables you to move (i.e. translate and rotate) the active part. You can access this function by activating the **3-D Standard** tab and selecting **Transform > Others**. You can also right-click the part and select this function from the context menu. You can use this function to align an obliquely positioned 3-D part parallel to the X-, Y, or Z-axis. You can also align a 3-D part parallel to another, arbitrarily positioned 3-D part.

- Enter 3 fitting point pairs in succession.
- The fitting points on the part are transformed in the direction of the corresponding fitting points of the drawing. The part retains its original dimensions.

Example: You want to align an obliquely positioned cuboid to a second cuboid, which has in turn already been aligned parallel to the X-, Y, and Z-axis (reference cuboid). The two cuboids can have different sizes. Select the **Move, via fitting points** function:

1st fitting point pair: Specify an arbitrary point on the obliquely positioned cuboid, e.g. the front, top, left point. Then select the corresponding comparable point on the reference cuboid, i.e. also the front, top, left point.

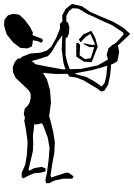
2nd fitting point pair: Specify another point on the obliquely positioned cuboid, e.g. the front, top, right point. Then select the corresponding comparable point on the reference cuboid, i.e. also the front, top, right point.

3rd fitting point pair: Specify a third point on the obliquely positioned cuboid. This point must be located in the same plane as the first two points and must not be located on the same axis as these points. Apart from these coordinates, it can be freely selected. You can, for example, select the back, top, or left point as third point on the obliquely positioned cuboid. Then select the corresponding comparable point on the reference cuboid, i.e. also the back, top, or left point.

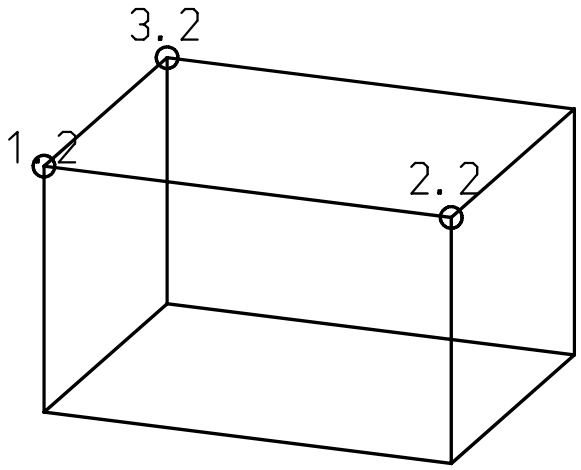
On the next page you will find an example showing you how to align an obliquely positioned cylinder to a cuboid which has already been aligned to the X-, Y- and Z-axis. Point 2.1. means: 2nd fitting point pair, 1st point. Point options “Centre” and (twice) “Quad point” were used for selection of the points on the cylinder.

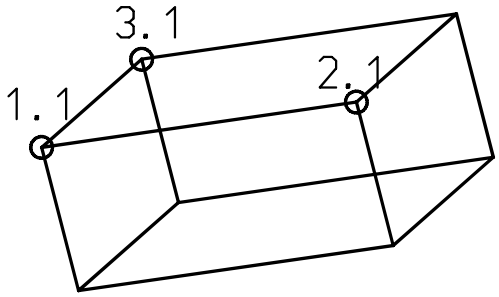
Tip: Let us assume that you have an obliquely positioned 3-D part, and that you want to align a second part parallel to this first part. This means that the first part would be our reference part. In this case, too, proceed as described above.

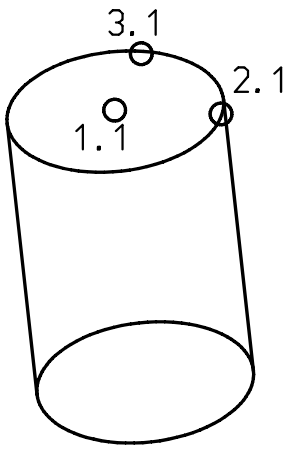
52.1 Exercise 12.1 (3D-GL-12.1) Align Parts to Coordinate Axes or to Other Part



- Tip: If there is not much time, you trainer should only show you this exercise.
- Learning target: Align parts parallel to X, Y-, or Z-axis (or to another part).
- Draw a cuboid or a cylinder and rotate these parts about one or several angles, so that the parts are obliquely positioned in space.
- Draw a reference cuboid which is positioned parallel to the X-, Y-, and Z-axis. This cuboid does not need to have the same size as the cuboid from the previous point.
- Align the obliquely positioned cuboid and cylinder to the reference cuboid. Use the **Move, via fitting points** function for this.
- Tip: In the same way, we could have aligned a 3-D part to another, arbitrarily positioned 3-D workpiece, with this workpiece being the reference part.







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25. 03. 2010

53 3-D Guidelines

3-D guidelines are used to create pipes with the **3-D Sketch** function, to describe motion paths, or – in the **Motion simulation** extension module – to perform translations along composite edges.

53.1 Create 3-D Sketch

In the **New** function group of the **3-D Standard** tab, you will find the **New dummy part** function. Use this function to create a new 3-D dummy part. You can also activate the Funktion **3-D Sketch** function in the **New** function group of the **Sketch** tab. In the **Line** fnction group of the **Sketch** tab, you will find the **New polyline** function, which enables you to draw 3-D polylines. To end the creation of the polyline, press the middle mouse button.

53.2 Process C-Edges

The **Process c-edges** function can be found in the **Others** function group of the **Sketch** tab. It You can use this function to trim lines (i.e. shorten them to the next intersection point). If you click the arrow on the icon, a pop-up menu with many processing functions, including the **Re-group** function:

53.2.1 Regroup

Use this function to break up and regroup composite edges. This enables you to automatically connect composite edges with identical end points.

You normally change the orientation of c-edges with this function. Furthermore, composite edges will be interrupted at “Y” points, i.e. at points where more than two edges intersect.

Very frequently you use this function to combine optically closed (but in fact still open) composite edges into polylines which are then actually closed in the internal data structure.

Example: You have drawn a polyline with 5 lines. You have however selected the **New polyline** function several times. This means that you have in fact created several (poly)lines, but not one closed polyline. If you apply the **Regroup** function, you obtain, also in the internal data structure, a closed polyline. If you now use the polyline for the creation of a pipe, you will obtain one continuous, uninterrupted pipe.

You can specifically connect two composite edges with the **Connect** function.

53.3 Fillet/Chamfer Composite Edge

The **Fillet** and **Chamfer** functions on the **Sketch** tab enable you to fillet or chamfer individual corners or entire polylines.

Fig. 142 Fillet/chamfer corners

54 Pipe Creation

54.1 C-Edge Sweep

Fig. 143 C-edge sweep

Use the **C-edge sweep** function to set cross-sections on 3-D guidelines. You will find this function in the **New** functiongroup of the **3-D Standard** tab.

The guideline can be a sketch, a 3-D composite edge (“3-D sketch”) or a hyperedge. Curved guidelines are also possible.

The cross-sections need to be sketches. If you activate the **Circular cross-section** option, you will require no cross-section.

Procees as follows:

- Activate the **Hollow** checkbox if you want to create a hollow body. In this case you can additionally activate the **To outside** checkbox to determine that the wall thickness is applied to the outside (instead of the inside, as is usually the case) of the body.
- In the **Type** area, select whether you want to set a circular or an arbitrary cross-section: Activate the corresponding option. For circular cross-sections, enter the diameter and identify the required guideline with the **Select guideline** function. If you have activated the **Arbitrary cross-section** option, identify the guideline and select the sketch for the cross-section. Use the **Select sketch** function for this. You are enabled to specify the direction in the process. Before you finally set the cross-section, you can position (i.e. translate, rotate, mirror) it as required.
- Activate/deactivate the **Reference** and **Feature** checkboxes.
- Enter the part name.
- If you first want to see a preview of the created part, click the **Preview** button, then click **Apply**. If you want to create the part without preview, click the **Create** button.

The following paragraph provides explanations of two other setting options:

- **Position sketch automatically**

  If you selected this option, the cross-section is placed into the plane vertical to the first identified guideline edge, so that the fitting point of the cross-section is located in the start point of this edge. If required, click the **Position sketch** button to translate, rotate or mirror the sketch.

- **With direction specification**

  Use this function to specify the direction for free cross-sections. If this option is active, the otherwise greyed out **Specify direction vector** icon is activated for selection. Identify the guideline and specify the direction by identifying an edge or selecting two points. The further procedure is identical to the procedure without direction specification.

This function can, for example, be useful if you want to create a stair stringer along a spiral. Normally, HiCAD would drag the cross-section along the guideline in such a way that the cross-section would always be positioned *perpendicular to the guideline*. For the stair stringer, in contrast, you would want the stringer cross-section to be *vertical at any point of the guideline*. To achieve this, activate the **With direction specification** option and use the **Specify direction vector** option to select an arbitrary vertical line, as shown in the figure below:

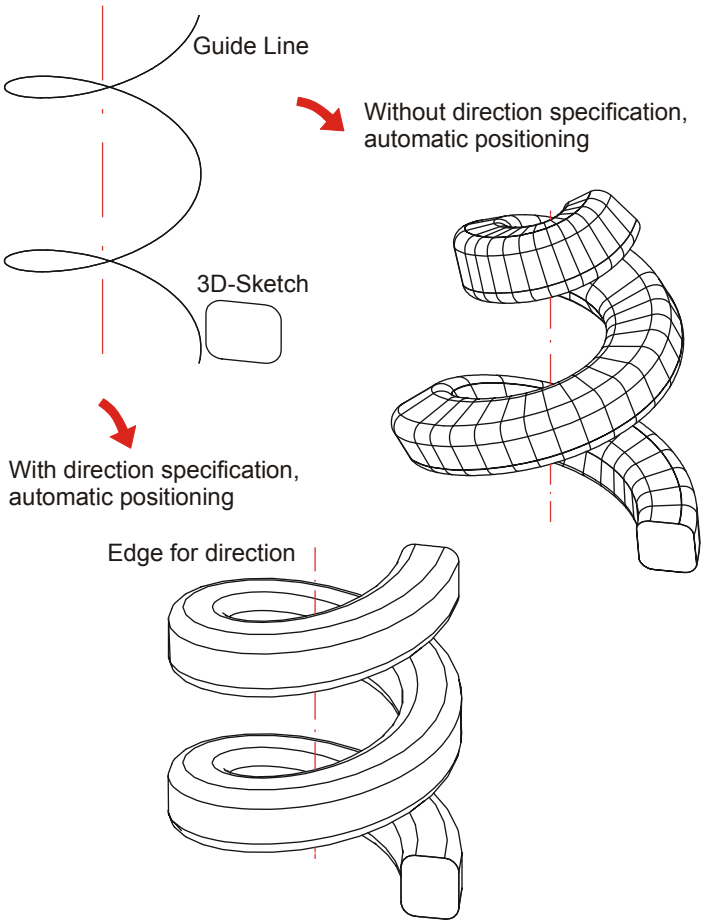


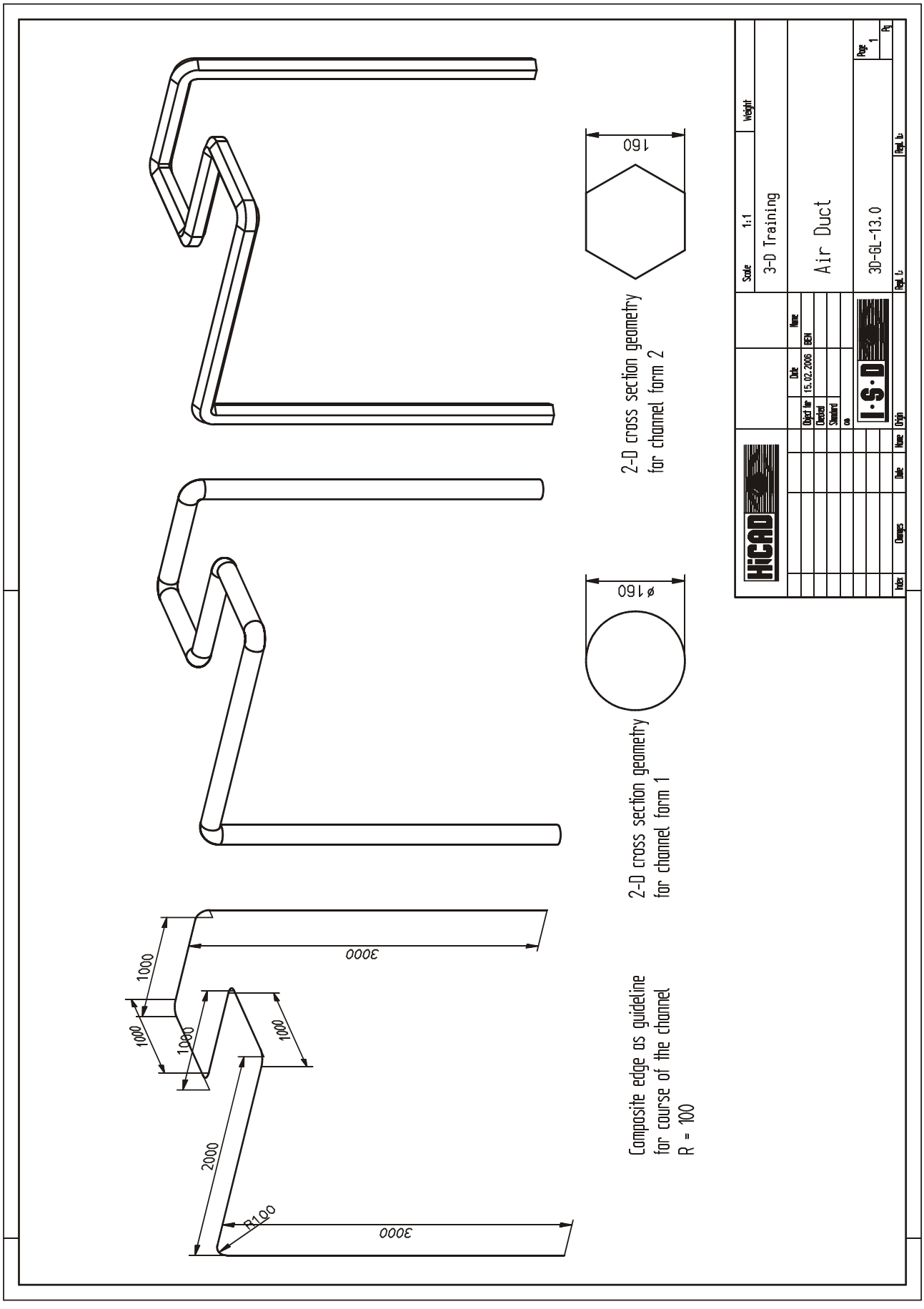
Fig. 144 Spiral with c-edge sweep

You can use up to 3 guidelines. Further information is provided in the Online Help.

This page is left blank to enable the next exercise to appear on a double page.

54.2 Exercise 13 (3D-GL-13.0) Air Duct

- Learning target: 3-D graphic elements; c-edge sweep .
- Create a 3-D dummy part or 3-D Sketch and draw the given 3-D guideline including the radius 100 fillet. Note: The “Regroup” function has the same effect as the “Sort GE” function in the 2-D module and in Sketches: Graphic elements with distance < tolerance are combined into a polyline
- Create the first pipe as a c-edge sweep with a circular cross section.
- Draw the cross-section of the hexagonal pipe in a sketch (not 3-D sketch). Create the second pipe as c-edge sweep with arbitrary cross section.
- Hint: The “Sketch” function creates a planar, i.e. 2-dimensiononal sketch. Such sketches are needed for functions such as “Extruded solid”, Revolved solid”, “Add”, “Subtract” etc. The 3-D Sketch (or 3-D dummy part) enables you to draw in 3-D. It can be used for guidelines which do not simply lie in a plane, but run in all 3 directions of the coordinate system. Typically, you would use the “C-edge sweep” function to create a solid from the guideline (e.g. a pipe, a staircase stringer or similar).
- Space for notes: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2-D cross section geometry
for channel form 2
ø160
2-D cross section geometry
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Composite edge as guideline
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55 Assemblies and Main Assembly

A 3-D drawing can already be defined as assembly drawing and entered into the database during its creation. For assembly drawings, HiCAD automatically creates a **3-D Main assembly**. All subsequently created 3-D parts are subordinated to this main assembly! This structuring provides maximum efficiency, in particular for referencing, itemisation, BOM creation, or a combined working with the PDM system HELiOS.

Main assemblies play an important role for many HiCAD automatisms. In case of itemisation or identical part search in Steel Engineering, for instance, the entire product structure of the main assembly is automatically transferred to the HELiOS product structure (if you use the PDM system HELiOS).

In the ICN, main assemblies are identified with the symbol. Each drawing can only have one main assembly! A drawing can however contain a large number of “normal” assemblies, as shown in the figure below:

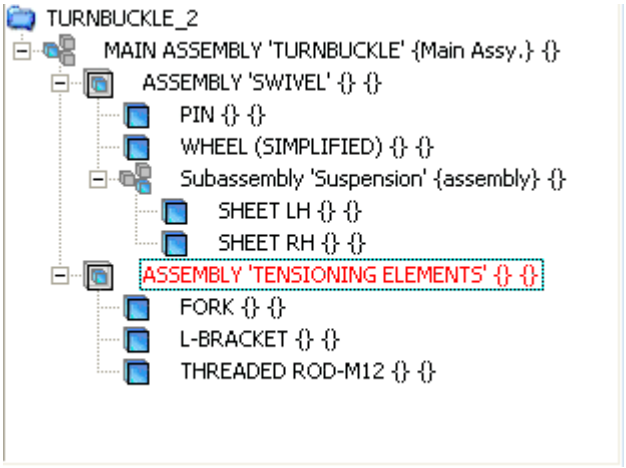


Fig. 145 Example of a part structure of an assembly drawing

55.1 Create Main Assembly

In the **New** function group of the **3-D-Standard** tab you will find the **Main assembly** function. This function enables to create the top part of all drawings which are no detail drawings. You can create only one main assembly for each drawing. In most cases, a main assembly consists of several (sub-)assemblies. The next chapter describes how to create such assemblies.

55.2 Create Assembly

In the **New** function group of the **3-D-Standard** tab you will find the **Assembly** function. Select this function and enter the name of the assembly. In contrast to the main assembly, you can create for your drawing as many assemblies as you want. Of course, you can also use this function to create sub-assemblies, e.g. by entering a name such as *Sub-assembly “Suspension”* in the “Article number” (or “Part name”) field. The figure shown above provides an example of the subdivision into assemblies and sub-assemblies.

Please note that the part name is only displayed in the ICN if the part has no article number. As soon as you enter an article number when creating a main assembly, it will have priority over the part name and will be displayed (instead of the part name) in the ICN.

56 Model Set

At the top right of the screen, you will find the **Settings** icon. Click the icon and select the **Toolbars** function. In the dialogue, activate the **Toolbars** tab, then activate the **3-D Model set** checkbox.

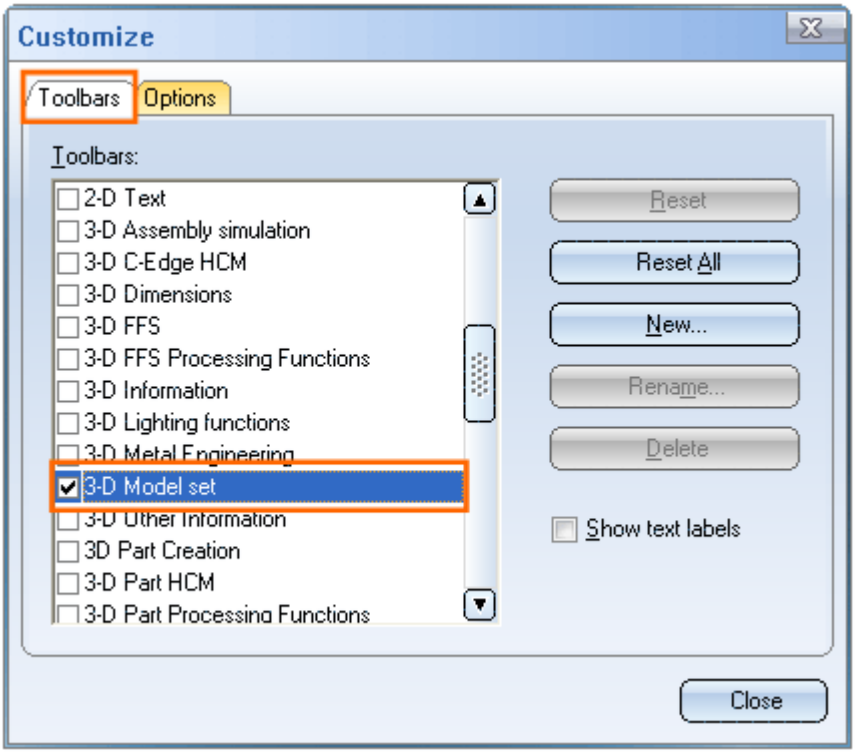


Fig. 146 Activate 3-D Model set

The **3-D Model set** toolbar is displayed at the bottom of the screen.

Fig. 147 The Model set toolbar

You can use the functions of the **3-D Model set** toolbar to drag parts from large assembly drawings and process them in your own drawing. This method provides a very fast and well-structured working, even when dealing with very large drawings. A requirement for this is a free window (thumbnail) in the **Switch drawing** tab.

