Version
September 2013

Add-on Module

HSS

Load-bearing Capacity of Welded Hollow Section Connections
According to EC 3

Program Description

All rights, including those of translations, are reserved.
No portion of this book may be reproduced – mechanically, electronically, or by any other means, including photocopying – without written permission of Dlubal Software GmbH.

© Dlubal Software GmbH
Am Zellweg 2 D-93464 Tiefenbach

Tel.: +49 (0) 9673 9203-0
Fax: +49 (0) 9673 9203-51
E-mail: info@dlubal.com
Web: www.dlubal.com
# Contents

<table>
<thead>
<tr>
<th>Contents</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Introduction</td>
<td>4</td>
</tr>
<tr>
<td>1.1 Add-on Module HSS</td>
<td>4</td>
</tr>
<tr>
<td>1.2 HSS Team</td>
<td>5</td>
</tr>
<tr>
<td>1.3 Using the Manual</td>
<td>5</td>
</tr>
<tr>
<td>1.4 Open the Add-on Module HSS</td>
<td>6</td>
</tr>
<tr>
<td>2. Input Data</td>
<td>7</td>
</tr>
<tr>
<td>2.1 General Data</td>
<td>7</td>
</tr>
<tr>
<td>2.2 Cross-sections</td>
<td>10</td>
</tr>
<tr>
<td>2.3 Geometry</td>
<td>13</td>
</tr>
<tr>
<td>2.4 Loads</td>
<td>14</td>
</tr>
<tr>
<td>2.4.1 Defining Loads Manually</td>
<td>15</td>
</tr>
<tr>
<td>3. Calculation</td>
<td>16</td>
</tr>
<tr>
<td>4. Results</td>
<td>17</td>
</tr>
<tr>
<td>4.1 Design by Nodes</td>
<td>17</td>
</tr>
<tr>
<td>4.2 Results Evaluation</td>
<td>18</td>
</tr>
<tr>
<td>5. Printout</td>
<td>20</td>
</tr>
<tr>
<td>5.1 Printout Report</td>
<td>20</td>
</tr>
<tr>
<td>6. General Functions</td>
<td>21</td>
</tr>
<tr>
<td>6.1 HSS Design Cases</td>
<td>21</td>
</tr>
<tr>
<td>6.2 Units and Decimal Places</td>
<td>23</td>
</tr>
<tr>
<td>6.3 Export of Results</td>
<td>23</td>
</tr>
<tr>
<td>A Literature</td>
<td>25</td>
</tr>
<tr>
<td>B Index</td>
<td>26</td>
</tr>
</tbody>
</table>
1. Introduction

1.1 Add-on Module HSS

DLUBAL SOFTWARE GMBH offers this powerful program to civil and structural engineers, providing them with a tool that they can use to design planar and spatial truss nodes consisting of hollow sections. HSS is able to calculate connections with round, square and rectangular hollow sections.

Checking the geometric conditions to be kept is essential for designing hollow section connections successfully. Various types of nodes are assigned by their geometric characteristics like overlaps or gaps between struts to different design types in the program. The program's automatic geometry recognition makes it easier to lay out node connections in an optimal way.

HSS performs the ultimate limit state design according to specifications described in the European Standard EN 1993-1-8:2005. The ultimate limit state of the designed connections is shown as the maximum load bearing capacity of the struts of a framework for axial forces and bending moments.

The add-on module HSS is not only an integral part of RSTAB in its appearance. Like other add-on modules, HSS is completely integrated in the main program. The module results can be included in the global printout report. In this way, both is possible: to easily perform and present the entire calculation in a consistent form.

Finally, separate design cases allow for a flexible analysis of individual structural components built in complex truss systems.

We hope you will enjoy working with HSS.

