<p>| Contacts                                                                                     | 8 |
| Opening the service Results                                                               | 10 |
| Selecting the 1D members for display                                                      | 11 |
| Selection                                                                                   | 11 |
| Selection: Advanced                                                                        | 11 |
| Filter                                                                                      | 11 |
| Structure                                                                                  | 12 |
| Section                                                                                     | 12 |
| Selecting the load for the display of results                                              | 13 |
| Selecting the geometry for display of results                                              | 14 |
| Structure                                                                                  | 14 |
| Adjusting the style of result diagrams                                                     | 16 |
| Representation                                                                              | 16 |
| Limits                                                                                      | 16 |
| Description                                                                                 | 18 |
| Angle of text                                                                               | 18 |
| Drawing of more components                                                                  | 18 |
| Regenerating the diagrams                                                                   | 19 |
| Preserving the results after model modifications                                           | 20 |
| Setting of result deleting prevention                                                      | 21 |
| Animation of results                                                                       | 22 |
| Controls of the animation window                                                           | 23 |
| Upgrade from 2D to 1D project                                                               | 25 |
| Type of export                                                                              | 25 |
| From template                                                                              | 25 |
| Results on beams                                                                           | 27 |
| Displaying the internal forces                                                             | 27 |
| Display parameters for diagrams of internal forces                                         | 27 |
| Extreme                                                                                    | 27 |
| Displaying the deformation on 1D members                                                   | 28 |
| Displaying the deformation of nodes                                                        | 28 |</p>
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displaying the resultant of reactions</td>
<td>28</td>
</tr>
<tr>
<td>Display parameters for diagrams of resultant of reactions</td>
<td>29</td>
</tr>
<tr>
<td>Resultant in intersecting linear supports</td>
<td>29</td>
</tr>
<tr>
<td>Displaying the nodal space support resultant</td>
<td>30</td>
</tr>
<tr>
<td>Nodal Support Space Resultant</td>
<td>31</td>
</tr>
<tr>
<td>Calculation principle</td>
<td>31</td>
</tr>
<tr>
<td>Possible application</td>
<td>32</td>
</tr>
<tr>
<td>Nodal Support Space Resultant table in the document</td>
<td>32</td>
</tr>
<tr>
<td>Displaying the reactions</td>
<td>33</td>
</tr>
<tr>
<td>Display parameters for diagrams of reactions</td>
<td>33</td>
</tr>
<tr>
<td>Displaying the foundation table</td>
<td>34</td>
</tr>
<tr>
<td>Parameters of Foundation table function</td>
<td>34</td>
</tr>
<tr>
<td>How are results in Foundation table calculated?</td>
<td>36</td>
</tr>
<tr>
<td>Displaying the bill of material</td>
<td>36</td>
</tr>
<tr>
<td>Displaying the intensity</td>
<td>36</td>
</tr>
<tr>
<td>Displaying the stress on members</td>
<td>38</td>
</tr>
<tr>
<td>Displaying the shear stress</td>
<td>38</td>
</tr>
<tr>
<td>Shear stress calculated from the difference in axial forces</td>
<td>39</td>
</tr>
<tr>
<td>Selecting the joints for display of connection forces</td>
<td>41</td>
</tr>
<tr>
<td>Displaying the connection forces</td>
<td>42</td>
</tr>
<tr>
<td>Parameters of function Connection forces</td>
<td>43</td>
</tr>
<tr>
<td>Displaying the calculation report</td>
<td>43</td>
</tr>
<tr>
<td>Displaying the results in tabular form</td>
<td>43</td>
</tr>
<tr>
<td>Displaying the results in named fibres</td>
<td>44</td>
</tr>
<tr>
<td>Named fibres of a cross-section</td>
<td>44</td>
</tr>
<tr>
<td>Named items dialogue</td>
<td>44</td>
</tr>
<tr>
<td>Named fibres in function Results &gt; Member stress</td>
<td>44</td>
</tr>
<tr>
<td>Displaying the stress distribution over the cross-section</td>
<td>48</td>
</tr>
<tr>
<td>Fast selection of result quantities for the display</td>
<td>49</td>
</tr>
<tr>
<td>Displaying the natural frequencies</td>
<td>50</td>
</tr>
<tr>
<td>Displaying the eigenmodes</td>
<td>51</td>
</tr>
<tr>
<td>Evaluating the results for harmonic load</td>
<td>52</td>
</tr>
</tbody>
</table>
### Chapter 0

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculation of internal forces in ribs</td>
<td>52</td>
</tr>
<tr>
<td>Evaluation of the results on the phased CSS</td>
<td>53</td>
</tr>
<tr>
<td>3D displacement</td>
<td>55</td>
</tr>
<tr>
<td>Properties of the result</td>
<td>55</td>
</tr>
<tr>
<td>3D stress</td>
<td>59</td>
</tr>
<tr>
<td>Properties of the result</td>
<td>62</td>
</tr>
<tr>
<td>Calculation of stresses for 1D members</td>
<td>66</td>
</tr>
<tr>
<td>Results on slabs</td>
<td>72</td>
</tr>
<tr>
<td>Calculated results for 2D members</td>
<td>72</td>
</tr>
<tr>
<td>Internal forces</td>
<td>72</td>
</tr>
<tr>
<td>2. Stresses, group=3</td>
<td>76</td>
</tr>
<tr>
<td>Strains, group=14, Plastic strains, group=15</td>
<td>78</td>
</tr>
<tr>
<td>Displaying the deformation of nodes on slabs</td>
<td>80</td>
</tr>
<tr>
<td>Displaying the internal forces on slabs</td>
<td>81</td>
</tr>
<tr>
<td>Parameters for display of results</td>
<td>81</td>
</tr>
<tr>
<td>Type of forces</td>
<td>82</td>
</tr>
<tr>
<td>Basic magnitude</td>
<td>82</td>
</tr>
<tr>
<td>Principal magnitude</td>
<td>83</td>
</tr>
<tr>
<td>Design magnitude</td>
<td>84</td>
</tr>
<tr>
<td>Principal internal forces</td>
<td>84</td>
</tr>
<tr>
<td>Design internal forces</td>
<td>85</td>
</tr>
<tr>
<td>Displaying the stresses on slabs</td>
<td>86</td>
</tr>
<tr>
<td>Available stress values</td>
<td>87</td>
</tr>
<tr>
<td>Stresses</td>
<td>87</td>
</tr>
<tr>
<td>Displaying the contact stress on slabs</td>
<td>88</td>
</tr>
<tr>
<td>Calculated C parameters</td>
<td>88</td>
</tr>
<tr>
<td>Displaying the settlement</td>
<td>90</td>
</tr>
<tr>
<td>Displaying the soil structure strength</td>
<td>90</td>
</tr>
<tr>
<td>Results in membrane elements</td>
<td>91</td>
</tr>
<tr>
<td>Differences in the results between membrane and standard element</td>
<td>92</td>
</tr>
<tr>
<td>Displaying results for individual FE nodes or elements</td>
<td>93</td>
</tr>
<tr>
<td>Dialogue for the display of results for individual FE nodes or elements</td>
<td>93</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>Displaying the results in selected sections</td>
<td>144</td>
</tr>
<tr>
<td>1D member</td>
<td>144</td>
</tr>
<tr>
<td>Slab</td>
<td>146</td>
</tr>
<tr>
<td>Type of diagram in the section</td>
<td>150</td>
</tr>
<tr>
<td>Displaying the resultant in the section across a slab</td>
<td>153</td>
</tr>
<tr>
<td><strong>Table results</strong></td>
<td>155</td>
</tr>
<tr>
<td><strong>Table results - introduction</strong></td>
<td>155</td>
</tr>
<tr>
<td>How to load results into the Table results</td>
<td>156</td>
</tr>
<tr>
<td>Reset the Table results settings to the default</td>
<td>157</td>
</tr>
<tr>
<td><strong>Table results - toolbar description</strong></td>
<td>158</td>
</tr>
<tr>
<td>Get results</td>
<td>158</td>
</tr>
<tr>
<td>Get results on Refresh</td>
<td>158</td>
</tr>
<tr>
<td>Regenerate current tab</td>
<td>158</td>
</tr>
<tr>
<td>Delete all tabs</td>
<td>158</td>
</tr>
<tr>
<td>Column selector</td>
<td>158</td>
</tr>
<tr>
<td>Filtering row</td>
<td>159</td>
</tr>
<tr>
<td>Display detailed check of row in preview</td>
<td>159</td>
</tr>
<tr>
<td>Send table to the Engineering Report</td>
<td>159</td>
</tr>
<tr>
<td>Result properties info</td>
<td>160</td>
</tr>
<tr>
<td><strong>Table results - table</strong></td>
<td>160</td>
</tr>
<tr>
<td>Sorting</td>
<td>160</td>
</tr>
<tr>
<td>Filtering row</td>
<td>161</td>
</tr>
<tr>
<td>Copy values into clipboard</td>
<td>163</td>
</tr>
<tr>
<td>Columns manipulation</td>
<td>164</td>
</tr>
<tr>
<td>Highlight of extreme values</td>
<td>166</td>
</tr>
<tr>
<td>Selection link</td>
<td>167</td>
</tr>
<tr>
<td>3D model position highlight</td>
<td>170</td>
</tr>
<tr>
<td>Display detailed check of row in preview</td>
<td>171</td>
</tr>
<tr>
<td>Header context menu</td>
<td>173</td>
</tr>
<tr>
<td>Table context menu</td>
<td>173</td>
</tr>
<tr>
<td><strong>Table results - tabs with results</strong></td>
<td>174</td>
</tr>
<tr>
<td>Tabs</td>
<td>174</td>
</tr>
</tbody>
</table>
### Contacts

**Belgium Headquarters**

SCIA nv  
Industrieweg 1007  
B-3540 Herk-de-Stad  
Tel: +32 13 55 17 75  
E-mail: info@scia.net

**Support**  
CAE (SCIA Engineer)  
Tel: +32 13 55 09 90

CAD (Allplan)  
Tel: +32 13 55 09 80

**Support E-mail:**  
support@scia.net

---

**France**

SCIA France sarl  
Centre d'Affaires  
29, Grand' Rue  
FR-59100 Roubaix  
Tel.: +33 3 28.33.28.67  
Fax: +33 3 28.33.28.69  
E-mail: france@scia.net

**Agence commerciale**  
8, Place des vins de france  
FR-75012 Paris  
Tel.: +33 3 28.33.28.67  
Fax: +33 3 28.33.28.69  
E-mail: france@scia.net

---

**Brazil**

SCIA do Brasil Software Ltda  
Rua Dr. Luiz Migliano, 1986 - sala 702, CEP  
SP 05711-001 São Paulo  
Tel.: +55 11 4314-5880  
E-mail: brasil@scia.net

---

**USA**

SCIA North America  
7150 Riverwood Drive  
21046 Columbia, MD  
Tel.: +1 443-542-0638  
Fax: +1 410-290-8050  
E-mail: usa@scia.net

---

**Netherlands**

SCIA Nederland B.V.  
Wassenaarweg 40  
NL-6843 NW ARNHEM  
Tel.: +31 26 320 12 30  
Fax: +31 26 320 12 39  
E-mail: info@scia.net

---

**Switzerland**

SCIA Swiss Office  
Dürenbergstrasse 24  
CH-3212 Gurmels  
Tel.: +41 26 341 74 11  
Fax: +41 26 341 74 13  
E-mail: info@scia.ch

---

**Czech Republic**

SCIA CZ s.r.o. Praha  
Evropská 2591/33d  
160 00 Praha 6  
Tel.: +420 226 205 600  
Fax: +420 226 201 673  
E-mail: info.praha@scia.cz

SCIA CZ s.r.o. Brno  
Slavickova 827/1a  
638 00 Brno  
Tel.: +420 530 501 570  
Fax: +420 226 201 673  
E-mail: info.brno@scia.cz

---

**Slovakia**

SCIA SK s.r.o.  
Murgašova 1298/16  
SK-010 01 Žilina  
Tel.: +421 415 003 070  
Fax: +421 415 003 072  
E-mail: info@scia.sk
<table>
<thead>
<tr>
<th>Country</th>
<th>Address</th>
<th>Telephone</th>
<th>Fax</th>
<th>E-mail</th>
</tr>
</thead>
<tbody>
<tr>
<td>Austria</td>
<td>SCIA Datenservice Ges.m.b.H. Dresdnerstrasse 68/26/9 A-1200 WIEN Tel.: +43 1 7433232-11 Fax: +43 1 7433232-20 E-mail: <a href="mailto:info@scia.at">info@scia.at</a> Support Tel.: +43 1 7433232-12 E-mail: <a href="mailto:support@scia.net">support@scia.net</a></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Germany</td>
<td>SCIA Software GmbH Technologie Zentrum Dortmund, Emil-Figge-Strasse 76-80 D-44227 Dortmund Tel.: +49 231/9742586 Fax: +49 231/9742587 E-mail: <a href="mailto:info@scia.de">info@scia.de</a></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 SCIA nv. All rights reserved.

Document created: 27 / 05 / 2016

SCIA Engineer 16.0
Chapter 1

Opening the service Results

Service Results may be opened after a calculation has been successfully finished. Service Results can be opened using:

- tree menu item Results,
- menu function Tree > Results,
- icon Results ( ) on toolbar Project.

The service may look like:

As soon as the service is opened in the tree menu window, the Property window is filled with parameters corresponding to active function of service Results. The parameters in the Property window can be used to adjust “WHAT” is displayed and “HOW” it is displayed.

Common parameters are:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Load type</strong></td>
<td>Specifies what &quot;load type&quot; is considered for the display. Available load types are:</td>
</tr>
<tr>
<td></td>
<td>load cases, load case combinations, result classes.</td>
</tr>
<tr>
<td><strong>Load case / combination / class</strong></td>
<td>For each of the above specified load type a set of available items (load cases, combinations, result classes) is offered.</td>
</tr>
<tr>
<td><strong>Selection</strong></td>
<td>The user may display the results either on all or only selected 1D members.</td>
</tr>
<tr>
<td><strong>Filter</strong></td>
<td>The set of 1D members where the results are displayed may be specified by means of a filter.</td>
</tr>
<tr>
<td><strong>Values</strong></td>
<td>For each of the result groups (internal forces, deformations, etc.) a set of quantities id offered for display. The user may select which one is really shown.</td>
</tr>
<tr>
<td><strong>Drawing setup</strong></td>
<td>It is possible to adjust the style of the diagrams.</td>
</tr>
<tr>
<td><strong>Other specific parameters</strong></td>
<td>Some of the available result groups (internal forces, deformations, etc.) may have other group-specific parameters.</td>
</tr>
</tbody>
</table>

Note: If a calculation has not been performed yet or the structure has been somehow modified after the calculation has been carried out, service Results is not accessible (to be precise, it is not offered in the tree menu).

Note: The collection of functions offered in the service may vary according to the project type and authorised modules.
Selecting the 1D members for display

The result diagrams may be displayed on (i) all the 1D members in the structure, or (ii) selected 1D members only. Which variant is actually applied can be adjusted in the Property window by means of parameters Selection and Filter.

Selection

<table>
<thead>
<tr>
<th>Selection</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>If this option is selected, the result diagrams are displayed on all 1D members in the structure.</td>
</tr>
<tr>
<td>Current</td>
<td>The result diagrams are displayed on all the currently selected members.</td>
</tr>
<tr>
<td>Advanced</td>
<td>This option allows the user to display diagrams on selected members. It is similar to the previous option but offers something more. See below the table.</td>
</tr>
<tr>
<td>Named</td>
<td>This option allows the user to select one of the previously created, named and saved selections.</td>
</tr>
</tbody>
</table>

Selection: Advanced

With this option, you may select required members on which the results are to be displayed and review the results. Then you may clear the selection. The result diagrams, however, remain displayed. Now you may make a new selection and invoke the refresh of the screen. The program will ask you what to do. The available options are:

Use current selection

The result diagrams displayed during the last refresh are deleted. New result diagrams are displayed on the currently selected members only.

Add current selection to previous selection

The result diagrams displayed during the last refresh remain displayed. New result diagrams are shown on the currently selected members.

Use previous selection

The current selection is ignored. The result diagrams displayed during the previous refresh remain displayed.

Subtract current selection from previous selection

If there is a result diagram currently displayed on one of the currently selected members, this diagram is hidden. The result diagrams that are shown on members that are not in the current selection remain displayed.

Filter

<table>
<thead>
<tr>
<th>Filter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>No</td>
<td>No filter is applied.</td>
</tr>
<tr>
<td>Wildcard</td>
<td>The set of 1D members for display is defined by a wildcard expression. E.g. expression &quot;N*&quot; lists all entities whose name starts with letter N. The expression &quot;B??&quot; lists all entities whose name starts with letter B and is followed by two characters.</td>
</tr>
</tbody>
</table>
Chapter 2

Cross-section

- Diagrams are shown only on entities of selected cross-section.

Material

- Diagrams are shown only on entities of selected material.

Layer

- Diagrams are shown only on entities inserted into selected layer.

Structure

This parameter is useful especially for nonlinear analysis construction stages analysis.

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial</td>
<td>The diagrams of result quantities are drawn at the initial (non-deformed) shape of the analysed structure. The &quot;smoothness&quot; of the diagram is specified by the Number of sections on average member that can be adjusted in Solver setup.</td>
</tr>
<tr>
<td>Mesh</td>
<td>The diagrams of result quantities are drawn at the initial mesh for the evaluated construction stage. For the results of stage 1 or for results of a simple (non-staged) calculation it is identical with the previous option. However, for stage 2 and subsequent ones it represents the &quot;initial&quot; shape of the structure at the beginning of the evaluated constructions stage. The smoothness of the diagram is given by how fine the generated mesh is.</td>
</tr>
<tr>
<td>Deformed</td>
<td>The diagrams of result quantities are drawn at the final (deformed) shape of the analysed structure. The deformation of the structure uses fixed predefined scale 1:1.</td>
</tr>
</tbody>
</table>

Section

This parameter defines how detailed and smooth the diagram is.

<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>All</td>
<td>The checks are performed and displayed in all sections along the member. The number of sections is defined in Solver setup.</td>
</tr>
<tr>
<td>Input</td>
<td>The checks are performed and displayed ONLY in sections defined by the user. The sections can be defined in service Structure.</td>
</tr>
<tr>
<td>Ends</td>
<td>The checks are performed and displayed ONLY in end points of the member.</td>
</tr>
<tr>
<td>Input + Ends</td>
<td>The checks are performed and displayed in sections defined by the user and in the end points of the member. The sections can be defined in service Structure.</td>
</tr>
</tbody>
</table>
Selecting the load for the display of results

The results may have calculated for: (i) load cases, (ii) load case combinations, (iii) result classes.

In service Results, the user may specify which group (or set) should be taken into account for display. The selection can be made in item Type of loads in the Property window of service Results.

<table>
<thead>
<tr>
<th>type</th>
<th>description</th>
</tr>
</thead>
<tbody>
<tr>
<td>load case</td>
<td>Diagrams are drawn for specific load case.</td>
</tr>
<tr>
<td>combination</td>
<td>Diagrams are drawn for specific load case combination.</td>
</tr>
<tr>
<td>result class</td>
<td>Diagrams are drawn for specific result class.</td>
</tr>
</tbody>
</table>

The choice of a particular load case, combination, or result class can be then made in item located just below Types of loads in the Property window of service Results. Only one load case, combination or result class may be selected at a time.
Selecting the geometry for display of results

When displaying results on beams, it is possible to specify the geometry on which the result diagrams should be displayed.

Structure

The main purpose of this setting (parameter) is to define on which geometry the results are to be drawn. For internal forces, stresses, checks... it modifies basically the geometry on which the results are displayed. For displacements, in some cases, it also influences the displayed results.

The possible settings are listed below.

To illustrate the various possibilities, let’s use the following definitions, for the displacement of a particular node in a structure:

- X₀ is the original input coordinate of the considered node
- X₁ is the modified coordinate of the same node in the mesh of the structure (see explanation below, why it might differ from X₀)
- UX is the result value, i.e. the displacement of the node for the considered load case / combination / construction stage...

(the graphical output shown are from a nonlinear staged analysis; a load is applied at the end of the cantilever and there is an initial deflection coming from the previous stages)

**Initial**

The results are drawn on the initial geometry of the structure, as originally input by the user

- the results are displayed on the geometry X₀
- the displayed values and diagrams are UX

**Mesh**
The results are drawn on the geometry of the mesh that is used for the analysis. In some cases, the mesh may differ from the initial geometry. Typically:

- cable members may have a curved initial geometry, unlike the apparently straight line that is displayed at input time, defined by their nonlinear member settings
- global imperfection for nonlinear analysis uses a mesh that includes imperfections
- nonlinear construction stages use the resulting geometry from previous stage as starting point for the current stage, i.e. the mesh of a stage is the result of the previous stage

- the results are displayed on the geometry X1
- the displayed values and diagrams are UX

**Deformed**
The results are drawn on the final geometry, including the initial geometry and the displacements induced by the current loading.

- the results are displayed on the final geometry X1+UX
- the displayed values and diagrams are UX

**Global deformation**
The results are drawn on the initial geometry, but the displayed values are the cumulated results. This is relevant only for displacements, when the mesh differs from the initial geometry. In particular for nonlinear construction stages, it allows to display the cumulated displacement of the structure (each stage contains only its displacement increment).

- the results are displayed on the initial geometry X0
- the displayed values and diagrams are X1+UX
Adjusting the style of result diagrams

The style of the diagrams may be adjusted in the Drawing setup dialogue.

**Representation**

<table>
<thead>
<tr>
<th>Outline lines only</th>
<th><img src="diagram1.png" alt="Diagram" /></th>
</tr>
</thead>
<tbody>
<tr>
<td>Outline lines only with hatches in sections</td>
<td><img src="diagram2.png" alt="Diagram" /></td>
</tr>
<tr>
<td>Filled form</td>
<td><img src="diagram3.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

**Limits**

The limits may be adjusted to control the colour of the diagram. The user specifies two numerical values. Three colours are used to display the diagram. The colours may be adjusted in the Setup > Colours and lines dialogue. Rules for use of individual colour are explained in the enclosed table:

| Colour: Result if below min | This colour is used for those sections where the value of displayed component is lower than the minimum limit. |
| Colour: Result if above max | This colour is used for those sections where the value of displayed component is greater than the maximum limit. |
| Colour: Result if between min and max | This colour is applied for the sections where the value of displayed component is between the limits. |

Example of limits application

Let’s assume the following adjustment of limits and colours:
Adjusting the style of result diagrams

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Maximum</strong></td>
<td>1000</td>
</tr>
<tr>
<td><strong>Minimum</strong></td>
<td>-4000</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if below min</td>
<td>blue</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if above max</td>
<td>red</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if between min and max</td>
<td>green</td>
</tr>
</tbody>
</table>

The diagram will look like:

![Diagram Image]

Another example

The settings described above may be used to "hide" specific range of the result values. For example, if you want to see just the positive branch of the diagram, it is possible to use the following trick.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Maximum</strong></td>
<td>0</td>
</tr>
<tr>
<td><strong>Minimum</strong></td>
<td>0</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if below min</td>
<td>colour that is very close or identical to the background colour, e.g. very very light blue if white background is used</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if above max</td>
<td>e.g. blue</td>
</tr>
<tr>
<td><strong>Colour:</strong></td>
<td></td>
</tr>
<tr>
<td>Result if between min and max</td>
<td>does not matter</td>
</tr>
</tbody>
</table>

The diagram will look like:
Description

<table>
<thead>
<tr>
<th>Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Numerical values</td>
<td>Numerical values are printed next to the diagram.</td>
</tr>
<tr>
<td>Sections in labels</td>
<td>The relative co-ordinate of individual sections is printed next to the diagram.</td>
</tr>
<tr>
<td>Load case or combination in labels</td>
<td>The name of appropriate load case or combination is printed next to the diagram.</td>
</tr>
</tbody>
</table>

Angle of text

The user may specify the inclination of the text for diagram labels.