You can define and process up to 18 model sets, which may also have a hierarchical structure, while the modelling browser provides a clear overview of the model sets.

The working with model sets is meant to facilitate the processing of parts in complex drawings. For this purpose,

- the parts of the drawing that you want to process are collected, i.e. combined into a model set,
- copied to a new drawing,
- processed in the new drawing, and
- retransferred to the original drawing.

56.1 Procedure

Remove all parts from all sets

Use this function to remove the parts from all selected sets, i.e. all old model sets that might exist are deleted.

Add part

Use this function to select the parts that you want to take over to the active model set, i.e. you collect a new set of parts to be taken over into a new, empty drawing.

Separate model set

If you have defined the model set, you can transfer it temporarily to a new, initially empty drawing and process it there. This method is much faster and more convenient than applying the changes in the original main drawing. The new drawing contains only the parts in the model set which can now be modified with all 3-D functions. In this context, please note however the exception at the bottom of the page (highlighted in grey).

Retransfer active set

After completion of the processing, you can retransfer the modified parts to the original drawing by selecting the **Retransfer active set** function. Before the parts are inserted, HiCAD will ask you whether you really want to apply the changes to the original drawing.

Please do not rename the parts in the model set and do not change the part structure. Otherwise loss of data may occur. You are however allowed to add parts.

57 List View

HiCAD allows you to define part lists and then to display only those parts contained in these lists in the view. Such “list views” are views that contain only a few parts instead of all parts. You can, for instance, create a view consisting of only one long beam, which is to be displayed in shortened representation in the list view. In a list view, you can also to display only a few selected parts with a larger scale (e.g.. 5:1 instead of 1:1).

To access the functions for the creation of list views, activate the **Views** tab and select **New > Others > List view**. You can also right-click a view in the drawing area (pink dotted frame) and select the function from the context menu.

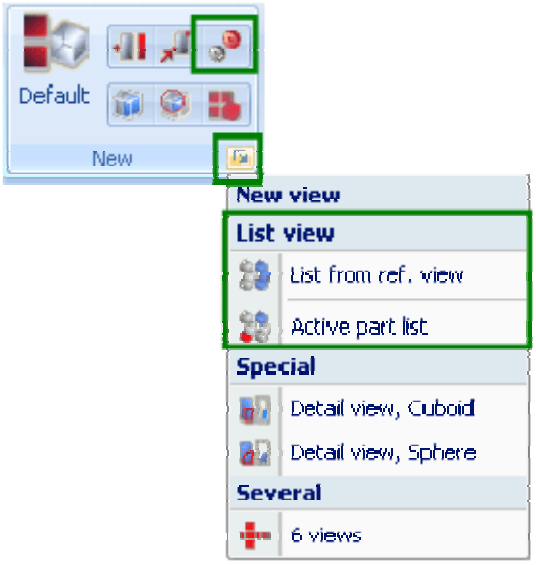


Fig. 148 List view functions

You have the following options for the creation of view lists:

Select parts
Individual parts are selected either by identification with the cursor or via the ICN. You end the selection with a right-click. Specify the position of the new view on the screen and select the desired type of projection.

List from reference view
The parts displayed in another view are taken over into the view list.

- In the *Information + Communication Navigator* all parts hidden in the active view are displayed with reduced colour intensity. The shown parts of a view list constitute the “view list”.
- Normal views become list views when you right-click parts in the ICN and select the **Hide** and **Show** functions.
- List views can be extended or narrowed at any time with the functions described above.

58 Create Detail Drawing

Right-click the part of which you want to create a detail drawing. Select the **Reference part, Save, Detail drawing** function.

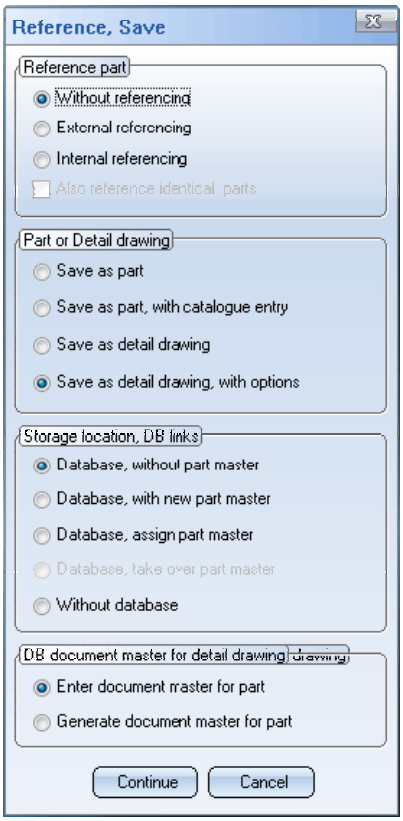


Fig. 149 Create detail drawing (here: without referencing)

The settings selected above are based on the assumption that you only want to create a detail drawing, without referencing the part (or maybe the part is already referenced). If you want to simultaneously create a detail drawing of the active part and reference the part, select the **External referencing** option instead of the **Without referencing** option. When you click **Continue**, the following prompt appears:

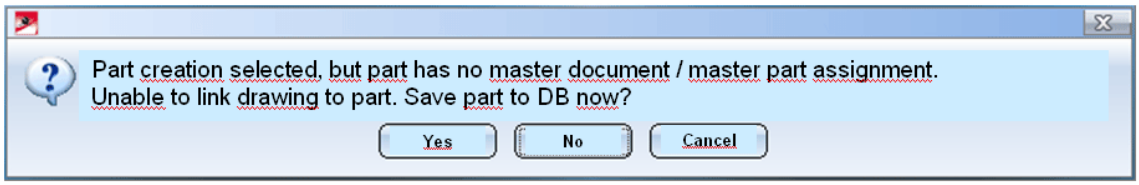


Fig. 150 further question

HiCAD suggests linking the document master of the new detail drawing to the document master of the part. This however requires that the part has a document master, i.e. that it is saved as a part to the database. Therefore you need to answer with **Yes**. After entering part number, designation etc. confirm with OK.

The following dialogue is displayed:

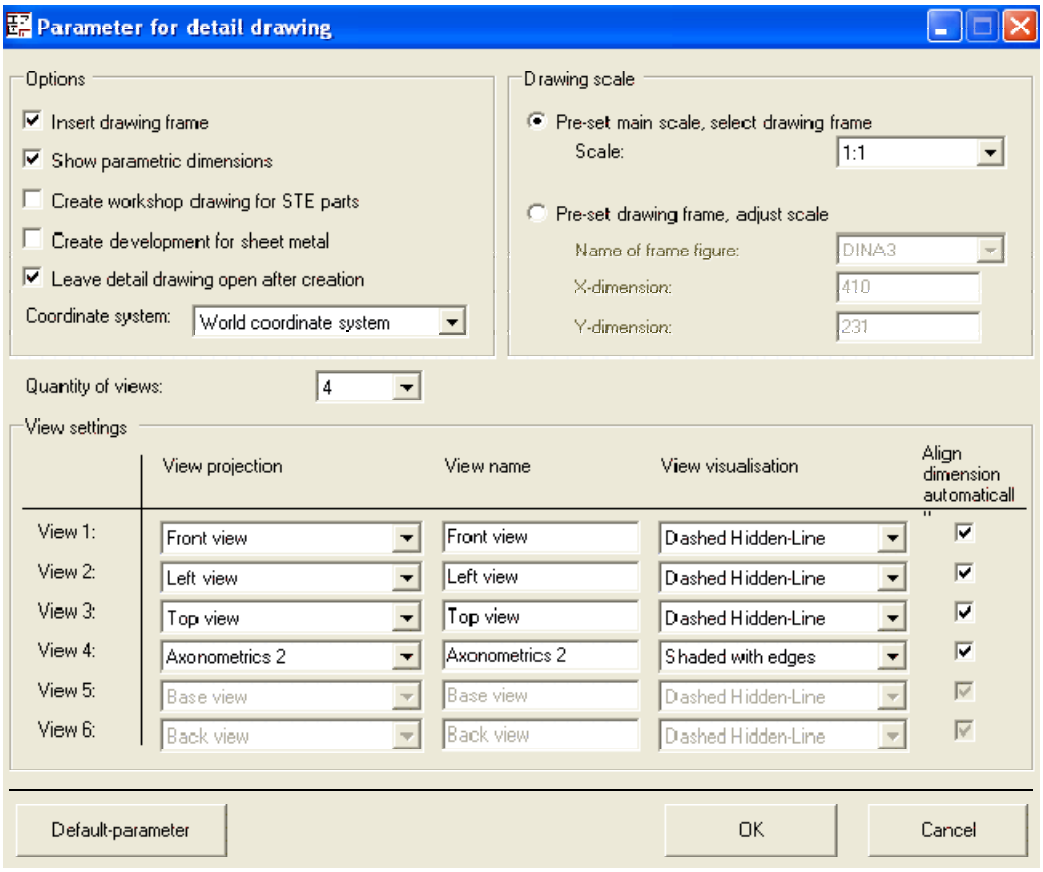


Fig. 151 Parameters for detail drawings

Select the required setting, in particular decide whether you want to preset the scale of the drawing or the size of the drawing frame. Confirm with **OK**. If you have preset the size of the drawing frame, you can confirm the scale calculated by HiCAD. If you have preset the scale, HiCAD will prompt you to select a drawing frame. After that, the document mask for the new detail drawing is displayed. Now enter the document number, designation etc. of the new detail drawing and confirm with **OK**.

Advance information: *Part* master data is a database-specific term. The relevant terminology is explained (if you are working with a database) in a separate database training, which also deals extensively with the “links” topic.

The following prompt does only appear if the part of which you want to create a detail drawing is a main part, and if this part is the only part of the drawing. In this case HiCAD offers to interpret this part as a detail drawing and create 4 views in the Sheet area of the same drawing:

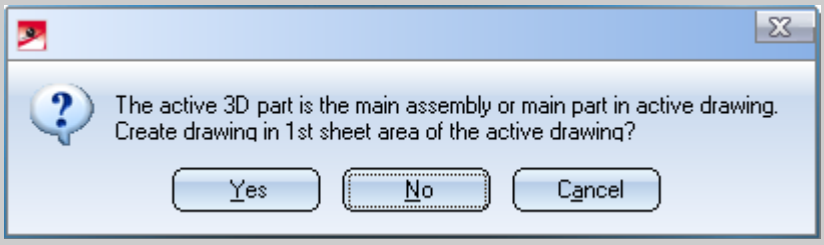


Fig. 152 Prompt appearing if part selected for detail drawing creation is a main part

Answer this question with “Yes” if you want to create the detail drawing in the Sheet area of the *same* drawing, answer with “No” if you want to create a new, separate detail drawing instead.

59 Assembling, Transport and Motion Simulations

You can use the functions of the **Assembling Simulation** (additional module) to perform and evaluate assembling, transport and motion simulations on a 3-D model. This enables you detect and eliminate possible errors at early design stages.

The simulation functions are also very useful for product presentations.

To access these functions, click the **Settings** icon at the top right of the screen, and select **Toolbars**.

In the dialogue, activate the **Toolbars** tab and select the required function.

A detailed description of these functions can be found in the Online Help.

The following figures demonstrate a simulation on the basis of a simple example:

59.1 Assembling Simulation

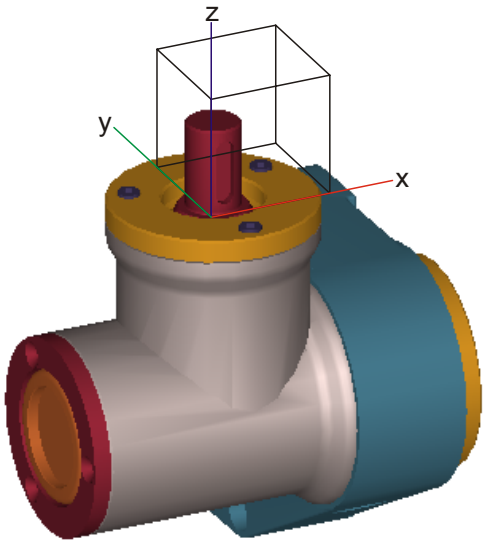


Fig. 153 The original model (assembled)

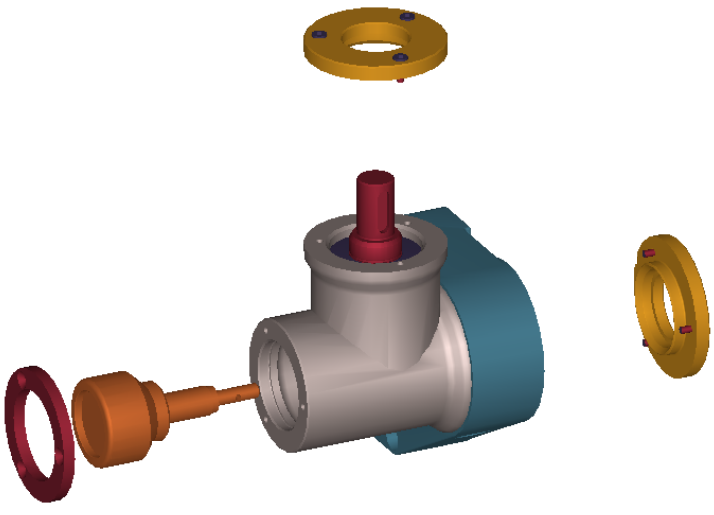


Fig. 154 Disassembling of the model

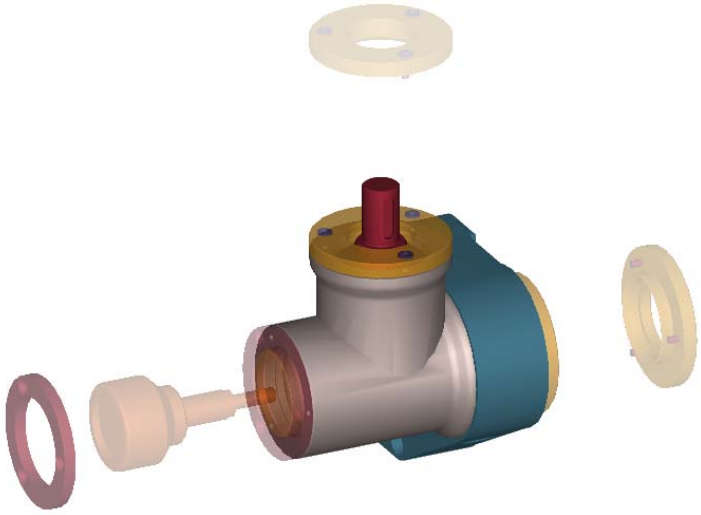


Fig. 155 Complete assembling simulation

The following figures show an assembling simulation with assembling track:

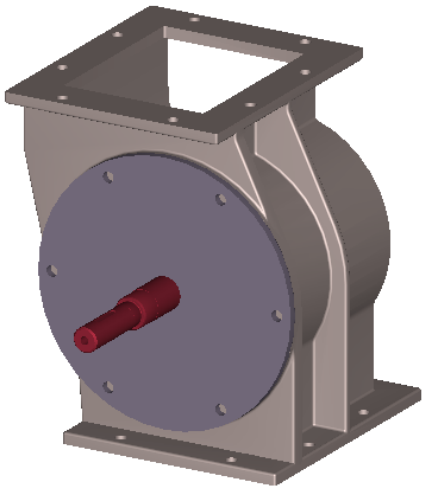


Fig. 156 Original model (assembled)

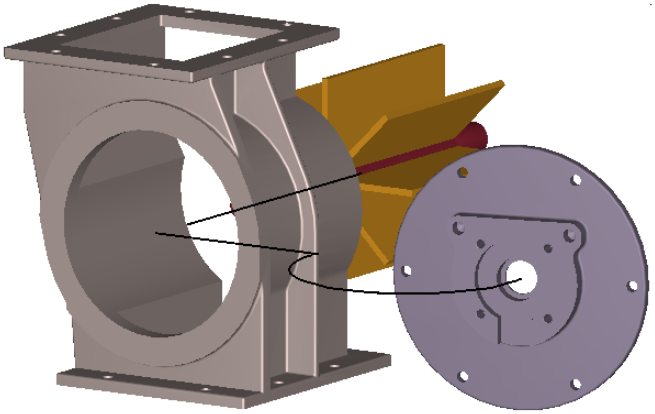


Fig. 157 Disassembled model with assembling tack

59.2 Transport Simulation

The example below simulates the rolling of a ball though a groove. The groove was created with the **C-Edge sweep** function. The spiral line is still present in the drawing.

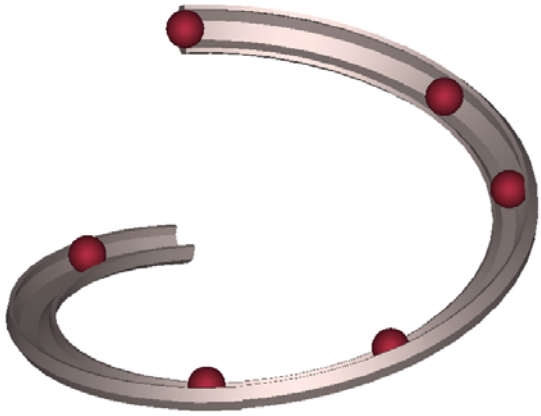


Fig. 158 Rolling ball

59.3 Motion Simulation

The below example simulates the lifting of an arm:

A conceivable sequence of motions would be the following one:

- Translate coordinate system into the centre of the shoulder joint
- X-rotation of the upper arm
- Translate coordinate system into the centre of the arm joint
- Z-rotation of the lower arm
- Translate coordinate system into the centre of the hand joint
- Y-rotation of the hand

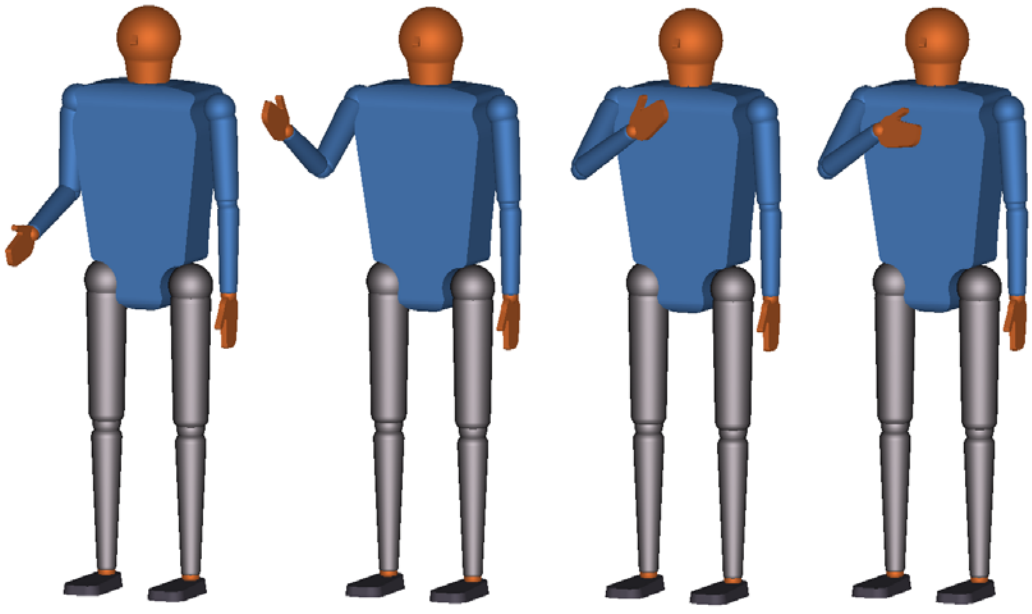


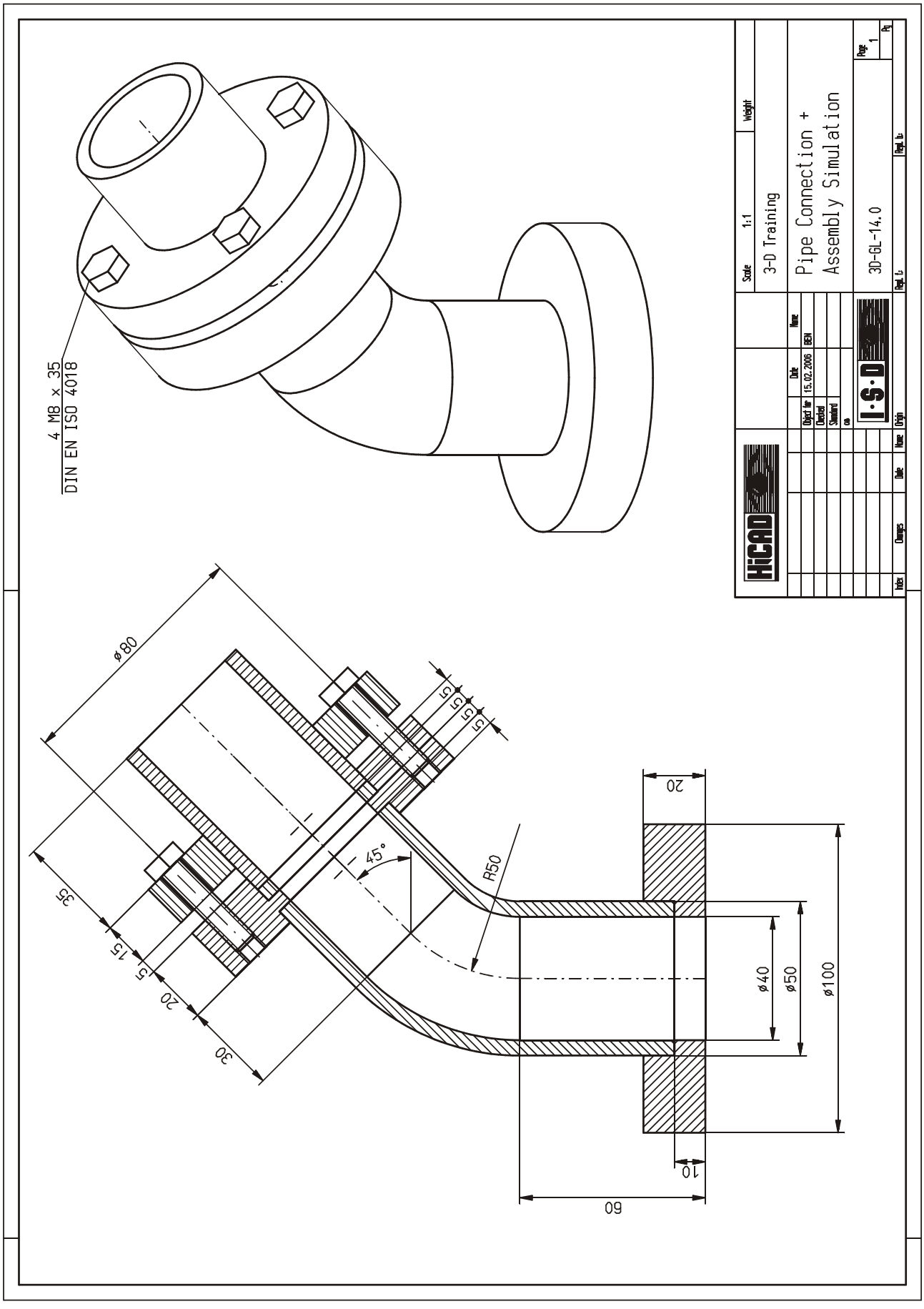
Fig. 159 Sequences of a motion simulation

The figure above shows (from left to right): The original model, the upper arm movement, the lower arm movement and the movement of the hand.