Your team from DLUBAL SOFTWARE GMBH
1 Introduction

1.2 HSS Team

The following people were involved in the development of HSS:

Program coordination
Dipl.-Ing. Georg Dlubal
Dipl.-Ing. (FH) Younes El Frem

Programming
Jan Fenár
Dipl.-Ing. Georg Dlubal
Ing. Milan Jaroš
Dipl.-Ing. Georg Dlubal

Cross-section and material database
Ing. Ph.D. Jan Rybin

Program design, dialog figures, and icons
Dipl.-Ing. Georg Dlubal
Ing. Jan Miléř
MgA. Robert Kolouch

Program supervision
Ing. František Knobloch
Dipl.-Ing. (FH) René Flori

Manual and translations
Ing. Jana Duníková
Ing. Jan Miléř
Ing. Ladislav Kábrt

Technical support and quality management
Dipl.-Ing. (BA) Markus Baumgärtel
Dipl.-Ing. (FH) Bastian Kuhn
Dipl.-Ing. Moritz Bertram
Dipl.-Ing. (FH) Ulrich Lex
Dipl.-Ing. (FH) Steffen Clauß
Dipl.-Ing. (BA) Sandy Matula
Dipl.-Ing. (FH) Frank Faulstich
M.Eng. Dipl.-Ing. (BA) Andreas Niemeier
Dipl.-Ing. (FH) René Flori
M.Eng. Dipl.-Ing. (FH) Walter Rustler
Dipl.-Ing. (FH) Stefan Frenzel
M.Sc. Dipl.-Ing. (FH) Frank Sonntag
Dipl.-Ing. (FH) Walter Fröhlich
Dipl.-Ing. (FH) Lukas Sühnel
Dipl.-Ing. (FH) Wieland Götzler
Dipl.-Ing. (FH) Robert Vogl

1.3 Using the Manual

Topics like installation, graphical user interface, results evaluation and printout are described in detail in the manual of the main program RSTAB. The present manual focuses on typical features of the HSS add-on module.

The descriptions in this manual follow the sequence of the module's input and results windows as well as their structure. The text of the manual shows the described buttons in square brackets, for example [Edit]. At the same time, they are shown on the left margin. In addition, expressions used in dialog boxes, tables and menus are set in italics to clarify the explanations.

Finally, you find an index at the end of the manual. However, if you don’t find what you are looking for, please check our website www.dlubal.com where you can go through our FAQ pages by selecting particular criteria.
1.4 Open the Add-on Module HSS

RSTAB provides the following options to start the add-on module HSS.

**Menu**

To start the program in the menu bar,

point to **Connections** on the **Add-on Modules** menu, and then select **HSS**.

![Figure 1.1: Menu: Add-on Modules → Connections → HSS](image1.png)

**Navigator**

To start HSS in the **Data** navigator,

select **HSS** in the **Add-on Modules** folder.

![Figure 1.2: Data navigator: Add-on Modules → HSS](image2.png)
2. Input Data

Data for defining design cases is entered in module windows.

After you have started HSS, a new window appears with a navigator on the left showing you all module windows that you can select. The pull-down list above the navigator contains the design cases that are already available (see chapter 6.1, page 21).

When you start HSS for the first time, only window 1.1 General Data is active. After entering a node number or using the [Select] function to select a node of the model, HSS imports the following data from RSTAB:

- Geometry
- Cross-section type
- Connection type
- Cross-sections
- Material

To select a window, click the corresponding entry in the HSS navigator, or page through the windows by using the buttons shown on the left. You can also use the function keys [F2] and [F3] to select the previous or subsequent window.

To save the entered data and quit HSS, click [OK]. To exit the add-on module without saving the data, click [Cancel].

2.1 General Data

In window 1.1 General Data (see Figure 2.1), you define the nodes that you want to design. Only geometrically identical connections can be calculated in a design case (see 6.1, page 21). HSS checks the cross-sections and geometry data of the connected members.

Geometry data includes the number of connected members and the angle enclosed by the chord and members. With HSS you can design planar and spatial truss nodes as represented in EN 1993-1-8:2005, figure 7.1.
2 Input Data

Connections on nodes
The nodes that you want to design can be entered manually into the input field. You can also use the [Select] button to select the nodes graphically in the RSTAB work window.

National annex (NA)
Use this selection field to set the national annex according to which you want to apply the parameters for the design.

Click the [Edit] button to access detail settings of the national annex that is set. A separate dialog box opens showing the partial safety factors that are valid for the selected national annex. It is not possible to change the parameters.

With the [New] button you can create a user-defined national annex where the partial safety factors can be adjusted individually.

To remove a user-defined annex, use the [Delete] button.
2 Input Data

Geometry

This window section defines the type of connection with regard to the position of the connected members (struts) in space. HSS is able to recognize the connection type automatically. The meaning of Planar and Spatial is made clear in the figures below.

