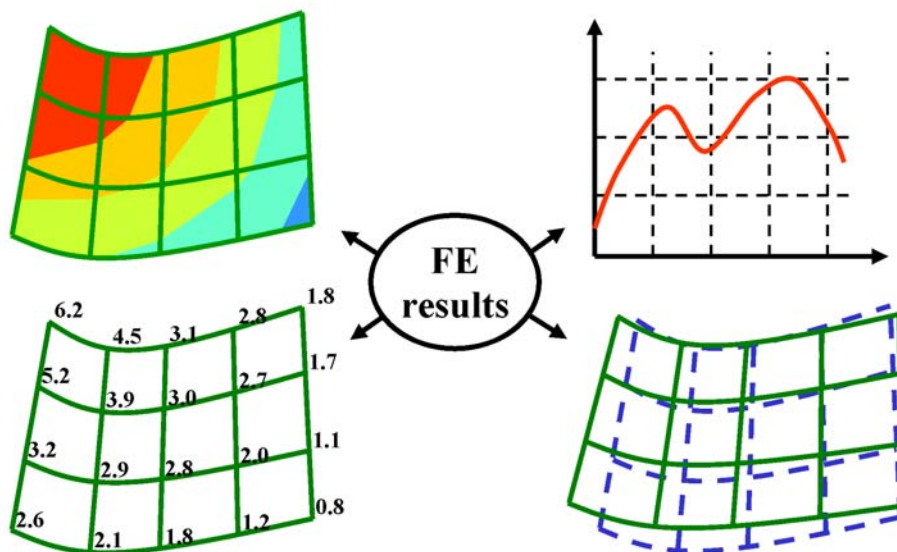


SESAM USER MANUAL

Xtract



Postprocessor for Presentation,
Animation and Reporting of Results

Developed by
ceetron
understanding by visualization

DET NORSKE VERITAS

Sesam User Manual Xtract

Postprocessor for Presentation,
Animation and Reporting of Results

February 1st, 2011

Valid from program version 3.0

Developed by



Marketed by

DET NORSKE VERITAS

DNV Software Report No.: 00-7004 / Revision 12, February 1st, 2011

Copyright © 2011 Det Norske Veritas

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher.

Published by:

Det Norske Veritas
Veritasveien 1
N-1322 Høvik
Norway

Telephone:	+47 67 57 99 00
Facsimile:	+47 67 57 72 72
E-mail, sales:	software.sesam@dnv.com
E-mail, support:	software.support@dnv.com
Website:	www.dnvsoftware.com

If any person suffers loss or damage which is proved to have been caused by any negligent act or omission of Det Norske Veritas, then Det Norske Veritas shall pay compensation to such person for his proved direct loss or damage. However, the compensation shall not exceed an amount equal to ten times the fee charged for the service in question, provided that the maximum compensation shall never exceed USD 2 millions. In this provision "Det Norske Veritas" shall mean the Foundation Det Norske Veritas as well as all its subsidiaries, directors, officers, employees, agents and any other acting on behalf of Det Norske Veritas.

Table of Contents

1	INTRODUCTION	1-1
1.1	Xtract – General Purpose Postprocessor.....	1-1
1.1.1	Structural Analysis Results	1-1
1.1.2	Hydrodynamic Analysis Results	1-1
1.2	Xtract in the Sesam System	1-1
1.3	How to read the Manual.....	1-3
1.4	Status List	1-3
1.5	Program Extensions	1-4
2	FEATURES OF XTRACT.....	2-1
2.1	Models and Results for Presentation (Input to Xtract)	2-2
2.2	Result Positions, Attributes and Components.....	2-4
2.3	Run Numbers, Result Cases and Occurrences.....	2-5
2.4	Combining and Scanning Results	2-5
2.5	Dynamic Frequency Domain Results – Complex Results.....	2-6
2.6	Superelements	2-6
2.7	Sets.....	2-6
2.8	Graphs.....	2-7
2.9	Animation	2-7
2.10	Plotting and Printing (Tabulating)	2-7
2.11	Viewing a 3D Model with Results in Xtract Viewer	2-8
2.12	Viewing Two 3D Models (Global Model plus Sub-Model) in Xtract Viewer	2-8
2.13	Automatic Report Generation – the Xtract Report Builder	2-8
2.14	Logging and Reading Commands.....	2-9
2.15	Transfer of Data through the Sesam Interface File System	2-9
2.16	Transfer of Data to Xtract through a VTF-File.....	2-10

3	USER'S GUIDE TO XTRACT	3-1
3.1	Getting Started	3-1
3.1.1	Starting Xtract and Opening the Results Interface File	3-1
3.1.2	The Graphical User Interface of Xtract	3-3
3.1.3	Basic Display and Model Manipulation	3-3
3.1.4	Viewing Results.....	3-6
3.1.5	View Nodes/DISPLACEMENT	3-6
3.1.6	View Nodes/G-STRESS.....	3-7
3.1.7	View Elements/G-STRESS	3-7
3.1.8	View Elements/P-STRESS.....	3-9
3.1.9	View Elements/G-FORCE	3-11
3.1.10	View Elements/B-STRESS	3-12
3.1.11	View Element average/G-STRESS	3-13
3.1.12	View all Components of an Attribute.....	3-14
3.1.13	View Resultpoints/G-FORCE	3-16
3.1.14	View Nodes/REACTION-FORCE.....	3-18
3.1.15	Creating a Graph.....	3-18
3.1.16	Selecting Result Case	3-23
3.1.17	Combining Result Cases.....	3-23
3.1.18	Scanning Result Cases.....	3-25
3.1.19	Transforming Stresses	3-29
3.1.20	Setting Contours	3-31
3.1.21	Unit Conversion.....	3-32
3.1.22	Sets	3-33
3.1.23	Multiple Viewports.....	3-38
3.1.24	Adding Text and Pointers	3-40
3.1.25	Display Settings, Labelling, Part Attributes and Information	3-40
3.1.26	Making Plots/Hard Copies and Adding Graphics to Reports.....	3-46
3.2	Structural Analysis Results	3-48
3.2.1	Result Positions	3-48
3.2.2	Upper and Lower Surfaces	3-58
3.3	Hierarchical Organization of Results.....	3-58
3.3.1	Run Number	3-58
3.3.2	Result Cases and Occurrences.....	3-59
3.4	Postprocessing Complex (Frequency Domain) Results.....	3-59
3.4.1	Evaluating a Single Complex Resultcase	3-60
3.4.2	Combining a Complex Resultcase with Static Resultcases.....	3-61
3.4.3	Scanning Complex Resultcases	3-64
3.4.4	Scaling a Complex Resultcase.....	3-66
3.4.5	Graphing Complex Resultcases.....	3-67
3.5	View Sections	3-67
3.6	List Model and Results Data.....	3-67
3.7	Animation	3-70

3.7.1	Animating Mode Shapes (and Static Results)	3-72
3.7.2	Animating Frequency Domain Results	3-73
3.7.3	Animating Time Domain Results.....	3-73
3.8	Hydrodynamic Analysis Results from Wadam or Wasim.....	3-74
3.8.1	Hydrodynamic Loads, Motion and Stresses – Wadam/Wasim and Sestra Results (R+G).....	3-74
3.8.2	Hydrodynamic Loads and Motion – Wadam/Wasim Results only (T+L+G).....	3-76
3.8.3	Hydrodynamic Loads – Wadam/Wasim Results only (T+L).....	3-78
3.9	Hydrodynamic Loads and Motion – Wasim Time Domain Results.....	3-80
3.10	Animating Jacket Launching – Installjac Time Domain Results	3-82
3.11	Riser and Mooring Analysis – Riflex Time Domain Results	3-83
3.12	Stochastic Fatigue – Stofat Results.....	3-85
3.13	Exporting Data for Further Processing and Reporting	3-86
3.13.1	Exporting 3D Model with Results to VTFx-File for Viewing in Free Programs.....	3-86
3.13.2	Combined export of Two 3D Models to VTFx-File – Sub-Model plus Global Model ..	3-90
3.13.3	Automatic Report Generation by the Xtract Report Builder.....	3-92
3.14	Notes on Graphing, Combining and Scanning	3-96
3.14.1	Graphing.....	3-96
3.14.2	Combining and Scanning	3-96
3.15	Special Features	3-96
3.15.1	Save Time for large Models by not Reading Beam Sections.....	3-96
3.15.2	Square Root of Sum-of-Squares Combination for Elements/G-STRESS.....	3-97
3.15.3	Ship Overall Vibration Level	3-97
3.16	Working with Line-Mode Commands and Command Input Files	3-99
3.16.1	Reading a Command Input File from inside Xtract	3-100
3.16.2	Notes on Result Case Identification	3-101
4	EXECUTION OF XTRACT.....	4-1
4.1	Program Environment.....	4-1
4.1.1	Files used by Xtract.....	4-1
4.1.2	Command Line Arguments	4-2
4.1.3	Registry Settings.....	4-2
4.2	Program Requirements	4-3
4.2.1	Execution Time	4-3
4.2.2	Storage Space	4-3
4.3	Program Limitations	4-4
5	COMMAND DESCRIPTION	5-1
5.1	File Commands.....	5-5
5.1.1	Open	5-5

5.1.2	Source	5-5
5.1.3	Plot.....	5-6
5.1.4	Plotall.....	5-6
5.1.5	Print	5-7
5.1.6	Exportvtfx.....	5-8
5.1.7	Exportcombined	5-8
5.1.8	Rep.....	5-8
5.1.9	Log.....	5-9
5.1.10	Exit	5-9
5.1.11	Resetsettings	5-9
5.2	Display Commands	5-10
5.2.1	Display.....	5-10
5.2.2	Displaysettings	5-11
5.2.3	Displayinfo	5-12
5.2.4	Label	5-13
5.2.5	Eye.....	5-13
5.3	Model Commands	5-15
5.3.1	Model.....	5-15
5.3.2	Section	5-16
5.4	Set Command.....	5-17
5.4.1	Set.....	5-17
5.5	Result Commands	5-20
5.5.1	Result.....	5-20
5.5.2	Contour	5-22
5.5.3	Vector	5-22
5.5.4	Shell.....	5-23
5.5.5	Evaluation.....	5-23
5.5.6	Combine	5-23
5.5.7	Combine add.....	5-24
5.5.8	Combine all	5-25
5.5.9	Combine sqrt	5-26
5.5.10	Scan	5-26
5.5.11	Scale	5-27
5.5.12	Createuservector.....	5-27
5.5.13	Overallvelocity	5-28
5.6	Graph Commands	5-28
5.6.1	Graph	5-28
5.7	Animation Commands	5-29
5.7.1	Animation	5-29

APPENDIX A TUTORIAL EXAMPLES..... A-1

A 1	Plate Connection with Bracket.....	A-1
A 1.1	GeniE Input	A-1

A.7.2	// GeniE input for creating a model of a plate connection with bracket.....	A-2
A 1.2	Prefem Input	A-5
A 1.3	Patran-Pre Input.....	A-8

APPENDIX B STRESSES AND OTHER RESULTS B-1

B 1	Element Local Coordinate Systems.....	B-1
B 1.1	Membrane and Shell Elements.....	B-1
B 1.2	Solid Elements.....	B-2
B 2	Result Attributes and Components	B-2
B 2.1	Result Attribute: DISPLACEMENT	B-2
B 2.2	Result Attribute: VELOCITY	B-2
B 2.3	Result Attribute: ACCELERATION.....	B-2
B 2.4	Result Attribute: REACTION-FORCE.....	B-3
B 2.5	Result Attribute: G-STRESS	B-3
B 2.6	Result Attribute: P-STRESS	B-3
B 2.7	Result Attribute: PM-STRESS.....	B-4
B 2.8	Result Attribute: D-STRESS.....	B-4
B 2.9	Result Attribute: R-STRESS	B-4
B 2.10	Result Attribute: G-FORCE	B-5
B 2.11	Result Attribute: B-STRESS	B-5
B 3	Derived Stresses for Membrane, Shell and Solid Elements	B-6
B 3.1	Decomposed Stresses	B-6
B 3.2	Stress Resultants.....	B-8
B 3.3	Principal Stresses.....	B-9
B 3.3.1	Principal Stresses for Thin Shell (2D Problem).....	B-9
B 3.3.2	Principal Stresses for Thick Shell.....	B-10
B 3.3.3	Principal Stresses for Solids (3D Problem)	B-10
B 3.4	von Mises Stress.....	B-11
B 3.5	Sequence of Computation of Stresses	B-12
B 4	Beam Stresses Derived from Beam Forces and Moments.....	B-13
B 5	Maximum and Minimum Stress Involving Complex Result Cases.....	B-15
B 5.1	Basic Concepts and Properties of Complex Results	B-15
B 5.1.1	Extracting Physical Response from Complex Results.....	B-15
B 5.1.2	Stress Variation through a Cycle (360 Degrees)	B-16
B 5.2	Manipulations of Complex Results	B-17
B 5.2.1	Presenting a Single Complex Result Case	B-17
B 5.2.2	Scaling of a Complex Result Case.....	B-17
B 5.2.3	Adding Complex Result Cases of Equal Frequencies	B-17
B 5.2.4	Combining Several Static and Several Complex Result Cases	B-18
B 5.2.5	Combining Several Static Result Cases with a Single Complex Result Case	B-18
B 5.2.6	Scanning Complex Result Cases	B-19

1 INTRODUCTION

1.1 Xtract – General Purpose Postprocessor

Xtract is the model and results visualisation program of Sesam. Xtract is owned and developed by Ceetron AS and marketed by DNV Software. It offers general-purpose features for selecting, further processing, displaying, tabulating and animating results from static and dynamic structural analysis as well as results from various types of hydrodynamic analysis. Its intuitive and high-performance 3D graphics enables easy and efficient interactive rotation, zooming and panning of the model while viewing and animating the results.

The graphical user interface operations may optionally be logged as command equivalents. This log may be edited and used as input to a new session for automated extraction and presentation of results.

1.1.1 Structural Analysis Results

Xtract presents results for truss, beam, membrane, plate, shell and solid models in alternative ways: deformed model, contour (iso-) curves, numeric data on model display, X-Y graphs and tabulated data. Based on stresses computed by the analysis program Xtract computes and presents derived stresses: stresses decomposed into membrane and bending parts, principal stresses and von Mises stress. The graphic results presentation may be printed directly or saved to various graphic file formats (Windows bitmap, GIF, etc.). A 3D model with results may even be exported and opened in a free program for interactive interrogation. Tabulated data may be exported to file for importing into spreadsheets for customised results manipulation.

1.1.2 Hydrodynamic Analysis Results

In addition to its general presentation features the animation feature of Xtract is especially useful for presenting results from hydrodynamic analyses. The motion of a vessel in waves may for example be animated with the resulting stresses in the hull.

1.2 Xtract in the Sesam System

Sesam is comprised of preprocessors, environmental analysis programs, structural analysis programs and postprocessors. An overview of Sesam is shown in Figure 1.1. Xtract is found in the postprocessor group.

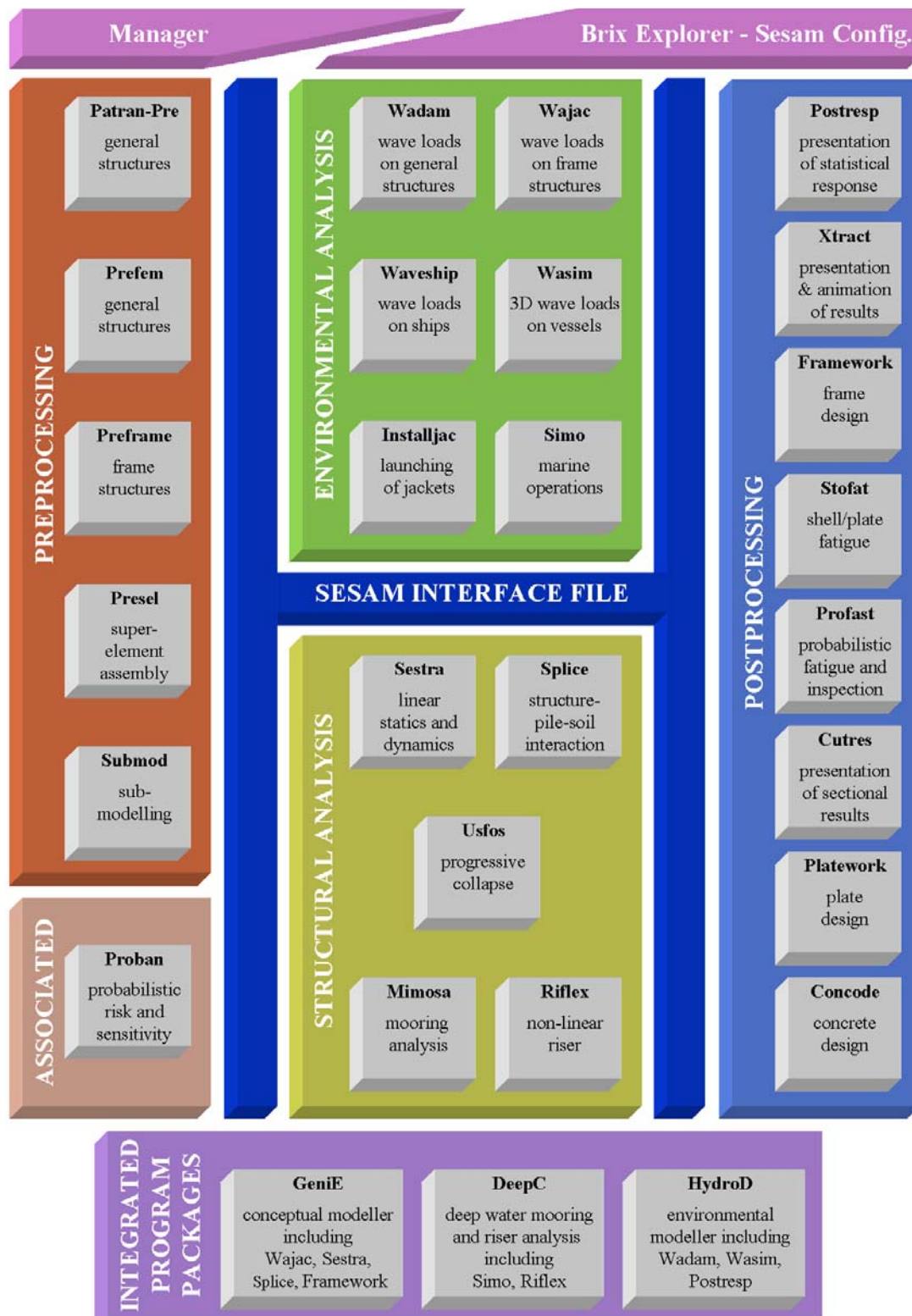


Figure 1.1 Sesam overview

1.3 How to read the Manual

- Read **Chapter 2 FEATURES OF XTRACT** to learn about the postprocessing features of Xtract.
- Read **Section 3.1 Getting Started** to learn the basics of using Xtract. Even though the section mostly refers to postprocessing structural analysis results the basics of other postprocessing tasks are the same.
- Read **Section 3.2 Structural Analysis Results** and **Section 3.3 Hierarchical Organization of Results** to learn about the organisation of structural analysis (FE) results data in Xtract.
- If you want to concentrate on presenting hydrodynamic analysis results in the frequency domain from Wadam or Wasim, possibly in combination with Sestra results, then proceed from Section 3.1 to **Section 3.8 Hydrodynamic Analysis Results from Wadam or Wasim**.
- If you want to concentrate on presenting hydrodynamic analysis results from Wasim then proceed from Section 3.1 to **Section 3.9 Hydrodynamic Loads and Motion – Wasim Time Domain Results**.
- If you want to concentrate on presenting hydrodynamic analysis results from Installjac then proceed from Section 3.1 to **Section 3.10 Animating Jacket Launching – Installjac Time Domain Results**.
- If you want to concentrate on presenting riser and mooring analysis results from Riflex then proceed from Section 3.1 to **Section 3.11 Riser and Mooring Analysis – Riflex Time Domain Results**.
- If you want to concentrate on presenting fatigue analysis results from Stofat then proceed from Section 3.1 to **Section 3.12 Stochastic Fatigue – Stofat Results**.
- If you want to learn about alternative ways of exporting data from Xtract for further processing by additional programs (free software) as well as for results presentation and reporting then read **Section 3.13 Exporting Data for Further Processing and Reporting**.
- Read the remaining sections of **Chapter 3 USER'S GUIDE TO XTRACT** to learn about more advanced ways of using Xtract.
- Read **Chapter 4 EXECUTION OF XTRACT** for information on files associated with Xtract and about alternative ways of running the program.
- Read **Chapter 5 COMMAND DESCRIPTION** for a full description of all line-mode commands (a command reference).
- Refer to **Appendix B STRESSES AND OTHER RESULTS** for background information enabling you to fully interpret all stress results presented. The methods used for extrapolating and interpolating stresses, computing stress derivatives and more are described.

1.4 Status List

There exists for Xtract as for all other Sesam programs a Status List providing additional information. This may be:

- Reasons for update (new version)
- New features

- Errors found and corrected
- Etc.

The most updated version of the Status List is found on our website www.dnvsoftware.com. Click Downloads followed by Sesam Status Lists to log into this service. Contact us for log-in information.

1.5 Program Extensions

A program extension is a feature or a set of features – normally of advanced or specialised nature – that has been screened off for users of the basic version of the program. Access to a program extension is subject to agreement and a proper configuration (password) file.

The basic version of Xtract only includes features for presenting a model (reading file T#.FEM), optionally with hydrodynamic loads computed by Wadam or Wasim (L#.FEM).

There are three extensions in Xtract:

- STRU – Features for postprocessing structural analysis results, required for reading a Results Interface File
- ANIM – Animation, required for accessing the Animation menu and for reading a VTF-file¹
- RPTP – Xtract Report Builder – a separate program associated with Xtract and used for automated report generation

Also see Section 2.1 for an overview of data available for presentation depending on program extension.

1. VTF is together with VTFx a proprietary file format of Ceetron AS.

2 FEATURES OF XTRACT

Xtract presents model and results data of the Sesam Interface Files (see Section 2.15). Furthermore, certain results stored on a VTF-file (see Section 2.16) may be presented. The main features of Xtract include:

- Quick and easy interactive rotation and zooming of model with results, during animation if relevant.
- Model display and manipulation features include:
 - Surface, lines and outline view, with and without finite element mesh
 - Light sources giving shading effects or ambient light
 - Orthographic and perspective views
 - Colouring of elements according to element type, material type and shell/plate thickness
 - Adding model coordinate system, local element axes, labels for node and element numbers
 - Retrieving sets created by the preprocessor and creating new sets
- Deformed and undeformed model
- All basic stress components plus stresses averaged within elements and averaged in nodes
- Derived stresses such as von Mises, decomposed (membrane and bending) and principal stresses
- Annotated contour lines and colour contouring (filling) for displacements and stresses
- Numeric values superimposed on model or only peak (maximum and minimum) values
- Vector presentation of principal stresses and deformation
- Transformation of stresses of shell, plate and membrane elements to a user defined coordinate system
- X-Y graphing of results
- Combinations of result cases and scanning for max/min over result cases
- Complete handling of complex (frequency domain) results including finding phase giving max/min

- Unit conversion of model data and results
- Splitting of the screen into one, two and four viewports and copying display between these
- Colour/grey shaded plots annotated with header, footer and freely positioned text and pointers
- Animation of results from structural dynamic and hydrodynamic analyses
- Export of 3D model with results for interactive results interrogation in the free program Xtract Viewer
- Export of two 3D models, typically a global model and a sub-model, for viewing one superimposed on the other in the free program Xtract Viewer
- Export of 3D model, image, table and other results data to the separate program Xtract Report Builder for automated report generation

2.1 Models and Results for Presentation (Input to Xtract)

In addition to results from a FE structural analysis Xtract may present other data. A complete list of alternative files for processing is provided below. The required program extension is also given (see Section 1.5 on program extensions). Refer to Section 2.15 for more information on Sesam Interface Files.

1 Structural Results Postprocessing – R#.SIN or R#.SIU or R#.SIF (R#.SI*)

- The Results Interface File in alternative formats, # is the (top level) superelement number.
- The file contains a FE model with structural analysis results.
- Program extension required: STRU

This kind of postprocessing is covered in Section 3.1 through Section 3.7.

2 Hydrodynamic & Structural Results Postprocessing – R#.SI* (above alternatives) + Gn.SIF

- The Results Interface File plus the Hydrodynamic Results Interface File, also termed Global Response File.
- The R#.SI* file contains a FE model with results from a structural analysis in frequency domain.
- The Gn.SIF contains the rigid body motion of a floating object, possibly including a sea surface description. n is a positive number, normally 1. The file is created by Wadam or Wasim.
- Program extension required: STRU

This kind of postprocessing is covered in Section 3.8.1.

3 Model & Loads Presentation – T#.FEM + L#.FEM

- The Input Interface File optionally appended by a Loads Interface File (automatically read if present). # may be a 1st level or top level superelement.
- The T#.FEM file contains a FE model (no analysis results).
- The optional L#.FEM contains hydrodynamic loads computed by Wadam or Wasim.
- Program extension required: none

This kind of postprocessing is covered in Section 3.8.3.

4 Hydrodynamic Results Postprocessing – T#.FEM + L#.FEM + Gn.SIF

- The Input Interface File appended by a Loads Interface File (automatically read) plus the Hydrodynamic Results Interface File, also termed Global Response File. # must in this case be a top level superelement.
- The T#.FEM + L#.FEM files contain a FE model and hydrodynamic loads computed by Wadam or Wasim.
- The Gn.SIF contains the rigid body motion of a floating object, possibly including a sea surface description. n is a positive number, normally 1. The file is created by Wadam or Wasim.
- Program extension required: ANIM (to allow animation)

This kind of postprocessing is covered in Section 3.8.2.

5 Animating Wasim Results – file.VTF (+ 2D-series.VTF) (+ T#.FEM)

- A VTF-file produced by Wasim containing models of a vessel and a sea surface plus time domain motion results
- A VTF-file containing 2D series data (component versus time) may optionally be opened
- A T#.FEM file containing a FE model (extra geometry) may optionally be opened
- Program extension required: ANIM

This kind of postprocessing is covered in Section 3.9.

6 Animating Installjac Results – file.VTF (+ 2D-series.VTF)

- A VTF-file produced by Installjac containing models of a jacket, barge and sea surface plus time domain motion results of the launching/floating stability/upending process
- A VTF-file containing 2D series data (component versus time) may optionally be opened
- Program extension required: ANIM

This kind of postprocessing is covered in Section 3.10.

7 Animating Riflex Results – file.VTF (+ 2D-series.VTF) (+ T#.FEM)

- A VTF-file produced by Riflex containing time domain results for conductors, risers and anchor lines
- A VTF-file containing 2D series data (component versus time) may optionally be opened
- A T#.FEM file containing a FE model (extra geometry) may optionally be opened
- Program extension required: ANIM

This kind of postprocessing is covered in Section 3.11.

8 Presenting Stofat Results – file.VTF

- A VTF-file produced by Stofat containing a stiffened plate model with stochastic fatigue analysis results
- Program extension required: ANIM

This kind of postprocessing is covered in Section 3.12.

2.2 Result Positions, Attributes and Components

The structural analysis results stored on the Results Interface File ('results file' or 'R-file' for short) are basically:

- Displacement (and optionally velocity and acceleration) components in the nodes
- Force and moment components for beam elements
- Stress components for membrane, shell and volume elements

Xtract presents these results and data based on these results in up to five *positions*:

- **Nodes** – the nodes of the model
- **Elements** – the nodes of the individual elements
- **Element average** – the midpoint of the elements
- **Resultpoints** – the points in which stresses and forces are found on the results file
- **Surface resultpoints** – available for 6 and 8 node shell elements only, these elements have their result points located inside the elements in the thickness direction, extrapolation to the surfaces is therefore provided

See Section 3.2.1 for more information on the result positions.

The result positions contain one or more of the following result *attributes* (the list is not exhaustive):

- **DISPLACEMENT** – nodal displacements
- **G-STRESS** – general stresses (as found on the results file)
- **P-STRESS** – principal stresses
- **PM-STRESS** – principal membrane stresses
- **D-STRESS** – decomposed stresses for shell elements
- **G-FORCE** – general forces for beam elements
- **REACTION-FORCE** – reaction forces in supported (fixed or prescribed) nodes

Finally, the result attributes include result *components*, for example:

- DISPLACEMENT includes the components **X**, **Y**, **Z**, **RX**, **RY**, **RZ** and **ALL**
- G-STRESS includes the components **SIGXX**, **SIGYY**, **TAUXY**, etc.
- P-STRESS includes the components **P1**, **P2** and **P3**.
- PM-STRESS includes the components **P1** and **P2**.

- D-STRESS includes the components **SIGMX**, **SIGMY**, **TAUMXY**, etc.
- G-FORCE includes the components **NXX**, **NXY**, **NXZ**, **MXX**, etc.
- REACTION-FORCE includes the components **X-FORCE**, **Y-FORCE**, **Z-FORCE**, **RX-MOMENT**, etc.

A particular results value available for presentation is therefore identified by a result position, result attribute and result component ('position/attribute/component' for short). See Appendix B 2 for a complete list of all attributes and components.

2.3 Run Numbers, Result Cases and Occurrences

The results are organised in a hierarchical manner: a run number contains a number of result cases which in turn contain a number of occurrences:

- **Run number** – In most cases there will initially be only one run number (=1). Merging of results files (performed in Prepost) will result in a run number 2. Creating combinations in Prepost may also yield run numbers higher than 1. Combinations, scan result cases and scaled complex result cases created in Xtract will be put in a run 'number' termed User¹.
- **Result case and Occurrence**
 - For a static analysis the result cases correspond to the load cases for which the model was analysed. There will be only one occurrence per result case which means that this bottom level of the results hierarchy may be neglected.
 - For a dynamic analysis the results are grouped as occurrences under result cases. Typically the result cases correspond to wave directions (frequency domain analysis) or time histories (time domain analysis). The occurrences correspond to frequencies (frequency domain) or time steps (time domain).

See Section 3.3 for more information on the organisation of results.

2.4 Combining and Scanning Results

Any number of result cases (but only one complex result case, see Section 2.5) may be included in a combination. This new result case will comprise either only the result attribute selected for the combination or, optionally, all available result positions and attributes. In the former case, additional result positions/attributes may be added to the combination. Read more about this in Section 3.1.17.

Any number of result cases may be scaled and scanned for the (absolute) maximum or minimum of a selected result component. Read more about this in Section 3.1.18.

1. In Xtract's database the run number 9999 is used even though it appears as 'User'. When making combinations in Prepost the run number 9999 should therefore be avoided.

2.5 Dynamic Frequency Domain Results – Complex Results

Xtract offers specialised features for handling results from a dynamic analysis in the frequency domain. The complex results from such an analysis cannot be manipulated and presented in the same way as results from a plain static analysis. When presenting a complex result case it needs to be evaluated for a specific phase of the wave (or other harmonic loading). Furthermore, special considerations must be made when combining and scanning complex result cases. See Appendix B 5 for some basic theory on postprocessing complex results.

Xtract will handle complex results as follows:

- When a result component is presented the complex result case may be evaluated for the maximum (magnitude) of the component as well as for any user chosen phase of the wave.
- A combination may include a single complex result case and any number of static (non-complex) result cases. The complex result case may be evaluated for the (absolute) maximum or minimum of any result component as well as for any user chosen phase of the wave.
- A scanning for the (absolute) maximum or minimum value of a component may be done for any number of complex result cases and any number of static (non-complex) result cases. In this process each complex result case is evaluated. The evaluation may be done for a given phase or for the same criterion as the scanning over the result cases. The latter evaluation is normally the desired one and involves that if you want to scan a number of result cases for the highest von Mises stress then the complex result cases need to be evaluated for the highest von Mises stress also.
- A complex result case may be scaled. I.e. the real and imaginary parts are simply multiplied by a given factor.

Read more about this in Section 3.4.

2.6 Superelements

Xtract handles large superelement hierarchies as well as single superelement models. Any kind of result may be presented in the same way for all superelements at any level. The way in which the model and results are presented may be controlled for the superelements individually (see Section 3.1.25).

A browser provides an overview of the superelement hierarchy making it easy to select and display the desired superelement whether this is a first, top or intermediate level superelement.

2.7 Sets

Sets defined by the preprocessors are stored on the results file and are available in Xtract.

Also, new sets may be defined by various graphic means (left/right/above/below a line, rubberband, polygon) as well as by referring to existing sets, element numbers, material type, element type, beam cross section, shell thickness, coordinate ranges and result values. Read more about sets in Section 3.1.22.

2.8 Graphs

Three types of graphs may be created:

- Nodes – a result component belonging to result position Nodes may be graphed over any number of selected nodes.
- Beams – a result component belonging to result position Elements and attribute G-FORCE, i.e. beam forces and moments, may be graphed over any number of selected beam elements.
- Occurrences – a result component belonging to result position Nodes or Elements may be graphed for any single node over any number of result cases and occurrences. This is mostly relevant for a time domain dynamic analysis for which the time history of for example a displacement component may be graphed.

A graph may include curves for many components by adding new graphs on top of the existing one.

Read about creating graphs in Section 3.1.15.

In addition to the above there is a feature for graphing 2D series results stored on a VTF-file when this is in connection with presenting time domain results found on a VTF-file as detailed in Section 3.9, Section 3.10 and Section 3.11.

2.9 Animation

Animation of results may be performed for:

- Static results – Animation of a static displacement may give an improved understanding of the results.
- Mode shapes – Animation is often the best way of presenting a mode shape (eigenvector).
- Frequency domain results – The harmonic results may be animated through the whole cycle (360°) or parts of it. The displacements may be superimposed by colour contouring of any result component. During the animation the model with results may be rotated and zoomed and manipulated in other ways.
- Time domain results – The time series results may be animated for all or selected time steps of the time history. The displacements may be superimposed by colour contouring of any result component. During the animation the model with results may be rotated and zoomed and manipulated in other ways.

Animating the time domain results stored on a VTF-file by Wasim, Installjac and Riflex falls into this category.

Read more about animations in Section 3.7.

2.10 Plotting and Printing (Tabulating)

The display may either be printed on an on-line printer, saved in a plot file using a selection of graphic formats or simply put on the clipboard for subsequent pasting into a document. Choose between exporting the display as is, with white background and in black and white mode. Export either the active viewport only or all viewports. Read more about this in Section 3.1.26.

Header and footer texts may be given for plot files. These will be added to the plot file with increased font size compared with other text. (Header and footer cannot be given when exporting all viewports.)

Numeric results may be tabulated on-screen and saved in a file suitable for importing into a spreadsheet. The currently selected component is tabulated initially. The table may then be extended to include all components, all result cases and corresponding coordinates. Read about tabulation in Section 3.6.

2.11 Viewing a 3D Model with Results in Xtract Viewer

A 3D model with a selected result component and result case (termed a display scene) may be exported to a VTFx-file¹. The display scene may be an animation of dynamic results. The VTFx-file may be opened in the free program Xtract Viewer² for interactive 3D viewing of the model with results. By use of the free GLview 3D Plugin³ embedded into MS Word and PowerPoint interactive 3D viewing of the model with results is also available in a Word report and a PowerPoint slideshow. A Sesam user may thus prepare data and send to a non-Sesam user who may view the data in an Xtract like environment. Read more about this in Section 3.13.1.

2.12 Viewing Two 3D Models (Global Model plus Sub-Model) in Xtract Viewer

Two 3D models may be exported to a common VTFx-file. The feature is general but it is particularly useful for comparing a sub-model with the global model in a sub-modelling type of analysis, i.e. involving use of the program Submod. The deformed shape of the two models may for instance be overlaid to verify that the deformations at the outer boundary of the sub-model match those of the global model. Such match of deformations is an absolute demand in a sub-modelling analysis. Read more about this in Section 3.13.2.

2.13 Automatic Report Generation – the Xtract Report Builder

A report of the analysis results may be created in an automated fashion. Rather than the traditional approach of saving stress contour images in Xtract (e.g. as gif-files) and subsequently manually import these into the report (e.g. an MS Word document) the stress contour images are saved to a repository (database) which in turn is opened by the Xtract Report Builder⁴. This program allows for automatic generation of a report by merging the data saved to the repository with a report template. Not only 2D images but also 3D models

-
1. VTFx is a proprietary file format of Ceetron AS. It is an XML based and enhanced version of the VTF format.
 2. Xtract Viewer is a free 3D viewer for presentation and distribution of 3D models and results. The product enables full 3D interactivity and high performance graphics. Xtract Viewer is developed and owned by Ceetron AS.
 3. GLview 3D Plugin is a free 3D viewer for presentation and distribution of 3D models and results. The product is embedded into MS Word, MS PowerPoint and MS Internet Explorer and enables full 3D interactivity and high performance graphics. GLview 3D Plugin is developed and owned by Ceetron AS.
 4. Xtract Report Builder is a program for easy capturing of information during postprocessing using Xtract and exporting the captured information to a report. Xtract Report Builder is developed and owned by Ceetron AS and marketed by DNV Software.

with associated data like maximum and minimum values as well tabular data may be saved to the repository for reproduction in the report. Read more about this in Section 3.13.3.

2.14 Logging and Reading Commands

The user interaction through the GUI as well as commands entered through the keyboard (line-mode commands) may be logged on a user given file name. This file name may be edited and used as a Command input file to a subsequent run. This feature allows an interactive session to be modified and re-run thereby saving time in e.g. creating load combinations and producing plots for a report. Read more about working with line-mode commands in Section 3.16.

2.15 Transfer of Data through the Sesam Interface File System

Xtract reads model and results data from Sesam Interface Files. These files are comprised of the following subsets:

- Input Interface Files (T#.FEM)
- Loads Interface Files (L#.FEM)
- Hydrodynamic Results Interface File, also termed Global Response File (Gn.SI*)
- Results Interface Files (R#.SI*)

The name of a Sesam Interface File may be written:

prefixX#.frm

where:

- ‘prefix’ is an optional character string.
- ‘X’ represents a single character taking the values T, L, G or R as indicated above.
- ‘#’ is the superelement number, or in case of the Hydrodynamic Results Interface File a data set identifier n, normally equal to 1.
- ‘frm’ is a file extension that takes different values:
 - FEM for T- and L-files
 - SIF (ASCII formatted), SIU (sequential unformatted) or SIN (database format) for G-and R-files

An example of a name of an Input Interface File with prefix ‘ABC’:

ABCT5.FEM

An example of a name of a Structural Results Interface File in database format and with prefix ‘ABC’:

ABCR5.SIN

2.16 Transfer of Data to Xtract through a VTF-File

The following Sesam programs produce VTF formatted files containing results for presentation: Installjac, Riflex, Stofat and Wasim. Xtract may open and process such VTF-files.

3 USER'S GUIDE TO XTRACT

This chapter explains how to use the program for the most typical postprocessing tasks. Section 3.1 is a tutorial in basic use of Xtract. Section 3.2 and Section 3.3 provides details on the organisation of the structural analysis results data (from Sestra). Section 3.8 addresses presentation and animation of hydrodynamic results (from Wadam or Wasim). Section 3.9 through Section 3.12 addresses how to process results stored on a VTF-file from Wasim, Installjac, Riflex and Stofat. The other sections (Section 3.4 through Section 3.7 plus Section 3.14 through Section 3.16) provide information on how to do specific and some specialised postprocessing tasks.

3.1 Getting Started

This section takes you through the basic operations of Xtract. That is, using its graphical user interface (GUI) for basic postprocessing tasks. You may want to perform these operations as an exercise using the same model as referred to here: a plate connection with stiffeners and bracket. Then you need to create the model in either GeniE, Patran-Pre or Prefem using input given in Appendix A 1 (the input may be copied from a pdf version of this manual) and run a static analysis in Sestra before proceeding with the tutorial.

Even though this tutorial is not directly applicable for presenting results from Wadam, Wasim, Installjac, Riflex and Stofat (that all create VTF-files for presentation in Xtract) several of the issues discussed herein are relevant for such use also. You are therefore advised to study this tutorial before proceeding to the relevant one of Section 3.8 through Section 3.12.

3.1.1 Starting Xtract and Opening the Results Interface File

Start Xtract from Sesam Manager, from Brix Explorer for Sesam, from GeniE or from HydroD. Alternatively, if you have set the SIN file type to be opened with Xtract you may double-click the SIN-file produced by the Sestra analysis. Depending on how you start Xtract some of the initial steps listed below may be performed automatically.

In Xtract do the following:

- 1 Open the Results Interface File by File > Open (or click the tool button ). If you have a superelement model then select the top level file whichever format this has (SIF, SIU or SIN, see Section 2.15).
- 2 Having opened the results file the Database dockable window will appear and the model name being the same as the name of the results file is shown. Double-click it and the model will be displayed. To view the FE mesh do Display > Show mesh (or click the tool button .
- 3 Now present the displacements of the first result case by clicking the Result Cases tab of the Database window (see Figure 3.2) and then double-click in sequence the result position Nodes (see Section 3.2.1 about result positions), the attribute DISPLACEMENT and the component ALL (see Appendix B 2 about attributes and components).

The Xtract window will open and display the model with displacements for the first result case as shown in Figure 3.1.

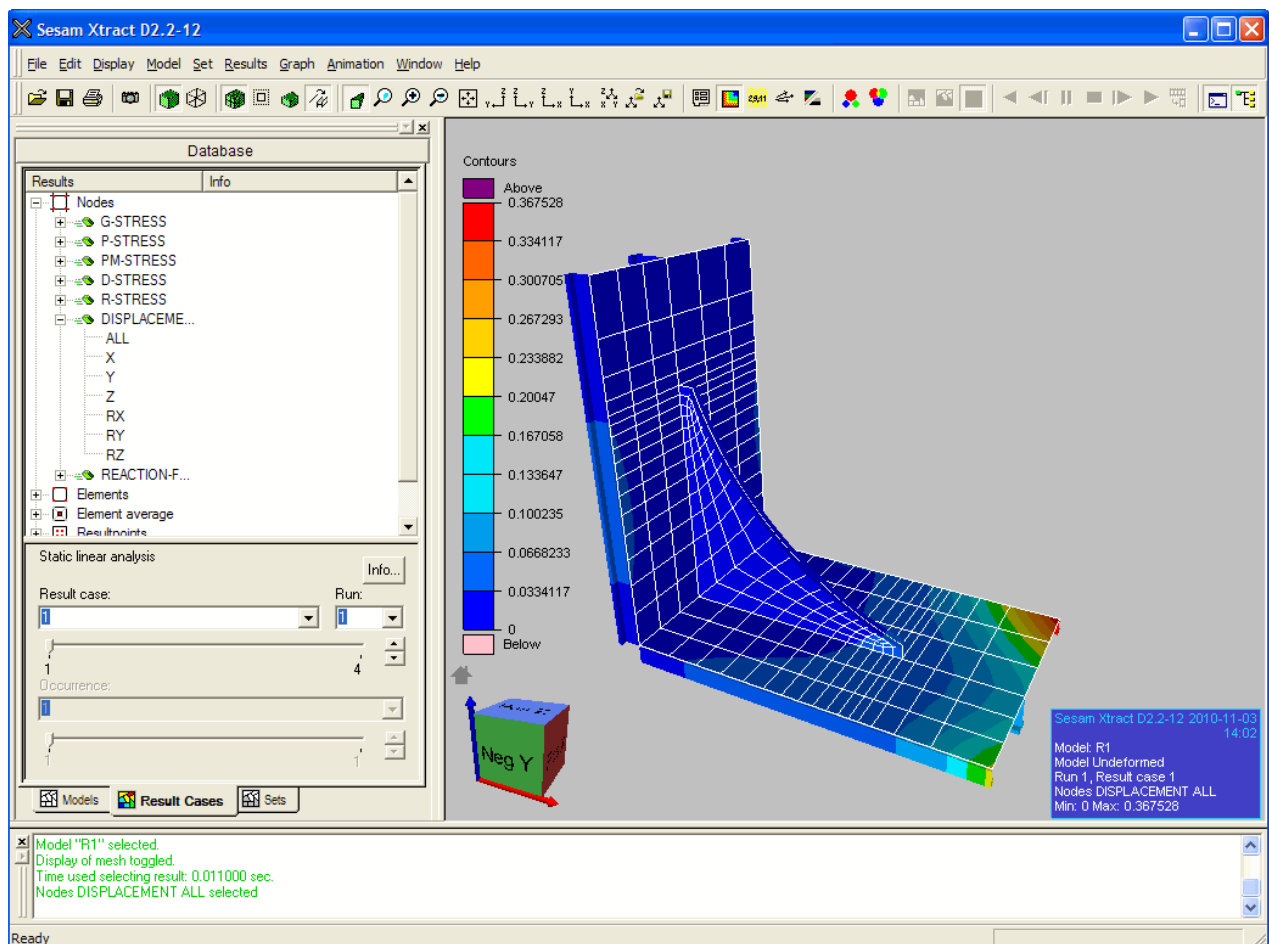


Figure 3.1 Xtract window with initial display of the model – displacements are contoured

3.1.2 The Graphical User Interface of Xtract

The Xtract window consists of various dockable windows, i.e. windows that may be docked along the edges of the Xtract window or torn loose and moved to any position on the screen. Figure 3.2 is a sketch illustrating the graphical user interface of Xtract with its dockable windows. Figure 3.31 shows an example of the Database window torn loose to increase the display area. Note that the Database window and the Command and message window may also be re-sized.

The dockable windows are used as follows:

- The menu bar and tool button bar for all general interactive operations
- The Database window for selecting model (superelement), result case, result component and set
- The Command and message window for entering commands through the keyboard (line-mode commands) and for feedback from the program

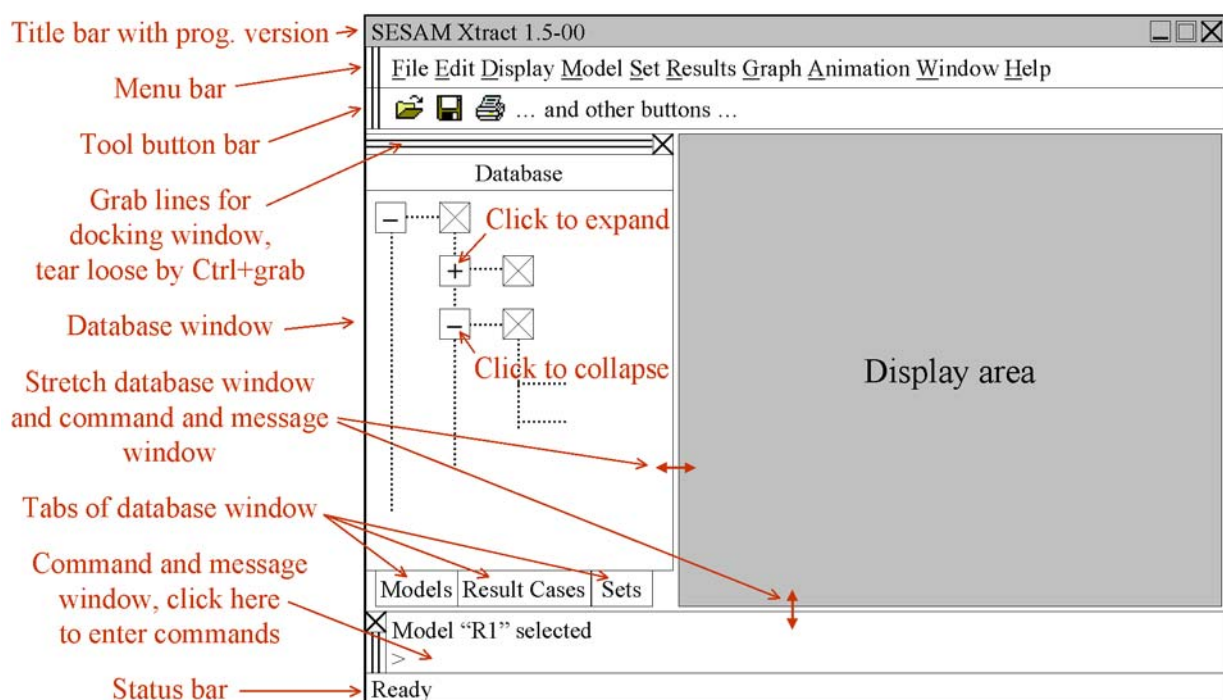


Figure 3.2 The graphical user interface of Xtract

3.1.3 Basic Display and Model Manipulation

You may manipulate the model display interactively (the most important methods of manipulation is highlighted by **bold text**):

- Rotate by either:
 - **Pressing the right mouse button (RMB) and moving the mouse up/down/left/right**, or

- **Clicking the navigation cube** in the lower left corner of the display area. Figure 3.3 shows the navigation cube in various situations: 1) initially, when positioning the mouse on the 2) edge, 3) corner and 4) side (click these to view the model from that position), 5) after clicking the side 'Neg Y' (in this situation arrows for rotation appear), and 6) after clicking the arrow for rotating 90 degrees clockwise.

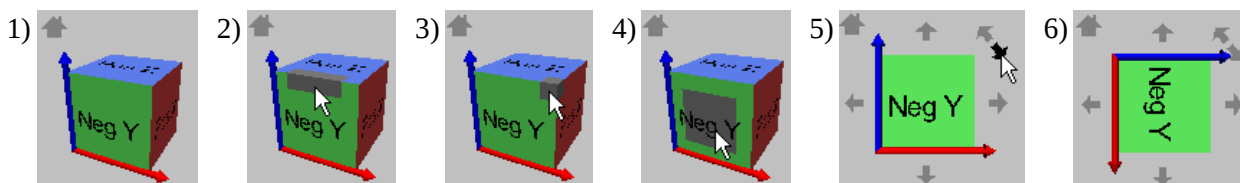


Figure 3.3 Use of the navigation cube

- Zoom by alternative means:
 - **Shift+RMB and move the mouse up (zoom in) and down (zoom out)**
 - **Rotate the mouse wheel up (zoom in) and down (zoom out)**
 - Zoom in and out by a factor of 1.5 by clicking the tool buttons and .
 - Zoom with rubberband by Display > Zoom (or click the tool button) and dragging a rectangle with the RMB.
 - **Zoom with rubberband by Shift+Ctrl+RMB and dragging a rectangle**
- **Pan by Ctrl+RMB and moving the mouse.**

You may jump to preset view positions by:

Display > View from > Global axes > ...

Display > View from > Isometric > ...

Notice that a selection of the preset view positions are available as tool buttons: . The latter two are return to initial (default) view position and overstore default view position by current position, respectively.