Drawing of more components

If more than one component is drawn at the same time, it is possible to define the style of the composed diagram.

<table>
<thead>
<tr>
<th>Description</th>
<th>Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Same scale</td>
<td>All the diagrams for the same 1D member use the same scale.</td>
</tr>
<tr>
<td>Same height</td>
<td>All the diagrams for the same 1D member use the same height.</td>
</tr>
<tr>
<td>Space between diagrams</td>
<td>Defines the “gap” between two adjacent diagrams.</td>
</tr>
<tr>
<td>Shift of the first diagram</td>
<td>Defines the shift of the first diagram from the 1D member.</td>
</tr>
</tbody>
</table>

The procedure for the adjustment of display style parameters

1. Open service Results.
2. Select required group (or set) for the display (e.g. internal forces, bill of material, etc.).
3. Click button [Drawing setup].
4. The Drawing setup dialogue opens on the screen.
5. Set required parameters.
6. Confirm with [OK].
Regenerating the diagrams

When the settings in the Property window of service Results are changed, the diagrams usually require regeneration. Because the fully automatic regeneration could be very slow for excessive models, it is up to the user to regenerate the drawing when necessary.

Any time the user makes a change that affects the display, the program paints the cell Refresh in red colour. Until the user presses the button [Refresh], the cell remains highlighted.
Chapter 7

Preserving the results after model modifications

Keeping of solver results is safer since version 14. After any modification of the model which leads to deleting of solver results, user can decide whether to really do the action or cancel the action and keep results.

After finishing of such an action the following message is displayed:

[OK] proceed the action and results are deleted.

[Cancel] cancels the action and keeps results available.

User can switch the message off by checking the checkbox or in the Setup of SCIA Engineer (see following chapter).
Setting of result deleting prevention

On the tab Other of the SCIA Engineer Setup it is possible to switch the result preservation completely off or to specify the Time threshold. Results whose calculation took more than time threshold will be preserved by the message with the question. Results whose calculation took less than defined threshold will be deleted immediately after the model modification.
Animation of results

Any result quantity that has been calculated and shown in the graphical window can be displayed also in the Animation window. This window, as the name suggest, provides for animation of the currently displayed quantity.

In practice, this may be useful e.g. when dynamic calculation was performed. The animation window enables the user to view the vibration "in action".

Procedure to activate the animation

1. If necessary, perform the calculation.
2. Open service results and display the quantity you want to be animated including the load case or combination.
3. Regenerate the window to see the result diagram.
4. Call function Edit > View > New animation window.
5. If required, set the parameters of the window (see below).
6. Start the animation through icon Start animation.
7. When satisfied, close the animation window.
Controls of the animation window

<table>
<thead>
<tr>
<th>Control</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Start animation</td>
<td>This button starts / stops the animation.</td>
</tr>
<tr>
<td>Pause animation</td>
<td>This button enables you to pause the animation.</td>
</tr>
<tr>
<td>Repeat the animation indefinitely</td>
<td>If OFF, just one &quot;cycle&quot; of animation is shown.</td>
</tr>
<tr>
<td></td>
<td>If ON, the animation is repeat until stopped manually.</td>
</tr>
<tr>
<td>Preset minimal ratio to invert MAX</td>
<td>If ON, the quantity is animated in both positive and negative direction.</td>
</tr>
<tr>
<td>Set initial view parameters</td>
<td>The view in the animation window can be adjusted using standard SCIA Engine</td>
</tr>
<tr>
<td></td>
<td>&quot;mouse+keyboard&quot; controls (shift view, rotate view, zoom in/out).</td>
</tr>
<tr>
<td>Display frame time</td>
<td>Specifies the time for how long each calculated screen is shown. The lower the number, the &quot;finer&quot; the animation is.</td>
</tr>
<tr>
<td></td>
<td>For large projects it may be necessary to adjust greater number in order to give the computer enough time to calculate the next screen.</td>
</tr>
<tr>
<td>Play time</td>
<td>The total time of the animation (i.e. of one &quot;cycle&quot; of the animation).</td>
</tr>
<tr>
<td>Mode of calculation</td>
<td>The interpolation of the diagram can be performed in two ways.</td>
</tr>
<tr>
<td></td>
<td>Linear</td>
</tr>
</tbody>
</table>
Standard linear interpolation is used.

Sinus

This interpolation gives nicer "motion" of the diagram.
Upgrade from 2D to 1D project

2D-1D Upgrade is a special export function which is available for beams of prefab slab and integration strips. This export function enables the user to select one or several beams from the slab and export them to a separate project including load cases, combinations and calculated internal forces that are exported as a load the exported beam is subjected to.

The user can control the export through a set of parameters.

<table>
<thead>
<tr>
<th>Type of export</th>
<th>Specifies the way of generation of 1D project</th>
</tr>
</thead>
<tbody>
<tr>
<td>Effective width slab</td>
<td>If ON, the effective width of the slab is taken into account and the beam is exported as a T-section.</td>
</tr>
<tr>
<td></td>
<td>If OFF, the beam is exported with the cross-sections that was specified for it in the project.</td>
</tr>
<tr>
<td>Export into Frame XZ</td>
<td>(available only if just one beam is exported)</td>
</tr>
<tr>
<td></td>
<td>If ON, the exported project is of Frame XZ type.</td>
</tr>
<tr>
<td>Export to UCS from member 1</td>
<td>If ON, the origin of the UCS in the exported project is set to the origin of the LCS of the exported beam.</td>
</tr>
<tr>
<td>Change self weight to standard LC</td>
<td>If ON, the load type of the LC with type Self-weight is in the exported project changed to Standard.</td>
</tr>
<tr>
<td>Upgraded internal forces</td>
<td>If ON, the calculated internal forces are exported.</td>
</tr>
<tr>
<td></td>
<td>If OFF, only the geometry is exported.</td>
</tr>
<tr>
<td>Load cases, Combinations</td>
<td>(available only if Upgraded internal forces is set to ON)</td>
</tr>
<tr>
<td></td>
<td>The user can select which load cases and combinations are exported.</td>
</tr>
</tbody>
</table>

**Type of export**

The options for parameter Type of export are described below

<table>
<thead>
<tr>
<th>New project</th>
<th>A new project is created and beam is exported into that</th>
</tr>
</thead>
<tbody>
<tr>
<td>From template</td>
<td>The user selects existing SEN project (template) which has to contain only one 1D member. During the export the existing beam in the template is updated and stored as a new project</td>
</tr>
</tbody>
</table>

**From template**

This group of parameters is available only if the above-mentioned parameter Type of export is set to From template.

<table>
<thead>
<tr>
<th>Template</th>
<th>Selection of a template (existing SEN project)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Target directory</td>
<td>Defines a directory where will be stored generated projects with exported 1D members</td>
</tr>
<tr>
<td><strong>Filename prefix</strong></td>
<td>Defines a prefix of a project name (created from a 1D member name) which will be generated</td>
</tr>
<tr>
<td>---------------------</td>
<td>--------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td><strong>Filename suffix</strong></td>
<td>Defines a suffix of a project name (created from a 1D member name) which will be generated</td>
</tr>
<tr>
<td><strong>Length export by</strong></td>
<td>If Member length, a length of a member in the template is updated If Parameters, length parameters in the template are updated</td>
</tr>
<tr>
<td><strong>Update CSS width</strong></td>
<td>If ON, CSS dimension width is updated (available only for rectangular CSS)</td>
</tr>
<tr>
<td><strong>Update CSS height</strong></td>
<td>If ON, CSS dimension height is updated (available only for rectangular CSS)</td>
</tr>
</tbody>
</table>
Results on beams

Displaying the internal forces

The procedure to display diagrams of internal forces

1. Open service Results.
2. Select function Internal forces on beams.
3. Select the beams for the display of results.
4. Select the required type of loads.
5. Adjust the diagram style.
6. Set other display parameters.
7. If necessary, regenerate the diagrams.

Display parameters for diagrams of internal forces

<table>
<thead>
<tr>
<th>Values</th>
<th>Specifies the values, i.e. the components, which are displayed. Either one or multiple components can be displayed at a time.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Principal</td>
<td>Specifies whether the components are evaluated in principal or local axes of 1D members.</td>
</tr>
<tr>
<td>Extreme</td>
<td>Specifies the position on diagrams where numerical values are attached.</td>
</tr>
<tr>
<td>Drawing setup</td>
<td>It is possible to adjust the style of the diagrams. Read chapter Adjusting the style of result diagrams.</td>
</tr>
<tr>
<td>Section</td>
<td>Defines whether the diagram is drawn for defined sections or only for end-sections on the 1D member.</td>
</tr>
</tbody>
</table>

Extreme

The individual options for parameter Extreme are explained below.

   **No**
   The values of calculated results are displayed in every drawn section. For a load case result, there is just one value for each section. However, in the case of envelope combinations, there are as many values displayed in each section as there are load cases in the combination.

   **Section**
   The values of calculated results are displayed in every drawn section. Both for load case and combination only one value (extreme section value) is displayed in each section.

   **Local**
   The values of calculated results are displayed for every local extreme of a member. It means, if the distribution is smooth along the member, there is just one local minimum and one local maximum per member. If the result quantity undergoes step changes or has several inflexion points along the member, there are several local extremes per a member.

   **Member**
   The values of calculated results are displayed for the extreme sections of each member.

   **Interval**
   The diagrams are displayed only for those intervals of a member where the result quantity falls within the specified interval.

   **Cross-section**
   The values of calculated results are displayed only in the extreme sections for each type of cross-section.
Global
The values of calculated results are displayed only in the global extreme sections - i.e. there is just one minimum and one maximum displayed for the whole structure.

Displaying the deformation on 1D members

The procedure to display diagrams of deformation on 1D members

1. Open service Results.
2. Select function Deformation on beams.
3. Select the beams for the display of results.
4. Select the required type of loads.
5. Adjust the diagram style.
6. Set other display parameters (Display parameters for diagrams of deformation on 1D members are analogous to parameters for internal forces on beams).
7. If necessary, regenerate the diagrams.

Displaying the deformation of nodes

The procedure to display diagrams of deformation of nodes

1. Open service Results.
2. Select function Deformation of nodes.
3. Select the beams for the display of results.
4. Select the required type of loads.
5. Adjust the diagram style.
6. Set other display parameters (Display parameters for diagrams of deformation of nodes are analogous to parameters for internal forces on beams).
7. If necessary, regenerate the diagrams.

Displaying the resultant of reactions

The procedure to display diagrams of resultant of reactions

1. Open service Results.
2. Select function Supports >Resultant of reactions.
3. Select the supports for the display of results.
4. Select the required type of loads.
5. Set the display parameters.
6. Refresh the screen
Display parameters for diagrams of resultant of reactions

Values
Specifies the values, i.e. the components, which are displayed.
Either one or multiple components can be displayed at a time.

Extreme
Specifies the position on diagrams where numerical values are attached.
Possible options are: No, Node, Global

Rotated supports
Selects the type of supports.

Resultant in intersecting linear supports
When the resultant is displayed for a linear support and if the selected linear support intersects another linear support, one must be aware of the following.

If several linear supports meet in one point or if they intersect each other, the resultant calculated for one of the supports takes into account also the results from other supports. As a result, if you display in turns the resultants for individual supports, the sum of these resultants will not be equal to the resultant calculated for all the supports selected simultaneously.

Let us assume a structure whose one part is supported by three linear supports that all meet in one point of intersection (red support A, blue support B and green support C).

![Diagram showing supports A, B, and C intersecting at a point]

Let us define some load (the exact size and distribution is not important as the example is just illustrative). That load produces the following resultants in individual supports (i.e. when these supports are selected separately):

red A = 111.1 kN
blue B = 60.1 kN
green C = 51.9 kN

The sum of these three resultants is 223.1 kN. When, however, all the three supports are selected at the same time, the total resultant is 189.2 kN.

The reason is that the resultants in individual supports take into account also the results from finite elements located in the two remaining supports (for example elements e2 and e3 if we evaluate the resultant in the support with element e1 in the picture below).
Note: Resultant of reactions cannot be displayed in case there is a surface support in the project.

Displaying the nodal space support resultant

SCIA Engineer can display standard resultants of reactions – see chapter Results > Results on beams > Displaying the resultant of reactions. However, for certain types of structures other representation of the reaction may be suitable.

Example

Reactions in node
**Nodal Support Space Resultant**

Function Nodal Support Space Resultant (NSSR) calculates the total resultant of a given reaction. In addition, the function also calculates the total horizontal component of the reaction.

**Calculation principle**

For each selected (nodal) support the program does the following:

1. The minimum and maximum extreme of reaction values $R_x$, $R_y$ and $R_z$ is found.
2. For each extreme reaction value the complementary values are calculated:
   1. horizontal component;
   
   $H_x = \sqrt{R_x^2 + R_y^2}$

   1. total resultant;

   $R = \sqrt{H_x^2 + R_z^2}$

   1. direction (angle to diagonal) of the total resultant;
   2. slope of the total resultant ($= R_z / HR$);
   3. the combination in which the extreme reaction value was achieved is shown.

If there are more than one combination that have the same extreme reaction value, the combination in which the maximal total resultant is calculated is displayed.

---

Note: The function has been designed to give results for load case combinations and result classes. It can be however applied also to load cases, and the resultant is calculated as well, but the search for extreme is irrelevant.
**Possible application**

The primary application of the function is in the design of masts. When the total resultant is known, it is possible to determine the direction of the foundation poles under supports. The slope determines the inclination of the foundation pole. It can be also found whether the pole is under tension or compression. The angle with the diagonal is required for the following reason. If the angle is too large, an additional shear force has to be taken into account for the calculation of the foundation.

**Nodal Support Space Resultant table in the document**

The layout of the default Nodal Support Space Resultant table will be explained on an example of the table generated for support Sn1 located in node N21.

A table for one support has six lines. Each line contains the extreme value (minimum / maximum) of one reaction component (Rx / Ry / Rz) and the corresponding calculated values of the Nodal Support Space Resultant.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Wind I/1</td>
<td>Sn1/N21</td>
<td>Rx</td>
<td>33,92</td>
<td>83,31</td>
<td>-139,21</td>
<td>2,24</td>
<td>6,37</td>
<td>-33,32</td>
<td>76,09</td>
</tr>
<tr>
<td>Wind I/2</td>
<td>Sn1/N21</td>
<td>Rx</td>
<td>33,54</td>
<td>119,55</td>
<td>-118,60</td>
<td>3,42</td>
<td>16,99</td>
<td>-28,92</td>
<td>114,75</td>
</tr>
<tr>
<td>Wind I/3</td>
<td>Sn1/N21</td>
<td>Ry</td>
<td>43,31</td>
<td>87,65</td>
<td>-139,43</td>
<td>1,76</td>
<td>7,23</td>
<td>-42,71</td>
<td>76,09</td>
</tr>
<tr>
<td>Wind II/1</td>
<td>Sn1/N21</td>
<td>Ry</td>
<td>29,81</td>
<td>90,16</td>
<td>-135,00</td>
<td>2,85</td>
<td>7,23</td>
<td>-28,92</td>
<td>85,09</td>
</tr>
<tr>
<td>Wind II/2</td>
<td>Sn1/N21</td>
<td>Rz</td>
<td>33,92</td>
<td>83,31</td>
<td>-139,21</td>
<td>2,24</td>
<td>6,37</td>
<td>-33,32</td>
<td>76,09</td>
</tr>
</tbody>
</table>

| Case       | Shows the load case combination + load case index from the combination key in which the corresponding extreme reaction component was reached. |
| Support    | Shows the name of the support and name of the node where the support is located. |
| Extreme    | Indicates the component to which the extreme value refers to. |
| Horizontal component | Contains the calculated horizontal component of the Nodal Support Space Resultant. |
| Resultant  | Displays the total value of the Nodal Support Space Resultant. |
| Angle      | Shows the orientation of the resultant in plan view. |
|            | Contrary to many other functions, this function does not measure the angle from the axis of the coordinate system, but from a diagonal line. |
|            | What is measured is the deviation between the direction of the resultant force and the diagonal line (because of the connection of the supports). To have a general solution, the diagonal line is the line from the support in question to the point (0,0,0) in GCS. |
| Slope      | Shows the inclination of the resultant from the horizontal plane. |
| Rx, Ry, Rz | Displays the appropriate reaction component extreme value. |

**Note:** Some texts in the table header in the figure have been modified (in comparison with original headers in the real document table) in order to reduce the width of the table to fit the page in this documentation.
Note: It is convenient to add the Combination key table into the document too, as (in case of the results for load case combinations and result classes) it provides useful information about the particular load case in which the extreme value of the reaction component was reached.

### 1. Combination key

<table>
<thead>
<tr>
<th>Name</th>
<th>Description of combinations</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>LC1<em>1.00 + LC2</em>1.00 + LC3*1.00</td>
</tr>
<tr>
<td>2</td>
<td>LC1<em>1.00 + LC2</em>1.00 + LC4*1.00</td>
</tr>
<tr>
<td>3</td>
<td>LC1<em>1.00 + LC2</em>1.00</td>
</tr>
</tbody>
</table>

**The procedure to display the nodal space support resultant**

1. Open service Results.
2. Select function Nodal space support resultant.
3. Select the supports for the display of results.
4. Select the required type of loads.
5. Set other display parameters.
6. Refresh the screen.

### Displaying the reactions

**The procedure to display diagrams of reactions**

1. Open service Results.
2. Select function Supports.
3. Select the beams for the display of results.
4. Select the required type of loads.
5. Adjust the diagram style.
6. Set other display parameters.
7. If necessary, regenerate the diagrams.

**Display parameters for diagrams of reactions**

<table>
<thead>
<tr>
<th>Values</th>
<th>Specifies the values, i.e. the components, which are displayed. Either one or multiple components can be displayed at a time.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Extreme</td>
<td>Specifies the position on diagrams where numerical values are attached. Possible options are: No Node</td>
</tr>
</tbody>
</table>
Chapter 10

| Rotated supports | Selects the type of supports. |

Displaying the foundation table

Service Reactions contains, among others, the possibility to generate a table with reactions in foundations.

This option is only available for load cases.

A coefficient may be entered for each load case. The reactions in the table are multiplied by this coefficient. This may be used with advantage to consider a safety factor in reactions.

The table generated in the Preview window contains in general four main parts:

| Permanent loads | All permanent load cases are considered together. Only the total reaction (sum of all permanent load cases) is given. |
| Variable load case - not exclusive | Variable load cases which are not in an exclusive group. They can act simultaneously with all other variable loads. |
| Variable load cases - exclusive | These load cases cannot act simultaneously with other load cases of the same exclusive group. |
| Extremes | This section contains extreme values composed from all permanent and variable load cases. |

The procedure for the generation of a foundation table

1. Open service Results.
2. Select function Foundation table (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function (see below).
4. Use function Print / Preview data to create a table in the Preview window.
   1. either use menu function File > Print data > Print / Preview data,
   2. or use function Print data > Print / Preview data on toolbar Project.
5. Review the results.

Parameters of Foundation table function

| Selection | The results may be shown for either All or User-defined entities. |
| Filter | Here, the user may limit the selection to specific entities only. |
| Coefficient | This option enables the user to select and / or define a set of coefficients for individual load cases. The reactions in the foundation table are multiplied by these coefficients. |
| Rotated support | This option has influence only when rotated supports exist in the project. If this option is not marked, the reactions in the global axes are drawn. If this option is marked, the reactions in the axes of the support are drawn. |
Results on beams

More information about display settings for results may be found in chapter Opening the service Results and Displaying the internal forces.

Example of a foundation table

<table>
<thead>
<tr>
<th>Loadcase/Node</th>
<th>Rx [kN]</th>
<th>Ry [kN]</th>
<th>Rz [kN]</th>
<th>Mx [Nm]</th>
<th>My [Nm]</th>
<th>Mz [Nm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>LC1: LC2</td>
<td>-1.74</td>
<td>2.06</td>
<td>2.1906</td>
<td>-2.93</td>
<td>-6.67</td>
<td>-0.14</td>
</tr>
<tr>
<td>LC3: LC3</td>
<td>8.38</td>
<td>-0.08</td>
<td>11.62</td>
<td>14.62</td>
<td>19.18</td>
<td>14.38</td>
</tr>
<tr>
<td>LC4: LC4</td>
<td>0.12</td>
<td>-0.03</td>
<td>-0.05</td>
<td>16.70</td>
<td>0.13</td>
<td>-0.00</td>
</tr>
</tbody>
</table>

**Permanent loads**

**Variable loads - not exclusive**

**Extreme values**

Max Rz [kN] | 247.06 |
Min Rz [kN] | 219.66 |
Max Rx [kN] | 5.76   |
Min Rx [kN] | -1.74  |
Max Ry [kN] | 2.06   |
Min Ry [kN] | -3.89  |
Max Mx [Nm] | 11.62  |
Min Mx [Nm] | -2.95  |
Max My [Nm] | 12.64  |
Min My [Nm] | -6.67  |
Max Mz [Nm] | 14.24  |
Min Mz [Nm] | -0.14  |
How are results in Foundation table calculated?

The extreme values are calculated as a combination of load cases. It takes into account properties of load groups. Load cases for extreme calculation (min and max value):

- Permanent load cases are always added.
- Exclusive load cases are added only separately to get the biggest min or max value.
- Seismic load cases can be added as positive or negative to get the biggest min or max value.

Example:

Permanent load cases - LC1 = -10;
Variable (not exclusive) - LCV1 = -3;
Variable (exclusive) - LCVE1 = 5, LCVE2 = -5;
Extreme MaxRx = LC1 + LC2 + LCVE1 = -10 + 5 = -5;
Extreme MinRx = LC1 + LCV - +LCVE2 = -10 - 3 - 5 = -18;

Displaying the bill of material

*The procedure to display diagrams of bill of material*

1. Open service Results.
2. Select function Bill of material.
3. Select the beams for the display of results.
4. Select the required type of loads.
5. Adjust the diagram style.
6. Set other display parameters (Display parameters for diagrams of bill of material are analogous to parameters for internal forces on beams, although their number is considerably lower).
7. If necessary, regenerate the diagrams.