![Planar and Spatial Connections](image)

Figure 2.3: Representations for planar and spatial connections

The setting in this window section affects the appearance of window 1.3 Geometry described in chapter 2.3 on page 13.

Hollow section

After selecting the node, HSS shows you the cross-section types recognized as available in the RSTAB model. In accordance with EN 1993-1-8, you can only design sections of similar types connected on one node. Therefore, HSS checks the model data during the import: If unacceptable cross-sections like round and square hollow sections are detected together on one node, the add-on module displays the following error message immediately after starting the program.

![Error Message](image)

Figure 2.4: Error message for unacceptable cross-sections on a node

In this case, it is recommended to adjust the RSTAB model and to import the node to HSS again.

Input data

If you want to analyze a node irrespective of the data from RSTAB, you can enter input data for HSS with the option Define manually. In this case, window 1.3 Geometry is hidden, and window 1.4 Loads is adjusted accordingly (see Figure 2.11, page 15).

Type of connection

The node type recognized by HSS is set. The standard describes seven different types of connection.

Criteria for the assignment are the number of connected struts, the angle enclosed by the chord member and the struts as well as the arrangement among each other. The current type is shown in a small template graphic to the right. When you define the geometry data manually, you can also select the connection type in the pull-down list with the thumbnails.
2.2 Cross-sections

When the automatic data import from RSTAB has been set, the recognized cross-sections of the chord and struts are listed in module window 1.2 *Cross-Sections*.

![Window 1.2 Cross-Sections](image)

With the button shown on the left you can select a different section from the [Cross-Section Library]. Please note that the library enables only those cross-section types that have been set below *Hollow Section* in module window 1.1.

The [Edit] button allows for direct access to the cross-section table of the currently set cross-section. In the table, you can select another circular section that will be used for the design. During the transfer HSS checks certain limiting criteria that are given by the connection's geometry: For example, an information appears if a chord section has smaller dimensions or wall thicknesses than a strut section.

![RSTAB Error No. 1303](image)

In case of rectangular chord sections, it is possible to rotate the cross-section by 90° to adjust the geometric conditions between chord and strut.
Info about cross-section

By using the [Info] button you can display additional cross-section information. A dialog box with design-relevant cross-section properties appears. In addition, it is possible to look at the cross-section's Stress points and (c/t)-Parts used for the stress analysis and the width-to-thickness analysis. The c/t analysis according to EN 1993-1-8 is carried out during checking the range of validity. Here, only cross-sections that can be assigned to class 1 or 2 are allowed.

Figure 2.7: Dialog box Info About Cross-Section

In the right part of the dialog box, the currently selected cross-section is displayed graphically. The buttons below have the following functions:

<table>
<thead>
<tr>
<th>Button</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>![Display stress points]</td>
<td>Displays or hides the stress points.</td>
</tr>
<tr>
<td>![Display (c/t) cross-section parts]</td>
<td>Displays or hides the (c/t) cross-section parts.</td>
</tr>
<tr>
<td>![Display numbering]</td>
<td>Displays or hides the numbering of stress points or (c/t) cross-section parts.</td>
</tr>
<tr>
<td>![Display details]</td>
<td>Displays or hides the details of stress points or (c/t) cross-section parts.</td>
</tr>
<tr>
<td>![Display dimensions]</td>
<td>Displays or hides the dimensions of the cross-section.</td>
</tr>
<tr>
<td>![Display principal axes]</td>
<td>Displays or hides the principal axes of the cross-section.</td>
</tr>
</tbody>
</table>

Table 2.1: Buttons of cross-section graphic
Materials
This section of window 1.2 lists the materials imported from RSTAB. In accordance with EN 1993-1-8, clause 7.1.1 (4), only steel grades with a yield strength of not more than 460 N/mm² are allowed. Use the list to change the material. If an unacceptable material was defined in RSTAB, an error message appears when you open HSS.

Use the [Info] button to look at the material properties in detail.

![Figure 2.8: Dialog box with material properties](image)

For square and rectangular sections you can also define the type of steel production. The setting affects the imperfection factor $\alpha$ according to EN 1993-1-1, table 6.1 used for the determination of the slenderness ratio for flexural buckling.

Member assignment
Below the window graphic on the right, you see a table overview listing the members that are connected on the selected node. Use the [Select] buttons below the structural components to look at them in the graphic window of RSTAB.