You may also open the Viewbox dialog, see Figure 3.4, by which the model may be rotated, panned and zoomed in user-controlled increments. Clicking the Rotate and Zoom Default buttons will bring you back to the initial view position. The Viewbox dialog is opened by:

Display > Viewbox

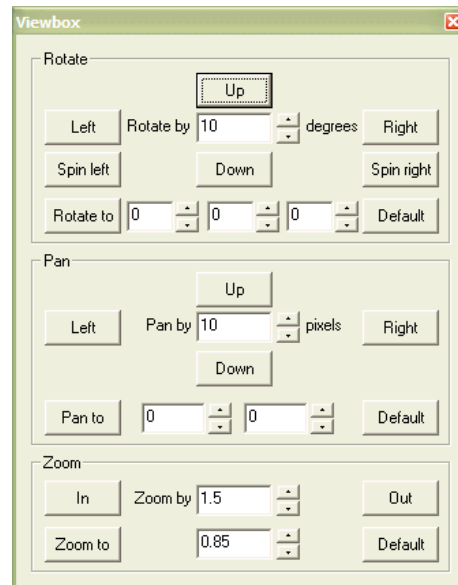


Figure 3.4 Viewbox dialog for rotating, panning and zooming

Make the complete model fit into the display area by:

Display > Frame (or click the tool button  or Ctrl+F)

Add and remove (toggle) display of the FE mesh:

Display > Show mesh (or click the tool button  or Ctrl+Alt+M) – only relevant with Show as surface

Show the FE mesh as lines only (wireframe), as outline (model edges only), as lines with hidden lines removed and thereafter return to Show as surface:

Display > Show as lines (or Ctrl+Alt+L)

Display > Show as outline (or click the tool button  or Ctrl+Alt+O)

Display > Show as hidden lines (or Ctrl+Alt+H)

Display > Show as surface (or click the tool button  or Ctrl+Alt+S)

When Display > Show as surface is active beam elements are shown with their sections (I or H, pipe, box, L (angle), channel, bar, un-symmetric I and double bottom). Show beams as lines by:

- Clicking the tool button  (this takes effect for all superelements in a superelement model)
- Alternatively, for a superelement model, open the Part attributes dialog by Display > Part attributes, select the appropriate part(s) – in most cases this will be the superelement(s) – and switch off Expanded (see Figure 3.41).

When not being expanded beam elements are displayed as thick lines between nodes (i.e. not showing the cross section and neglecting any eccentricities). Adjust the line thickness by:

Display > Beam width > *pixels* (default number of pixels is 3)

Note: Display > Show as (hidden) lines will display beam elements with thin lines irrespective of the 'Expanded' and 'Beam width' settings.

3.1.4 Viewing Results

Now we want to view some more results. Click the Result Cases tab of the Database window and see that the following five result positions are shown (Section 3.2.1 provides more information on result positions):

- Nodes
- Elements
- Element average
- Resultpoints
- Surface resultpoints – only for models containing 6 and 8 node shell elements

Section 3.1.5 through Section 3.1.14 deal with viewing various results.

Note: Use of Sesam is based on choosing a consistent set of units: SI, Imperial, American or any other consistent set. This choice is made when creating the modelling. However, the time unit is seconds and the rotational unit is radians.

Note: The results file contains no information on the chosen set of units. Results interpretation in Xtract is therefore based on knowing which set of units were chosen in the modelling stage. See Section 3.1.21 Unit Conversion.

3.1.5 View Nodes/DISPLACEMENT

Notice that the result position Nodes is expanded (the attributes G-STRESS, P-STRESS, etc. are listed) while the result positions Elements, Element average, Resultpoints and Surface resultpoints are collapsed, see Figure 3.5. Furthermore, the attribute DISPLACEMENT is expanded (the components ALL, X, Y, etc. are listed).

View any component (X, Y, etc.) of the total displacement (ALL) by double-clicking the component.

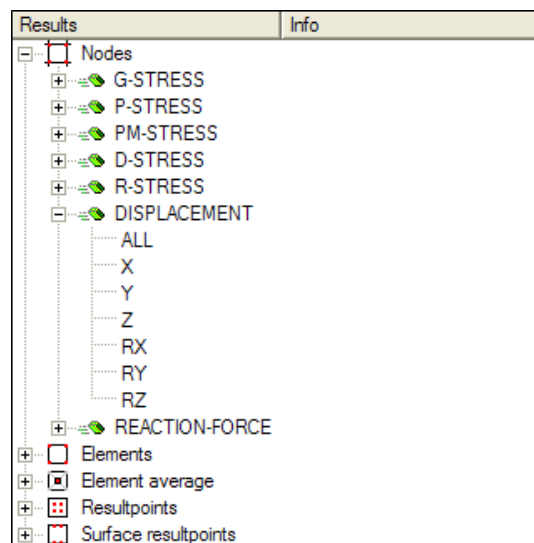


Figure 3.5 Result positions shown in Database window – Nodes/DISPLACEMENT expanded

3.1.6 View Nodes/G-STRESS

Expand the G-STRESS attribute of Nodes by clicking the plus sign and then double-click the VONMISES component to see the von Mises stresses averaged between coplanar adjoining elements. See section “Result Position: Nodes” on page 3-55 about nodal averaging.

Note: The von Mises stress is computed as follows: Basic stress components are averaged between coplanar adjoining elements. Von Mises is then computed according to Appendix B 3.4.

Display the model deformed at the same time by:

Model > Deformed (or click the tool button  or Ctrl+Alt+D)

Increase the scaling of the deformations and add an undeformed wireframe of the model by:

Model > Deformation scaling

Model > Add undeformed wireframe (toggle for adding and removing wireframe)

By default the results are shown for the upper plate/shell surface (as stated in the Info box in the lower right corner) which is on positive local z-axis side. See Section 3.2.2 for more information on the concept of upper and lower surfaces.

3.1.7 View Elements/G-STRESS

Now, expand the result position Elements and thereafter the G-STRESS attribute and double-click the VONMISES component to see the von Mises stresses of the individual elements. I.e. stresses *not* averaged between adjoining elements. See Figure 3.6.

Note: Both expanding (clicking the plus) and collapsing (clicking the minus) of the result positions and attributes may be done by double-clicking the result position/attribute.

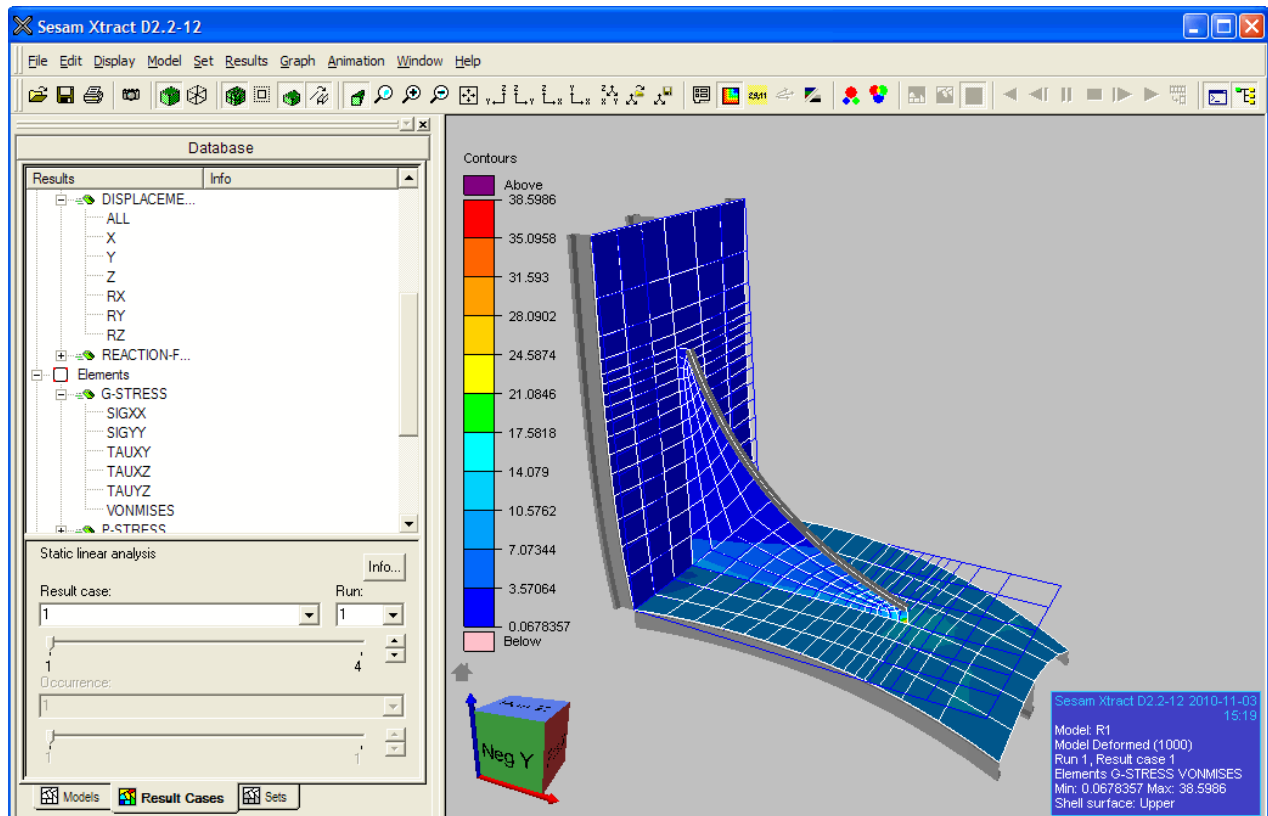


Figure 3.6 Result position Elements and attribute G-STRESS expanded and display of von Mises stress for deformed model

Note: Shell/plate and membrane stresses refer to local element coordinate systems. See Appendix B 2 for more information on this. See Section 3.1.25 about adding local element coordinate systems to the display.

If you want the exact values of the von Mises stress you may add numeric values to the display:

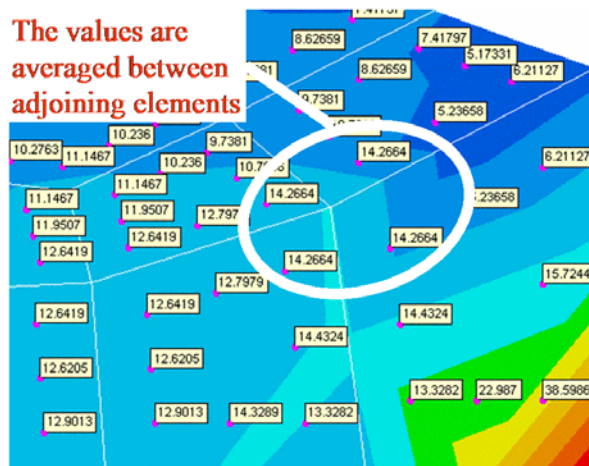
Results > Show values (or click the tool button )

Showing result values will easily clutter the screen. Zoom in to see the values as shown in Figure 3.7.

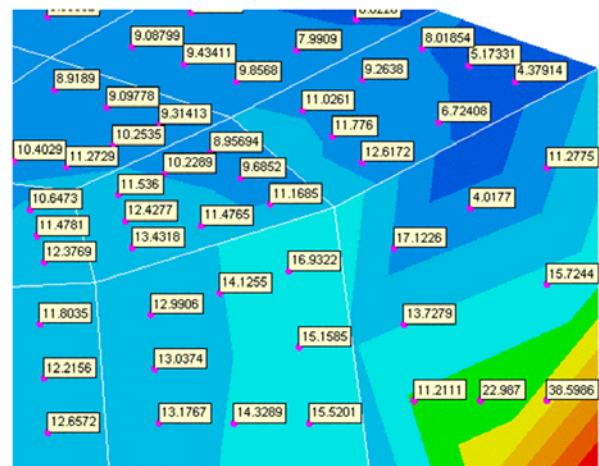
Note: Display > Labeling allows limiting the number of numeric values shown. By default maximum 10.000 values are shown. The user given number of values are distributed over the model. Setting the number to e.g. 100 prevents the screen from being cluttered. Zoom in and see more and more values, and finally all within the region, pop up. You may also select Show all.

Note: Display > Labeling also allows switching off the background box of the labels. See the lower left of Figure 3.7.

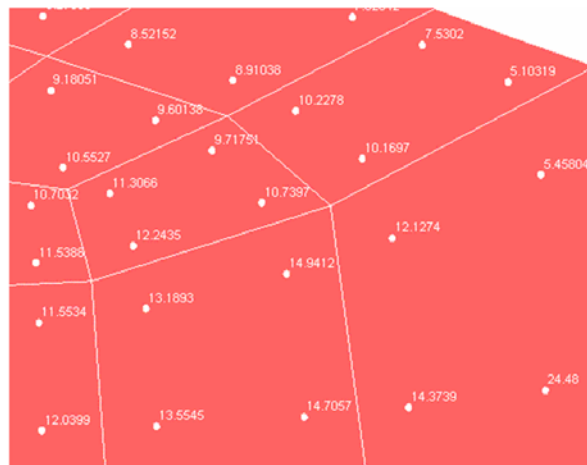
Note: Element related numeric values that refer to the nodes (e.g. Nodes G-STRESS and Elements G-STRESS) are pulled inside the elements so as not to overlap each other. See Figure 3.7.



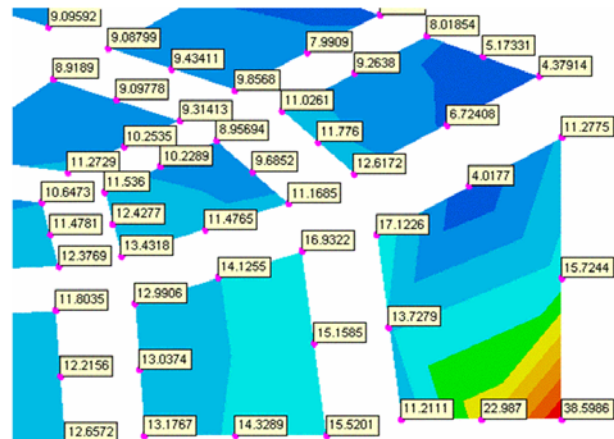
Nodes G-STRESS values refer to nodes but are pulled inside the elements to avoid overlapping.



Elements G-STRESS values refer to nodes but are pulled inside the elements to avoid overlapping.



Resultpoints G-STRESS values for 8 node shell elements are shown inside elements in the true position of the result points.



With shrunken elements the Elements G-STRESS values are positioned in the 'shrunken nodes'.

Figure 3.7 Result values are pulled inside the elements (here 8 node shell) to avoid overlapping

3.1.8 View Elements/P-STRESS

Expand the result attribute P-STRESS of position Elements and double-click the P1 component. The first (highest, most positive) principal stress is shown. Zoom in on an area of interest (ref. Figure 3.8) and show the principal stresses as vectors by:

Results > Show vectors (or click the tool button )

Note: Labels (result values, principal stress vectors, etc.) are lifted towards the eye in order not to be hidden by the model itself. But in certain cases labels may still be partly hidden. Labels may then be lifted more by the Lift factor in the Labeling dialog (see Figure 3.39).

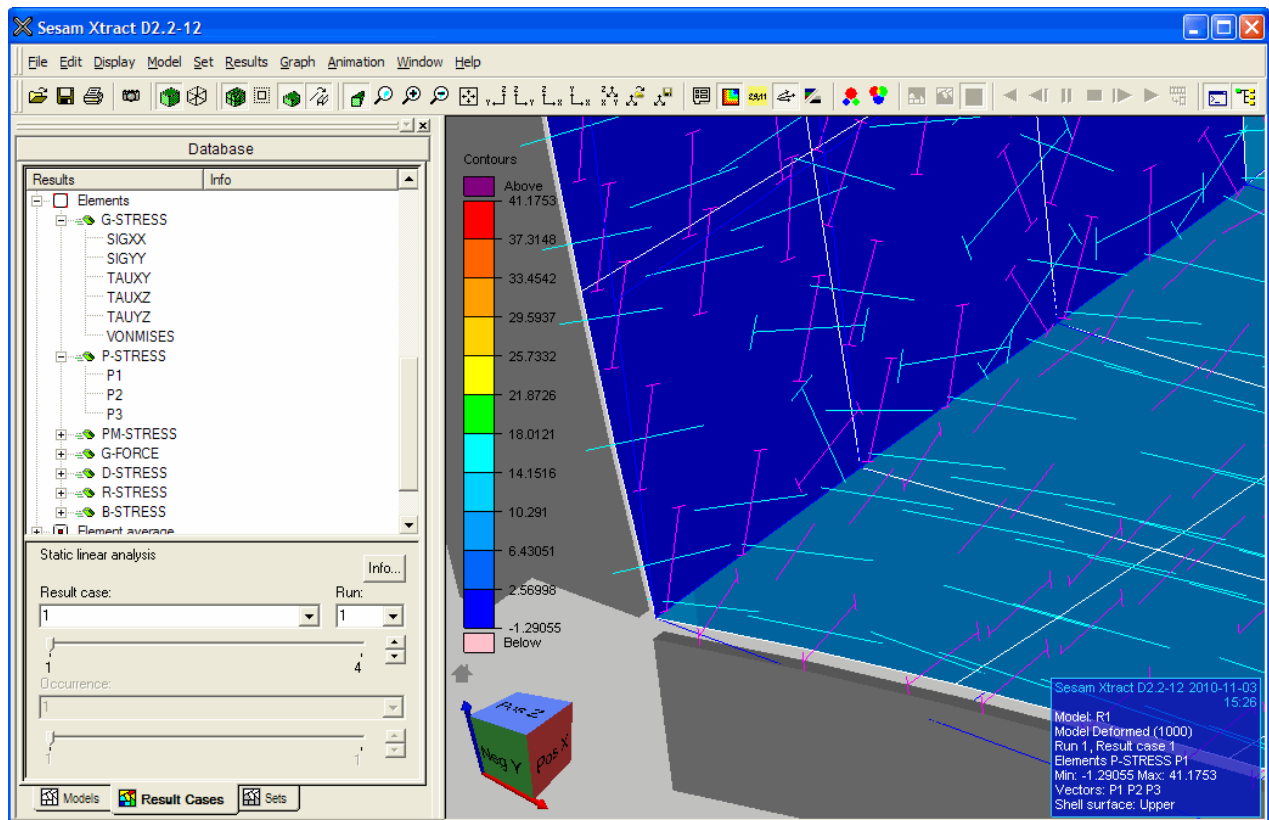


Figure 3.8 Principal stress P1 (Elements/P-STRESS) contoured and vectors for P1 and P2 shown

The principal stress vectors are shown as orthogonal intersecting lines. Negative (compressive) stresses are indicated by small crosslines at both ends of the lines. The three principal stresses are distinguished by colours and lengths as explained in Figure 3.9.

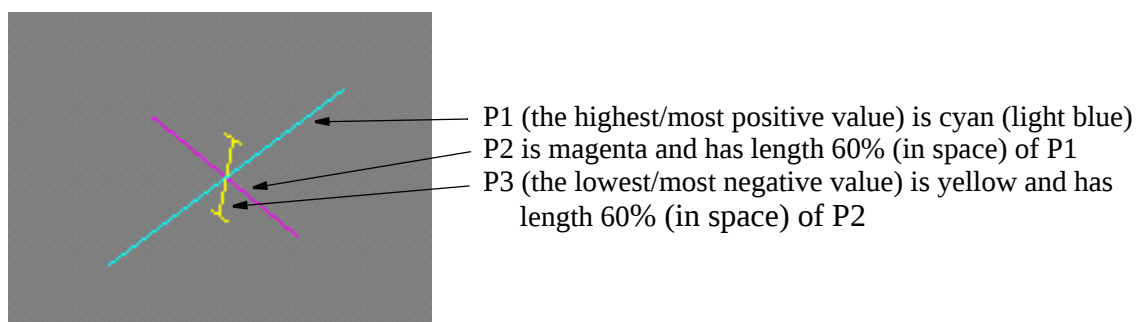


Figure 3.9 The three principal stress vectors shown as intersecting coloured lines

The vector lengths are by default as explained above but may be set to be proportional with value by:

Results > Settings > Vectors > Scale P-stress vectors according to value

Note: Principal stress vectors are by default shown for both P1 and P2 (and P3 when relevant) even though stress contours are only shown for one of them. The Info box in the lower right corner contains information about which principal stress is selected (contoured) and which principal stress vectors are shown.

The 6 and 8 node shell elements have three principal stresses, the Info box will therefore list all three. However, P3 is zero on the shell surfaces and its vector is therefore not shown in Figure 3.8. (Selecting a principal stress for position Resultpoints that are found inside the shell will display also P3.) You may switch off (and on again) display of vectors for P2 by:

Results > P-stress vectors > P2

Note: Principal stresses based on membrane stresses only are available through the attribute PM-STRESS. I.e. the principal stresses are based on the D-STRESS components SIGMX, SIGMY and TAUMXY and with no contribution from the bending components SIGBX, SIGBY, TAUBXY, TAUBXZ and TAUBYZ.

3.1.9 View Elements/G-FORCE

The forces and moments of the beam elements may also be shown. Expand the G-FORCE attribute of the result position Elements and double-click the NXX component to present the axial force. Results are only shown for beam elements, shell elements that do not have this particular result component are grey.

The positions and values of the maximum and minimum values are found by:

Results > Show peaks

See the model from another view point:

Display > View from > Isometric > -X+Y-Z

Figure 3.10 shows the axial force of the girders and stiffeners.

Note: The positioning of the maximum and minimum values does not account for any eccentricities.

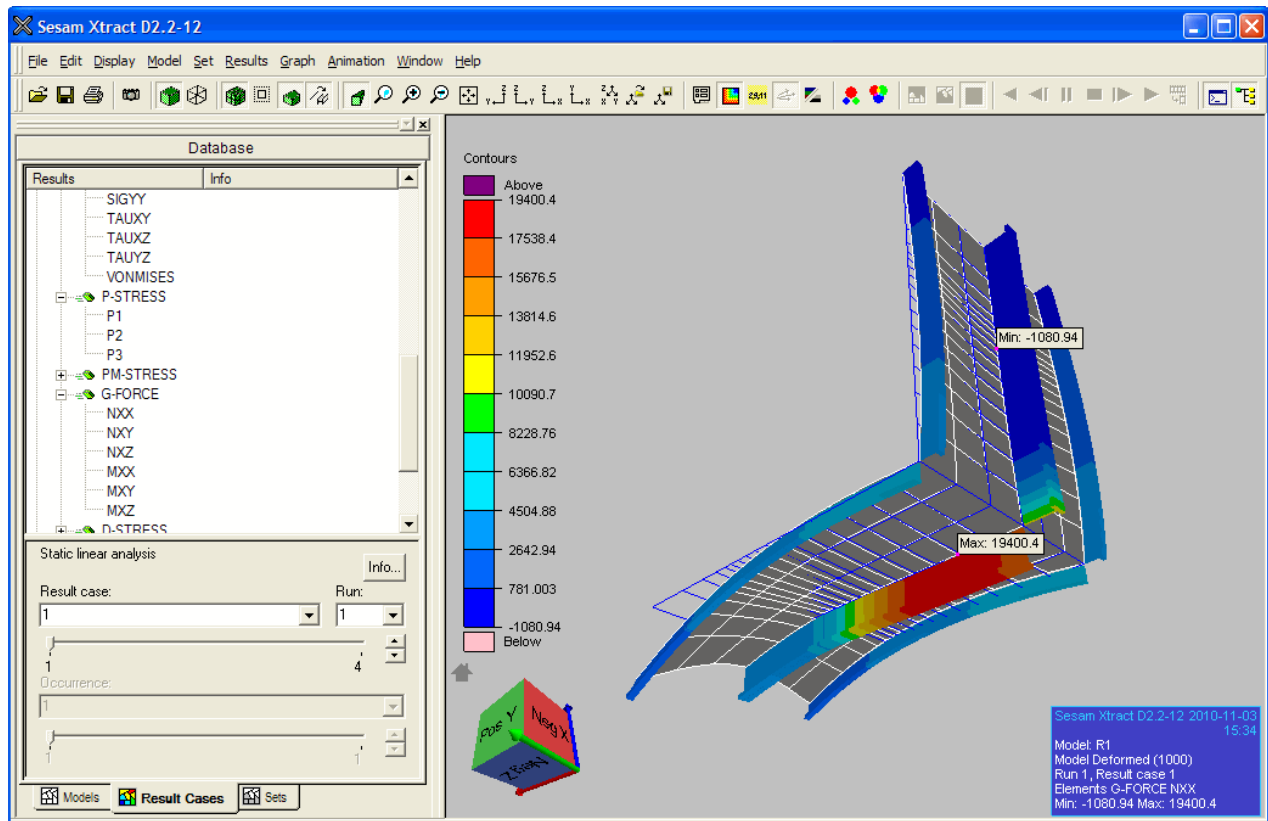


Figure 3.10 Axial force NXX (Elements/G-FORCE) contoured and peaks shown

3.1.10 View Elements/B-STRESS

Xtract computes axial and shear stresses in the beam elements based on the forces and moments (G-FORCE attribute). See Appendix B 4 for details on how the beam stress components are computed.

Expand the B-STRESS attribute of result position Elements and double-click the SIGBYX2 component to present the normal stress in (the most extreme fibre of) the flange of the girders and stiffeners due to the moment MXV.

Figure 3.10 shows the axial force of the girders and stiffeners.

Note: The whole beam section is coloured even though the B-STRESS components generally refer to the extreme fibres only. (The SIGXX component is constant over the section though.)

Note: Xtract does not provide information as to where in the section (for which extreme fibres) the B-STRESS components are found.

Note: See notes in Appendix B 4 about how to interpret the B-STRESS components.

Note: The B-STRESS attribute will not appear if reading beam sectional data is switched off using Display > Read beam expansion; see Section 3.15.1.

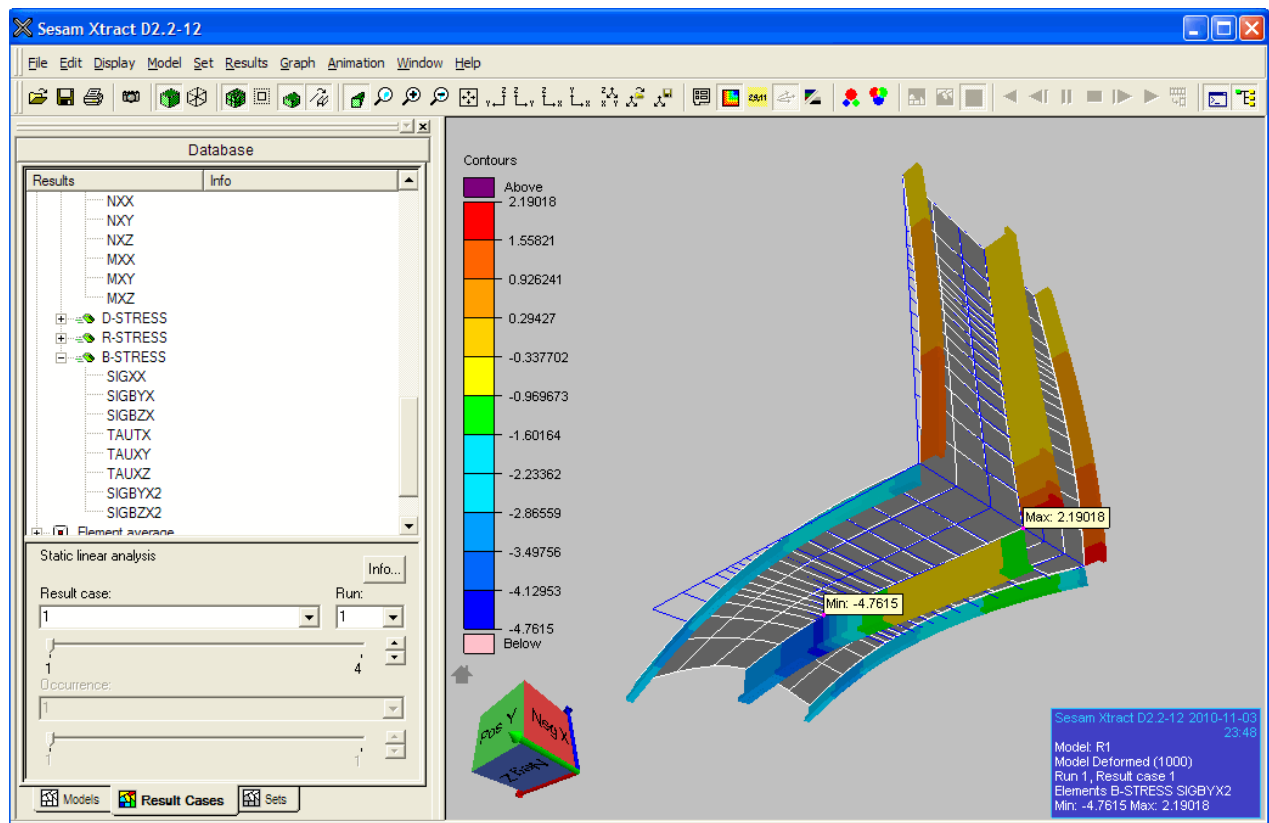


Figure 3.11 Normal stress SIGBYX2 (Elements/B-STRESS) contoured and peaks shown

3.1.11 View Element average/G-STRESS

Xtract computes element average values as the average of corner values in position Elements. Expand the result position Element average and thereafter the G-STRESS attribute and double-click the VONMISES component to see a single von Mises stress value per element. See Figure 3.12.

Note: Contour type Lines cannot be used for Element average results as it has no meaning.

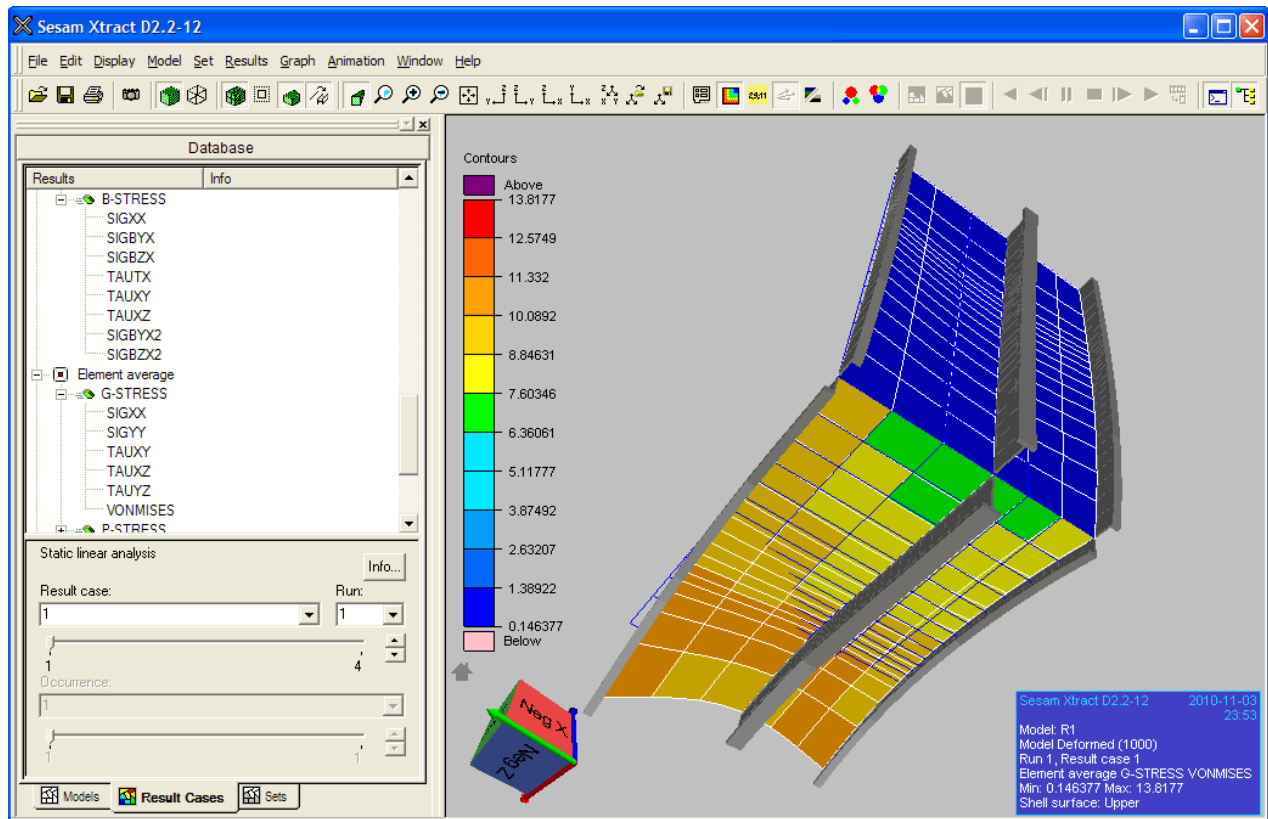


Figure 3.12 Von Mises stress in result position Element average

3.1.12 View all Components of an Attribute

In the previous sections a single results component is presented at the time. This section explains how you can present numeric values for all components of an attribute at the same time. This is done by entering a mode allowing you to click (select) nodes or elements, one at the time, for which all components will be presented in a box next to the nodes/elements. The element values presented are those belonging to the result position Element average, see Section 3.1.11.

Present all components of a Nodes attribute by either:

- 'Results > Show values for the following nodes' followed by LMB clicking nodes, or
- Ctrl+LMB click a node and continue LMB clicking more nodes.

Present all components of an Element average attribute by either:

- 'Results > Show values for the following elements' followed by LMB clicking elements, or
- Shift+Ctrl+LMB click an element and continue LMB clicking more elements.

Figure 3.13 shows an example of showing values for selected elements. The cursor may be positioned anywhere inside the element and the box will appear with its lower left corner in the element midpoint. To remove a box click the element once more.

When showing values for nodes click anywhere inside an element and the nearest node will be selected. The box will appear with its lower left corner in the node. To remove a box click near the node once more.

To remove all boxes give:

Results > Clear selected elements/nodes

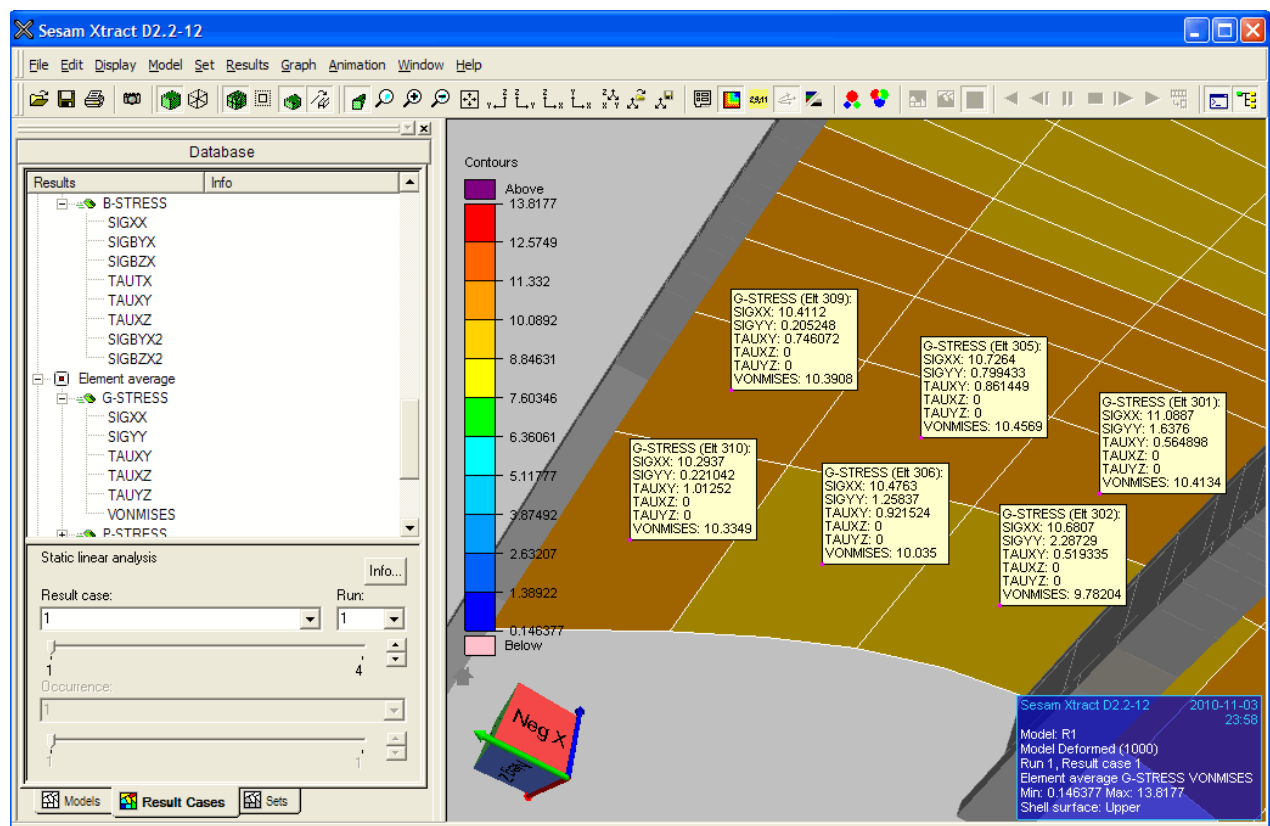


Figure 3.13 Boxes showing all components of an Element average attribute for selected elements

Note: Showing result values for elements and nodes requires 'Result > Show result values in labels' to be set (the default condition). Without this setting only the element/node number is shown.

Note: When showing element values: Components of an attribute of the position 'Element average' is presented in the box irrespective of which result position is selected, i.e. colour contoured. E.g. selecting a component of Nodes/G-STRESS or Elements/G-STRESS both result in Element average/G-STRESS components to be shown. Be aware of the possible confusion in this.

Note: When showing nodal values: Components of an attribute of the position Nodes is presented in the box irrespective of which result position is selected, i.e. colour contoured. E.g. selecting a

component of Elements/G-STRESS or Element average/G-STRESS both result in Nodes/G-STRESS components to be shown. Be aware of the possible confusion in this.

Note: For combinations and scan cases showing all components of an attribute cannot be done unless the combination/scan case includes the relevant Nodes/Element average attribute.

3.1.13 View Resultpoints/G-FORCE

The values as computed by the analysis program, i.e. neither interpolated, extrapolated nor averaged, are found in the Resultpoints position. These data represent the pure and non-manipulated data and may only be shown with their numeric values.

Expanding position Resultpoints and attribute G-FORCE and double-clicking NXX, see Figure 3.14, shows the numeric values of the axial force. To avoid cluttering the display the values shown may be limited to selected result point numbers. Figure 3.15 shows how Results > Settings (or click the tool button ) and selecting the Misc tab allows limiting values to selected result points: Check the 'Show result labels for selected resultpoints only' box, enter a result point number in the small list box and click Add to move the number to the larger list box. (See Section 3.2.1 for more information on result points and their numbering.)

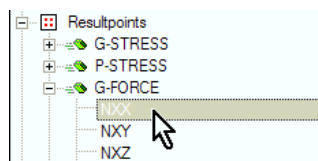


Figure 3.14 Selecting Resultpoints/G-FORCE/NXX

Note: Element related numeric values that refer to points positioned in the nodes (in this case Resultpoints G-FORCE) are pulled inside the elements so as not to overlap each other.

Note: As an alternative to limiting values shown to selected result points be aware of the possibility to show numeric values for the result position Element average, i.e. a single value for each element. See Section 3.1.11.

You may customize the display in various ways. You may for example set the background to white and increase the font size by Display > Settings. The colour of the superelement may be changed from red to white (or rather tones of grey due to light source and shading) by Display > Part attributes. You may remove the background box of the labels (the numeric values) by Display > Labeling. Read more about these settings in Section 3.1.25. When printing or saving graphics (plots) you may save (export) the graphics as appearing on screen (as is), with white background and in black & white mode. Read more about this in Section 3.1.26. Figure 3.16 illustrates such customization.

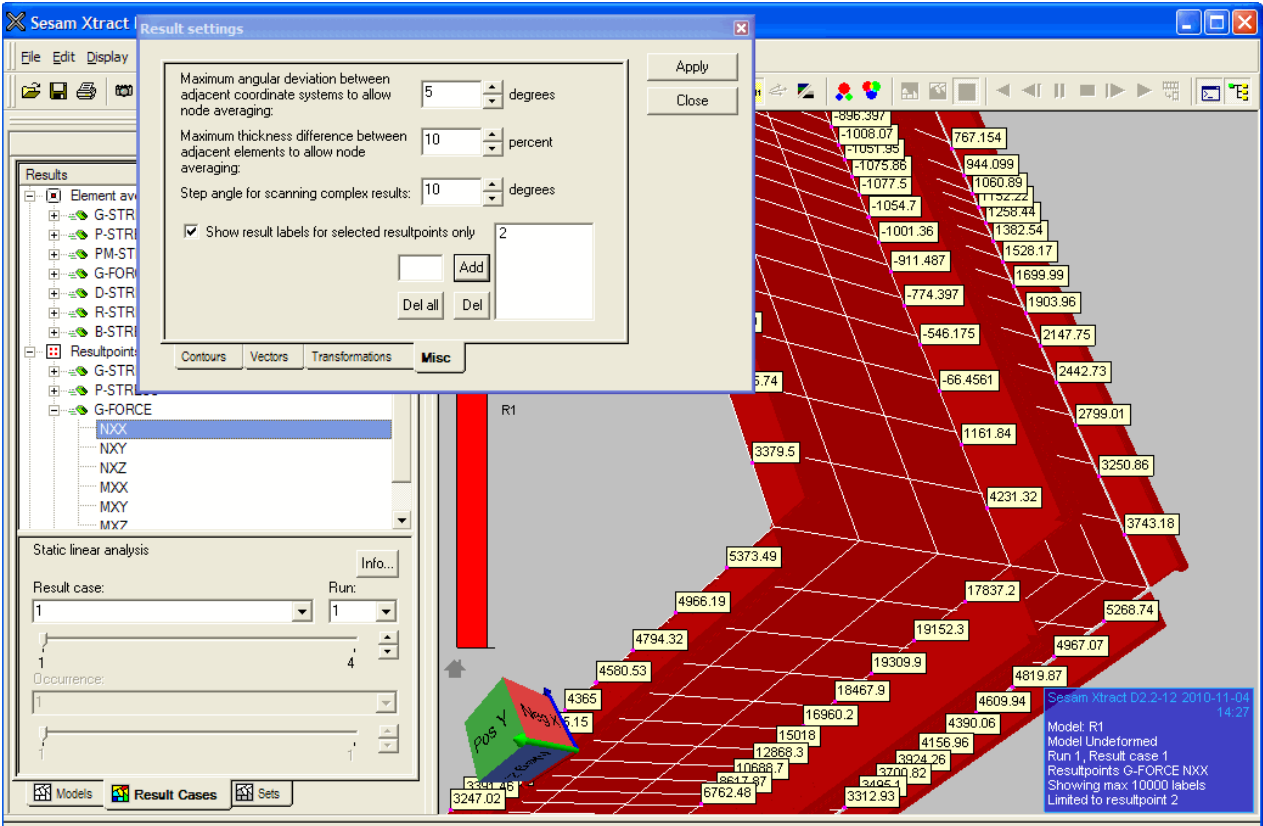


Figure 3.15 Axial force NXX (Resultpoints/G-FORCE) values shown for result point 2

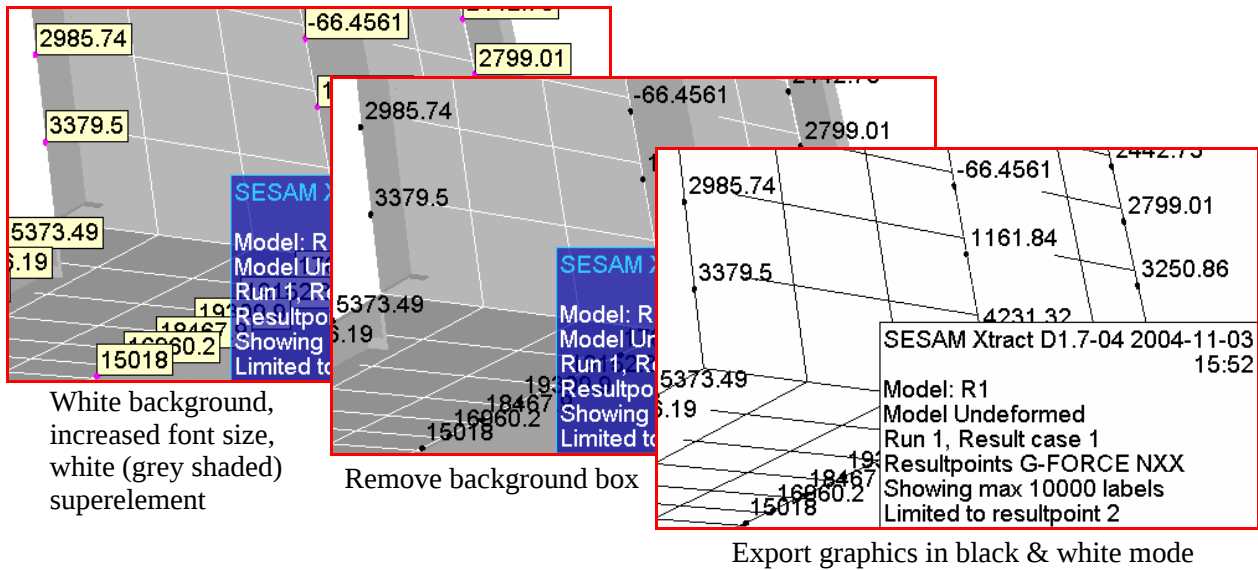


Figure 3.16 Customizing the display

3.1.14 View Nodes/REACTION-FORCE

Expand the REACTION-FORCE attribute of Nodes and double-click the X-FORCE component. As colour contour presentation is irrelevant for reaction forces (see note below) switch to numeric presentation by:

Results > Show values (or click the tool button )

Note: The value zero will appear for all nodes but the fixed ones (zero value means no reaction force) as seen in Figure 3.17. These zeros are irrelevant.

Note: The reaction forces are by default shown by colour contours (as for all Nodes and Elements results). This form of presentation is strictly irrelevant as the reaction forces cannot be interpolated and contoured inside the elements. The colours may be interpreted as a mere highlighting of fixed nodes. Alternatively, the colours may be switched off by Results > Show contours (or click )

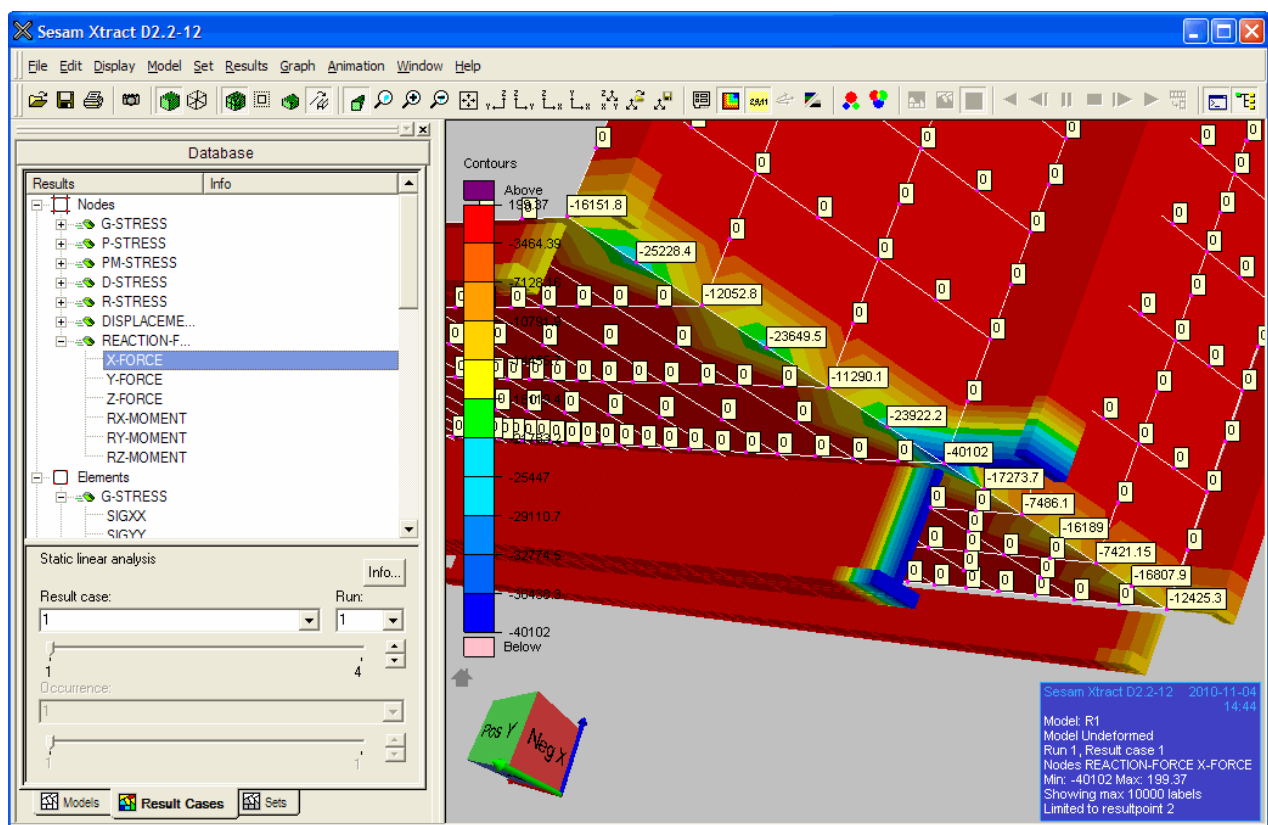


Figure 3.17 Reaction force X-FORCE (Nodes/REACTION-FORCE) values shown for fixed nodes

3.1.15 Creating a Graph

First, switch off showing results by Display > Results.

A graph may be presented for the axial force along the beam elements constituting the curved flange of the bracket. Open the Graph construction dialog by:

Graph > Generate graph

In the Graph construction dialog click the Item type Beams. The only attribute available for graphing for beams is G-FORCE in position Elements. Select the NXX component. Select a result case (or occurrence if relevant). Adjust the text for Curve name as desired.

Select all beam elements by checking the option Expand selection in the Graph construction dialog and then click the first and last beam of the flange (see Section 3.14.1 for a note on selecting beam elements). The selected beams are labelled by their element numbers and they appear in the Selected items box of the Graph construction dialog. Verify that all beams along the flange have been picked up. If not then click Clear in the Graph construction dialog and perform a new selection this time clicking as many times along the flange as needed to pick up all beams. (The Expand selection feature attempts to find a straight as possible path between the items you click.)

The graph appears in the upper right corner of the display area when Create is clicked. Adjust the appearance of the graph through:

Graph > Options

In the Graph options dialog you may change the title of the graph and its axes, change the background colour of the graph (when set to white the axes and text change to black), make it transparent, take control over the scaling of the axes and more. Figure 3.18 shows the Graph construction and Graph options dialogs together with the Xtract window with the graph. The view direction has been changed to another isometric view by Display > View from > Isometric > +X -Y +Z followed by zooming in a bit.

Other result cases may be added to the same graph by selecting a new result case (the Occurrences box), checking the Leave old curves option and clicking Create. You may fill the entire viewport with the graph through:

Graph > Draw graph in > Entire viewport

Making the graph transparent (Transparent plot in Graph options dialog) and having suitable colours for the graph background (Background button in Graph options dialog) as well as the display background (in Display settings dialog) allow you to present the graph on top of the model as exemplified in Figure 3.19 where NXX is graphed for four result cases.