Displaying the intensity

If a member of a structure is laid on foundation, it is possible to display the intensity (reaction per meter of length) in the footing surface.

*The procedure for displaying of intensity*

1. Open service Results.
2. Select function Intensity (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function.
4. If required, redraw the screen using button Redraw in the Property window.
5. Review the results.

Example
The procedure for displaying of intensity in Preview window

1. Open service Results.
2. Select function Intensity (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function.
4. Use function Print / Preview data to create a table in the Preview window.
   1. either use menu function File > Print data > Print / Preview data,
   2. or use function Print data > Print / Preview data on toolbar Project.
5. Review the results.

Example

<table>
<thead>
<tr>
<th>Case</th>
<th>Line support</th>
<th>dx [m]</th>
<th>Rx [kN/m]</th>
<th>Ry [kN/m]</th>
<th>Rz [kN/m]</th>
<th>Mx [kN.m]</th>
<th>My [kN.m]</th>
<th>Mz [kN.m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>LC1</td>
<td>LS1</td>
<td>0,000</td>
<td>0,03</td>
<td>0,00</td>
<td>-18,53</td>
<td>0,00</td>
<td>0,00</td>
<td>0,00</td>
</tr>
<tr>
<td>LC1</td>
<td>LS1</td>
<td>1,618</td>
<td>0,01</td>
<td>0,00</td>
<td>8,37</td>
<td>0,00</td>
<td>0,00</td>
<td>0,00</td>
</tr>
<tr>
<td>LC1</td>
<td>LS1</td>
<td>5,000</td>
<td>0,00</td>
<td>0,00</td>
<td>-3,60</td>
<td>0,00</td>
<td>0,00</td>
<td>0,00</td>
</tr>
<tr>
<td>LC1</td>
<td>LS1</td>
<td>8,152</td>
<td>-0,01</td>
<td>0,00</td>
<td>8,37</td>
<td>0,00</td>
<td>0,00</td>
<td>0,00</td>
</tr>
<tr>
<td>LC1</td>
<td>LS1</td>
<td>10,000</td>
<td>-0,03</td>
<td>0,00</td>
<td>-18,53</td>
<td>0,00</td>
<td>0,00</td>
<td>0,00</td>
</tr>
</tbody>
</table>
Displaying the stress on members

SCIA Engineer calculates various stress components for each member. Simple stress is given here considering neither code checks nor stability check (buckling, lateral-torsional buckling).

<table>
<thead>
<tr>
<th>Normal</th>
<th>Normal stress in the member.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shear</td>
<td>Shear stress in the member.</td>
</tr>
<tr>
<td>Von Mises</td>
<td>Von Mises (or equivalent) stress in the member.</td>
</tr>
<tr>
<td>Fatigue</td>
<td>The stress variation between the maximum and minimum stress in each fibre for the selected load cases or combinations.</td>
</tr>
<tr>
<td>Kappa</td>
<td>The stress ratio. This ratio is used in some fatigue check rules (e.g. DIN).</td>
</tr>
</tbody>
</table>

The procedure for displaying of stress

1. **Open service Results.**
2. Select function Member stress (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function (see Note below).
4. If required, redraw the screen using button Redraw in the Property window.
5. Review the results.

The procedure for displaying of stress in Preview window

1. **Open service Results.**
2. Select function Member stress (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function (see Note below).
4. Use function Print / Preview data to create a table in the Preview window.
   1. either use menu function File > Print data > Print / Preview data,
   2. or use function Print data > Print / Preview data on toolbar Project.
5. Review the results.

Displaying the shear stress

SCIA Engineer calculates shear stress in joint of phased cross-sections.
dx/h  See paragraph Shear stress calculated from the difference in axial forces 
(below)

Values  Two values are available: shear stress and increment of normal forces

The procedure for displaying of stress

1. Open service Results.
2. Select function Shear stress (just click the function, do not open it by double-clicking).
3. Adjust the parameters of the function.
4. If required, redraw the screen using button Redraw in the Property window.
5. Review the results.

Shear stress calculated from the difference in axial forces

The shear stress calculated by function Shear stress is determined from the difference of axial forces along the length of the beam. The calculation of shear stress in a joint from the difference of axial forces is the same as the Grasshof theory, but it is applicable only to lateral load. This theory cannot be applied for the effects of a longitudinal load (e.g. shrinkage and creep of concrete between two cross-section phases).

![Diagram showing shear stress calculation](image)

where \( F_{c1} \), \( F_{c2} \), \( F_{c3} \) are resultants of axial forces in individual cross-section phases.

Joint 1

\[
F_{c1} + \tau_{x1} * b_{j1} * dx = F_{c1} + dF_{c1}
\]

\[
\tau_{x1} = \frac{dF_{c1}}{b_{j1} * dx}
\]

Joint 2

\[
F_{c1} + F_{c2} + \tau_{x2} * b_{j2} * dx = F_{c1} + dF_{c1} + F_{c2} + dF_{c2}
\]

\[
\tau_{x2} = \frac{(dF_{c1} + dF_{c2})}{b_{j2} * dx}
\]

or

\[
F_{c3} + \tau_{x2} * b_{j2} * dx = F_{c3} + dF_{c3}
\]

\[
\tau_{x2} = \frac{dF_{c3}}{b_{j2} * dx}
\]
Procedure to calculate the shear stress in a joint

1) In the Result service it is possible to obtain the values of axial forces in individual cross-section phases.

2) These values of axial forces are averaged along the length of the finite element (if the values in the start-point and the end-point differ) and this averaged value is put to the centre of the finite element.

\[ N_i = \frac{N_{i-1} + N_i}{2} \]

3) A function is created that passes through the centres of the finite elements (to make it simpler, the y-axis is placed to the centre of the i-th element). The value of IFE is the distance between the adjacent elements Ni and Ni+1 ==> the length of the finite element.

\[ N_i(x) = \left( \frac{N_{i+1} - N_i}{l_{FE}} \right) \cdot x + N_i \]

4) The above described function is used to calculate the value of axial force in the section-being-evaluated between the finite elements (in the distance equal to the half of the finite element length).

\[ N_i(dx) = \left( \frac{N_{i+1} - N_i}{l_{FE}} \right) \cdot \frac{l_{FE}}{2} + N_i \]

5) The value of Ni(dx) is the starting value for the calculation of shear in the joint in the section-being-evaluated.

6) Next, the difference in the axial force \( \Delta N \) along the segment length dx. We obtain the value of \( \Delta N \) when we subtract the values of axial forces in the distance of dx/2 to the right and to the left from the section-being-evaluated, respectively.

\[ N_i(dx)^- = \left( \frac{N_{i+1} - N_i}{l_{FE}} \right) \cdot \left( \frac{l_{FE}}{2} - \frac{dx}{2} \right) + N_i \]

\[ N_i(dx)^+ = \left( \frac{N_{i+1} - N_i}{l_{FE}} \right) \cdot \left( \frac{l_{FE}}{2} + \frac{dx}{2} \right) + N_i \]

\[ \Delta N = N_i(dx)^- - N_i(dx)^+ \]

the segment dx is determined using the user-defined ratio dx/h (see the property dialogue of the Shear stress function), where h is the total height of the cross-section.

7) The shear stress is then calculated:

\[ \tau = \frac{\Delta N}{dx \cdot b_j} \]

with bj being the width of the joint.
Selecting the joints for display of connection forces

Usually, the user will probably use module Connections for design and checking of connections in the structure. However, it may be sometimes useful to perform a manual design and fast checking of an individual joint manually.

SCIA Engineer enables the user to select required joints (or nodes), define the "configuration" of the connection and review easily the internal forces acting in these connections (or nodes).

The term "configuration" in this context means the basic arrangement of the connection. If no "configuration" is adjusted, the internal forces in the connection are equal to zero, as each joint of the structure must be in equilibrium, which is one of the principles of numerical method applied for calculation. In order to obtain the required internal forces, it is necessary to define:

- which 1D member (entering the joint) is the owner of the connection,
- which other 1D members contribute to the connection (i.e. the internal forces of which 1D members are transferred into the connection).

Let’s assume a node where four 1D members meet. Two 1D members are vertical and two are horizontal. The joint then looks like a simple cross. If such a joint is selected and no other adjustment is made, the resultant internal forces will be equal to zero and won’t be shown.

If, however, one of the 1D members (e.g. the bottom vertical 1D member) is selected as the owner of the connection, the function shows internal forces that are transferred into the joint from the remaining three 1D members. The connection should be then designed to resist these forces.
**Chapter 10**

*The procedure for selection of required nodes and definition of the "configuration" of connection*

1. **Open service Results.**
2. **Activate (doubleclick) function Connection input.**
3. If required, type the name of the connection.
4. **Select the co-ordinate system. The internal forces will be then determined in the selected system.**
5. **Confirm with [OK].**
6. **Select node or nodes where the connection forces should be displayed. A circular mark is drawn around each of the selected nodes.**
7. **Close the function.**
8. **If more than one connection has been defined, clear the selection and select the first one.**
9. **The Property window displays the parameters of the connection including all the entering 1D members.**
10. **Select, i.e. unmark, the 1D member that is the "owner" of the connection.**
11. **Select, i.e. mark, all the 1D members that contribute to the connection.**
12. **If necessary, clear the selection and select another connection.**

**Displaying the connection forces**

Usually, the user will probably use module Connections for design and checking of connections in the structure. However, it may sometimes be useful to perform a manual design and fast checking of an individual joint manually. SCIA Engineer provides for a simple and fast determination of internal forces acting in selected joints (nodes).

Once the node and the configuration of the connection are defined, the internal forces may be displayed.

*The procedure to display the forces in connection on a screen*

1. **Open service Results.**
2. **Select function Connection forces (just click the function, do not open it by double-clicking).**
3. **Adjust the parameters of the function (see below).**
4. **If required, redraw the screen using button Redraw in the Property window.**
5. **Review the results.**

*The procedure to display the forces in connection in the Preview window*

1. **Open service Results.**
2. **Select function Connection forces (just click the function, do not open it by double-clicking).**
3. **Adjust the parameters of the function (see below).**
4. **Use function Print / Preview data to create a table in the Preview window.**

---

Note: The internal forces in the connection may be then displayed using function Connection forces.
1. either use menu function File > Print data > Print / Preview data,
2. or use function Print data > Print / Preview data on toolbar Project.
3. Review the results.

Parameters of function Connection forces

<table>
<thead>
<tr>
<th>Redraw</th>
<th>This item invokes a regeneration of the screen when the button is pressed.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selection</td>
<td>The results may be shown in either All or User-defined entities.</td>
</tr>
<tr>
<td>Type of load</td>
<td>The results for load case, load case combination or class may be displayed.</td>
</tr>
<tr>
<td>Load case / Com-</td>
<td>This item provides for selection of a particular load case or combination for the display.</td>
</tr>
<tr>
<td>bination</td>
<td></td>
</tr>
<tr>
<td>Filter</td>
<td>Here, the user may limit the selection to specific entities only.</td>
</tr>
<tr>
<td>Values</td>
<td>Either all or only selected quantities may be shown.</td>
</tr>
<tr>
<td>Individual com-</td>
<td>If the previous item is set to More components, the user may specify which particular component should be drawn.</td>
</tr>
<tr>
<td>components</td>
<td></td>
</tr>
<tr>
<td>Drawing setup</td>
<td>This item enables the user to adjust the view parameters for the result diagrams.</td>
</tr>
<tr>
<td>Extreme</td>
<td>This item performs no action for this function.</td>
</tr>
<tr>
<td>Section</td>
<td>This item performs no action for this function.</td>
</tr>
</tbody>
</table>

Note: More information about display settings for results may be found in chapter Opening the service Results and Displaying the internal forces.

Displaying the calculation report

If required, the user may display (and subsequently print) a report summarising all the important about carried out calculation.

The procedure to display the calculation report

1. Open service Results.
2. Select function Calculation protocol (just click the function, do not open it by double-clicking).
3. Select the type of calculation you require to be reported.
4. Use function Print > Print / Preview table to create the report.
5. A brief summarising table is shown in the Preview window.

Note: If you double-click the Calculation protocol function in service Results, a small preview window is opened on the screen. This window contains the required information about the last performed calculation.

Displaying the results in tabular form

Any of available results can be displayed in tabular form in the Preview window.
Chapter 10

The basic principle is explained in chapter Document > Creating the document > Inserting a new section into document from the graphical window.

The same approach is applied to get the required result values into the Preview window. The user just has to use function Print/Preview table.

Displaying the results in named fibres

Named fibres of a cross-section

SCIA Engineer enables the user to name selected fibres of selected cross-sections. These named fibres can be then referred to in the Results service in function Member Stress and the calculated stresses can be displayed in these named fibres.

The standard procedure consists of two steps: (i) naming the required fibres in the Editing dialogue for the selected cross-section, (ii) referring to the named fibres in function Member stress in service Results.

Named items dialogue

The first step is made in the Named Items dialogue that consists of the following parts and controls.

| Cross-section parts | This window lists all parts of the selected cross-section. For "normal" cross-sections there is just one line here. For composite cross-sections consisting of multiple partial cross-sections the list is longer. |
| Cross-section fibres | This window contains all the vertices of the selected cross-section. The list offers the vertices for all the cross-section parts listed in the top window. |
| Graphical window | This window shows the graphical representation of the cross-section shape. It also highlights the vertex and/or part that is currently selected in the lists. |
| [OK] and [Cancel] buttons | These buttons close the dialogue. [OK] confirms the changes made, [Cancel] abandons them. |

Named fibres in function Results > Member stress

Property window in function Results > Member stress is extended by a couple of options to enable you to refer to the named fibres.

- All
  The stress is displayed in all fibres (i.e. the "envelope" for the stress is displayed)
- Top
  The stress is displayed in the top fibres of the cross-section.
- Bottom
  The stress is displayed in the bottom fibres of the cross-section.
- Named fibre
  You may specify the fibre in which the stress is to be displayed.

Note: You must remember the names of the fibres defined in the Named Items dialogue as you are required to type the name in the
Procedure to name the fibres

1. Open the Cross-section manager.
2. Select the required cross-section.
3. Open the Cross-section editing dialogue.
4. In the properties table find item Edit named items and press the three-dot button [..] next to it.
5. The Named items dialogue is opened on the screen.
6. If required, type the names of the cross-section parts (you are not obliged to name all the parts unless you want so).
7. If required, type the names of the selected fibres (you are not obliged to name all the fibres unless you want so).
8. If required, you may:
   1. invoke a pop-up menu in the graphical window of the dialogue and employ some basic display-related functions, or
   2. use combination "Press-and-hold keys Ctrl+Shift" + "Press-and-hold the mouse right button" and zoom-in or zoom-out the drawing, or
   3. use combination "Press-and-hold key Shift" + "Press-and-hold the mouse right button" and move the drawing around the graphical window of the dialogue.
9. Confirm the Named items dialogue with [OK].
10. Confirm the Cross-section editing dialogue with [OK].
11. Close the Cross-section manager.

Procedure to display the results in the given named cross-section part

1. You must have the named cross-section parts defined.
2. Run the calculation and open service Results.
3. Select function Beams > Member stress.
4. In the Property table go to item Cross-section parts.
5. Select option Named item.
Chapter 10

6. A new input box called Named item is added to the table.
7. Type the name of the required cross-section part.
8. Refresh the screen using the action button.

| Note: You must remember the names of the cross-section parts defined in the Named items dialogue as you are required to type the name in the input field. |

Procedure to display the results in the named fibre

1. You must have the named fibres defined.
2. Run the calculation and open service Results.
3. Select function Beams > Member stress.
4. In the Property table go to item Fibres.
5. Select option Named item.
6. A new input box called Named item is added to the table.
7. Type the name of the required fibre.
8. Refresh the screen using the action button.

| Note: You must remember the names of the fibres defined in the Named items dialogue as you are required to type the name in the input field. |

| Note: Option Named fibres is of higher priority that the option Named cross-section parts. Therefore, once you select Named fibres in the property table, the item Cross-section parts is hidden. |

Example

Let us have a solid rectangular cross-section as in figure below. Further, let us name the fibre number 4 "MY TOP" and fibre number 8 "MY BOTTOM".

| Note: The vertex (fibre) numbers are generated automatically by the program and cannot be altered by the user. |
Let input a beam fully fixed on both its ends and subject it to the self-weight. The deflection diagram clearly indicates which part of the top and bottom surface of the beam is subjected to tension and which interval of the top and bottom surface is under compression. (Top surface: towards the end tension occurs, in the middle the face is under compression. Bottom surface: it is vice versa).

When displaying stress Normal + and Normal- for the user-defined MY TOP and MY BOTTOM fibres, the diagrams look like:

bottom – compression:

bottom – tension:
Note: The named fibres and named cross-sections work for stresses in 1D members only.

Displaying the stress distribution over the cross-section

SCIA Engineer provides for the display of stress distribution over the cross-section of 1D members.

The procedure to display the stress distribution over the cross-section

1. Open service Results.
2. Select function Beams > Member stress.
3. Set Fibres to All.
4. Set Drawing to 3D stress diagram.
5. Click the three-dot button in item Selection tool and select the 1D members and sections in these 1D members where the stress diagram is to be drawn.
6. Select the quantity to be displayed.
7. If required, make other adjustments (e.g. Extreme, Drawing setup, etc.).
8. Refresh the screen.

Fast selection of result quantities for the display

Whenever service Results is opened, a new toolbar appears at the top of command line. Individual buttons switch on appropriate diagrams.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>N</td>
<td>axial force</td>
</tr>
<tr>
<td>Vy</td>
<td>shear force Vy</td>
</tr>
<tr>
<td>Vz</td>
<td>shear force Vz</td>
</tr>
<tr>
<td>Mx</td>
<td>bending moment Mx</td>
</tr>
<tr>
<td>My</td>
<td>bending moment My</td>
</tr>
<tr>
<td>Mz</td>
<td>bending moment Mz</td>
</tr>
<tr>
<td>ux</td>
<td>displacement ux on 1D members</td>
</tr>
<tr>
<td>uy</td>
<td>displacement uy on 1D members</td>
</tr>
</tbody>
</table>
### Displaying the natural frequencies

The calculated eigenfrequencies (natural frequencies) may be displayed in summarised form in a preview table.

**The procedure to display the table with eigenfrequencies**

1. If it is not the case, perform dynamic calculation of the project.
2. Open service Results.
3. Double click function Eigen frequencies

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>uz</td>
<td>displacement uz on 1D members</td>
</tr>
<tr>
<td>deformed structure</td>
<td>displacement of nodes</td>
</tr>
<tr>
<td>Rx</td>
<td>force reaction Rx</td>
</tr>
<tr>
<td>Ry</td>
<td>force reaction Ry</td>
</tr>
<tr>
<td>Rz</td>
<td>force reaction Rz</td>
</tr>
<tr>
<td>Mx</td>
<td>moment reaction Mx</td>
</tr>
<tr>
<td>My</td>
<td>moment reaction My</td>
</tr>
<tr>
<td>Mz</td>
<td>moment reaction Mz</td>
</tr>
</tbody>
</table>

**Note:** If the command line is hidden, the toolbar does not appear. In order to see the toolbar, display the command line first using function View > Toolbars.
**Displaying the eigenmodes**

The calculated eigenmodes may be displayed as any other result.

*The procedure to see the modes on the screen*

1. If it is not the case, perform dynamic calculation of the project.
2. Open service **Results**.
3. Select function **Deformations of nodes**.
4. Item **Type of loads** set to **Mass combinations**.
5. Item **Value** set to **Deformed mesh**.
6. In item **Mass combination** select the required eigenmode.
7. Press button **[Redraw]** to see the result.

**Example**

First five eigenmodes of a simple frame whose horizontal bar is subject to a distributed load are shown in the following pictures. Mass has been **automatically generated from the load**.

### Results on beams

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>10,05</td>
<td>63,14</td>
<td>3987,19</td>
<td>0,10</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>25,48</td>
<td>160,10</td>
<td>25631,16</td>
<td>0,04</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>62,77</td>
<td>364,35</td>
<td>155514,45</td>
<td>0,02</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>69,42</td>
<td>436,17</td>
<td>190246,49</td>
<td>0,01</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>92,12</td>
<td>578,19</td>
<td>334995,30</td>
<td>0,01</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>1,59</td>
<td>10,02</td>
<td>100,38</td>
<td>0,63</td>
</tr>
<tr>
<td></td>
<td>2</td>
<td>3,51</td>
<td>22,05</td>
<td>488,05</td>
<td>0,28</td>
</tr>
<tr>
<td></td>
<td>3</td>
<td>10,30</td>
<td>64,87</td>
<td>4209,65</td>
<td>0,10</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>20,75</td>
<td>130,36</td>
<td>16993,85</td>
<td>0,05</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>34,79</td>
<td>218,58</td>
<td>47775,86</td>
<td>0,03</td>
</tr>
</tbody>
</table>
Evaluating the results for harmonic load

Once the calculation has been finished, the user may review the results the same way s/he is accustomed to doing for static calculations.

In addition to standard result quantities, some additional result can be found in the calculation report. These are:

- \( \omega, \) period, frequency,
- participation coefficients: \( wx, i/wx, tot, wy, i/wy, tot, wz, i/wz, tot. \)

The above-mentioned values are stated for every calculated eigenmode.

Calculation of internal forces in ribs

When calculating internal forces in a rib (see the procedure below to learn how to switch this feature on), a substitute T-section is used to calculate the results. The web of this T-section is formed by the rib-beam itself, the flange of the T-section is made of the appropriate effective width of the slab. The effective width of the slab is then used to determine internal forces from the slab that must be added to the internal forces calculated in the rib itself. The internal forces in the slab are transformed into the local coordinate system of the rib before the integration.

- \( T \) the centre of gravity of the whole substitute T-section
- \( T1 \) the centre of gravity of the left hand side part of the effective width
- \( T2 \) the centre of gravity of the right hand side part of the effective width
- \( T3 \) the centre of gravity of the original rib

\[
\begin{align*}
Lever \; Arm \; Z1 &= T1z - Tz \\
Lever \; Arm \; Y1 &= T1y - Ty
\end{align*}
\]

The coordinates of centres of gravity are used to determine lever arms in Y and Z direction:
The final internal forces in the rib can be calculated from the formulas below:

\[
N = N_{\text{beam}} + N_{\text{slab, left}} + N_{\text{slab, right}}
\]

\[
V_y = V_y_{\text{beam}} + V_y_{\text{slab, left}} + V_y_{\text{slab, right}}
\]

\[
V_z = V_z_{\text{beam}} + V_z_{\text{slab, left}} + V_z_{\text{slab, right}}
\]

\[
M_x = M_x_{\text{beam}} + M_x_{\text{slab, left}} + M_x_{\text{slab, right}} + V_z_{\text{slab, left}} \times (Lever\ \text{Arm}\ \text{Y}_1) + V_z_{\text{slab, right}} \times (Lever\ \text{Arm}\ \text{Y}_2)
\]

\[
M_y = M_y_{\text{beam}} + M_y_{\text{slab, left}} + M_y_{\text{slab, right}} + N_{\text{slab, left}} \times (Lever\ \text{Arm}\ \text{Z}_1) - N_{\text{slab, right}} \times (Lever\ \text{Arm}\ \text{Z}_2) + N_{\text{beam}} \times Lever\ \text{Arm}\ \text{Z}_3;
\]

\[
M_z = M_z_{\text{beam}} + M_z_{\text{slab, left}} + M_z_{\text{slab, right}} + N_{\text{slab, left}} \times (Lever\ \text{Arm}\ \text{Y}_1) - N_{\text{slab, right}} \times (Lever\ \text{Arm}\ \text{Y}_2) + N_{\text{beam}} \times Lever\ \text{Arm}\ \text{Y}_3;
\]

**The procedure to recalculate internal forces in the rib**

1. Open service Results.
2. Select function Beams > Internal forces on beams.
3. Select the 1D member(s) where the results should be displayed.
4. Select the quantity to be displayed.
5. In the property dialogue select option Rib.
6. Press button Refresh to see the result diagram.