Use the [Delete] button to delete the entries in the Member Assignment table including the node defined in module window 1.1. Then, a query appears asking you to select a new node.

If input data from RSTAB is not imported, you have to define the cross-section properties of all members manually by means of the cross-section library. Then, the materials can be defined by using the lists. The window section for Member Assignment is not available when cross-sections are defined manually.
2.3 Geometry

Module window 1.3 is displayed if the import of input data from RSTAB has been set in window 1.1 General Data, or if entries for a manual definition of the connection are already available in window 1.2 Cross-Sections.

Angles between struts and chord member

The angles enclosed by the struts and the chord member are set in this window section. The angle values can be manually adjusted: Select a value and replace it by another. A value definition using the integrated calculator or the Formula Editor is also possible: Click the button \([\text{]}\) shown on the left. The context menu opens offering the mentioned functions. In addition, it is possible to specify data imported from RSTAB with Full Precision.

Gap/overlap and eccentricity

The struts’ geometric relation to each other is important for selecting the appropriate design. The relation is specified by defining a gap between the struts or an overlap of the struts. The position of the struts in RSTAB is defined by means of the centroid position: The centroidal lines of all members connected to the node get together in one point. This allows HSS to determine the position of the struts to each other and to the chord member. Therefore, the Eccentricity \(e\) is always set to zero after accessing the module window.

Depending on the cross-sections’ dimensions, the setting results in a positive Gap \(g\) with a positive value between the struts, or a negative gap value which is then called Overlap \(q\).

Thus, the eccentricity is defined as perpendicular distance between the point of intersection of the struts’ center lines and the center line of the chord member.

Fully fitted

If an overlap (negative gap value) is defined, the options are enabled. For the determination of the overlap degree \(\lambda_{ov}\) it is necessary to specify the strut that is completely welded to the chord.
2 Input Data

Additional geometry parameters
In the right part of the module window, additional geometry parameters are displayed for the cross-sections. The individual parameters are represented in the graphical scheme above which is automatically adjusted depending on the node type and the type for gap and overlap.

To the left you can see an interactive graphic showing the node selected for design in rendering mode. Input of module windows 1.2 and 1.3 is visualized dynamically. Use the wheel button of your mouse to zoom in and out the node graphic. To set the view in Y or Z and the isometric view, use the buttons shown on the left.

2.4 Loads

Existing load cases
Load combinations and result combinations
The two window sections list all load cases, load and result combinations that are defined in RSTAB. Use the button [►] to transfer selected entries to the Design of list on the right. You can also double-click the items. To transfer the entire list, click [►►].

Load cases that are marked by an asterisk (*) like load case LC 4 in the figure above cannot be designed: No loading is defined in the load case, or it is an imperfection load case as the one in the example above.

In addition, a multiple selection of load cases is possible with the common Windows function where you hold down the keyboard key [Ctrl]. In this way, you can transfer several load cases at the same time.

Design of
The column on the right lists the load cases and combinations that you have selected for the design. Use the button [◄] to remove a selected item from the list. You can also double-click the items. To empty the entire list, click [◄◄].
2.4.1 Defining Loads Manually

When the input option Define manually has been selected in module window 1.1 General Data (see chapter 2.1, page 9), the module window 1.4 looks different.

You can enter the Internal Forces of a load case directly into the input fields that are available for the forces and moments of the chords and struts.

Use the [New] button to create another load case. Furthermore, it is possible to import data from an existing load case. Each load case can be specified by its own description. Click the buttons [▼] and [▲] to switch between the load cases. Additionally created load cases can be selected for deletion in a separate dialog box by clicking the button shown on the left.
3 Calculation

To start the calculation, click the [Calculation] button that is available in all input windows of HSS.

HSS searches for the results of the load cases, load and result combinations to be analyzed. If they cannot be found, the RSTAB calculation starts in order to determine the design-relevant internal forces. During determination HSS falls back on the calculation parameters that are valid for the current model in RSTAB.

You can also start the calculation of HSS results out of the RSTAB user interface. The add-on modules are listed like load cases or combinations in the dialog box To Calculate. To open the dialog box in RSTAB, select To Calculate on the Calculate menu.

![Figure 3.1: Dialog box To Calculate](image)

If you miss the HSS design cases in the Not Calculated list, set the selection field below to All or Add-on Modules.