Switch off the graph by the toggle:

Graph > Show graph

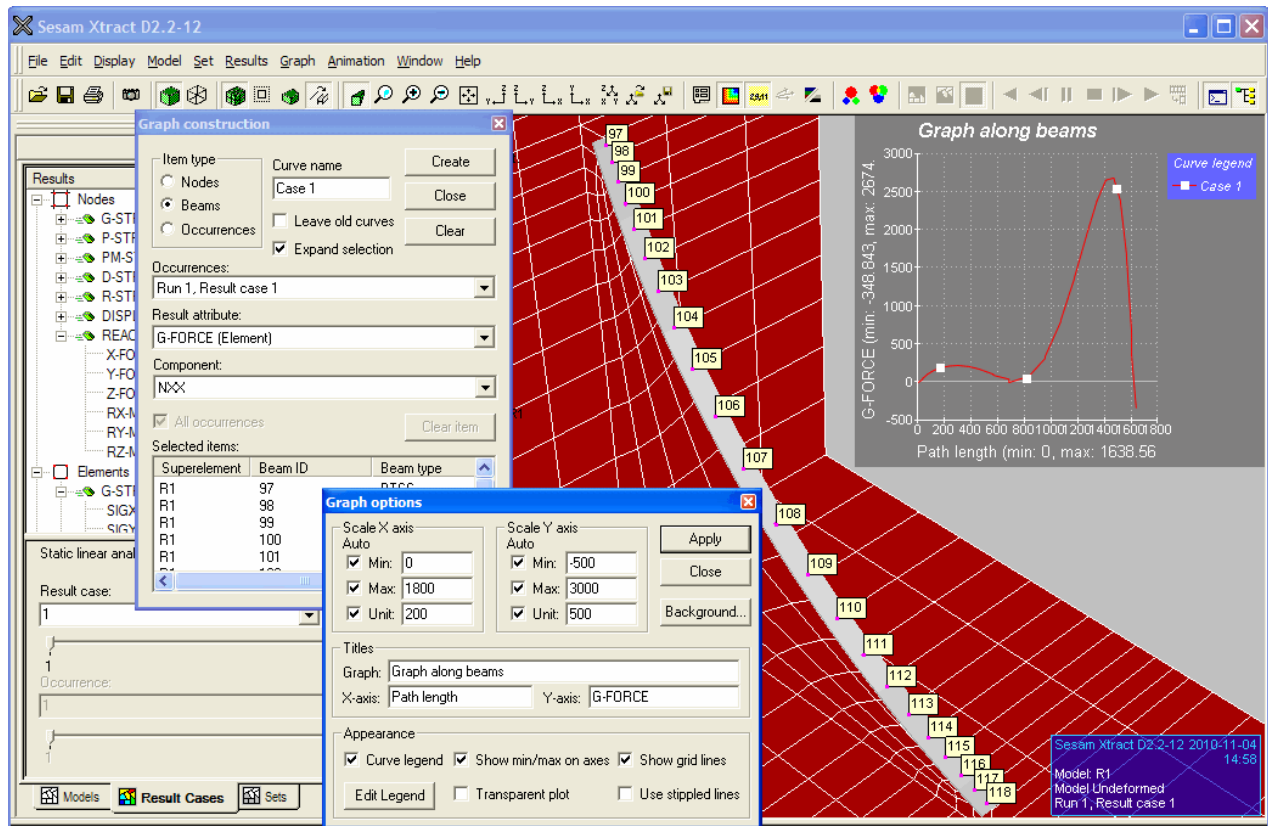


Figure 3.18 Graph for beam force created through Graph construction and Graph options dialogs

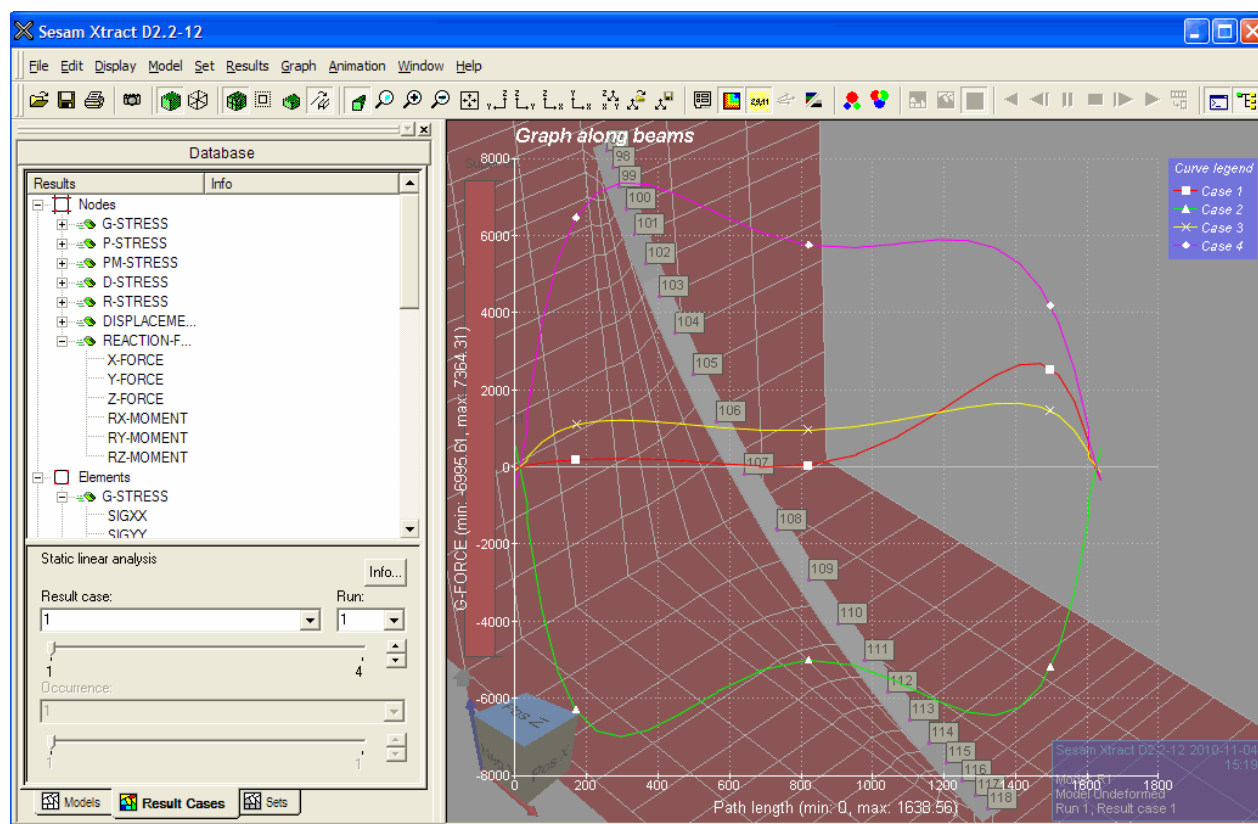


Figure 3.19 Transparent graph filling entire viewport

You may also create a graph for various result components over selected nodes by selecting Item type Nodes in the Graph construction dialog; see Figure 3.20.

Note: The Expand selection check box, see Figure 3.18 and Figure 3.20, allows selecting all beam elements or nodes on a straight line (or a line as straight as possible) between two selected items.

Note: See Section 3.14.1 for more notes on graphing.

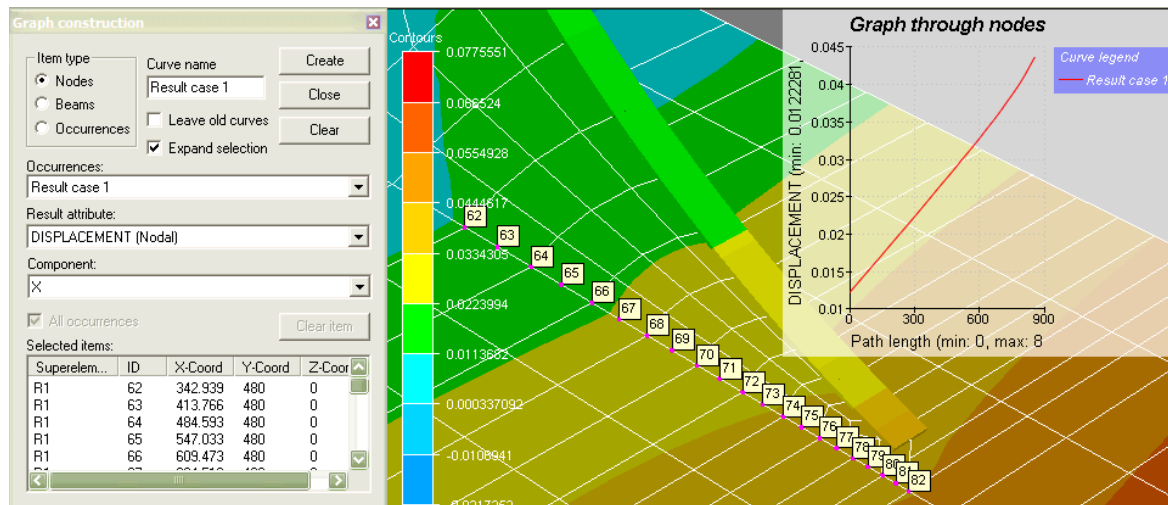


Figure 3.20 Graph for displacement over nodes (the figure is cropped)

Finally, you may create a graph for result components in a single node over selected occurrences of a result case (see Section 3.3.2 for an explanation of occurrences) by selecting Item type Occurrences in the Graph construction dialog. This type of graph is relevant for results from frequency and time domain analyses.

Note: When graphing results from frequency and time domain analyses it may be relevant to select all occurrences of a result case. Rather than selecting the occurrences one-by-one select only one, for example the first, and check the All occurrences check box as shown in Figure 3.21.

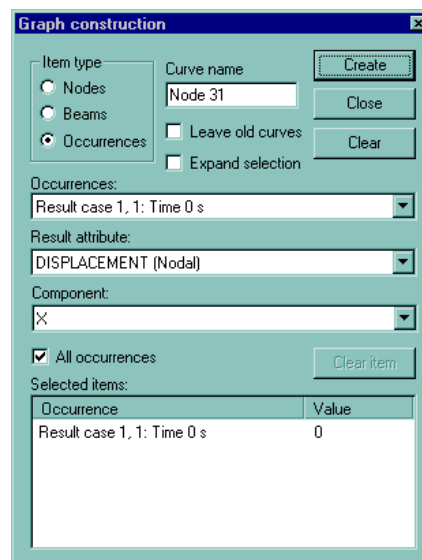


Figure 3.21 Selecting all occurrences of a result case in the Graph construction dialog

Having created a graph the values of all the curves of the graph may be exported to a file by the line-mode command (include the file type txt in the file name, e.g. values.txt):

graph export *filename*

3.1.16 Selecting Result Case

Now we want to view results for another result case. At the bottom of the Database window (Result Cases tab) you may select a new result case in three alternative ways: (1) through the Result case combo box, (2) through the Result case slider and (3) through the Result case spin (up/down) buttons. See Figure 3.22.

Note: When dragging the Result case slider the command log, see Section 3.16, may include logging of some arbitrary intermediate result cases. It may also include multiple logging of the same result selection command depending on the speed by which the slider is dragged. You may want to remove such superfluous logging when using the log as a command input file or documentation of work done.

Note: The combo boxes for Run and Occurrence are inaccessible (grey) here as this analysis only contains a single Run number and each Result case contains only a single Occurrence.

The text above the Result case combo box provides basic information on the result case. Click the Info button to get more information (result case properties). For a combination or scan result case all contributing result cases will be listed; see Figure 3.24 and Figure 3.27.

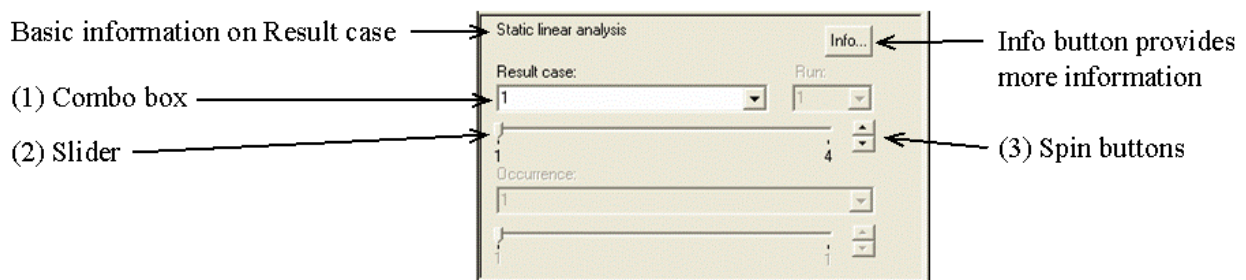


Figure 3.22 Result case selection in three alternative ways – Info button for result case information

3.1.17 Combining Result Cases

Combining (adding with scaling factors) result cases is done through:

Results > Combine (or click the tool button )

The Combine dialog, see Figure 3.23, appears in which Source result cases are transferred to the Combination result cases list box. Select result cases in the Combination result cases list box and adjust the Scaling factor (default 1), give the combination a name and click Apply.

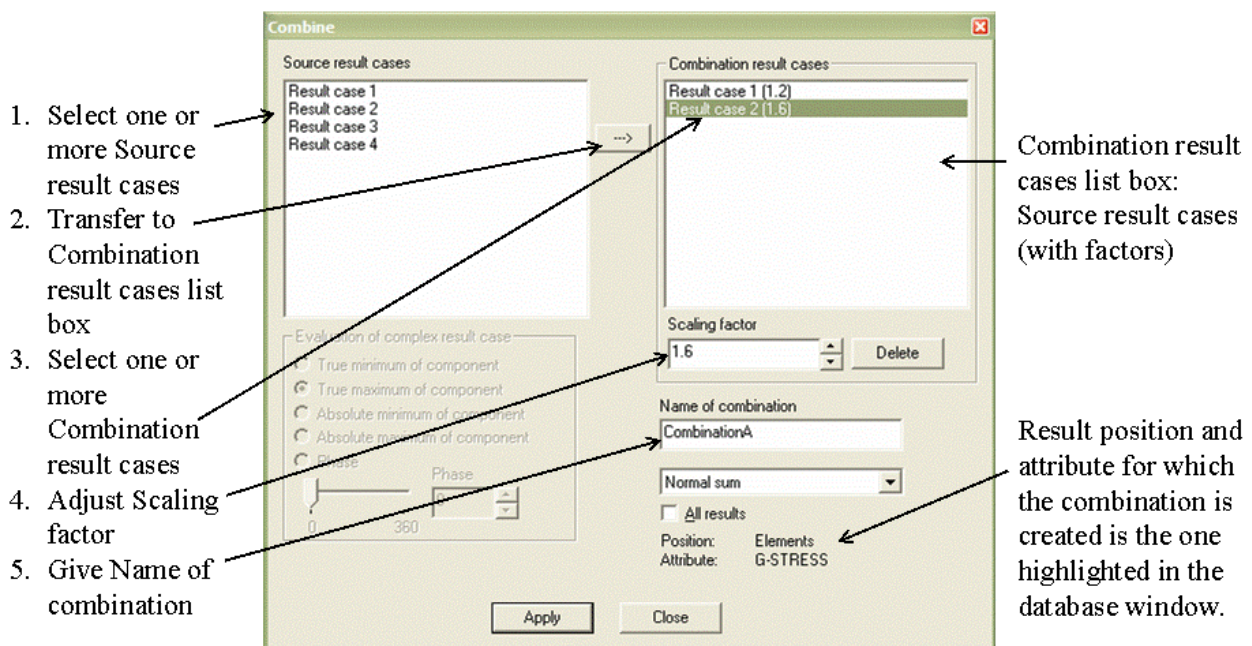


Figure 3.23 Combining result cases

Note: You may select and transfer two or more Source result cases at the time.

Note: You may select and change Scaling factor for two or more Combination result cases at the time.

Note: The combination is established for the result position and attribute (all its components) highlighted in the database window and listed to the lower right in the Combine dialog. Change result position and attribute by selecting (single-click to highlight) the appropriate one (or any component of the attribute) and return to the Combine dialog. The new result position and attribute will be listed in the Combine dialog.

Note: You may want to establish the combination for all available results (all attributes of all positions). Do so by checking the All results option in the Combine dialog. Alternatively, you may add extra attributes to an existing combination by the line-mode command 'combine add'.

Note: When checking the All results option and one of the selected result cases is complex then the evaluation should be performed for a specific phase only, i.e. do not choose True/Absolute Minimum/Maximum.

Combinations are put in a Run named User (see Section 3.3.1 for an explanation of the concept of Run numbers). To select the combination you therefore need to select the Run named User (see Figure 3.24), thereafter select the relevant Result case (i.e. the combination name) and finally expand to find and double-click the desired component.

Click the Info button to get information on a combination.

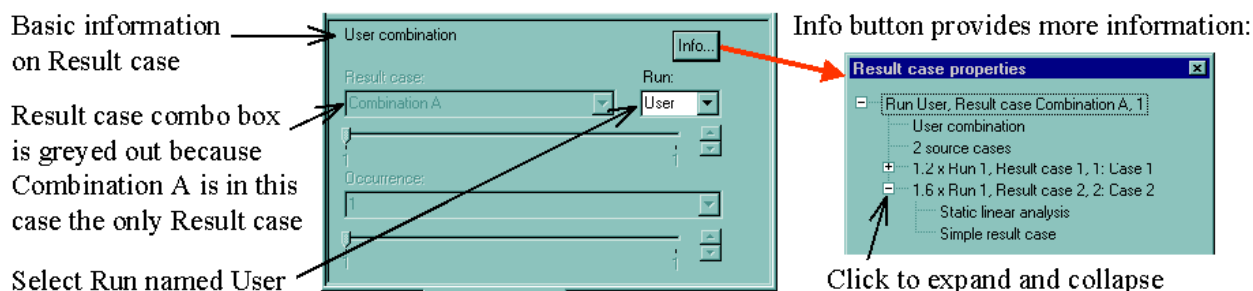


Figure 3.24 Combinations are found in a Run named User

Delete a combination by the line-mode command:

```
result deletecase name-of-case
```

Note: Do not delete a combination that is referred to by another combination or scan result case.

Note: Deformed model is unavailable for a combination not containing displacements. Use 'combine add *combination* nodes displacement' and reselect the combination to enable deformed model.

Note: For a superelement model a combination may be created for any superelement at any level. The combination will, however, only be available for the chosen superelement and lower level superelements contained in it.

Square Root of Sum-of-Squares Combination for Elements/G-STRESS

A specialized feature is available for combination of Elements/G-STRESS components: As an alternative to a normal linear combination a square root of sum-of-squares type of combination may be performed. See Section 3.15.2 for further explanation of this.

Change Normal sum in the Combine dialog to Square root of sum-of-squares to perform this type of combination.

Note: Square root of sum-of-squares combination is available only for the linear components (i.e. not von Mises) of Elements/G-STRESS and Elements/D-STRESS.

Note: The square root of sum-of-squares combination cannot be used when one of the resultcases is complex (see Section 3.4.2).

3.1.18 Scanning Result Cases

Scan selected (and scaled) result cases for the maximum or minimum (absolute) values of a selected result position/attribute/component by:

Results > Scan (or click the tool button )

The Scan dialog, Figure 3.25, appears in which Source result cases are transferred to the Selected source result cases list box. Select result cases in the Selected source result cases list box and adjust the Scaling factor (default 1). When determining the Scan criterion note that:

- (True) maximum(3,-5) = 3
- Absolute maximum(3,-5) = -5 (the value, positive or negative, when the absolute value is maximum)
- (True) minimum(3,-5) = -5
- Absolute minimum(3,-5) = 3 (the value, positive or negative, when the absolute value is minimum)

Give the scan result case a name and click Apply.

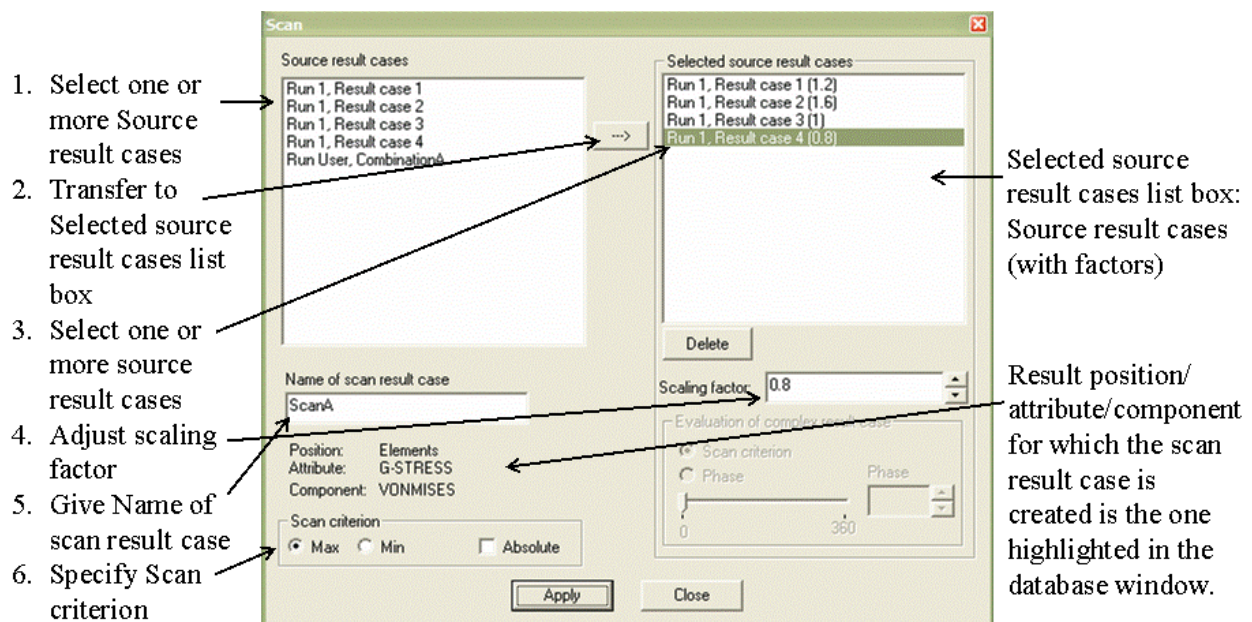


Figure 3.25 Scanning result cases

Note: You may select and transfer two or more Source result cases at the time.

Note: You may select and change Scaling factor for two or more Selected source result cases at the time.

Note: The scanning is performed for the result position/attribute/component highlighted in the database window and listed to the lower left in the Scan dialog. Change result position/attribute/component by selecting (single-click to highlight) the appropriate one in the database window and return to the Scan dialog. The new result position/attribute/component will be listed in the Scan dialog.

Note: Even though the scanning is performed for a specific component the scan result case will contain all components of the attribute in question. I.e. if you scan for Elements/G-STRESS/VON-

MISES then the scan result case will also contain SIGXX, SIGYY and TAUXY (assuming four node element). These other components are normally of little interest though.

Note: The scan result case created will at any point in the model contain the (absolute) maximum/minimum of the selected component over all selected (and scaled) result cases.

Note: Deformed model for a scan result case is normally unavailable and meaningless.

Scan result cases are put in a Run named User (see Section 3.3.1 for an explanation of the Run numbers). To select the scan result case you therefore need to select the Run named User, thereafter select the relevant Result case (i.e. the scan result case) and finally expand to find and double-click the scanned component.

In addition to the attribute with components resulting from the scanning, the scan result case will contain an attribute named MISC which in turn contains the component SCANINDEX. Selecting (double-clicking) this component gives labels (text strings) for all scanned points that identify the result cases that 'won' the scanning. By default this label will include the full result case identification which typically may look like:

- For static (non-complex) results: Run 1, Result case 2, 2 Case 2
- For complex results: Run 2, Result case 1, 3 Excit. freq. 5 Hz, phase 170

The text string of the SCANINDEX label may be modified through

Display > Labeling

For static results you may want to uncheck the Run option, change the Result case prefix to 'Result case' (i.e. remove the comma in front since the Run number has been removed) and set the Occurrence option to 'None'. For complex results you may want to uncheck the Run and Result case options, set the Occurrence option to 'Show all' and change the Occurrence prefix to for example 'Occ'. (See Section 3.4.3 about scanning complex results.) If you also check 'Show SCANINDEX together with result label' the SCANINDEX will be appended to the numeric display of results as shown in Figure 3.26. There are in addition to the default setting two preset versions of the SCANINDEX string:

- A 'minimum string' for static results, click the 'Min. static results' button
- A 'minimum string' for complex results, click the 'Min. complex results' button

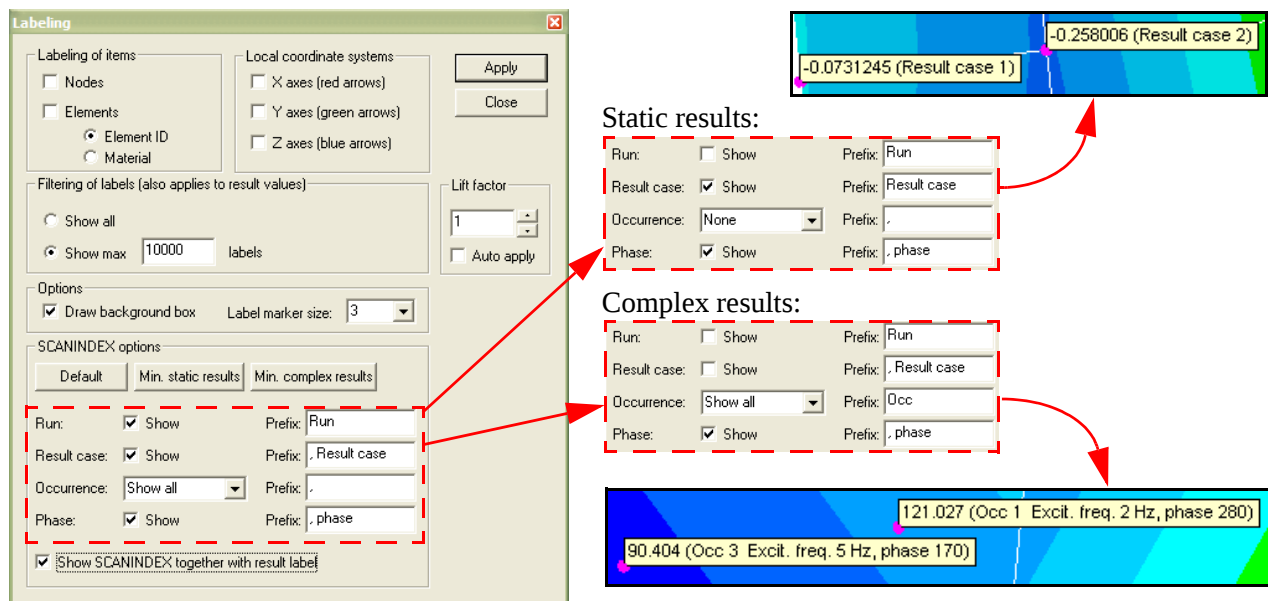


Figure 3.26 Showing adjusted SCANINDEX (result case identification) together with scan result

Click the Info button to get information on a scan result case. Figure 3.27 shows a rather elaborate example of a scan over 24 source result cases, each being a combination a few static and a single complex result case.

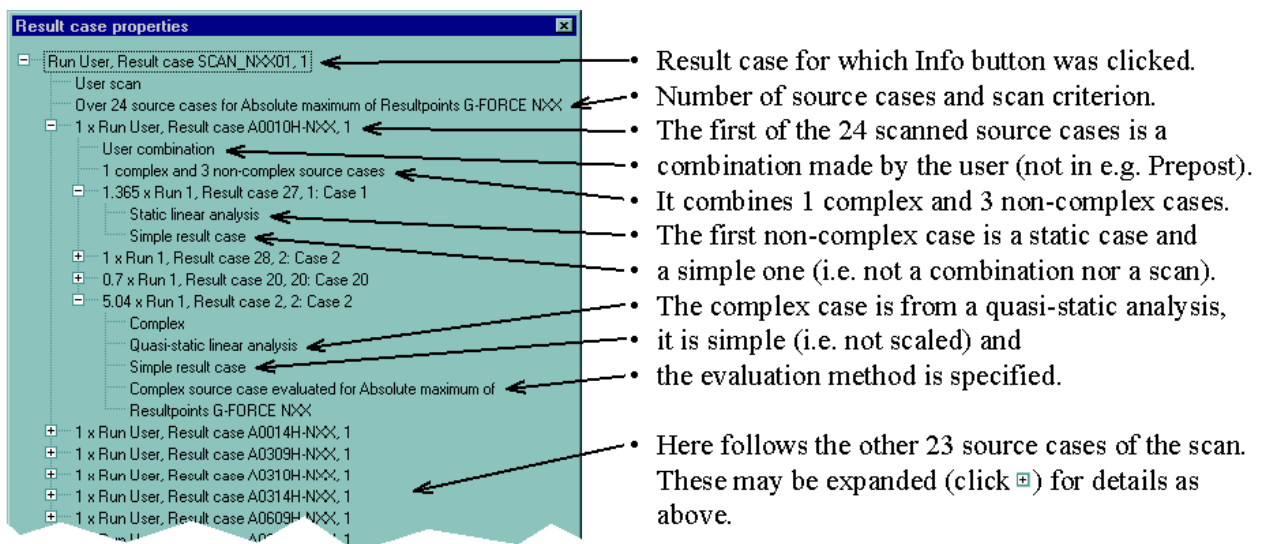


Figure 3.27 Result case properties obtained by clicking Info button

Delete a scan result case by the line-mode command:

result deletecase *name-of-case*

Note: Do not delete a scan result case that is referred to by another combination or scan result case as this may lead to inconsistency.

Note: For a superelement model a scan result case may be created for any superelement at any level. The scan result case will, however, only be available for the chosen superelement and lower level superelements contained in it.

3.1.19 Transforming Stresses

Shell/plate and membrane stresses refer to local element coordinate systems. See Appendix B 2 for more information on this.

You may transform the stresses to a new local element coordinate system by:

Results > Settings (or click the tool button ) > Transformations

The transformation is limited to a rotation about the surface normal, i.e. the local z-axis remains the same. The transformation may be done in two alternative ways:

- A rotation about the surface normal in terms of degrees
- The projection of a vector onto the surface defines either the new local x-axis or the new local y-axis. The vector may be given in the coordinate system of the top level superelement or the 1st level superelement.

Figure 3.28 shows Elements G-STRESS components for the bottom plate (set named DECK) before stress transformation, i.e. in the local element coordinate system of the analysis program Sestra. Viewport 3 (lower left) shows the local x- and y-axes. See Section 3.1.23 about adding local element coordinate systems to the display.

Figure 3.29 shows the same stress components after a transformation, a 30 degree rotation about the surface normal (local z-axis). The rotation is seen in viewport 3. Notice that the SIGXX and SIGYY components change while the VONMISES component, as expected, does not change. Such a transformation will not have any effect on the principal stresses either.

The transformation is performed for the current set only (if any) and is valid for all stress components in all positions and for all relevant attributes.

Note: The transformation of stresses is only performed when presenting results. The underlying data are not transformed. Scanning and combining result cases will thus always be performed based on the untransformed stresses. This means that you cannot first transform stresses and then scan for the highest stress in a rotated direction.

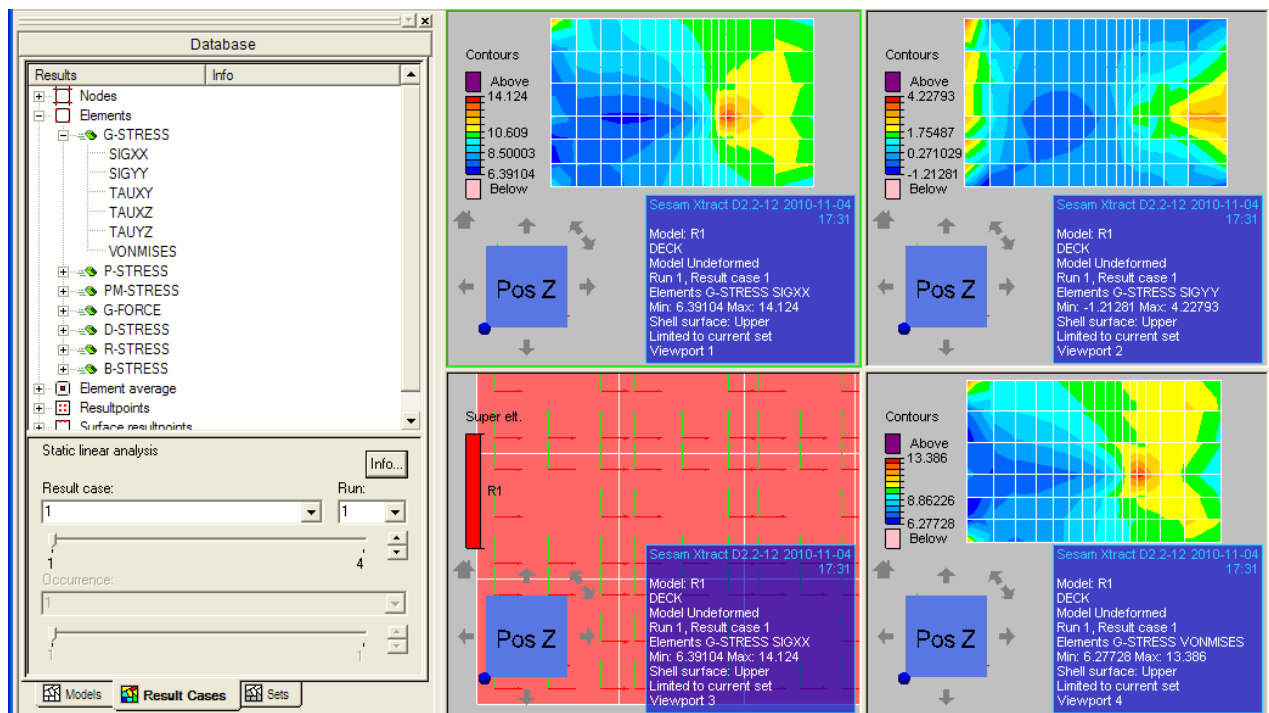


Figure 3.28 Stresses before transformation

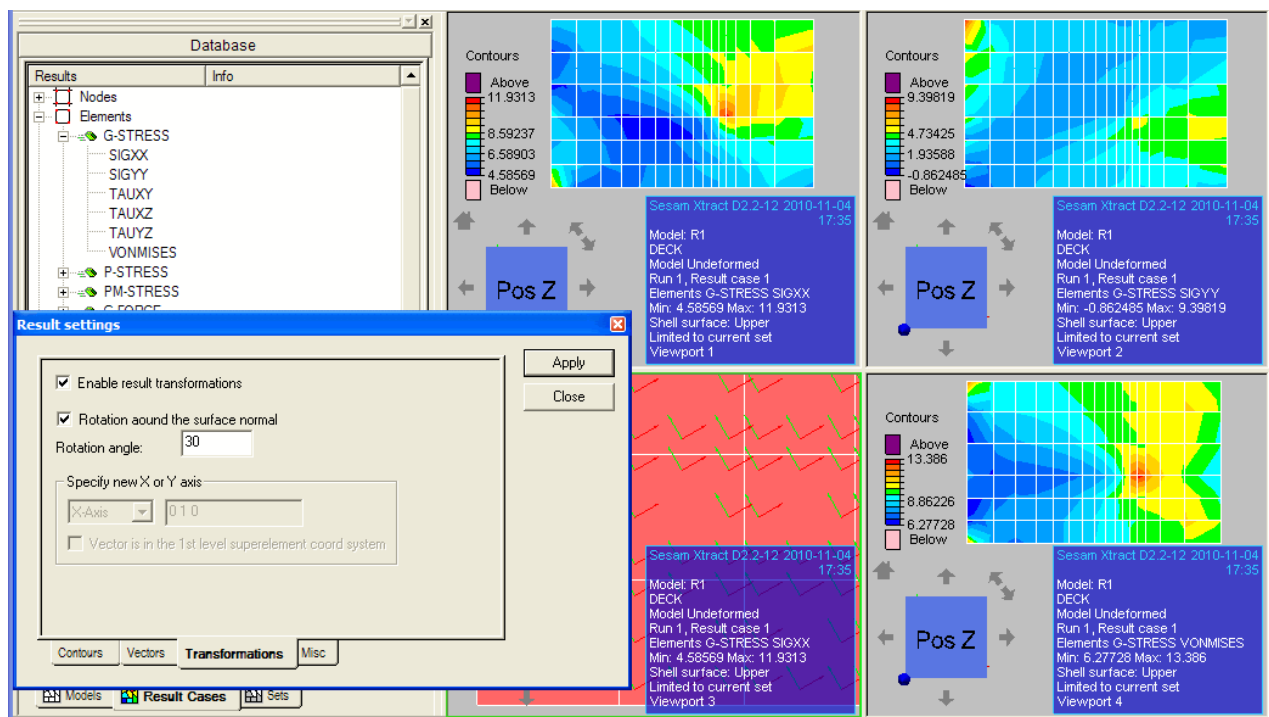


Figure 3.29 Stresses after transformation

3.1.20 Setting Contours

When presenting results Xtract will by default display 10 equidistant contours (levels) between the minimum and maximum values using filled contours. This setting may be modified through:

Results > Settings (or click the tool button ) , clicking the Contours tab and selecting among four contouring modes, see Figure 3.30

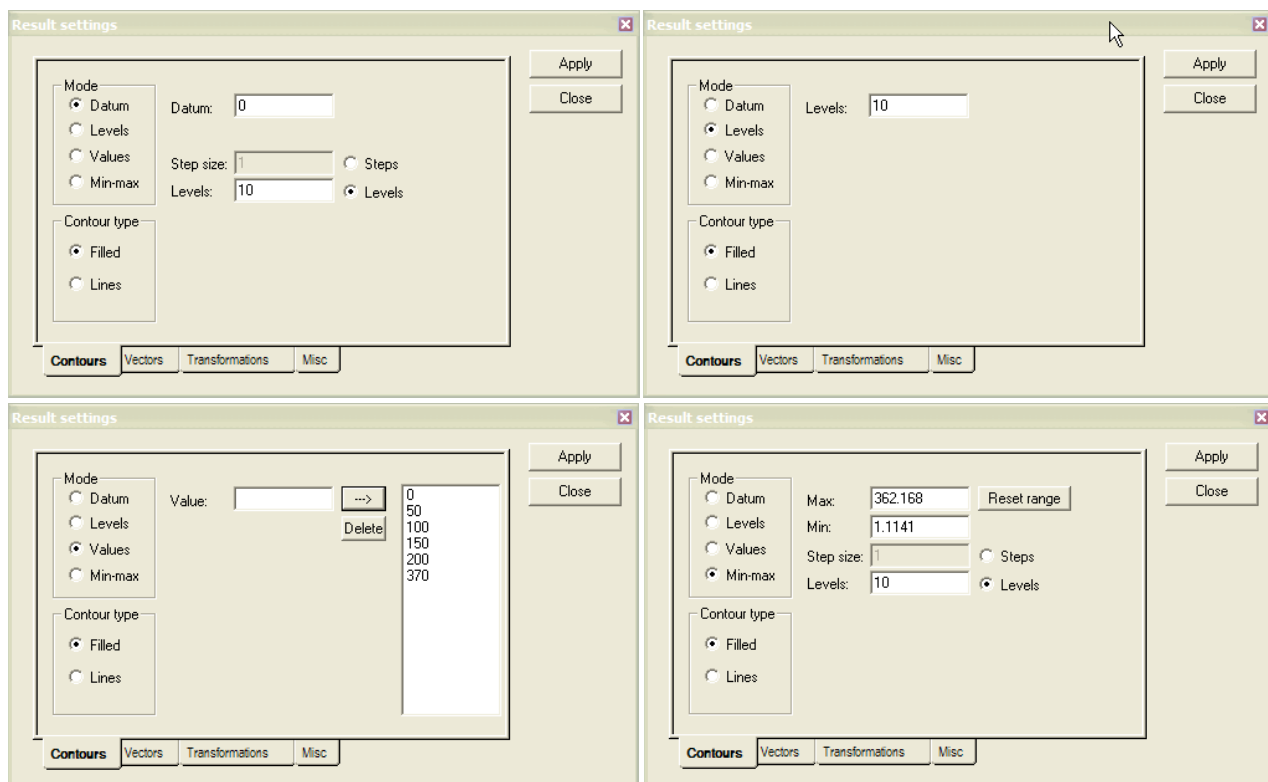


Figure 3.30 Result setting – Contours

The four contouring modes are:

- Datum mode for setting a certain contour value (datum) and either a step value or a number of levels below and above this datum.
- Levels mode for setting number of equidistant levels.
- Values mode for setting all levels explicitly. Areas with values above the maximum are violet and below the minimum are pink.
- Min-max mode for setting minimum and maximum values and in between these either a step value or a number of levels. Areas with values above the maximum are violet and below the minimum are pink. If result values are shown then only labels for values within the min-max range will appear.

In addition to the contouring modes you may select Contour type Lines in addition to the default type Filled. Lines may be the preferred alternative for creating black & white plots, see Section 3.1.26. The colour lines will in black & white mode change to black lines annotated by characters.

Figure 3.31 shows an example of use Values mode. Notice that the Database dockable window has been torn loose to increase the display area. Also notice that perspective view has been switched off by Display > Perspective view (or clicking the tool button ).

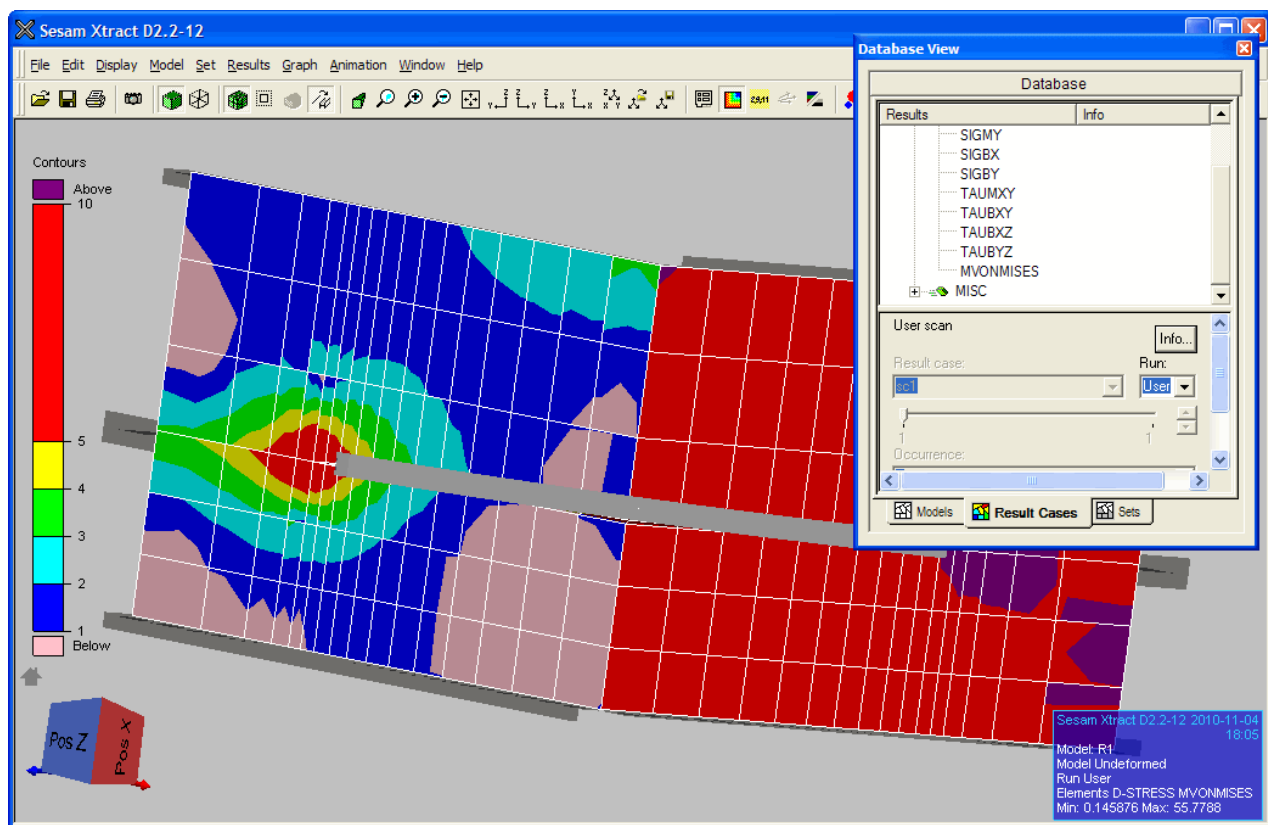


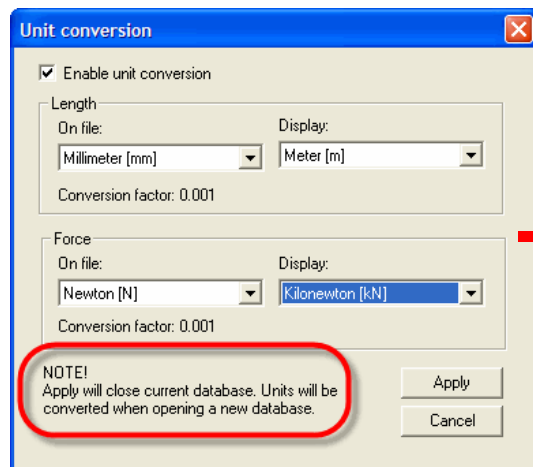
Figure 3.31 Setting contours and undocking database window

3.1.21 Unit Conversion

Convert from one set of length and force units to another through

Model > Unit conversion

The dialog shown to the left in Figure 3.32 appears. Specify the current (On file) and desired (Display) units for Length and Force and click Apply. (The current units must be specified since the results file does not contain such information.) The database then automatically closes and you must reopen the results file and reselect the model as well as the result case and component. You basically need to restart your session. The reason for this procedure is that all data are converted from the current to the desired units at the opening stage of the results file. This ensures that all data are properly converted. When presenting results after the conversion the new units are shown in the Info box as shown to the right in Figure 3.32.



Clicking Apply in the Unit conversion dialog causes closing of the database and you must reopen the results file and reselect the model. Note that the Info box in the lower right corner now specifies the new units.

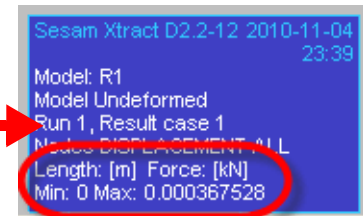


Figure 3.32 Unit conversion

Note: The unit of mass is a consequence of the length and force units and is therefore not given.

Note: The unit of rotational displacements is always radians and cannot be converted.

Note: Unit conversion cannot be done for results fetched from a VTF-file, i.e. results from programs like Wasim, Installjac, Riflex and Stofat.

3.1.22 Sets

Sets stored on the results file (i.e. defined in the preprocessor) may be displayed in Xtract. New sets may also be defined.

Note: A set may only contain elements. I.e. a set cannot contain nodes.

Click the Sets tab of the Database window and find all sets listed. Double-click a set to display it, see Figure 3.33. Alternatively, merely click a set and hit Return. Notice that selecting a set lifts the tool button for Set > Show both () and depresses the tool button for Set > Show working set (). Give Set > Show complement (or click the tool button ) to display the complement of the current set.

Notice that a check mark indicates the current set in the browser.

When selecting a set (or its complement or both) the currently selected result component is automatically reloaded so that the results range (min and max values) is limited to the displayed part of the model.

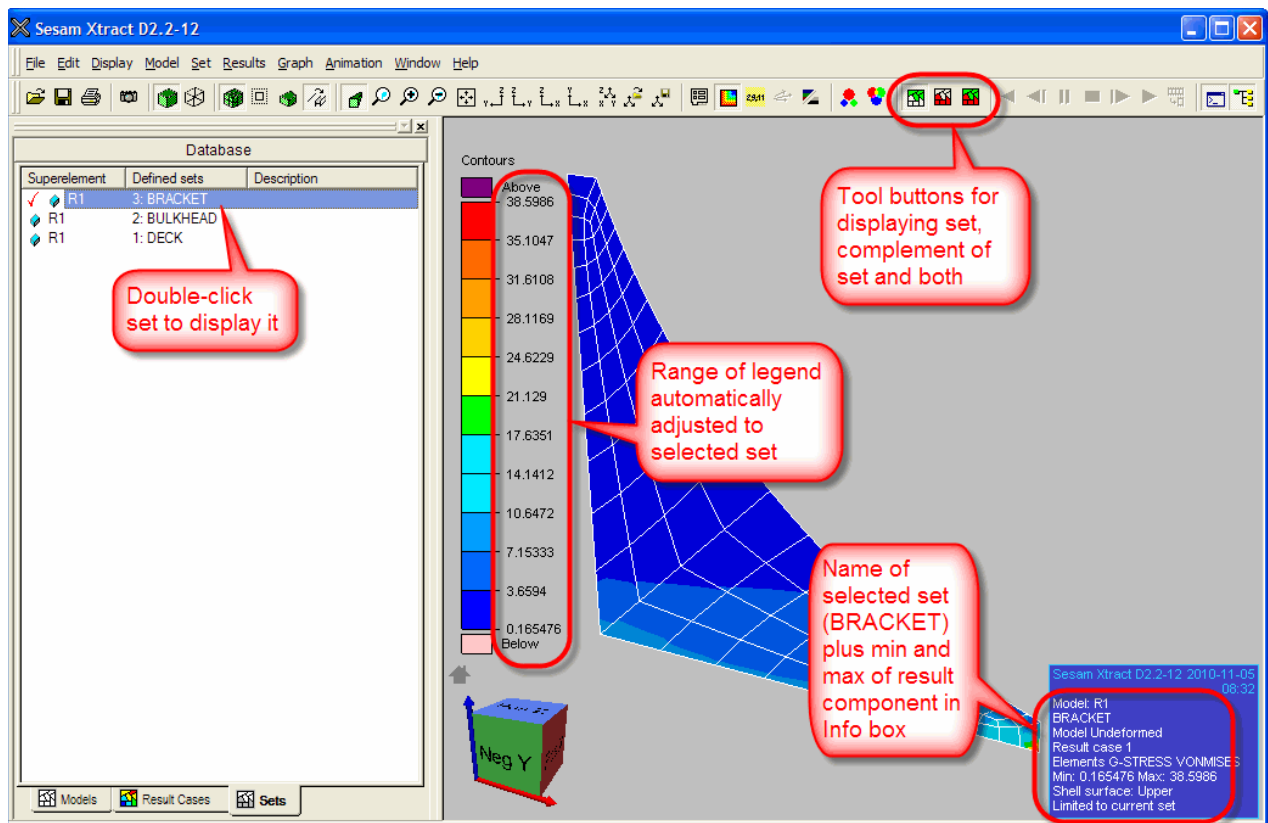


Figure 3.33 Double-click a set to display it – or click and hit Return

Select two or more sets by clicking one set and Ctrl+clicking and/or Shift+clicking more sets and concluding the selection by hitting Return. Figure 3.34 shows an example for a superelement model. Notice that the sets are sorted alphabetically superelement by superelement.