This page is left blank to enable the next exercise to appear on a double page.

59.4 Exercise 14 (3D-GL-14.0) Pipe Connection + Assembling Simulation

- Learning target: Create assemblies, derive detail drawings from the assembly (or vice versa); automatic alignment of assembly and detail drawings via referencing, list view; 3D Model set; assembling simulation; exploded view.
- Create a drawing with database, create one main assembly called “Main assembly Pipe connection” and two (subordinate) assemblies called “Assembly Pipe parts” and “Assembly Flanges”
- Draw in a 3-D Sketch the 3-D guideline of the pipe (tangent with a 45 degree angle at a circular arc with a radius of 50, etc.). Draw ALL following parts as referenced parts!
- Create the pipe as a (hollow) c-edge sweep.
- Divide the pipe into the three pieces shown in the drawing and create the 5 mm gap with “Subtract part, via Translation”.
- Draw the lowest flange as a hollow cylinder and subtract the pipe. Position a processing plane on the upper end of the pipe. If you draw the two upper flanges as hollow cylinders, they are both angled by 45 degrees when inserting them. Please remember that the wall thicknesses of the two flanges are different!
- Subtract the three pipe parts from the middle flange and execute the standard part processings.
- Check whether all pipe parts are below the “Assembly Pipe parts“ and all flanges below the “Assembly Flanges“.
- Save the drawing!
- Derive two detail drawings: One from the bottom flange and one from the upper section of the pipe (with referencing and via the database)
- Load both detail drawings. Apply some minor modifications to the detail drawings (e.g. fillet, respectively chamfer) and save them. Change to the assembly drawing and take over the modifications of the referenced parts.
- Now apply a minor modification to the lower flange in the assembly drawing (e.g. a further fillet). Save the assembly drawing.
- Switch to the detail drawing and take over the modifications here as well.
- Assign a Material to the parts and create, in the Sheet area, the sectional view. Change the hatching of individual surfaces. Add dimensionings and insert a drawing frame in the Sheet view.
- In the Sheet area, create a list view of the 3 sections of the pipe, and a second list view of the 3 flanges.
- Use the Model set option to separate the upper section of the pipe and the upper flange into another drawing. Apply a minor change (e.g. fillet) and retransfer the change to the main drawing.
- Make an Assembling Simulation (Settings > Toolbars > Toolbars > 3-D Assembling simulation). If the parts will also move in the sectional view and the 2 list views, freeze the sectional view and the 2 list views (“View Functions” function group in the “Views“ tab).
- Create a new drawing. Copy the main assembly “Pipe connection” into this new drawing. De-reference the parts to prevent possible repercussions of the planned exploded view on the original drawing. Activate the pipe and create the exploded view.
- Hint: If you have several views and want to create an exploded view it makes sense to freeze all views, except for the one in which you want to display the exploded view.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HiCAD
Scale 1:1
Weight
3-D Training
Pipe Connection +
Assembly Simulation
3D-GL-14.0
Page 1
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Object type
Created
Standard
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I·S·D
Index
Changes
Date
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Origin
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60 Part Offset

Use this function to create an offset of the active part. You can find the **Part offset** function in the **+/-** function group of the **3-D Standard** tab.

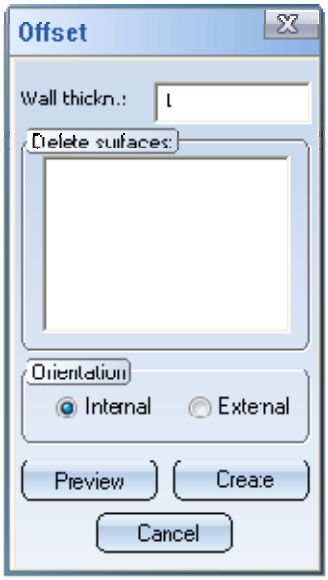


Fig. 160 The **Offset** dialogue

Enter the wall thickness and select whether you want to create an internal or external offset.

HiCAD prompts you to select surfaces. You have the following two options:

- Select an arbitrary number of surfaces that you want to delete. Click the **Preview** button. Select **Preview** again to change wall thickness and the direction of the offset, if required. Then select **Create**.
- Select no surface, but click the **Preview** button, then the **Create** button. HiCAD immediately creates an offset, without deleting any surface.

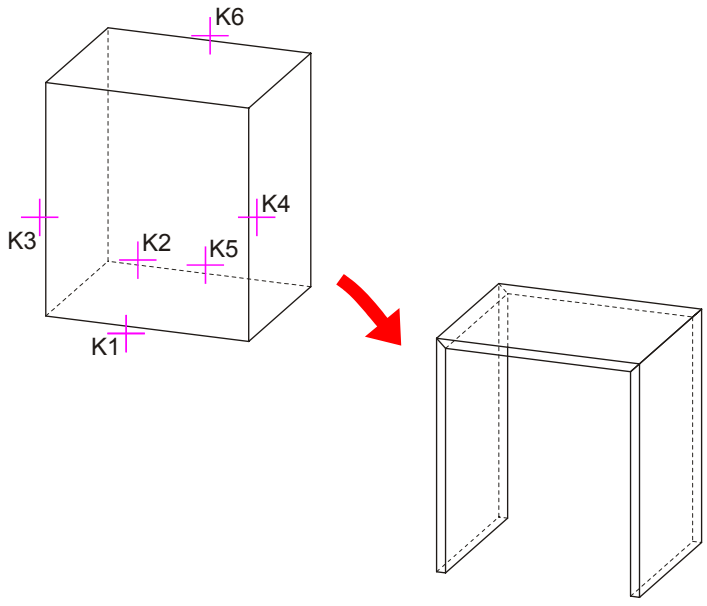


Fig. 161 Internal offset

Tip: You can also use the **Part offset** function to assign wall thicknesses to *surface* parts.

61 Create Graphics

2-D and 3-D drawings can also be used for presentations and brochures, or can be inserted as graphics into a letter. Before converting the drawing to a graphic, you can assign RAL-colours or custom colours to entire parts or individual surfaces, or use “layer 40” to make them transparent. Use the Reality Studio (additional module) to insert light sources (lamps) and spots into your drawing. Further options are shades, light reflections, textures, and much more. The Reality Studio also enables the simulation of camera panning as well as animated motion simulations.

The following paragraphs provide explanations of various commands for the conversion of drawings to graphics. Detailed information on the Colour Editor can be found in the Online Help. For further information on the Reality Studio, please contact your sales agent.

61.1 CTRL+C

On your keyboard, select the CTRL+C shortcut. Select the EMF or Bitmap format and enter the desired size of the image in pixels. Confirm with **OK**. Specify the required drawing detail in a rectangle and select a background colour. The created graphic is then copied to the Windows clipboard and can be loaded into any picture processing program such as Paint, Corel-Draw etc. The program “Paint“ can be accessed, (depending on the operating system used), e.g. via **Start > Programs > Accessories …**

61.2 CTRL+W

On your keyboard, select the CTRL+W shortcut. Draw a rectangle around the drawing detail that you want to convert to a graphic. Select **Process with Paint Brush**. After confirming with OK, the selected detail of the drawing is displayed as a bitmap. You can now either save the bitmap or process it further.

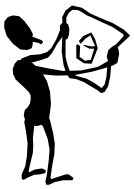
61.3 PrtSc Key

You find the PrtSc key to the right of the F12 key on you keyboard. If you select PrtSc, the entire screen (including toolbars) is converted to a bitmap. You can then load this bitmap to an image processing program from the clipboard.

Fig. 162 Awning and kitchen, drawn in HiCAD and processed with Reality Studio

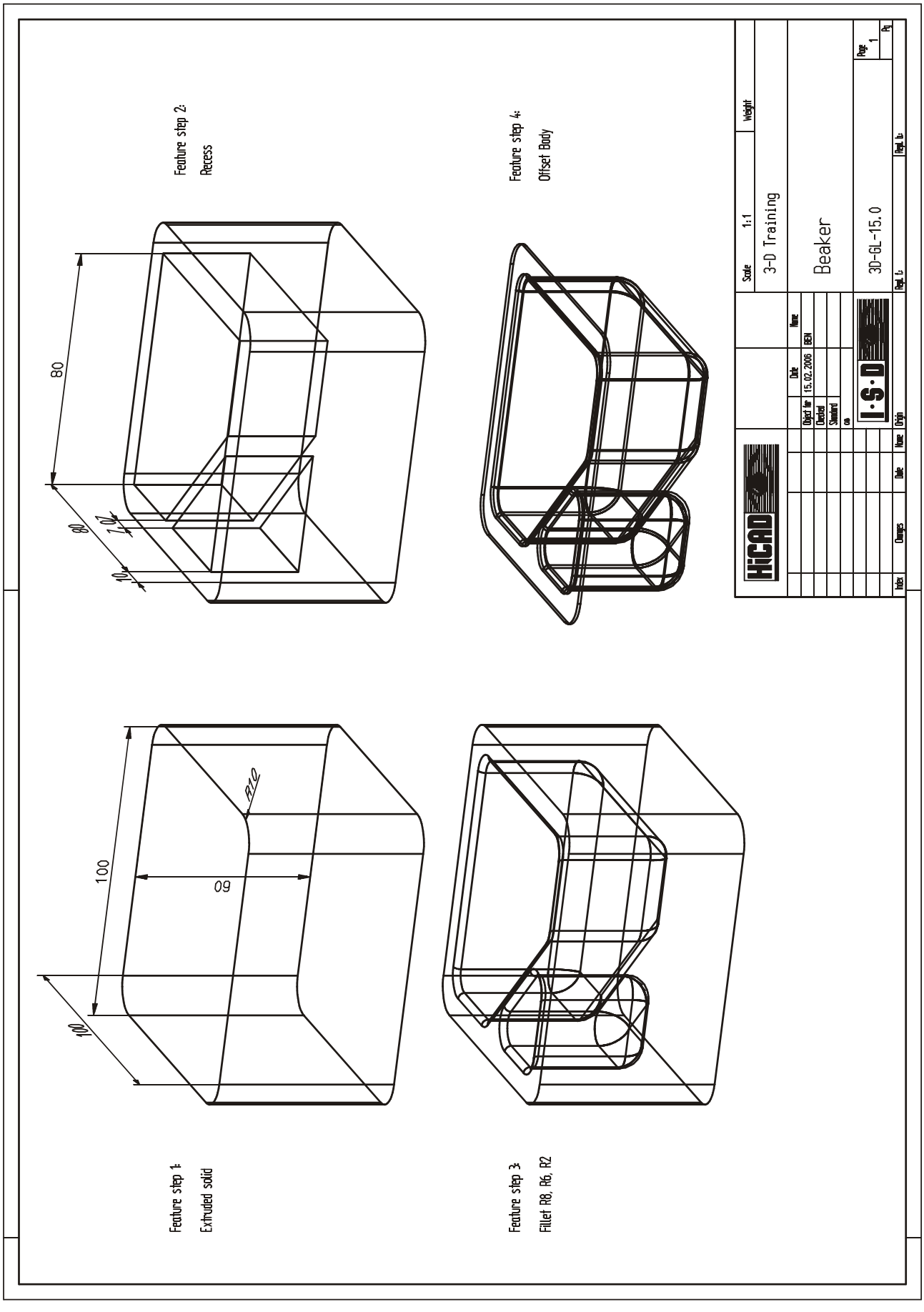
61.4 Create EMF File

To access the **Create Enhanced Metafile (EMF)** function, activate the **Drawing** tab and select **Others > Extras > Tools**.



61.5 Exercise 15 (3D-GL-15.0) Beaker

- Learning target: Offset; options of the Feature log, derive graphics from HiCAD drawings.
- Create the beaker as an extruded solid (Referencing and Feature: Yes). Create the subtractions and fillets.
- Delete all generated surfaces and the floor space, but not the top surface.
- Allocate a wall thickness of 1 mm with the “Offset” function.
- The trainer will explain the options of the Feature log. You can, e.g., move the red arrow to the start of the Feature log and go through the log step by step. You can also move or deactivate Feature steps etc.
- Shade and colour the beaker. Position it on layer 40, to make it transparent.
- Create a graphics form this drawing in three different ways:
  - CTRL+C
  - CTRL+W
  - PRINT key
  - Create Enhanced-Metafile (EMF)
- Space for notes:


Feature step 2:
Recess
80
80
10
Feature step 4:
Offset Body
Feature step 1:
Extruded solid
100
100
R12
60
Feature step 3:
Fillet R8, R6, R2
HiCAD
Scale 1:1
Weight
3-D Training
Beaker
3D-GL-15.0
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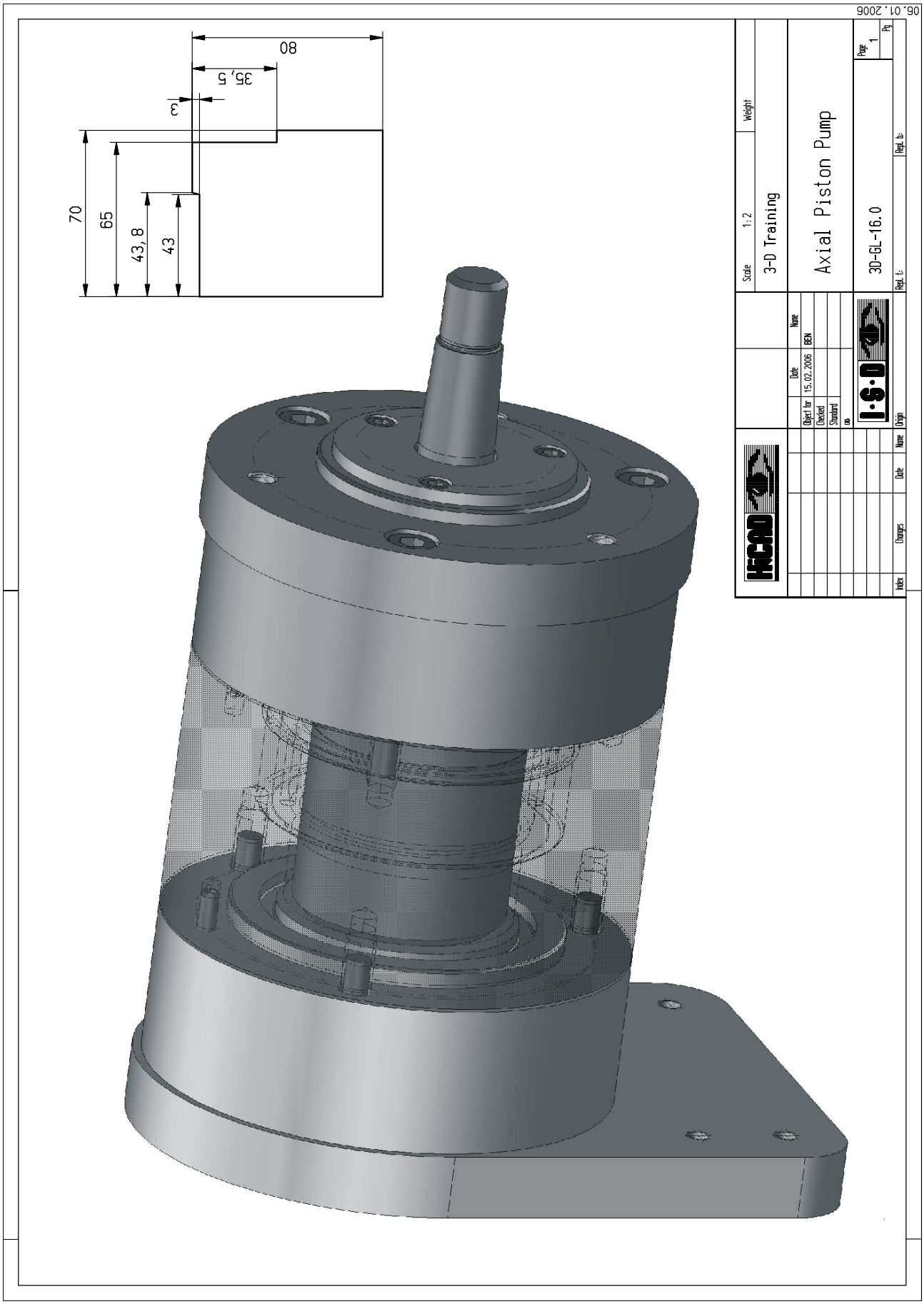
62 Designing in Assemblies

The first exercises in this 3-D training book dealt with the creation of detail drawings. In the exercise “Pipe Connection + Assembling Simulation” we have created an assembly and derived detail drawings from it. The following exercise shows how to design within an assembly:

62.1 Exercise 16 (3D-GL-16.0) Axial Piston Pump

- Learning target: Changing and modelling of parts in an assembly drawing.
- Load the drawing “Axial Piston Pump” from the hicad / Szenen / Examples directory.
- Save the drawing to the database (to prevent an accidental overwriting of the original drawing)!
- Activate the part “Casing hub”. You find it below “Hub complete”. Translate it 500 mm to the right. We are now going to draw this flange in an *approximated* form.
- Change to the front view and draw the rough contour of your flange.
- Correct the sketch with the “Process Sketch” function.
  - Freeze the bottom and the left line of the contour of the sketch with the help of the 3-D c-edge HCM.
  - Change the diameter of the top left and top centre horizontal lines by dynamic dragging.
  - Define an exact distance for those two horizontal lines. Use the function “Offset - Parallel to Y-Axis“.
  - Allocate an appropriate width and height to the revolved solid. For example, at the left side, the outer edge of the flange needs to be flush with the part “Case bearing part” (you will find it below “Casing”). A further help is the dimensioned 2-D contour in the drawing.
  - Take over the sketch.
- Create at least one of the subtractions.
- Add hole pattern, countersink and bolting. Use the flange as an orientation; you translated it 500 mm to the right.
- Change the assembly via the context menu functions.
- Add at least one more hole pattern.
- If your flange has not been referenced, save subsequently as a referenced part.
- Create a new (individual part) drawing with the flange you drew.
- Chamfer an optional point of the flange you drew in the assembly drawing. Check if the individual part drawing has been up-dated automatically.
- The angular hollow area to the left can be created with the part “hicad / Szenen / Examples / AX-GEH-SUBTRAKT.KRA (position to ABS 0 0 0).
- You will find two additional exercises on the following pages.

Tip: The additional exercise 3D-GL-Z32 is highly recommended if you want to practise the working with processing planes.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Weight
3-D Training
Axial Piston Pump
3D-GL-16.0
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Checked
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Origin

63 Designing with Other HiCAD Modules

This training book mainly dealt with functions of the **3-D** module, which is one of very many HiCAD modules. Below please find short descriptions of these modules. If you require further information on any HiCAD modules, please contact your sales agent.

2-D: Besides the 3-D module, HiCAD also offers the traditional 2-D module. This module, too, facilitates your daily work by many automatisms and a 2-D part structure.

HELiOS Document Management: Enables you to create and efficiently manage drawings as new, modified, derived, or follow-on sheets drawings. More than 20 attributes are available allowing you to search for particular drawings. For an efficient project management, assignment of rights, workflows, approval and release processes, etc.