To transfer the selected HSS cases to the list on the right, use the [►] button. Then, start the calculation by using the [OK] button.

You can also use the list in the toolbar to calculate an HSS case: Select the relevant design case, and then click [Show Results].

![Figure 3.2: Direct calculation of a HSS design case in RSTAB](image)

Now, you can observe the calculation process in a separate dialog box.
4. Results

4.1 Design by Nodes

Module window 2.1 Design by Nodes appears after the calculation. The upper part of the results window shows a summary of designs for the individual load cases, load and result combinations. The lower part lists detailed information about material data, cross-section properties, analyzed internal forces, validity limits and performed designs for the table row that is selected above.

Click [OK] to save the results and quit the add-on module HSS.

![Module window 2.1 Design by Nodes](image)

**Node type**
The table column displays the node type symbol. Its denotation depends on the node geometry, the type of strut forces and the acting of additional loads.

**Governing load case**
This column displays the numbers of those load cases, load and result combinations (Max or Min) that prove to be decisive for the respective designs.

**Design / design criterion**
The output shows for each type of design and each load case respectively each load and result combination the design criteria according to EN 1993.

**Design according to formula**
Table column E shows the equations according to the standard used (EN 1993 in our example) that have been applied to perform the design.
4 Results

You see four buttons below the upper results table. They have the following functions:

<table>
<thead>
<tr>
<th>Button</th>
<th>Description</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Display color scales" /></td>
<td>Display color scales</td>
<td>Shows or hides the colored relation scales</td>
</tr>
<tr>
<td><img src="image" alt="Exceeding" /></td>
<td>Exceeding</td>
<td>Shows only rows with a ratio &gt; 1 (design not fulfilled)</td>
</tr>
<tr>
<td><img src="image" alt="View mode" /></td>
<td>View mode</td>
<td>Jumps to the RSTAB work window to set another view.</td>
</tr>
<tr>
<td><img src="image" alt="Node selection" /></td>
<td>Node selection</td>
<td>Enables the selection of a node in the RSTAB window to display its results in the table</td>
</tr>
</tbody>
</table>

Table 4.1: Buttons of results window

4.2 Results Evaluation

In the lower part of module window 2.1, all intermediate results of the design are shown. These Details refer to the results of the table above and the respective row where the cursor is placed. Details are refreshed when you click into another table row.

For a successful design of a hollow section connection it is necessary to meet the geometric validity limits. They are defined in EN 1993-1-8, table 7.1 for pipes, respectively tables 7.8 and 7.9 for square and rectangular hollow sections. Furthermore, table 7.9 provides supplementary conditions to facilitate a simplified design if they are met.

If validity limits are not met, the failure is shown in module window 2.1. The design won't be performed. In case only one additional condition is not met, you can see the relevant values highlighted in red under Validity Limits in the Details table.

Figure 4.2: Module window 2.1 Design by Nodes with error for validity limits
If several geometrically identical nodes have been selected for the design, window 2.1 lists the designs sorted by node numbers.

Figure 4.3: Module window 2.1 Design by Nodes for design of several nodes
5. Printout

5.1 Printout Report

First, a printout report for the design results determined with HSS is generated which can be completed by graphics and descriptions. In the report, you also decide which details of the HSS results window 2.1 appear in the printout.

The printout report is described in detail in the RSTAB manual. In particular, chapter 10.1.3.5 Selecting Data of Add-on Modules on page 232 provides information about the selection of input and output data in add-on modules.

When your structure is quite extensive, it is advisable to split the data into several small reports. If you create a separate printout report only for the data of a design case from HSS, the printout report will be generated relatively quickly.
6. General Functions

This chapter describes some menu functions as well as export options for the design results.

6.1 HSS Design Cases

You can group nodes in separate design cases. In this way, it is possible to integrate nodes of the same geometry. As connections with different geometries or different cross-sections cannot be calculated together in one design case, this option proves to be very useful.

The HSS cases are available in the RSTAB workspace and can be displayed like load cases or combinations by using the toolbar list.

Create a new HSS case

To create a new design case,

select New Case on the File menu in the HSS add-on module.

The following dialog box appears.

![New HSS Case dialog box](image)

In this dialog box, enter a *No.* (which is not yet assigned) and a *Description* for the new design case. When you click [OK], module window 1.1 General Data opens where you can enter the new design data.