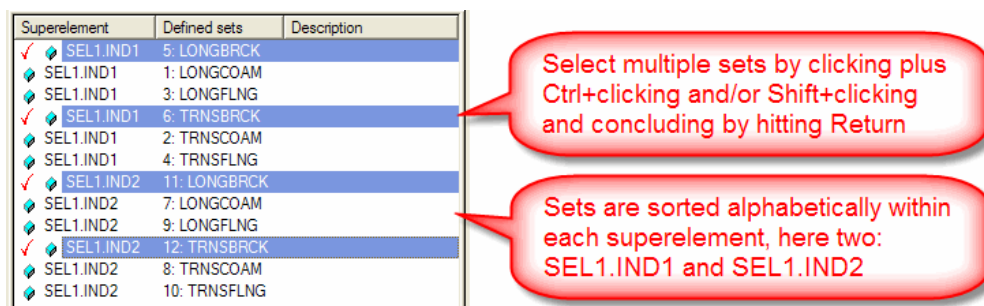


Figure 3.34 Select multiple sets by clicking and hitting Return – sets are sorted alphabetically

How to define a new set, modify it and finally revert to the complete model is done as follows:

- Set > New

The new set appears in the list of sets with a default name.

- Set > Name (or right-click the new set)
In the Name and description of set dialog enter the desired information and click OK.
- Give the set content (fill it with elements) by Set > Union (or right-click the set) and select an alternative:
 - Existing set – set stored on the results file or previously defined in Xtract
 - Element type – elements of a given element type
 - Material type – elements having a given material type
 - Elements – single element number or range of element numbers
 - Beam cross section – beam elements having a given cross section
 - Model – all elements of a given superelement
 - Shell thickness – plate/shell elements with thickness within a specified range
 - Coordinate ranges – elements within specified coordinate ranges
 - Result values – elements containing result values within a specified range
 - Top/Bottom/Right/Left – use the left mouse button to drag a line across the model and all elements completely above/below/to the right of/to the left of the line are put into the set
 - Rubberband – use the left mouse button to drag a rubberband (rectangle) over the model, all elements completely inside the rubberband are put into the set
 - Polygon – use the left mouse button to make a polygon, close it by either clicking the first point or hitting the space key (the Delete key removes the last point while right-click and Esc aborts the operation), all elements completely inside the polygon are put into the set

The upper part of Figure 3.35 illustrates how the rubberband option is used to create a set containing elements in the vicinity of the upper part of the bracket.

- Modify the content of the set by:
 - Set > Subtract by
 - Set > Intersect with

The lower part of Figure 3.35 is the result of the following action: Set > Subtract by > Element type. In the Select list item dialog appearing you may select item BTSS (23) (the name of the 3 node beam element) to subtract beam elements leaving only 8 node shell elements (SCQS (28)) in the set.

- You may also add and remove single elements:
 - Set > Pick to add – the complement is shown allowing you to click elements to add to the set, when done leave the picking mode by giving Set > Pick to add once more and see the set by clicking 
 - Set > Pick to remove – click elements to remove from the set, when done leave the picking mode by giving Set > Pick to remove once more
- You may modify the way the graphical means for modifying a set (options Top, Bottom, Right, Left, Rubberband and Polygon) works by:
 - Set > Select outside/opposite – this allows you to select elements outside an area defined by Rubberband and Polygon (it is less relevant for Top, Bottom, Right, Left as it merely exchanges the functionalities of Top and Bottom and likewise for Right and Left)

— Set > Include touched by – in addition to elements completely above/below/to the right of/to the left of a line and completely inside a rubberband/polygon also elements touched by the line/rubberband/polygon are selected

The ‘Select outside/opposite’ and ‘Include touched by’ are options that are switched on and off as needed. These two options may also be used in combination.

- Having created a set you may add it to, subtract it from and intersect it with a previously defined set by:
 - Set > Add elements shown to existing set – the current set is added to another set, i.e. the other set is modified
 - Set > Subtract elements shown from existing set – the current set is subtracted from another set, i.e. the other set is modified
 - Set > Intersect elements shown with existing set – the current set is intersected with another set, i.e. the other set is modified
- See the position of the current set in the whole model by:
 - Set > Show rest of model as wire-frame – this adds the mesh of the whole model in dashed wireframe mode
- Return to displaying the complete model by either of the following:
 - Set > Show both – the set is still active but both the set and its complement are shown
 - Set > Cancel – all sets are made inactive involving that the whole model is shown

You may list the elements contained in a set or the nodes of these elements by Model > List model data.

Use Set > Delete or right-click a set to delete it.

Note: When viewing a set the nodal averaging of stresses (all stresses in position Nodes) will not account for elements not included in the set. Nodal stresses on the edge of a set may therefore differ from the values presented for the whole model (or another set). See Figure 3.36.

Note: When using graphical means for modifying a set (options Top, Bottom, Right, Left, Rubberband, Polygon and picking one at a time by clicking) then avoid using a deformed shape of the model since the graphical selection operates on the undeformed model display.

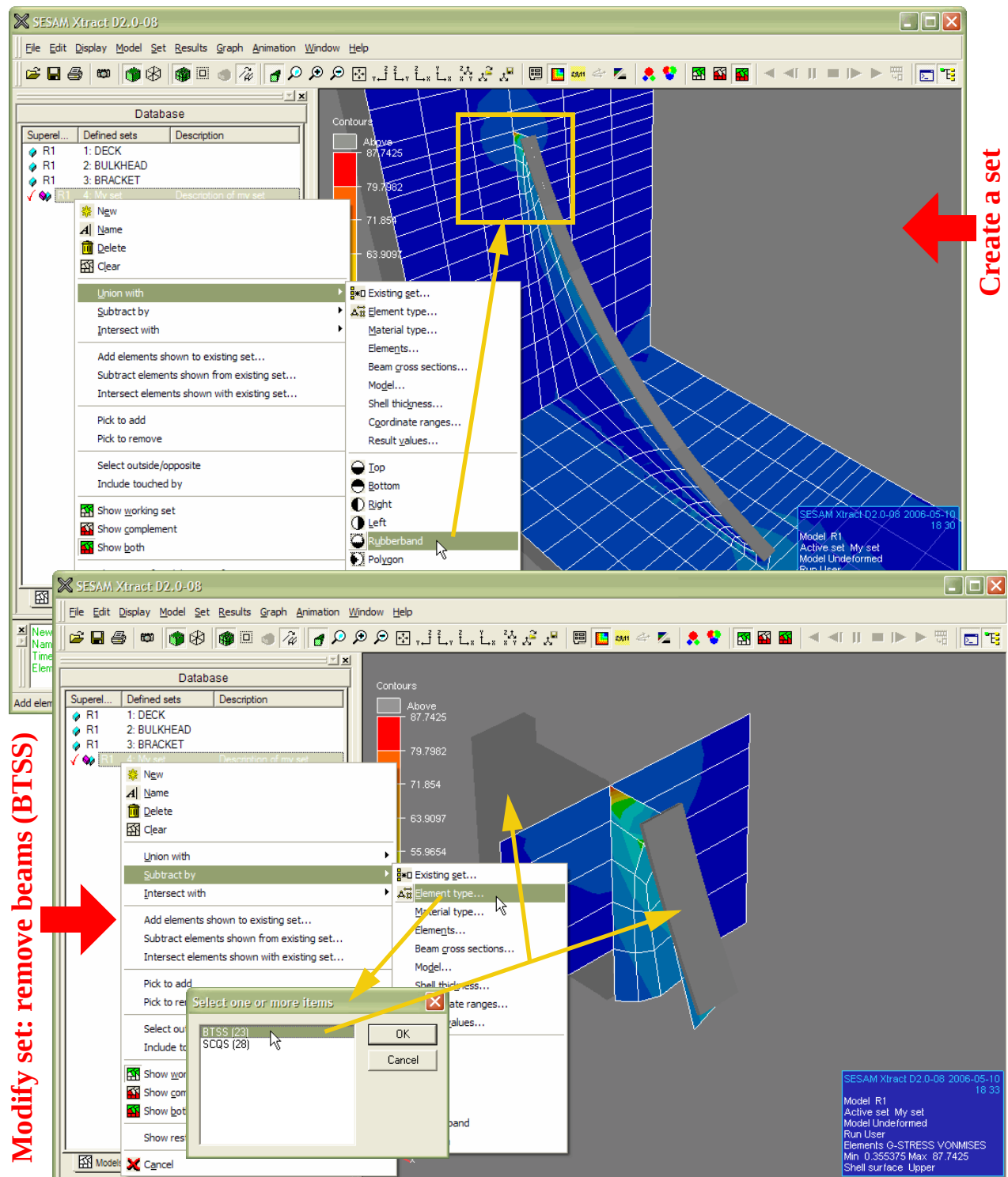


Figure 3.35 Create (Union with) and modify (Subtract by) a set

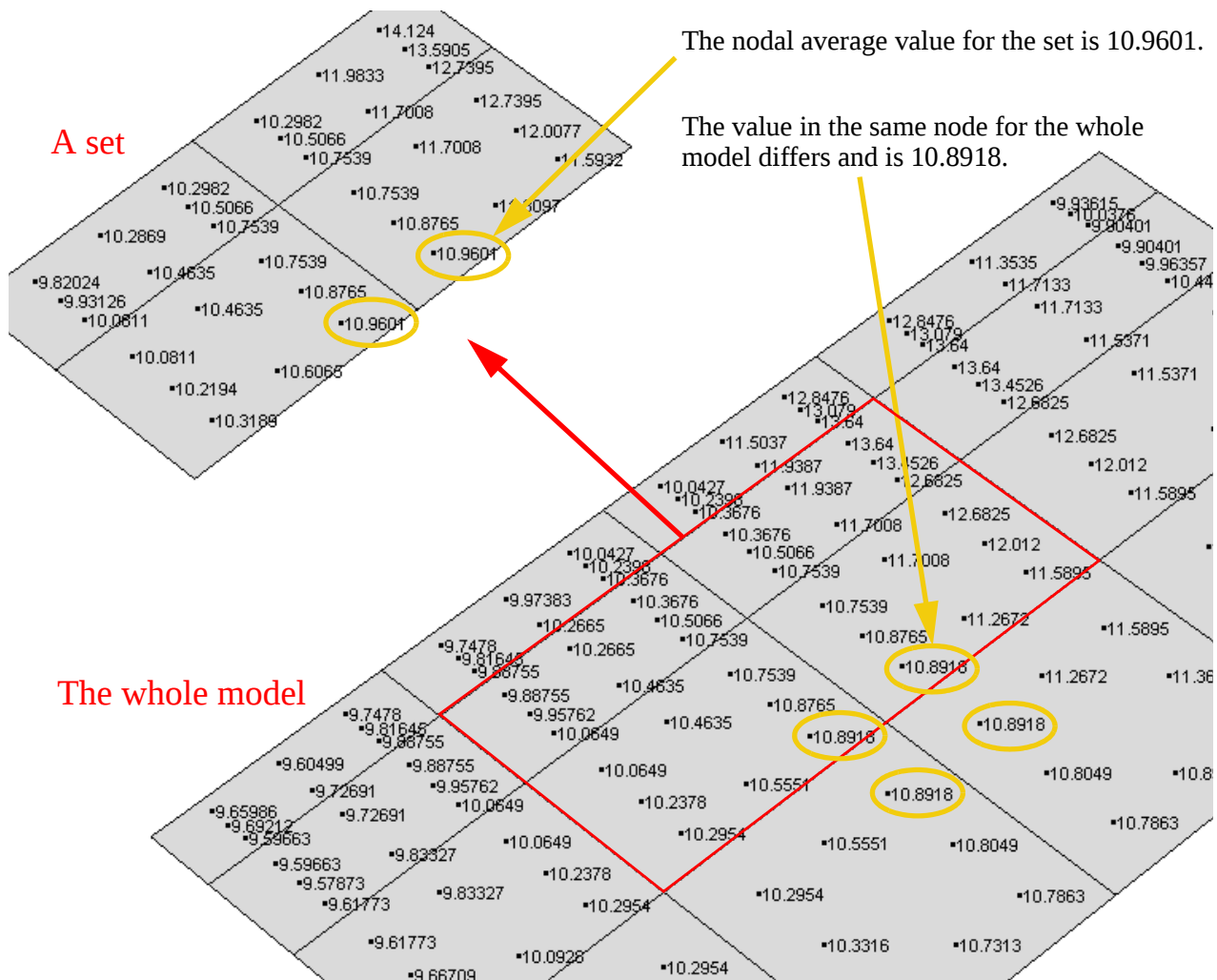


Figure 3.36 Nodal stresses on the edge of a set may differ from those of the whole model

3.1.23 Multiple Viewports

The display area may be split into multiple viewports. The alternatives are two vertical, two horizontal and four viewports. These viewports function as independent areas for displaying the model with results.

Split into four viewports by:

Display > Viewports > Four (or Ctrl+4)

The current display will be put into the upper left viewport. Click another viewport to make it active. The active window is framed by a green line (the lower right in Figure 3.37). Copy from a selected viewport to the active one by:

Display > Copy visual attributes > From viewport *i* – this copies result case, component, display settings and result settings, i.e. everything except the model, from viewport *i*

Display > Copy all > From viewport *i* – this copies also the model, i.e. everything, from viewport *i*

You may grab the corner between four viewports or the line between two viewports and adjust their sizes. Figure 3.37 shows an example of four viewports. Notice that the viewport number is given in the Info box.

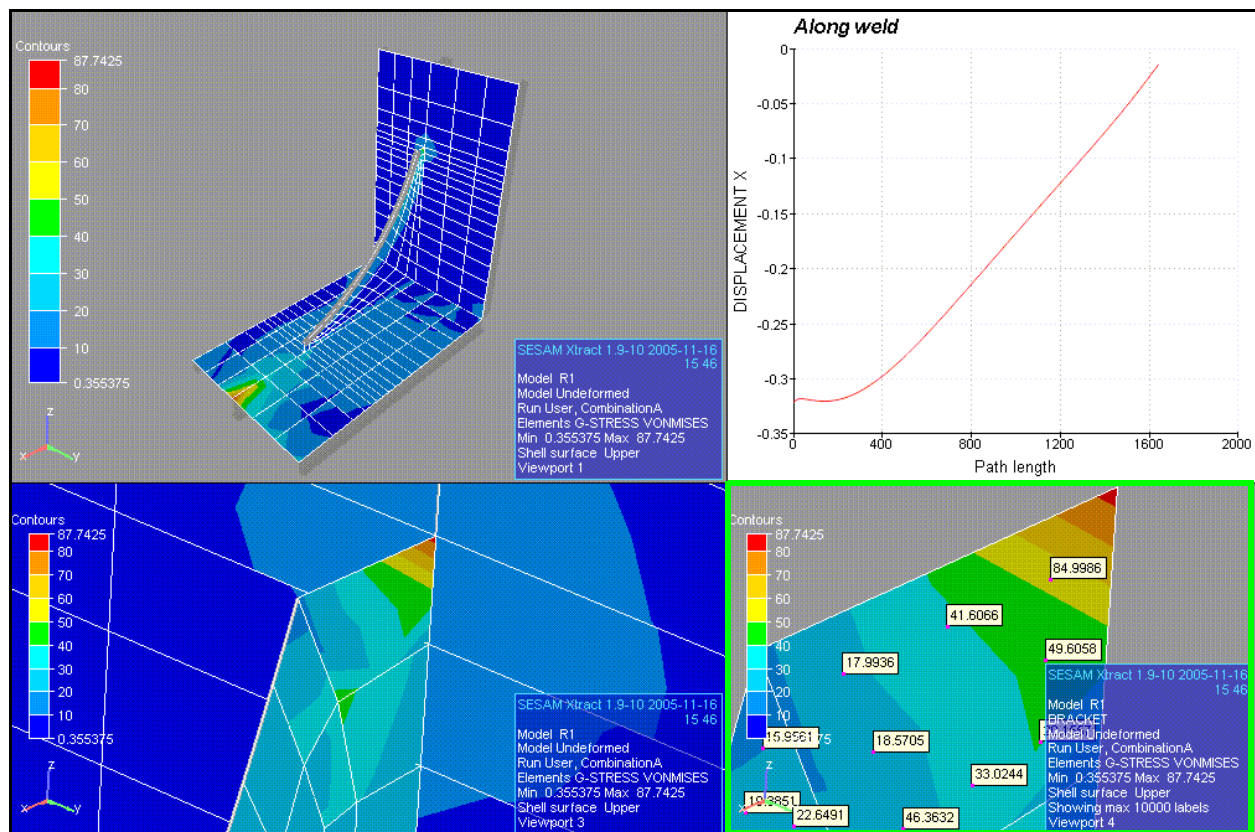


Figure 3.37 Multiple viewports

Note: The dialog for saving the graphics (File > Plot) has an option 'Plot all visible views'. When not checked then only the active viewport is saved.

Note: 'Edit > Copy view to clipboard' saves the active viewport to the clipboard while 'Edit > Copy visible views to clipboard' saves all viewports to the clipboard.

When displaying the same model/set then navigate (rotate/zoom) synchronously all 2/4 viewports by:

Display > Synchronous navigation

Return to a single viewport by (the active one of the four viewports will be shown):

Display > Viewports > One (or Ctrl+1)

When returning from four to two viewports (Ctrl+2 or Ctrl+3) then the active and previously active viewports will be shown.

3.1.24 Adding Text and Pointers

Add text and pointers to the display (for example in connection with producing plots for a report) by:

Display > Draw toolbox

In the Draw dialog there are tools for adding, moving and deleting text and pointers. See Figure 3.38. Positioning, moving and deleting texts and pointers are done by clicking at the start of the item.

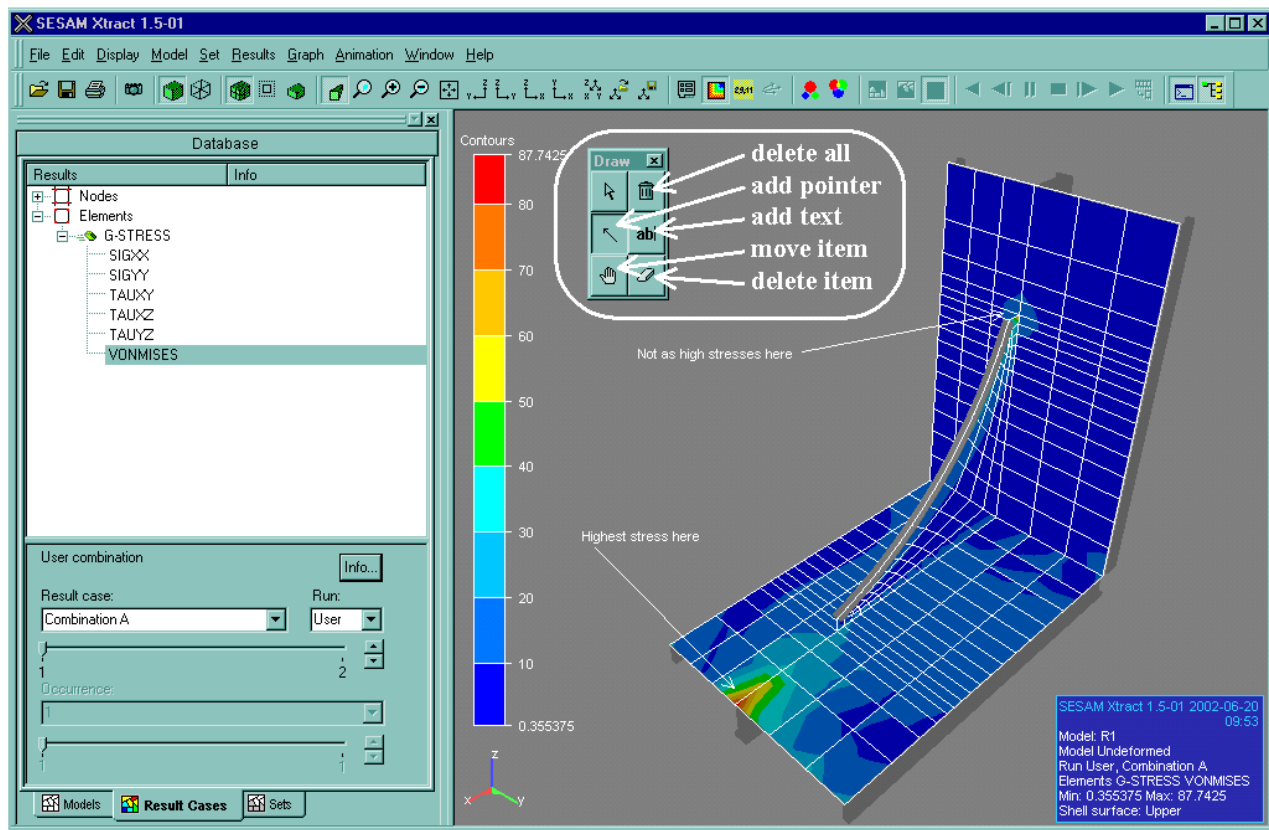


Figure 3.38 Adding text and pointers

Note: If you have added text and pointers as shown in Figure 3.38 and when plotting you set the plot size (see Section 3.1.26) then be aware of that the text strings and pointers may shift somewhat. A pointer may then not be positioned exactly as intended.

3.1.25 Display Settings, Labelling, Part Attributes and Information

By:

Display > Settings (see Display settings dialog in Figure 3.39)

you can change the number of significant digits to be used in result presentation, adjust the element shrink factor (Display > Shrunken elements), set the font size for all text in the display, change the colour used for

undeformed wireframe mesh (Model > Add undeformed wireframe) as well as change the background of the viewport. (Click a colour box to change colour.) Check the Top and bottom box to make the background a smooth vertical transition from one colour to another as shown in Figure 3.39.

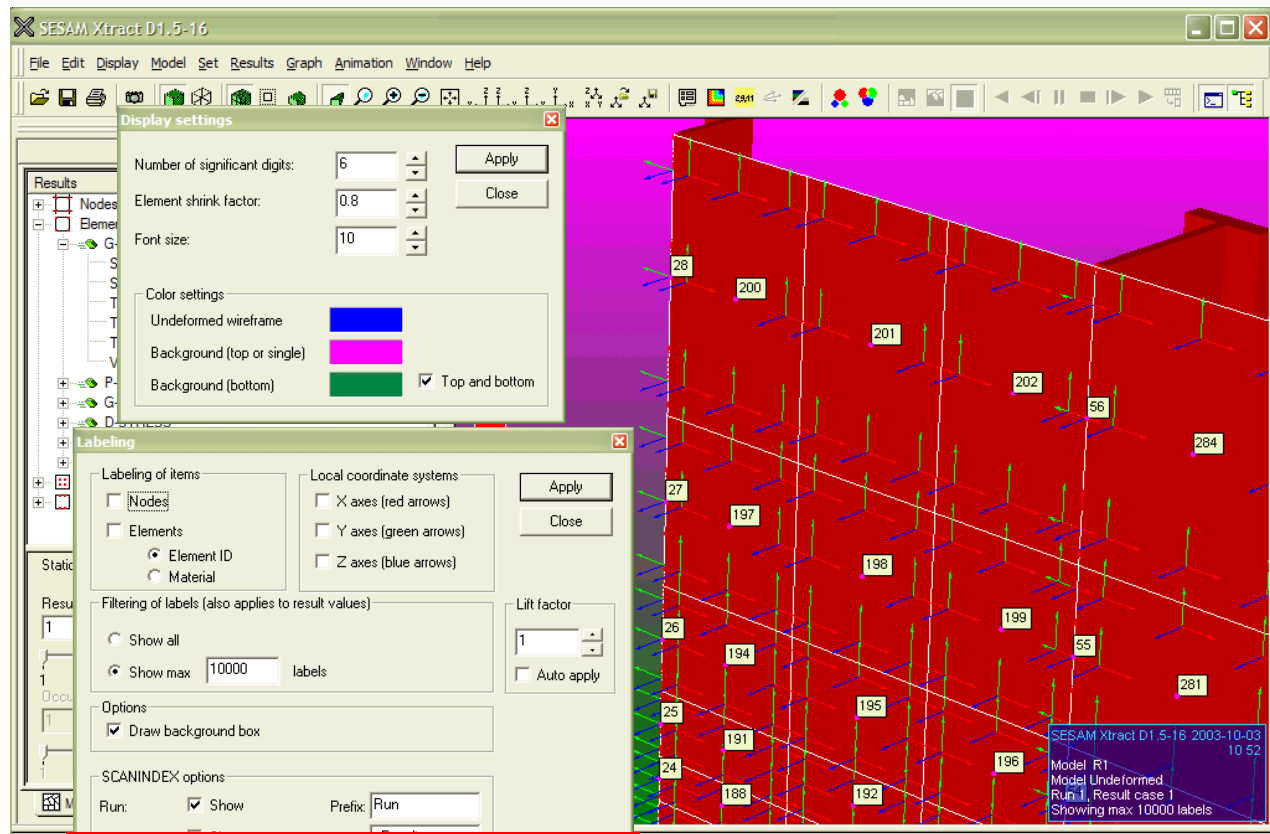


Figure 3.39 Adding local coordinate axes and labels, changing background and more

Note: You don't need to set the background to white for plotting purposes as exporting graphics (saving a plot file) with white background may be chosen in the save image process (see Section 3.1.26).

You can add node and element numbers as well as element local coordinate systems by:

Display > Labeling (see Labeling dialog in Figure 3.39)

A maximum of 10.000 labels (node or element numbers as well as result values) will be shown by default. (This limit which is higher than any practical use is only to avoid Xtract hanging while cluttering the screen with labels for a very big model.) The Labeling dialog allows you to adjust this maximum number or to allow all labels to be shown. Notice that the Info box in the lower right corner tells whether there currently is a limit to the number of labels shown.

The Draw background box option allows switching on/off the light background box of the numeric values. See Figure 3.7.

Note: Labels (result values, principal stress vectors, etc.) are lifted towards the eye in order not to be hidden by the model itself. But in certain cases labels may still be partly hidden. Labels may then be lifted more by the Lift factor in the Labeling dialog.

Note: When beams are shown expanded (showing cross sections) labels along the beams will easily be hidden inside the beams. Apart from switching off the expansion (see below) you can make the labels visible by lifting them more. Check the Auto apply for Lift factor and use the spin button to increase the factor until all labels are visible.

Note: Some of the settings of the Display settings and Labeling dialogs are stored in the user registry. This involves that these settings are stored when exiting Xtract and they take effect when restarting the program with the same user and PC. Briefly, these are: number of significant digits, element shrink factor, font size, background colour, label background box shown or not and SCANINDEX options. See more detailed information in Section 4.1.3.

In addition to the above, nodes and elements may be labelled individually by graphic means. The element and node numbers are accompanied by result values depending on the setting of 'Results > Show result values in labels'. See Section 3.1.12 about showing result values for selected elements and nodes.

Shift+Ctrl+LMB for the first click and LMB thereafter toggles labels for element numbers

Ctrl+LMB for the first click and LMB thereafter toggles labels for node numbers

Note: When toggling node and element numbers you must click *inside* the model. E.g. to label a node on an edge you cannot click outside the model no matter how close to the node you click.

To remove all element/node labels give:

Results > Clear selected elements/nodes

Show shell/plate thicknesses by colours as exemplified in Figure 3.40 by:

Display > Show plate thickness

Show material types by colours by:

Display > Show materials

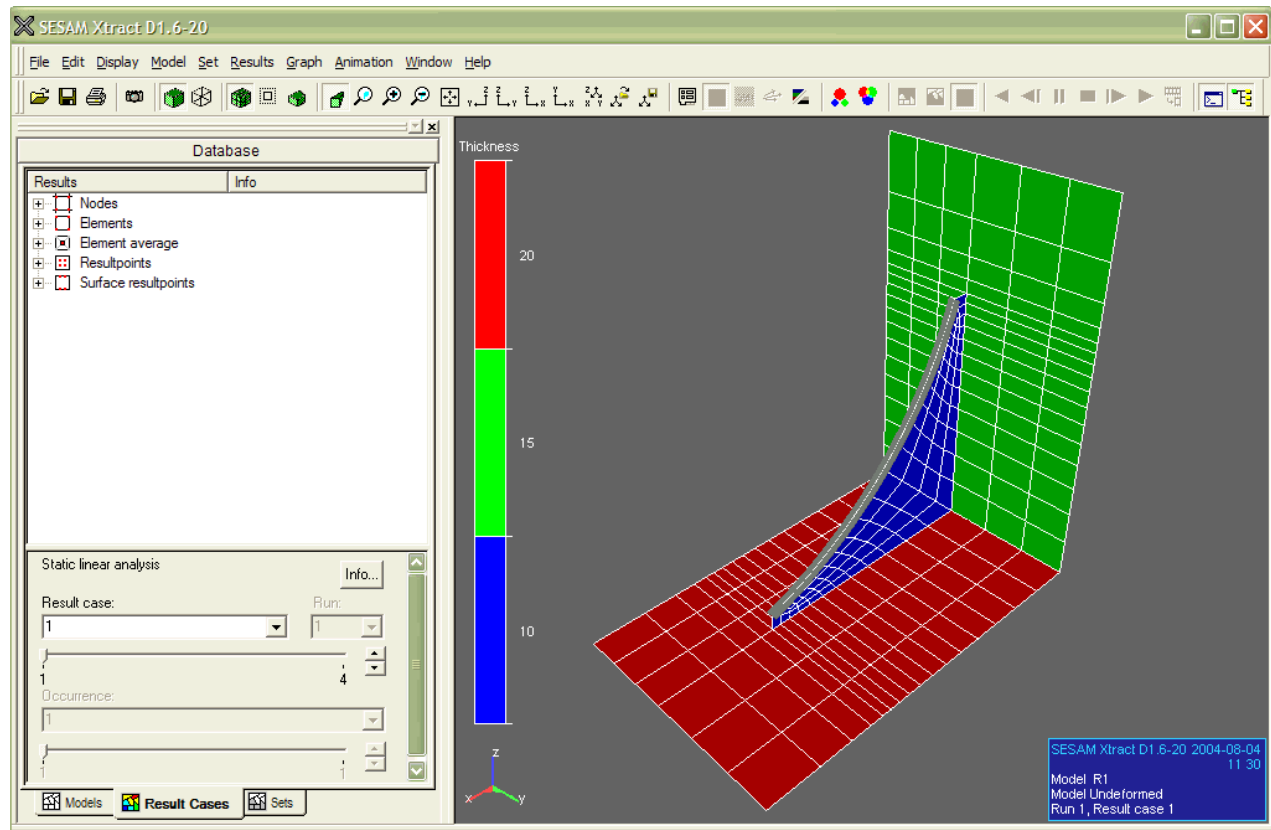


Figure 3.40 Showing plate thicknesses

Beam elements are by default expanded (shown with their cross sections). You can switch off expansion by:

Display > Part attributes – the Expanded option (see Part attributes dialog in Figure 3.41) or for all superelements (parts) by the tool button

In addition to controlling expansion of beam elements the Part attributes dialog controls several other display settings for superelements (parts) individually. For example, the various display alternatives Show as surface, Show mesh, etc. that are set for the whole model by the tool buttons, etc. may be set for individual superelements through the Draw style alternatives of the Part attributes dialog.

Moreover, the following settings are controlled (first select superelements by click, Shift+click and Ctrl+click and then check/uncheck the options):

- Visible – When unchecked the superelement (part) will not be visible.
- Smooth – Shading will be smooth.
- Deformed – Superelement (part) will be shown deformed.
- Contours – The current results component will be contoured.
- Vectors – Present deformation (DISPLACEMENT/ALL) as vectors.

- Mesh – Display the mesh.
- Outline mesh – Outline the mesh.
- Expanded – Present beam elements with their sections and in their eccentric (offset) positions.
- Transparent – Display the superelement (part) transparent. Transparency Alpha value is set in the lower right corner: 0 is fully transparent (invisible), 255 is opaque. The Contours option must be switched off.

An example in which one of three superelements has been made transparent (and Contours and Mesh switched off) is shown in Figure 3.41. (A superelement model is shown instead of the single superelement model of this tutorial to show how display settings are given for superelements individually.)

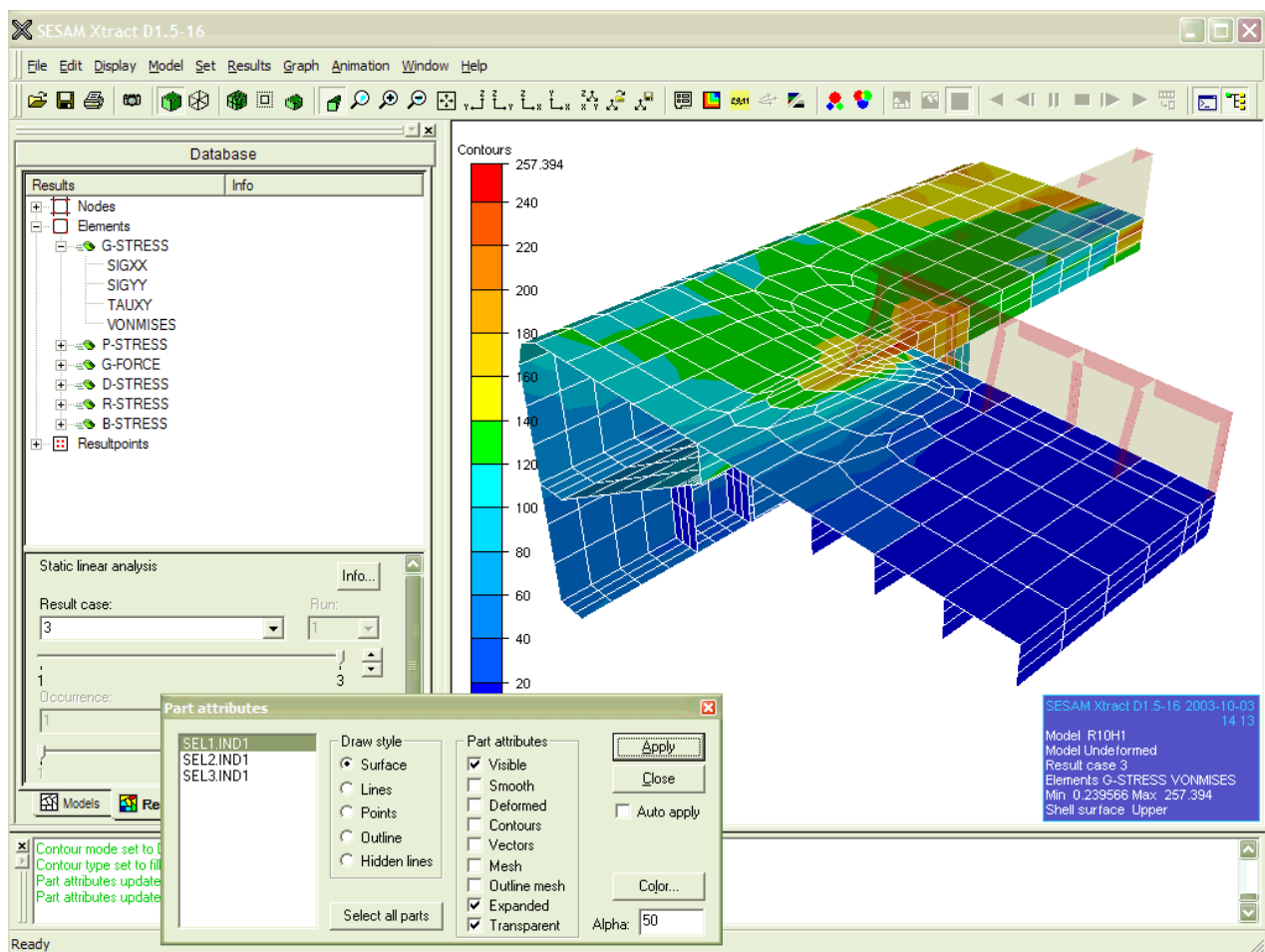


Figure 3.41 Part attributes

The content of the Info box in the lower right corner may be modified and removed altogether. Moreover, the navigation cube in the lower left corner may be replaced by a more traditional model coordinate system, or if both are switched on the traditional model coordinate system will be in the lower left and the navigation cube in the upper right corner. Finally, the coordinate axes drawn in the origin and the legend may be added and removed. These manipulations are available through:

Display > Information

Figure 3.42 shows the options in the Display information dialog and explains all possible information provided by the Info box.

Note: In addition to the Legend check box in the Display information dialog (which is equivalent to the line-mode command 'displayinfo legend') there is a line-mode command 'displayinfo superelementlegend' that switches on and off the legend for the superelement colours. This is relevant when showing the model with no results in which case the superelement colour legend appears by default.

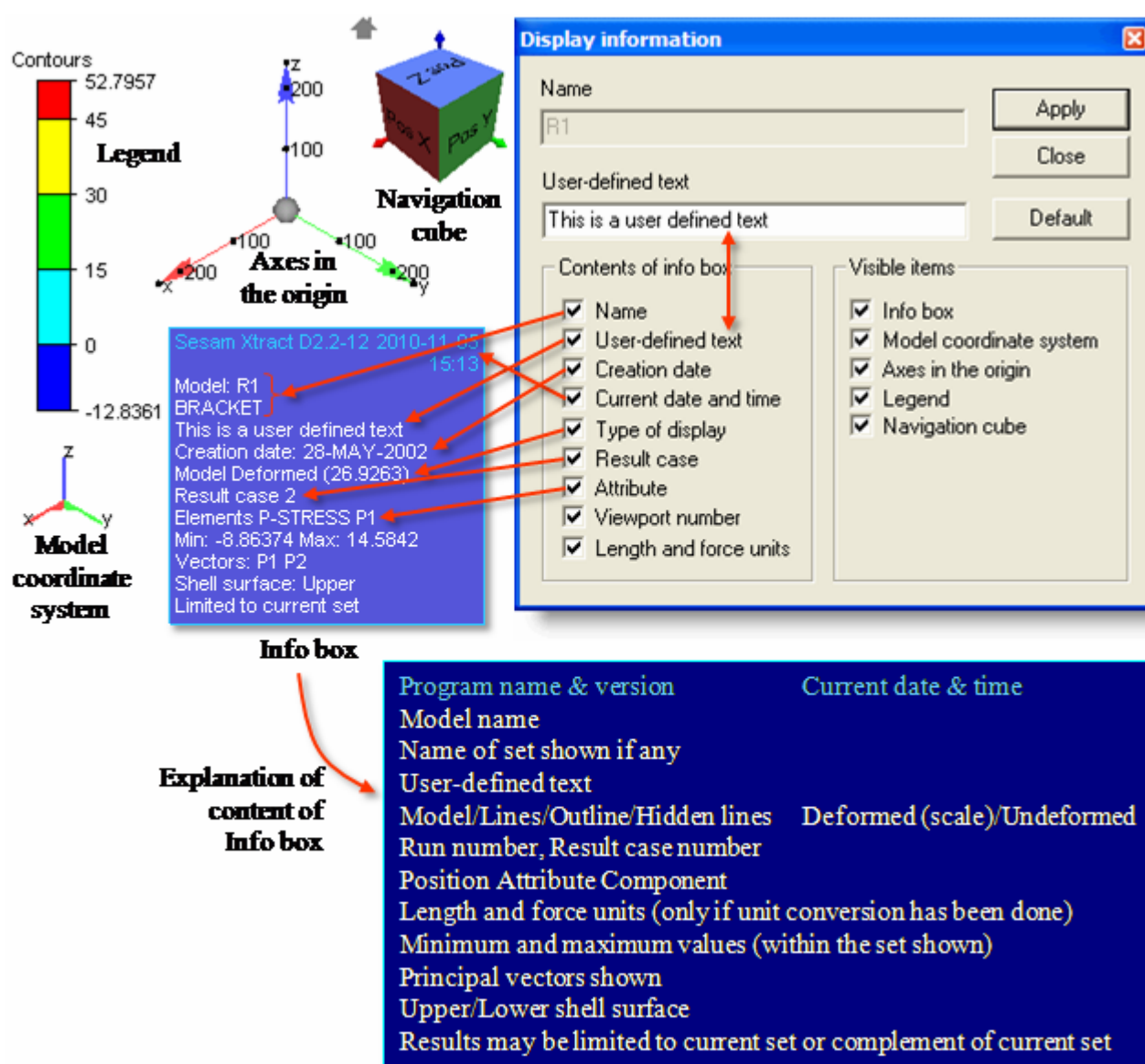


Figure 3.42 Display information

3.1.26 Making Plots/Hard Copies and Adding Graphics to Reports

There are alternative ways of printing/saving graphics created in Xtract:

- File > Print (or click the print tool button  or Ctrl+P) – for printing the active viewport (send it directly to an on-line printer)
- Edit > Copy view to clipboard (or click the camera tool button  or Ctrl+C) – for copying the active viewport to the clipboard for subsequent pasting into a report
- Edit > Copy visible views to clipboard – for copying all viewports to the clipboard
- File > Plot (or click the save tool button  or Ctrl+S) – for saving to file with a chosen graphics format, only the active viewport is saved unless the ‘Plot all visible views’ option is checked, see Figure 3.43

Note: As an alternative to printing/saving graphics for manual inclusion in a report Xtract also offers automated report generation through the program Xtract Report Builder. Read more about this in Section 3.13.3.

When printing directly and copying to clipboard increase the quality (resolution) by maximising the Xtract window and maybe also closing down the Database and Command and message windows; see Figure 3.2.

File > Plot opens the Save image dialog, see Figure 3.43. A choice of graphics formats are available:

- The GIF format which requires limited disk space (typically less than 10 percent of TIFF and BMP) and may be imported into word processors, presentation tools, HTML files (for viewing by web browsers), XML files and other. This is the best format for normal work.
- The TIFF format which represents colours and colour shading better than GIF but requires more disk space (about the same as BMP). It may be imported into word processors but it is not supported by web browsers.
- The Windows BMP format which gives large files (1 - 10 MB depending on the X size and Y size; see below) but after import into Word the document size does not increase correspondingly (Word compresses the file). The quality is somewhat better than GIF as there is no compression involved.
- The EPS format – this format is less suitable as the pixel-based display of Xtract is unsuitable for translation into the vector based Encapsulated PostScript format.
- The PPM format – this Portable Pixmap format is less suitable for normal work.
- The RGB format is less suitable for normal work.

Note: When the slight quality reduction of the compressed GIF-file is unacceptable consider using the Edit > Copy alternative rather than saving a BMP-file for pasting graphics into a report.

The size (resolution) given in pixels of the image saved is by default equal to the display area (which may be increased by maximising the Xtract window and closing down the Database and Command and message windows; see Figure 3.2). Increase the X size (horizontal) and Y size (vertical) to increase the resolution and thereby the quality of the plot. Suitable values for X size and Y size are in the range 500 - 2000, the upper end of the range gives high quality plots.

The Save image dialog includes a field for entering a File name. By default this will be Xtract001 and after saving a plot file (clicking Apply) the name will be incremented to Xtract002, Xtract003, etc. This feature simplifies producing several plot files for reporting purposes. If you enter another name, say Bracket1, the name will be incremented to Bracket2, Bracket3, etc. A name without counter, say Abc, will after clicking Apply be 'incremented' to Abc001. Closing the Save image dialog resets the File name to Xtract001 (next time the dialog is opened).

The Save image dialog includes Header and Footer fields for text strings to be reproduced at the top and bottom of the plot image. The font size of the header and footer is somewhat larger than other text and if the font size is increased (see Section 3.1.25) then the header and footer are increased proportionally. The header and footer may be left, centre and right adjusted.

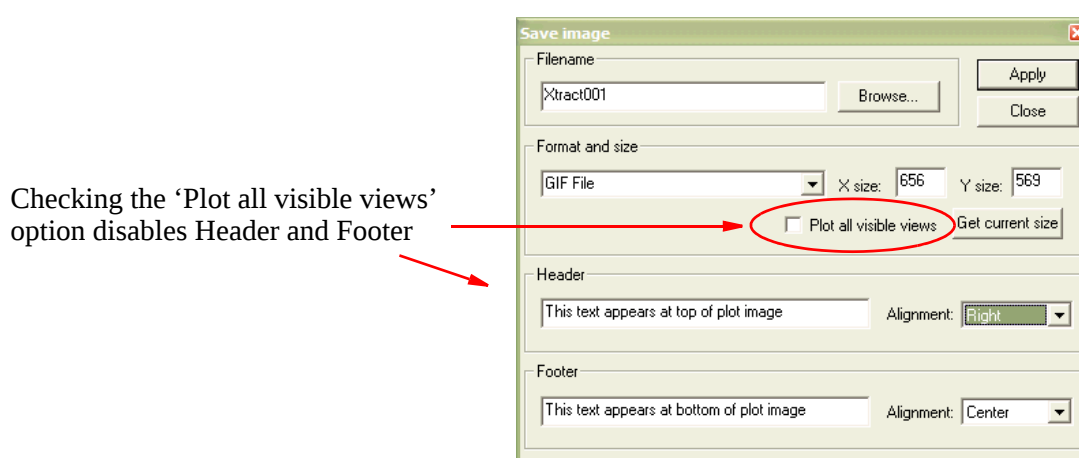


Figure 3.43 Plotting – Save image

Note: If you have added text and pointers (see Figure 3.38) then a setting of the plot size should approximately maintain the width over height quotient of the display area or else the text and pointer positioning may not be as intended.

For either of the alternatives for printing/saving graphics (File > Print, Edit > Copy view to clipboard, Edit > Copy visible views to clipboard and File > Plot) a dialog appears: Export View Mode. Hitting Return accepts the default mode: exporting colours with white background and black text. Alternatively you may choose black & white mode and 'as is', i.e. as appearing on the screen. See Figure 3.44.

Note: The tool button  offers a preview of the graphics exported in black & white mode. This is a preview only and not a working mode. I.e. when selecting a new results component colours will reappear in an imperfect manner. Click  once more to return to normal colour mode.

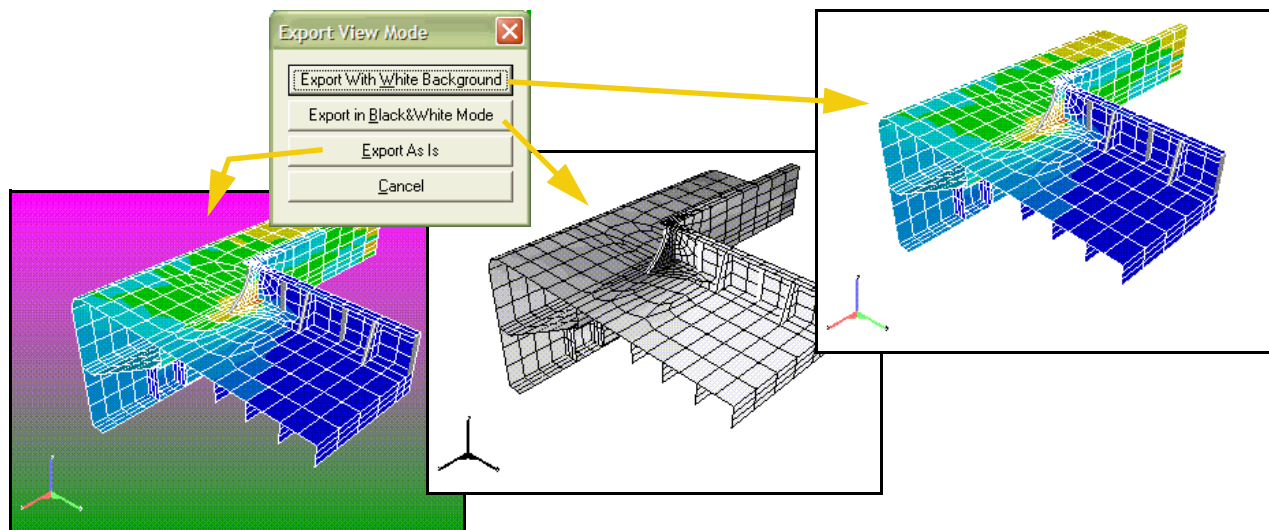


Figure 3.44 Export View Mode dialog

3.2 Structural Analysis Results

In the general case structural analysis results are organised as follows:

- Run numbers
 - Each run number contains a number of result cases
 - And finally, each result case contains a number of occurrences

The run and occurrence numbers are normally irrelevant for a plain static analysis. Such an analysis will include a number of result cases under which all results data are found.

Section 3.3 explains the meaning of the run and occurrence numbers and in which cases these are relevant.

The results data under each run number, result case and occurrence are organised in a hierarchical manner:

- Result position (Nodes, Elements, Element average, Resultpoints and Surface resultpoints)
 - Result attribute (DISPLACEMENT, G-STRESS, D-STRESS, ...)
 - Result component (DISPLACEMENT ALL, DISPLACEMENT X, G-STRESS SIGXX, G-STRESS VONMISES, ...)

A particular (occurrence of a) result case will contain a subset of all possible sets of result positions, attributes and components depending on the type of analysis, type of element and other factors. The complete set of result positions and attributes is presented in the following.

3.2.1 Result Positions

The result positions Nodes and Elements are both located in the nodes of the model. The values in position Nodes are the averaged values of adjoining elements. More specifically, the values in position Elements as

illustrated for the 8 node shell element in Figure 3.54. The values in position Element average are the averages within each element (of the values in position Elements), i.e. a single value for each element. The Resultpoints and Surface resultpoints positions are found inside the elements as illustrated for the most common elements in Figure 3.45 through Figure 3.56. The sketches also show the Resultpoint numbering as appearing in Xtract. The Sestra user manual contains more information on the elements.

The 2 node beam element has 3 Resultpoints: on the neutral axis in the ends plus in the midpoint.

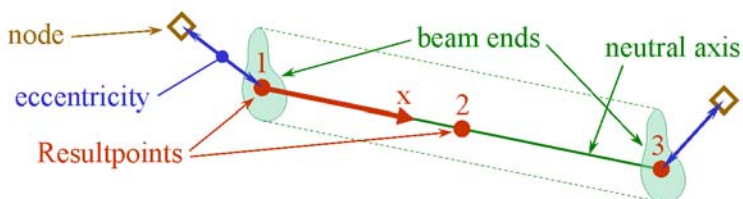


Figure 3.45 Resultpoints for 2 node beam element (BEAS)

The 3 node beam element has 2 Resultpoints: on the neutral axis as shown.

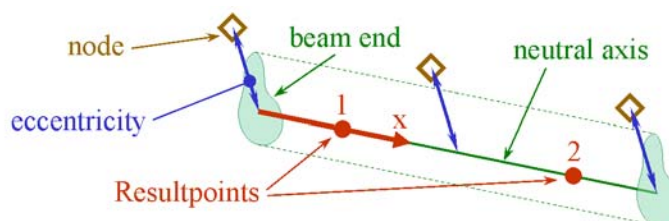


Figure 3.46 Resultpoints for 3 node beam element (BTSS)

The 3 node membrane element has 1 Resultpoint: in the midpoint.

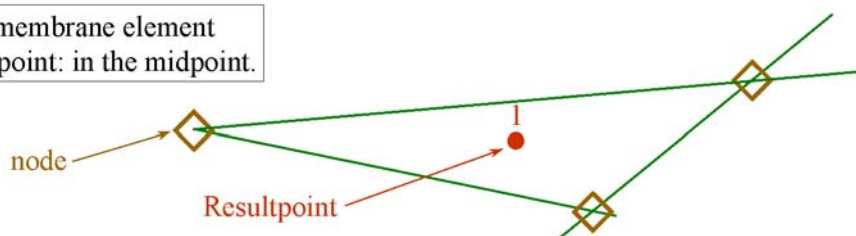


Figure 3.47 Resultpoints for 3 node membrane element (CSTA)

The 4 node membrane element has 5 Resultpoints: in each node plus in the midpoint.