HELiOS Part and BOM Management: Enables you to assign master data to (2-D or 3-D) parts. BOM creation at the push of a button. Allows the creation of dynamic BOMs, parts where-used lists, user-defined BOM formats. The combination of HELiOS Document Management and HELiOS Part and BOM Management is also called “Product Data Management” (PDM).

Sheet Metal: Enables an efficient creation and development of sheet metal constructions. Parts from the 3-D module can also be developed. Allows a selection of material and development procedures (outer/inner leg, neutral chamfer etc.), as well as user-defined development procedures. Provides many sheet metal-specific functions, from mitre cuts and overbend tables to punching and forming tools. Automatic covering of 3-D parts with sheet plates. Reconstruction of sheet metal parts from 2-D developments (even if not created with HiCAD), and much more.

Steel Engineering: Provides steel engineering-specific standard parts from rolled profiles to gratings, and steel engineering processing functions such as mitre cuts, notch and DAST connections. Automatic itemisation of all parts, automatic detail drawing and workshop drawing creation, creation of user-defined steel beam series. Optional extension modules: Stairs module, Metal & Facade Engineering, including logiKal interface.

Stairs: Enables the creation of straight and winding stairs. Straight stairs are already included in the steel engineering module.

Metal and Facade Engineering: Enables the creation of glass facades, conservatories etc. Loading of parts from logiKal is optionally available. All HiCAD 3-D and Steel Engineering functionalities can also be used for Metal Engineering constructions.

Plant Engineering: Provides plant engineering-specific standard parts from pipes and vessels to small chemical plants. Enables an individual expansion of the plant engineering part catalogues by user-defined parts and the creation of plant engineering BOMs. Offers many automatisms enabling a significant reduction of calculation times.

P+ID: Enables the creation of Piping and Instrumentation Diagrams (P+IDs), which can be linked to 3-D plant engineering layout plans.

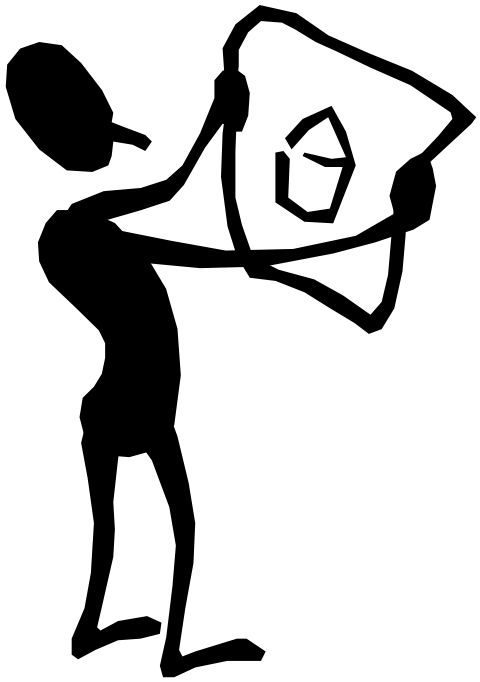
Freeform Surfaces: Module for all parts which are not analytically describable. This module is particularly interesting for the Automotive Industry and casting manufacturers.

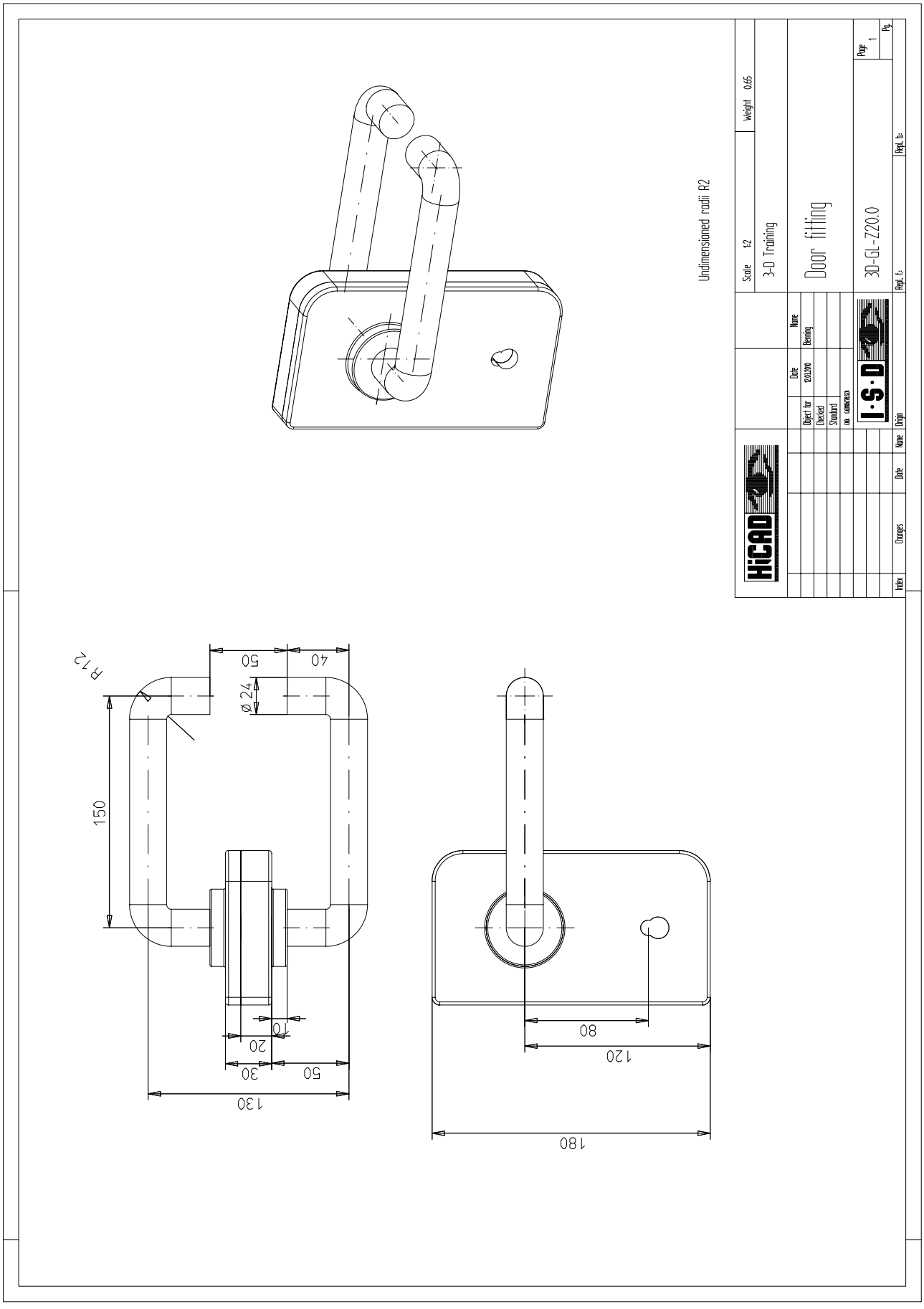
Macro Programming: Allows the creation of user-defined successions of commands and call them from the GUI / the keyboard. This is possible for almost all HiCAD modules. The macro module enables you to add further commands to the “normal” user interfaces of the individual modules.

Variant Technology: If a company uses part which only differ in their dimensions, these parts need to be drawn only once, as so-called “variants”. For a second creation, only the entering of dimensions, number of bores etc. will be required. The variant is then drawn automatically.

For information on **further modules** please contact your sales agent.

64 Additional Exercises




Undimensioned radii R2
Scale 1:2
Weight 0.65
3-D Training
Door fitting
3D-GL-Z20.0
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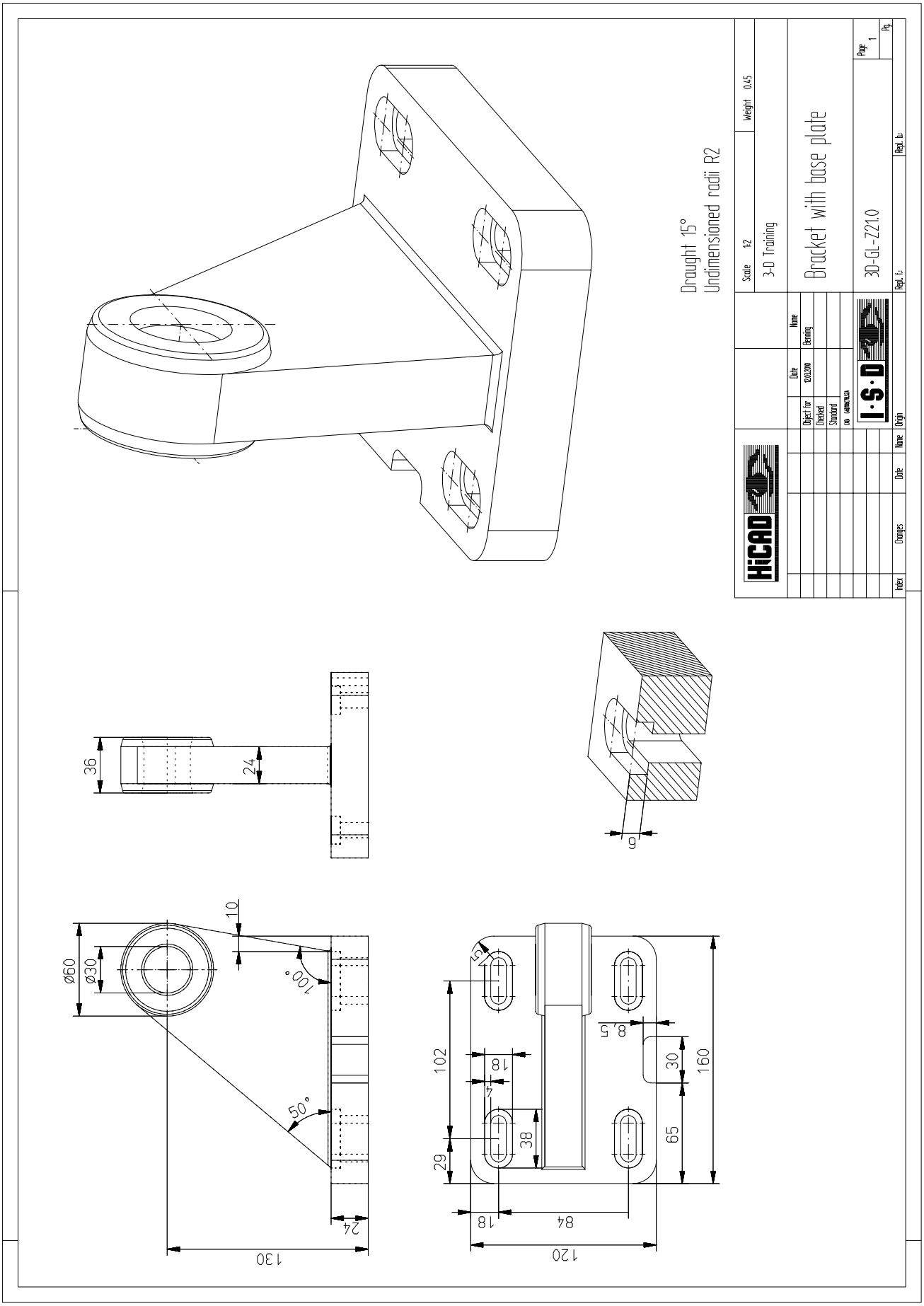
Draught 15°
Undimensioned radii R2
Scale 1:2
Weight 0.45
3-D Training
Bracket with base plate
3D-GL-Z210
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29
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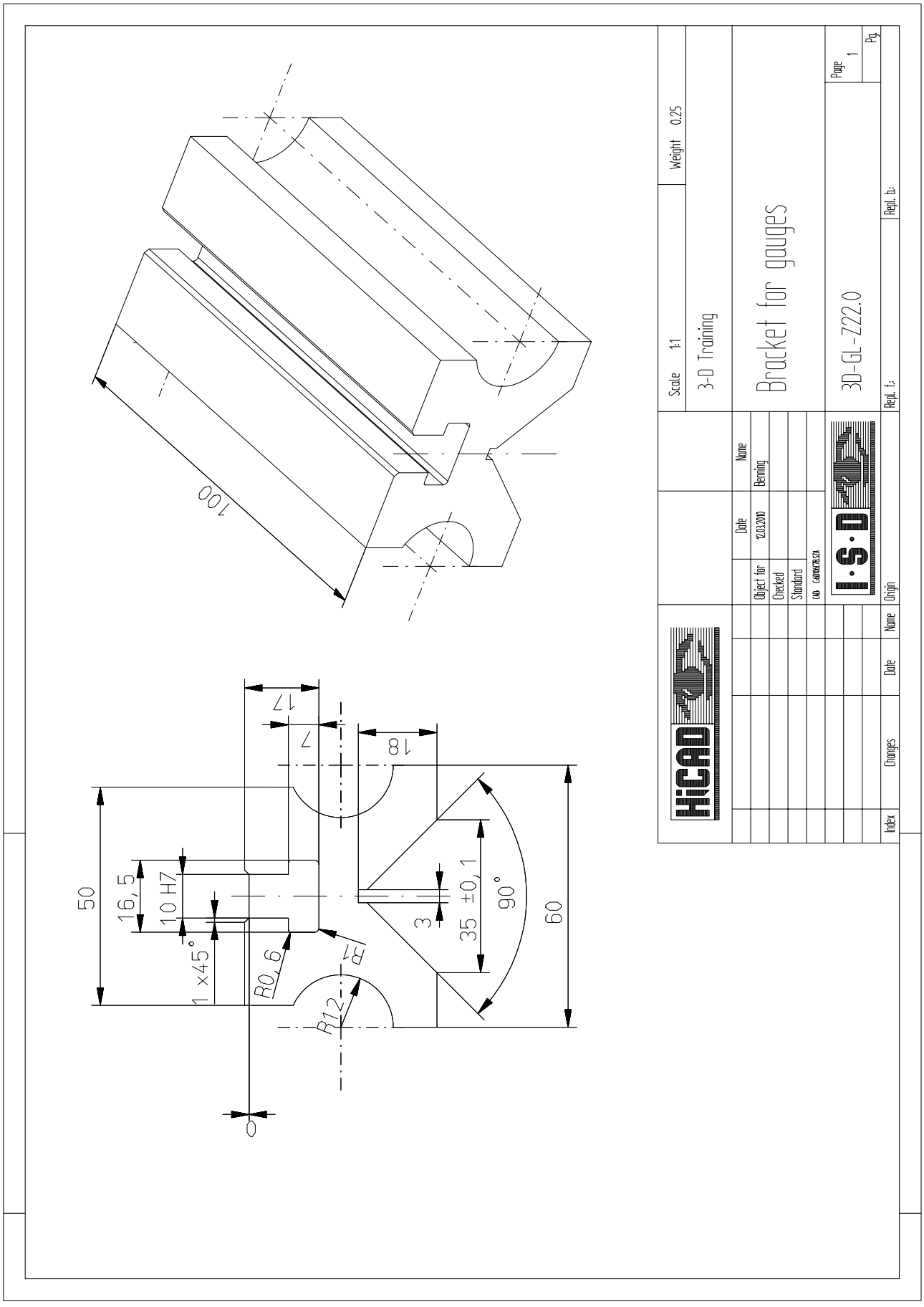
120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R0,6
R12
3
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60
Scale 1:1
Weight 0.25
3-D Training
Bracket for gauges
3D-GL-Z22.0
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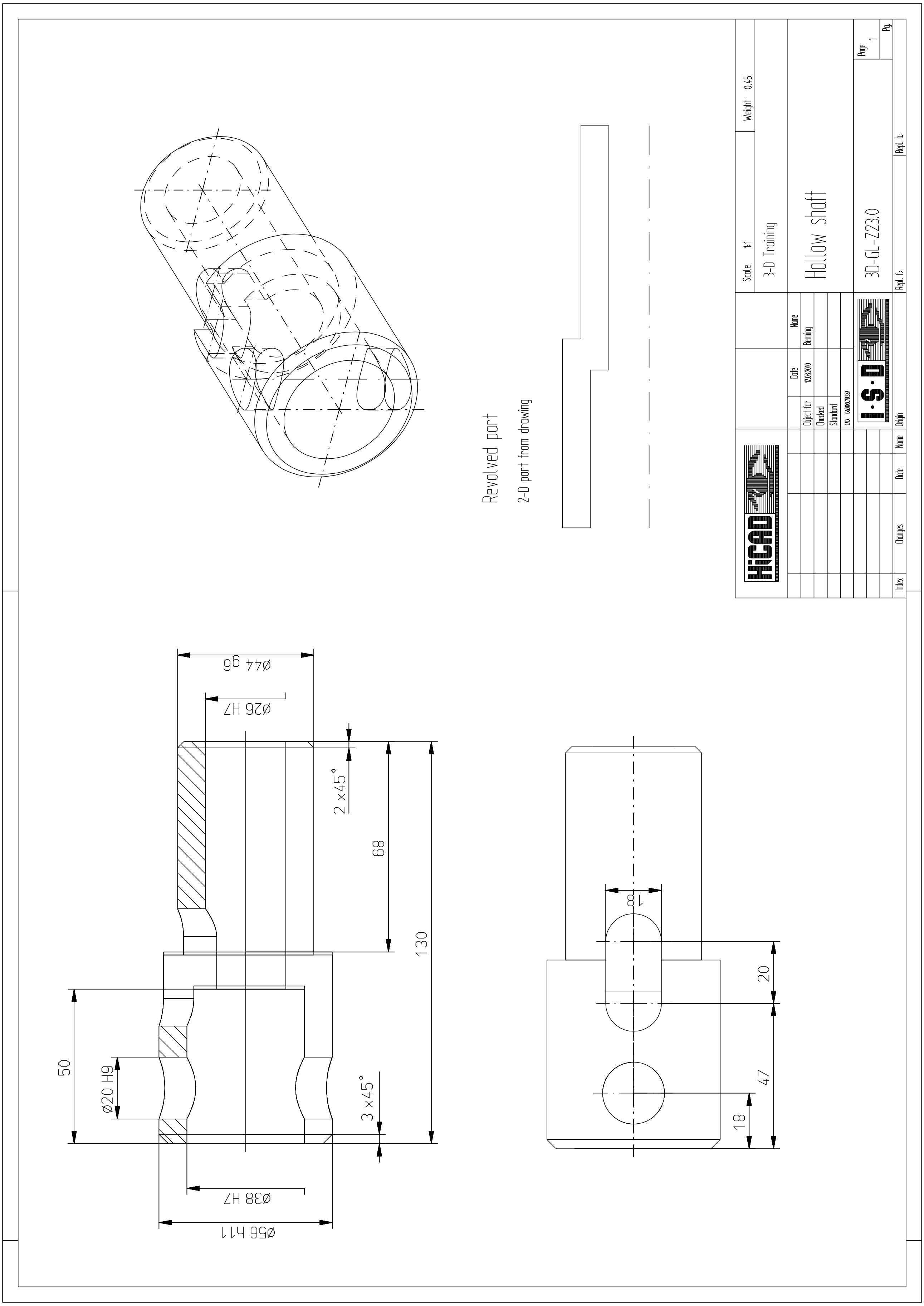
Revolved part
2-D part from drawing
HiCAD
Scale 1:1
Weight 0,45
3-D Training
Hollow Shaft
3D-GL-Z23.0
Object for
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Standard
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Changes
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Ø20 H9
3×45°
Ø38 H7
Ø56 h11
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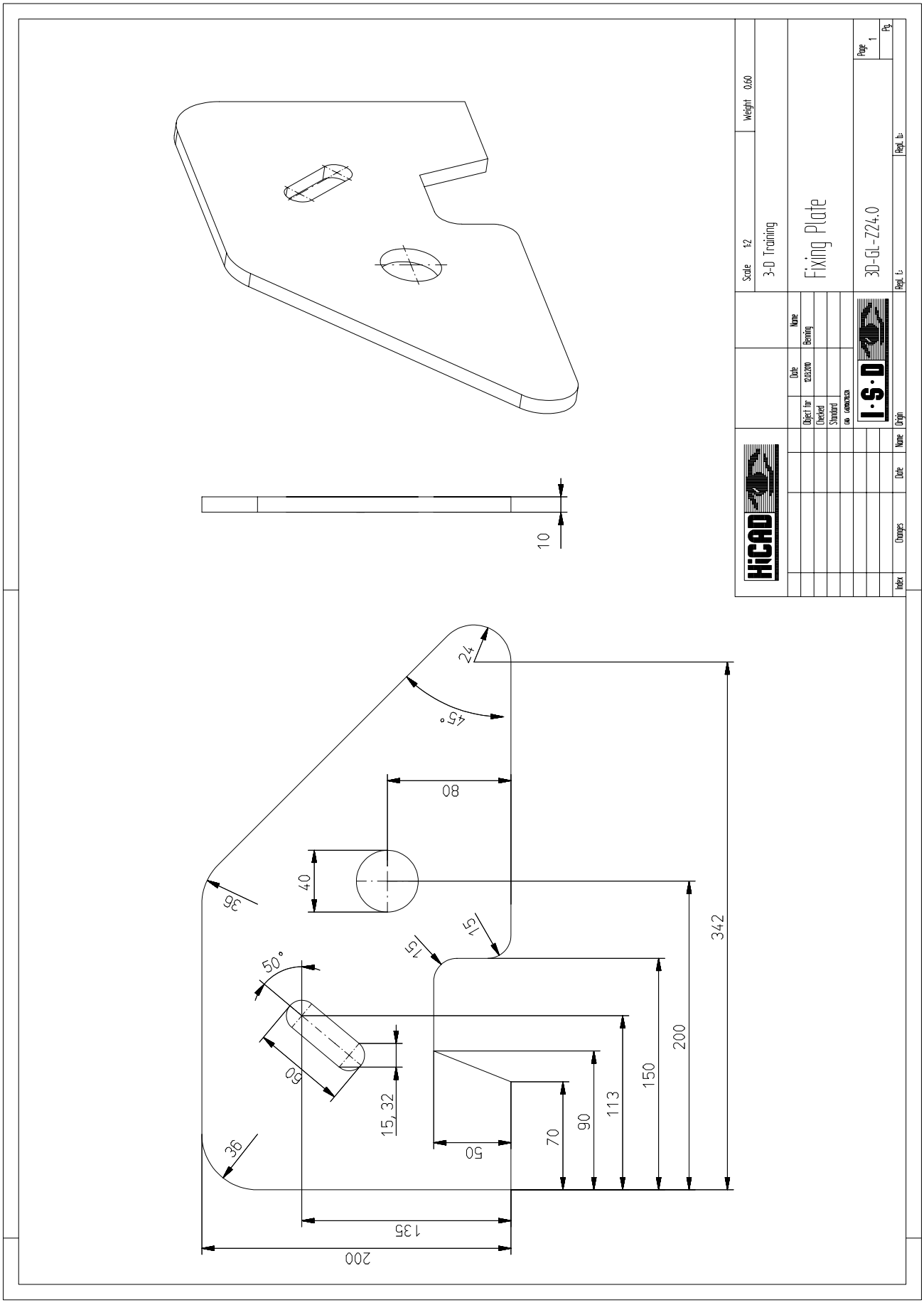
18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Weight 0.60
3-D Training
Fixing Plate
3D-GL-Z24.0
Page 1
Name
Date
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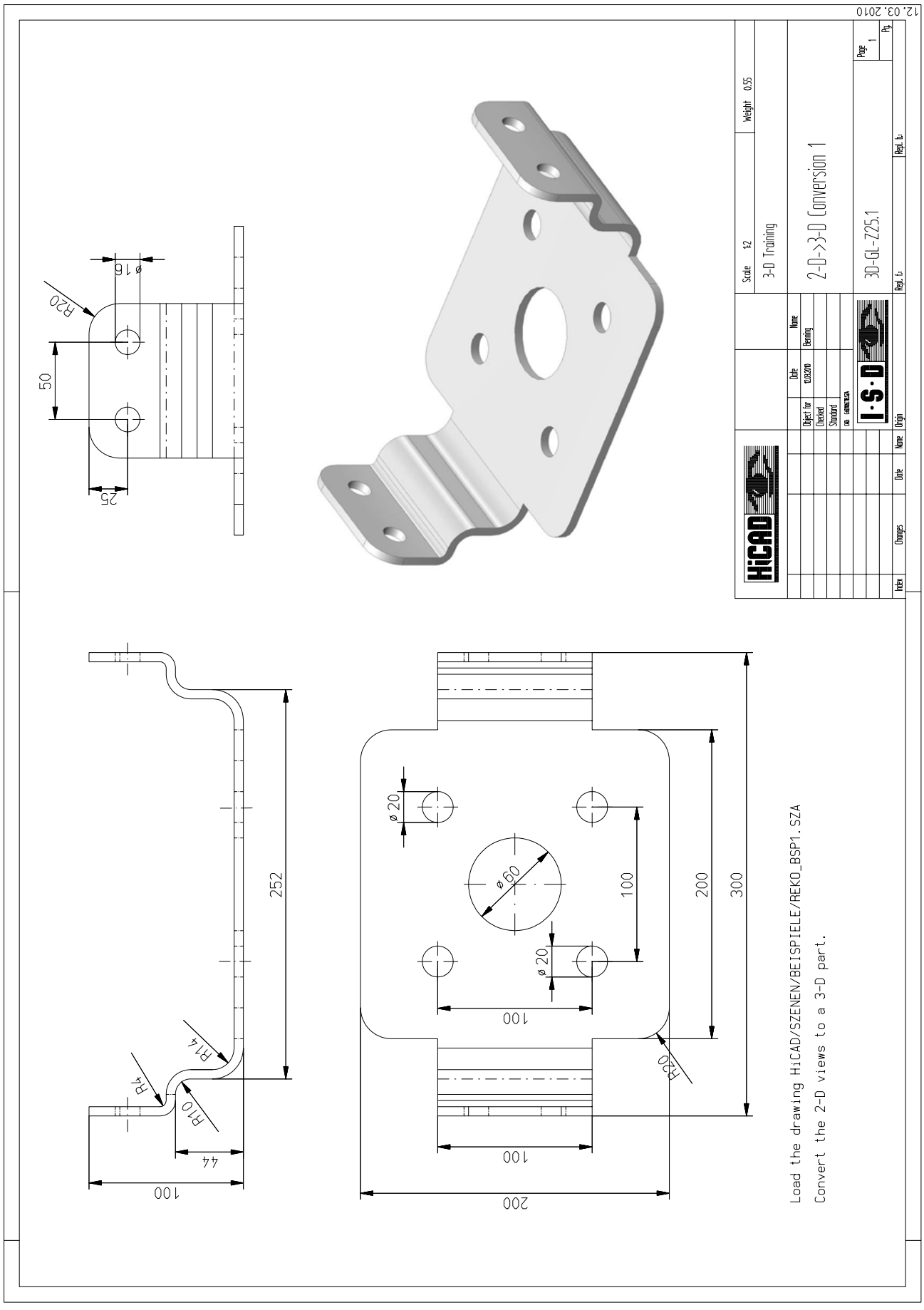
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Scale 1:2
Weight 0.55
3-D Training
2-D->3-D Conversion 1
3D-GL-Z25.1
Page 1
HiCAD
I·S·D
Load the drawing HiCAD/SZENEN/BEISPIELE/REKO_BSP1.SZA
Convert the 2-D views to a 3-D part.
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R14
R4
R10
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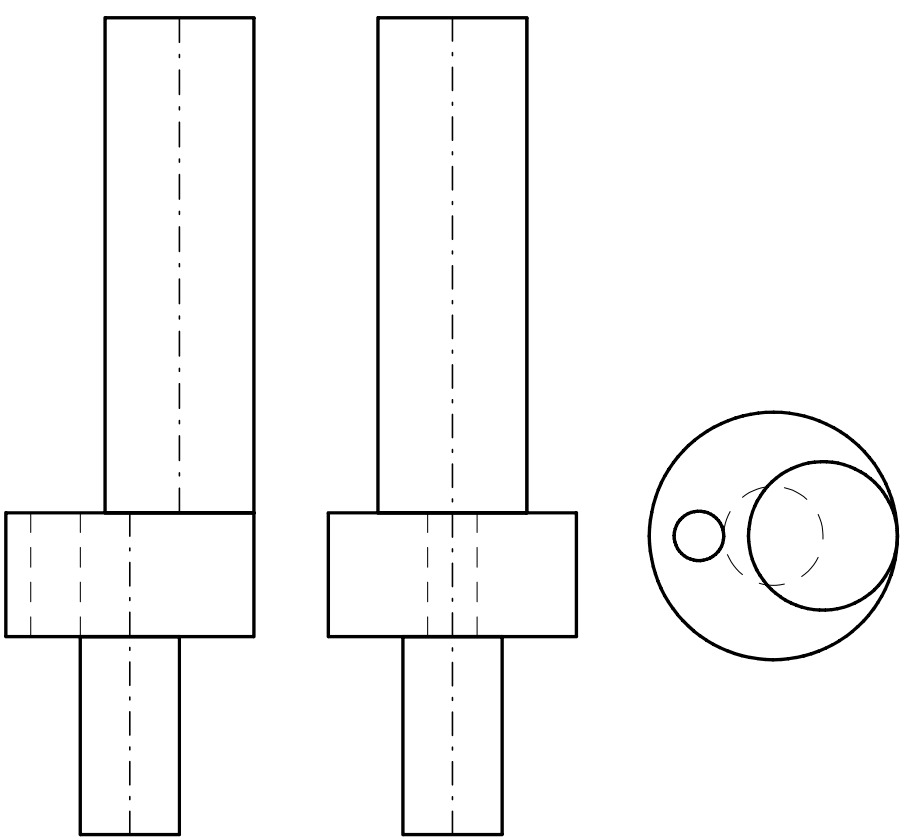
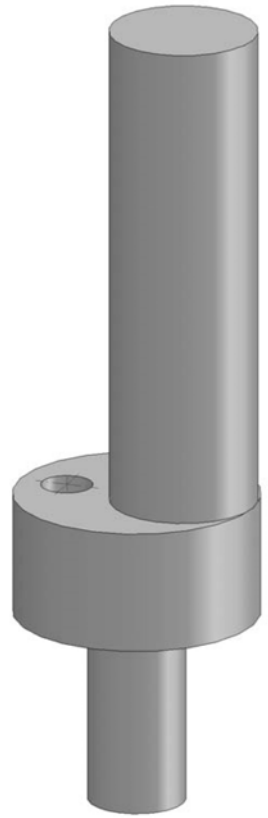
R20