Rename a HSS case

To change the description of a design case subsequently,

select Rename Case on the File menu in the HSS add-on module.

The dialog box Rename HSS Case appears.

![Rename HSS Case dialog box](image)
Copy a HSS case

To copy the input data of the current design case,

select **Copy Case** on the **File** menu in the HSS add-on module.

The dialog box **Copy HSS Case** appears where you can specify the number and description of the new case.

![Copy HSS Case dialog box](image)

**Figure 6.3: Dialog box Copy HSS Case**

Delete a HSS case

To delete a design case,

select **Delete Case** on the **File** menu in the HSS add-on module.

In the dialog box **Delete Cases**, you can select a particular design case in the list **Available Cases** and delete it by clicking **[OK]**.

![Delete Cases dialog box](image)

**Figure 6.4: Dialog box Delete Cases**
6.2 Units and Decimal Places

The units and decimal places for RSTAB and all add-on modules are managed in one global dialog box. In the add-on module HSS, you can use the menu to define the units. To open the corresponding dialog box,

select **Units and Decimal Places** on the **Settings** menu.

The program opens the following dialog box that you already know from RSTAB. The add-on module HSS is preset.

![Dialog box Units and Decimal Places](image)

Figure 6.5: Dialog box **Units and Decimal Places**

The settings can be saved as user profile to reuse them for other models. The corresponding functions are described in the RSTAB manual, chapter 11.1.3.

6.3 Export of Results

It is possible to make HSS designs available also for other programs in various ways.

**Clipboard**

To copy cells selected in the HSS results window to the clipboard, use the keyboard keys [Ctrl]+[C]. To insert the cells, for example in a word processing program, press [Ctrl]+[V]. The headers of the table columns will not be transferred.

**Printout report**

Data of the HSS add-on module can be printed into the global printout report (see chapter 5.1, page 20) from where it can be exported. In the printout report,

select **Export to RTF** on the **File** menu of the printout report.

The function is described in the RSTAB manual, chapter 10.1.11.
Excel

HSS provides a function for the direct data export to MS Excel, OpenOffice.org Calc or as CSV file. To open the corresponding dialog box, select **Export Tables** on the **File** menu in the HSS add-on module.

The following export dialog box appears.

![Figure 6.6: Dialog box Export - MS Excel](image)

When you have selected the relevant parameters, start the export by clicking **[OK]**. It is not necessary to run MS Excel in the background because it starts automatically.

![Figure 6.7: Result in MS Excel](image)
A Literature

B Index

A
Angle ........................................................... 13

C
c/t cross-section parts .................................. 11
Chord ......................................................... 10, 13, 15
Connection type ............................................ 9
Cross-section type .......................................... 9
Cross-sections .............................................. 10

D
Decimal places ............................................. 23
Design ......................................................... 17
Design case ............................................ 21, 22

E
Eccentricity .................................................. 13
Excel ............................................................ 24
Export .................................................... 21, 23
External loads .............................................. 15

F
Fitted ........................................................... 13

G
Gap .............................................................. 13
General data .................................................. 7
Geometry .................................................. 9, 13, 14

I
Info about cross-section .............................. 11
Input data ...................................................... 9
Installation .................................................... 5
Internal forces .......................................... 15

L
Load case .................................................. 14, 17
Load combination ........................................ 14, 17
Loads ....................................................... 14

M
Material ....................................................... 12
Member assignment ................................ 12

N
National annex .............................................. 8
Navigator ..................................................... 7
Nodal load .................................................. 15
Node type ................................................... 17
Nodes ......................................................... 8
Overlap ...................................................... 13

P
Partial safety factors ................................... 8
Planar ......................................................... 9
Printout report ........................................... 20
Quit HSS ...................................................... 7

R
Relation scales ........................................... 18
Result combination ...................................... 17
Results .................................................. 17, 18
Selecting module windows ..................... 7
Spatial ......................................................... 9
Start HSS ..................................................... 6
Start program ............................................. 6
Steel production ........................................ 12
Stress point .............................................. 11
Strut .................................................. 9, 10, 13, 15

U
Units ............................................................. 23
User profile .................................................. 23

V
Validity limits ............................................ 18
View mode ................................................... 18

W
Windows ................................................... 7