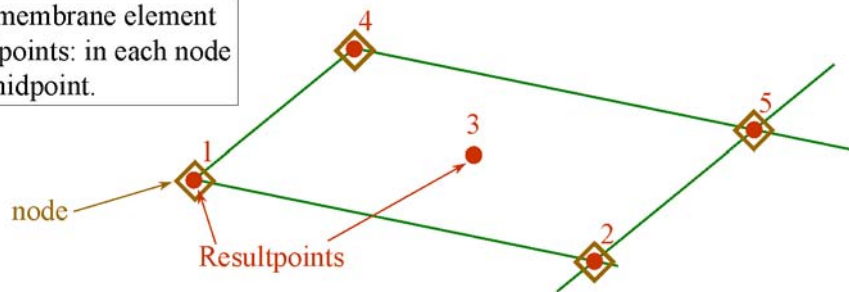


Figure 3.48 Resultpoints for 4 node membrane element (LQUA)

The 6 node membrane element has 4 Resultpoints: inside the element as show.

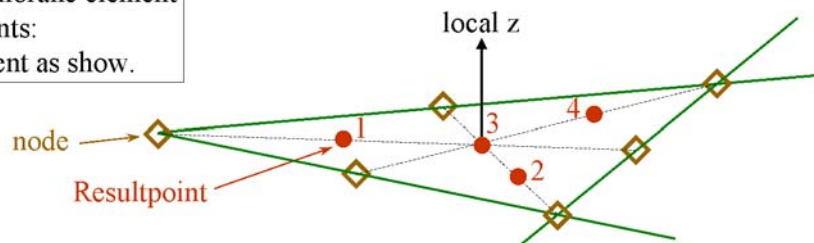


Figure 3.49 Resultpoints for 6 node membrane element (ILST)

The 8 node membrane element has 4 Resultpoints: inside the element as shown (the Gauss points).

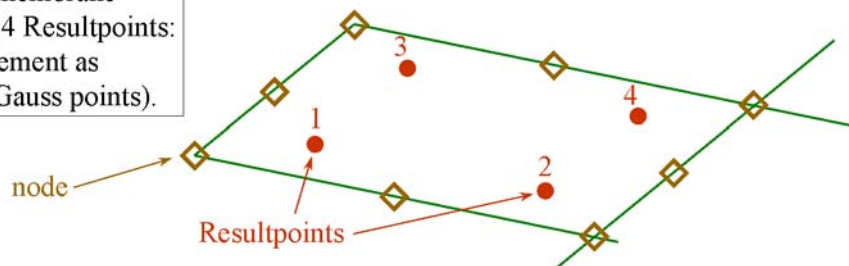


Figure 3.50 Resultpoints for 8 node membrane element (IQQE)

The 3 node shell element has 8 Resultpoints: in the upper (top) and lower (bottom) surfaces for each node plus in the midpoint.

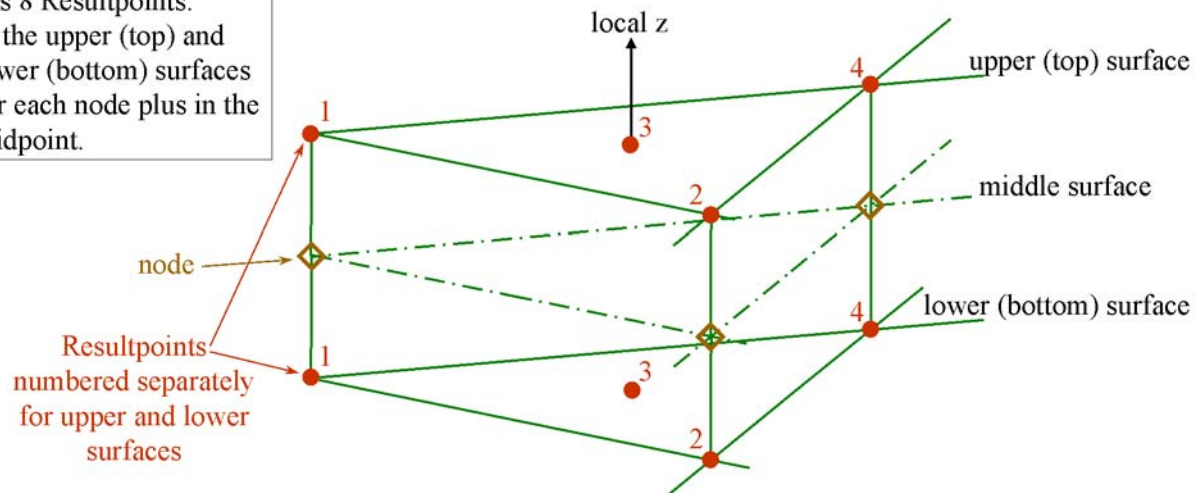


Figure 3.51 Resultpoints for 3 node shell element (FTRS and FTAS)

The 4 node shell element has 10 Resultpoints: in the upper (top) and lower (bottom) surfaces for each node plus in the midpoint.

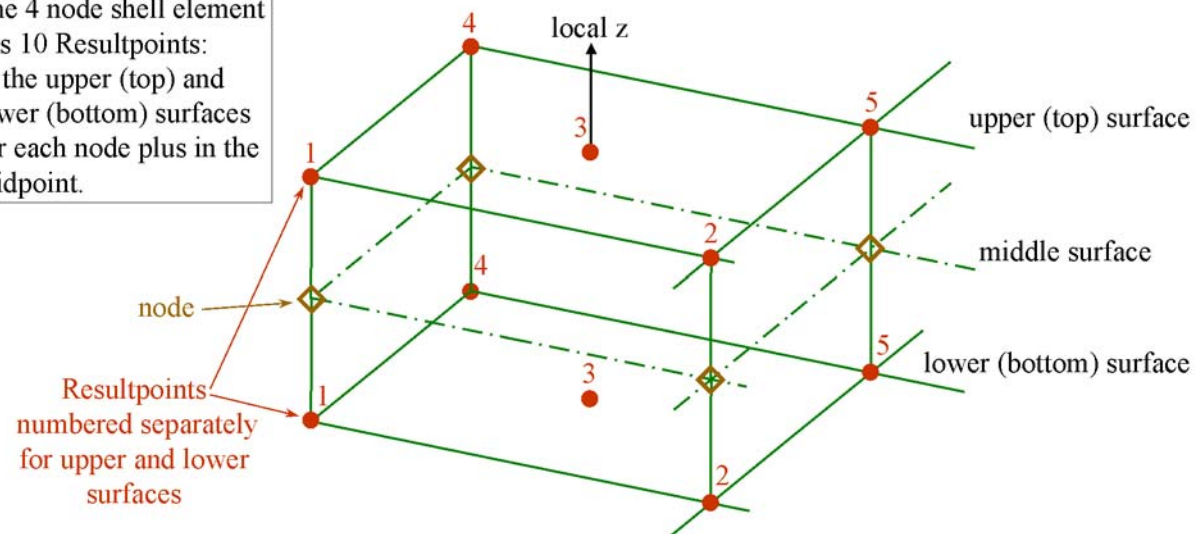


Figure 3.52 Resultpoints for 4 node shell element (FQUS and FQAS)

The 6 node shell element has 6 Resultpoints: in the corners of a 'prism' inside the element' as shown (the Gauss points).

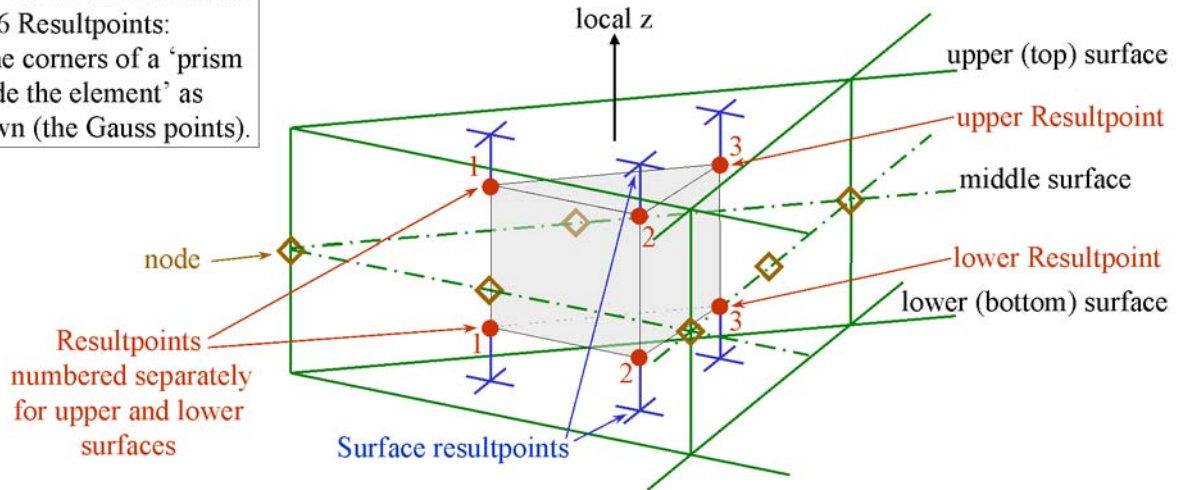


Figure 3.53 Resultpoints for 6 node shell element (SCTS)

σ_a is Elements position stress (extrapolated to node in top surface from the Resultpoints).
 σ_b , σ_c , σ_d are the corresponding stresses for the neighbouring elements.
 The Nodes position stress in the top surface is: $\sigma_{average} = (\sigma_a + \sigma_b + \sigma_c + \sigma_d)/4$

The 8 node shell element has 8 Resultpoints: in the corners of a 'box' inside the element' as shown (the Gauss points).

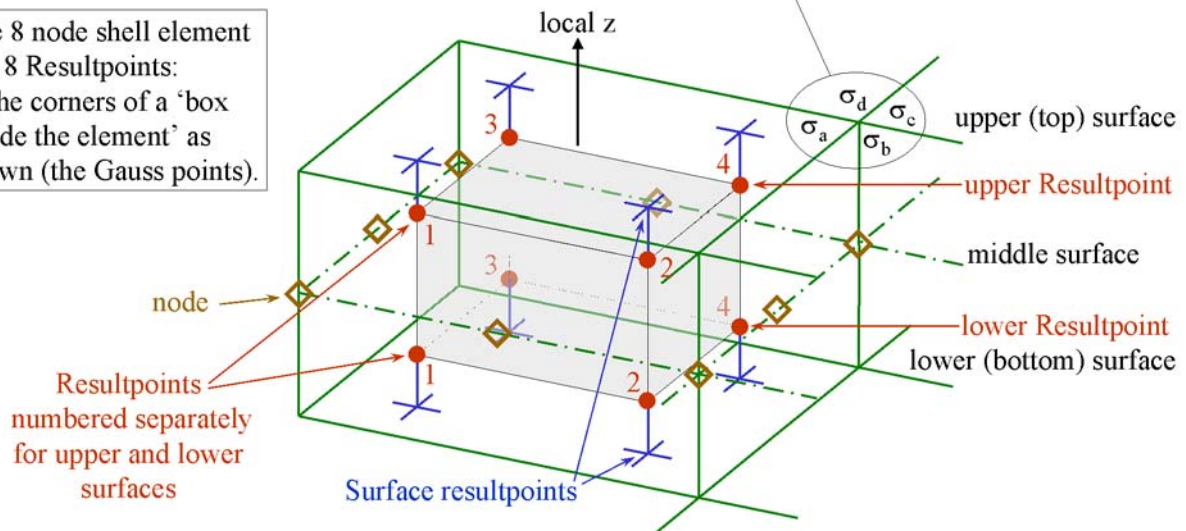


Figure 3.54 Resultpoints and Surface resultpoints for 8 node shell element (SCQS)

The 8 node solid element has 8 Resultpoints: in the corners of a 'box' inside the element' as shown (the Gauss points).

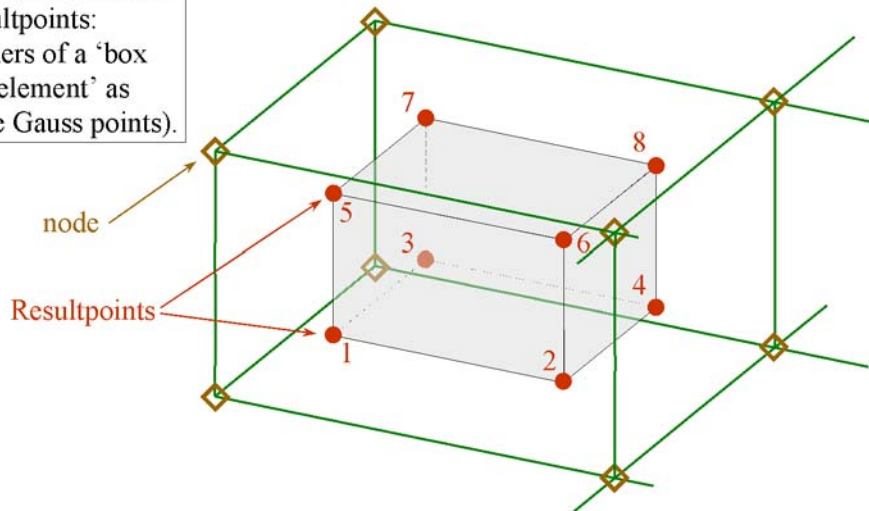


Figure 3.55 Resultpoints for 8 node solid element (LHEX)

The 20 node solid element has 8 Resultpoints: in the corners of a 'box' inside the element' as shown (Gauss points).

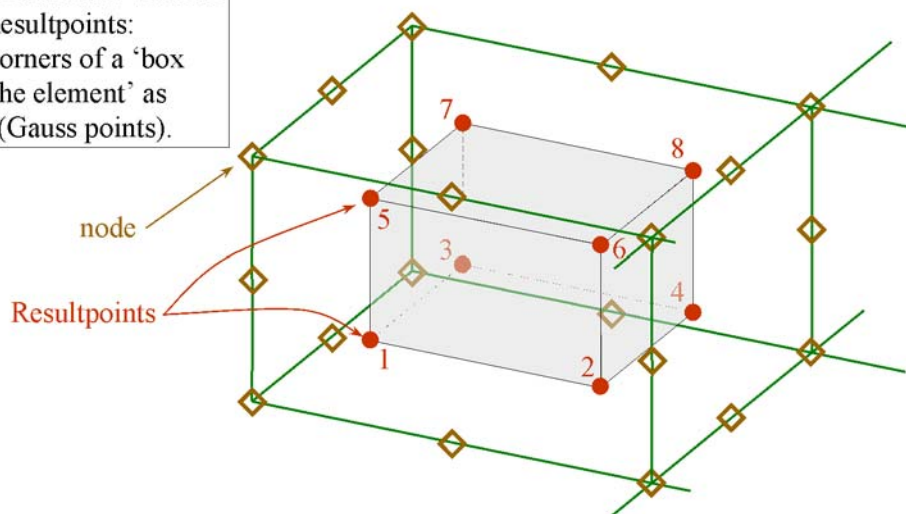


Figure 3.56 Resultpoints for 20 node solid element (IHEX)

The result attributes contained in the various result positions are shown in Table 3.1 and briefly described below. The components contained in each result attribute are explained in Appendix B 2.

Table 3.1 Result positions and attributes versus element types

Element type	Position	Attributes								
		DISPLACEMENT VELOC. & ACCEL. REACTION-FORCE	G- STRESS	P- STRESS	PM- STRESS	G- FORCE	D- STRESS	R- STRESS	B- STRESS	SURFACE- LOADS
Truss	Nodes	√								
	Elements		√							
	Element average		√							
	Resultpoints		√							
2 & 3 node beam	Nodes	√								
	Elements					√			√	
	Element average					√			√	
	Resultpoints					√			√	
3 & 4 node membrane 6 & 8 node membrane	Nodes	√	√	√	√			√		
	Elements		√	√	√			√		
	Element average		√	√	√			√		
	Resultpoints		√	√						
3 node shell 4 node shell	Nodes ^a	√	√√	√√	√		√	√		√
	Elements ^a		√√	√√	√		√	√		√
	Element average ^a		√√	√√	√		√	√		
	Resultpoints ^a		√√	√√			√			√
6 & 8 node shell	Nodes ^a	√	√√	√√	√		√	√		√
	Elements ^a		√√	√√	√		√	√		√
	Element average ^a		√√	√√	√		√	√		
	Resultpoints ^a		√√	√√						√
	Surface resultpoints ^a		√√	√√			√			√

Table 3.1 Result positions and attributes versus element types

Element type	Position	Attributes								
		DISPLACEMENT VELOC. & ACCEL. REACTION-FORCE	G- STRESS	P- STRESS	PM- STRESS	G- FORCE	D- STRESS	R- STRESS	B- STRESS	SURFACE- LOADS
6 & 8 node layered shell	Nodes ^{b c}	√	√√√√	√√√√	√√√		√√√			√
	Elements ^{b c}		√√√√	√√√√	√√√		√√√	√		√
	Element average ^{b c}		√√√√	√√√√	√√√		√√√	√		
	Resultpoints ^{b c}		√√√√	√√√√				√		√
	Surface resultpoints ^{b c}		√√√√	√√√√			√√√			√
Solid elements	Nodes	√	√	√						√
	Elements		√	√						√
	Element average		√	√						
	Resultpoints		√	√						√

a. Two tick marks imply that the attribute is available in upper and lower surfaces.

b. Three tick marks imply that the attribute is available in all layers.

c. Four tick marks imply that the attribute is available in upper and lower surfaces of all layers.

Result Position: Nodes

The attributes available in the nodes are:

- Nodal displacements (DISPLACEMENT)
- Nodal velocities and accelerations if relevant (VELOCITY and ACCELERATION)
- Reaction forces in fixed and prescribed nodes (REACTION-FORCE)
- Nodal stresses averaged between adjoining elements, these are available in the form of:
 - General stresses (G-STRESS)
 - Principal stresses (P-STRESS)
 - Principal membrane stresses (PM-STRESS)
 - Decomposed stresses (D-STRESS) – general stresses decomposed into membrane and bending parts
 - Stress resultants (R-STRESS) – general stresses integrated through the shell/membrane thickness

Averaging of stresses between adjoining elements (i.e. coupled to a common node) is performed only for those of any number of elements for which all of the following conditions are fulfilled:

- The element coordinate systems do not deviate more than 5 degrees (a default value adjusted by Results > Settings, the Misc tab).
- The element thicknesses do not deviate more than 10% (a default value adjusted by Result > Settings, Misc tab).
- The material type is the same.

This implies that:

- An element with an element coordinate system deviating more than 5 degrees compared with *all* its neighbour elements will have nodal stresses equal to the element stresses.
- For a node on the intersection line between two intersecting plates where 8 plate elements meet two average values will be computed: one value for the four coplanar elements of one of the plates and another value for the four coplanar elements of the other plate.

Note: For shell elements some of the stresses are strictly not positioned in the nodes: While the nodes are located in the middle surface of the shell the stresses are available for the upper and lower surfaces (in the nodes projected onto the surfaces).

Note: A change in stress from one element to the next may be caused by a concentrated force or a line/surface load acting at the border between the elements. In such cases the averaging performed by Xtract will be incorrect.

Note: When viewing a set the nodal averaging will not account for elements not included in the set. Nodal stresses on the edge of a set being a cut-out of a plate/shell will thus be incorrect.

Result Position: Elements

The attributes available in the nodes of the individual elements are:

- Beam forces and moments (G-FORCE) – more precisely the position is in the beam ends, see note below
- Beam stresses (B-STRESS) – G-FORCE components divided by area and minimum section moduli
- Stresses extrapolated from the result points to the element nodes and pertaining to the individual elements (i.e. no averaging between adjoining elements), these are available in the form of:
 - General stresses (G-STRESS)
 - Principal stresses (P-STRESS)
 - Principal membrane stresses (PM-STRESS)
 - Decomposed stresses (D-STRESS) – general stresses decomposed into membrane and bending parts
 - Stress resultants (R-STRESS) – general stresses integrated through the shell/membrane thickness
- Surface loads (SURFACE-LOADS) – normal pressure loads (positive when pointing in the direction of the element z-axis for plate/shell elements, positive when pointing into volume elements)

Note: For shell elements some of the stresses are strictly not positioned in the nodes: While the nodes are located in the middle surface of the shell the stresses are available for the upper and lower surfaces (in the nodes projected onto the surfaces).

Note: The G-FORCE attribute is positioned in the beam ends rather than in the nodes. This has significance for beam elements with eccentricities.

Result Position: Element average

The attributes available as average values in the midpoints of the elements are:

- Beam forces and moments (G-FORCE)
- Beam stresses (B-STRESS) – G-FORCE components divided by area and minimum section moduli
- Stresses available in the form of:
 - General stresses (G-STRESS)
 - Principal stresses (P-STRESS)
 - Principal membrane stresses (PM-STRESS)
 - Decomposed stresses (D-STRESS) – general stresses decomposed into membrane and bending parts
 - Stress resultants (R-STRESS) – general stresses integrated through the shell/membrane thickness

Note: For shell elements some of the stresses are positioned in the upper and lower surfaces at the midpoint.

Result Position: Resultpoints

These are the points in which the results are found on the results file, the location of these points depends on the type of element. The attributes available are:

- Beam/truss forces and moments (G-FORCE) – these are the forces computed by the analysis program
- Stresses in the form of:
 - General stresses (G-STRESS) – these are the stress components computed by the analysis program (except for the VONMISES component which is based on the other components)
 - Principal stresses (P-STRESS)
 - Decomposed stresses (D-STRESS) – general stresses decomposed into membrane and bending parts, relevant for 3 and 4 node shell elements (for 6 and 8 node shell elements these stresses are available under Surface resultpoints). The term ‘Resultpoints’ is somewhat misleading as the D-STRESS attribute relates to a line through the shell thickness in the position of the result points rather than to the individual result points.

Note: For the result points of 6 and 8 node shell elements the choice between the upper and lower surfaces (see Section 3.2.2) has a modified meaning: rather than meaning the upper and lower external shell surfaces they mean the result point closest to the given surface. Confer the section Result Position: Surface Resultpoints below.

Result Position: Surface Resultpoints

This position is available for 6 and 8 node shell elements only. These elements have their result points located inside the elements in the thickness direction. The surface result points are therefore the result points extrapolated to the upper and lower shell surfaces. The attributes available are:

- Stresses in the form of:
 - General stresses (G-STRESS)
 - Principal stresses (P-STRESS)
 - Decomposed stresses (D-STRESS) – general stresses decomposed into membrane and bending parts. The term ‘Surface resultpoints’ is somewhat misleading as the D-STRESS attribute relates to a line through the shell thickness in the position of the surface result points rather than to the individual surface result points.

3.2.2 Upper and Lower Surfaces

For plate/shell elements there is a concept of upper and lower surfaces. When presenting for example Elements/G-STRESS/SIGXX (position/attribute/component) the upper surface is selected by default. The upper surface is the one on the positive local z-side, see Figure 3.54.

Switch between the upper and lower surfaces by:

Results > Shell surface and layer

Which surface is currently selected is stated in the Info box.

3.3 Hierarchical Organization of Results

Rather than having a plain and flat list of result cases the results are organised in a hierarchical manner: a run number contains a number of result cases which in turn contain a number of occurrences. This is a practical way of organising results, particularly results from a dynamic analysis.

See Section 3.16.2 for how to refer to a result case.

3.3.1 Run Number

The run number is used to separate logically independent sets of results.

Combinations, scan result cases and scaled complex result cases created in Xtract are stored in a run ‘number’ termed User¹. In most cases this will be the only other run number than run number 1 in which all result cases of the results file are organised.

In certain cases, however, there will be additional run numbers on the results file:

1. In Xtract’s database the run number 9999 is used even though it appears as ‘User’. When making combinations in Prepost the run number 9999 should therefore be avoided.

- Results files produced by separate analyses (runs) of (more or less) the same model may be merged into a single results file using the auxiliary program Prepost. The run number (1, 2, ...) will identify the separate analyses.
- In a forced dynamic analysis (frequency or time domain) using the Modal Superposition method the mode shapes (eigenvectors) may optionally be stored on the results file (i.e. they are retracked together with the dynamic forced response analysis results – see the Sestra User Manual about this). In such case the mode shapes will belong to run number 1 and the forced response results will belong to run number 2.
- The auxiliary program Prepost may be used to create combinations that are stored on the results file. Such combinations may be given any run number.

Note: Even when there is only one run number the command log and correspondingly the Command input file must always include the run number (which will be 1).

3.3.2 Result Cases and Occurrences

The result cases and occurrences are used to logically organize the results. Logically related results are grouped as occurrences under the same result case. Here is an overview of how the result cases and occurrences are employed:

- For a static analysis the result cases correspond to the load cases (load vectors) for which the model was analysed. There will be only one occurrence per result case so the occurrence may be neglected.
- For a free vibration (eigenvalue) analysis the modes shapes are organised as occurrences under a single result case.
- For a frequency domain analysis, whether it be a quasi-static (neglecting dynamic effects) or true dynamic analysis, the excitation frequencies belonging to the same wave heading (wave direction) will be occurrences of the same result case. The wave headings (the LOHI commands in the Sestra input) will correspond to result cases.
- For a time domain dynamic analysis the time steps belonging to the same load history will be occurrences of the same result case. The load histories (the LOHI commands in the Sestra input) will correspond to result cases.

Note: Even when there is only one occurrence (in which case this bottom level of the results hierarchy may be neglected when using Xtract in GUI mode) the command log and correspondingly the Command input must always include the occurrence (which will be 1).

3.4 Postprocessing Complex (Frequency Domain) Results

A frequency domain analysis produces complex results, i.e. the results are comprised of a real and an imaginary part. For example, in a certain point there are two numbers expressing the stress σ , σ_{real} and σ_{imag} , and to determine the stress we need to select a phase of the incoming wave, ωt , (a point of time) and then compute the stress according to the formula:

$$\sigma = \sigma_{\text{real}} \cos(\omega t) - \sigma_{\text{imag}} \sin(\omega t)$$

Computing the value of a component for a certain phase of the wave is termed evaluating the component.

If you are unfamiliar with postprocessing complex results then go to Appendix B 5 for some basic theory on this subject.

3.4.1 Evaluating a Single Complex Resultcase

When presenting a complex component (by double-clicking it) Xtract will automatically evaluate it for phase zero ($\omega t=0$). At the same time the Complex evaluation dialog will appear, see Figure 3.57. You may then choose to evaluate the component for any specific phase by entering the phase, or using the slider, followed by clicking Apply. In addition, the Complex evaluation dialog has the following alternatives for evaluating a component depending on whether it is harmonic or not:

- For all harmonic basic components (e.g. DISPLACEMENT X, G-STRESS SIGXX, G-FORCE NXX, etc.):
 - Real (R), corresponding to phase = 0
 - Imaginary (I), corresponding to phase = 270 (or -90)
 - Magnitude = $\text{SQRT}(R^2 + I^2)$, the maximum value of the component through the cycle
 - Phase shift = $\text{ATAN}(I/R)$, the phase lag between the wave and the component (entering the negative phase shift for phase yields a value of the component equal to the magnitude)
 - Give phase in degrees, the phase of the incoming wave (for wave loading produced by Wajac, Wadam and Wasim phase = 0 corresponds to the crest of the wave being in $X=0$, $Y=0$)
- For non-harmonic components (VONMISES, P1, P2 and P3):
 - Give phase, the phase of the incoming wave
 - Maximum through cycle, the maximum is found by stepping through the cycle (360 degrees) at a 10 degree interval (a default value that may be modified by Results > Settings, the Misc tab)
 - Phase angle (in degrees) of maximum, the angle found by the stepping procedure described above

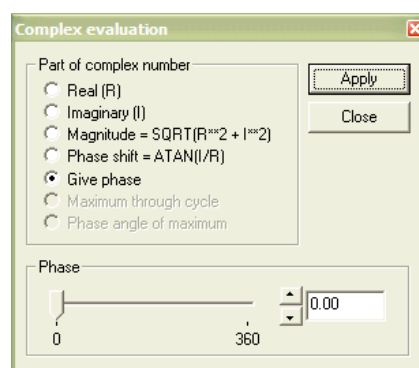


Figure 3.57 The Complex evaluation dialog

The displacement (Nodes/DISPLACEMENT/ALL) deviates from the harmonic components in that the Phase shift is unavailable. This is because the displacement will always be a positive value and therefore not

a normal harmonic component for which a phase shift may be found by $\text{atan}(I/R)$. (The displacement forms an ellipse in space.)

Note: Deformed model is available only when the evaluation is performed for a specific phase, i.e. Real, Imaginary and given phase. This is because the Magnitude and Maximum alternatives involve having a phase varying over the model.

Figure 3.58 explains the information provided by the Info box for a complex result case.

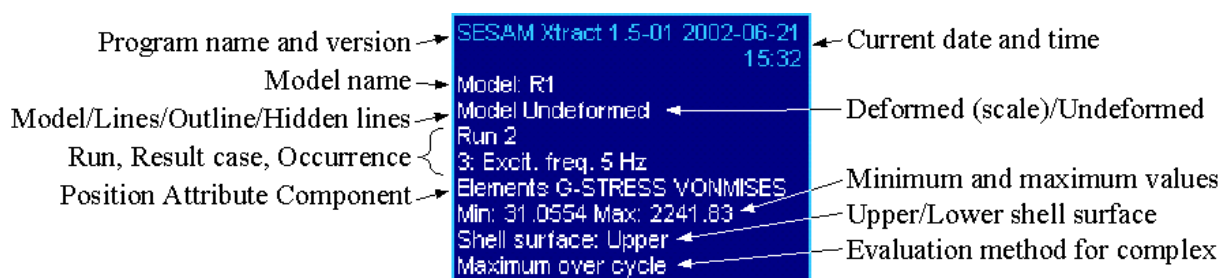


Figure 3.58 The Info box content for a complex result case

3.4.2 Combining a Complex Resultcase with Static Resultcases

When creating a combination involving complex results only a single complex result case is allowed. (See Appendix B 5.2 for an explanation of this limitation which is introduced to prevent the user creating meaningless combinations.) The single complex result case may be combined with any number of static (non-complex) result cases.

Combining (adding with scaling factors) result cases is done through:

Results > Combine (or click the tool button )

The Combine dialog, see Figure 3.59, appears in which a single complex Source result cases is transferred to the Combination result cases list box. Subsequent to this any number of static (non-complex) Source result cases may be transferred. Select result cases in the Combination result cases list box and adjust the Scaling factor (default 1), give the combination a name and click Apply.

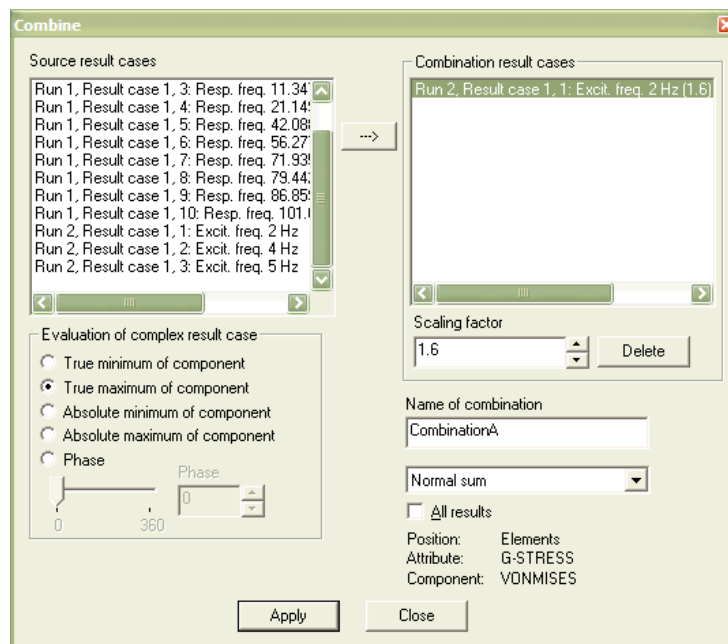


Figure 3.59 The Combine dialog for complex results

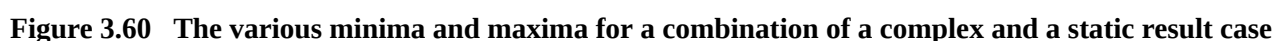
For the complex Source result case a method for Evaluation of complex result case must be selected. The alternatives are:

- True minimum of component
- True maximum of component
- Absolute minimum of component (the value, positive or negative, when the absolute value is minimum)
- Absolute maximum of component (the value, positive or negative, when the absolute value is maximum)
- Phase, selecting this option a phase (in degrees) must also be given

Note: The true/absolute minimum/maximum of component above is found for the complex and static result cases combined as illustrated in Figure 3.60. See also Section B 5.2.5.

The true/absolute minimum/maximum of a component are all found by stepping through the cycle (360 degrees) at a 10 degree interval (a default value that may be modified by Results > Settings, the Misc tab). The position/attribute/component in question appears in the lower right area of the Combine dialog.

For the Phase option only position/attribute is relevant and appears in the Combine dialog.



Note: Deformed model is available (or meaningful) for a combination including a complex result case only when it is evaluated for a specific phase (not true/absolute minimum/maximum) and when the combination includes nodal displacements.

Combinations are put in a Run named User (see Section 3.3.1 for an explanation of the concept of Run numbers). To select the combination you therefore need to select the Run named User, see Figure 3.24, thereafter select the relevant Result case (i.e. combination name) and finally double-click the desired component.

Click the Info button to get information on a combination.

Delete a combination by the line-mode command:

```
result deletecase name-of-case
```

Note: Do not delete a combination that is referred to by another combination or scan result case as this may lead to inconsistency.

3.4.3 Scanning Complex Resultcases

Scanning for the maximum or minimum (absolute) value of a component may be performed for any number of (scaled) complex and static (non-complex) result cases by:

Results > Scan (or click the tool button )

The Scan dialog, see Figure 3.61, appears in which Source result cases are transferred to the Selected source result cases list box. Select result cases in the Selected source result cases list box and take two actions: (1) adjust the Scaling factor (default 1) and (2) select a method for Evaluation of complex result case (normally you want the same evaluation method for all result cases). The alternatives are:

- Scan criterion, this involves that the complex result case is evaluated for the phase giving (absolute) maximum/minimum value of the component in question according to the Scan criterion. The evaluation is performed by stepping through the cycle (360 degrees) at a 10 degree interval (a default value that may be modified by Results > Settings, the Misc tab). Scan criterion is normally the desired option.
- Phase, selecting this option a phase (in degrees) must also be given.

The specification of Scan criterion (in the lower left area of the dialog) is common for all complex and static result cases to be scanned. The alternatives are:

- Max (Absolute), the (absolute, see Figure 3.60) maximum value of the component over all Selected source result cases is picked.
- Min (Absolute), the (absolute, see Figure 3.60) minimum value of the component over all Selected source result cases is picked.

Conclude by giving the scan result case a name and click Apply.

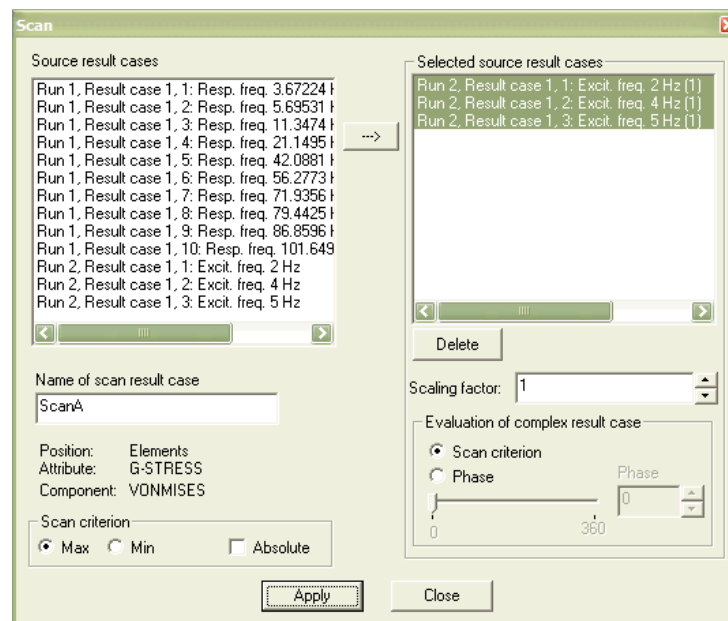


Figure 3.61 The Scan dialog for complex result

Note: You may select and transfer two or more Source result cases at the time.

Note: You may select and change Scaling factor and Evaluation of complex result case for two or more Selected source result cases at the time.

Note: The Scan criterion is common for all scanned result cases whereas the method for Evaluation of complex result case may in principle vary.

Note: The scanning is performed for the result position/attribute/component highlighted in the database window and listed to the lower left in the Scan dialog. Change result position/attribute/component by selecting (single-click to highlight) the appropriate one in the database window and return to the Scan dialog. The new result position/attribute/component will be listed in the Scan dialog.

Note: Even though the scanning is performed for a specific component the scan result case will contain all components of the attribute in question. I.e. if you scan for Elements/G-STRESS/VONMISES then the scan result case will also contain SIGXX, SIGYY and TAUXY (assuming four node element). These other components are normally of little interest though.

Note: The scan result case created will at any point in the model contain the (absolute) maximum/minimum of the selected component over all selected (and scaled) result cases.

Note: Deformed model for a scan result case is normally unavailable and meaningless.

Scan result cases are put in a Run named User (see Section 3.3.1 for an explanation of the concept of Run numbers). To select the scan result case you therefore need to select the Run named User, thereafter select the relevant Result case (i.e. scan result case) and finally double-click the desired component.

In addition to the attribute with components resulting from the scanning, the scan result case will contain an attribute named MISC which in turn contains the component SCANINDEX. Selecting (double-clicking) this component gives labels (text strings) for all scanned points that identify the result cases that 'won' the scanning. By default this label will include the full result case identification which typically may look like:

Run 2, Result case 1, 3 Excit. freq. 5 Hz, phase 170

The text string of the SCANINDEX label may be adjusted and appended to the numeric display of results as shown in Figure 3.26.

Click the Info button to get information on a scan result case.

Delete a scan result case by the line-mode command:

```
result deletecase name-of-case
```

Note: Do not delete a scan result case that is referred to by another combination or scan result case as this may lead to inconsistency.

3.4.4 Scaling a Complex Resultcase

A complex result case may be scaled resulting in a new complex result case through:

Results > Scale complex result case

The Scale complex result case dialog, see Figure 3.62, appears in which a single Complex result case with Scaling factor is specified. Give the new result case a name and click Apply.

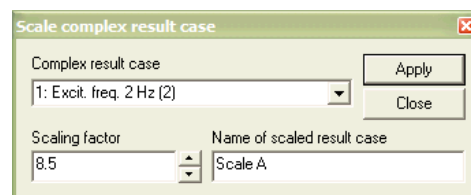


Figure 3.62 The Scale complex result case dialog

Scaled complex result cases are put in a Run named User (see Section 3.3.1 for an explanation of the concept of Run numbers). To select it you therefore need to select the Run named User, thereafter select the relevant Result case (i.e. scaled complex result case) and finally double-click the desired component. This is shown for combinations in Figure 3.24.

Click the Info button to get information on a scaled complex result case.

Delete a scaled complex result case by the line-mode command:

```
result deletecase name-of-case
```

Note: Do not delete a scaled complex result case that is referred to by combination or scan result case as this may lead to inconsistency.

3.4.5 Graphing Complex Resultcases

Creating a graph for a complex resultcase is done in the same way as for a static (non-complex) resultcase, see Section 3.1.15. The complex result case is evaluated by the current (last used) evaluation method, see Section 3.4.1.

3.5 View Sections

Any number of plane sections through the model may be defined by:

Model > View section

The View section dialog, see Figure 3.63, appears. Clicking Add automatically creates a plane through a point (Point in the plane) in the middle of the model (in the middle between the lowest and highest of each coordinate value) and with the normal vector [0 0 1] (pointing in the Z-direction). To see the plane switch to outline display of the model by:

Display > Show as outline (or click the tool button )

Or you may switch off display of the model showing only the plane by:

Display > Model

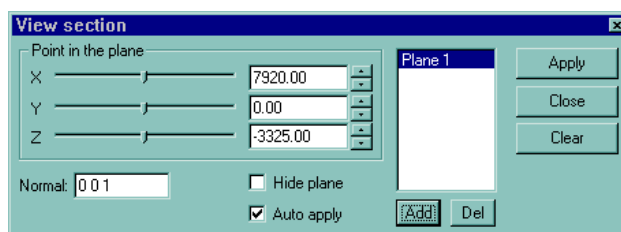


Figure 3.63 The View section dialog

Note: Only nodal displacements, velocities and accelerations may be presented for a section. For other results the section will be presented in grey colour.

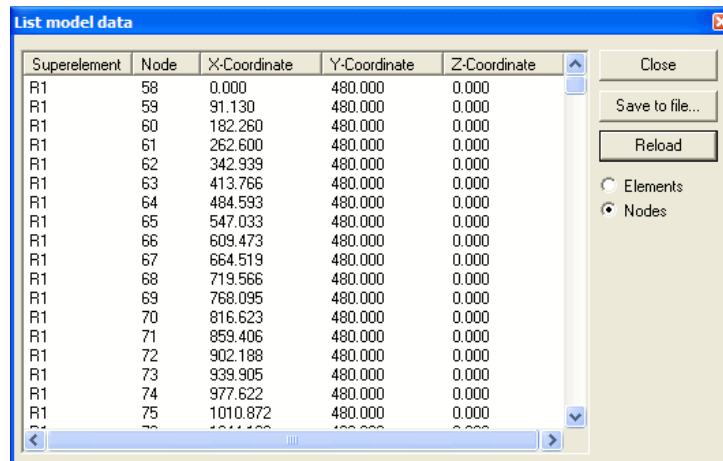
Having checked the Auto apply box the section may be dragged through the model by the appropriate slider for the Point in plane. Note that if the normal vector is [0 0 1] changing the Z-coordinate only for the Point in the plane will have effect.

3.6 List Model and Results Data

Model data in terms of nodal coordinates and element data may be listed (tabulated) by:

Model > List model data

The List model data window, see Figure 3.64, appears. Select Elements or Nodes and click Reload to switch between the two. The data is limited to the currently shown set, if any. Click Save to file to save a file that may be viewed, printed or imported into spreadsheets.



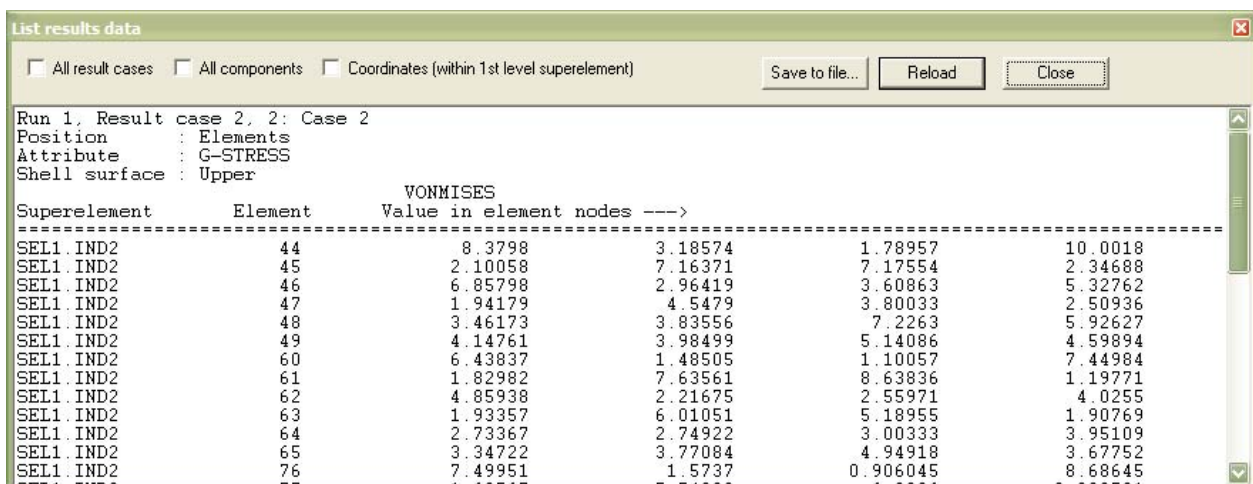
Superelement	Node	X-Coordinate	Y-Coordinate	Z-Coordinate
R1	58	0.000	480.000	0.000
R1	59	91.130	480.000	0.000
R1	60	182.260	480.000	0.000
R1	61	262.600	480.000	0.000
R1	62	342.939	480.000	0.000
R1	63	413.766	480.000	0.000
R1	64	484.593	480.000	0.000
R1	65	547.033	480.000	0.000
R1	66	609.473	480.000	0.000
R1	67	664.519	480.000	0.000
R1	68	719.566	480.000	0.000
R1	69	768.095	480.000	0.000
R1	70	816.623	480.000	0.000
R1	71	859.406	480.000	0.000
R1	72	902.188	480.000	0.000
R1	73	939.905	480.000	0.000
R1	74	977.622	480.000	0.000
R1	75	1010.872	480.000	0.000

Figure 3.64 The List model data window

The currently displayed component may be listed (tabulated) by:

Results > List results

The List results data window, see Figure 3.65, appears. The results listed is limited to the current set, if any, and the selected resultpoints (Results > Settings, Misc tab). The data may be saved to a text file. Select and present (double-click) another component or result case followed by clicking Reload to list a new component or result case.



Superelement	Element	VONMISES Value in element nodes --->
SEL1.IND2	44	8.3798
SEL1.IND2	45	2.10058
SEL1.IND2	46	6.85798
SEL1.IND2	47	1.94179
SEL1.IND2	48	3.46173
SEL1.IND2	49	4.14761
SEL1.IND2	60	6.43837
SEL1.IND2	61	1.82982
SEL1.IND2	62	4.85938
SEL1.IND2	63	1.93357
SEL1.IND2	64	2.73367
SEL1.IND2	65	3.34722
SEL1.IND2	76	7.49951

Figure 3.65 The List results data window

In the List results data window click all or any of the boxes ‘All result cases’, ‘All components’ and ‘Coordinates (within 1st level superelement)’ followed by Reload to add more data to the table. In Figure 3.66 is shown an example of tabulating all components of the current attribute as well as the coordinates of the points of the results.

List results data

☐ All result cases
☒ All components
☒ Coordinates (within 1st level superelement)

Save to file...

Reload

Close

Run 1, Result case 1, 1: Case 1

Position : Elements

Attribute : G-STRESS

Shell surface : Upper

SIGXX

SIGYY

TAUXY

TAUXZ

TAUYZ

VONMISES

X-coord

Y-coord

Z-coord

Superelement	Element	Value in element nodes --->						
R1	23	80.9277	46.7396	63.3537	55.0314			
		17.5523	-7.98076	-0.408293	9.92427			
		17.825	-8.24435	0.878951	6.64289			
		0	0	0	0			
		0	0	0	0			
		79.9381	53.1527	63.577	52.0883			
		0	121.507	121.507	0			
		0	0	66.6667	66.6667			
R1	24	55.7395	56.7839	59.6315	60.488			
		-0.157063	-0.828038	-1.646	3.85574			
		-0.367905	-1.02934	-1.168	0.00170058			
		0	0	0	0			
		0	0	0	0			
		55.8218	57.2302	60.5051	58.6553			
		121.507	228.626	228.626	121.507			
		0	0	66.6667	66.6667			

Figure 3.66 The List results data window with all components and coordinates added

Note: The element nodal values as shown in Figure 3.65 and Figure 3.66 are tabulated for each element in a sequence corresponding to the sequence in which the nodes are listed in the table created by Model > List model data. Switch on labels showing the node numbers in the Labeling dialog (Display > Labeling) to see the position of each single value. For 4 node plate elements the node sequence is anticlockwise about the element z-axis. For 8 node shell elements the sequence is also anticlockwise but with corner nodes first followed by mid-side nodes.

Note: In addition to the fact that the tabulated results are limited to the currently selected set you may limit the table further by Ctrl+LMB clicking nodes and Shift+Ctrl+LMB clicking elements as described in Section 3.1.12 followed by clicking Reload in the List results data window.

Note: When tabulating averaged stresses in position Nodes and limiting the selection of elements by either selecting a set or Shift+Ctrl+LMB clicking elements then the tabulated stresses are averaged based on the selection only. This corresponds to the way results are presented graphically (by colours or numerical values) as shown in Figure 3.36.

3.7 Animation

An animation is showing how a selected result component changes (over time) with the deformation. The Animation menu provides animation for:

- Mode shapes (and static results)
- Frequency domain results
- Time domain results (Result case animation)

The procedure in all cases is to set up the animation for the appropriate component and thereafter to run the animation. Setting up involves first selecting and presenting (i.e. double-clicking) a component of the desired run number, result case and occurrence and thereafter selecting the appropriate Animation menu. The alternative animation setups are described in Section 3.7.1 through Section 3.7.3.

Depending on the size of the model, the number of steps (frames) in the animation and the speed of your computer the setup may require some time. Selecting a part of the complete model, e.g. a lower level superelement or a set will speed up the setup of the animation. Upon completion of the setup the animation control buttons will be accessible (no longer grey), see Figure 3.67.



Buttons from left to right: Play backward, Step backward, Pause, Stop, Step, Play, End animation

Figure 3.67 The animation control buttons

Note: During an animation (without pausing it) the model may be manipulated (rotated, zoomed, panned, change to shrunken view, etc.) and the result settings may be changed (contour settings, etc.). Furthermore, existing sets may be retrieved and new sets may be defined.

Note: During an animation the current frame (step) number is given in the lower left corner of the viewport.

Note: During an animation you may slow down and, up to a limit decided by the capacity of your hardware, speed up an animation by Animation > Max FPS. In the dialog set the maximum number of frames to show per second. The default is 25.

Save an animation to either of the formats:

- AVI – for running an animation using a media player (e.g. Quick Time Player, RealOne Player and Windows Media Player)
- Animated GIF – for running an animation through a web browser

To save an animation stop it (to save all resources for the saving process) and do:

Animation > Save animation

In the Save animation dialog, see Figure 3.68, select File format and size, Frame options, give a Filename and click OK. The Frame options are:

- Frame repeat – the number of times each frame of the animation is repeated
- First frame repeat – number of repeats for the first frame (the animation stands still for a while in the beginning)
- Last frame repeat – number of repeats for the last frame (the animation rests a while at the end)
- Framerate – the target number of frames per second when played. To which degree this setting takes effect depends on the size of the animation in relation to the resources of your computer: Your computer may not be able to process the data quickly enough to play the animation at the desired speed. Or it will be too quick. In the latter case increasing the Frame repeat may help. Besides, this setting is disregarded by some media players.
- Cycle animation – choose whether the animation is set to be run back and forth

Note: Whether to run the animation once or repeatedly is determined by the media player.

Which data to give for the options above depend on the speed at which you want the animation to run, the resources of your computer and the animation size. You may need to do some experimenting to find the values suitable for your case.