Load the drawing
HiCAD/Szenen/REKO_BSP5.SZA.
Convert the 2-D views to a
3-D part.

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Scale 1:2.5 | Weight 0.25

3-D Training

2-D -> 3-D Conversion

3D-GL-Z25.2

Page 1 | Pg.

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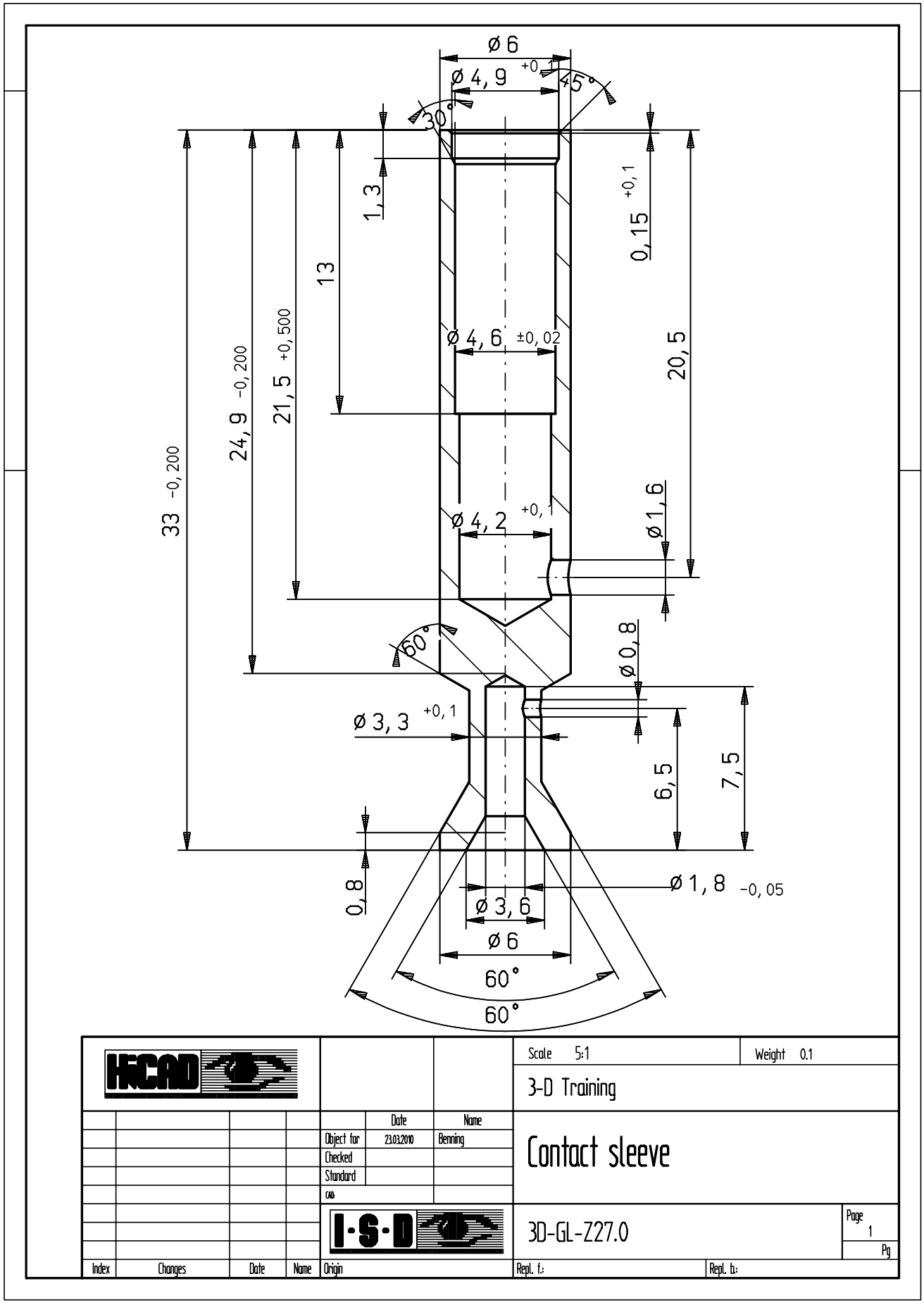
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0, 15 +0, 1
ø 4, 6 ±0, 02
20, 5
24, 9 -0, 200
21, 5 +0, 500
33 -0, 200
ø 4, 2 +0, 1
ø 1, 6
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ø 0, 8
ø 3, 3 +0, 1
6, 5
7, 5
0, 8
ø 1, 8 -0, 05
ø 3, 6
ø 6
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60°
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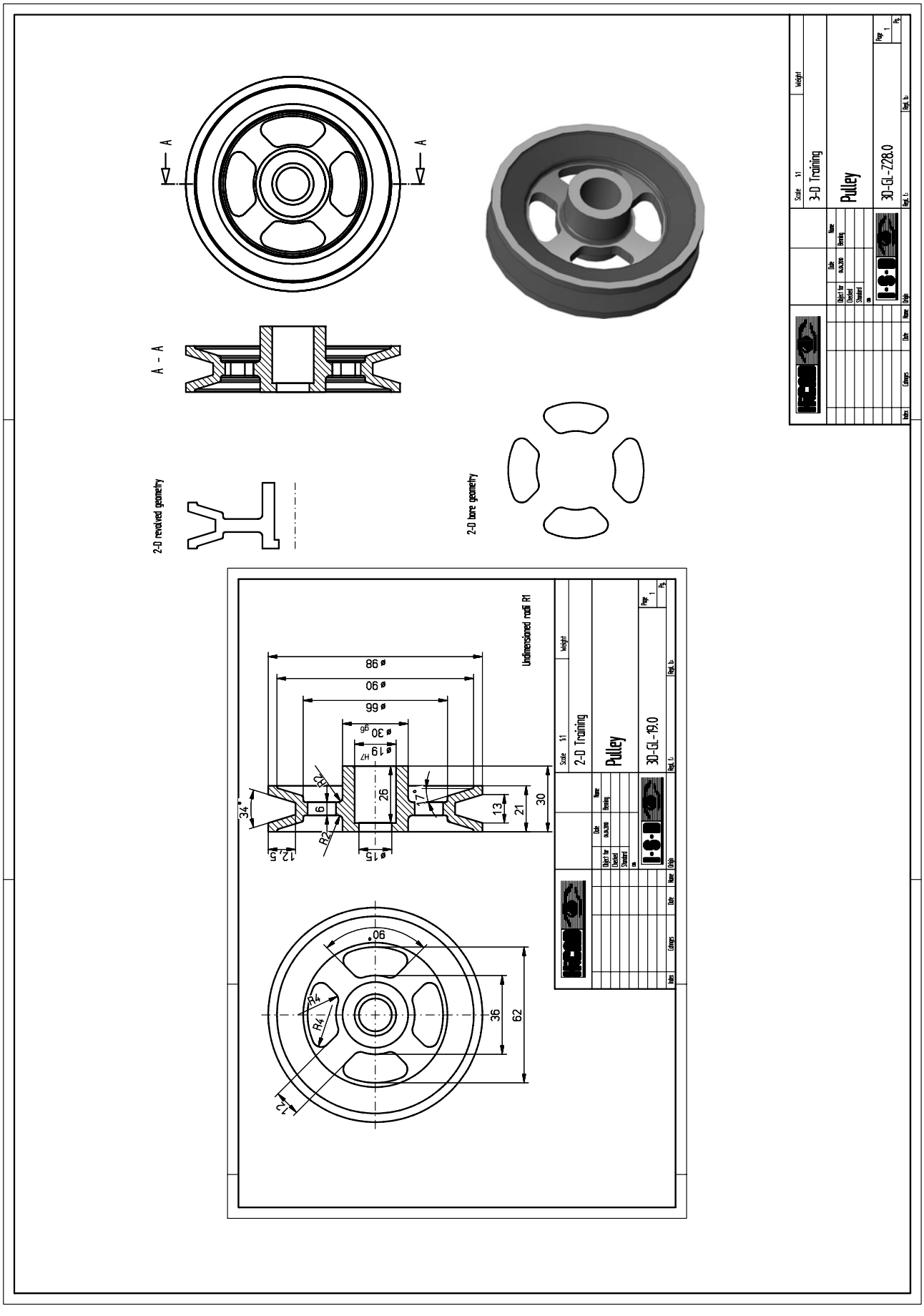
A
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A - A
2-D revolved geometry
2-D bore geometry
Undimensioned radii R1
Scale 1:1
Weight
3-D Training
Pulley
3D-GL-728.0
Page 1
Scale 1:1
2-D Training
Pulley
3D-GL-19.0
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66 ø
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6
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R2
R2
12,5
15
90°
R4
R4
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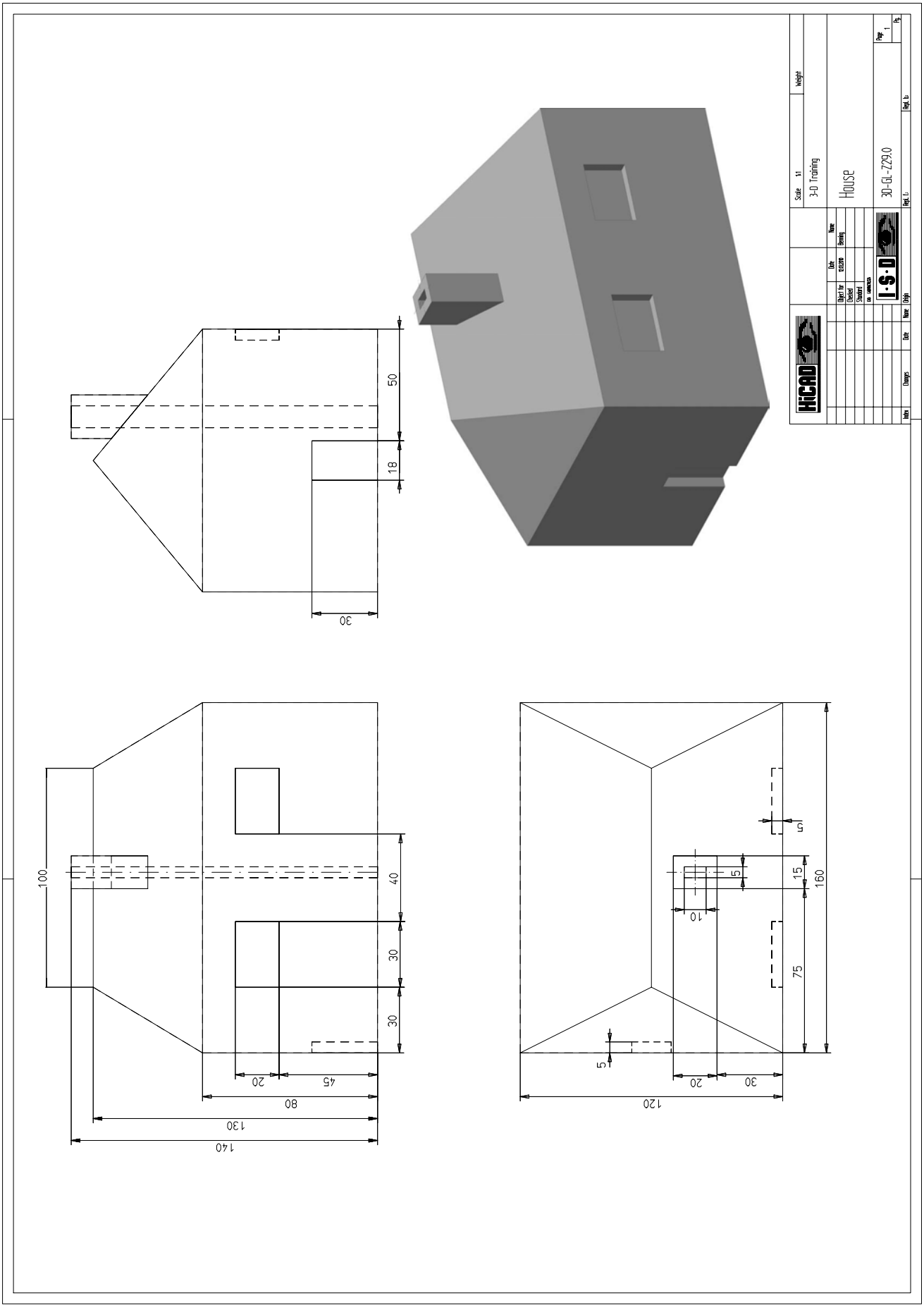
Scale 1:1
Weight
3-D Training
HOUSE
3D-GL-Z29.0
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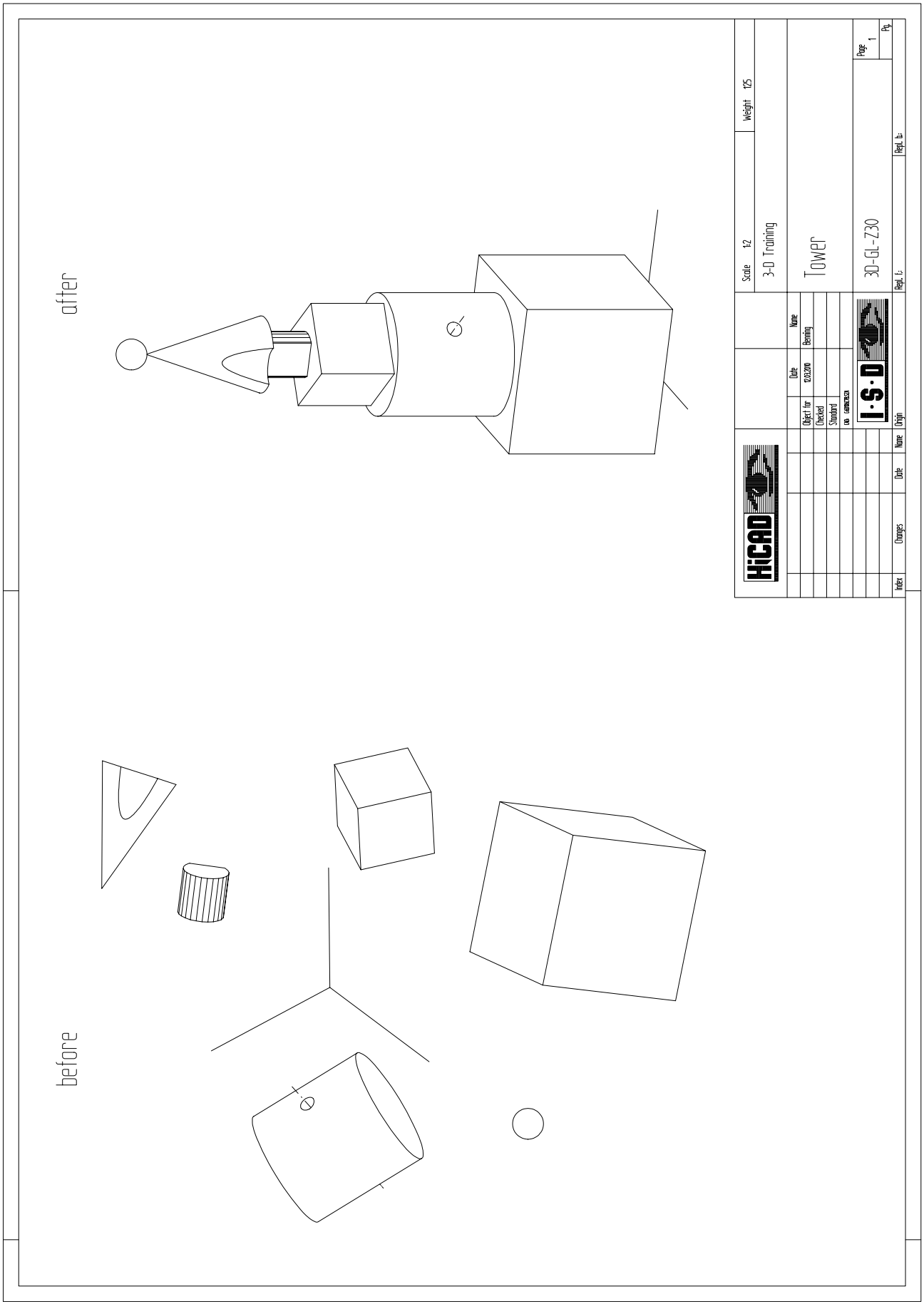
after
before
Scale 1:2
Weight 125
3-D Training
Tower
3D-GL-Z30
Page 1
Repl. to:
Repl. by:
HiCAD
I·S·D
Object for
Checked
Standard
as commercial
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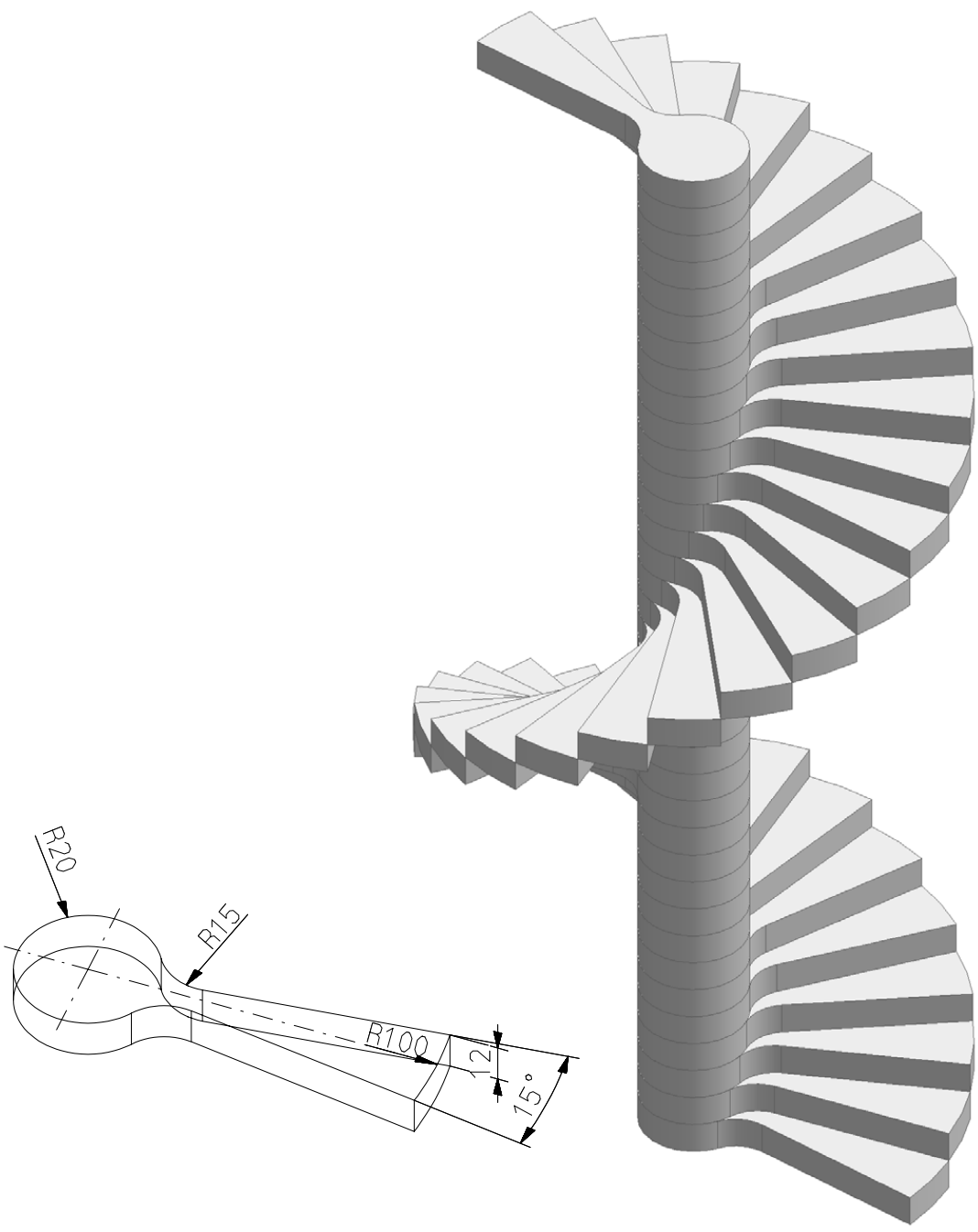
Index
Changes
Origin