**Evaluation of the results on the phased CSS**

This chapter briefly explains how the program handles with the evaluation of the internal forces on the phased CSS. General prescription is that the internal forces on the phased CSS are evaluated to centre of gravity of whole phased CSS. The most efficient way how to explain is to use the following examples. Imagine two phased CSS. The centre of the gravity is marked in the figure. During the definition of the structure we can move the eccentricity of the centre axis. Then if Point force -10kN is acting on the beam and only the 1st phase of CSS exists only the torsional moment are following.
Chapter 10

\[ M_x = 10 \times 0.15 - 20 \times 0.15 = -1.5 \text{kNm} \]
\[ M_y = 30 \times 0.15 = 4.5 \text{kNm} \]
\[ M_z = -30 \times 0.15 = -4.5 \text{kNm} \]

\[ M_x = 10 \times 0 - 20 \times 0 = 0 \text{kNm} \]
\[ M_y = 30 \times 0 = 0 \text{kNm} \]
\[ M_z = -30 \times 0 = 0 \text{kNm} \]

\[ M_x = 10 \times 0.25 - 20 \times 0.15 = -0.5 \text{kNm} \]
\[ M_y = 30 \times 0.15 = 4.5 \text{kNm} \]
\[ M_z = -30 \times 0.25 = -7.5 \text{kNm} \]
The same results you will get if you evaluate the same point force in the stage when also the 2nd phase of CSS exists.

3D displacement

Since version 14.1 it is possible to display displacements also on surfaces of 2D members

The new result 3D displacement for checking the displacements of the structure is available in version 14.

The major possibilities of this result are:

- the result enables to show displacements of surfaces of 1D and 2D members as an isolines.
- the result enables to show deformed surface of 1D and 2D members

Available magnitudes which can be displayed are:

- $ux, uy, uz$ - displacements of particular fibres in local coordinate system of the member
- $\varphi_x, \varphi_y, \varphi_z$ - rotations of section in local coordinate system
- $U_{global}$ - global displacement of particular fibre. $U_{global} = \sqrt{(ux^2+uy^2+uz^2)}$

Properties of the result

<table>
<thead>
<tr>
<th>Properties</th>
<th>3D displacement (1)</th>
</tr>
</thead>
</table>

- **Name**

- **Result case**

- **Type of load**
- Combosinations

- **Combination**
- C01

- **Envelope**
- Maximum

- **Selection**

- **Type of selection**
- All

- **Filter**
- No

- **Value**
- U global

- **Extreme**
- Global

- **Deformed structure**

- **Deformed structure**
- Yes

- **Member deformation**
- for global maximum

- **For drawing use linear rotation**
- No

- **Use user scale isolines**
- No

- **Drawing setup 2D**

Result case

The property group Result case enables to specify the loading conditions for which the result is calculated. It enables to select from available Types of load (Load case, Combination, Nonlinear combination, Result class, ...) and then the
particular instance for selected Type of load.

For envelope combinations or for result classes it is possible to switch between maximal or minimal values. This selection influences the values displayed as isolines. In each node of the isolines the maximum or minimum is taken as the result value.

**Selection**

The group selection enables to modify the range of entities where the results are displayed. It is possible to combine available selection types with filters. The final range of entities is the intersection of selection and the filter.

**Value and Extreme**

User can select between available magnitudes which are described above.

The selected values is also used as the primary magnitude in the output tables. The extremes are searched for this selected value.

Setting of Extreme is respected in the tabular output only. The extreme setting for the graphical output is inside the Drawing setup 2D.

The displacement of particular fibres $U$ are influenced also by rotation of sections. Therefore there are e.g. non zero displacements caused by the pure torsion:

![Diagram showing displacement and rotation](image)

The displacements are calculated using the small displacement assumption. The translations of particular fibres caused by the rotation of sections are linear.

The rotation $\phi$ causes translation of point "0" to the point "2":

- 56 -
Draw result on...

It is possible to switch ON/OFF drawing of results on 1D members or faces of 2D members.

Positive face is the face which is on +z local coordinate of the 2D member, negative face is on -z local coordinate of the 2D member.

Deformed structure

The result values can be displayed also on deformed surface of the structure.
Chapter 10

The deformed surface is displayed for the combination from selected result case which provides the extreme value of selected magnitude. In case the result type contains envelope combination (different combinations can give extreme values for each member of the structure) it is possible to select between "Member deformation for global / local maximum".

- **Member deformation for global maximum** means that the deformation of surfaces is calculated just for one exact combination which gives global extreme of selected magnitude.
• **Member deformation for member maximum** means that the extreme combination is evaluated for each member separately. In this case the deformed surface can be displayed for different combinations per each member and there can be discontinuities in deformed surface.

The checkbox "For drawing use linear rotation" enables to switch between small displacement assumption based linear translations (which provides result consistent with used assumption) and circular translations (which keeps the original shape of cross section).

• Linear translation ON

![Image of deformed surface with linear translation ON]

• Linear translation OFF

![Image of deformed surface with linear translation OFF]

**3D stress**
Since version 14.1 it is possible to display stresses also on surfaces of 2D members.

Since version 15.3 it is possible to disable calculation of additional torsion effect from eccentric shear forces (see paragraph "Torsion from shear force eccentricity").

The new result 3D stress for checking stresses on the structure is available in version 14.

The major possibilities of this result are:
- the result enables to show stresses on surfaces of 1D members and 2D members as an isolines.
- stresses can be shown on deformed surface of 1D members and 2D members

There are available three types of magnitudes:
- Basic (magnitudes in local coordinate system of finite elements)
- Principal magnitudes
- Membrane magnitudes (only for 2D members)

<table>
<thead>
<tr>
<th>Magnitude</th>
<th>1D members</th>
<th>2D members</th>
</tr>
</thead>
<tbody>
<tr>
<td>Basic magnitudes</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\sigma_x$</td>
<td>Normal -</td>
<td>( \sigma_x )</td>
</tr>
<tr>
<td>$\sigma_y(2D)$</td>
<td>Normal +</td>
<td>( \sigma_y )</td>
</tr>
<tr>
<td>$\sigma_{xy}(2D)$</td>
<td>shear stresses in local coordinates (thick walled cross sections) or total shear flow (thin walled cross sections)</td>
<td>( \sigma_{xy} )</td>
</tr>
<tr>
<td>$\tau_{xy}/\tau_{xz}(1D)$</td>
<td></td>
<td>( \tau_{xy}/\tau_{xz} )</td>
</tr>
<tr>
<td>$\tau_{xz}/\tau_{yz}(1D)$</td>
<td></td>
<td>( \tau_{xz}/\tau_{yz} )</td>
</tr>
<tr>
<td>$\tau_{xx}(2D)$</td>
<td></td>
<td>( \tau_{xx} )</td>
</tr>
<tr>
<td>$\tau_{yz}(2D)$</td>
<td></td>
<td>( \tau_{yz} )</td>
</tr>
<tr>
<td>Principal magnitudes</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\sigma_E$</td>
<td>Equivalent stress (von Misses) ( \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_{xy}^2} )</td>
<td>Equivalent stress (von Misses) ( \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_{xy}^2} )</td>
</tr>
<tr>
<td>$\sigma_1$</td>
<td>( \sigma_1 = \sigma_x )</td>
<td>Principal stress ( \frac{1}{2}(\sigma_x + \sigma_y + \sqrt{(\sigma_x - \sigma_y)^2 + 4\tau_{xy}^2}) )</td>
</tr>
<tr>
<td>$\sigma_2$</td>
<td></td>
<td>Principal stress ( \frac{1}{2}(\sigma_x + \sigma_y - \sqrt{(\sigma_x - \sigma_y)^2 + 4\tau_{xy}^2}) )</td>
</tr>
</tbody>
</table>
### Results on beams

<table>
<thead>
<tr>
<th></th>
<th>Angle of principal stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>$\pm \sqrt{\tau_{xy}^2 + \tau_{xz}^2}$</td>
</tr>
<tr>
<td>$\tau_{xy}/\tau_{xs}$</td>
<td>$\pm \sqrt{\tau_{xy}^2 + \tau_{xs}^2}$</td>
</tr>
<tr>
<td>$\tau_{xz}/\tau_{xs}$</td>
<td>$\pm \sqrt{\tau_{xz}^2 + \tau_{xs}^2}$</td>
</tr>
</tbody>
</table>

Shear stresses in local coordinates (thick walled cross sections) or total shear flow (thin walled cross sections)

Maximum shear stress perpendicular to the plane

<table>
<thead>
<tr>
<th>Membrane magnitudes</th>
<th>Membrane equivalent stress (Mises)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\sigma_m E$</td>
<td>$\sqrt{(\sigma_m^E)^2 + \sigma_{m1}^2 + \sigma_{m2}^2}$</td>
</tr>
<tr>
<td>$\sigma_m 1$</td>
<td>$\frac{1}{2}(\sigma_{m1} + \sigma_{m2} - \sqrt{\sigma_{m1}^2 + \sigma_{m2}^2})$</td>
</tr>
<tr>
<td>$\sigma_m 2$</td>
<td>$\frac{1}{2}(\sigma_{m1} + \sigma_{m2} + \sqrt{\sigma_{m1}^2 + \sigma_{m2}^2})$</td>
</tr>
<tr>
<td>$\alpha_m$</td>
<td>$\pm \frac{1}{2}(\tau_{xy} + \tau_{xz} - \tau_{xs})$</td>
</tr>
</tbody>
</table>

The shear stresses are evaluated differently for **thick-walled** and **thin-walled** cross sections. For thick-walled (solid) sections the real $\tau_{xy}$ or $\tau_{xz}$ are calculated and displayed. For thin-walled sections the final shear flow from unit forces in Y or Z directions are evaluated and displayed as total shear flow $\tau_{xs} = \sqrt{(\tau_{xy}^2 + \tau_{xz}^2)}$. See also chapters in manual related to calculation of sectional characteristics.
Properties of the result

**Result case**

The property group Result case enables to specify the loading conditions for which the result is calculated. It enables to select from available Types of load (Load case, Combination, Nonlinear combination, Result class, ...) and then the particular instance for selected Type of load.

For envelope combinations or for result classes it is possible to switch between maximal or minimal values. This selection influences the values displayed as isolines. In each node of the isolines the maximum or minimum is taken as the result value.

**Selection**

The group selection enables to modify the range of entities where the results are displayed. It is possible to combine available selection types with filters. The final range of entities is the intersection of selection and the filter.

**Value and Extreme**

User can select between available magnitudes which are described above.
The selected magnitude is also used as the primary magnitude in the output tables. The extremes are searched for this selected value.

Setting of Extreme is respected in the tabular output only. The extreme setting for the graphical output is available inside the Drawing setup 2D.

**Torsion from shear force eccentricity**

Shear stresses on 1D members are calculated by default from shear forces and torsion moment, where shear forces are considered to act in the shear centre of the beam's cross-section. In case of cross-section with different position of shear centre with respect to the centroid of gravity this approach leads to discrepancy in the position of acting forces. Typical example of such cross-section is hot-rolled U shaped cross-section:

![Cross-section diagram](image)

It is possible now to calculate and use the values of combined shear forces, bending moment and torsion moment with respect to the centroid of gravity. This functionality is enabled in result service “3D Stress” by check box “Consider torsion due to shear centre eccentricity”.

- 63 -
With this option on, the shear forces are considered to act in the centre of gravity and therefore the shear stresses are magnified by additional torsion effect from eccentricity of shear forces.

In versions prior 15.3 this option is always considered as turned on by default.

**Draw result on...**

It is possible to switch ON/OFF drawing of results on 1D members or faces of 2D members.

**Positive face** is the face which is on +z local coordinate of the 2D member, **negative face** is on -z local coordinate of the 2D member.
Deformed structure

The result stress values can be displayed also on the deformed surface of the structure.
Chapter 10

For more details please see 3D displacement.

Calculation of stresses for 1D members

Result values are calculated in fibres. Position of fibres is visible in the cross section editing dialogue.

- $\sigma_x$ is calculated on assumption of linear distribution of the strain within the section. In the first step the plane of deformation is calculated using iterative algorithm () so strain in all fibres is known. In the next step adequate stresses are assigned using stress-strain curve. Remember that linear calculation uses linear approximation ($E$ modulus) of stress-strain curve.

- $\tau_{xy}/\tau_{xs} \cdot \tau_{xz}/\tau_{xs}$ are calculated from the shear force and torsional moment using unit distribution of shear stresses. Those unit distributions of shear stresses are also available in the cross section editing dialogue.

The $\tau_{xs}$ magnitude represents the total shear flow which is in the thin-walled cross section. Distribution of this shear flow from horizontal a vertical unit shear forces can be checked in the cross section manager.
Thin walled cross section:

Results on beams

\[ V_z = 1 \text{kN} \]

Unit shear stresses for a unit shear force \( V_z \)
Chapter 10

Thick walled cross section:
Limitations

It is necessary to take into account that results are calculated in fibres only. Values between fibres are linearly interpolated. In some cases this precondition results in omitting of some local extremes. Example of this phenomenon is visible on following pictures:

There is a thick-walled I cross section exposed to horizontal shear force:
The third picture shows distribution of shear XY on the edge between fibres 7, 9. The theoretical distribution is shown by the dashed light blue line, the red line shows distribution which is presented by the 3D stress result. The reason is quite clear. The distribution of stresses is linear interpolation between values in fibres and the only fibre between fibres 7 and 9 is fibre 8 which has quite low value. Therefore the higher shear stress on exceeding parts of flanges are not respected on this edge.

This higher shear stress from exceeding part of flanges is displayed correctly on the inner edge 10-11 as is also visible on the picture.
# Results on slabs

## Calculated results for 2D members

### Internal forces

#### Principal moments – plate

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>m1</td>
<td>Principal moment (max)</td>
<td>( \frac{1}{2} \left( m_x + m_y + \sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}^2} \right) )</td>
</tr>
<tr>
<td>m2</td>
<td>Principal moment (min)</td>
<td>( \frac{1}{2} \left( m_x + m_y - \sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}^2} \right) )</td>
</tr>
<tr>
<td>alfa (bending)</td>
<td>Angle of principal moment m1</td>
<td>( \frac{1}{2} \left( \arctan \left( \frac{2m_{xy}}{m_y - m_x} \right) \right) )</td>
</tr>
<tr>
<td>mtmax</td>
<td>Maximum torque moment</td>
<td>( \sqrt{(m_x - m_y)^2 + 4 \cdot m_{xy}^2} )</td>
</tr>
<tr>
<td>qmax (bending)</td>
<td>Maximum shear force perpendicular</td>
<td>( \sqrt{u_x^2 + u_y^2} )</td>
</tr>
<tr>
<td>beta</td>
<td>Angle of maximum shear force</td>
<td>( \arctan \left( \frac{u_y}{u_x} \right) )</td>
</tr>
</tbody>
</table>

#### Principal forces – wall

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>n1</td>
<td>Principal force (max)</td>
<td>( \frac{1}{2} \left( n_x + n_y + \sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}^2} \right) )</td>
</tr>
<tr>
<td>n2</td>
<td>Principal force (min)</td>
<td>( \frac{1}{2} \left( n_x + n_y - \sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}^2} \right) )</td>
</tr>
<tr>
<td>alfa (membrane)</td>
<td>Angle of principal force n1</td>
<td>( \frac{1}{2} \left( \arctan \left( \frac{2n_{xy}}{n_y - n_x} \right) \right) )</td>
</tr>
<tr>
<td>qmax (membrane)</td>
<td>Maximum shear force in the plane</td>
<td>( \sqrt{(n_x - n_y)^2 + 4 \cdot n_{xy}^2} )</td>
</tr>
</tbody>
</table>

### Design moments from EC2 – plate

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
</table>
| mxo+    | Design moment in x-direction on positive surface | (1) \(-m_x + |m_{xy}|\) if \(m_x \leq m_y\) and \(m_y \leq |m_{xy}|\)  
(2) \(-m_x + |m_{xy}|\) if \(m_x > m_y\) and \(m_x \leq |m_{xy}|\)  
(3) \(-m_x + \frac{m_{xy}^2}{|m_y|}\) if \(m_x \leq m_y\) and \(m_y > |m_{xy}|\)  
(4) 0 if \(m_x > m_y\) and \(m_x > |m_{xy}|\) |
### Upper and lower surface of 2D member is determined by the Z axis direction of local coordinate system (LCS). Upper surface is in the positive direction of the Z axis and on the other hand. Lower surface is in negative direction of Z axis. Upper surface values are marked with + and lower values are marked with -.

### Design moments, classical method – plate

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>myD+</td>
<td>Design moment in y-direction on positive face</td>
<td>$m_y +</td>
</tr>
<tr>
<td>mxD+</td>
<td>Design moment in x-direction on positive face</td>
<td>$m_x +</td>
</tr>
<tr>
<td>mcD+</td>
<td>Design moment in the concrete on positive face</td>
<td>$-2 \cdot</td>
</tr>
<tr>
<td>mxO-</td>
<td>Design moment in x-direction on negative face</td>
<td>$-m_x +</td>
</tr>
<tr>
<td>myO-</td>
<td>Design moment in y-direction on negative face</td>
<td>$-m_y +</td>
</tr>
<tr>
<td>mcD-</td>
<td>Design moment in the concrete on negative face</td>
<td>$-2 \cdot</td>
</tr>
</tbody>
</table>
### Design forces from EC2 – wall

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>nxD</td>
<td>Design force in x-direction</td>
<td>[ n_x + n_{xy} \text{ if } n_x \leq n_y \text{ and } n_x \geq -n_{xy} ]  [ n_x + n_{xy} \text{ if } n_x &gt; n_y \text{ and } n_x \geq -n_{xy} ]  [ 0 \text{ if } n_x \leq n_y \text{ and } n_x &lt; -n_{xy} ]  [ n_x + \frac{(n_{xy})^2}{n_y} \text{ if } n_x &gt; n_y \text{ and } n_x &lt; -n_{xy} ]</td>
</tr>
<tr>
<td>nyD</td>
<td>Design force in y-direction</td>
<td>[ n_y + n_{xy} \text{ if } n_x \leq n_y \text{ and } n_x \geq -n_{xy} ]  [ n_y + n_{xy} \text{ if } n_x &gt; n_y \text{ and } n_x \geq -n_{xy} ]  [ 0 \text{ if } n_x &gt; n_y \text{ and } n_x \leq n_y &lt; -n_{xy} ]  [ n_y + \frac{(n_{xy})^2}{n_x} \text{ if } n_x \leq n_y \text{ and } n_x &lt; n_{xy} ]</td>
</tr>
<tr>
<td>ncD</td>
<td>Design force in the concrete</td>
<td>[ -2n_{xy} \text{ if } n_x \leq n_y \text{ and } n_x \geq -n_{xy} ]  [ -2n_{xy} \text{ if } n_x &gt; n_y \text{ and } n_x \geq -n_{xy} ]  [ -n_x - \frac{(n_{xy})^2}{n_x} \text{ if } n_x \leq n_y \text{ and } n_x &lt; -n_{xy} ]  [ -n_y - \frac{(n_{xy})^2}{n_y} \text{ if } n_x &gt; n_y \text{ and } n_y &lt; -n_{xy} ]</td>
</tr>
</tbody>
</table>

### Design forces, classical method – wall

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>nxD</td>
<td>Design force in x-direction</td>
<td>[ n_x + n_{xy} ]</td>
</tr>
<tr>
<td>nyD</td>
<td>Design force in y-direction</td>
<td>[ n_y + n_{xy} ]</td>
</tr>
</tbody>
</table>
### Results on Slabs

<table>
<thead>
<tr>
<th>nD</th>
<th>Design force in the concrete</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$-2n_{xy}$ if $n_x \leq n_y$ and $n_x \geq -n_{xy}$</td>
</tr>
<tr>
<td></td>
<td>$-n_x - \left(\frac{n_{xy}}{n_z}\right)^2$ if $n_x \leq n_y$ and $n_x &lt; -n_{xy}$</td>
</tr>
<tr>
<td></td>
<td>$-n_y - \left(\frac{n_{xy}}{n_z}\right)^2$ if $n_x &gt; n_y$ and $n_y &lt; -n_{xy}$</td>
</tr>
</tbody>
</table>

The calculation of the design forces for the plates and shells according to the EC2 algorithm uses the flow diagram in ČSN P ENV 1992–1–1 (731201), Appendix 2, par. A2.8. Following rules are valid for the indices:

$m_a \geq m_b : a = x, b = y$

$m_a < m_b : a = y, b = x$

---

The calculation of the design forces for the walls and shells according to EC2 algorithm uses the flow diagram in ČSN P ENV 1992–1–1 (731201), Appendix 2, par. A2.9. Following rules are valid for the indices:

$n_a \geq n_b : a = x, b = y$

$n_a < n_b : a = y, b = x$
Chapter 11

**Magnitudes** $m_xD$ and $m_yD$ (or $n_xD$ and $n_yD$) are the design moments (or forces) in the reinforcement. The negative values of the moments or forces have no practical meaning and they are featured only by reason of integrity.

Magnitude $m_cD$ (or $n_cD$) is design moment (or force) in the concrete and together with design moments (or forces) in the reinforcement it forms unity triplet with respect to invariant:

$$m_x + m_y = m_i + m_z = m_{xD} + m_{yD}$$

$$- m_x - m_y = -m_i - m_z = m_{xD} + m_{yD}$$

$$n_x + n_y = n_i + n_z = n_{xD} + n_{yD}$$

The design force in the concrete $n_cD$ is used at concrete crushing check (see ČSN P ENV 1992–1–1 (731201), Appendix 2, par. A2.9). The design moments in the concrete $m_cD$ are not mentioned in this code, but their meaning is analogical and they are stated by reason of integrity.

The values of the design moments and forces according to „classical“ algorithm are calculated according to the left branches of the flow diagrams stated above, i.e. without respect to the relation between $m_x$, $m_y$ and $m_{xy}$ (or $n_x$, $n_y$ a $n_{xy}$). This calculation is on the safe side, but in some cases it could be less optimal.