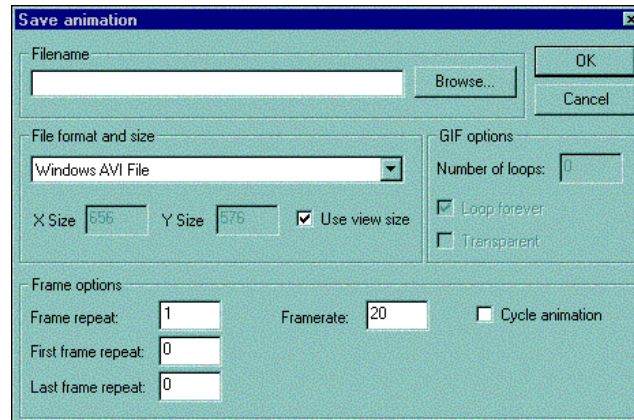


Figure 3.68 The Save animation dialog

Having clicked OK in the Save animation dialog Xtract will spend some time creating a number of bitmap files (that are deleted once the saving process is complete). The Video compression dialog, see Figure 3.69, appears in which you need to choose a video compression algorithm. Which algorithm is available to you depends on software installed on your computer.

The following compression algorithms have been tested and with given result:

- Cinepak Codec by Radius, Compression quality 100 – very good playback quality
- Microsoft Video 1, Compression quality 75 – lower playback quality for rich graphic information

- Microsoft MPEG-4 Video Codec V2 – good playback quality, small animation file, incompatible with Quick Time Player (and RealOne Player?)

Note: The size of the animation file and its quality (resolution) will depend on the video compressor.

Note: You need to use a video compressor – do not use Full Frames (Uncompressed) – as the animation file will then easily become prohibitively large.

Note: The temporary consumption of disk space during the saving process may be substantial: A bit-map file of size between 1 and 4 MB (depending on the display area size) will be created for each frame of the animation. The temporary disk consumption for saving a frequency animation with 10 degree stepping will thus be about 40 MB. Increasing the display area size and reducing the step size may demand several hundred MB of temporary disk space.

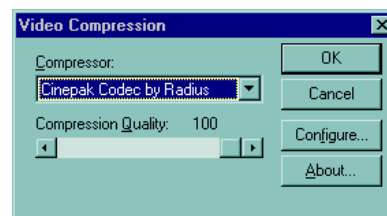


Figure 3.69 The Video compression dialog

Having clicked OK in the Video compression dialog Xtract will be locked while the compression of all the temporary bitmap files into the animation file is being done.

3.7.1 Animating Mode Shapes (and Static Results)

Having performed a free vibration (eigenvalue) analysis the results file will contain mode shapes stored as occurrences under a common result case. Prepare an animation by:

Animation > Mode shape animation

The Mode shape animation setup dialog, see Figure 3.70, appears. Set the desired Number of frames, i.e. the number of steps through a cycle (360°) and click OK. The steps are equidistant in time and the deformation will be harmonic (sinusoidal). The more steps the smoother animation.

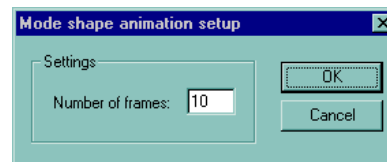


Figure 3.70 The Mode shape animation setup dialog

Note: The mode shape animation may also be used for static results. The static deformation is in such case assumed to be harmonic like a mode shape.

3.7.2 Animating Frequency Domain Results

Having performed a frequency domain analysis the results file will contain complex results describing the harmonic response. Typically, there will be result cases (wave directions) and several occurrences (frequencies) for each result case. The following menu will then be available:

Animation > Frequency animation

The Setup frequency animation dialog, see Figure 3.71, appears. Set the start and stop angles (normally these should be left as 0 and 359, respectively) and the Skip by parameter which is the step angle between start and stop. The default of 10 will give 36 steps (frames). The more steps the smoother animation.

Note: If you set up an animation for a certain component, say von Mises stress, you cannot change to another component without leaving the animation and setting up a new one.

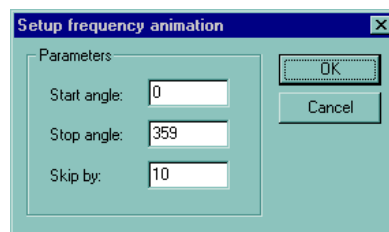


Figure 3.71 The Setup frequency animation dialog

3.7.3 Animating Time Domain Results

Having performed a time domain analysis the results file will contain a result case with occurrences being the time steps of a time history. If more than one time history has been analysed there will be more result cases. Prepare an animation by:

Animation > Result case animation

The Setup animation over result cases dialog, see Figure 3.72, appears. In the Occurrences list box all occurrences of the current result case will be listed. You have the option to exclude certain occurrences (time steps) by putting them into the Except list box (select the occurrence and click the arrow button). (Cancel an exclusion by selecting the occurrence in the Except list box and clicking Del.) You may also use the Step by parameter to run an animation for only every second, third, etc. time step. The default values (all occurrences and Step by 1) give the smoothest animation.

Note: Once the animation reaches the final time step it will automatically restart causing a discontinuity in the animation.

Note: If you set up an animation for a certain component, say von Mises stress, you cannot change to another component without leaving the animation and setting up a new one.

Note: The result case animation may also be performed for (quasi-)static results. I.e. results from an analysis neglecting inertia effects and in which the result cases are in reality time steps.

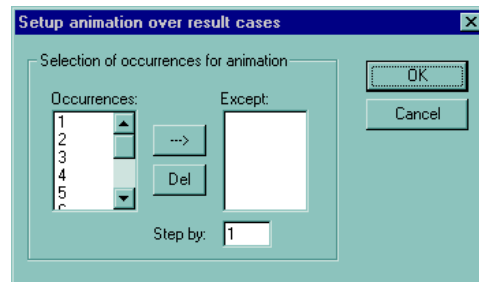


Figure 3.72 The Setup animation over result cases dialog

3.8 Hydrodynamic Analysis Results from Wadam or Wasim

As outlined in Section 2.1 (list items 2, 3 and 4) results from the hydrodynamic analysis programs Wadam and Wasim may be presented, optionally in combination with FE structural analysis results. Presenting Wadam and Wasim results thus falls into three cases:

- Hydrodynamic pressure loads and rigid body motion (frequency domain results) for a floating object as computed by Wadam or Wasim plus structural analysis results computed by Sestra (R+G-files). See Section 3.8.1.

The next two cases represent subsets of this case with respect to presentation options.

- Hydrodynamic pressure loads and rigid body motion for a floating object as computed by Wadam or Wasim (T+L+G-files). See Section 3.8.2.
- Hydrodynamic pressure loads for a floating object as computed by Wadam or Wasim (T+L-files). See Section 3.8.3.

The use of Xtract for presenting the above results differs to some extent from presenting FE results. The following sections describe the procedure for the most typical postprocessing task for each case.

Note: You should be familiar with the basics of using Xtract as covered in Section 3.1 before presenting and animating hydrodynamic analysis results.

3.8.1 Hydrodynamic Loads, Motion and Stresses – Wadam/Wasim and Sestra Results (R+G)

Floating and fixed large volume structures (barges, ships, FPSOs, semi-submersibles, gravity base structures, etc.) are analysed in Sesam by the hydrodynamic analysis programs Wadam or Wasim and the structural analysis program Sestra. These programs produce files with results that may be postprocessed by Xtract.

- The hydrodynamic analysis produces frequency domain results. (Wasim performs the analysis in time domain but is capable of transforming the results to frequency domain.) Hydrodynamic pressure loads are stored on Loads Interface Files (L-files) while the rigid body motion is stored on the Hydrodynamic Results Interface File (G-file), also termed Global Response File. The total wave elevation which is the

sum of incoming, reflected and diffracted waves may optionally have been stored as additional information on the G-file (Wadam only).

- The Sestra analysis stores the structural results on the Results Interface File (R-file). The R-file will contain the loads (L-file) as well.

This section presupposes basic knowledge in postprocessing complex results as covered in Section 3.4.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open. Open both the R-file (for the top level superelement) and the G-file by click and Ctrl+click. Both files may have any of the formats SIF, SIU and SIN.
- Double-click the model name appearing in the Database window of Xtract (R*) and the model (optionally in sea) is displayed. The various parts (superelements and sea surface) are colour coded.
- Click the Result Cases tab and select a result case and an occurrence and double-click a stress result component, e.g. Nodes/G-STRESS/SIGXX.

- Relevant for Wadam and not for Wasim:

Having double-clicked a result component other than a displacement, velocity and acceleration component the sea will turn gray. This is because the program switches Contours on (Part attributes dialog) in an attempt to display the results but when no results are found (there are no stress components for the sea) it is left gray. You must switch the Contours off.

Open the Part attributes dialog (Display > Part attributes), select the Sea surface and change how to display it:

— Switch off Contours.

— Switch on Mesh.

— Change the colour (to sea green or marine blue?) by clicking the Color button.

— Make the sea Transparent, maybe also adjust the transparency Alpha value (lower right corner), 0 is fully transparent, 255 is opaque.

- You may want to use the Part attributes dialog to change the display of the structure too (for example: Smooth colouring? Mesh? Beams Expanded?). See Section 3.1.25 for more details on Part attributes.
- Give Display > Settings and click the colour box for the Background and change it to grey or white or another light colour of your liking. A light background is better in combination with a transparent sea.
- You may want to rotate and zoom to see the structure from below.
- Present the rigid body displacement of the model and the wave elevation by Model > Deformed or the corresponding tool button. Note that the structural deformation is not included.
- Scale (increase) the deformations by Model > Deformation scaling.
- Give Animation > Frequency animation to set up an animation (see Section 3.7 for general information on animation). Set the Skip by value and click OK. Setting up the animation may require some time depending on the size of the model and the Skip by value. Once the animation control buttons (see Figure 3.67) appear you may start the animation.

- Run the animation by clicking the animation buttons.
- During the animation you may rotate, zoom and pan, change part attributes and more. But to change result case and occurrence you must end the current animation and set up a new one.

Figure 3.73 is a snapshot from an animation of a ship. The contours show a stress component.

Note: Opening a G-file (together with the R-file) involves that the rigid body motion of the model (plus the wave elevation) replaces the structural deformations as the components available under Nodes/DISPLACEMENT as well as the results presented by Model > Deformed. Structural deformations are unavailable in this case (open only the R-file only if you want to see such results).

Note: The normal pressure load on a shell is positive when pointing in the direction of the element z-axis. For a hull with the z-axis pointing outwards the pressure values will thus be negative.

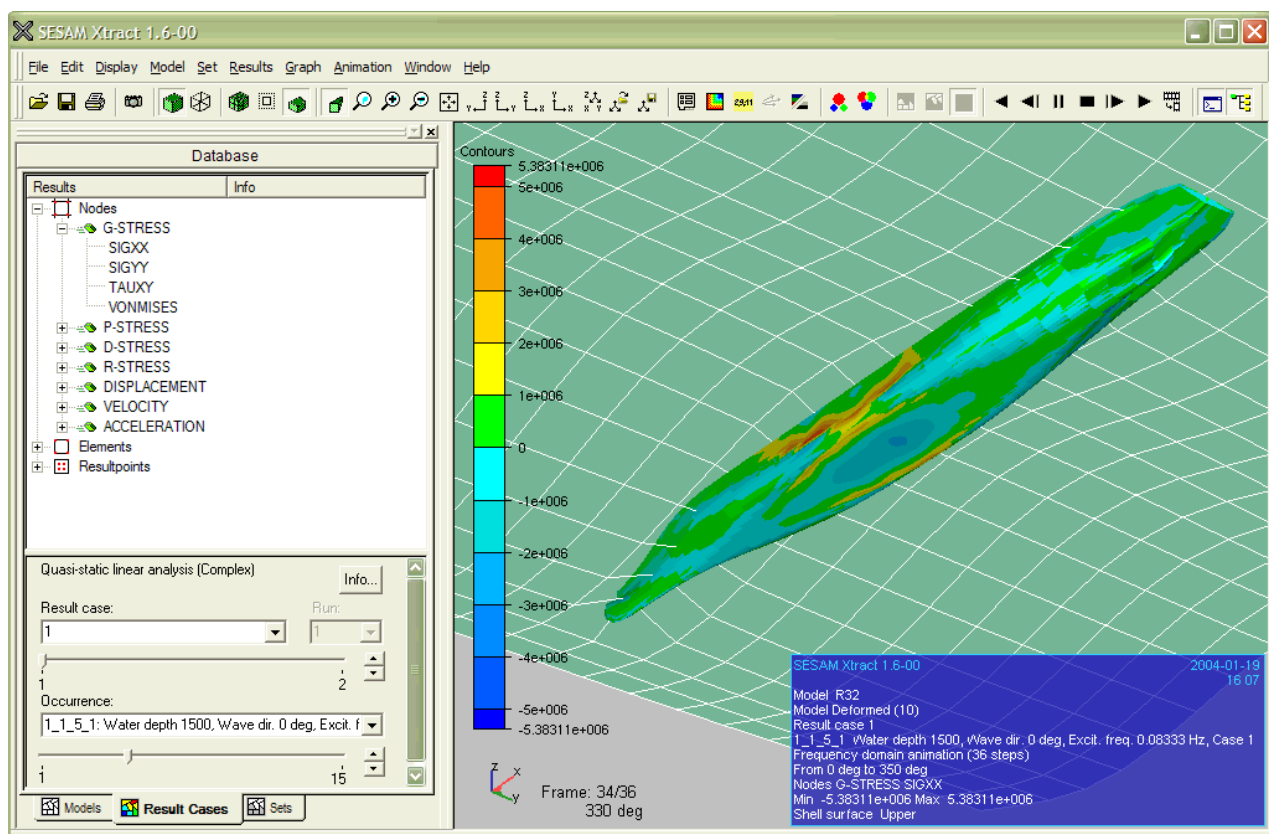


Figure 3.73 Animation of a ship (R+G-files) – stresses, motion and waves – from Wadam/Sestra

3.8.2 Hydrodynamic Loads and Motion – Wadam/Wasim Results only (T+L+G)

Floating and fixed large volume structures are analysed in Sesam by the hydrodynamic analysis programs Wadam or Wasim. The hydrodynamic analysis produces frequency domain results. (Wasim performs the

analysis in time domain but is capable of transforming the results to frequency domain.) Hydrodynamic pressure loads are stored on Loads Interface Files (L-files) while the rigid body motion is stored on the Hydrodynamic Results Interface File (G-file), also termed Global Response File. The total wave elevation may optionally have been stored as additional information on the G-file (Wadam only).

This section presupposes basic knowledge in postprocessing complex results as covered in Section 3.4.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open. Open both the T-file (for the top level superelement) and the G-file by click and Ctrl+click. The G-file may have any of the formats SIF, SIU and SIN. Corresponding L-files are automatically opened. These may be either formatted or unformatted.
- Double-click the model name appearing in the Database window of Xtract (T*) and the model (optionally in sea) is displayed. The various parts (superelements and sea surface) are colour coded.
- Click the Result Cases tab and select a result case and an occurrence and double-click Elements/SURFACE-LOADS/NORMAL-PRESSURE.
- At this stage go to Section 3.8.1 and follow the procedure from the bullet item 'Relevant for Wadam ...'.

Figure 3.74 is a snapshot from an animation of a ship. The contours show the hydrodynamic pressure load.

Note: Do not open L-files by File > Open as these are automatically opened when the corresponding T-files are opened.

Note: The normal pressure load on a shell is positive when pointing in the direction of the element z-axis. For a hull with the z-axis pointing outwards the pressure values will thus be negative.

Note: For a superelement model viewing surface pressure loads should be restricted to having selecting the top level superelement. I.e. avoid opening a top level T-file, selecting a first or lower level superelement followed by presenting surface pressure loads.

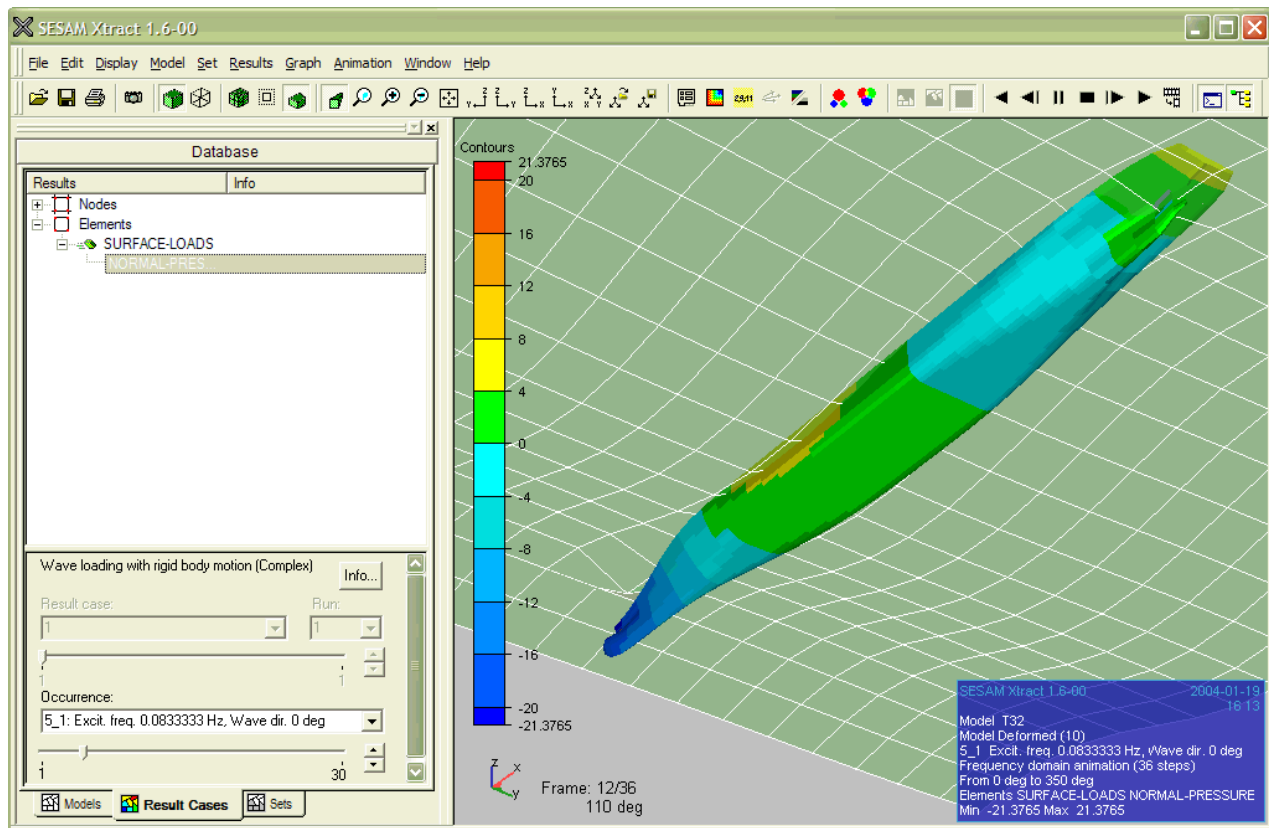


Figure 3.74 Animation of a ship (T+L+G-files) – pressure, motion and waves – from Wadam

3.8.3 Hydrodynamic Loads – Wadam/Wasim Results only (T+L)

Floating and fixed large volume structures are analysed in Sesam by the hydrodynamic analysis programs Wadam or Wasim. The hydrodynamic analysis produces frequency domain results. (Wasim performs the analysis in time domain but is capable of transforming the results to frequency domain.) Hydrodynamic pressure loads are stored on Loads Interface Files (L-files).

This section presupposes basic knowledge in postprocessing complex results as covered in Section 3.4.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open. Open the T-file (for the top level superelement). Corresponding L-files are automatically opened. These may be either formatted or unformatted.
- Double-click the model name appearing in the Database window of Xtract (T*) and the model is displayed. The various parts (superelements) are colour coded.
- Click the Result Cases tab and select a result case and an occurrence and double-click Elements/SURFACE-LOADS/NORMAL-PRESSURE.
- You may want to rotate and zoom to see the structure from below.

- Give Animation > Frequency animation to set up an animation (see Section 3.7 for general information on animation). Set the Skip by value and click OK. Setting up the animation may require some time depending on the size of the model and the Skip by value. Once the animation control buttons (see Figure 3.67) appear you may start the animation.
- Run the animation by clicking the animation buttons.
- During the animation you may rotate, zoom and pan, change part attributes and more. But to change result case and occurrence you must end the current animation and set up a new one.

Figure 3.75 is a snapshot from an animation of a ship. The contours show the hydrodynamic pressure load.

Note: Do not open L-files by File > Open as these are automatically opened when the corresponding T-files are opened.

Note: The normal pressure load on a shell is positive when pointing in the direction of the element z-axis. For a hull with the z-axis pointing outwards the pressure values will thus be negative.

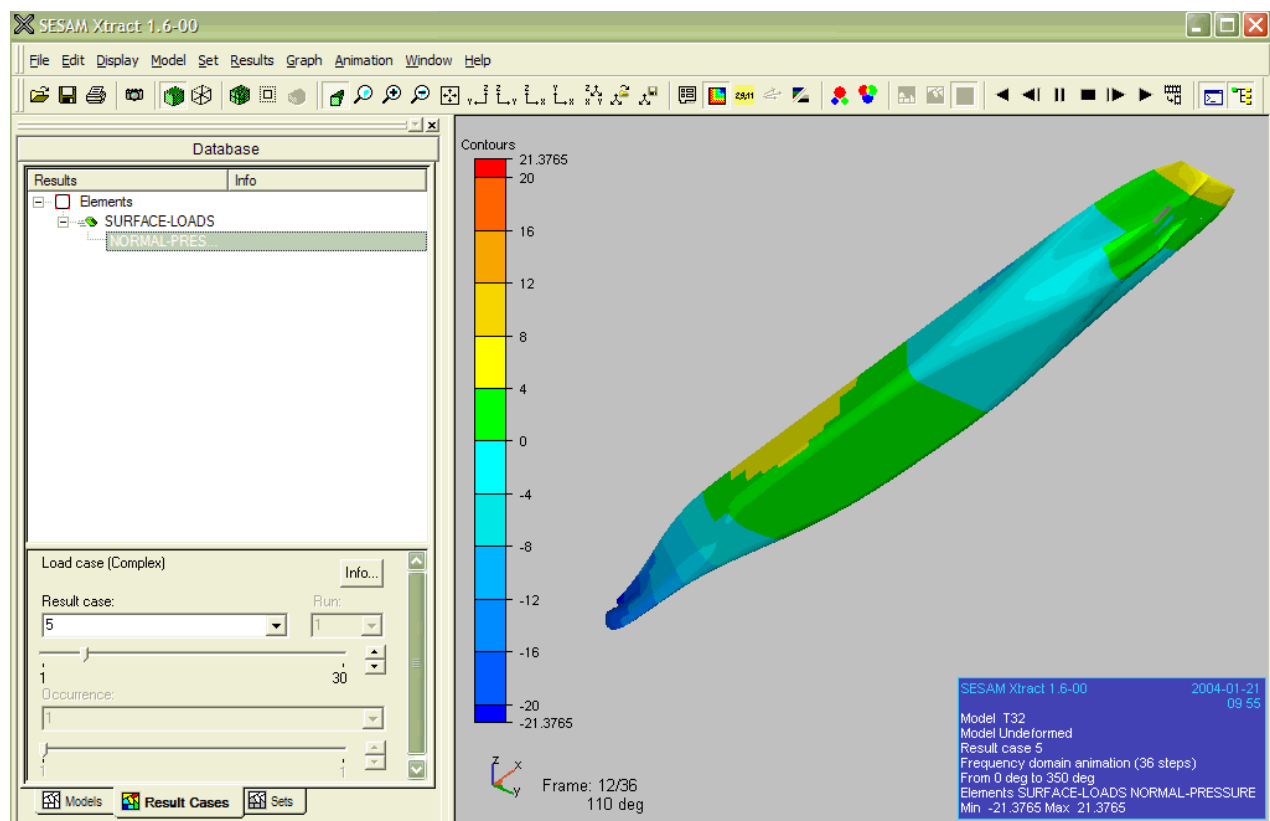


Figure 3.75 Animation of a ship (T+L-files) – pressure only – from Wadam

3.9 Hydrodynamic Loads and Motion – Wasim Time Domain Results

Ships and ship-like structures with forward speed are analysed in Sesam by the hydrodynamic analysis program Wasim. The analysis is performed in the time domain. The analysis model, the hydrodynamic pressure loads, the rigid body motion and the wave elevation (split into incoming and radiated waves) are stored in a VTF-file. Optionally, the VTF-file may also contain result components versus time or this data may be found in a separate VTF-file. In addition to this VTF-file you may for illustration purposes have a (superelement) model, a so-called ‘extra geometry’, in the form of a (set of) T-file(s) representing the superstructure. This superstructure will in an animation rigidly follow the motion of the Wasim analysis model (which normally is limited to ship hull).

Note: You should be familiar with the basics of using Xtract as covered in Section 3.1.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open and open the VTF-file. If relevant open also the (set of) T-file(s) representing the superstructure by Ctrl+clicking the (top level) T-file.
- Double-click the model name appearing in the Database window of Xtract and the model in sea is displayed. The various parts (of the ship model and sea surface) are colour coded.
- If relevant and needed you may position and scale the superstructure using the line-mode command ‘model extramodeltransformation’. Preferably, though, the superstructure is modelled in the proper coordinate system and with proper scaling making this command superfluous.
- Click the Result Cases tab, let the Result case be ‘Total wave elevation/Displacement’ and double-click a component of Elements/SCALARS, e.g. Radiation Pressure.
- Open the Part attributes dialog (Display > Part attributes), select the parts representing the Sea surface by click and Ctrl+click (if these parts do not have meaningful names they tend to be parts 1 and 3, uncheck the Visible option for these parts and click Apply to confirm this, then check the Visible option and click Apply) and change how to display the sea:
 - Switch off Contours (you may want to keep the Contours but this cannot be combined with transparent sea) and click Apply.
 - Switch on Mesh and click Apply.
 - Change the colour (to sea green or marine blue?) by clicking the Color button.
 - Make the sea Transparent, maybe also adjust the transparency Alpha value (lower right corner), 0 is fully transparent, 255 is opaque. Click Apply and Close.
- You may want to use the Part attributes dialog to change the display of the wet and dry parts of the hull as well as the superstructure too (for example: Smooth colouring? Mesh? Contours off and change colour for dry part and superstructure?). See Section 3.1.25 for more details on Part attributes.
- Give Display > Settings and click the colour box for the Background and change it to grey or white or another light colour of your liking. A light background is better in combination with a transparent sea.
- You probably need to zoom in to see the ship model properly.
- Present the rigid body displacement of the model and the wave elevation by Model > Deformed or the corresponding tool button.

- Give Animation > Result case animation to set up an animation (see Section 3.7 for general information on animation). You may want to skip time steps (increase the Step by value) and skip the first or last part of the simulation period (put Occurrences into the Except list box). Setting up the animation may require some time. Once the animation control buttons (see Figure 3.67) appear you may start the animation.
- Run the animation by clicking the animation buttons.
- During the animation you may rotate, zoom and pan, change part attributes and more.
- If the VTF-file also contain result components versus time then use Graph > VTF-Plot to open the VTF-file once more(!). A dialog named Plot Setup then appears in which you may select one or more time series results and click Apply. A graph window will then open in the upper right area of the display area. See Section 3.10 for more details on this.

Figure 3.76 is a snapshot from an animation of a ship in waves coming at angle from astern.

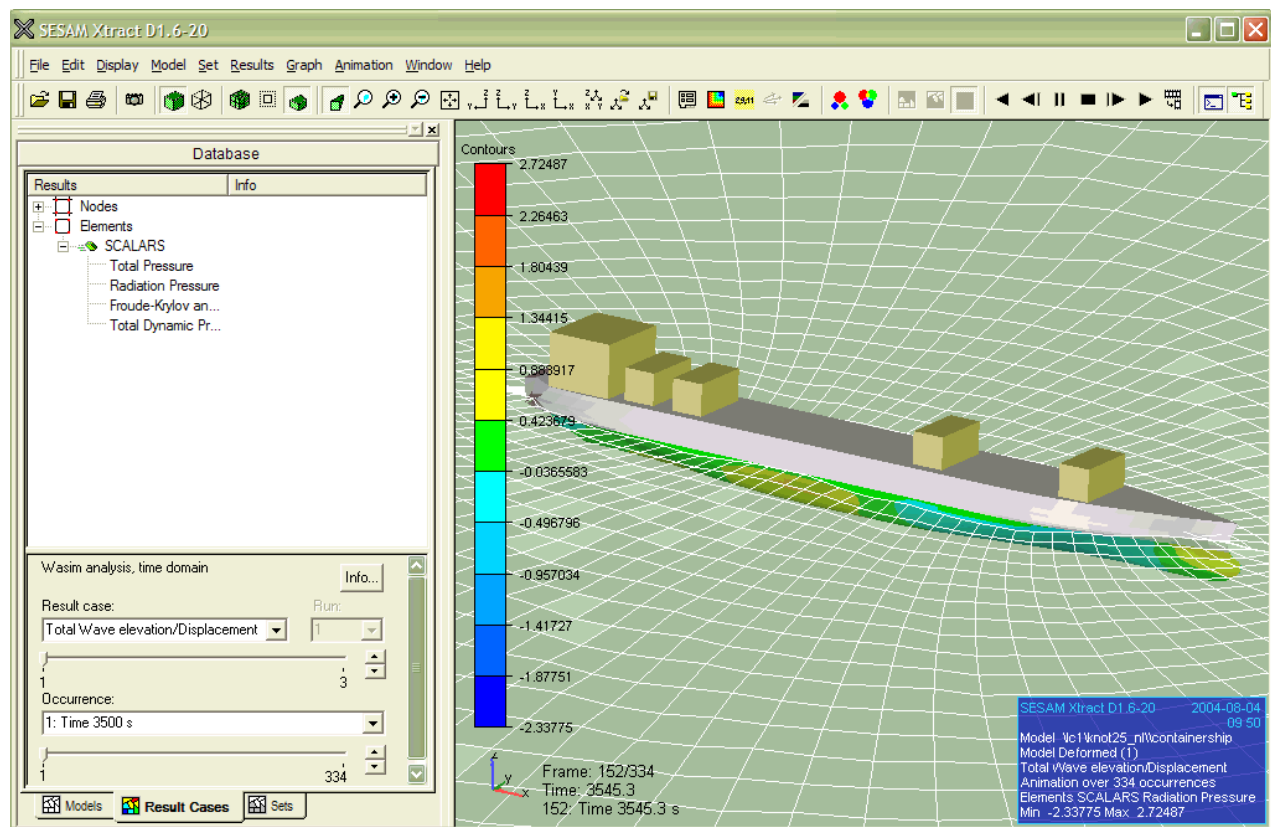


Figure 3.76 Animation of a ship (VTF-file) – pressure, motion and waves – from Wasim

Note: Do not scale the deformation if you have a superstructure included as the scaling only applies to the hull causing the two parts to disconnect.

Note: Tabulation as explained in Section 3.6 is not available for typical Wasim results like wave elevations and pressures.

Note: Unit conversion as explained in Section 3.1.21 is not available for typical Wasim results like wave elevations and pressures.

3.10 Animating Jacket Launching – Installjac Time Domain Results

A jacket launched from a barge is analysed in Sesam by Installjac. The analysis is performed in time domain. The analysis model and the rigid body motions of the jacket and the barge are stored in a VTF-file. Optionally, the VTF-file may contain time series results or this data may be found in a separate VTF-file.

Note: You should be familiar with the basics of using Xtract as covered in Section 3.1.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open and open the VTF-file.
- Double-click the model name appearing in the Database window of Xtract and the model in sea is displayed. The various parts (of the ship model and sea surface) are colour coded.
- Open the Part attributes dialog (Display > Part attributes), select the sea surface and change how to display the sea:
 - Change the colour (to sea green or marine blue?) by clicking the Color button.
 - Make the sea Transparent, maybe also adjust the transparency Alpha value (lower right corner), 0 is fully transparent, 255 is opaque. Click Apply and Close.
- You may want to use the Part attributes dialog to change the display of the structure too, for example changing colours. See Section 3.1.25 for more details on Part attributes.
- Give Display > Settings and click the colour box for the Background and change it to grey or white or another light colour of your liking. A light background is better in combination with a transparent sea.
- Give Animation > Result case animation to set up an animation (see Section 3.7 for general information on animation). You may want to skip time steps (increase the Step by value) and skip the first or last part of the simulation period (put Occurrences into the Except list box). Setting up the animation may require some time. Once the animation control buttons (see Figure 3.67) appear you may start the animation.
- Run the animation by clicking the animation buttons.
- During the animation you may rotate, zoom and pan, change part attributes and more.

Figure 3.77 is a snapshot from an animation of a jacket launching just after the jacket is free from the barge. The barge rocks back while the jacket sinks into the water.

An Installjac run may have created 2D series data, i.e. results component versus time, stored in the VTF-file containing the model or in a separate VTF-file. The 2D series data may be graphed in Xtract.

Open the VTF-file (which may be the same as the one containing the model) by:

Graph > VTF-Plot

Having opened the VTF-file with time series data you should select one or more of the series available: Select in the Available series list box, add them to the Selected series list box and click Apply (and Close but

not Close File). Use the Graph options dialog to modify the graph. You may for instance want to rename the curves by the Edit Legend button. See Section 3.1.15 for help on the graph feature. During an animation a vertical line will move through the graph in parallel with the model animation. See Figure 3.77.

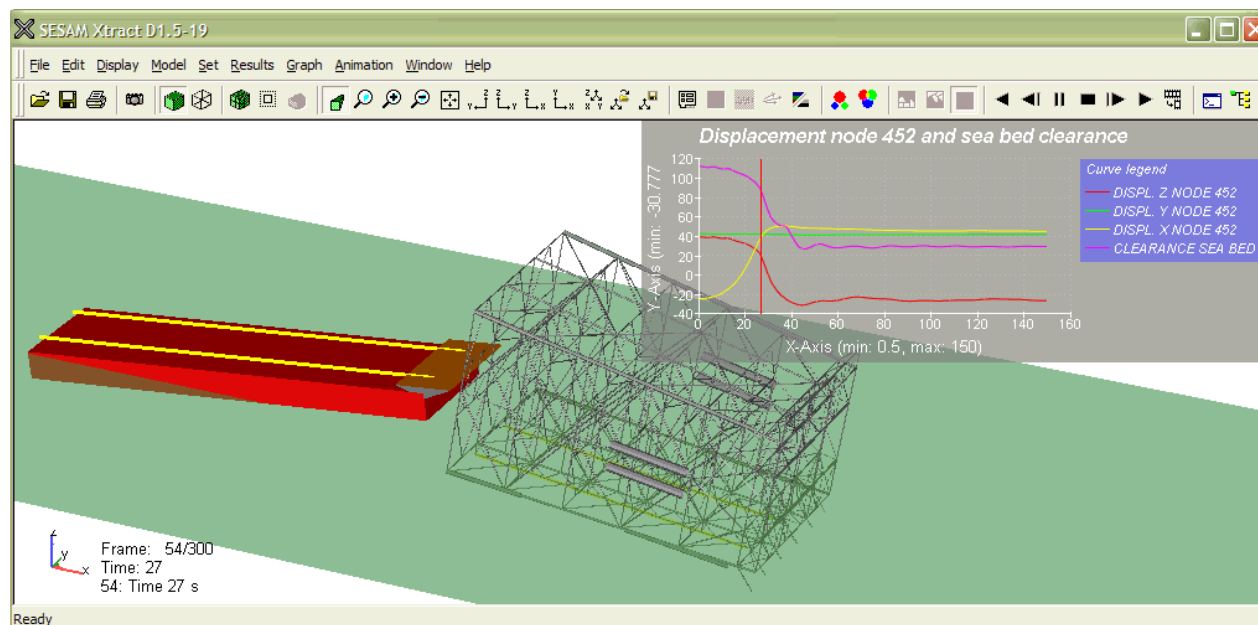


Figure 3.77 Animation of a jacket launching (VTF-file) – from Installjac

3.11 Riser and Mooring Analysis – Riflex Time Domain Results

Riser and mooring analysis in Sesam is performed by Riflex. The analysis is performed in the time domain. The analysis model and rigid body motion of the riser, mooring system and floater are stored in a VTF-file. Optionally, the VTF-file may also contain result components versus time or this data may be found in a separate VTF-file. In addition to the VTF-file you may for illustration purposes have a (superelement) model, a so-called 'extra geometry', in the form of a (set of) T-file(s) representing some additional structure. This additional structure will in an animation rigidly follow the motion of the Riflex analysis model.

Note: You should be familiar with the basics of using Xtract as covered in Section 3.1.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open and open the VTF-file. If relevant open also the (set of) T-file(s) representing the additional structure by Ctrl+clicking the (top level) T-file.
- Double-click the model name appearing in the Database window of Xtract and the model is displayed. The various parts are colour coded.
- However, as the risers and mooring lines are displayed with their true dimension they may hardly be visible (the lines are very thin compared with the whole system). Therefore, open the Part attributes dialog (Display > Part attributes), click Select all parts, uncheck the Expanded option and click Apply.

- Click the Result Cases tab and select a component of Elements SCALARS, e.g. Efficient axial force.
- Present the rigid body displacement of the model by Model > Deformed or the corresponding tool button.
- Give Animation > Result case animation to set up an animation (see Section 3.7 for general information on animation). You may want to skip time steps (increase the Step by value) and skip the first or last part of the simulation period (put Occurrences into the Except list box). Setting up the animation may require some time. Once the animation control buttons (see Figure 3.67) appear you may start the animation.
- Run the animation by clicking the animation buttons.
- During the animation you may rotate, zoom and pan, change part attributes and more.
- If the VTF-file also contain result components versus time then use Graph > VTF-Plot to open the VTF-file once more(!). A dialog named Plot Setup then appears in which you may select one or more time series results and click Apply. A graph window will then open in the upper right area of the display area. See Section 3.10 for more details on this.

Figure 3.78 is a snapshot from an animation of a riser and mooring system.

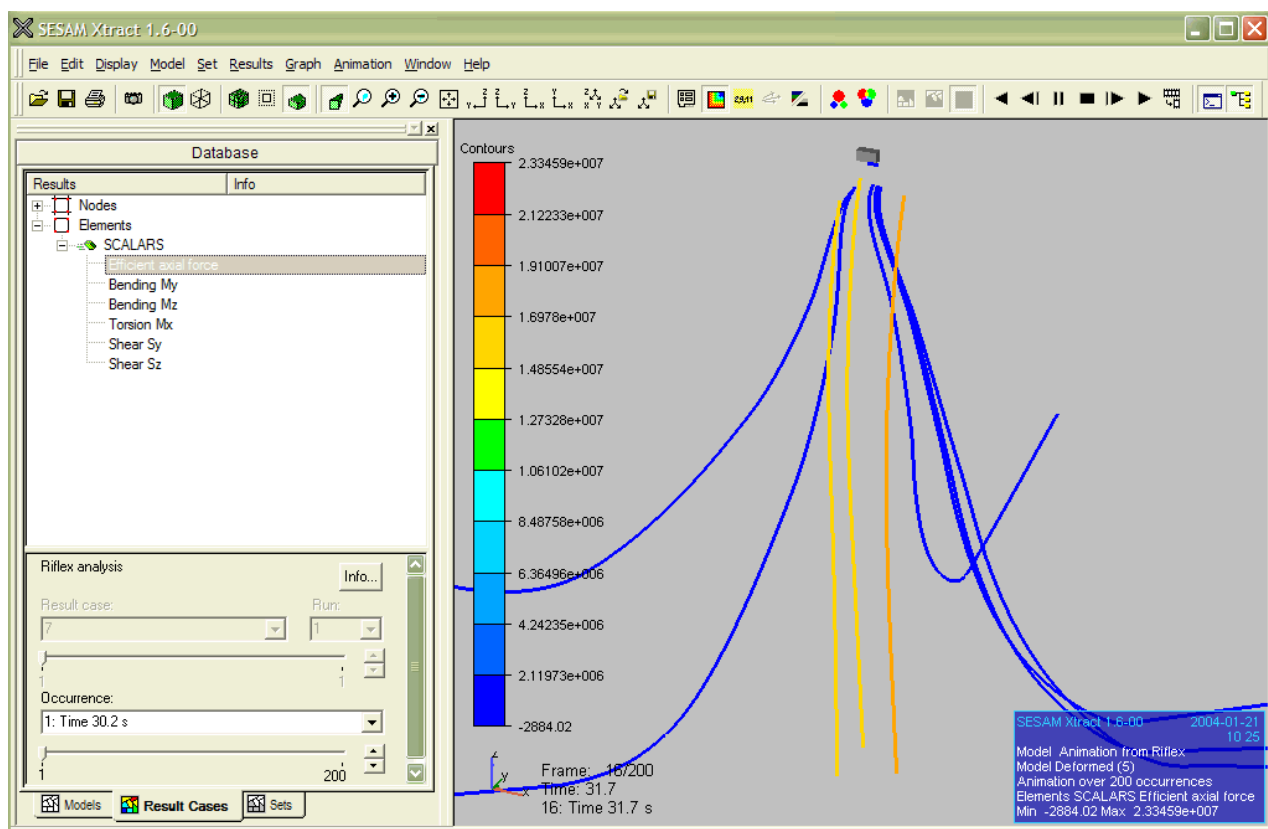


Figure 3.78 Animation of risers (VTF-file) – from Riflex

Note: Do not scale the deformation if your VTF-file includes a floater (having rigid body motion) as the scaling only applies to the risers/mooring lines (having nodal deformations) causing them to part from the floater.

3.12 Stochastic Fatigue – Stofat Results

Stochastic fatigue analysis of stiffened plate structures are preformed in Sesam by Stofat. The analysis model and fatigue lives are stored in a VTF-file.

Note: You should be familiar with the basics of using Xtract as covered in Section 3.1.

The procedure for presenting and animating such results is:

- Start Xtract and click File > Open and open the VTF-file.
- Double-click the model name appearing in the Database window of Xtract and the model is displayed.
- The VTF-file may contain results for more than one fatigue analysis. The various analyses will appear as colour coded sub-models of the top level model. Select one of the sub-models by double-clicking it.
- Click the Result Cases tab and double-click a component of Elements/SCALARS, e.g. a usage factor.
- You may want to add the mesh and rotate the model to see the areas of highest interest.

Note: Tabulation as explained in Section 3.6 is not available for typical Stofat results like usage factors and fatigue lives.

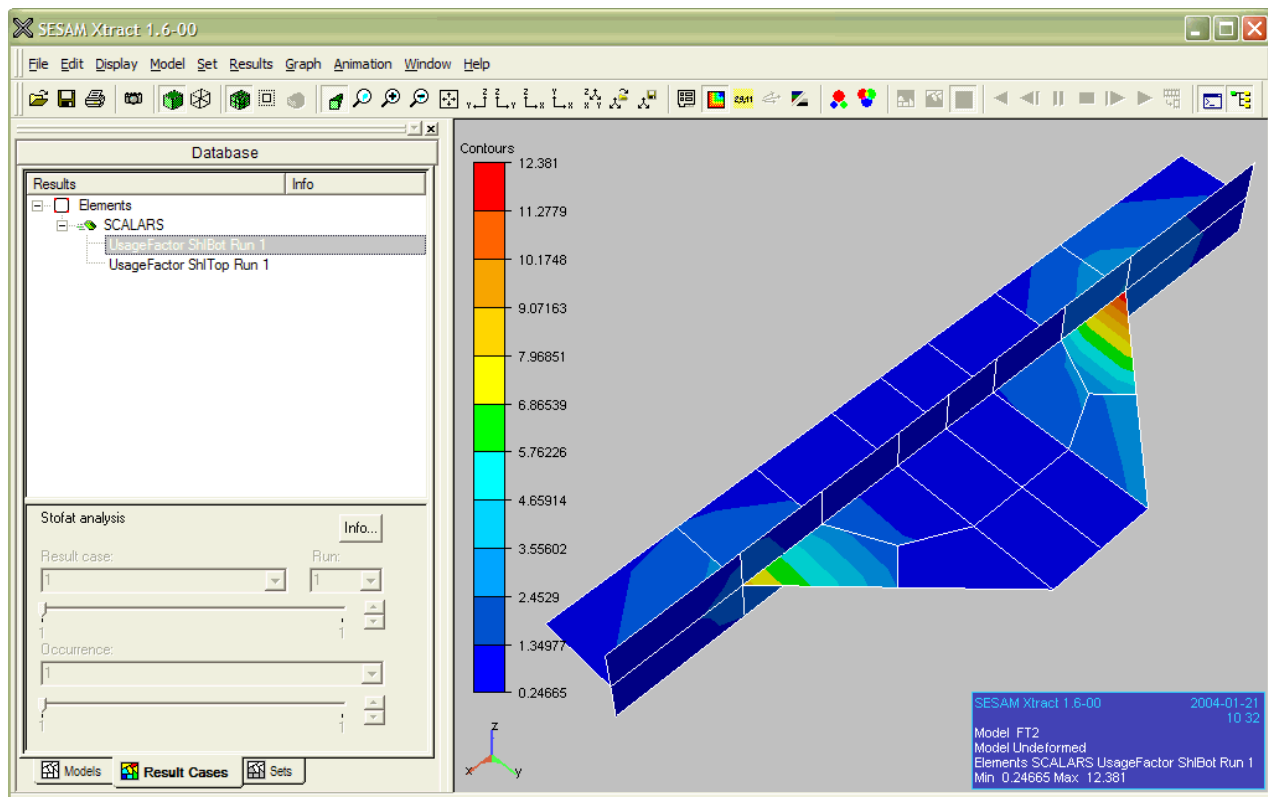


Figure 3.79 Presenting stochastic fatigue results (VTF-file) – fatigue lives – from Stofat

3.13 Exporting Data for Further Processing and Reporting

In the following are presented some features for exporting model with results to files for further processing by other software as well as for report generation. These features are:

- Exporting 3D model with results to a VTFx-file that in turn is opened in the free program Xtract Viewer for interactive 3D viewing. Alternatively, the VTFx-file may be opened in MS Word and PowerPoint through installation of the free GLview 3D Plugin. Read about this in Section 3.13.1.
- Exporting two 3D models to a common VTFx-file for viewing a sub-model on top of the global model in Xtract Viewer. Read about this in Section 3.13.2.
- Exporting data like 3D models with results, images, tables and single result values to a repository (database) belonging to the program Xtract Report Builder. This program is used for automatic generation of reports in for example MS Word and PowerPoint. Read about this in Section 3.13.3.

3.13.1 Exporting 3D Model with Results to VTFx-File for Viewing in Free Programs

Export a 3D model with a selected result component and result case (a display scene) to a VTFx-file by:

- File > Export to VTFx (or Ctrl+E)

This opens the Export to VTFx file dialog, see Figure 3.80, in which a Case name and a Case description may be entered. The exported VTFx-file may be opened in the free Xtract Viewer for interactive 3D viewing of the model with results. By use of the free GLview 3D Plugin embedded into MS Word and PowerPoint interactive 3D viewing of the model with results is also available in a Word report and a PowerPoint slideshow. The Save button saves the VTFx-file as a new file while the Append button adds the display scene to an existing VTFx-file.

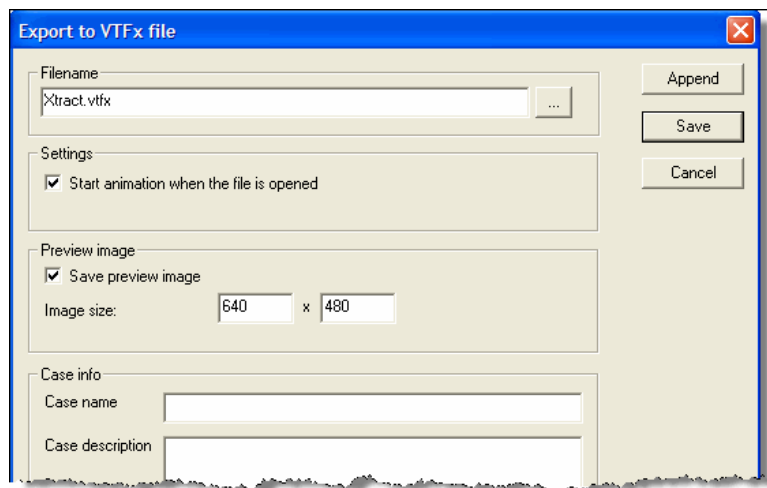


Figure 3.80 Exporting to VTFx-file for viewing

Checking 'Save preview image' is relevant only for PowerPoint and involves that an image of the model, of specified Image size if desired, is displayed initially before starting the slideshow. Checking 'Start animation when the file is opened' is also relevant for PowerPoint only and only when the display scene is an animation. I.e. an animation has been set up as described in Section 3.7 prior to exporting the VTFx-file.

Export of results to a VTFx-file allows a Sesam user to send a model with results to anyone, who after installing the free programs Xtract Viewer and/or the GLview 3D Plugin, can view the model with results in an interactive and Xtract-like user interface.

Xtract Viewer

Open the VTFx-file in Xtract Viewer, see Figure 3.81. Choose among the models saved by double-clicking in the Table of Cases pane to the left. In addition to rotating, panning and zooming in an Xtract-like graphical user interface the Xtract Viewer has several useful features like toggling display information (these buttons: ) , one to four viewports (this button: ) , showing detailed information about nodes and elements (press this button  to open a pane and then use this button  to select Element or Node followed by clicking elements/nodes), running an animation, and finally measuring distances (press this button:  and click two nodes). Read more about these and other features through the Help (F1). (In this help documentation read 'GLview Express' as 'Xtract Viewer'.)