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						3-D Training	
					Date	Name	
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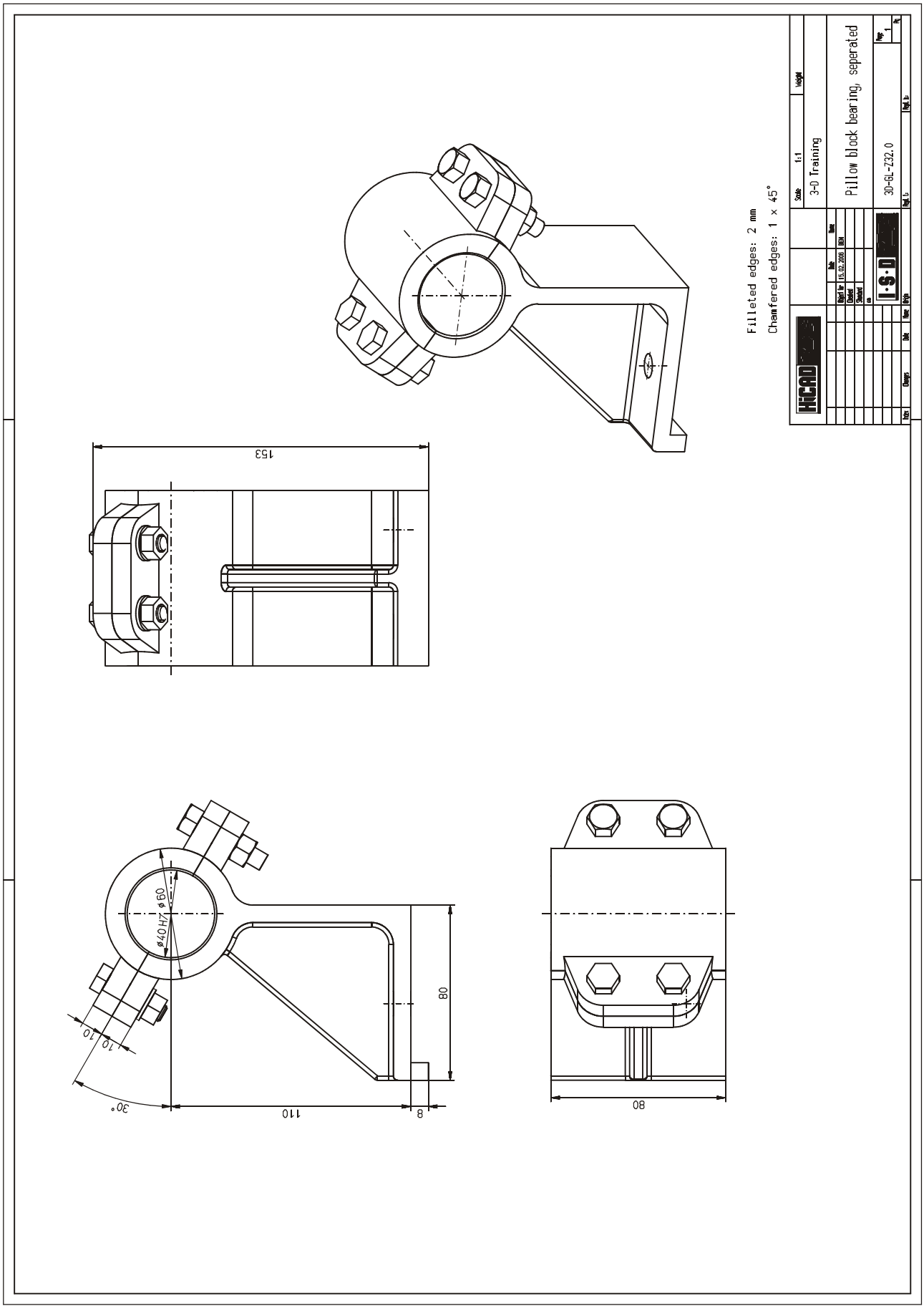
				Checked			
				Standard			
				CAD: C:\HICAD\TRS.SZA			
				I·S·D			3D-GL-Z31.0
							Page 1
							Pg.
Index	Changes	Date	Name	Origin		Repl. f.:	Repl. b.:


Filleted edges: 2 mm
Chamfered edges: 1 x 45°
HiCAD
Scale 1:1
Weight
3-D Training
Pillow block bearing, separated
3D-GL-232.0
I·S·D
153
ø40 H7
ø60
10
10
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30°
110
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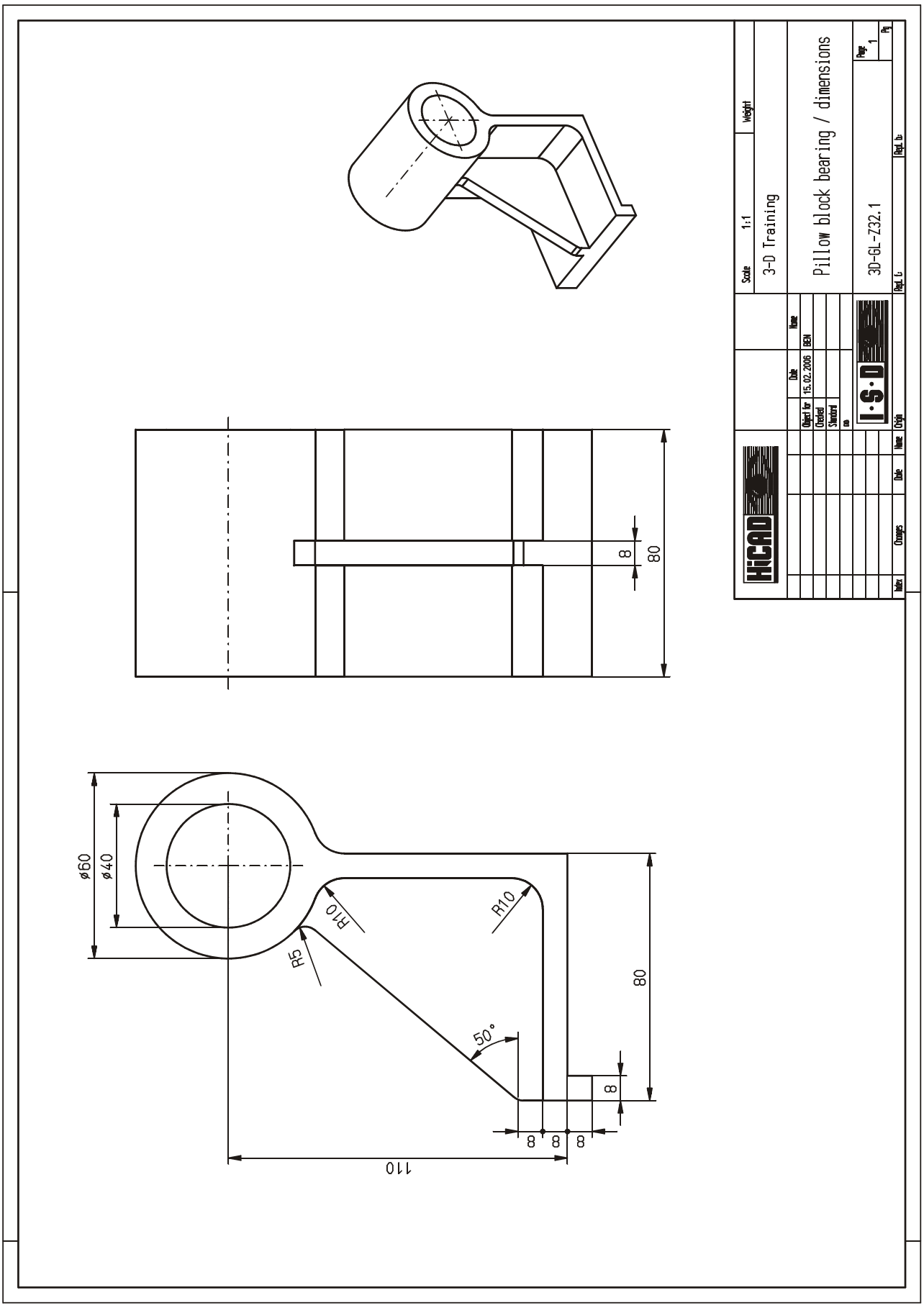
80
80


Scale 1:1
Weight
3-D Training
Pillow block bearing / dimensions
3D-GL-732.1
Page 1
HiCAD
I·S·D
Name
Date
Checked
Standard
15.02.2006
BEN
ø60
ø40
R10
R10
R5
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80
8
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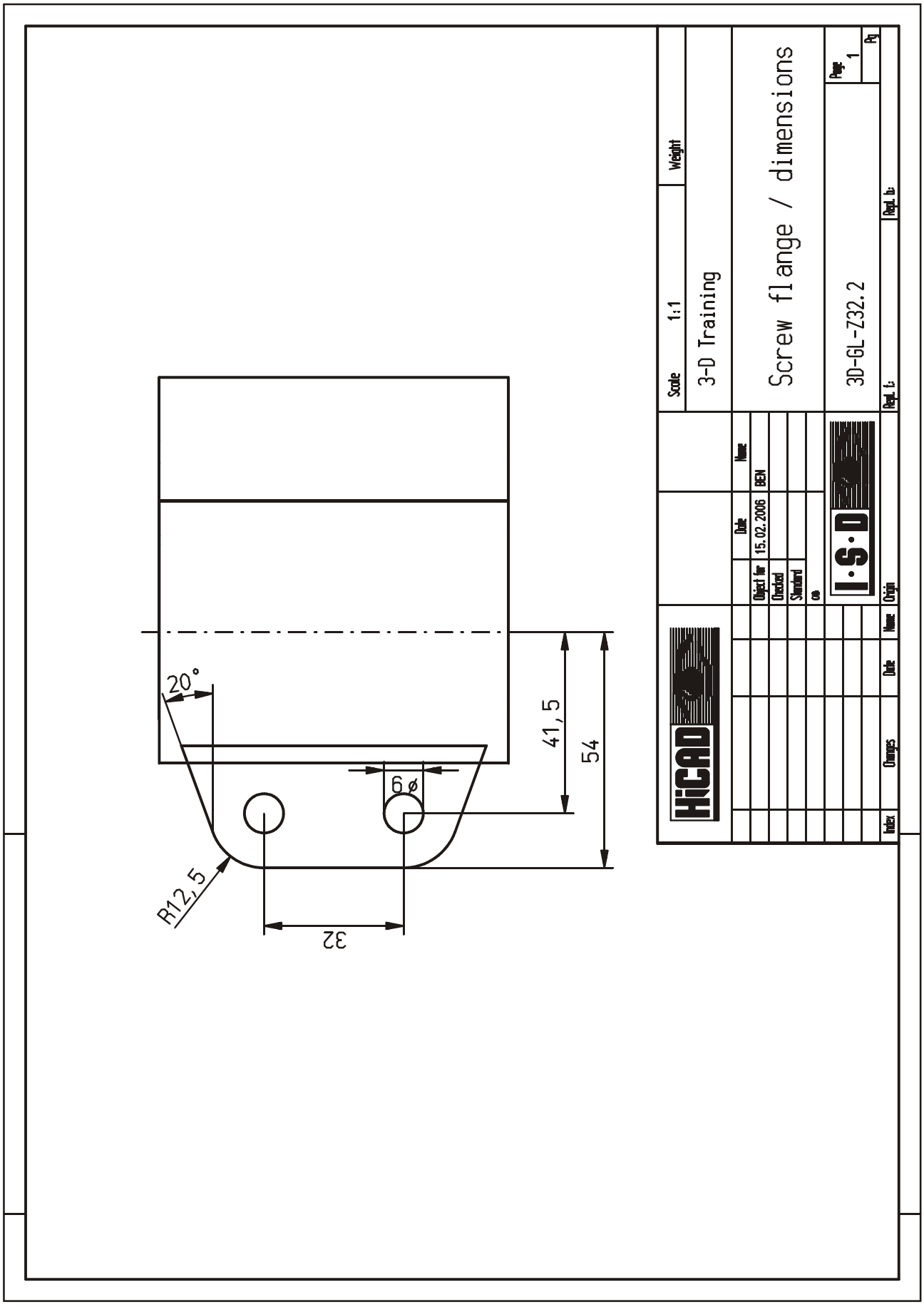
8