The right branch is used if some of the directions of the reinforcement calculated according to the left branch is pressed (negative value of the correspondent design magnitude). In this case a zero value of the design magnitude is assigned to this direction, and the other direction will have lower value of design magnitude (and consequently lower necessary reinforcement area) than by calculation according to the left branch (the condition of the capacity is granted in both cases). The calculation according to the right branch causes higher pressure in the concrete (magnitudes $m_cD$ and $n_cD$) than according to the left branch. By this point of view the EC2 algorithm could be considered as more cost-effective.

### 2. Stresses, group=3

#### Basic stresses on 2D

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>sigxb (not disp.)</td>
<td>Stress by bending moments</td>
<td>$\frac{6 \cdot m_x}{h^2}$</td>
</tr>
<tr>
<td>sigyb (not disp.)</td>
<td>Stress by bending moments</td>
<td>$\frac{6 \cdot m_y}{h^2}$</td>
</tr>
<tr>
<td>sigzb (not disp.)</td>
<td>Stress by bending moments</td>
<td>$\frac{6 \cdot m_{xy}}{h^2}$</td>
</tr>
<tr>
<td>sigxm (not disp.)</td>
<td>Stress by normal forces</td>
<td>$\frac{m_x}{h}$</td>
</tr>
<tr>
<td>sigym (not disp.)</td>
<td>Stress by normal forces</td>
<td>$\frac{m_y}{h}$</td>
</tr>
<tr>
<td>sigz furnish (not disp.)</td>
<td>Stress by normal forces</td>
<td>$\frac{m_{xy}}{h}$</td>
</tr>
<tr>
<td>sigx+</td>
<td>Stress on positive face</td>
<td>$\frac{n_x}{h} + \frac{6 \cdot m_x}{h^2}$</td>
</tr>
<tr>
<td>sigy+</td>
<td>Stress on positive face</td>
<td>$\frac{n_y}{h} + \frac{6 \cdot m_y}{h^2}$</td>
</tr>
<tr>
<td>sigxy+</td>
<td>Stress on positive face</td>
<td>$\frac{n_{xy}}{h} + \frac{6 \cdot m_{xy}}{h^2}$</td>
</tr>
<tr>
<td>sigx-</td>
<td>Stress on negative face</td>
<td>$\frac{n_x}{h} - \frac{6 \cdot m_x}{h^2}$</td>
</tr>
<tr>
<td>Label</td>
<td>Description</td>
<td>Calculation</td>
</tr>
<tr>
<td>-------</td>
<td>-------------</td>
<td>-------------</td>
</tr>
<tr>
<td>sig1+</td>
<td>Principal stress on positive face (max)</td>
<td>( \frac{1}{2} \left( \sigma_{xx} + \sigma_{yy} + \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + 4 \sigma_{xy}^2} \right) )</td>
</tr>
<tr>
<td>sig2+</td>
<td>Principal stress on positive face (min)</td>
<td>( \frac{1}{2} \left( \sigma_{xx} + \sigma_{yy} - \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + 4 \sigma_{xy}^2} \right) )</td>
</tr>
<tr>
<td>alfa+</td>
<td>Angle of principal stress ( \sigma_{1+} )</td>
<td>( \frac{1}{2} \arctan \left( \frac{2 \sigma_{xy}}{\sigma_{xx} - \sigma_{yy}} \right) )</td>
</tr>
<tr>
<td>sigE+</td>
<td>Equivalent stress on positive face (Mises)</td>
<td>( \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 - 2 \sigma_{xy} \sigma_{xy} + 3 \sigma_{xy}^2} )</td>
</tr>
<tr>
<td>sig1-</td>
<td>Principal stress on negative face (max)</td>
<td>( \frac{1}{2} \left( \sigma_{xx} + \sigma_{yy} + \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + 4 \sigma_{xy}^2} \right) )</td>
</tr>
<tr>
<td>sig2-</td>
<td>Principal stress on negative face (min)</td>
<td>( \frac{1}{2} \left( \sigma_{xx} + \sigma_{yy} - \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + 4 \sigma_{xy}^2} \right) )</td>
</tr>
<tr>
<td>alfa-</td>
<td>Angle of principal stress ( \sigma_{1-} )</td>
<td>( \frac{1}{2} \arctan \left( \frac{2 \sigma_{xy}}{\sigma_{xx} - \sigma_{yy}} \right) )</td>
</tr>
<tr>
<td>sigE-</td>
<td>Equivalent stress on negative face (Mises)</td>
<td>( \sqrt{\sigma_{xx}^2 + \sigma_{yy}^2 - 2 \sigma_{xy} \sigma_{xy} + 3 \sigma_{xy}^2} )</td>
</tr>
<tr>
<td>tau_maxb</td>
<td>Maximum shear stress perpendicular to the plane</td>
<td>( \sqrt{\tau_{max}^2 + \tau_{y,z}^2} )</td>
</tr>
<tr>
<td>sigZ</td>
<td>Stress for plane stress</td>
<td>( \mu \left( \sigma_{xx} + \sigma_{yy} \right) )</td>
</tr>
<tr>
<td>sigmE</td>
<td>Membrane equivalent stress (Tresca,Rankine)</td>
<td>( \max \left( \sqrt{\left( \sigma_{xxm} - \sigma_{yym} \right)^2 + 4 \sigma_{xyzm}^2}, \frac{1}{2} \left( \sigma_{xxm} + \sigma_{yym} + 2 \sigma_{xyzm} \right) \right) )</td>
</tr>
<tr>
<td>sigEmax</td>
<td>Maximum equivalent stress (Mises)</td>
<td>( \max(\sigma_{xx} + \sigma_{yy}, \sigma_{xx} - \sigma_{yy}) )</td>
</tr>
<tr>
<td>sigE</td>
<td>Membrane equivalent stress (Mises)</td>
<td>( \sqrt{\sigma_{zz}^2 + 2 \sigma_{zx}^2 - \sigma_{zy}^2} )</td>
</tr>
<tr>
<td>sigT+</td>
<td>Equivalent stress on positive face (Tresca)</td>
<td>( \max(\sigma_{1+},0) - \min(\sigma_{2+},0) )</td>
</tr>
<tr>
<td>sigT-</td>
<td>Equivalent stress on negative face (Tresca)</td>
<td>( \max(\sigma_{1-},0) - \min(\sigma_{2-},0) )</td>
</tr>
<tr>
<td>sigTmax</td>
<td>Maximum equivalent stress (Tresca)</td>
<td>( \max(\sigma_{1+}, \sigma_{1-}) )</td>
</tr>
<tr>
<td>sigmT</td>
<td>Membrane equivalent stress (Tresca)</td>
<td>( \max(\sigma_{1m},0) - \min(\sigma_{2m},0) )</td>
</tr>
<tr>
<td>sigR+</td>
<td>Equivalent stress on positive face (Rankine)</td>
<td>( \max(\sigma_{1+}, \sigma_{2+}) )</td>
</tr>
<tr>
<td>sigR-</td>
<td>Equivalent stress on negative face (Rankine)</td>
<td>( \max(\sigma_{1-}, \sigma_{2-}) )</td>
</tr>
<tr>
<td>sigRmax</td>
<td>Maximum equivalent stress (Rankine)</td>
<td>( \max(\sigma_{1+}, \sigma_{1-}) )</td>
</tr>
<tr>
<td>sigmR</td>
<td>Membrane equivalent stress (Rankine)</td>
<td>( \max(\sigma_{1m}, \sigma_{2m}) )</td>
</tr>
</tbody>
</table>
Chapter 11

**Stresses on 3D**

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>sig1, sig2, sig3</td>
<td>Principal stresses</td>
<td>Eigenvectors of stress matrix σ</td>
</tr>
<tr>
<td>τmaxB</td>
<td>Maximum shear stress</td>
<td>[ \frac{1}{2}(\text{max} - \text{min}) ]</td>
</tr>
<tr>
<td>sigEM</td>
<td>Equivalent stress (Mises)</td>
<td>[ \frac{1}{2} \left( \sigma_1 + \sigma_2 + \sigma_3 \right) ]</td>
</tr>
<tr>
<td>sigET</td>
<td>Equivalent stress (Tresca)</td>
<td>[ \max \left( \sigma_1 - \sigma_2, \sigma_2 - \sigma_3, \sigma_3 - \sigma_1 \right) ]</td>
</tr>
<tr>
<td>sigER</td>
<td>Equivalent stress (Rankine)</td>
<td>[ \max \left( \sigma_1, \sigma_2, \sigma_3 \right) ]</td>
</tr>
<tr>
<td>sigEB</td>
<td>Equivalent stress (Bach)</td>
<td>[ \max \left( \sigma_1 - \mu(\sigma_2 + \sigma_3), \sigma_2 - \mu(\sigma_3 + \sigma_1), \sigma_3 - \mu(\sigma_1 + \sigma_2) \right) ]</td>
</tr>
</tbody>
</table>

**Strains, group=14, Plastic strains, group=15**

**Strains on 2D**

Basic strains are calculated using following formula:

\[ \varepsilon = T D^{-1} T^T m \]

a) **Bending strains**

\[ \varepsilon = \begin{bmatrix} \frac{\partial \varepsilon_x}{\partial x} & \frac{\partial \varepsilon_y}{\partial x} & \frac{\partial \gamma_{xy}}{\partial x} \\ \frac{\partial \varepsilon_x}{\partial y} & \frac{\partial \varepsilon_y}{\partial y} & \frac{\partial \gamma_{xy}}{\partial y} \end{bmatrix} = \begin{bmatrix} \cos^2 \alpha & \sin^2 \alpha & \sin \alpha \cos \alpha \\ \sin^2 \alpha & \cos^2 \alpha & -\sin \alpha \cos \alpha \\ -2 \sin \alpha \cos \alpha & 2 \sin \alpha \cos \alpha & \cos^2 \alpha + \sin^2 \alpha \end{bmatrix} \begin{bmatrix} D_{11} & D_{12} & 0 \\ D_{12} & D_{22} & 0 \\ 0 & 0 & D_{33} \end{bmatrix} \begin{bmatrix} m_{x1} \\ m_{x2} \end{bmatrix} \]

b) **Shear strains**

\[ \varepsilon = \begin{bmatrix} \frac{\partial \gamma_{xy}}{\partial x} & \frac{\partial \gamma_{yx}}{\partial y} \\ \frac{\partial \gamma_{xy}}{\partial y} & \frac{\partial \gamma_{yx}}{\partial x} \end{bmatrix} = \begin{bmatrix} \cos \alpha & \sin \alpha \\ -\sin \alpha & \cos \alpha \end{bmatrix} \begin{bmatrix} 0 & 0 \\ D_{44} & D_{55} \end{bmatrix} \begin{bmatrix} m_{y} \end{bmatrix} \]

c) **Membrane strains**
\[ \varepsilon = \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \end{bmatrix} = \begin{bmatrix} \cos^2 \alpha & \sin \alpha \cos \alpha & \sin \alpha \cos \alpha \\ \sin^2 \alpha & \cos^2 \alpha & -\sin \alpha \cos \alpha \\ -2 \sin \alpha \cos \alpha & 2 \sin \alpha \cos \alpha & -\sin^2 \alpha \end{bmatrix} D \begin{bmatrix} d_{1x} \\ d_{2x} \\ 0 \\ 0 \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix} \]

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\varepsilon_{x+})</td>
<td>Strain on positive surface</td>
<td>(\frac{\partial v}{\partial x} + \frac{1}{2} \frac{\partial \varphi_y}{\partial x})</td>
</tr>
<tr>
<td>(\varepsilon_{y+})</td>
<td>Strain on positive surface</td>
<td>(\frac{\partial v}{\partial y} + \frac{1}{2} \frac{\partial \varphi_x}{\partial y})</td>
</tr>
<tr>
<td>(\gamma_{x+y+})</td>
<td>Slope on positive surface</td>
<td>(\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{1}{2} \frac{\partial \varphi_x}{\partial y} - \frac{1}{2} \frac{\partial \varphi_y}{\partial x})</td>
</tr>
<tr>
<td>(\varepsilon_{x-})</td>
<td>Strain on negative surface</td>
<td>(\frac{\partial v}{\partial x} + \frac{1}{2} \frac{\partial \varphi_y}{\partial x})</td>
</tr>
<tr>
<td>(\varepsilon_{y-})</td>
<td>Strain on negative surface</td>
<td>(\frac{\partial v}{\partial y} + \frac{1}{2} \frac{\partial \varphi_x}{\partial y})</td>
</tr>
<tr>
<td>(\gamma_{x+y-})</td>
<td>Slope on negative surface</td>
<td>(\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{1}{2} \frac{\partial \varphi_x}{\partial y} + \frac{1}{2} \frac{\partial \varphi_y}{\partial x})</td>
</tr>
<tr>
<td>(\varepsilon_{1+})</td>
<td>Principal strain on positive surface (max)</td>
<td>(\varepsilon_{x+} + \frac{1}{2} \left( (\varepsilon_{x+} - \varepsilon_{y+})^2 + \gamma_{x+y+}^2 \right))</td>
</tr>
<tr>
<td>(\varepsilon_{2+})</td>
<td>Principal strain on negative surface (min)</td>
<td>(\varepsilon_{x-} - \frac{1}{2} \left( (\varepsilon_{x-} + \varepsilon_{y-})^2 + \gamma_{x+y-}^2 \right))</td>
</tr>
<tr>
<td>(\alpha_{x+})</td>
<td>Angle of principal strain on positive surface</td>
<td>(\frac{1}{2} \arctan \left( \frac{\gamma_{x+y+}}{\varepsilon_{x+} - \varepsilon_{y+}} \right))</td>
</tr>
<tr>
<td>(\alpha_{x-})</td>
<td>Angle of principal strain on negative surface</td>
<td>(\frac{1}{2} \arctan \left( \frac{\gamma_{x+y-}}{\varepsilon_{x-} - \varepsilon_{y-}} \right))</td>
</tr>
<tr>
<td>(\varepsilon_{M+})</td>
<td>Equivalent strain on positive surface (Mises)</td>
<td>(\left( \varepsilon_{x+} - \varepsilon_{y+} \right)^2 + \left( \varepsilon_{x+} + \mu \varepsilon_{y+} \right)^2 + \left( \frac{\mu + \varepsilon_{x+}}{1 - \mu} \right)^2 \left( \frac{\mu + \varepsilon_{x+}}{1 - \mu} \right)^2 + \frac{3}{2} \gamma_{x+y+}^2 - 2(1 + \mu))</td>
</tr>
<tr>
<td>(\varepsilon_{M-})</td>
<td>Equivalent strain on negative surface (Mises)</td>
<td>(\left( \varepsilon_{x-} - \varepsilon_{y-} \right)^2 + \left( \varepsilon_{x-} + \mu \varepsilon_{y-} \right)^2 + \left( \frac{\mu + \varepsilon_{x-}}{1 - \mu} \right)^2 \left( \frac{\mu + \varepsilon_{x-}}{1 - \mu} \right)^2 + \frac{3}{2} \gamma_{x+y-}^2 - 2(1 + \mu))</td>
</tr>
<tr>
<td>(\varepsilon_{M})</td>
<td>Maximum equivalent strain (Mises)</td>
<td>(\max(\varepsilon_{M+}, \varepsilon_{M-}))</td>
</tr>
<tr>
<td>(\varepsilon_{R+})</td>
<td>Equivalent strain on positive surface (Rankine)</td>
<td>(\varepsilon_{x+} + \frac{1}{2} \left( (\varepsilon_{x+} - \varepsilon_{y+})^2 + \gamma_{x+y+}^2 \right))</td>
</tr>
<tr>
<td>(\varepsilon_{R-})</td>
<td>Equivalent strain on negative surface (Rankine)</td>
<td>(\varepsilon_{x-} + \frac{1}{2} \left( (\varepsilon_{x-} - \varepsilon_{y-})^2 + \gamma_{x+y-}^2 \right))</td>
</tr>
<tr>
<td>(\varepsilon_{R})</td>
<td>Maximum equivalent strain (Rankine)</td>
<td>(\max(\varepsilon_{R+}, \varepsilon_{R-}))</td>
</tr>
</tbody>
</table>
Chapter 11

### Strains on 3D

<table>
<thead>
<tr>
<th>Label</th>
<th>Description</th>
<th>Calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\varepsilon_{1}$, $\varepsilon_{2}$, $\varepsilon_{3}$</td>
<td>Principal strains</td>
<td>Eigenvectors of strain matrix</td>
</tr>
<tr>
<td>$\varepsilon_{M}$</td>
<td>Equivalent strain (Mises)</td>
<td>$1 \sqrt{\frac{\varepsilon_{1}^2 + \varepsilon_{2}^2 + \varepsilon_{3}^2 - \varepsilon_{1} \varepsilon_{2} - \varepsilon_{2} \varepsilon_{3} - \varepsilon_{3} \varepsilon_{1}}{3(1-\mu)}}$</td>
</tr>
<tr>
<td>$\varepsilon_{T}$</td>
<td>Equivalent strain (Tresca)</td>
<td>$\max(\varepsilon_{1}, \varepsilon_{2}, \varepsilon_{3})$</td>
</tr>
<tr>
<td>$\varepsilon_{R}$</td>
<td>Equivalent strain (Rankine)</td>
<td>$\max(\varepsilon_{1}, \varepsilon_{2}, \varepsilon_{3})$</td>
</tr>
<tr>
<td>$\varepsilon_{B}$</td>
<td>Equivalent strain (Bach)</td>
<td>$\max(\varepsilon_{1}, \varepsilon_{2}, \varepsilon_{3})$</td>
</tr>
</tbody>
</table>

$R_1, R_2, R_3$ are the eigenvalues of the matrix $R$:

$$R = \frac{1 - \mu}{1 + \mu} \begin{pmatrix} \frac{1}{2} (1 - \mu) \varepsilon_{1}^2 + \mu \varepsilon_{2}^2 + \varepsilon_{3}^2 & \frac{1}{2} (1 - \mu) \varepsilon_{2}^2 + \mu \varepsilon_{3}^2 + \varepsilon_{1}^2 & \frac{1}{2} (1 - \mu) \varepsilon_{3}^2 + \mu \varepsilon_{1}^2 + \varepsilon_{2}^2 \\ \frac{1}{2} \varepsilon_{1} \varepsilon_{2} & \frac{1}{2} \varepsilon_{2} \varepsilon_{3} & \frac{1}{2} \varepsilon_{3} \varepsilon_{1} \\ \frac{1}{2} \varepsilon_{1} \varepsilon_{3} & \frac{1}{2} \varepsilon_{2} \varepsilon_{1} & \frac{1}{2} \varepsilon_{3} \varepsilon_{2} \end{pmatrix}$$

### Displaying the deformation of nodes on slabs

*The procedure to display the deformation of nodes*

1. **Open service Results.**
2. Select function 2D members > Deformation of nodes.
3. Select the slabs for the display of results.
4. Select the required type of loads.
5. Select the quantity to be displayed.
6. Select the drawing style.
7. If required, change the Drawing setup.
8. Set any other parameter.
9. If necessary, regenerate the diagrams.
Note: This function displays deformation of both slabs and 1D members.

The isolines/isobands of deformation can be displayed either on the original (nondeformed) structure or on the deformed one. This can be selected in option Standard in the Property window when the function 2D members > Deformation of nodes is opened.

See also Style of isolines.

Displaying the internal forces on slabs

The procedure to display the internal forces

1. Open service Results.
2. Select function 2D members > Internal forces.
3. Select the slabs for the display of results.
4. Select the required type of loads.
5. Select the quantity to be displayed.
6. Select the drawing style.
7. If required, change the Drawing setup.
8. Set any other parameter.
9. If necessary, regenerate the diagrams.

Parameters for display of results

Name
Specifies the name of the current result quantity.

Selection
Specifies on which slabs the results are to be displayed. Read chapter Selecting the 1D members for display for more information.

Type of loads
The results can be displayed for calculated load cases or combinations or classes.

Load cases / Combinations / Class
This item select the particular load case / combination / class for the display.

Filter
The display can be limited to slabs of certain name, material, thickness, etc.

System
The result quantities (except those displayed in principal directions) can be displayed in several coordinate systems.
Local = local coordinate system of individual finite elements.
UCS = user-defined coordinate system
UCS polar = user-defined polar coordinate system
LCS - Member 2D = local coordinate system of the 2D element

Rotation
The results can be displayed in the direction that is rotated by the given angle from the direction specified above.
Averaging of peaks
If ON, the peak values in the corners of 2D members are averaged.

Location
The program calculates result values in the nodes of individual finite elements. If required, these results can be further processed to obtain "better" displayed values. For more read chapter Averaging of results in FE nodes.

Type of forces
It is possible to select from three types of result values:
Basic magnitude = Results in local slab axes are displayed.
Principal magnitude = Results in principal axes are evaluated.
Dimensional magnitude = Quantities for design are calculated.

Envelope
For envelope combinations and for result classes, it is possible to select the minimum or maximum "branch" of the envelope should be displayed.

Drawing
The results can be displayed using several different techniques:
Standard = isolines / isobands are used.
Section = distribution of the quantity along defined section(s) is displayed
Resultant = the resultant over defined sections is displayed
Section+ Standard = combines the two above-mentioned techniques
Trajectories = the trajectories of the quantity are displayed (useful e.g. for principal magnitudes).
Read also chapter Isolines Setup for more information and illustrative examples

Values
Here the required quantity is selected.

Text output
This parameter is available only if type of load is set to "class".

Extreme
This parameter says what type extreme is indicated in the screen.

Drawing setup
This button can be used to set additional parameter for the display style.

Type of forces
As mentioned above, there are three different types of force. The following tables summarise individual options.

Basic magnitude

Project: plate

mx
bending moment of 2D member in x direction
my
bending moment of 2D member in y direction
mxy
torsion moment of 2D member
vx
shear force perpendicular to plane of 2D member in x direction
vy
shear force perpendicular to plane of 2D member in y direction
Results on slabs

Project: wall

\( n_x \)
normal force of 2D member in x direction

\( n_y \)
normal force of 2D member in y direction

\( n_{xy} \)
membrane shear force of 2D member

Project: general (shell)

\( m_x \)
bending moment of 2D member in x direction

\( m_y \)
bending moment of 2D member in y direction

\( m_{xy} \)
torsion moment of 2D member

\( v_x \)
shear force perpendicular to plane of 2D member in x direction

\( v_y \)
shear force perpendicular to plane of 2D member in y direction

\( n_x \)
normal force of 2D member in x direction

\( n_y \)
normal force of 2D member in y direction

\( n_{xy} \)
membrane shear force of 2D member

Principal magnitude

Note: Lower index "m" at the quantity name means the membrane component. Lower index "b" at the quantity name means the bending component.