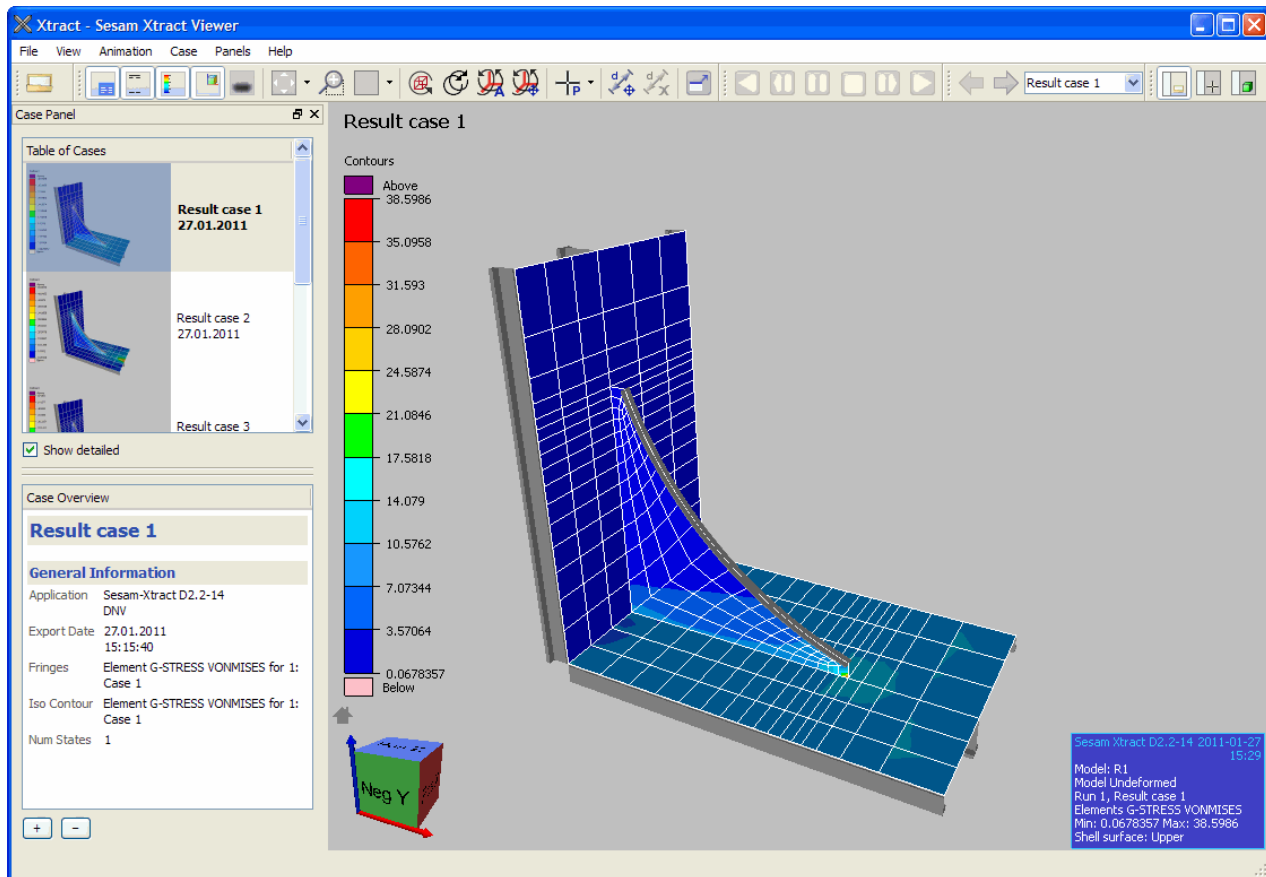


Figure 3.81 The free Xtract Viewer offers an Xtract-like GUI for viewing results

Note: Associate a VTFx-file with Xtract Viewer by starting the program and using **File > Preferences** to open the Preferences dialog. In this dialog click the Associate button.

GLview 3D Plugin

Having downloaded and installed the free GLview 3D Plugin a new tab appears in MS Word and PowerPoint with some new buttons including these two: Load VTFx and Insert. Use the former button to insert a frame for the graphics and, with this frame selected, use the latter button to load a VTFx-file. Then in Word and a PowerPoint slideshow you may interactively manipulate (rotate, pan, zoom) the 3D model with results in an Xtract-like graphical user interface as illustrated in Figure 3.82.

Note: In PowerPoint the interactive manipulation only works during a slideshow.

Note: Interactive manipulation in GLview 3D Plugin differs somewhat from Xtract. RMB is for rotation and mouse wheel for zooming as in Xtract. Panning, however, is done by LMB while in Xtract it is Ctrl+RMB. (Ctrl+RMB in GLview 3D Plugin toggles display of the model.)

Analysis results

- Result case 1
- Membrane von Mises stress
- Highest stress is 38
- Found in bracket toe

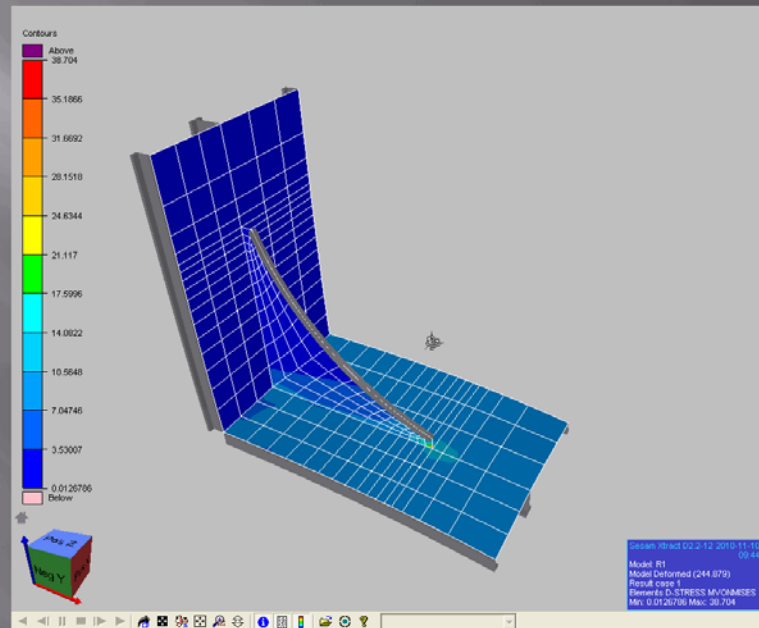


Figure 3.82 GLview 3D Plugin in PowerPoint offers an Xtract-like GUI during a slideshow

If the VTFx-file contains more than one model (by use of the Append button in the dialog shown in Figure 3.80) then choose among the models using the case selection field at the bottom of the GLview 3D Plugin window as shown in Figure 3.83.

During a slideshow in PowerPoint you may also open the VTFx-file in Xtract Viewer as a separate process by clicking the button: .

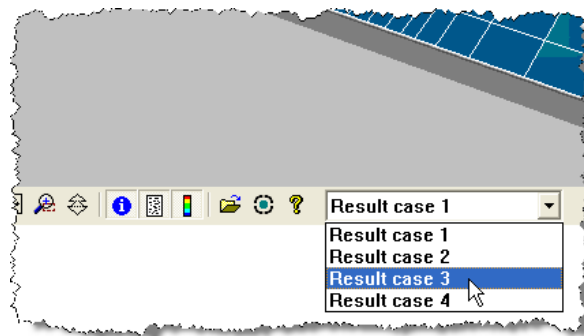


Figure 3.83 Select among cases in the GLview 3D Plugin window in the PowerPoint slideshow

3.13.2 Combined export of Two 3D Models to VTfX-File – Sub-Model plus Global Model

Sub-modelling analysis is a technique for refined and quick analysis of a detail of a ‘global’ analysis. It involves creating a model of the detail, a sub-model, and imposing prescribed (forced) displacement on the sub-model boundaries. These displacements are taken from the global analysis at positions corresponding to the sub-model boundaries. In Sesam the program Submod is used to extract such displacements from the global analysis and impose them on the sub-model boundaries.

It is a crucial quality assurance measure of a sub-modelling analysis to verify that the prescribed displacements of the sub-model boundaries are correct. This may be verified by comparing the deformations of the sub-model and global model.

Xtract allows such comparison by overlaying the two deformed 3D models. This is done by exporting the two 3D models to a common VTfX-file which in turn is opened by the Xtract Viewer. In this free program the sub-model is superimposed on the global model thus allowing interactive inspection of the overlaid deformed 3D models. Do this by following the procedure outlined below.

Open the global model results file in Xtract and select a result case. Switch on deformed model by Model > Deformed (or Ctrl+Alt+D). Use Model > Deformation scaling to select a suitable scaling factor, e.g. round up the program chosen factor. See the dialog to the left in Figure 3.84. Then export the deformed global to a VTfX-file by:

- File > Combined export

This opens the dialog shown to the right in Figure 3.84. Notice that when selecting Model type = Global model the model is by default drawn in surface mode using a blue colour.

Note: Any currently shown colour contour presentation of a result component will not be exported.

Note: At this stage do not check the Launch Sesam Xtract Viewer box. Give a file name and click Export.

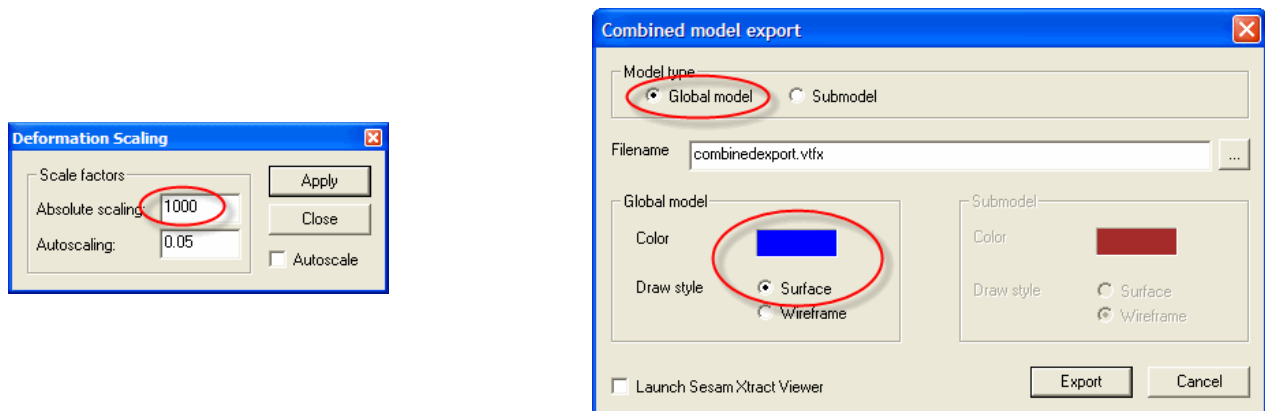


Figure 3.84 Combined export of global model to VTFx-file

Then open the sub-model results file (you don't have to exit Xtract) and select the same result case as the one selected for the global model. Switch on deformed model by Model > Deformed (or Ctrl+Alt+D) and use Model > Deformation scaling to select the *same* scaling factor as the one chosen for the global model. See the dialog to the left in Figure 3.85. Then export the deformed sub-model to the *same* VTFx-file as the one of the global model by:

- File > Combined export

This opens the dialog shown to the right in Figure 3.85. Notice that when selecting Model type = Submodel the model is by default drawn in wireframe mode using a marron colour. This is to enable distinguishing the two models when overlaid. At this stage check the Launch Sesam Xtract Viewer box. Make sure the file name is the one given for the global model and click Export.

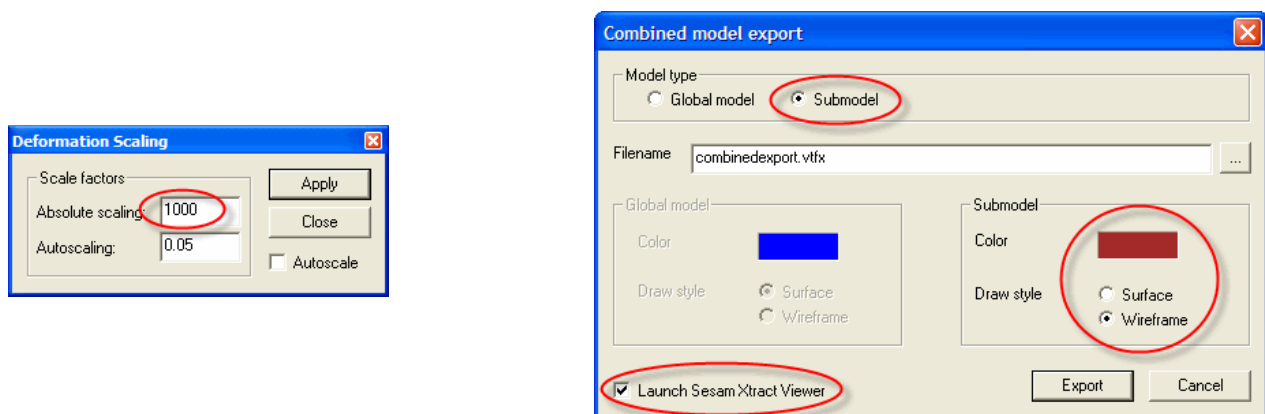


Figure 3.85 Combined export of sub-model to VTFx-file

The Xtract Viewer will now open, see Figure 3.86. Superimpose the sub-model on the global model (and not the other way around as this may produce a distorted perspective) by right-clicking it as shown in the figure.

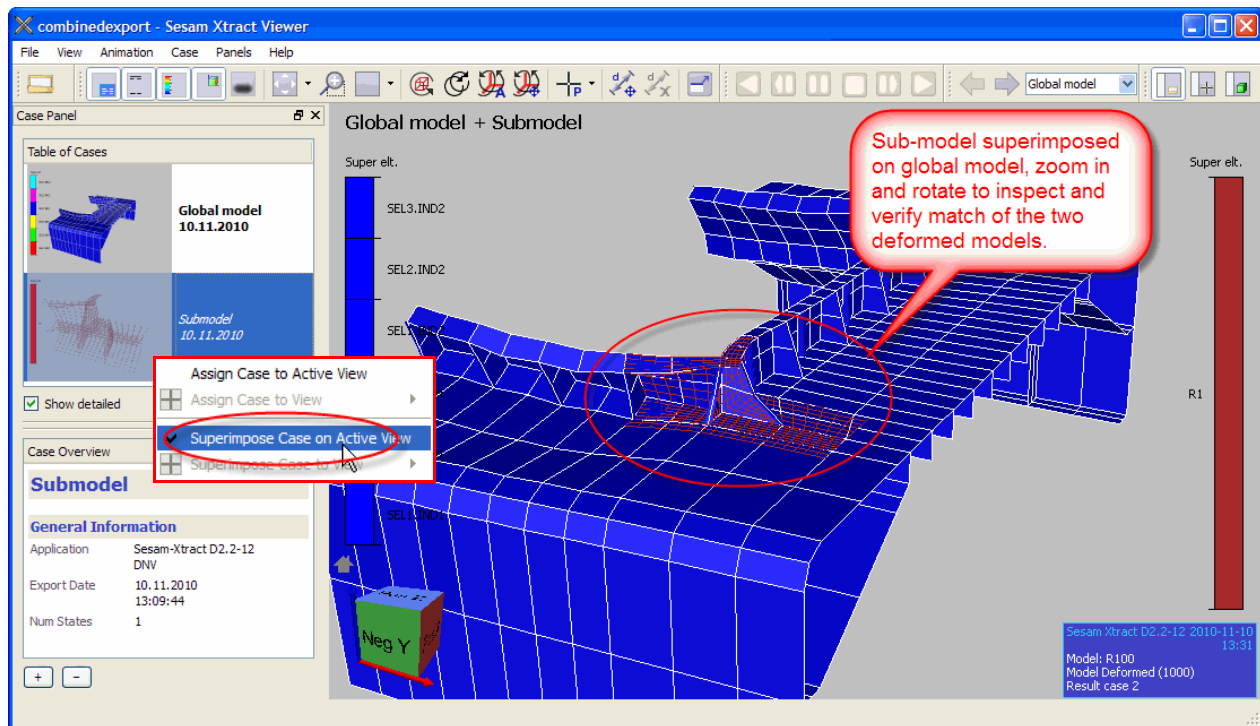


Figure 3.86 Xtract Viewer allows viewing sub-model superimposed on global model

Note: Associate a VTfX-file with Xtract Viewer by starting the program and using **File > Preferences** to open the Preferences dialog. In this dialog click the Associate button.

3.13.3 Automatic Report Generation by the Xtract Report Builder

The purpose of the Xtract Report Builder is to facilitate the creation of a report for an analysis performed in Sesam. The Xtract Report Builder is especially useful for creating standardised reports and report revisions. A new version or revision of a report may be created in mere seconds. The delay between introducing a change in the model and presenting a completely new report is practically no longer than what it takes to run the analysis anew plus doing the postprocessing in Xtract by reading the log of commands from the previous session.

The Xtract Report Builder is a separate program accompanying Xtract and is subject to a separate license, see Section 1.5.

The use of the Xtract Report Builder requires MS Office 2007 or later. You also need to install the software component GLview 3D Plugin that will integrate with MS Word and PowerPoint.

The process of using the Xtract Report Builder is illustrated in Figure 3.87 and works like this:

- The postprocessing of analysis results is performed as normal in Xtract.

- Graphics, graphs, tables and other data to include in the report are during the Xtract postprocessing saved (captured) to a repository (database). This repository belongs to the Xtract Report Builder which is automatically opened when such capturing is performed.
- The graphics, graphs and tables that are captured in Xtract are termed situations. A situation may optionally include a complete 3D representation of the model with results and not merely a static 2D picture. This allows interactive manipulation of the model in an Xtract-like GUI in the final report.
- Accompanying each situation are a title, a text and some parameters (termed field values). Examples of field values are name of the current result case, names of the current result position/attribute/component, maximum and minimum values of the current result component and upper/lower surface.
- Any number of situations may be captured in Xtract and all will appear in the Xtract Report Builder.
- In the Xtract Report Builder the situations may be deleted, reordered, made inactive and even edited to some extent.
- The final step is to create the report in e.g. MS Word or PowerPoint. The creation is based on a template that decides where all the graphics, graphs and tables – the situations – with their accompanying titles, texts and field values should go in the final report.
- The template determining the appearance of the final report is itself a Word/PowerPoint document. A previous report may thus with some editing be used as such a template.

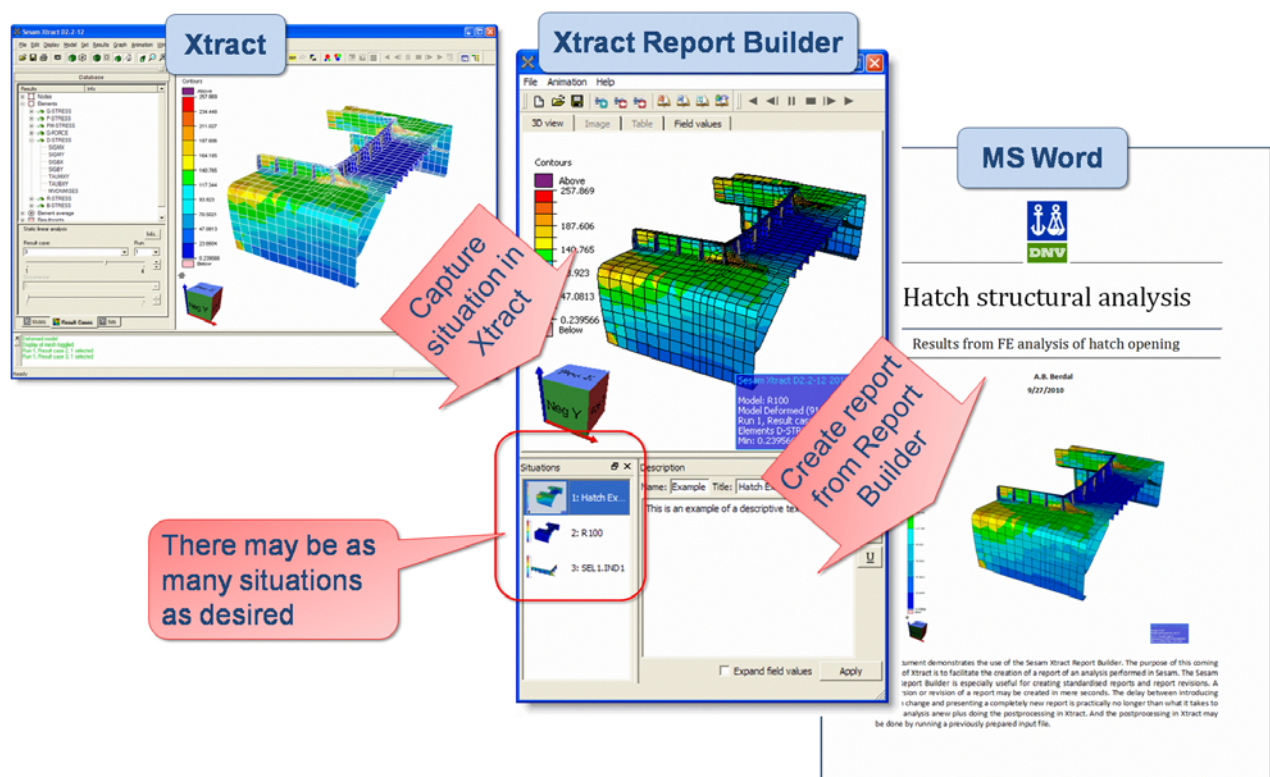


Figure 3.87 Xtract Report Builder process

Save a situation in Xtract to an Xtract Report Builder repository – a repx-file – by:

- File > Capture situation (or F4)

Or by:

- File > Quick Capture situation (or F5)

The former opens the dialog shown in Figure 3.88 for determining the contents of the situation to capture while the latter captures a situation based on the settings of the previous capture (no dialog will open).

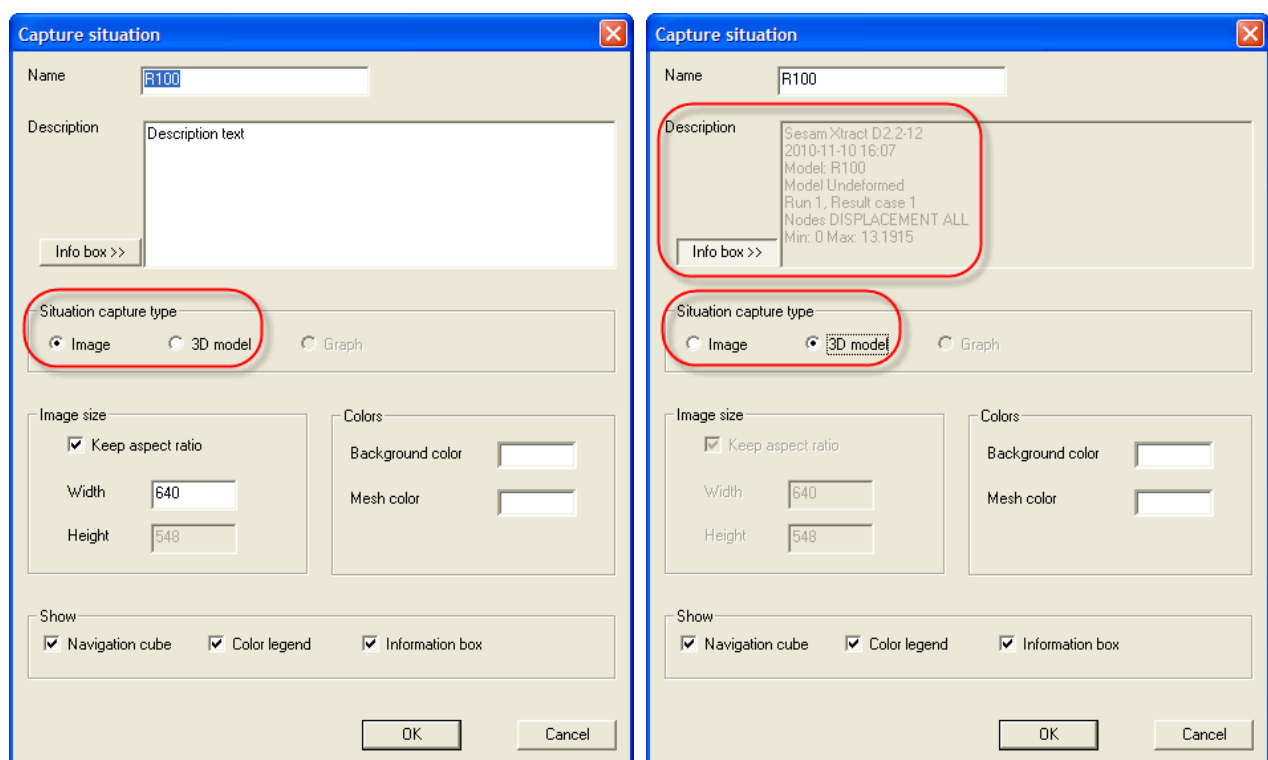


Figure 3.88 Capturing a situation for a report by F4

Capturing a situation involves storing the current display as either a 2D image or a 3D model according to the choice made in the Capture situation dialog to a repository (repx-file) and at the same time starting (if not already running) the program Xtract Report Builder shown in Figure 3.89. The figure shows a case where several situations have already been captured. Situation number 3 is the one currently selected. This situation is a 3D model as can be seen from the active tab with text '3D view'. For a situation containing an image the 'Image' tab will be active.

Figure 3.89 also shows the Xtract Report Builder when selecting the tab 'Field values'. These are parameters captured together with the 3D model or image. As can be seen these are name of the results file (database), i.e. the SIN-file, and when this was created, which run number and result case are shown, which position, attribute and component are shown, minimum and maximum values of the selected component and more.

In the Xtract Report Builder you may select any of the situations and inspect the model and the field values. If the situation is a 3D model you may also rotate and zoom it. The Title and Description that were given in the capturing process in Xtract may be modified. You may even use the bold, italic and underline buttons to highlight text in the Description.

You may right-click a situation as shown to the left in Figure 3.89 to move it up or down, to deactivate it (meaning it will not be exported to the report) and remove (delete) it.

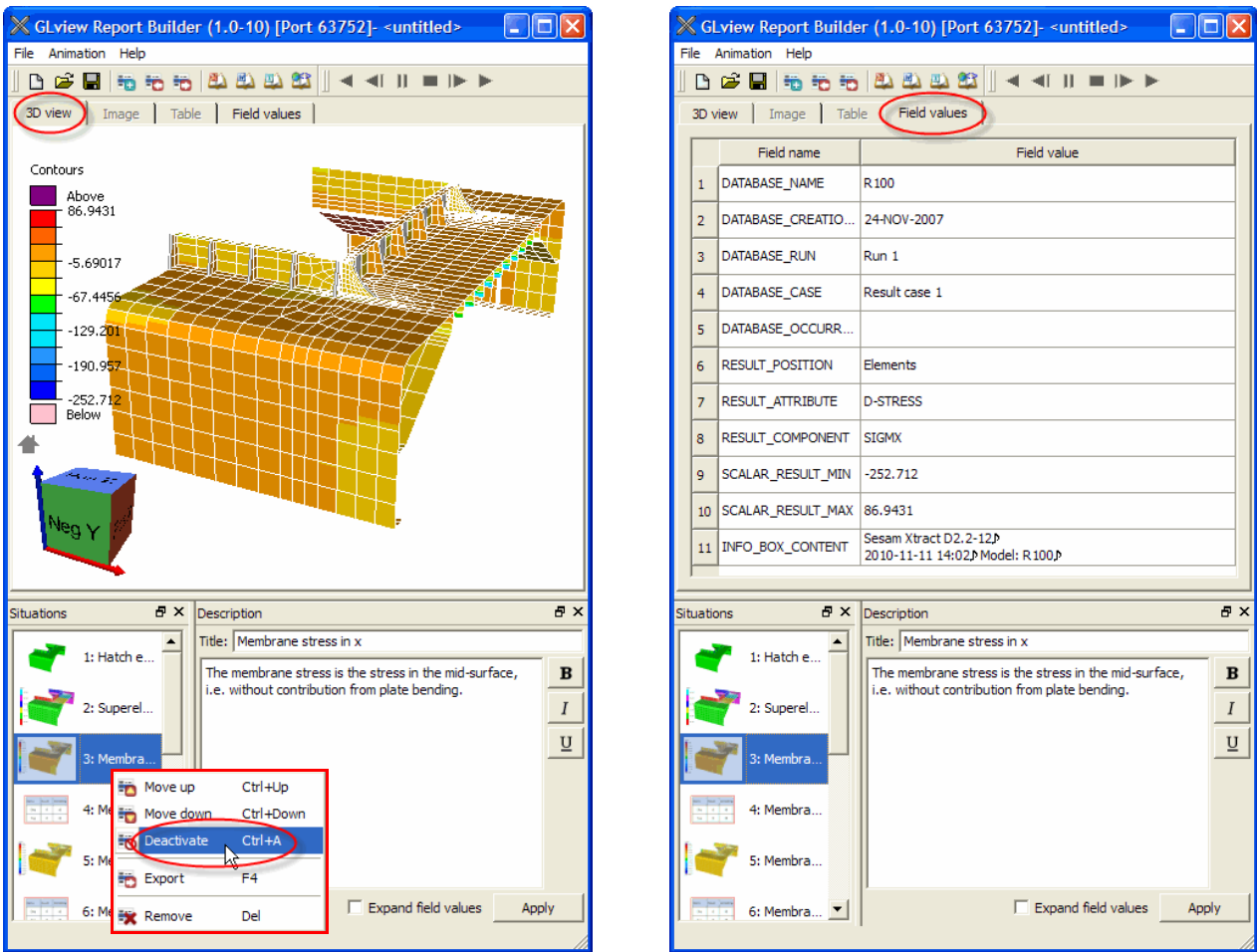


Figure 3.89 Xtract Report Builder

Provided you already have a suitable template you may then create the report by clicking either of the buttons , from left to right: PowerPoint, Word, Text and Html.

Read more about how to create automated reports in the Xtract Report Builder User Manual.

3.14 Notes on Graphing, Combining and Scanning

3.14.1 Graphing

Note: When graphing Item types Nodes (in Graph construction dialog, see Figure 3.18) care should be taken when selecting attributes other than displacements, velocities and accelerations (which are the only true nodal results data). If, for instance, Nodes/G-STRESS/SIGXX is graphed along an intersection between two plates where the values differ for the two plates due to difference in local (element) coordinate system then Xtract will choose the values belonging to the *lowest element number*.

Note: When graphing item types Beams (in Graph construction dialog, see Figure 3.18) beams on a line should be selected in the direction of their element x-axis or else the graphs of the individual beams will be flipped (individual beam graphs always follow the direction of the x-axis).

3.14.2 Combining and Scanning

Note: Both a combination and a scan result case are established for all components of the chosen attribute and for that attribute only.

Note: Unless a combination has been created for all available results by the All results option in the Combine dialog it may be extended with additional attributes (and all components of these) by the line-mode command 'combine add'. (There is no GUI equivalence to this command.) A combination including a complex result case cannot be extended.

Note: Combinations and a scan result cases are put in a run named User. The occurrence will always be 1.

3.15 Special Features

This section includes some special features.

3.15.1 Save Time for large Models by not Reading Beam Sections

Large models with a number of different beam sections, typically ship models with many different stiffeners, may demand quite some time reading the sectional data. Save time by skipping reading sectional data as follows:

- Start Xtract and open the results file by File > Open.
- Use Display > Read beam expansion to switch off reading beam sections, this is by default set to on.
- Select the model by double-clicking it in the browser.

Then when displaying the model the beams will be drawn as thick lines only (adjust the line thickness by Display > Beam width). Furthermore, the B-STRESS attribute will not be available.

Note: Display > Read beam expansion must be done *after* opening the results file and *before* selecting the model.

To revert to displaying the model with beam sections (and having the B-STRESS attribute available) you need to reopen the results file (the Read beam expansion will then automatically be switched back on) and then select the model.

3.15.2 Square Root of Sum-of-Squares Combination for Elements/G-STRESS

A specialized feature is available for combination of Elements/G-STRESS and Elements/D-STRESS components: As an alternative to a normal linear combination a square root of sum-of-squares (SRSS) type of combination may be performed. That is, the following combination is performed for each of the components:

$$\sigma_{\text{Combination}} = \sqrt{(f_1\sigma_1)^2 + (f_2\sigma_2)^2 + (f_3\sigma_3)^2 + \dots}$$

where f is the factor, σ is the stress component (axial and shear) and 1, 2, 3, ... are the result cases included in the combination.

Change Normal sum in the Combine dialog, see Figure 3.23, to Square root of sum-of-squares to perform this type of combination.

Note: Square root of sum-of-squares (SRSS) combination is available only for the linear components (i.e. not von Mises) of Elements/G-STRESS and Elements/D-STRESS.

Note: The stress components of an SRSS combination cannot be interpreted as true stresses. E.g. a result cases with SIGXX = 100 ‘SRSS-combined’ with a result case with SIGXX = -100 yields a SIGXX = 141 (100 times square root of 2) whereas the true stress produced by a normal combination would be zero (100 - 100). For this reason the von Mises stresses are not calculated for an SRSS combination.

Note: The square root of sum-of-squares combination cannot be used when one of the resultcases is complex (see Section 3.4.2).

3.15.3 Ship Overall Vibration Level

This section explains how Xtract computes the overall vibration level of a ship (or any other structure for which you may want to use this feature) as a sort of a norm of velocities taken from a dynamic forced response analysis. How to use this feature is also explained.

The computation is based on velocities computed in a dynamic forced response analysis in the frequency domain. The velocity results are thus complex. The results file will contain a certain number of (normally two or three) result cases. Each result case includes a number of frequencies (occurrences). The frequencies typically constitute a range around, e.g. $\pm 10\%$, a resonance frequency of the structure.

The procedure is as follows.

- 1 Scan each of the (2 or more) result cases as follows:

- Evaluate each occurrence (frequency) for the maximum absolute value (largest value no matter which sign) of each of the translational (X, Y and Z) components of the nodal velocities.
- Scan over all frequencies (or limited to a specified frequency range) of the result case for the maximum absolute value of each of the translational (X, Y and Z) components of the nodal velocities.
- The above two steps create three scan result cases (for X, Y and Z). (Each scan result case contains all components but only the component scanned for is of interest.)
- Scale each scan result case with a factor depending on the result case. This factor is user specified.
- Let the scaled scan result cases be termed $v_{\text{scan_x_i}}$, $v_{\text{scan_y_i}}$ and $v_{\text{scan_z_i}}$ where 'i' is the result case.

2 Compute the norms:

$$v_{\text{norm_x}} = \frac{1}{\sqrt{2}} \sqrt{v_{\text{scan_x_1}}^2 + v_{\text{scan_x_2}}^2 + v_{\text{scan_x_3}}^2 + \dots} \quad (3.1)$$

$$v_{\text{norm_y}} = \frac{1}{\sqrt{2}} \sqrt{v_{\text{scan_y_1}}^2 + v_{\text{scan_y_2}}^2 + v_{\text{scan_y_3}}^2 + \dots} \quad (3.2)$$

$$v_{\text{norm_z}} = \frac{1}{\sqrt{2}} \sqrt{v_{\text{scan_z_1}}^2 + v_{\text{scan_z_2}}^2 + v_{\text{scan_z_3}}^2 + \dots} \quad (3.3)$$

3 Add user specified constants to the norms:

The constants – one for each component X, Y and Z – are specified for previously defined sets. I.e. for one set the three constants $v_{\text{tip_x}}$, $v_{\text{tip_y}}$ and $v_{\text{tip_z}}$ are given. For another set three new constants are given and so on.

Add these constants to the norms to find the overall vibration levels:

$$v_{\text{overall_x}} = v_{\text{norm_x}} + v_{\text{tip_x}} \quad (3.4)$$

$$v_{\text{overall_y}} = v_{\text{norm_y}} + v_{\text{tip_y}} \quad (3.5)$$

$$v_{\text{overall_z}} = v_{\text{norm_z}} + v_{\text{tip_z}} \quad (3.6)$$

An example input is presented below. It is based on a frequency domain analysis with two result cases found under run number 2. Each of the result cases has a number of frequencies. The frequency range of the first result case is 290 - 310 Hz and the frequency range of the second result case is 320 - 340 Hz. The scaling factors for the scan result cases are 1.1 for result case 1 and 1.2 for result case 2. The constant $v_{\text{tip_z}}$ has value 1.12 for a set named SetA and 1.18 for a set named SetB. We assume that the sets SetA and SetB have already been created. Furthermore, we want to limit the frequency ranges to be considered for the two result cases to 300 - 308 Hz and 322 - 335 Hz, respectively. The input for computing the overall vibration level will be:

```
createuservector \
  "tip_v" \
  "SetA" 0 0 1.12 \
  "SetB" 0 0 1.18
overallvelocity \
  "v_overall" \
  "tip_v" \
  300 308 "2" "1" 1.1 \
  322 335 "2" "2" 1.2
```

As seen the constants v_{tip_x} , v_{tip_y} and v_{tip_z} to be added to the norms are stored in the new result case *tip_v* created by the command *createuservector*. The result case *tip_v* is referred to in the subsequent command *overallvelocity* that computes the overall vibration levels and stores the data in the new result case *v_overall*.

Both *tip_v* and *v_overall* will be found under run 'number' User. The *tip_v* result case contains the attribute VELOCITY under position Nodes with components X, Y and Z as entered (1.12 and 1.18 for sets SetA and SetB, respectively). The *v_overall* result case will contain the attribute VELOCITY under position Nodes. The attribute will include the components X, Y, Z, RX, RY and RZ being the main result of this feature. In addition, the following result cases will be created by the command *overallvelocity*: *v_overall_SCAN_X_1*, *v_overall_SCAN_Y_1*, *v_overall_SCAN_Z_1*, *v_overall_SCAN_X_2*, *v_overall_SCAN_Y_2* and *v_overall_SCAN_Z_2*. These correspond to the terms $v_{scan_x_i}$, $v_{scan_y_i}$ and $v_{scan_z_i}$ above ($i = 1$ and 2). Also the result case *v_overall_NORM* containing v_{norm_x} , v_{norm_y} and v_{norm_z} above will be created.

3.16 Working with Line-Mode Commands and Command Input Files

If you want to enter commands through the keyboard (line-mode commands) then click anywhere in the Command and message window (see Figure 3.2) and start typing. Pressing Enter will execute a complete command (the backslash character before Enter allows an incomplete command to be continued). A command reference is found in Chapter 5. See the beginning of Chapter 5 (page 5-1) for details on how to enter commands.

There are certain features that are only available through line-mode commands (e.g. the 'result deletecase *name-of-case*' command).

Whether you use the graphical user interface or enter line-mode commands your actions will be logged in a file. Depending on how Xtract is started the name of the log file will be *Xtract.jnl* or *Xtract#.jnl* where # is the (top level) superelement number. You may stop logging by:

File > Stop logging commands

You may start logging in a new file (and stop logging to the current file, if any) by:

File > Start logging commands

The appearance of a log may be:

```
open sin "<path of file>R1.SIN"
model select R1
display mesh
result select "Elements" "G-STRESS" "VONMISES"
eye isometric + - +
plot GIF "Plot-stress-case1.GIF" 656 576 white "<header text>" 2 "<footer text>" 0
result selectcase "1" "3" "1"
plot GIF "Plot-stress-case3.GIF" 656 576 white "<header text>" 2 "<footer text>" 0
```

The first couple of lines above opens the results file and displays the (top level) superelement. The 3rd line adds the FE mesh to the display. The 4th line presents the component Elements/G-STRESS/VONMISES. Thereafter, a new view direction is set and a plot is produced. Then result case 3 (run 1, result case 3, occurrence 1) is selected and another plot is produced.

If you want to create an input file to rerun the above session this can be done in two ways as described in the subsequent sections.

Note: Xtract always logs result positions, attributes and components as well as result case identifications plus all text strings in double-quotes. In a Command input file the double-quotes may be skipped unless the word/string includes spaces.

3.16.1 Reading a Command Input File from inside Xtract

The above interactive session may be repeated by creating a file, named for example Input.jnl, with the following content:

```
model select R1
display mesh
result select Elements G-STRESS VONMISES
eye isometric + - +
plot GIF Plot-stress-case1 1000 1000 white "<header text>" 2 "<footer text>" 0
result selectcase 1 3 1
plot GIF Plot-stress-case3 1000 1000 white "<header text>" 2 "<footer text>" 0
```

See notes on result case identification in Section 3.16.2 and details on how to enter line-mode commands in Chapter 5, page 5-1.

The plot commands have been revised somewhat by increasing the plot file size to 1000 by 1000 pixels thereby creating plots of higher resolution.

Run Xtract using the input file by starting Xtract, opening the results file (the command for opening the results file could very well have been part of the input file) and giving:

File > Open (or click the tool button )

The Open dialog, see Figure 3.90, appears. In the Files of type box select Xtract Journal Files (*.jnl) and see that all files in the folder of type .jnl are listed. Select the file Input.jnl and click Open. The file will then be read into Xtract and processed.

Note: If a Command input file contains an error then Xtract stops the processing of commands and points to the erroneous line-mode command.

Note: You may also use cut and paste to drop several commands from a text editor into Xtract.

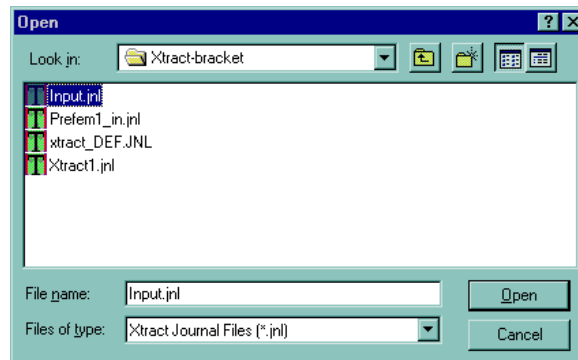


Figure 3.90 The Open dialog for reading a Command input file

3.16.2 Notes on Result Case Identification

- Note:** Even in cases where there is only one run number and one occurrence (e.g. a static analysis) a result case is always referred to by the full identification: Run number, result case and occurrence. This concerns the log as well as a Command input file.
- Note:** The run number, result case and occurrence are strings that in general are enclosed by double-quotes: "2" "4" "3: Excit. freq. 5.3 Hz". But if a string does not include blanks the double-quotes may be skipped: 2 4 "3: Excit. freq. 5.3 Hz".
- Note:** Sub-string matching is performed for strings. The occurrence "3: Excit. freq. 5.3 Hz" may be referred to as "3:" (or 3: since there are no blanks) or "5.3 Hz".
- Note:** The colon in the sub-string "3:" is normally required to match uniquely an occurrence of frequency domain results as "3" alone will easily match a number of occurrences (it may occur in the frequency value). The full identification "2" "4" "3: Excit. freq. 5.3 Hz" is therefore normally best referred to as 2 4 3:.

4 EXECUTION OF XTRACT

4.1 Program Environment

Note: Xtract is available on Windows only.

Go to Section 3.1 for information on how to start Xtract.

Xtract may be started with some command line arguments that will control the execution, see Section 4.1.2. This is mostly relevant when starting Xtract from another program.

4.1.1 Files used by Xtract

Xtract may open and read several files as described in Section 2.1. Xtract does not modify these files.

Xtract will open its own database (model file) named Xtract#.mod and a corresponding index file named Xtract#.idx where # is the (top level) superelement number. Combinations, scan result cases and scaled complex result cases will be stored on this database. Sets defined in Xtract (as opposed to sets read from files) are also stored on the Xtract database.

When opening a new results file within the same Xtract execution then the current database will be closed and a new opened on the directory of the new results file.

Commands (both actions through the GUI and line-mode commands) may be logged to a user given file name. See File > Start logging commands, the equivalent line-mode command 'log on *file-name*' and the command line argument /log.

Line-mode commands may be read from a Command input file. See File > Open, the line-mode command 'open jnl *filename*' and the command line argument /command-file.

Files containing tabulated data may be saved (see 'Model > List model data' and 'Results > List results') and plots may be saved to various formats.

3D models may be saved to a VTFx-file by the commands File > Export to VTFx and File > Combined export. These features are described in Section 3.13.1 and Section 3.13.2, respectively.

4.1.2 Command Line Arguments

The command line arguments are:

/eye-x=value	X-coordinate of an initial viewpoint
/eye-y=value	Y-coordinate of an initial viewpoint
/eye-z=value	Z-coordinate of an initial viewpoint
/software	Do not use graphics accelerator
/hardware	Use graphics accelerator installed, if any
/command-file=filename	Read the specified Command input file.
/log=filename	Log commands to the specified file.
/nolog	Disable logging from start. (By default logging is performed to the file Xtract#.jnl.)
/pixelformat=value	Force Xtract to use a certain pixel format rather than the default format. This command line argument should only be used when there are problems with the graphics driver as the pixel format value is highly dependent on the graphics driver. Therefore, use only after consulting with the Sesam support.

Command line arguments should be preceded either by a slash (as shown above) or by a minus sign. The argument name, which may be abbreviated as long as it is unique, should follow immediately after the slash (or minus). The equal sign and qualifier should follow immediately after the argument. One or more spaces are used to separate the individual arguments.

4.1.3 Registry Settings

Xtract saves certain settings in the user registry:

- In HKEY_CURRENT_USER\Software\DNVS\Xtract\Layout are stored:
 - Position and size of the Xtract window
 - Maximize/Minimize/Restore status
- In HKEY_CURRENT_USER\Software\DNVS\Xtract\ResultListing parameters related to tabulation of results as described in Section 3.6 are stored.
- In HKEY_CURRENT_USER\Software\DNVS\Xtract\Settings are stored:
 - Number of significant digits (Display settings dialog)
 - Element shrink factor (Display settings dialog)
 - Font size (Display settings dialog)
 - Undeformed wireframe colour (Display settings dialog)
 - Background colour (top/bottom/gradient) (Display settings dialog)

- Label filtering (show all or a maximum number of labels) (Labeling dialog)
- Label background box shown or not (Labeling dialog)
- SCANINDEX options (Labeling dialog)
- In HKEY_CURRENT_USER\Software\DNVS\Xtract\Recent File List are stored:
 - Recently opened files
- In HKEY_CURRENT_USER\Software\DNVS\Xtract\CaptureSituation are stored:
 - Settings of the Capture situation dialog, see Figure 3.88.

Note: The settings in HKEY_CURRENT_USER\Software\DNVS\Xtract\Settings are set back to default by File > Reset settings.

Note: The settings in HKEY_CURRENT_USER\Software\DNVS\Xtract\Layout can only be reset to default by deleting these registry settings: Start > Run, type 'regedit' and hit Return, expand the branch and delete the parameters named 'WindowBottom', 'WindowLeft', etc.

This involves that all these settings are stored when exiting Xtract and they take effect when restarting the program with the same user and PC.

4.2 Program Requirements

4.2.1 Execution Time

Xtract utilises the graphics accelerator OpenGL if such is installed. The speed of graphical manipulation (e.g. animation) will improve with this hardware. Utilisation of the graphics accelerator may be switched off by the command line argument /software, see Section 4.1.2.

Execution time is for most commands limited to a few seconds. Depending on the hardware and the model size creating combinations and scan result cases may require some time though. The progress bar in the lower right corner of the Xtract window indicates the progress of the combine and scan commands. Also notice that the Command and message window displays execution time of various operations. Setting up and running animations may also be slow on computers with modest performance.

Note: Reading large models with many beam stiffeners, e.g. a ship model, may require some time due to the amount of beam sectional data. It is selecting the model (reading data and displaying the model) and not opening the results file that takes time. To reduce this time reading the beam sectional data may be skipped, see Section 3.15.1. This involves that beam sections cannot be displayed. Moreover, the B-STRESS attribute will not appear as data for computing this result type is unavailable.

4.2.2 Storage Space

The database (model file) and index file of Xtract are initially empty. When combinations, sets, etc. are created the size of the model file will increase. Except for large models for which many combinations, sets, etc. are created the size will be modest.

4.3 Program Limitations

Xtract employs the general high-performance graphics software OpenGL for presenting results with filled contours. This has a couple of consequences as explained below that may for the keen observer be somewhat confusing. One should bear in mind, however, that the accuracy of an analysis normally does not allow an interpretation at a very detailed level (like gradients within an element) anyway. Consequently, the matters discussed in the notes may in certain cases cause some bewilderment but it should have no practical significance.

Note: To facilitate support for large result variations in an element the positioning of the filled contours may deviate from the exact position when the user manually enters the contours, e.g. by the Values mode in Result settings. This is due to the limitation of texture size supported by the graphics board as well as the linear interpolation scheme for texture coordinates of OpenGL.

Note: OpenGL handles the task of linear interpolation between the four corner values of quadrilateral elements. As a bi-linear field cannot be established between four corner values (for the same reason as a plane through four arbitrary points cannot be found) an assumption must be made. In other words, there is not one definite way of interpolating the result values (e.g. a stress component) inside a quadrilateral element. This has the consequence that the exact position of the contour lines inside an element may depend on the view position. In certain situations this gives an abrupt change of contours when the model is slightly zoomed or rotated.

Note: Opening a database (model file) with a newer version of Xtract than the one creating it will in most cases work. But in certain cases the data will be inconsistent. As a general rule it is therefore best to save the command log (journal) file from a session and use this as input in a later session using a new program version.

Note: There is a limitation to how many combinations, sets, etc. that can be defined in Xtract. This limitation is related to the size of the model as explained below.

$$\text{NSET} + \text{NCOMB} \times \text{NSEL} \leq 500,000$$

where:

NSET is number of sets defined

NCOMB is number of combinations, 'combine add's', scan result cases and scaled complex result cases

NSEL is number of first level superelements for which the combinations, etc. have been created for

Note: A maximum of 3125 result cases on the results file may be handled.

Note: Names of sets, result combinations, scan result cases and scaled complex result cases are limited to 80 characters.

Note: Avoid shrinking elements (Display > Shrunk mesh) while showing results with Contour type Lines (selected through Results > Settings) as these contour lines will not be shrunk (they will not follow the elements). Contour type Filled is, however, properly shrunk.

See the Status List for possible other program limitations.

5 COMMAND DESCRIPTION

This chapter describes the Xtract line-mode commands, i.e. commands entered by the keyboard through the Command and message window; see Figure 3.2. Commands deemed relevant only through the GUI are excluded, e.g. window docking/undocking, print preview, making annotations etc. See Section 3.16 for information on command logging and command input files.

Details on the command syntax and how to enter commands:

- **Keywords:** All commands consist of one or two keywords. Examples of one-keyword commands are plot, scan, combine and evaluation. Examples of two-keyword commands are ‘display mesh’, ‘model deformed’ and ‘result select’.
- **Parameters:** The one or two keywords are followed by one or more parameters that may be words, names (character strings) or numeric values. Examples in which the parameters are underlined are ‘plot gif plot1’, ‘scan maxvonm max Elements G-STRESS VONMISES ...’, ‘model select R1’, ‘result select-case 1 3 1’, ‘result select Nodes DISPLACEMENT ALL’.
- The keywords and parameters are separated by one or more blanks.
- **Backslash continuation:** A command may be continued by the backslash character. For example:

scan scan1 max Elements G-FORCE NXX 1 1 1_1: criterion 1 1 1 1_2: criterion 1 1 1 1_3: criterion 1
may be split into several lines:

```
scan \  
scan1 \  
max \  
Elements G-FORCE NXX \  
1 1 1_1: criterion 1 \  
1 1 1_2: criterion 1 \  
1 1 1_3: criterion 1
```

Note that comments are not allowed in between lines belonging to the same command.