20°
41,5
54
6 ø
R12,5
32
Scale 1:1
Weight
3-D Training
Screw flange / dimensions
3D-GL-732.2
Page 1
Pg
Name
Date
15.02.2006
BEN
Object for
Checked
Standard
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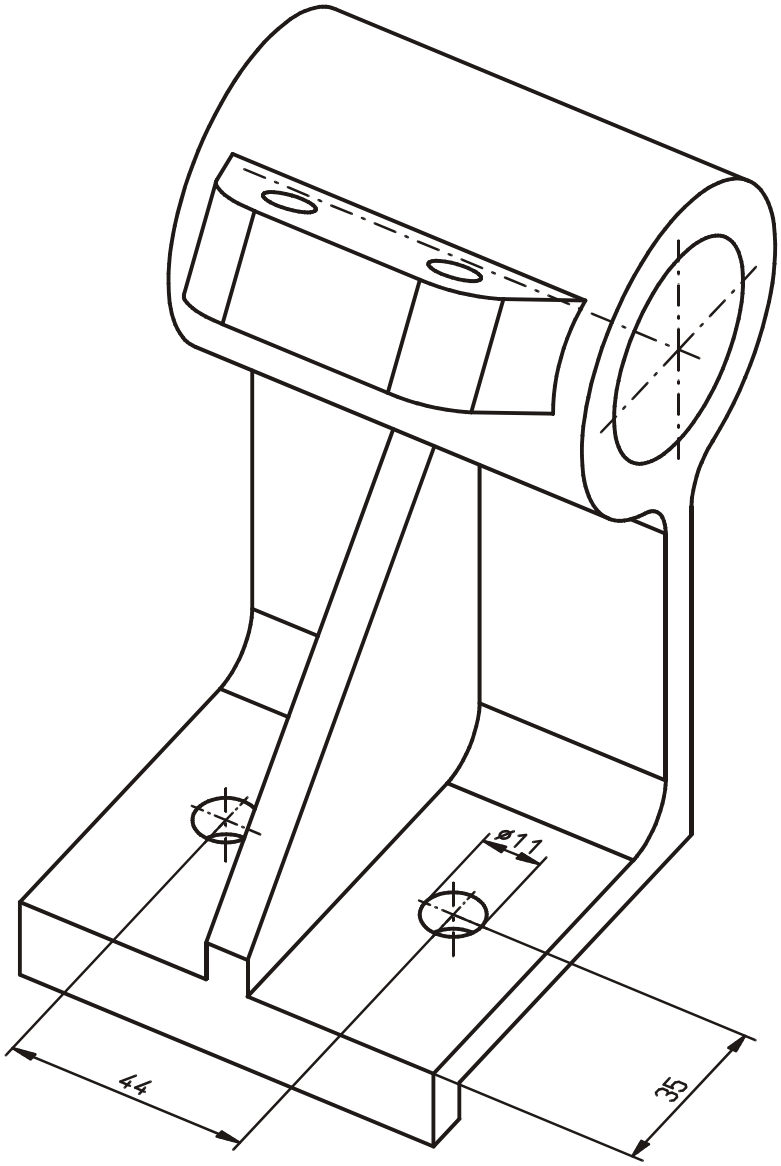
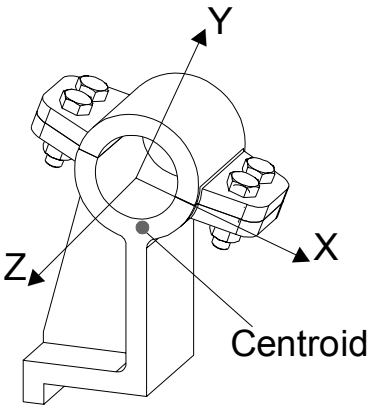
Repl. f.:
Index
Changes
Date
Name
Origin
HiCAD
I·S·D

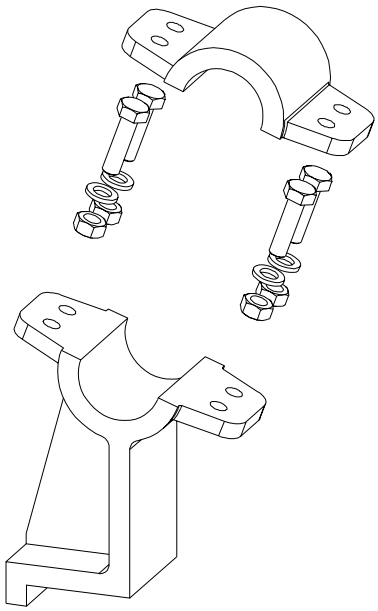


HiCAD					Scale 1:1	Weight	
					3-D Training		
				Date	Name		
				Object for	15.02.2006	BEN	Bearing / flange drilled
				Checked			
				Standard			
				on			
				I·S·D	3D-GL-Z32.3		Page 1 Pg
Index	Changes	Date	Name	Origin	Repl. f.:	Repl. b.:	

a)



b)



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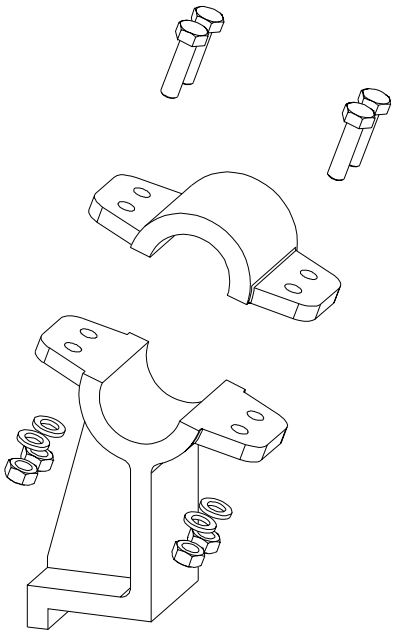
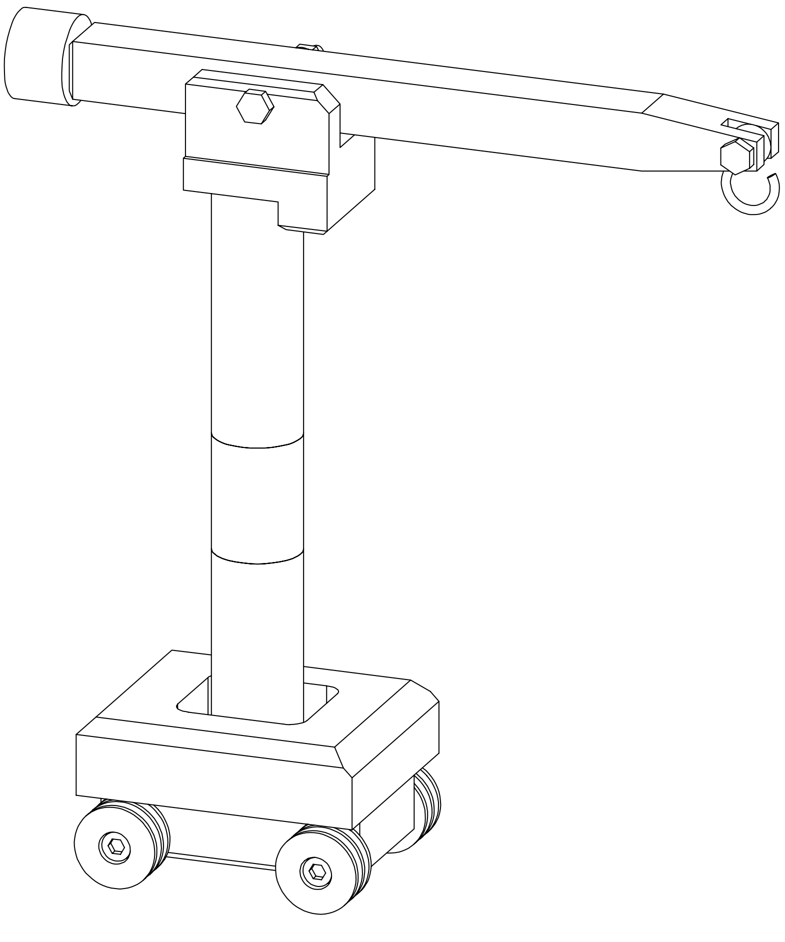
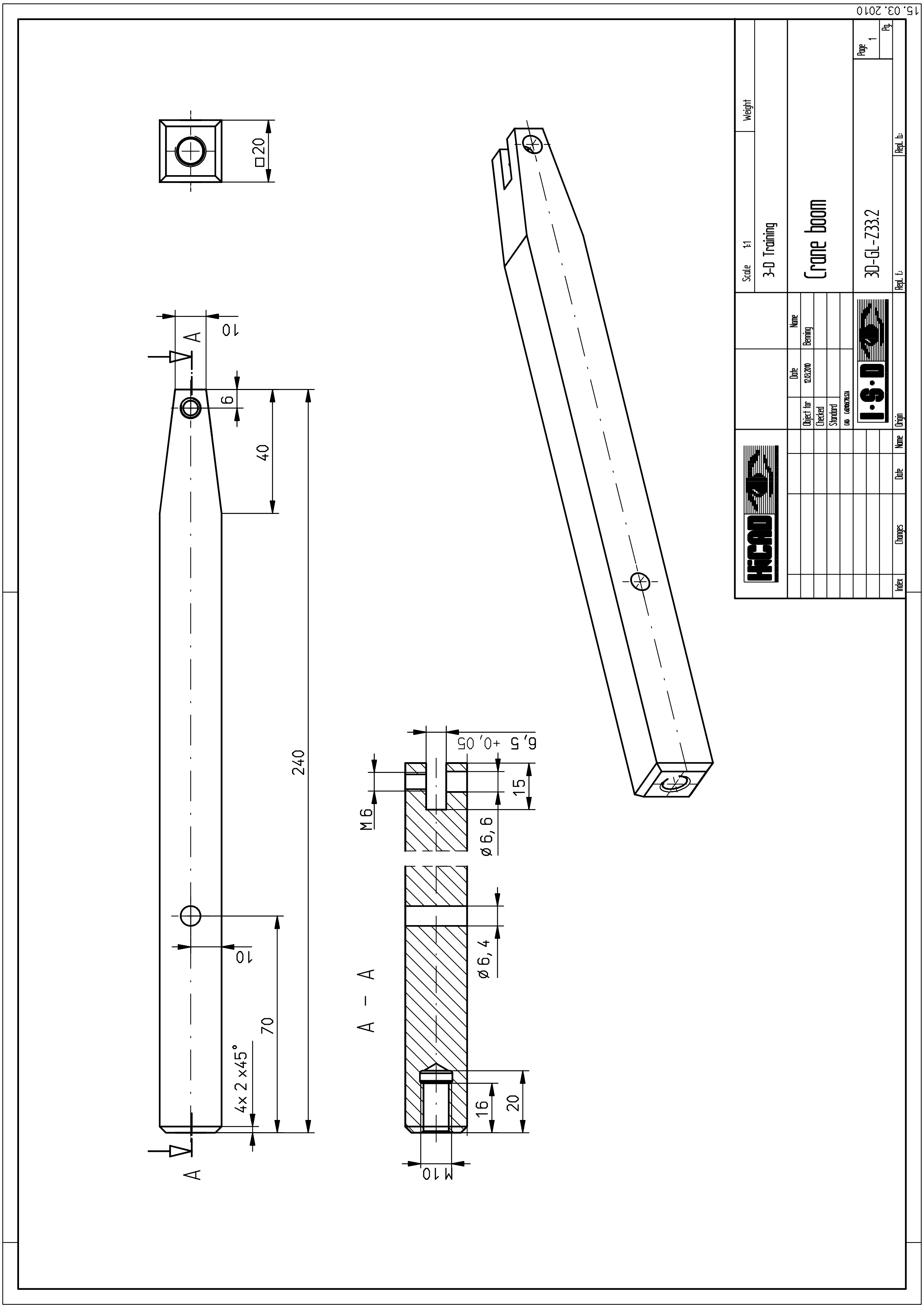
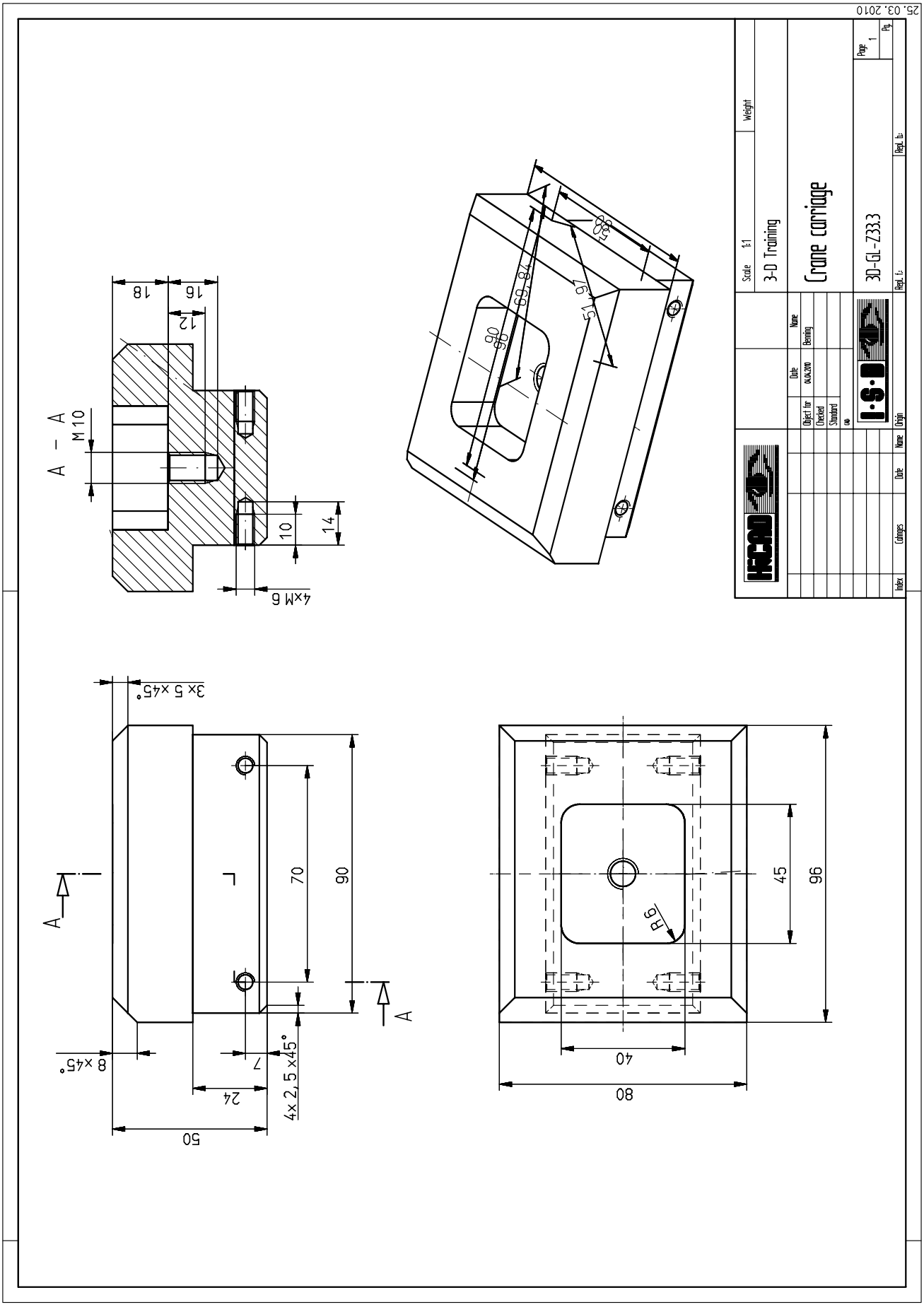
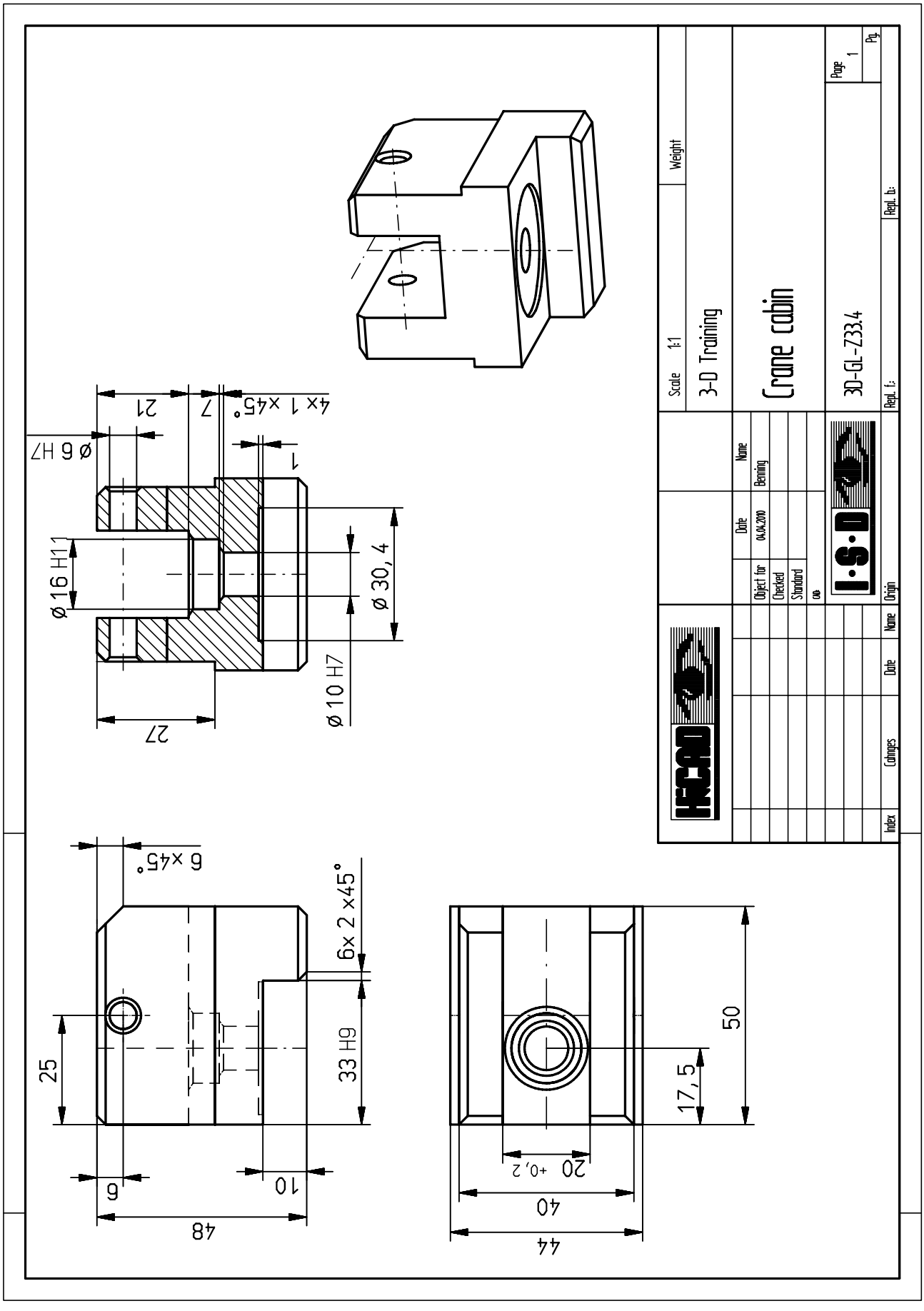




Scale	1:2	Weight	
3-D Training			
	Date	Name	
Object for	12.03.2010	Benning	Crane
Checked			
Standard			
CAD: C6D106C7B.SZA			
			3D-GL-Z33.1
Page 1			
Pg.			
Index	Changes	Date	Name
Origin			
Repl. f.:		Repl. b.:	


15.03.2010
Scale 1:1
Weight
3-D Training
Crane boom
3D-GL-Z33.2
A - A
240
70
40
□20
M 6
M 10
ø 6,4
ø 6,6
6,5 +0,05
4x 2 x45°


25.03.2010
Crane carriage
3D-GL-2333
3-D Training
Scale 1:1
Weight
Page 1
Pg
Name
Date
Bensing
04.04.2010
Object type
Checked
Standard
HiCAD
I·S·D
Index
Changes
Date
Name
Origin
A – A
M10
18
16
12
10
14
4xM 6
90
89,84
87
51
88
3x 5 x45°
70
90
A
8 x45°
7
24
4x 2,5 x45°
50
R9
45
96
40
80