Project: plate

<table>
<thead>
<tr>
<th>( m_1, m_2 )</th>
<th>principal moments</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>angle between the direction of ( m_1 ) and planar axis ( x_P )</td>
</tr>
<tr>
<td>( q_{\text{max}} )</td>
<td>maximal shear force</td>
</tr>
</tbody>
</table>

Project: wall

<table>
<thead>
<tr>
<th>( n_1, n_2 )</th>
<th>principal axial forces</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>angle between the direction of ( n_1 ) and planar axis ( x_P )</td>
</tr>
</tbody>
</table>

Project: general (shell)

| \( m_1, m_2 \) | principal moment |
Design magnitude

<table>
<thead>
<tr>
<th>Project: plate</th>
<th>mxD+, myD+, mxD-, myD-, mcD-</th>
</tr>
</thead>
<tbody>
<tr>
<td>Project: wall</td>
<td>nxD, nyD, ncD</td>
</tr>
<tr>
<td>Project: general (shell)</td>
<td>mxD+, myD+, mxD-, myD-, mcD-, nxD, nyD, ncD</td>
</tr>
</tbody>
</table>

Design moments in slabs that are related to the surface with positive element coordinate and are marked with + (plus sign). Dimension moments in slabs that are related to the surface with negative element coordinate and are marked with – (minus sign).

Design forces in a wall are in the middle plane.

Corresponding surface of action of design moments in shells is given directly by the sign of the moment.

See also chapters Principal internal forces and Design internal forces.

See also chapter Style of isolines.

Note: To activate the use of redistribution strips read chapter Results > Results on slabs > Redistribution strips > Displaying the redistributed results.

**Principal internal forces**

The calculation of principal bending forces is performed to the formula below.

The calculation of principal membrane forces is performed to the formula below.

\[
m_{12} = \frac{1}{2} \left( m_x + m_y \pm \sqrt{(m_x - m_y)^2 + 4 m_x^2} \right)
\]

\[
\alpha = \frac{1}{2} \arctg \frac{2m_y}{m_x - m_y} \quad (m_y \leq 0, \quad \alpha \in (-90, 0); \quad m_y > 0, \quad \alpha \in (0, 90))
\]

\[
g_{ax} = \sqrt{g_x^2 + g_y^2}
\]
Design internal forces

The calculation of design moments for plates and shells according to the EC2 algorithm (option EC2 is selected) follows the flow chart from CSN P ENV 1992–1–1 (731201), Annex 2, paragraph A2.8.

The following rule is used for indexes:

\[
\begin{align*}
m_y \geq m_x &: a = x, b = y \\
m_y < m_x &: a = y, b = x
\end{align*}
\]

---

The calculation of design moments for walls and shells according to the EC2 algorithm (option EC2 is selected) follows the flow chart from CSN P ENV 1992–1–1 (731201), Annex 2, paragraph A2.9.

The following rule is used for indexes:

\[
\begin{align*}
n_y \geq n_x &: a = x, b = y \\
n_y < n_x &: a = y, b = x
\end{align*}
\]
Quantities $m_{xD}$ and $m_{yD}$ (respectively $n_{xD}$ and $n_{yD}$) are design moments (respectively forces) in the reinforcement. Negative design moments have no practical meaning and are stated just for the reason of completeness.

Quantity $m_{cD}$ (resp. $n_{cD}$) is design moment (resp. force) in concrete and these two quantities form an integral trio with design moments (resp. forces) in the reinforcement in terms of invariant.

$$m_{cD} = m_x + m_y - m_1 + m_2 - m_{xD} + m_{yD} - m_{zD}$$

Design force in concrete $n_{cD}$ is used for checking of concrete crushing (see CSN P ENV 1992–1–1 (731201), Annex 2, paragraph A2.9). The standard does not mention the design moments in concrete $m_{cD}$, but their meaning is analogous and are stated for the reason of completeness.

Values of design moments and forces according to the standard algorithm (option EC2 is NOT selected) are calculated according to the left branch of the above mentioned flow charts, i.e. no account is taken of the relation between $m_x$, $m_y$ and $m_{xy}$ (respectively $n_x$, $n_y$ and $n_{xy}$). This approach is on the safe side (see below) but is less optimal.

The right branch of the flow charts is used if the left branch of the flow charts would lead to one reinforcement direction in compression (negative value of the corresponding quantity). This direction is assigned zero value of the design quantity, the value in the other direction (and also the necessary reinforcement area) is then lower than it would be if the right branch of the flow charts were followed (the condition of completeness is met in both variants). The difference is in increased compression in concrete ($m_{cD}$ and $n_{cD}$). In this respect the EC2 algorithm can be considered as more economic.

Displaying the stresses on slabs

The procedure to display the stresses

1. Open service Results.
2. Select function 2D members > Stresses.
3. Select the slabs for the display of results.
4. Select the required type of loads.
5. Select the quantity to be displayed.
6. Select the drawing style.
7. If required, change the Drawing setup.
8. Set any other parameter.
9. If necessary, regenerate the diagrams.

**Available stress values**

**Project: plate, shell**

| \( \sigma_{1+}, \sigma_{2+} \) | Principal stress at the surface with positive planar z-coordinate |
| \( \alpha_+ \) | Angle between the direction of \( \sigma_{1+} \) and planar axis \( x_P \) |
| \( \sigma_{E+} \) | Equivalent stress at the surface with positive planar z-coordinate |
| \( \sigma_{1-}, \sigma_{2-} \) | Principal stress at the surface with negative planar z-coordinate |
| \( \alpha_- \) | Angle between the direction of \( \sigma_{1-} \) and planar axis \( x_P \) |
| \( \sigma_{E-} \) | Equivalent stress at the surface with negative planar z-coordinate |
| \( \tau_{\text{maxb}} \) | Maximal transverse shear stress in middle plane |

**Project: wall**

| \( \sigma_{1}, \sigma_{2} \) | Principal stress in middle plane |
| \( \alpha \) | Angle between the direction of \( \sigma_{1} \) and planar axis \( x_P \) |
| \( \sigma_{E} \) | Equivalent stress in middle plane |
| \( \tau_{\text{maxb}} \) | Maximal membrane shear stress in middle plane |

### Stresses

Principal and maximal shear stresses are calculated by means of widely known formulas:

\[
\sigma_{12} = \frac{1}{2} \left( \sigma_{xx} + \sigma_{yy} + \sqrt{(\sigma_{xx} - \sigma_{yy})^2 + 4 \tau_{xy}^2} \right)
\]

\[
\alpha = \frac{1}{2} \arcsin \left( \frac{2 \tau_{xy}}{\sigma_{xx} - \sigma_{yy}} \right) \quad (\tau_{xy} \leq 0, \ \alpha \in (-90, 0], \ \tau_{xy} > 0, \ \alpha \in (0, 90])
\]

\[
\tau_{\text{maxb}} = \sqrt{\tau_{xx}^2 + \tau_{yy}^2}
\]

Equivalent stress is calculated by means of Huber–Mises–Hencky theory:

\[
\sigma_{E} = \sqrt{\sigma_{1}^2 + \sigma_{2}^2 - \sigma_{1} \sigma_{2}}
\]
Chapter 11

Displaying the contact stress on slabs

The procedure to display the contact stresses

1. **Open service Results.**
2. Select function 2D members > Contact stress.
3. Select the slabs for the display of results.
4. Select the **required type of loads.**
5. Select the quantity to be displayed.
6. Select the **drawing style.**
7. If required, change the Drawing setup.
8. Set any other parameter.
9. If necessary, regenerate the diagrams.

See also **Style of isolines.**

Calculated C parameters

The calculated C parameters can be reviewed in 2D data viewer or in service Results.

The procedure to view the C parameters in 2D Data viewer

1. Perform the calculation
2. Open tree Calculation, mesh.
3. Start function 2D data viewer.
4. Select function Subsoil.
5. Select the required parameter.
6. Adjust other drawing parameters.
7. Invoke the refresh of the screen (through button [Refresh] in the property window)
Note: This function offers all five C parameters. The two that are not calculated (C1x and C1y) are constant across the whole ground slab. The other ones may have an arbitrary distribution depending on input boundary conditions.

The procedure to view the C parameters in service Results

1. Perform the calculation
2. Open service Results.
3. Start function Subsoil – C parameters.
4. Select the required parameter.
5. Adjust other drawing parameters.
6. Invoke the refresh of the screen (through button [Refresh] in the property window)
Displaying the settlement

The procedure to display settlement

1. Open service Results.
2. Select function Subsoil - Other data.
3. Select the slabs for the display of results.
4. Click action button [Preview] to see a table with settlement results.

Displaying the soil structure strength

The procedure to display the soil structure strength

1. Open service Results.
2. Select function Subsoil - Other data.
3. Select the slabs for the display of results.
4. Click action button [Soil Stress Diagram].
5. The program generates points in centroids of 2D mesh elements (Vertexes are only generated on those 2D elements which have Soilin support data defined).
6. Select the required point where the diagram is to be displayed.
7. The program opens a dialogue with the diagram.
8. If required, use button Previous or Next to move to another point.
9. Alternatively, select a borehole from the combo box to review the results for a locations of the required borehole.
The Soilin module calculates two stresses: the overstress $\sigma_{z}$ and the original soil stress $\sigma_{or}$. According to theory, settlement will occur if $\sigma_{z} > m * \sigma_{or}$.

The $m$-value is code dependent: (i) for the CSN code it can vary, for EC & DIN it is fixed at 0.2. It practically means that settlement occurs in case the overstress is bigger than 20% of the original soil stress.

The picture shows these two lines: $\sigma_{z}$ in blue and $m * \sigma_{or}$ in brown. The program is looking for the intersection of the two lines: all layers above have $\sigma_{z} > m * \sigma_{or}$ and settlement occurs in them and all the layers below have $\sigma_{z} < m * \sigma_{or}$, which means that no settlement is there. The depth at which the lines intersect is called the "limit depth".

In case the user has not input a sufficient geological profile i.e. not deep enough, the intersection point cannot be determined. It means that the calculated settlement will be too small since there are still deeper layers which will also be compressed and will thus settle. Therefore, the program gives a warning that the geology is "Insufficient".

**Results in membrane elements**

With regard to the theoretical assumptions some of the internal forces in membrane elements are not defined (are zero):

$$mx \quad \text{zero (see the fig. below)}$$

$$my \quad \text{zero}$$
Differences in the results between membrane and standard element

The difference in the obtained results resulting from the application of the membrane behaviour can be best demonstrated on a simple example. Let us assume a rectangular plate made of a very thin sheet of steel. The left-hand side of the figure shows the results obtained for a standard 2D element. The right-hand side then contains the results for the membrane elements.

Moment mx
Displaying results for individual FE nodes or elements

SCIA Engineer offer the option to display in the screen and in the document the numerical result for all loadcases in a selected finite element node or element.

This function is available for all types of results in slabs, walls and shells (i.e. 2D members).

Dialogue for the display of results for individual FE nodes or elements

The dialogue consists of the following parts.

<table>
<thead>
<tr>
<th>Value in</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>element</td>
<td>the values in the nodes of the selected element are displayed,</td>
</tr>
<tr>
<td>node</td>
<td>the value in the given node is displayed.</td>
</tr>
<tr>
<td>Get values</td>
<td>This button reads the appropriate values and displays them in the preview window.</td>
</tr>
<tr>
<td>Preview / Document</td>
<td>This button inserts a table into the document with the appropriate result for the selected node/element.</td>
</tr>
<tr>
<td>Preview window</td>
<td>In this small window the selected results are shown.</td>
</tr>
</tbody>
</table>

The procedure to display the results for individual FE node or element

1. Open service Results.
2. Select the required type of results for 2D members.
3. Click action button Values for loadcase.
Chapter 11

4. The function dialogue is opened on the screen.
5. Specify if the values should be shown for a specific node or element.
6. Click button [Get values] to see the values in the small preview window.
7. If required, click the other button to insert the results into the Document.
8. When ready, close the dialogue.

**Integration strips**

The integration strip is useful when you need to represent results on slabs as on beam members. For example, you may need to display results in a wall like on a column. Another example is a ceiling composed of prefabricated slabs. You need the results per one prefabricated element and treat them as results on a beam. The images below demonstrate some examples of practical application.
The procedure to input an integration strip

1. Start function:
   a) either Structure > 2D member > 2D member component > Integration strip,
   b) or Results > 2D members > Integration strip.
2. Adjust the required parameters (see below) and confirm with [OK].
3. Select the master plate.
4. Define the starting and end point of the strip. If required, you may input a polygonal strip as well.

Properties of the integration strip

Name
This property identifies the integration strip.

**2D member**
(informative) It contains the name of the master 2D member.

**Create mesh nodes**
If ON, nodes of the finite element mesh are generated along the strip (the mesh and results, if exist, are deleted). If OFF, the integration strip is independent on the FE mesh (the mesh and results, if exist, remain intact).

**Effective width definition**
Width = the width of the integration strip is input directly by the user.
Number of thicknesses = the width of the strip is input by the user indirectly as the multiple of the thickness of the slab.

**Effective width geometry**
Constant symmetric = the width of the strip on the left and on the right of the definition line (curve) is the same. The user inputs the total width.
Constant nonsymmetric = the width of the strip is different on the left and right side. The user must specify separately the left and right thickness.

**Width / No. of thicknesses**
Here the user inputs the actual width of the integration strip. Depending on the Effective width geometry either one (total) or two (left + right) values are required.

**Check of geometry**
Geometry check is available for the Integration strip. It can detect strips that are fully outside the master slab. Such strips are deleted.
Use function Check structure data. It can be called from service Structure or from service Calculation, mesh.

**Constraints**
There are no special constraints concerning the input of Integration strip in comparison with ribs.
- overlapping of integration areas is allowed,
- full coverage of slab by integration areas of Integration strips is not required,
- integration areas can extend outside the slab,
- integration areas can overlap with openings,
- integration areas can overlap with adjacent slab(s).

**Displaying of Integration strips**

**Colour + style**
Integration strips are drawn using the same pen as for 1D member. It can be adjusted in Setup > Colours / lines.

**View parameters**
The Integration strip has its own view parameters.
Tab Structure

**Centreline**
ON = the strip is displayed according to view flag Effective width
OFF = the strip is not drawn

**Effective width**
Results on slabs

No = only the definition line of the strip is displayed
Wired = only the contour of the strip is drawn
Rendered = the strip is rendered

Tab Labels

\textbf{Display integration strip label}

ON = the name of the strip is displayed
OFF = no label is shown

Analysis

When property “Create mesh nodes” is ON then an internal edge is generated automatically along the Integration strip centre-line. Then the mesh is generated in the same way as with an ordinary internal edge.

The integration strip does not have any cross section and does not influence the stiffness of the model.

Results

Results on Integration strips are displayed as results on ribs (1D member results). It is necessary to check checkbox “Rib” to see results along the Integration strip.

Internal forces are integrated from 2D member stresses within effective width (as for ribs).

Resulting 2D member internal forces (not included in Integration strips) can be displayed (as for ribs).

If the checkbox “Rib” is not checked, then zero values will be displayed along Integration strips.

Member stress and code checks

Member stress and code checks are not available for Integration strips.

Document

Integration strip can be inserted into document from “New document item”. This way you can insert the table with the definition of Integration strips.

Results on Integration strips are displayed together with other 1D members.

XML

Integration strip can be inserted into XML.

\begin{quote}
Note: Internal forces for the integration are taken only from the "parent slab". The parent slab is displayed in the properties of the integration slab. When an integration strip is copied or moved from one slab to another, the proper parent slab is found, which means that integration strips can be freely copied from one slab to another.
\end{quote}

Isolines, isobands, etc.

Averaging of results in FE nodes

The program calculates result values in the nodes of individual finite elements. If required, these results can be further processed to obtain "better" displayed values.
The user may select from the following options (the options are demonstrated on a slab composed of two halves of different thickness with one half subjected to a uniformly distributed load and the other half without any loading):

<table>
<thead>
<tr>
<th>In centres</th>
<th>The values in centres (centres of gravity) are calculated as an arithmetic average of nodal values of the finite element. The result is a single value for one finite element.</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Diagram" /></td>
<td><img src="image2.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>In nodes, no averaging</th>
<th>These are values provided by the solver. The results are kept pure without any processing.</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image3.png" alt="Diagram" /></td>
<td><img src="image4.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>In nodes, averaging</th>
<th>Nodal values from adjacent finite elements are averaged in every node. The result is a single value for each node and the distribution becomes continuous. Moreover, extrapolation of values is carried out on free edges (the values on the free edge are so modified so that the average value in the element was preserved).</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image5.png" alt="Diagram" /></td>
<td><img src="image6.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>
**Results on slabs**

In nodes, averaging on macro

This is similar to the option above, but the values are not averaged if:

- elements belong to a different 2D member,
- elements are located on different sides of an internal edge,
- an internal point is defined.

For these situations, the distribution is non-continuous and a possible discontinuity (step-change) in the distribution of internal is taken into account, which may result from applied loads, supports, changes of physical properties.

**Isolines setup**

The values set here are used as a default option in Drawing setup dialogues when the results on slabs are drawn in the form of isolines.

*Note:* The averaging may not be available for every result quantity. Only some results may be subject to this type of "postprocessing".
Chapter 11

Styles – isolines / isobands

The style of isolines can be adjusted independently for different kind of representation of results (in centres of finite elements, averaged in nodes, non-averaged in nodes, averaged on macros). These settings are used if the results are displayed with the parameter Drawing set to Standard:

![Settings for Drawing](image)

Averaged values in nodes

![Averaged values](image)
Chapter 11

Isobands

Labelled isolines

Numbers
Constant (centre) values on elements
Non-averaged values in nodes

One colour

Smooth

isolines
Styles – arrows / vectors

The style of isolines can be adjusted independently for different kind of representation of results (in centres of finite elements, averaged in nodes, non-averaged in nodes, averaged on macros). These settings are used if the results are
displayed with the parameter Drawing set to Trajectories:

Averaged values in nodes

One colour

Colour
Results on slabs

Arrow

Coloured arrows

Constant (centre) values on elements

One colour
Non-averaged values in nodes

- One colour
- Colour
- Arrow
Chapter 11

Coloured arrows

Common properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display mesh</td>
<td>If ON, the finite element mesh is displayed.</td>
</tr>
<tr>
<td>Lighting</td>
<td>If ON, a light above the displayed surface is switched on. The colours get brighter.</td>
</tr>
<tr>
<td>Flat shading</td>
<td>The effect of shading is applied.</td>
</tr>
<tr>
<td>Number of isolines</td>
<td>Specifies the number of isolines used.</td>
</tr>
<tr>
<td></td>
<td>The number must be from interval &lt;1, 99&gt;.</td>
</tr>
</tbody>
</table>

Surfaces with isolines

The isolines may be drawn on a "transparent" slab, on a slab in "background" or on a slab of "rendered" colour.

This option is useful if the slabs are in several levels and the view is so adjusted that one slab overlaps the other and hides a part of that slab from your view. See the pictures below.
Isobands style

Palette properties

<table>
<thead>
<tr>
<th>Font</th>
<th>Defines the font of the palette.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Size</td>
<td>Specifies the font size for the palette.</td>
</tr>
</tbody>
</table>
Local extremes

This option allows the user to mark places where the displayed quantity reaches its local extreme. It is possible to display only “minimum peaks” or only the “maximum peaks” or both. Various description options are available.

<table>
<thead>
<tr>
<th>Extreme</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>None</td>
<td>No values are displayed.</td>
</tr>
<tr>
<td>Local minimum and maximum</td>
<td>Both minimum and maximum are displayed.</td>
</tr>
<tr>
<td>Local minimum</td>
<td>Only minimum is displayed.</td>
</tr>
<tr>
<td>Local maximum</td>
<td>Only maximum is displayed.</td>
</tr>
</tbody>
</table>

**Style**

- Transparent description

![Diagram showing a color scale with values from 0.00 to 5.30 kN/m and a graph representing m1.](image)
**Procedure to adjust the isolines parameters**

1. Start menu function Setup > Colours/Lines.
2. Select tab Isolines.
3. Press button [Detailed setup].

**Drawing setup for isolines**

The Drawing setup dialogue is accessible from the property window of any function for display of results. Usually, it is the last item in the Property window.
The three-dot button opens a separate dialogue with settings for a particular quantity and for a particular display style. The Drawing setup dialogue will look differently for the results on 1D members and 2D members, which is quite logical.

**IMPORTANT NOTE**

Moreover, the contents and layout of the Drawing setup dialogue for the results on 2D members will differ depending on some other settings such as:

- type of averaging of results on 2D members (in centres of finite elements, averaged in nodes, non-averaged in nodes, averaged on macros),
- drawing style (standard, section, resultant, trajectories),
- type of result quantity.

The following text will try to summarize the settings that may be present in the Drawing setup dialogue when isolines (standard drawing style) or trajectories are used to display the results.

The Drawing setup dialogue for the drawing style set to section or resultant is identical to the Drawing setup dialogue for 1D members.

**Drawing setup dialogue for the standard drawing style (isolines)**

The dialogue is divided into four parts: Display, Minimum and maximum settings, Ground value, Local extremes. The items in individual parts may differ depending on the style selected in the Display part. The following text focuses on settings that are exclusive to the Drawing setup dialogue and are not available in the Isolines setup dialogue. The meaning of individual options that will not be explained below is shown in chapter **Isolines setup**.

**Display**

<table>
<thead>
<tr>
<th>Display style</th>
<th>The list of options depends on the type of averaging of results on 2D members. Individual options are presented in chapter Isolines setup.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Display mesh</td>
<td>If ON, the finite element is drawn as well. This option is available for relevant display styles.</td>
</tr>
<tr>
<td>Lightning</td>
<td>If ON, the effect of a light is applied.</td>
</tr>
</tbody>
</table>
Results on slabs

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Number of isolines</strong></td>
<td>Defines the number of isolines, i.e. the refinement of the “map” of the result. This option is available for relevant display styles.</td>
</tr>
<tr>
<td><strong>Colour</strong></td>
<td>Defines the colour used for the display. This option is available only if the display style is set to one colour.</td>
</tr>
<tr>
<td><strong>Surfaces with isolines</strong></td>
<td>The isolines may be drawn on a “transparent” slab, on a slab in “background” or on a slab of “rendered” colour. This option is useful if the slabs are in several levels and the view is so adjusted that one slab overlaps the other and hides a part of that slab from your view. Transparent</td>
</tr>
<tr>
<td><strong>Background</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Rendered</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Palette values</strong></td>
<td>The explanation of this parameter is given in a separate chapter Palette values for isobands/isolines (see below).</td>
</tr>
</tbody>
</table>

**Advanced Display settings**

The advanced settings may differ according to the selected display style.

**Advanced settings for isobands**

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Number of isobands</strong></td>
<td>Defines the “refineness” of the scale.</td>
</tr>
</tbody>
</table>
Chapter 11

<table>
<thead>
<tr>
<th>Style</th>
<th>Specifies the style.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Filled</td>
</tr>
<tr>
<td></td>
<td>The bands are fully in colour.</td>
</tr>
<tr>
<td></td>
<td>Inserted isolines</td>
</tr>
<tr>
<td></td>
<td>The bands are not filled with the adjusted colour, just intermediate isolines are drawn in each band (the final display is similar to &quot;labelled isolines&quot;).</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Isoband contour: Display</th>
<th>If ON, the band border is drawn as a solid line.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Isoband contour: Label</td>
<td>If ON, appropriate scale value is attached to each band.</td>
</tr>
<tr>
<td>Predefined palette colours</td>
<td>The user may select one of several predefined colour schemes.</td>
</tr>
<tr>
<td></td>
<td>This is available only if option User-adjustable palette values is selected in the main Drawing setup dialogue.</td>
</tr>
<tr>
<td>Palette colours</td>
<td>It is possible to adjust a user-defined colour for each band.</td>
</tr>
<tr>
<td></td>
<td>This is available only if option User-adjustable palette values is selected in the main Drawing setup dialogue.</td>
</tr>
</tbody>
</table>

Numerical values for individual isobands or isolines can be adjusted by the user. For more read chapter [Palette values for isobands/isolines](#).

**Advanced settings for labelled isolines**

<table>
<thead>
<tr>
<th>Number of isolines with description</th>
<th>Determines the number of drawn labelled isolines – see the examples below.</th>
</tr>
</thead>
<tbody>
<tr>
<td>6 labelled isolines + 1 inserted</td>
<td><img src="image" alt="Diagram of labelled isolines" /></td>
</tr>
</tbody>
</table>

For more information see the examples below.
Results on slabs

Number of inserted isolines
Determines the number of drawn non-labelled isolines inserted between labelled ones – see the examples below.

Coloured isolines
If ON, isolines are in colour. If OFF, isolines are black & white.

Letters
If ON, letters are used instead of numerical values to describe individual isolines.

Minimum and maximum settings
It is possible to define the range of the scale. Normally, the program calculates the range on the basis of the result values. If required, however, the user may decide to change the top and bottom limit value of the scale.

User defined values
If ON, the user-input minimum and maximum values are used for the isolines palette.
If OFF, the automatically calculated minimum and maximum values are used.

Minimum
The (disabled) edit box on the left shows the automatically calculated minimum value.
The user may input the required minimum value to the edit on the right.

Maximum
ditto for the maximum value
### Ground value

If ON, the user may specify a value (zero by default) that is marked in the diagram. Sometimes the zero value may be useful to see where in the structure a specific quantity passes from negative to positive values. Sometimes a specific non-zero value may quickly reveal a place where some quantity exceeds a certain limit.

The picture for example clearly shows where the deformation exceeds 10 mm.

<table>
<thead>
<tr>
<th>Use value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Value</td>
<td>If ON, the user may specify a value (zero by default) that is marked in the diagram. Sometimes the zero value may be useful to see where in the structure a specific quantity passes from negative to positive values. Sometimes a specific non-zero value may quickly reveal a place where some quantity exceeds a certain limit.</td>
</tr>
<tr>
<td></td>
<td>The picture for example clearly shows where the deformation exceeds 10 mm.</td>
</tr>
</tbody>
</table>

**Draw isoline**

This option accompanies the option above. If ON then a line marking the "border" is drawn.

**Use +/- palette**

If ON, not only the border (i.e. the ground value) is drawn but only two colours are used for the diagram – one for "up-to-the-ground-value" interval and the other one for the "above-the-ground-value" interval. (see the picture above and the pictures below)

<table>
<thead>
<tr>
<th>+/- ON</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>If ON, not only the border (i.e. the ground value) is drawn but only two colours are used for the diagram – one for &quot;up-to-the-ground-value&quot; interval and the other one for the &quot;above-the-ground-value&quot; interval. (see the picture above and the pictures below)</td>
</tr>
</tbody>
</table>
Local extremes
This option allows the user to mark places where the displayed quantity reaches its local extreme. It is possible to display only “minimum peaks” or only the “maximum peaks” or both. Various description options are available.

<table>
<thead>
<tr>
<th>Extreme</th>
<th>None</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>No values are displayed.</td>
</tr>
<tr>
<td></td>
<td>Local minimum and maximum</td>
</tr>
<tr>
<td></td>
<td>Both minimum and maximum are displayed.</td>
</tr>
<tr>
<td></td>
<td>Local minimum</td>
</tr>
<tr>
<td></td>
<td>Only minimum is displayed.</td>
</tr>
<tr>
<td></td>
<td>Local maximum</td>
</tr>
<tr>
<td></td>
<td>Only maximum is displayed.</td>
</tr>
</tbody>
</table>

| Style                    | Transparent description       |
Description colour
Selects the colour of the printed description.

Drawing setup dialogue for the trajectories drawing style
The dialogue is divided into three parts: Display, Minimum and maximum settings, Local extremes. The items in individual parts may differ depending on the style selected in the Display part.

The meaning of individual parameters is analogous to the meaning for the standard drawing style (isolines) – see above.

More information about some settings can be found in chapter Isolines setup.

Palette values for isobands/isolines
The numerical values used in isoline/isoband palette are normally calculated automatically by the program and respect the range of the evaluated quantity (i.e. the minimal and maximal value) and the adjusted number of isolines/isobands.

Under certain circumstances, it may be more convenient to adjust the palette values manually. For example, when designing the reinforcement in concrete 1D members it may be useful to adjust the values so that they correspond to the area of let’s say 1, 2, 3, … n bars of a certain diameter.

Types of palette values

<table>
<thead>
<tr>
<th>Automatic palette values–normal</th>
<th>The palette values are calculated automatically and used as is, i.e. the decimal digits are used in the length specified in Setup &gt; Units.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>E.g.:</td>
</tr>
<tr>
<td></td>
<td>28.47229688  [Red]</td>
</tr>
<tr>
<td></td>
<td>21.43614005  [Orange]</td>
</tr>
<tr>
<td></td>
<td>14.39999222  [Yellow]</td>
</tr>
<tr>
<td></td>
<td>7.363850391  [Green]</td>
</tr>
<tr>
<td></td>
<td>0.3277015625 [Blue]</td>
</tr>
<tr>
<td></td>
<td>-6.700447266 [Light Blue]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Automatic palette values–rounded</th>
<th>The palette values are calculated automatically and rounded, so that the user can better “read” the results.</th>
</tr>
</thead>
</table>

- 121 -
User-adjustable palette values

By default, the palette is identical to Automatic palette values—rounded option. But, the user can edit the values and adjust such values that best meet his/her needs. Moreover, the palette becomes "frozen" and is used for every result quantity.

**The procedure to adjust the type of palette values**

1. Open the required service (e.g. Results, Concrete, etc.).
2. Select the required function for evaluation of results (e.g. Internal forces in Results).
3. In the Property window, click three-dot button [...] to open the Drawing setup dialogue.
4. In the combo box in the left-hand side part of the dialogue select the required palette values type.
5. If necessary, use button [Advanced settings] to edit the values or to make other adjustments.
6. Confirm with [OK].

**Important Note**: The option with user adjustable palette values requires that maximum and minimum value of the corresponding quantity is know. These two values become known only after the quantity has been displayed on the screen at least once. Therefore, until you display the result diagrams on the screen using action button [Refresh], it is not possible to select the type of palette values.

**Saving the palette for later use**

The palette with User-adjustable palette values can be saved and later read into another project or service or function.

**The procedure to save the palette**

1. Open the required service (e.g. Results, Concrete, etc.).
2. Select the required function for evaluation of results (e.g. Internal forces in Results).
3. In the Property window, click three-dot button [...] to open the Drawing setup dialogue.
4. In the combo box in the left-hand side part of the dialogue select User-adjustable palette values.
5. The [Save palette …] and [Load palette …] buttons appear in the dialogue.
6. If necessary, use button [Advanced settings] to edit the values or to make other adjustments.
7. Use button [Save palette …] to save the adjusted palette for later re-use.
8. Close the dialogue.
Averaging strips

Averaging strips

This functionality provides for automatic averaging of peak results around defined points or along defined line strips on slabs. The users can define several styles how to calculate the averaged values. The averaging can be applied to internal forces on slabs and to required reinforcement areas used in the design of reinforcement in concrete slabs.

The averaging strips are defined as what is termed additional data. This fact together with some other characteristics of the averaging strips leads to the following rules concerning the manipulation with the already defined strips:

- No geometrical manipulation is supported (i.e. the averaging strip cannot be copied, moved, etc.) The only exception is the direct editing of the coordinated of the definition points in the Property Window.
- The averaging strip can be normally deleted.
- The removal or editing of the defined averaging strip DOES NOT influences the results.
- If the slab that contains the averaging strip is moved, copied, etc. the averaging strip "goes with" its master slab.
- The averaging strips react to the activity of the slabs. It means that only averaging strips that are defined on active slabs are visible.
- Check of data verifies the position of the strips and all invalid strips (e.g. located out of the master slab) are deleted.

Averaging strips versus finite element mesh

The averaging algorithm uses only the FE nodes that are located inside the averaging strip. This may cause certain inaccuracies especially in the combination with larger finite elements. Therefore, it is recommended to define internal edges along the averaging strips. This ensures that finite element nodes are generated along the edge of the averaging strip, which may significantly improve the accuracy.

The recommended procedure is thus:

1. Define the model of the structure.
2. Perform the calculation.
3. Review the results.
4. Define averaging strips.
5. Review the averaged results.
6. Decide on the final location and number of averaging strips.
7. Define internal edges along the averaging strips.
8. Repeat the calculation to obtain the improved results.

"Density" of averaging strips

The averaging strips can be defined almost arbitrarily. For the purpose of this paragraph we will distinguish two situations. Averaging strips defined with a gap between individual strips and averaging strips defined one next to another (e.g. strip above support and strip in the "middle" of the span defined without any gap in between).

The possible effect of these configurations can be best explained in the following pictures.
Separate strips (i.e. gap between strips)  
If the averaging strips are defined as separate, the algorithm can meet the condition that the distribution of the quantity should as much constant across the span as possible. In other words, the quantity is constant (more or less) across the whole width of the strip. The vertical white line indicates the strip.

Adjacent strips (i.e. no gap between strips)  
On the other hand, if the averaging strips are defined closely one next to another, there is no space between them for the algorithm to handle the change of the magnitude of the given quantity, as the magnitude cannot change in step, it must be gradual. Thus one of the strips must be affected by the change in the magnitude. This is shown in the figure below where the value of the result quantity varies along the width of the strip.

In averaging strip only basic values (mx, my, mxy, vx, vy, nx, ny, nxy) are always averaged, principal and other values (e.g. required areas of reinforcement in concrete checks) are always calculated using these averaged basic values. Each value has some direction (e.g. mx act in the direction of x axis) and it is averaged only in the perpendicular direction, i.e. mx is averaged only in y direction.

Averaging strip, which is rotated towards LCS of the slab, is useful e.g. in case of slant line support. In the first step the basic values are transformed to rotated coordinate system of averaging strip ("Angle" property), in this system they are averaged and then they are transformed back to LCS of the slab.
Results on slabs

Averaging of mxy and vxy is slightly different. Averaged value is calculated in standard way but the signs are preserved, i.e. on part where the value has been positive there is average value with positive sign and vice versa. In addition, averaging of mxy and nxy is done in both directions (longitudinal and perpendicular), so if the averaging strip is also set to average in both directions ("Direction" property) the final value is constant across the whole strip (with respect to original signs).

Defining a new averaging strip

Procedure to define a new averaging strip

1. Define the project and perform the calculation. The completion of the calculation is necessary to get access to service Results that contains the function for the definition of the averaging strips. See also note below.

2. Open service Results.

3. Open branch 2D members.

4. Start (double-click) function Averagingstrip.

5. Fill in the parameters in the input dialogue – see below.

6. Input the strip in the graphical window.

7. End the function.

Note: Alternatively, the same function can be accessed from service Concrete. The procedure described above is useful when you want to review averaged internal forces. The alternative is suitable for the design of required reinforcement areas with the averaging taken into account. This function is accessible even prior to the completion of calculation. On the other hand, it is available only to users who purchase the module for the design of concrete structures and on condition that the material concrete has been defined in the project.

Averaging strip parameters

<table>
<thead>
<tr>
<th>Name</th>
<th>Specifies the name of the strip.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Strip</td>
<td>The averaging strip is defined by a line with a specified width.</td>
</tr>
<tr>
<td>Type</td>
<td></td>
</tr>
<tr>
<td>Point</td>
<td>The averaging strip is defined by a point, width, length, and angle (that specifies the direction of the strip).</td>
</tr>
<tr>
<td>Width</td>
<td>Defines the width of the averaging strip.</td>
</tr>
<tr>
<td>(only if Type = Point)</td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>Defines the length of the averaging strip.</td>
</tr>
<tr>
<td>(only if Type = Point)</td>
<td></td>
</tr>
<tr>
<td>Angle</td>
<td>Defines the direction of the averaging strip.</td>
</tr>
<tr>
<td>Direction</td>
<td>Specifies the direction in which the averaging is to be calculated.</td>
</tr>
</tbody>
</table>
Longitudinal

The averaging is done along the defined strip. We can imagine that the strip represents a 1D member and we want the program to smooth the distribution of the result along that 1D member.

Perpendicular

The averaging is performed in the direction that is perpendicular to the length of the strip. This option is for special purposes only.

Both

The averaging is made in both directions. Again, this option is for special purposes only, e.g. heads of columns.

None

No averaging is made. This option may be useful if one (or several) defined averaging strip(s) should be temporarily ignored while other strips are still required to be used.

Practical demonstration

The following example demonstrates the meaning and effect of parameter Direction.

Let us have a simple plate supported by nine columns placed symmetrically in both directions and review moment \( mx \) calculated without averaging strips and with different variants of the strips.

First, let us define horizontal averaging strips placed just over the supports (the support means the head of the column).
Second, let us define horizontal averaging strips placed both above the supports and between them.

Third, let us define vertical averaging strips just over the supports.

Fourth, let us define vertical averaging strips placed both above the supports and between them.

Let us subject the plate to the self-weight and to a uniformly distributed load placed over the whole plate. The result diagram for $m_x$ (horizontal direction) without any averaging looks like this.
Now, let us adjust longitudinal direction for the averaging. It means that the results will be averaged along the length of the strip and will be more or less constant across the strip.

The result for the horizontal averaging strips defined only above the supports will be like this.

The result for the horizontal averaging strips defined above both the supports and between them will be like this.

The following picture shows that vertically oriented strips has almost no impact on the distribution of moment $m$. 

- 128 -
Now, let us try to change the direction of averaging to Perpendicular. The following picture represents the results for the horizontal averaging strips defined above both the supports and between them. You can see almost no difference in comparison with the unaveraged results.

On the other hand, the vertical averaging strips placed over the supports give the following result for moment $mx$.

The vertical averaging strips placed both over the supports and between them give the following result.
The averaging affects also the results drawn in the section (i.e. not using the isolines / isobands). Let us define a section in the middle of the plate parallel to the x-axis. Let us adjust the longitudinal direction for the averaging strips and look at the results for the horizontal averaging strips placed above the supports.

The first picture shows the result without averaging.

The second picture the result with the averaging.
Editing the existing averaging strip

Procedure to edit the properties of the defined averaging strip

1. Select the required averaging strip.
2. The properties of the strip are displayed in the Property Window.
3. Change the required parameter (Direction, node coordinate, etc.)
4. The change is automatically immediately accepted.
5. Clear the selection.

Tip: Open service Result. Open branch 2D members. Select function Averaging strip (just select, do not double-click). In bottom right corner of the screen, on the status bar, click the "filter field" and select Filter for tree. With these settings, the only entity the cursor can select is just the averaging strip. This may simplify the process of selection.

Deleting the averaging strip

Procedure to delete the defined averaging strip

1. Select the required averaging strip.
2. The properties of the strip are displayed in the Property Window.
3. Press key [Del].
4. The strip is deleted.

Tip: Open service Result. Open branch 2D members. Select function Averagingstrip (just select, do not double-click). In bottom right corner of the screen, on the status bar, click the "filter field" and select Filter for tree. With these settings, the only entity the cursor can select is just the averaging strip. This may simplify the process of selection.

Displaying the averaged results

In order to activate the averaging algorithm, the user must select the appropriate option in service Results.

Procedure to activate the averaging of internal forces

1. Open service Results.
2. Select function 2D members > Member 2D – Internal forces.
3. In the property Window adjust the required parameters for the display of the results.
4. Select option Averaging of peak (without this option being selected, the results are NOT averaged even when averaging strips have been defined).
5. Refresh the screen.
**Procedure to activate the averaging of required reinforcement areas**

1. Open service Concrete.
2. Depending on the needs, select function 2D member > Member design – Design – ULS or 2D member > Member design – Design – crack width.
3. In the property Window adjust the required parameters for the display of the results.
4. Select option Averaging of peak (without this option being selected, the results are NOT averaged even when averaging strips have been defined).
5. Refresh the screen.

**View parameters related to averaging strips**

A special view parameter can be found in the View parameters setting dialogue.

**Procedure to display / hide the defined averaging strips**

1. Open View parameters settings dialogue.
2. Select Tab Structure.
3. Look into group Averaging strips.
4. Tick option Display to see the averaging strips (default) or clear the option to hide them.
5. Confirm with [OK].

**Averaging strips example**

To illustrate the algorithm of averaging strips, a manual verification is performed.

A square slab is inputted with dimensions 2m x 2m.

The mesh size is set to 0,5m.

An averaging strip was inputted in Y-direction with averaging set to ‘Perpendicular’
Results on slabs

Result of mx in nodes not averaged:

Result of mx in nodes not averaged with averaging strip activated:
Manual verification:

Step 1: Create sections perpendicular to the inputted averaging direction.

In this example, the averaging was set to ‘perpendicular’ => create sections in longitudinal direction.

section A is inputted just outside the strip from (0,499;0) to (0,499;2)

section B is inputted just inside the strip from (0,501;0) to (0,501;2)

section C is inputted in the middle of the slab from (1;0) to (1;2)

Result of mx in nodes not averaged:
Step 2: Look at the average result in each section by setting the course to 'uniform':

- Section A: 0.08
- Section B: -0.03
- Section C: -0.16

These are the results which are shown when activating averaging strips!

Step 3: Result of mx in nodes not averaged with averaging strip activated:
Refreshing the results

Principle

An ideal state would be if everything could be fully automatic and made immediately without any delay. This is even more true with reference to software and its interaction with the user.

Unfortunately, this is only an ideal state that can hardly be achieved in practice. What’s more, sometimes the immediate response to any user’s action can be even undesirable, especially when a set of successive steps is necessary to complete a particular action.

SCIA Engineer therefore presents a well-thought-out compromise solution in service Results and also in the Document.

The implemented solution consists of two separate steps:

1. The user can freely select "WHAT" should be displayed and also adjust "HOW" it should be displayed.
2. The user then gives the command "refresh (or display or regenerate) everything NOW".

Refresh of results

In order to refresh (regenerate the display of) results on the screen a special Action button is located in the Property Window of SCIA Engineer user interface. This button is called Refresh.

The procedure for refresh of results on the screen

1. Open service Results.
2. Select required function (e.g. Internal forces on beam, deformation of nodes, etc.)
3. In the Property Window, select required load case or combination, quantity and adjust other parameters defining the display style.
4. When finished with the settings, press Action button [Refresh].
Chapter 12

5. The screen is regenerated.
6. Evaluate the displayed diagrams.
7. When finished, select another result quantity and/or change the display settings and press Action button [Refresh] again.
8. The screen is regenerated once more in order to reflect the latest adjustment.
9. Repeat steps 7 to 8 as many times as required.

**Note:** Whenever a change made in the Property Window requires a subsequent refresh of the screen, item Refresh in Action buttons is highlighted in red.

---

**Example for refresh of results**

The following example demonstrates the use of [Refresh] button from service Results.

A model must be created and calculated. A simple frame will be used in this example.

The frame is subject to:

<table>
<thead>
<tr>
<th>self weight</th>
</tr>
</thead>
<tbody>
<tr>
<td>a vertical force located in the middle of the span of the beam</td>
</tr>
</tbody>
</table>
Refreshing the results

When service Results is opened and function Internal forces on beam is selected, no result diagrams appear on the screen.

In the Property window, make required adjustments, e.g. set Type of loads to Load cases, and under Load cases select LC1 (i.e. the self weight). Press button [Refresh] in the Action buttons.

Once button [Refresh] is pressed, the diagram is displayed (this time, bending moment diagram for self weight load).
In order to see the diagram for another load case, make the required setting in the Property Window.

And press button [Refresh] again.

The diagram is regenerated.

And the same may be repeated once more for the last load case.
Button [Refresh] must be pressed again.

The selected result is drawn on the screen.

The same may procedure may be now be repeated for any other result quantity, load case, load case combination, or for any other display-style related adjustments.
Selected sections for result diagrams

1D members

Diagrams of result quantities are normally displayed in sections whose density is defined in the Solver setup dialogue. If the need arises, the density may be reduced significantly by means of "user-defined" sections. The user may simply define a very limited (or excessive, if s/he likes) set of specific points on the structure (called sections) and the calculated results will be shown in these particular points only.

A section on a 1D member has the following parameters.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Identifies the section.</td>
</tr>
<tr>
<td>Position x</td>
<td>Defines the position of the section on the 1D member.</td>
</tr>
</tbody>
</table>
| Coordination definition | Specifies the coordinate system used for the definition. Either absolute or relative coordinates may be used. The relative coordinate must lie within interval <0; 1>. The absolute coordinate must lie within interval <0; the length of a particular 1D member>.
| Origin | Defines whether the position is measured from the beginning or end of the 1D member. |
| Repeat | Specifies the number of sections defined at the same time.        |
| Delta x | If Repeat is greater than 1 (one), this value defines the distance between individual sections on the 1D member. |

Slabs

Similarly to 1D members, it is possible to define a specific section or section across the slab where the results should be displayed.

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Identifies the section.</td>
</tr>
<tr>
<td>Draw</td>
<td>Defines the plane in which the section is drawn.</td>
</tr>
<tr>
<td></td>
<td>Upright to element = the plane of the result-diagram is perpendicular to the plane of the slab</td>
</tr>
<tr>
<td></td>
<td>Element plane = the result-diagram is drawn in the plane of the slab</td>
</tr>
<tr>
<td></td>
<td>X direction = the result-diagram is drawn in the direction of the global X-axis</td>
</tr>
<tr>
<td></td>
<td>Y direction = the result-diagram is drawn in the direction of the global Y-axis</td>
</tr>
<tr>
<td></td>
<td>Z direction = the result-diagram is drawn in the direction of the global Z-axis</td>
</tr>
<tr>
<td>Direction of cut</td>
<td>The section plane is defined by a line (input graphically in the graphical window) and by this &quot;in-plane&quot; vector.</td>
</tr>
<tr>
<td></td>
<td>See the example below.</td>
</tr>
</tbody>
</table>

Let us assume a single slab as in the picture.
There are two sections defined. The lines defining the sections are not situated in the plane of the slab but are 1 metre below the slab.