Crane cabin
3-D Training
3D-GL-Z33.4
Scale 1:1
Weight
Page 1
Name
Date
Benning
04.04.2000
Object for
Checked
Standard
Origin
Name
Date
Index
Changes
Repl. to:
Repl. fr:
4x 1 x45°
21
7
1
ø 6 H7
ø 16 H11
ø 30, 4
ø 10 H7
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6x 2 x45°
33 H9
25
6
10
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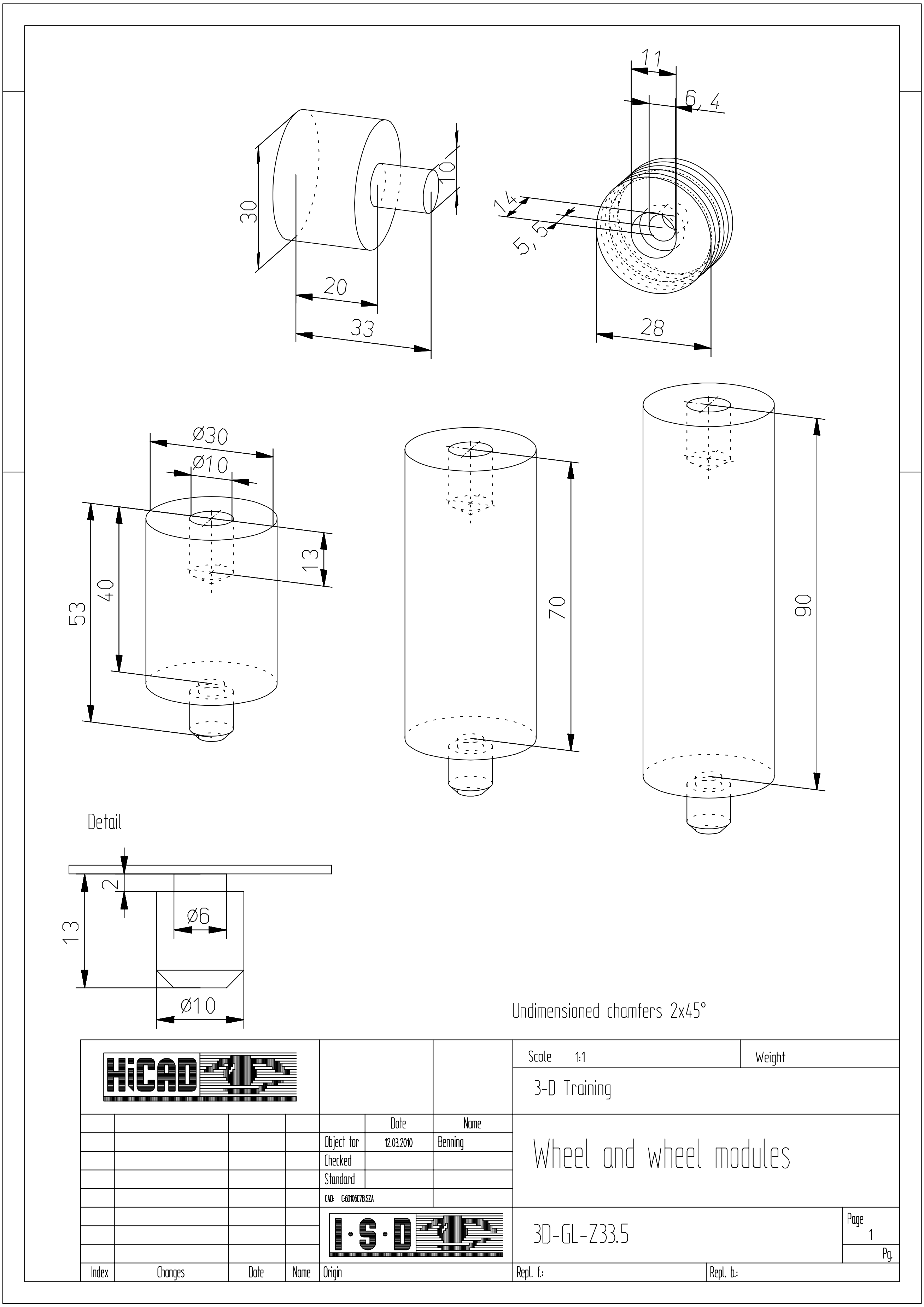
20 +0, 2
40
44


11
6,4
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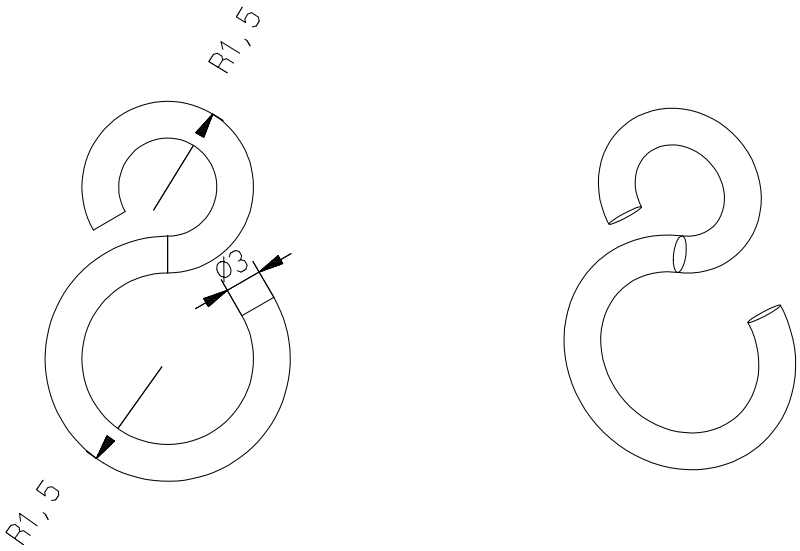
10
14
5,5
20
33
28
Ø30
Ø10
53
40
13
70
90
Detail
2
13
Ø6
Ø10
Undimensioned chamfers 2x45°
HiCAD
Scale 1:1
Weight
3-D Training
Date
Name
Object for 12.03.2010 Benning
Checked
Standard
Wheel and wheel modules
I·S·D
3D-GL-Z33.5
Page 1
Pg.
Index
Changes
Date
Name
Origin
Repl. f.:
Repl. b.:



Hexagon socket bolts acc. to DIN 6912

HiCAD						Scale 2:1	Weight	
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					Date	Name		
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				Checked				
				Standard				
				CAD C:\HICAD\STB\SZA				
				I·S·D			3D-GL-Z33.6	Page 1
								Pg.
Index	Changes	Date	Name	Origin		Repl. f.:	Repl. b.:	

65 Glossary

3-D C-Edge Constraint Manager
The C-Edge Constraint Manager is based on the 3-D HCM and is a tool enabling you to position and move composite edge elements. For this purpose, the c-edge elements are linked to each other through so-called “constraints”, i.e. dimensional and logical relations and restrictions. The C-Edge Constraint Manager then transforms the appropriate elements in such a way that the defined constraints are fulfilled.

3-D Part HCM
Use the 3-D Part HCM to define constraints between 2 objects (e.g. parts) The corresponding elements are linked to each other by dimensional/positional constraints and logical relations. The 3-D Part HCM transforms the corresponding elements in such a way that the specified constraints are fulfilled. *See also 3-D C-Edge HCM.*

3-D Part
3-D parts are normally solid bodies. They consists of 3-D graphical elements (short: “GE”), which constitute the lowest organisational units of a 3-D drawing. These can, for instance, be 3-D points, 3-D (poly)lines, 3-D circles/arcs or surfaces.

3-D Polyline
A 3-D polyline is a structurally coherent (i.e. continuous and uninterrupted) succession of individual 3-D lines. The end point of the previous line coincides with the start point of the next line.

Absolute coordinates
Coordinates referring to the point of origin of the current coordinate system (short: “CS”).

Active attributes
HiCAD enables you to define particular attributes as active attributes. For all manipulations applied at a later point, only those elements possessing these attributes are taken into account.

Arithmetic expressions
Valid algebraic combinations of constants, numeric variables, arithmetic operators and parenthesis.

Assembly
Several combined (“assembled”) parts under one superordinate part. This superordinate part serves only the purpose of structuring (enabling, for instance, a simultaneous moving of the sub-parts below it, or the allocation of part master data to the superordinate part) and can therefore be an empty dummy part, i.e. it needs not necessarily contain lines, dimensionings or similar.

Associativity
“2-D/3-D Associativity” basically describes the possibility of a combined and simultaneous working in 2-D and 3-D. A HiCAD drawing files can both contain 2-D and 3-D objects. 3-D views can be added to technical drawings in 2-D, 2-D objects can be used as a basis for 3-D drawings etc.

Attributes
Characteristics such as colour, line type and line thickness, influencing the appearance and shape of elements.

Autopilot
A tool for the identification of points and parts. If the Autopilot is switched on, the cursor provides information on snappable points (e.g. start points, end points or mid points of edges, perpendicular base points, intersection points or centre points of circles) when it is moved over the drawing objects. These points can be taken over directly, i.e. without activating a point option.

Basic elements
The lowest organisational units of a HiCAD drawing. In 2-D, these are line elements, in 3-D, these are edges, surfaces and 3-D points.

Bill of Materials (BOM)
List of all parts of which a product or an assembly is made up, indicating quantities and designations of the parts.

C-Edge Constraint Manager
see 3-D C-Edge Constraint Manager

Clone
Parts which are required several times in a drawing can be copied. These copies are called “clones” in HiCAD. You can choose whether you want to combine all parts into one part or if you want to create one superordinate part above the copied parts instead.

Closed contour
Continuous contour with coinciding start and end point.

Colour table
Enables you to pre-set the allocation of available colours.

Contour
Contours link graphic elements that are optically but not structurally coherent – they need not belong to the same part.

Contour edges
“Apparent” edges of the analytical 3-D model.

Contour hatching
Hatches a surface defined by a contour. Lines which are integrated in the boundary line can belong to different parts.

Copy
see Clone

Crosshairs
Cursor symbol which is moved by the mouse, indicating the current position on the screen.

Cursor
See Crosshairs

Data structure
The HiCAD drawing structure. The highest organisational unit in HiCAD is the **Drawing**. The Drawing contains **Main parts** which may in turn consist of **Sub-parts**. The individual parts consist of the lowest organisational units: In 2-D, these are the graphic elements (GE) and in 3-D, these are edges, surfaces and 3-D points. *See also 2-D part, 3-D part, Drawing.*

Default value
The value of a parameter which is set when the program is started.

Detail
An arbitrary drawing detail can be defined and inserted separately on the screen in any scale.

Display lists
Graphic card drivers for an accelerated output of graphic elements on the screen.

Drawing
Drawing file in HiCAD with the file extension .SZA. A Drawing is the highest organisational unit in HiCAD. It is composed of 2-D and/or 3-D parts. These can be assemblies, parts or bores. Each drawing is assigned an unambiguous name, enabling the user to retrieve it.

Drawing detail
The detail of a drawing that is displayed on the screen (alterable by the “Rectangular zoom window” and “View all” functions).

Drawing layout
Several HiCAD drawings on one drawing sheet for print output.

Engineering axonometry
A parallel projection, in which the y and z axes are full length and the x axis shortened. The x axis is at an angle of approx. 41°, and the y axis at an angle of approx. 7° to the horizontal of the screen plane. The z axis is perpendicular to the screen plane.

Error tolerance
Rounding errors in numerical values. Point coordinates, e.g., can be checked for a specific tolerance. Errors are calculated as the difference between the exact and the rounded value. The error tolerance, i.e. the allowed maximum deviation, can be preset within a specific range of values. Two points are recognised as one if the distance between them is less than the given value.

ESM (European Solid Modeler)
ISD’s self-developed 3-D kernel.

Feature
When designing in 3-D, Feature Technology is active by default. Feature Technology records all processing steps (“Features”) in a Feature log. Activate the Feature tab in the ICN to view the processing steps applied to the active part. With the help of Feature Technology you are enabled to conveniently change parts via the Feature log, and even correct processing steps that were applied long ago, without having to repeat these steps once again. You can also derive Feature variants from the Feature log. If Feature Technology is switched on, HiCAD automatically generates the parametric dimensions of the part. These are visible if the part and the Feature step are active. Such parametric dimensions provide, besides the Feature log, an additional option to change parts. *See also Parametric dimensions.*

File group
Short term for a HiCAD directory path, defined in the FILEGRUP.DAT file.

File name
The name under which the operating system (e.g. Windows or UNIX) manages the HiCAD data, such as CAD-drawings, parts, macros etc.

File path
A short designation for a HiCAD directory path. Paths are defined in the FILEGRUP.DAT file.

Fitting points
Points used for reference, e.g. when transforming parts.

Folder
Folders can be used in HELiOS to simulate and manage order processes. You can save all documents and parts of an order to a folder. One distinguishes between mandatory and optional objects. Mandatory objects are requested by Folder Workflows, optional objects can be created or dragged into the element list (with Drag & Drop or similar functions), if desired. You can create a Folder Workflow for a Folder: In such a workflow you can, for instance, define the steps required for the processing of an order. For example, if you have determined for the Workflow step “Request check-up” that a document named “Test Report” would be required, this configuration would automatically lead to the opening of a document mask in this Workflow step. Alternatively, you can also search for existing documents or parts. Documents/Parts in the Folder or in related links are listed in further tabs.

Freehand symbol technique
Special input method based on the definition of a symbol and assigning it to a specific command.

Graphic elements (GE)
Lowest organisational 2-D unit. Graphic elements can be points, symbols, lines, circles/arcs or conic sections.

HCM HiCAD Constraint Manager
Parametric designing on the basis of Constraints, linking individual objects to each other by means of geometrical relations.

Identical part search
Automatic detection of parts with an identical geometry in the drawing.

Identification
Selection of an object with the cursor.

Intelligent Cursor
Context-sensitive cursor providing you with functions appropriate to current processing requirements. If you process objects, for instance, you are prompted to select the object you want to process. When you now move the cursor on an object, this object is highlighted and can be selected for processing by a mouse-click. Only those functions which are useful for the processing of this object are then displayed.

Interfaces
Software modules enabling data interchange with other programs.

Isolated points
Independent points that can be inserted freely into the drawing. They need not be located on a drawing object. Isolated points can be used as auxiliary point during drawing creation, or as fitting points during transformations.

Kinematics
Functions for modelling, simulating and analysing complex freely definable linkages.

Kinetostatics
Analysis function for bearing and element stress.

Layer
The Working with layers can be compared to the working with transparent foils on a conventional drawing board.

Level
see Overlap and Visualisation level.

Library
A “pool” of parts (part library), macros or variants (macro library). A library can be added to the screen menu. In this case, the calling of the parts, macros or variants takes place via selection of the appropriate library entry.

Local 3-D coordinate system
Temporary 3-D coordinate system, which can also be rotated.

Logical relational expression
Simple arithmetic comparison, e.g. a > b (a is greater than b). The value of the expression is TRUE or FALSE.

Logical variables
Variables which always return either TRUE or FALSE as a result. They are frequently used in macro technology, especially in loop and IF conditions.

Macro
Sub-program accessing HiCAD functions and creating variants in cooperation with the user. Macros serve the purpose of reproduction, i.e. the repetition of permanently returning HiCAD processes.

Main part
The second-highest organisational unit after the Drawing. A Main part can be an Assembly or an individual part. An unlimited number of Sub-parts can be assigned to a Main part to provide the logical structure of a real construction. *See also Part structure and Sub-part.*

Mask
A form that is temporarily displayed for the input or selection of data. Masks usually contain input fields or selection fields for parameter specification, as well as explanatory texts.

Model area
A view in which all other views, sectional views, detail views, drawing frames etc. are hidden. This enables a fast and structured working. Besides, you can rotate the model view without any repercussions on the production views in the sheet area. *See also Sheet area.*

Mouse
A device that is manually moved over the table, thus moving and controlling the crosshairs on the screen.

Moving of objects
Simultaneous translation and rotation of objects.

Named points
Points to which a name has been assigned. This is possible for start points and end points of lines and arcs, as well as for isolated points.

Natural coordinates
All 2-D data contained in the drawing refer to the carthesic coordinate system. These coordinates are also referred to as natural coordinates.

Object selection
A choosing of objects for processing, either via identification, name entry, or with the structure browser.

Overlap contour
The Overlap contour defines the overlap area in 2-D. This is the area in which the overlapped lines are to be shown or hidden. *See also Overlaps and Overlap level.*

Overlap level
You can also create overlaps in 2-D by assigning so-called “overlap levels” to parts. *See also Overlap contour and Overlaps.*

Overlaps
You can create overlaps in 2-D by assigning a so-called “overlap level” to a 2-D part. *See also Overlap contour and Overlap level.*

Parametric dimension
Dimensioning appearing on the active part if the Feature is switched on. If you right-click a dimensioning, you are enabled to change the part. *See also Feature.*

Part
A drawing consists of very many graphical objects, such as lines, texts, dimensionings, etc. All of these graphical objects are assigned to parts. One part can, for instance, be called “Flange”, another part can be called “Plate”. All lines, dimensionings, texts and processing symbols of the flange are assigned to the part “Flange”, and all lines, dimensionings etc. of the plate are assigned to the part “Plate”. If you translate, rotate, mirror or delete such a part, all dimensionings, texts and processing symbols are also translated, rotated, mirrored or deleted. If you create a part as main part or sub-part depends on whether the part is an individual, independent part, or if it belongs to another part. For instance, an individual haunched plate in a detail drawing would, as an independent part, be created as a main part. If this haunched plate is welded to a steel beam in an assembly drawing, it would be a sub-part of the beam. *See also Main part, Sub-part.*

Part hatching
Hatching of a surface bordered by a closed polyline. All lines of the border need to belong to one part.

Part list
Multiple selection of parts. The aim is to process several parts in one step

Part structure
Subdivision of the drawing into main parts and sub-parts. The part structure describes the logical composition of the real construction. It constitutes, for instance, the basis for the automatic Bill of Materials (BOM), the automatic creation of subtractions for part hatchings, and part overlaps.

Patterning
Special hatching of a surface, with arbitrary symbols instead of lines for the hatching.

Pixel graphic
An image the data structure of which consists of pixels. Pixel graphics are created, for example, when scanning images. Common formats are BMP, JGP, GIF, PCX and TIF.

Point options
Functions for the specification of point coordinates. The functions are subordinated to all other functions, i.e. they can be activated within other functions.

Point specification
See Point options.

Polyline
A polyline is a structurally coherent (i.e. continuous and uninterrupted) succession of individual lines, circular, elliptical, hyperbolic or parabolic curves. The end point of the previous line coincides with the start point of the next line.

Redo
Reverses the previous UNDO.

Reference point
The point to which a later subsequent transformation or point specification refers. *See also Fitting point.*

Relative coordinates
The coordinates refer to a point in the drawing that is not necessarily the origin of the coordinate system. For many functions, this reference point is automatically defined by the last set point. Reference points can however also be defined by the user.

Representation
HiCAD offers several options for the representation of the 3-D parts in a drawing, e.g. shaded, as glass model, with or without hidden edges.

Roles
In contrast to the already known User/Group Management via edbsetup.exe, you can, in conjunction with Role Workflows, create so-called “Roles”. Such “Roles” are a combination of special rights within the Role Workflows. You can create different Roles and assign different (already existing) users or entire groups (from the normal User and Group Management) to them. The Role Management enables you to provide Users and/or Groups with different/other rights than those which are already available via the normal User/Group Management. The already existing rights remain unaffected and continue to exist. Within the Role Workflows, the rights can now be customised and can be used in addition to the, still available, existing rights.

Role (as opposed to Group)
In contrast to the User/Group concept, the utilisation of Roles enables a dynamic management of rights. Access rights are defined in HELiOS for individual Roles, and the access rights are given to Users by assigning them to specific Roles. The allocation of Users to Roles needs not necessarily be unique, central and system-wide, but can take place at the moment of the creation of an object possessing a Workflow, or even stepwise, during the lifecycle of the object. This means that access rights of users can be controlled individually on object level and via the User<->Role allocation. In addition, you are enabled to change the allocation of Users to Roles during the execution of a Role Workflow.

Screen coordinate system (Screen CS)
Coordinate system the XY-plane of which is the screen.

Screen menu
The area(s) of the graphic screen containing the icons, symbols and texts for function calls.

Screen plane
The XY-plane of the screen coordinate system.

Screenshot
Saving of the current screen or screen details as pixel graphic.

Sheet area
The area in which you can create and name production views. These can be normal views, sectional views, detail views, drawing frames etc. You can also create several sheet areas, e.g. one for the quotation drawing, one for the production drawing etc. *See also Model area.*

Similarity transformation
Simultaneous translation, rotation and scaling of a part.

Sketches
3-D part with free edges that lies in a plane. Sketches can be used, e.g., for the derivation of extruded solids, revolved solids, bores, subtractions or C-edge sweeps. The name “3-D SKETCH” can be changed if desired. After creating a sketch the appropriate sketch functions are displayed. The sketch is drawn in the active processing plane or, if no processing plane is available, in the XY-plane of the active coordinate system. Before drawing a closed contour in the sketch, you can use the “New plane, …“ functions to define or activate a different processing plane.

Snap circle
If snap mode is active, points within the radius of the displayed circle are automatically detected.

Snap mode
If this mode is active, points can be specified directly, without calling a point option. *See also Autopilot.*

Snap radius
Determines the size of the snap circle.

Solid primitives
Basic 3-D bodies such as cuboids, cylinders, spheres etc.

Status bar
Displays information about the current program status, i.e. the name of the active part, active view and active coordinate system, etc.

String
A sequence of alphanumeric and special characters.

Structure browser
Offers a graphic representation of parts as well as the opportunity of processing component structure.

Sub-part
A part that is subordinated to another part (Main part or Sub-part). *See also Part structure and Main part.*

System variable
HiCAD variables with fixed allocations.

Transformation
Umbrella term for positional and size changes of drawing parts, such as translations, rotations, mirrorings, enlargements, downsizings etc.

Trim
Limits or shortens lines with respect to given conditions.

Undo
Reverses the previous action.

Unit of measurement
Preset unit of measurement (mm, cm, etc.) for all value entries.

User-specific variable
Variables which have been defined by the user and to which, depending on the type, a numerical value or a character string has been assigned.

Value entry
Input of numbers, arithmetic expressions and variables.

Variables memory
Function enabling a rapid definition and saving of variables.

Variant
You use Variant Technology to create “Variants”. The creation of 2-D or 3-D variants makes sense if you frequently draw similar parts, differing only in their dimensions, the number of bores, etc. After calling a variant you only need to specify the required parameters (width, height, number of bores etc.), and HiCAD then draws the part automatically. Parts created with Variants (or Macros) can be simple sheets or plates, but also complex assemblies.

Vector graphic
An image constructed with vectors. In this type of graphic you can delete, move, change or scale the individual graphic elements. Vector graphics usually require less memory than pixel graphics.

Views
Projections, i.e. two-dimensional representations of 3-D objects. One distinguished between standard views (top view, side view, front view), isometric and axonometric views. View generation is based on projection principles.

View All
Displays the entire drawing on the screen.

Window
The subdivision of the screen in several processing windows, enabling you to process various details of a drawing.

Zoom
Functions enabling you to enlarge or downsize the objects on the screen.

ZTL file
A 2-D drawing file with the extension ZTL; used in older HiCAD versions.

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