To indicate the vertical direction and help you to understand the picture, four vertical columns supporting the slab are defined. The columns intersect the lines defining the sections: (i) the line defining the left-hand section intersects the two columns on the left and (ii) the line defining the right-hand section intersects the two columns on the right.

The vector defining the left-hand section is set to: \(1/0/1\).

The vector defining the right-hand section is set to: \(0/0/1\).

It is clearly seen that while the diagram for the right-hand section is drawn directly above the section defining line, the left-hand section is moved to the right in the direction of the X axis. In fact, the diagram is displayed along a line that forms the intersection of the slab and the plane coming out from the section-defining line inclined by 45 degrees from the vertical.

**Defining a new section for display of results**

*Procedure to define a new section on a 1D member*

1. Open service Structure.
2. Start function Section on beam.
3. Adjust the section parameters.
4. Confirm with [OK].
5. Select the 1D members where the section / sections should be defined.
6. Close the function.
7. Close the service.
Procedure to define a new section on a slab

1. Open service Results.
2. Start function 2D Members > Section on 2D member.
3. Adjust the section parameters.
4. Confirm with [OK].
5. Define the section by its starting and end point.
6. Close the function.
7. Close the service.

Displaying the results in selected sections

1D member

By default any result diagram is displayed in all calculated sections. If required, it is however possible to limit the diagram to a limited set of explicitly defined points – user defined sections.

Whenever a function displaying some result quantity is started, the parameters controlling the behaviour of this function are displayed in the Property window. One of the parameters is called Section. The meaning and consequences of this parameter will be demonstrated on a simple continuous beam.

Let’s assume a simple three-span continuous beam subject to uniformly distributed load. The beam is defined as a set of three beams attached to each other.

Further, let’s define one section in the middle of each span.
Finally, let’s display the diagram of calculated bending moment $M_y$. Using the default setting (parameters Section set to Ends), the diagram may look like:

Now, let’s change the setting of parameter Section to Ends. The calculated values of bending moment will be drawn in end points of each defined beam.

Now, let’s change the setting of parameter Section to Input+Ends. The calculated values of bending moment will be drawn in end points of each defined beam and in three defined sections.
Chapter 13

Now, let's change the setting of parameter Section to Input. The calculated values of bending moment will be drawn only in the defined sections.

Slab

By default any result diagram is displayed by means of isolines / isobands. If required, it is however possible to limit the display to a diagram along a defined section – user defined sections.

Whenever a function displaying some result quantity on slabs is started, the parameters controlling the behaviour of this function are displayed in the Property window. One of the parameters is called Drawing. The meaning and consequences of this parameter will be demonstrated on a simple slab.

Let's define a rectangular slab subject to any load.
Further, let's define a section cutting the section e.g. in the middle.

Let's calculate the slab and display e.g. internal forces.
Let's try all the options for the Drawing parameter.

| Standard | The results are shown using isolines/isobands. A legend is displayed in the top right corner of the graphical window. |
The results are drawn along the defined section across the slab. Function Setup > Scales can be used to control the size of the diagram.

The resultant along the section is calculated. Again, function Setup > Scales can be used to control the size of the diagram.
Options Standard and Section are combined together.

This option works for principal quantities only. The direction (trajectory) of the quantity is shown. A legend is displayed in the top right corner of the graphical window.
Type of diagram in the section

According to the needs of a particular calculation, SCIA Engineer allows you to select the most appropriate type of representation of the result in a section across the slab.

To understand more the individual options, let us input two identical slabs subjected to the identical load. Further, let us define a section across each of the slabs. The first slab has one section defined across the whole width. In the second slab, let us divide the section into eight intervals to have finer results - see the image below.

Precise
The precise distribution of the displayed result quantity is draw along the section.

In our example of two identical slabs, diagrams on both slabs look identical as well.
Uniform
The average value of the result is displayed.
This option may be useful to see the effect of the structure and loads to the particular section.

The example of two identical slabs produces the following results. The "precise" area must be equal to the area of the "uniform" rectangular diagram.
Trapezoid
The distribution of the quantity along the section is approximated by a trapezoid.
This option may be useful if you model your structure in parts and use the reactions of upper parts as load for lower parts. It may be practical to idealise the effect of the upper part by this trapezoidal distribution.

The example of two identical slabs produces results where the force resultant and moment resultant of the trapezoidal diagram are equal to the resultants determined from the “precise” diagram.
Procedure to select the type of diagram in the section across a slab

1. Have the project calculated.
2. Open service Results.
3. Call a function that displays the results in slabs in sections, i.e.:
   1. 2D members > Deformation of nodes,
   2. 2D members > Member 2D – Internal forces,
   3. 2D members > Member 2D – Stresses.
4. Note that there are two items named Drawing in the property window – on condition that the first Drawing is set to Section (otherwise there is just one Drawing item in the property window).
5. Set the first Drawing to Section.
6. Set the second Drawing to the required type of diagram (Precise, Uniform, Trapezoid).
7. Select the quantity to be displayed.
8. If required, adjust other display parameters.
9. Refresh the screen.

Displaying the resultant in the section across a slab

When displaying the results in sections across slabs, you may select between two options. Either the distribution over the section is displayed or the resultant for that section is calculated and shown.

Procedure to display the resultant in the section across a slab

1. Have the project calculated.
2. Open service Results.
3. Call a function that displays the results in slabs in sections, i.e.:
   1. 2D members > Deformation of nodes,
   2. 2D members > Member 2D – Internal forces,
   3. 2D members > Member 2D – Stresses.
4. Set the Drawing to Resultant.
5. Select the quantity to be displayed.
6. If required, adjust other display parameters.
7. Refresh the screen.
Table results

Table results - introduction

The Table results is a dock-able interactive window with a simple grid for presenting results in numerical form. It provides a preview of basic results and checks (Concrete, Steel, Aluminium etc.). It is created similarly to the Table input, thus all basic tools works in the same way.

Standard result:

<table>
<thead>
<tr>
<th>Support</th>
<th>Case</th>
<th>Rx [kN]</th>
<th>Rz [kN]</th>
<th>My [kN.m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Sle1/S1</td>
<td>-21.13</td>
<td>207.82</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>Sle1/S1</td>
<td>41.29</td>
<td>339.68</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>Sle2/S2</td>
<td>12.80</td>
<td>177.85</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>Sle2/S2</td>
<td>2.29</td>
<td>287.06</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>Sle2/S2</td>
<td>2.71</td>
<td>66.13</td>
<td>0.00</td>
</tr>
<tr>
<td>6</td>
<td>Sn1/N5</td>
<td>-0.58</td>
<td>147.82</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Check (Concrete service):

<table>
<thead>
<tr>
<th>Member</th>
<th>d x [m]</th>
<th>Case</th>
<th>Type of reinforcement</th>
<th>Design ULS</th>
<th>Design calculation</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>0.000</td>
<td>ENV2</td>
<td>Required</td>
<td>NOT OK</td>
<td></td>
</tr>
<tr>
<td>B2</td>
<td>0.000</td>
<td>ENV2</td>
<td>Required</td>
<td>NOT OK</td>
<td></td>
</tr>
<tr>
<td>B3</td>
<td>0.000</td>
<td>ENV2</td>
<td>Required</td>
<td>NOT OK</td>
<td></td>
</tr>
<tr>
<td>B4</td>
<td>0.000</td>
<td>ENV2</td>
<td>Required</td>
<td>NOT OK</td>
<td></td>
</tr>
<tr>
<td>B5</td>
<td>0.000</td>
<td>ENV2</td>
<td>Required</td>
<td>OK</td>
<td></td>
</tr>
</tbody>
</table>

Purpose of Table results is similar to the preview window, but it provides some benefits making the work with numerical results more intuitive and interactive:

- Filter the content by some value
- Sorting of content according to the selected column
- Copy the content to the clipboard
- Simple moving and hiding of columns
- Keeping more sets of results on several tabs always at your fingertips
- Selection of members
Chapter 14

- Highlight of relevant section in model
- Refresh of outdated results (marked by validity status)
- Detailed output for SCIA Design Formsopen-checks

The table can be filled by results when the calculation is finished. The columns order and its content is the same as it is in the Engineering Report.

Table Results can handle results based on both the TLX and IP files.

How to load results into the Table results

The common use case:

1. The model is calculated and the results service is available in the main tree.
2. User displays the empty Table results.
3. User selects the result type in the "Results" service or the check type in a material service (Concrete, Steel, ...).
4. User adapts properties of the result in the property window - Load case, selection etc..
5. User loads results using one of following methods:
   - User clicks the "Refresh" action button in the "Properties" window to display results and than clicks the "Get results" on the next page button on the toolbar.
   - User clicks the "Refresh" action button in the "Properties" window while the "Get results on Refresh" on the next page button is switched on.
   - User clicks the "Table results" action button in the "Properties".
6. The grid is filled by values.

See the step-by-step tutorial here.

Reset the Table results settings to the default
Settings used in the Table results is saved to the special XML file. Use the Reset GUI option to get the default settings.

Go to Setup / Options / Environment / Reset GUI. The restart of the application is required. See the "Environment settings" chapter.

The XML file "TableResultSettings.xml" containing the setting is located in the "User setting files" directory, see the "Directories settings" chapter.
Table results - toolbar description

Toolbar can be found in the upper part of Table results frame. Toolbar contains many buttons which are necessary for effective work with Table results.

Get results

This button is used for loading the numerical results into Table results. Results are displayed according to properties from the property window. Results are stored into the newly created tab (see related chapter). See the tutorial related to the proper use of this button.

Get results on Refresh

If this button is switched on the new tab with results is automatically created every-time the "Refresh" action button is pressed, but only if the Table results is visible.

Regenerate current tab

This button launch the refresh of results contained in the selected tab. When done, the "Validity status" on page 175 of particular tab is updated.

Delete all tabs

This button deletes all tabs containing results which can be found in the lower-most part of the Table results - see related chapter. This command can also be called from Cleaner.

Column selector

This button opens the Column selector. Using this feature the user can select which columns are visible in table and which are hidden.

See the "Column selector" on page 165 chapter for more details.
Filtering row

This button displays the Filtering row which is used for live filtering of the content of table. See the "Filtering row" on page 161 chapter for more details.

Display detailed check of row in preview

Using this button the user can send the detailed check of row selected in the table into the preview window. Combo-box offers several layouts for detailed output. See the "Display detailed check of row in preview" on page 171 chapter for more details.

Send table to the Engineering Report

This button allows user to send the current table to Engineering Report using standard "Table to Engineering Report" dialogue.
Please note that the table is not send to Engineering Report in the 1:1 way. The sorting, filtering of the table, rows layout, etc. is not respected.

Result properties info

Internal forces on member, Linear calculation, Extreme: No, System: Principal, Selection: All, Combinations: CO1

This text row lists some basic properties of result/check loaded into Table results.

Table results - table

The functionalities of table in the Table results are similar to Table input. If you are familiar with the Table input functionality it will be easy to use the Table results as well.

Table contains numerical results and consists of rows and columns. Columns represents the type of results and the row represents the geometrical position of result on the structure and combination key.

Sorting

Clicking on the column's header user can define the ascending/descending sorting of table. The small arrow in the header specifies whether the table is sorted ascending or descending.

1. The first click sorts the column from the top to the bottom.

2. The second click sorts the column from the bottom to the top.

3. The third click clears the sorting and default order of rows is displayed.
Filtering row allows user to filter the content of table according to specified criterion in particular columns.

Filtering row can be activated using appropriate button "Filtering row" on page 159.

Value in the grid can be easily inserted into Filtering row using the context menu of selected cell.

Basic mathematical operators "<" and ">" are also accepted.

There are 3 modes of filtering:

- **simple comparison**
  Table displays only rows which have in the selected column value which contain value entered in Filtering row. See following example:
• "<" and ">" for numerical values
  Table displays only rows which have in the selected column value 'bigger than...' or 'lower than...' specified value. See following example:

  ![Example Table 1](image)

• "<" and ">" for text values
  Table displays only rows which have in the selected column value 'bigger than...' or 'lower than...' specified string value. The advanced logic can also handle the naming system often used in civil engineering (e.g. naming of columns - first symbol (letter) according to x line-grid axis, the second symbol (number) according to y line-grid axis). See following examples:

  ![Example Table 2](image)
Copy values into clipboard

User can easily copy the content of selected cell range into clipboard and insert it into spreadsheet afterwards.

1. Select some cell range in the table.
2. Copy values to the clipboard:
   a. using the context menu of selected cell area
   b. using the short-cut CTRL+C.

All cells can be selected by the top left cell (corner empty cell).
Columns manipulation

Columns layout

**TLX type of results**

The default order of the result is taken from the predefined files which are used in the Engineering Report (TLX files).

When the order or the width of columns is changed, the file with a new layout is saved. The last setting is used when the same result type is displayed again. The file is stored in the User folder as TableResultSettings.xml - see the "Directories settings" chapter.

If the XML is deleted, the layout is generated again according to the default TLX file.

The Reset GUI option may be used for the loading the default layout to the Table results and Table input. See the "Environment settings" chapter.
**IPfile type of results**

The grid layout is not saved for this type of results. Instead of that there is the automatic width of column prediction that sets the sufficient width for particular columns. User can adjust the widths afterwards but this is not saved and after switching to another tab the changes are lost and than the default automatically predicted widths are restored to calculated value.

**Column selector**

Column selector contains the list of hidden columns.

The default order and visibility of columns is defined in the TLX file that is loaded when results are loaded into Table results. When user make some change in order or visibility, it is saved to a special file and used when this type of result is displayed again later.

The column selector may be displayed:

- using the button on the toolbar (see "Table results - toolbar description" on page 158 chapter)
- using the context menu on the column header (see "Header context menu" on page 173 chapter)

How to add a hidden column to the table:

1. Open column selector
2. Click on the required column and hold the left mouse button
3. Drag and drop it between two column headers
4. The small red arrow is displayed
5. Release the mouse button now

**Hiding of columns**

Column can be hidden by:

- context menu
• moving the column header out of the table header

<table>
<thead>
<tr>
<th>X [inch]</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>0,0</td>
<td>0,0</td>
</tr>
<tr>
<td>75,000</td>
<td>15</td>
</tr>
<tr>
<td>90,000</td>
<td>18</td>
</tr>
<tr>
<td>90,000</td>
<td>0,0</td>
</tr>
</tbody>
</table>

Empty columns are hidden by default.

Order of columns

Order of columns can be easily adjusted using the drag and drop method:

1. Click on the column header to select it.
2. Drag the column header to its new position.

Modify the width of the particular column

The column can be resized:

• manually by moving the column border - cursor is changed to this shape:

 Highlight of extreme values

Values which are extreme for particular magnitude and member are highlighted by bold font:
Selection link

For some types of results the ‘selection link’ between the Table results and model is available. This link works in both directions:

- members selected in the model are selected in Table results as well
- members selected in Table results are selected in the model as well

Selection from Table result is initiated by mouse click on the header of the row (index number) - see attached picture.

Every row in Table results presents results for some geometrical position in the structure for some load case. In general, results for one member (beam, slab, ...) are represented by many rows in Table results. This functionality causes that all rows related to members selected in the model are highlighted by yellow colour in Table results. And vice versa - selection of one row (using click on the header of the row) causes selection of related member in the model which causes the highlight of all rows related to this member in Table results.

User can use also multi-selection using CTRL and SHIFT keys.

All members selected /highlighted on current tab can be unselected using the ESC key.
Tip&Trick: It is very effective to use the “Zoom to selection” button together with selection link in Table results. Imagine that you have a large and complicated project and you’re listing through the table with results and you see some interesting value on member. But you don’t know where exactly this model is on the graphical presentation of the complicated and large model. Just select the row (by clicking on the header of the row) and use “Zoom to selection” (toolbar “Selection of objects”).
3D model position highlight
This functionality shows you the geometrical position that is related to the particular row in Table results.

Basically, every row in Table results contains values that are valid for some geometrical position - for node in case of 0D results, for section in case of 1D results and for mesh-node in case of 2D results. When this functionality is used, the small symbol is placed in the 3D window showing the exact position for which is particular row valid.

You can also place multiple symbols using standard multi-selection ways (CTRL and SHIFT).

You can activate this functionality in several ways:

- clicking on the header of the row
- using the context menu
- double-clicking any cell
- pressing ENTER with any cell selected in the grid

So far, this functionality is supported only for new results:

- 0D / 1D / 2D SCIA Design Forms Open Checks
- Basic results: 3D stress, 3D displacement, etc.
- Concrete
- Composites
Tip: This functionality may be useful together with "Filtering row". You can filter the grid to get only some interesting values (e.g. places where the tension occurs - $N > 0$ or $\sigma > 0$), place the symbol to all filtered rows and this way you can visualize all those interesting places in the model.

Tip: The colour of the symbol is bound to the colour of structural nodes. This colour can be changed in Palette settings (Setup -> Colours/Lines).

Display detailed check of row in preview

For open-checks using Design Forms technology results, it is possible to execute the detailed check for picked row using the double-click on the cell or using the ENTER or using the button "Display detailed check of row in preview" on page 159 on toolbar.
Every row in Table results shows results for some geometrical position (0D nodes, section on 1D members, mesh node on 2D member) and some load case / combikey. Using this functionality user is able to execute detailed check for any member for any load case / combikey. The detailed output is displayed in preview window.

Once the detailed output is displayed in the preview, it can also be sent to Engineering Report using the context menu in the 3D window. But be careful, this functionality is not intended to be used in this way and therefore the reference between the model and the detailed output from Table results is held to the row index number in the parental brief table of related check - see following warning for detailed description.

Although every row in Table results represents some geometrical position in the model, the reference between the detailed output from Table results and the parental brief table is based on the row index number. It means that if you send the Detailed output to Engineering Report, the detailed output is not related to the geometrical position in the model but is related to the index number of row in the parental brief table of considered result / check in the actual model. Basically it means that if you make some change in project that causes reordering of rows in parental brief table (e.g. delete some member, change the mesh, change the number of sections or mesh-nodes or elements, change the load, change boundary conditions, etc.), the existing detailed outputs from Table results will be still linked to the row index number in parental table but this row might not be valid to original geometrical position and combikey any more. After refresh of detailed output from Table result in Engineering Report you might get values for another member, section, node and combikey. Using other words, if you create detailed output from Table results for the 1000th row and send it to Engineering Report, the detailed output shows the same values as the 1000th row in parental brief table. If you make some significant change, recalculate, and refresh the detailed output in Engineering Report, it still shows the same values as the 1000th row of parental brief table, but this row might represent another position in the model and combikey than before.
Table results

Header context menu

The rest of the tools are in the context menu opened by right-click on the column header:

- Column selector - display the list with available columns;
- Add all columns - add all columns available in the Column selector;
- Hide column - hide the particular column from the table and move them to the Column selector;
- Clear sort - sorting of table is cancelled;
- Extend all columns - the width of all columns is increased by 20 %;
- Narrow all columns - the width of all columns is decreased by 20 %;

Table context menu

This context menu is opened by right-click on the cell in the grid. It is dynamic and offers only features that are currently available in the grid:

- Highlight in model - this command places the highlight symbol into the 3D window to show the position related to the particular value in the grid. See the "3D model position highlight" on page 170 for more.
- Select related member - this command selects the member related to the particular value in the grid. See the "Selection link" on page 167 for more.
- Copy - this item copies the values in the selected grid area into clipboard. See the "Copy values into clipboard" on page 163 for more.
- Copy value to filter - this item shows the "Filtering row" on page 161 and fill the selected values in.
- Run detailed check - this command executes the detailed check for selected cell in the grid. See the "Display detailed check of row in preview" on page 171 for more.

Table results - tabs with results

Results in Table results are stored to the tabs. The row with the tabs can be found in the lower part of Table results. Every tab represents one set of values which has been created using some result/check entity with some set of properties. Results can be loaded into the new tab using several ways - see the "How to load results into the Table results" on page 156 for more details.

There are 2 rows with the tabs:

- The lower row contains the primary tabs related to the type of results
- The upper row contains the secondary tabs representing particular tables inside one set of values (stored in primary tab). It is displayed only if primary tab contains several secondary tabs

Tabs

Create

The tab is created automatically during the loading results into the table - see the "How to load results into the Table results" on page 156 for more details.
Close

The tab can be closed using the "X" button in the right part of every tab.

![Tab close example]

All tabs can be closed using the button in the toolbar - see "Delete all tabs" on page 158 chapter.

Move

The order of the tabs within the row can be modified using the "drag and drop". To move the tab, just hold the left mouse-button on it, move it to desired position and release the button.

Features

Validity status

If the model is modified, the numerical results presented in the Table results may not be valid any more. Tabs containing such values are marked by red exclamation mark:

![Validity status example]

This validity status tells the user that results contained in this tab may be wrong and should not be considered. Similar functionality can be found in the Engineering Report - see "Validity status" chapter for more info.
Chapter 14

Tutorial: How to fill the Table results with results

1. Open attached file "Table_results.esa".

2. Add a window Table results to the application if it is not displayed yet.

3. Start the linear calculation.

4. Go to the service Results.

5. Click on the Internal forces on beam and check that the properties are displayed.

6. Select the combination CO1 in properties.

7. Pick one of following ways for loading results:
   - User clicks the "Refresh" action button in the "Properties" window to display results and than clicks the "Get results" on page 158 button on the toolbar.
   - User clicks the "Refresh" action button action button in the "Properties" window while the "Get results on Refresh" on page 158 button is switched on.
   - User clicks the "Table results" action button in the "Properties".
8. Check that grid is filled with results.

<table>
<thead>
<tr>
<th>Member</th>
<th>Case</th>
<th>dx [m]</th>
<th>Case</th>
<th>N [kN]</th>
<th>Vx [kN]</th>
<th>Vz [kN]</th>
<th>Mx [kNm]</th>
<th>My [kNm]</th>
<th>Mz [kNm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>B13</td>
<td>0.000</td>
<td>C01/1</td>
<td>67.74</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
</tr>
<tr>
<td>2</td>
<td>B11</td>
<td>0.000</td>
<td>C01/1</td>
<td>8.79</td>
<td>0.00</td>
<td>-2.82</td>
<td>0.00</td>
<td>19.84</td>
<td>0.00</td>
</tr>
<tr>
<td>3</td>
<td>B1</td>
<td>4.000</td>
<td>C01/1</td>
<td>0.75</td>
<td>0.00</td>
<td>-14.51</td>
<td>0.00</td>
<td>-14.62</td>
<td>0.00</td>
</tr>
<tr>
<td>4</td>
<td>B7</td>
<td>0.000</td>
<td>C01/1</td>
<td>-17.63</td>
<td>0.00</td>
<td>18.00</td>
<td>0.00</td>
<td>-35.15</td>
<td>0.00</td>
</tr>
<tr>
<td>5</td>
<td>B7</td>
<td>4.000</td>
<td>C01/1</td>
<td>-11.75</td>
<td>0.00</td>
<td>12.00</td>
<td>0.00</td>
<td>21.81</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Try to move one column to the new position, to hide the column, to add new column from "Column selector " on page 158 etc.

9. The final project is "Table-results-final.esa".