- **Truncate keywords:** The keywords may be truncated as long as they are unique. Truncate for example 'plot gif plot1' to 'pl gif plot1'. A two-keyword command is seen as a whole meaning that only the latter keyword may be truncated. Truncate for example 'result selectcase "2" "4" "3: Excit. freq. 5Hz"' to 'result selectc 2 4 3:' (see items on skipping double-quotes and sub-string matching below).
- **Tab-completion:** Having typed one or more characters of a keyword pressing the Tab-key fills in the remainder of the keyword or all characters until the string branches off into alternative keywords. For example: type 'd', press the Tab-key and the program fills in characters so that it reads 'display'. This is, however, not a unique keyword. Typing 's' and pressing the Tab-key gives 'displaysettings' which is unique. Or typing 'i' and pressing the Tab-key which gives 'displayinfo'.
- **Guidance in Status bar:** Once the keyword(s) of a command are unique the Status bar at the bottom of the Xtract window will provide guidance on the parameters to enter. Note that while this manual uses italic font for parameters (*parameter*) the Status bar encloses parameters in 'greater/less-than-brackets' like this: <parameter>.
- **Double-quotes when spaces:** Character strings entered for parameters, e.g. names of sets, result cases and occurrences, must be enclosed by double-quotes if they include spaces. Examples: The set name "Upper web", the combination "Combination North A" and the occurrence "3: Excit. freq. 5Hz". The command log will always enclose words as well as strings in double-quotes even when there are no spaces.
- **Sub-string matching:** Sub-string matching is performed for command words (Nodes, Elements, G-STRESS, SIGXX) and name strings (names of sets, result cases, occurrences). Examples: 'Elements G-STRESS SIGXX' may be referred to as 'El G-STRE XX'. The occurrence "3: Excit. freq. 5 Hz" may be referred to as "3:" (or 3: since there are no blanks) or "5 Hz".
- **Exact match vs. sub-string match:** An exact match of a string will take priority over a sub-string match. An example: There are two result cases: 'combine' and 'comb' sorted in this order. Alt. 1: The string 'comb' is given. This is a sub-string of the first result case 'combine' but it matches the second result case 'comb' exactly. The second result case 'comb' is therefore selected. Alt. 2: The string 'com' is given. This is a sub-string of both result cases 'combine' and 'comb'. The first result case 'combine' is therefore selected.
- **Comments in input:** Comments may be entered by preceding the text with either of the following: exclamation mark (!), hash symbol (#) and percentage sign (%).

Commands are described in Section 5.1 through Section 5.7 as follows:

- The commands are described in a logic rather than alphabetic order: They are described in sub-sections named after the GUI menu commands to which they belong.
- Commands (keywords and parameter words) are in the command description written in normal font while parameters that can take different values are written in *italic* font.
- Braces and a vertical bar are used for alternatives of which one must be selected: {nodes | elements}
- Brackets are used for optional input: [text]
- Brackets and a vertical bar are used for alternatives of which none or one can be selected: [on | off]

There are seven categories of commands: (1) File, (2) Display, (3) Model, (4) Set, (5) Result, (6) Graph and (7) Animation. These are tabulated with their subcommands in Table 5.1 and explained in detail in Section 5.1 through Section 5.7.

Table 5.1 Overview of line-mode commands

Command category	Subcommand (click link)	Explanation
File Commands	<u>Open</u>	Open file(s) of specific type. In the case of JNL process the file.
	<u>Source</u>	Open and process a Command input file (.JNL). Use Open instead.
	<u>Plot</u>	Plot current view.
	<u>Plotall</u>	Plot all (2 or 4) viewports.
	<u>Print</u>	Print current view.
	<u>Exportvtfx</u>	Export display scene to a VTFx-file.
	<u>Exportcombined</u>	Export a global model or a sub-model to a common VTFx-file.
	<u>Rep</u>	Capture a situation and store in a Xtract Report Builder database.
	<u>Log</u>	Start or stop logging of commands to file.
	<u>Exit</u>	Exit from Xtract.
	<u>Resetsettings</u>	Resets settings in the registry to default.
Display Commands	<u>Display</u>	Display model and results in alternative ways.
	<u>Displaysettings</u>	Set display settings such as colours, significant digits and fonts.
	<u>Displayinfo</u>	Set display information such as content of Info box and legends.
	<u>Label</u>	Enable/disable labelling of nodes, elements and materials.
	<u>Eye</u>	Zoom, rotate and pan the model.
Model Commands	<u>Model</u>	Change how the model is displayed.
	<u>Section</u>	Make a section (cut) through the model.
Set Command	<u>Set</u>	Make new, show existing and change sets.

Table 5.1 Overview of line-mode commands

Command category	Subcommand (click link)	Explanation
Result Commands	Result	Select result case, position/attribute/component sets display modes.
	Contour	Set contour settings such as datum, levels, values and min/max.
	Vector	Set vector settings such as scaling and arrow heads.
	Shell	Select shell surface or layer for results presentation.
	Evaluation	Set evaluation method for complex result cases.
	Combine	Combine result cases into new result case for one attribute/component.
	Combine add	Add an attribute to a previously created combination.
	Combine all	Combine result cases into new case for all attributes/components.
	Combine sqrt	Create a square root of sum-of-squares type of combination.
	Scan	Scan result cases for max or min of a result component.
	Scale	Scale a complex result case.
	Createuservector	Create a vector result with given values in given sets.
	Overallvelocity	Create a result case containing the overall velocity level.
Graph Commands	Graph	Draw graph and modify graph settings.
Animation Commands	Animation	Set up, play and save an animation.

5.1 File Commands

These commands access and produce files. Also resetting registry settings is found in this section.

5.1.1 Open

Open file(s) of specific type. In the case of JNL (Command input file) the file is also processed.

Syntax:

open	<i>file-type-code</i>	<i>filename</i>	[<i>global-response-file</i>]
------	-----------------------	-----------------	---------------------------------

Explanation of parameters:

<i>file-type-code</i>	JNL	Command input file Note: ‘open JNL <i>filename</i>’ is equivalent to ‘source <i>filename</i>’.
	FEM	Input Interface File – T-file (prefixT#.FEM)
	SIF	Sequential formatted (ASCII) Results Interface File – R-file (prefixR#.SIF)
	SIU	Sequential unformatted Results Interface File – R-file (prefixR#.SIU)
	SIN	Database (NORSAM) format of Results Interface File – R-file (prefixR#.SIN)
	VTF	VTF-file (produced by Wasim, Riflex, Stofat or Installjac)
<i>filename</i>	The full name of the file to open including the file type and if necessary also the directory/folder path. <i>file-type-code</i> pertains to <i>filename</i> .	
[<i>global-response-file</i>]	Optionally a Hydrodynamic Results Interface File (also termed ‘Global Response File’) may be opened together with an Input Interface File or a Results Interface File. See Section 2.1 for more details on postprocessing of various file types.	

5.1.2 Source

Open and process a Command input file (.JNL).

Syntax:

source	<i>filename</i>
--------	-----------------

filename should be the full name of the file including the file type (preferably .jnl) and if necessary also the directory/folder path.

5.1.3 Plot

Plot active viewport to a file of specific type with optionally specified size. Choose between colour plot with white background, black and white plot and plot as shown on screen. Optionally add header and footer texts.

Syntax:

plot	<i>file-type-code</i>	<i>filename</i>	[<i>x-size</i>] [<i>y-size</i>]	[normal white bw]	[<i>header h-align</i>]	[<i>footer f-align</i>]
------	-----------------------	-----------------	-------------------------------------	-----------------------	---------------------------	---------------------------

Explanation of parameters:

<i>file-type-code</i>	BMP	Windows bitmap
	EPS	Encapsulated PostScript
	GIF	Graphics Interchange Format
	TIF	Tag Image File Format (TIFF)
	PPM	Portable Pixmap bitmap
	RGB	Silicon Graphics RGB
<i>filename</i>	Name of the plot file. It need not include the file type as this will be added according to the chosen <i>file-type-code</i> .	
[<i>x-size y-size</i>]	Plot file size given in pixels. <i>x-size</i> is horizontal and <i>y-size</i> is vertical. If the size is not given the plot size will be equal to the current size of the display area. Unless you have resized the display area this size is about 600 by 600. 400 by 400 will be a low quality (low resolution) plot, 1000 by 1000 and higher will be a high quality (high resolution) plot.	
[normal white bw]	The plot will be: - normal ('Export As Is' in GUI) which means as appearing on the screen - white ('Export With White Background' in GUI) which means exporting colours with white background and black text - bw ('Export in Black&White Mode' in GUI) which means exporting in black & white mode	
[<i>header h-align</i>]	Header text (within double-quotes when spaces) that may be aligned (adjusted) centre, left and right by setting <i>h-align</i> to 0, 1 and 2 respectively, e.g. "Main deck" 0	
[<i>footer f-align</i>]	Footer text (within double-quotes when spaces) that may be aligned (adjusted) centre, left and right by setting <i>f-align</i> to 0, 1 and 2 respectively, e.g. "Main deck" 1	

5.1.4 Plotall

Plot all viewports to a file of specific type with optionally specified size. Choose between colour plot with white background, black and white plot and plot as shown on screen.

Syntax:

plotall	<i>file-type-code</i>	<i>filename</i>	[<i>x-size</i>] [<i>y-size</i>]	[normal white bw]
---------	-----------------------	-----------------	-------------------------------------	-----------------------

Explanation of parameters:

<i>file-type-code</i>	BMP	Windows bitmap
	EPS	Encapsulated PostScript
	GIF	Graphics Interchange Format
	TIF	Tag Image File Format (TIFF)
	PPM	Portable Pixelmap bitmap
	RGB	Silicon Graphics RGB
<i>filename</i>	Name of the plot file. It need not include the file type as this will be added according to the chosen <i>file-type-code</i> .	
[<i>x-size y-size</i>]	Plot file size given in pixels. <i>x-size</i> is horizontal and <i>y-size</i> is vertical. If the size is not given the plot size will be equal to the current size of the display area. Unless you have resized the display area this size is about 600 by 600. 400 by 400 will be a low quality (low resolution) plot, 1000 by 1000 and higher will be a high quality (high resolution) plot.	
[normal white bw]	The plot will be: - normal ('Export As Is' in GUI) which means as appearing on the screen - white ('Export With White Background' in GUI) which means exporting colours with white background and black text - bw ('Export in Black&White Mode' in GUI) which means exporting in black & white mode	

5.1.5 Print

Print active view.

Syntax:

print	[normal white bw]
-------	-----------------------

Explanation of parameters:

[normal white bw]	The print will be: - normal ('Export As Is' in GUI) which means as appearing on the screen - white ('Export With White Background' in GUI) which means exporting colours with white background and black text - bw ('Export in Black&White Mode' in GUI) which means exporting in black & white mode
-----------------------	---

5.1.6 Exportvtfx

Export the 3D model with results (the current display scene) to a VTFx-file. See Section 3.13.1 about this feature.

Syntax:

exportvtfx	<i>filename</i>	[startanim]	[append]	[previewimage <i>x-size y-size</i>]
------------	-----------------	-------------	----------	--------------------------------------

filename may include the file type in which case it should be given as vtfx. The optional command ‘startanim’ starts an animation when the file is opened (in Xtract Viewer or in MS Word or PowerPoint through GLview 3D Plugin). ‘append’ appends the VTFx-file to an existing VTFx-file. Otherwise any existing VTFx-file will be overwritten. ‘previewimage’ displays a preview picture with the given pixel size when the file is opened.

5.1.7 Exportcombined

Export the 3D model to a VTFx-file. This feature is designed for viewing a global model and a sub-model in superimposed mode, read more about this feature in Section 3.13.2.

Syntax:

exportcombined	global sub	<i>filename</i>	<i>red green blue</i>	[surface wireframe]	[launchviewer]
----------------	--------------	-----------------	-----------------------	-----------------------	----------------

filename should include the file type which should be given as vtfx. When selecting command ‘global’ then a new *filename* should be entered. When selecting command ‘sub’ then an existing *filename* should be entered. The colour of the exported 3D model is specified by *red*, *green* and *blue* in range 0-255. Select draw model as ‘surface’ or ‘wireframe’. The program default is to use ‘surface’ for the global model and ‘wireframe’ for the sub-model. The optional command ‘launchviewer’ starts the program Xtract Viewer.

5.1.8 Rep

Capture a situation to the Xtract Report Builder. See Section 3.13.3 about this feature.

Syntax:

rep	<i>subcommand</i>
-----	-------------------

Alternatives for *subcommand*:

add3d <i>title description</i> [background <i>red green blue</i>] [mesh <i>red green blue</i>] [text <i>red green blue</i>] [navigationcube] [colorlegend] [infobox]	Capture a 3D model situation
addgraph <i>title description</i> [capturesize <i>x-size y-size</i>]	Capture a graph situation
addimage <i>title description</i> [capturesize <i>x-size y-size</i>] [background <i>red green blue</i>] [mesh <i>red green blue</i>] [text <i>red green blue</i>] [navigationcube] [colorlegend] [infobox]	Capture an image situation
addtable <i>title description</i>	Capture a table situation

Explanation of parameters:

<i>title</i>	Title of situation
<i>description</i>	Description of situation
[capturesize <i>x-size y-size</i>]	Size in pixels of the captured image. <i>x-size</i> is horizontal and <i>y-size</i> is vertical. If the size is not given the image size will be equal to the current size of the display area. Unless you have resized the display area this size is about 600 by 600. 400 by 400 will be a low quality (low resolution) image, 1000 by 1000 and higher will be a high quality (high resolution) image.
[background <i>red green blue</i>]	Set colour of background, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
[mesh <i>red green blue</i>]	Set colour of mesh, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
[text <i>red green blue</i>]	Set colour of text, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
[navigationcube]	Optionally include the navigation cube in the capture.
[colorlegend]	Optionally include the colour legend in the capture.
[infobox]	Optionally include the Info box in the capture.

5.1.9 Log

Start logging of commands to file or stop logging. If no file name is given then the default log file name is used. When starting Xtract logging is by default set on.

Syntax:

log	{ on off }	[filename]
-----	--------------	------------

filename should include the file type which for general consistency should be .jnl.

5.1.10 Exit

Exit from Xtract.

Syntax:

exit

5.1.11 Resetsettings

Resets settings in the registry to default. See Section 4.1.3 for which registry settings are defined.

Syntax:

resetsettings

5.2 Display Commands

These commands display model and results as well as set various display modes for such.

5.2.1 Display

Display model and results. Some of the commands work both as a toggle and as an explicit command. For example: 'display model' will display the model (switch model display on) if the model is not displayed and remove the model (switch model display off) if the model is displayed.

Syntax:

display	<i>subcommand</i>
---------	-------------------

Alternatives for *subcommand*:

model [on off [<i>from_part to_part</i>]]	Toggle or set display of selected parts.
results [on off]	Toggle or set display of results.
perspective [on off]	Toggle between perspective and orthographic view, or set view mode (on: perspective, off: orthographic).
surfaces [<i>from_part to_part</i>]	Display model as surfaces for selected parts.
lines [<i>from_part to_part</i>]	Display model as lines for selected parts.
points [<i>from_part to_part</i>]	Display model as points for selected parts.
outline [<i>from_part to_part</i>]	Display model as outline for selected parts.
hidden [<i>from_part to_part</i>]	Display model as hidden lines for selected parts.
mesh [on off [<i>from_part to_part</i>]]	Toggle or set display of mesh for selected parts.
meshcolor <i>red green blue</i>	Set mesh colour. <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
outlinemesh [on off [<i>from_part to_part</i>]]	Toggle or set display of outline mesh for selected parts.
expanded [on off [<i>from_part to_part</i>]]	Toggle or set display of expanded (fully drawn) cross sections for selected parts.
smooth [on off [<i>from_part to_part</i>]]	Toggle or set smooth shading for selected parts.
transparent [on off [<i>from_part to_part</i>]]	Toggle or set transparent display for selected parts.
alpha <i>alpha</i> [<i>from_part to_part</i>]	Set transparency alpha value in range 0-255 (invisible-opaque) for selected parts.
color <i>red green blue</i> [<i>from_part to_part</i>]	Set colours in range 0-255 for selected parts (when no results).
elementcolors [on off]	Toggle or set element type colouring.
shellthickness [on off]	Toggle or set display of plate/shell thickness.
materials [on off]	Toggle or set display of materials.
shrunk [on off]	Toggle or set shrunken mesh display mode.
labellift <i>factor</i>	Lifts (moves towards you) labels (result values shown on model), local coordinate systems and principal stress vectors. By default these are lifted somewhat so as not to be hidden by the surface to which they belong. The factor allows you to scale the default (=1) lifting.
viewports {1 2h 2v 4}	Show one, two horizontal, two vertical or four viewports.

setviewport {1 2 3 4}	Set active viewport (when four: 1 is upper left, 2 is upper right, 3 is lower left and 4 is lower right).
copyall {1 2 3 4} {1 2 3 4}	Copy visual attributes and model/set from one viewport to another.
copyattributes {1 2 3 4} {1 2 3 4}	Copy visual attributes from one viewport to another.
lighting [on off]	Toggle or set lighting.
lights {1 0}{1 0}{1 0}{1 0}{1 0}{1 0}	Turn each of 6 light sources on (1) or off (0). The sources are in the sequence: front, back, lower left, lower right, upper right and upper left. The default setting is '1 0 0 0 0 0', i.e. front on and other light sources off.

Explanation of parameters:

<i>from_part to_part</i>	The command is valid only for parts of the model (a 1st level superelement is a part) identified by their sequence numbers according to the hierarchy of the Database window. No specification of parts or 0 (zero) for both parameters means all parts. See Section 3.1.25 on parts.
--------------------------	---

5.2.2 Displaysettings

Set display settings.

Syntax:

displaysettings	<i>subcommand</i>
-----------------	-------------------

Alternatives for *subcommand*:

numdigits <i>number</i>	Set number of significant digits in the range 1-17 (results are stored on the results file as 32 bit words that can represent about 7 digits only).
shrinkfactor <i>factor</i>	Set element shrink factor in range 0-1.
fontsize <i>size</i>	Set font size in the range 2-126. Default is 10.
scaleeccentricity <i>factor</i>	Set eccentricity scaling factor
bgcolor {single topbottom}	Use a single background colour or interpolate between top and bottom colours.
topcolor <i>red green blue</i>	Set top colour of background, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
bottomcolor <i>red green blue</i>	Set bottom colour of background, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
wireframecolor <i>red green blue</i>	Set colour of undeformed wireframe, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
beamwidth <i>pixels</i>	Set line thickness for beam elements in range 1-10. 3 is default.

blacktext [on off]	Toggle or set text, numbers and pointers to black and display to black & white (grey-shading). This command corresponds to the Color / Black and White tool button and is a preview mode only and not a working mode. I.e. when selecting a new results component colours will reappear in an imperfect manner.
readbeamexpansion [on off]	Toggle or set reading of beam expansions when loading model. Give this command after opening the results file and before selecting the model. Note: A large model with many beams (stiffeners) may take some time to read due to the amount of sectional data. Use this command to skip this data. This involves that beam sections cannot be displayed (beam expansion is disabled). Moreover, the B-STRESS attribute will not appear as data for computing this result type is unavailable.

5.2.3 Displayinfo

Set display information. See Figure 3.42 for explanations of the various information items.

Syntax:

displayinfo	<i>subcommand</i>
-------------	-------------------

Alternatives for *subcommand*:

text {on off} [text]	Set user-defined text or switch it off.
name [on off]	Toggle or set display of name.
creationdate [on off]	Toggle or set display of creation date.
datetime [on off]	Toggle or set display of current date and time.
displaytype [on off]	Toggle or set display of type of display
resultcase [on off]	Toggle or set display of result case name.
attribute [on off]	Toggle or set display of attribute name.
viewportnumber [on off]	Toggle or set display of viewport number.
lengthandforceunits [on off]	Toggle or set display of length and force units.
infobox [on off]	Toggle or set display of Info box in lower right corner.
legend [on off]	Toggle or set display of legend. (Both superelement legend and result legend.)
superelementlegend [on off]	Toggle or set display of superelement legend. Legend must also be on for the superelement legend to be shown.
modelaxes [on off]	Toggle or set display of model (superelement) coordinate axes.
originaxes [on off]	Toggle or set display of origin axes.

5.2.4 Label

Enable/disable labelling of nodes, elements and materials.

Syntax:

label	<i>subcommand</i>
-------	-------------------

Alternatives for *subcommand*:

nodes [on off]	Toggle or set labelling of nodes.
elements [on off]	Toggle or set labelling of elements.
materials [on off]	Toggle or set labelling of materials.
filter [on off] [<i>max-to-draw</i>]	Toggle or set filtering of labels limited to <i>max-to-draw</i> number of labels.
coordsyst [X] [Y] [Z] [on off]	Draw element coordinate systems with given axes or turn off display of the given axes. The coordinate system is drawn for the nodes of the elements.
backgroundbox [on off]	Toggle or set drawing background box for labels
markersize <i>size</i>	Set the label marker (the violet dot to which labels are attached) size in the range 1-10. Default is 3.
resandscanindex [on off]	Toggle or set showing SCANINDEX together with results label.
scanindexformat <i>format</i>	Set format for the SCANINDEX, i.e. the result case identification.

Explanation of parameters:

<i>format</i>	{1 0} <i>run-prefix</i> {1 0} <i>resultcase-prefix</i> {a p n} <i>occurrence-prefix</i> {1 0} <i>phase-prefix</i>
	Give 1 for showing and 0 for hiding the run and result case numbers. Give prefixes for each.
	Give a, p or n for Show all, Show partial and None occurrence, respectively (see Figure 3.26).
	Give a prefix for the occurrence. Give 1 for showing and 0 for hiding the phase (complex results only). Give a prefix for the phase.

5.2.5 Eye

Zoom, rotate and pan the model. Some of the commands expect viewport coordinates. These span the pixel part of a given window defined by coordinates (0,0) in the upper left corner, the first value is horizontal.

Syntax:

eye	<i>subcommand</i>
-----	-------------------

Alternatives for *subcommand*:

setview <i>eye vrp vup rot zoom</i>	Set viewpoint and view direction. This command is for logging interactive actions and less suitable for direct use. The parameters are explained below.
rotleft <i>degrees</i>	Rotate model to the left.
rotright <i>degrees</i>	Rotate model to the right.
rotup <i>degrees</i>	Rotate model up.
rotdown <i>degrees</i>	Rotate model down.
spinleft <i>degrees</i>	Spin model to the left.

<i>spinright degrees</i>	Spin model to the right.
<i>rotto deg deg deg</i>	From position with model X-axis pointing right and Y-axis pointing up rotate about screen axes in sequence: horizontal, vertical, in/out.
<i>isometric {+ -} {+ -} {+ -}</i>	Rotate eye to isometric view from +/- of each model axis
<i>rotdefault</i>	Rotate to default angles.
<i>panleft pixels</i>	Pan model to the left.
<i>panright pixels</i>	Pan model to the right.
<i>panup pixels</i>	Pan model up.
<i>pandown pixels</i>	Pan model down.
<i>panto xpixelsl ypixels</i>	Pan to position relative to the default position in the centre.
<i>pandefault</i>	Pan to default position (centre).
<i>syncnav [on off]</i>	Toggle or set synchronous navigation of the viewports.
<i>zoomin fac</i>	Zoom in by given factor.
<i>zoomout fac</i>	Zoom out by given factor.
<i>zoomto fac</i>	Zoom to specified factor.
<i>zoomdefault</i>	Zoom to default factor (which is 1).
<i>frame</i>	Make the model fit inside the viewport.
<i>storedefault [x y z]</i>	Set default eye position vector. Stores current eye position if no arguments.

Explanation of parameters:

<i>eye</i>	X-, Y- and Z-coordinates of the eye position
<i>vrp</i>	X-, Y- and Z-coordinates of a 'view reference point', the view direction is from <i>eye</i> toward <i>vrp</i>
<i>vup</i>	X-, Y- and Z-coordinates of a 'view up vector', a point defining what is up (on screen)
<i>rot</i>	X-, Y- and Z-coordinates of a 'rotation point', a point around which interactive rotations will be performed
<i>zoom</i>	Zoom factor

5.3 Model Commands

These commands select model and whether to display it with deformations. Also, they list model data and save model data to file.

5.3.1 Model

Change display of the model.

Syntax:

model	<i>subcommand</i>
-------	-------------------

Alternatives for *subcommand*:

select <i>model-name</i>	Select model.
deformed [<i>from_part to_part</i>]	Show deformed model for selected parts.
undeformed [<i>from_part to_part</i>]	Show undeformed model for selected parts.
scaled deformations <i>factor</i> [auto]	Set deformation scale to given factor or auto-scale to a fraction of maximum model extent.
add undeformed wireframe {on off}	Show or hide an undeformed wireframe model.
list {nodes elements}	List model data (nodes or elements).
save {nodes elements} <i>filename</i>	Save model data (nodes or elements) to a file.
extramodel transformation <i>t11 t12 t13 t14 t21 ... t44</i>	Set transformation matrix from the main geometry to 'extra geometry'. Such 'extra geometry' is geometry opened as a secondary file together with a VTF-file created by Wasim or Installjac. The 'extra geometry' may be fixed or it may follow the animation of the main geometry. In both cases its position relative to the main geometry is defined by this command.
unit conversion {on off} <i>file-length-unit display-length-unit</i> <i>file-force-unit display-force-unit</i>	Set unit conversion for length and force.

Explanation of parameters:

<i>from_part to_part</i>	The command is valid only for parts of the model (a 1st level superelement is a part) identified by their sequence numbers according to the hierarchy of the Database window. No specification of parts or 0 (zero) for both parameters means all parts. See Section 3.1.25 on parts.
--------------------------	---

<i>t11 t12 t13 t14 t21 ... t44</i>	<i>t11</i>	<i>t21</i>	<i>t31</i>	<i>t41</i>	- The upper left 3 by 3 matrix (<i>t11 t12 t13 t21 t22 t23 t31 t32 t34</i>) is the cosine transformation matrix. - <i>t14 t24 t34</i> are entered as 0 (zero) - <i>t41 t42 t43</i> are the translations in X, Y and Z in the main geometry coordinate system - <i>t44</i> is the inverse scaling factor for the ‘extra geometry’, i.e. it is scaled by 1/ <i>t44</i>
	<i>t12</i>	<i>t22</i>	<i>t32</i>	<i>t42</i>	
	<i>t13</i>	<i>t23</i>	<i>t33</i>	<i>t43</i>	
	<i>t14</i>	<i>t24</i>	<i>t34</i>	<i>t44</i>	
<i>file-length-unit</i> <i>display-length-unit</i>	Specify the length unit of the file opened and the desired (display) length unit. Legal values are: m, mm, cm, in and ft.				
<i>file-force-unit</i> <i>display-force-unit</i>	Specify the force unit of the file opened and the desired (display) force unit. Legal values are: N, kN, MN, kgf, lbf and kipf.				

5.3.2 Section

Create a section through the model.

Syntax:

section	<i>subcommand</i>
---------	-------------------

Alternatives for *subcommand*:

define <i>ID xp yp zp xn yn zn</i>	Define section through model by position and normal vector.
add <i>ID</i>	Add a section with the given ID.
delete { <i>ID</i> all}	Delete the section with the given ID, or delete all.
show <i>ID</i>	Show the section with the given ID.
hide <i>ID</i>	Hide the section with the given ID.

5.4 Set Command

5.4.1 Set

The command manipulates sets.

Sets read from file (e.g. the results file) are available for selection in the same way as sets defined within Xtract.

Note: Sets read from file should not be modified by Xtract.

Note: A set name is always accompanied by a number, a counter over all sets. This counter forms a part of the full set name. For example, the full name of a set named Myset may be "8: Myset". When referring to a set the counter may be skipped (sub-string matching) unless there is more than one set with the same name. This will happen in a superelement model where a 1st level superelement with sets defined by the preprocessor has been included more than once. To distinguish between sets with the same name (in for example the 'set select' or 'set addexisting' commands) use the full name within quotes.

Syntax:

set	<i>subcommand</i>
-----	-------------------

Alternatives for *subcommand*:

select { <i>setname</i> none}	Select a set by its name or select no set.
show {workingset complement both}	Show the working set, its complement or both.
new [<i>setname</i>]	Create a new set, optionally giving it a name.
name <i>newname</i> [<i>description</i>]	Change name of current set, optionally giving it a description.
clear	Clear the content of the current set.
delete <i>setname</i>	Delete a set.
rubberbandselectionmode enclosedby outside [touched]	Set rubberband/polygon/line selection mode.
showcomplementmesh [on off]	Specify if the rest of the model shall be shown as wire-frame when using sets.
addbybeamcrosssections <i>sections</i>	Add beam elements with given cross sections to the current set.
subtractbybeamcrosssections <i>sections</i>	Subtract beam elements with given cross sections from the current set.
intersectbybeamcrosssections <i>sections</i>	Intersect the current set with beam elements having the given cross sections.
addbycoordinates <i>minX maxX minY maxY minZ maxZ</i>	Add all elements located within the given coordinate ranges to the current set.
subtractbycoordinates <i>minX maxX minY maxY minZ maxZ</i>	Subtract all elements located within the given coordinate ranges from the current set.

intersectbycoordinates <i>minX maxX minY maxY minZ maxZ</i>	Intersect all elements located within the given coordinate ranges with the current set.
addbyplanes <i>numplanes planes</i>	Add elements within the region specified by the given planes to the current set. The parameters are explained below. This command is for logging of Set > Union with > Top/Bottom/Right/Left/Rubberband. The command is less suitable for user specification.
subtractbyplanes <i>numplanes planes</i>	Subtract elements within the region specified by the given planes from the current set. The parameters are explained below. This command is for logging of Set > Subtract by > Top/Bottom/Right/Left/Rubberband. The command is less suitable for user specification.
intersectbyplanes <i>numplanes planes</i>	Intersect the current set with elements within the region specified by the given planes. The parameters are explained below. This command is for logging of Set > Intersect with > Top/Bottom/Right/Left/Rubberband. The command is less suitable for user specification.
addbypolypartitioners <i>numtris numedges partitioner-planes</i>	Add elements within the regions specified by the given planes to the current set. This command is for logging of Set > Union with > Polygon. The command is less suitable for user specification.
subtractbypolypartitioners <i>numtris numedges partitioner-planes</i>	Subtract elements within the regions specified by the given planes from the current set. This command is for logging of Set > Subtract by > Polygon. The command is less suitable for user specification.
intersectbypolypartitioners <i>numtris numedges partitioner-planes</i>	Intersect the current set with elements within the regions specified by the given planes. This command is for logging of Set > Intersect with > Polygon. The command is less suitable for user specification.
addbyresultvalues <i>min max {above below} value</i>	Add elements with result values within the given range to the current set.
subtractbyresultvalues <i>min max {above below} value</i>	Subtract elements with result values within the given range from the current set.
intersectbyresultvalues <i>min max {above below} value</i>	Intersect elements with result values within the given range with the current set.
addbyshellthickness <i>min max</i>	Add shell elements having thickness within the given range to the current set.
subtractbyshellthickness <i>min max</i>	Subtract shell elements having thickness within the given range from the current set.
intersectbyshellthickness <i>min max</i>	Intersect shell elements having thickness within the given range with the current set.
addelement <i>model element</i>	Add element to the current set. Compared with command 'set addelements' the current command adds a single element. It corresponds to Set > Pick to add.
subtractelement <i>model element</i>	Remove element from the current set. Compared with command 'set subtractelements' the current command subtracts a single element. It corresponds to Set > Remove to add.

addelements <i>model elements</i>	Add elements of the given model to the current set. <i>elements</i> is a list of ranges, e.g. '11 7 40-55 57 60-70'.
subtractelements <i>model elements</i>	Subtract elements of the given model from the current set.
intersectelements <i>model elements</i>	Intersect the current set with elements of the given model.
addelementtypes <i>element-types</i>	Add elements of given list of types to the current set.
subtraclementtypes <i>element-types</i>	Subtract elements of given list of types from the current set.
intersectelementtypes <i>element-types</i>	Intersect the current set with elements of given list of types.
addexisting <i>setname</i>	Add set with given name to the current set.
subtractexisting <i>setname</i>	Subtract set with given name from the current set.
intersectexisting <i>setname</i>	Intersect the current set with set with given name.
addmaterials <i>material-types</i>	Add elements of given list of materials to the current set.
subtractmaterials <i>material-types</i>	Subtract elements of given list of materials from the current set.
intersectmaterials <i>material-types</i>	Intersect the current set with elements of given list of materials.
addmodel <i>model-name</i>	Add all elements of a given model to the current set.
subtractmodel <i>model-name</i>	Subtract all elements of a given model from the current set.
intersectmodel <i>model-name</i>	Intersect all elements of a given model with the current set.
addvisibleelements <i>setname</i>	Add all visible elements to the current set.
subtractvisibleelements <i>setname</i>	Subtract all visible elements from the current set.
intersectvisibleelements <i>setname</i>	Intersect all visible elements with the current set.

Explanation of parameters:

<i>numplanes</i>	Number of planes used in this command
<i>planes</i>	For each of the <i>numplanes</i> planes: four floating point (real) numbers followed by three flags. The four floating point numbers are: - <i>A</i> , <i>B</i> , <i>C</i> and <i>D</i> . These define a plane by the equation $Ax + By + Cz + D = 0$. - The three flags are <i>front</i> , <i>on</i> and <i>back</i> . Each has value 1 = yes or 0 = no to whether an element with this position relative to the plane will be selected.

5.5 Result Commands

These commands select and manipulate results and how to present them.

The latter two categories ‘createuservector’ and ‘overallvelocity’ are for a specialised feature for computing the overall vibration level for a ship. See Section 3.15.3.

5.5.1 Result

Select a result case and a position/attribute/component. The selected component is automatically displayed. The command also sets display modes and parameters as well as lists and saves results data.

Syntax:

result	<i>subcommand</i>
--------	-------------------

Alternatives for *subcommand*:

selectcase <i>run case occ</i>	Select result case by giving run number, result case and occurrence. See below for explanations of parameters.
select <i>pos attr comp</i>	Select and display result position/attribute/component. See below for explanations of parameters.
reload	Reload current result. Relevant e.g. after selecting a new set.
peaks [on off]	Toggle or set display of peaks.
values [on off]	Toggle or set display of values.
startlabelnodes	A label (box) with the Nodes values of all components of the selected attribute will be added for the subsequently selected or picked nodes. See Section 3.1.12.
selectnode <i>model node</i>	Select a node for which a label (box) will be shown.
startlabelelements	A label (box) with the Element average values of all components of the selected attribute will be added for the subsequently selected or picked elements. See Section 3.1.12.
selectelement <i>model element</i>	Select an element for which a label (box) will be shown.
showresultinlabel [on off]	Show or hide the result in picking labels.
clearlabels	Clear (remove) all node and element labels (boxes).
contours [on off [<i>from_part to_part</i>]]	Toggle or set display of contours for selected parts.
vectors [on off [<i>from_part to_part</i>]]	Toggle or set display of vectors for selected parts. See explanation for subcommand pstressvectors below.
pstressvectors [P1] [P2] [P3] [on off]	Turn on/off vectors for principal stress components. Note: This command does not display vectors. It is used for choosing which vectors to show when ‘result vectors on’ (or Results > Show vectors) is given.
rotangle <i>rot-angle</i>	Rotate the element axis system around the surface normal the given number of degrees.

rotclear	Remove any user-defined element axis system.
rotsetxaxis <i>element-x-definition-vector</i>	Specify a vector that projected onto the surface defines the new element x-axis of the rotated element coordinate system.
rotsetyaxis <i>element-y-definition-vector</i>	Specify a vector that projected onto the surface defines the new element y-axis of the rotated element coordinate system.
list [coords] [components] [resultcases]	List result data, optionally including coordinates, all components and all result cases.
save <i>filename</i> [coords] [components] [resultcases]	Save result data to a file, optionally including coordinates, all components and all result cases.
maxcoorddev <i>max-angular-deviation</i>	Set maximum angular deviation (in degrees) between coordinate systems of adjacent elements for nodal averaging of stresses to be allowed. Default is 5.
maxthicknessdev <i>max-thickness-deviation</i>	Set maximum thickness deviation (in percent) between adjacent elements for nodal averaging of stresses to be allowed. Default is 10.
scanstepangle <i>scan-step-angle</i>	Set step angle (in degrees) used for all cases where stepping through complex results is performed (evaluation, combining and scanning). Default is 10.
resultpoints {all <i>resultpoints</i> }	Show result point labels for all or selected result points. <i>resultpoints</i> is a list of ranges, e.g. '1-3 5'.
info	Show information about current result case.
deletecase <i>name</i>	Delete a user defined result case, i.e. a combination, scan result case or scaled complex case all created in Xtract

Explanation of parameters:

<i>run case occ</i>	The run, resultcase and occurrence numbers, see Section 3.3
<i>pos</i>	The result position, allowable values: Nodes, Elements, Element average, Result-points and Surface resultpoints. The result positions are explained in Section 3.2.1.
<i>attr</i>	The result attribute, see Section 3.2.1 and Appendix B 2
<i>comp</i>	The result component, see Appendix B 2
<i>from_part to_part</i>	The command is valid only for parts of the model (a 1st level superelement is a part) identified by their sequence numbers according to the hierarchy of the Database window. No specification of parts or 0 (zero) for both parameters means all parts. See Section 3.1.25 on parts.

Note: Xtract always logs result positions, attributes and components as well as result case identifications in double-quotes. In a Command input file the double-quotes may be skipped unless the string includes spaces.

5.5.2 Contour

Set contour settings. The following contour modes are available:

- Datum sets a value (*datum*) for one contour. The other contours are set relative to this one.
 - The step option sets the contour increment (*stepsize*) above and below *datum* bounded by the result range. *datum* may be outside the result range.
 - The levels option determines the contour increment above and below *datum* bounded by the result range based on the given number of levels.
- Levels sets the number of levels (contours) between the maximum and minimum result value.
- Values sets any number of values. Each value will correspond to a contour.
- Minmax sets values for the minimum contour (*min*) and maximum contour (*max*). Inside this range either a step size or the number of levels is given. Areas with values above the maximum are violet and below the minimum are pink.

Syntax:

contour	<i>subcommand</i>
---------	-------------------

Alternatives for *subcommand*:

datum <i>datum</i> {step <i>stepsize</i> levels <i>number-of-levels</i> }	Set contour mode to Datum.
levels <i>number-of-levels</i>	Set contour mode to Levels.
values <i>value</i> [<i>value</i> , ...]	Set contour mode to Values.
minmax <i>min max</i> {step <i>stepsize</i> levels <i>number-of-levels</i> }	Set contour mode to Minmax.
type [filled lines]	Toggle or set contour type to filled or lines.
default	Use default contour settings which is Levels mode with 10 levels.

5.5.3 Vector

Set vector settings.

Syntax:

vector	<i>subcommand</i>
--------	-------------------

Alternatives for *subcommand*:

scale <i>factor</i> [auto]	Set vector scale to given factor or auto-scale to a fraction of max model extent. Default is auto-scale 0.05.
scalepstress [on off]	Toggle or set scaling of principal stress vectors according to value.
color <i>red green blue</i>	Set colour of displacement vectors, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
arrows [on off]	Toggle or set display of arrow heads (crosslines for compressive principal stress vectors and heads for displacement vectors).
arrowsize <i>size</i>	Set size of arrow heads relative to the default size (=1). Relevant for vectors displayed by command vector arrows (see above) and not for local coordinate (element) axes.

5.5.4 Shell

Select shell surface or layer for results presentation. By default results for the upper surface (of layer 1 for layered element) is shown.

Syntax:

shell	[upper lower]	[layer-number]
-------	-----------------	----------------

Explanation of subcommands and parameters:

[upper lower]	The upper surface is the one on positive local z-side.
[layer-number]	Skip this parameter for non-layered (normal) shell elements. The layers are numbered consecutively from 1 and up according to their definition.

5.5.5 Evaluation

Set evaluation method for complex result cases. This setting is valid for subsequent presentations of complex results. See Section 3.4.1 for more information and Appendix B 5 for some basic theory on complex results.

Note: Depending on whether the subsequently selected component is linear (harmonic) or not only certain evaluation methods are relevant as explained below. Non-linear (non-harmonic) components are typically von Mises stress and principal stress.

Syntax:

evaluation	real imaginary magnitude phaseshift phase <i>angle</i> maximum phaseofmax
------------	---

Explanation of subcommands and parameters:

real	The real part of the result (corresponding to phase angle 0), for linear (harmonic) components only
imaginary	The imaginary part of the result (corresponding to phase angle 270 or -90), for linear (harmonic) components only
magnitude	The magnitude (amplitude) of the result ($\sqrt{R^2 + I^2}$), for linear (harmonic) components only
phaseshift	The phase shift (in degrees) of the result ($\text{atan}(I/R)$). I.e. the shift between the phase of the load and the phase of the result. For linear (harmonic) components only.
phase <i>angle</i>	The phase angle (in degrees) for evaluation is given explicitly. For all components.
maximum	The maximum through the cycle (360 degrees) of a non-linear (non-harmonic) component, i.e. a principal stress or von Mises stress. Also used for Nodes/DISPLACEMENT/ALL.
phaseofmax	The phase (in degrees) giving maximum through the cycle (360 degrees) of a non-linear (non-harmonic) component. Also used for Nodes/DISPLACEMENT/ALL.

5.5.6 Combine

Combine selected result cases with factors into a new result case. The new result case will contain the given result position/attribute only (and all components of this).

- Extend an already created and incomplete combination (i.e. 'combine all' has not been used) with new attributes by the 'combine add' command; see Section 5.5.7.
- For creating a complete combination including all result positions/attributes use the 'combine all' command; see Section 5.5.8.
- Create a square root of sum-of-squares type of combination (see subsection Square Root of Sum-of-Squares Combination for Elements/G-STRESS of Section 3.1.17) by the 'combine sqrt' command; see Section 5.5.9.

Note: Not more than one complex result case may contribute to a combination. See Appendix B 5 for some basic theory on postprocessing complex results.

Syntax:

combine	<i>name</i>	[[abs] {max min} <i>angle</i>]			<i>pos</i>	<i>attr</i>	[<i>comp</i>]	...
...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...			
...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...			

... continue for all result cases to be combined.

Explanation of subcommands and parameters:

<i>name</i>			Name of the combination to create. If the name includes blanks use double-quotes: "Combination A". The combination is put in a run named User. The occurrence will be 1.
[[abs] {max min} <i>angle</i>]			Evaluate the optional single complex result cases for the (absolute) maximum or minimum value of <i>pos/attr/comp</i> . Alternatively, evaluate for a given phase angle. Note that this evaluation is irrelevant and should be skipped when no complex result cases contribute to the combination. See Figure 3.60 for an illustration of the meaning of absolute maximum/minimum. Omitting ‘abs’ corresponds to true in the figure.
<i>pos</i>	<i>attr</i>	[<i>comp</i>]	Give the result position/attribute/component for which the combination will be created. Example: Elements G-STRESS SIGXX. Note that ‘ <i>comp</i> ’ is irrelevant and should be skipped when no complex result cases contribute to the combination.
<i>run</i>	<i>case</i>	<i>occ</i>	Each result case to include in the combination is identified by run number, result case and occurrence. Each may be enclosed by double-quotes. Double-quotes are required only if blanks are contained. Example: 1 3 "2: Excit. freq. 2 Hz". Occurrence may be abbreviated as long as it is unique: 1 3 2: is equivalent to the above.
<i>factor</i>			Each result case is multiplied (scaled) by ‘ <i>factor</i> ’ before being added to the combination.

5.5.7 Combine add

Add an attribute to a previously created combination. This is irrelevant for combinations including a complex result case.

Syntax:

combine add	<i>name</i>	<i>pos</i>	<i>attr</i>
-------------	-------------	------------	-------------

Explanation of parameters:

<i>name</i>	Name of the combination to extend. If the name includes blanks use double-quotes: "Combination A". This must previously have been created.		
<i>pos</i>	<i>attr</i>	Give the result position/attribute with which the combination will be extended. Example: Elements G-STRESS.	

5.5.8 Combine all

Combine selected result cases with factors into a new result case. The new result case will contain all result positions/attributes available in all selected result cases. (Use Combine in Section 5.5.6 to combine a single result position/attribute.)

Note: A specific phase angle must be given if one of the result cases is complex. I.e. the optional single complex result cases cannot be evaluated for the (absolute) maximum or minimum value of all its attributes/components at the same time. An attempt at violating this restriction by entering for example the command 'combine all Comb1 max ...' will result in phase angle zero being used.

Syntax:

combine all	<i>name</i>	{ <i>angle</i> nocomplex}	...
-------------	-------------	-----------------------------	-----

...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...
-----	------------	-------------	------------	---------------	-----

...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...
-----	------------	-------------	------------	---------------	-----

... continue for all result cases to be combined.

Explanation of subcommands and parameters:

<i>name</i>	Name of the combination to create. If the name includes blanks use double-quotes: "Combination A". The combination is put in a run named User. The occurrence will be 1.		
{ <i>angle</i> nocomplex}		Evaluate the optional single complex result cases for the given phase angle. If no complex result case is included give the command nocomplex.	
<i>run</i>	<i>case</i>	<i>occ</i>	Each result case to include in the combination is identified by run number, result case and occurrence. Each may be enclosed by double-quotes. Double-quotes are required only if blanks are contained. Example: 1 3 "2: Excit. freq. 2 Hz". Occurrence may be abbreviated as long as it is unique: 1 3 2: is equivalent to the above.
<i>factor</i>	Each result case is multiplied (scaled) by ' <i>factor</i> ' before being added to the combination.		

5.5.9 Combine sqrt

Create a square root of sum-of-squares type of combination of selected result cases with factors. The combination can only be created for Elements/G-STRESS and Elements/D-STRESS components. The square root of sum-of-squares type of combination is explained in Section 3.15.2.

Note: The square root of sum-of-squares combination cannot be used when one of the resultcases is complex.

Syntax:

combine sqrt	<i>name</i>	<i>pos</i>	<i>attr</i>	...	
...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...
...	<i>run</i>	<i>case</i>	<i>occ</i>	<i>factor</i>	...

... continue for all result cases to be combined.

Explanation of subcommands and parameters:

name			Name of the combination to create. If the name includes blanks use double-quotes: "Combination A". The combination is put in a run named User. The occurrence will be 1.
pos	attr		Give the result position/attribute for which the combination will be created. Currently only position Elements and attribute G-STRESS are allowed.
run	case	occ	Each result case to include in the combination is identified by run number, result case and occurrence. Each may be enclosed by double-quotes. Double-quotes are required only if blanks are contained. Example: 1 3 "2: Excit. freq. 2 Hz". Occurrence may be abbreviated as long as it is unique: 1 3 2: is equivalent to the above.
factor			Each result case is multiplied (scaled) by 'factor' before being added to the combination.

5.5.10 Scan

Scan selected result cases for the (absolute) maximum or minimum value of a given result position/attribute/component. A new result case will be created. This scan result case will contain all components of the given result position/attribute.

Syntax:

scan	<i>name</i>	[abs] {max min}	<i>pos</i>	<i>attr</i>	<i>comp</i>	...
...	<i>run</i>	<i>case</i>	<i>occ</i>	criterion <i>phase</i>	<i>factor</i>	...
...	<i>run</i>	<i>case</i>	<i>occ</i>	criterion <i>phase</i>	<i>factor</i>	...

... continue for all result cases to be scanned.

Explanation of subcommands and parameters:

<i>name</i>			Name of the scan result case to create. If the name includes blanks use double-quotes: "Scan A". The scan result case is put in a run named User. The occurrence will be 1.
[abs] {max min}			Scan the result cases for the (absolute) maximum or minimum value.
<i>pos</i>	<i>attr</i>	<i>comp</i>	Give the result position/attribute/component for which scanning will be performed. Example: Elements G-STRESS SIGXX
<i>run</i>	<i>case</i>	<i>occ</i>	Each result case to scan is identified by run number, result case and occurrence. Each may be enclosed by double-quotes. Double-quotes are required only if blanks are contained. Example: 1 3 "2: Excit. freq. 2 Hz". Occurrence may be abbreviated as long as it is unique: 1 3 2: is equivalent to the above.
criterion <i>phase</i>			This is irrelevant for non-complex (e.g. static) result cases, for such cases enter a dummy value 0 for ' <i>phase</i> '. Complex result cases, however, need to be evaluated (see 5.5.5 Evaluation). Subcommand 'criterion' involves that the evaluation is performed in accordance with the scan criterion itself. I.e. for [abs] {max min} of <i>pos/attr/comp</i> . This is normally what you want to do. Alternatively, entering a value for ' <i>phase</i> ' allows evaluation to be performed for a specific phase angle.
<i>factor</i>			Each result case is multiplied by ' <i>factor</i> ' prior to scanning.

5.5.11 Scale

Scale a complex result case. This is typically relevant for scaling results corresponding to the unit wave heights of a frequency domain analysis up to actual wave heights. A new result case will be created.

Syntax:

scale	<i>run</i>	<i>case</i>	<i>occ</i>	<i>name</i>	<i>factor</i>
-------	------------	-------------	------------	-------------	---------------

Explanation of parameters:

<i>run</i>	<i>case</i>	<i>occ</i>	The result case to scale is identified by run number, result case and occurrence. Each may be enclosed by double-quotes. Double-quotes are required only if blanks are contained. Example: 1 3 "2: Excit. freq. 2 Hz". Occurrence may be abbreviated as long as it is unique: 1 3 2: is equivalent to the above.
<i>name</i>			Name of the new complex result case. If the name includes blanks use double-quotes: "Scaled A". The scaled complex result case is put in a run named User. The occurrence will be 1.
<i>factor</i>			Scaling factor

5.5.12 Createuservector

Create a vector result with given values in given sets. This command is used in connection with the specialised feature for computing overall vibration level for a ship. See Section 3.15.3.

Syntax:

createuservector	<i>name</i>	<i>setname</i>	<i>x-value</i>	<i>y-value</i>	<i>z-value</i>	[<i>setname x-value y-value z-value [...]</i>]
------------------	-------------	----------------	----------------	----------------	----------------	--

Explanation of parameters:

<i>name</i>	Name of the new vector result
<i>setname</i>	Name of an existing set for which the subsequent values are defined
<i>x-value y-value z-value</i>	Values of the new vector result for the given set

5.5.13 Overallvelocity

Create a result case containing the overall velocity level. This command is used in connection with the specialised feature for computing overall vibration level for a ship. See Section 3.15.3.

Syntax:

overallvelocity	<i>name</i>	<i>tipvortexcase</i>	<i>minfreq</i>	<i>maxfreq</i>	<i>run</i>	<i>case</i>	<i>factor</i>	...
...	[<i>minfreq maxfreq run case factor [...]</i>]							

Explanation of parameters:

<i>name</i>	Name of the new result case containing the overall velocity level							
<i>tipvortexcase</i>	Name of a vector result case created by the createuservector command							
<i>minfreq maxfreq</i>	Frequency range contributing to the overall velocity level							
<i>run case factor</i>	Result case with factor contributing to overall velocity level							

5.6 Graph Commands

5.6.1 Graph

Draw graph or modify graph settings.

Syntax:

graph	<i>subcommand</i>
-------	-------------------

Alternatives for *subcommand*:

nodes [add] <i>name run case occ pos attr comp</i>	Draw a graph of the values of the previously selected nodes. <i>name</i> is the name of the curve. The curve is optionally added to a graph containing previously created curves.
beams [add] <i>name run case occ pos attr comp</i>	Draw a graph of the values of the previously selected beams. <i>name</i> is the name of the curve. The curve is optionally added to a graph containing previously created curves.
occurrences [add] <i>name pos attr comp occur</i>	Draw a graph of the values of the selected node over occurrences. See explanation of parameter <i>occur</i> below.
xscale <i>min max unit</i>	Set X-axis scaling.
yscale <i>min max unit</i>	Set Y-axis scaling.
title <i>title</i>	Set graph title.
xtitle <i>title</i>	Set X-axis title.
ytitle <i>title</i>	Set Y-axis title.

legend [on off]	Toggle or set display of graph legend.
transparent [on off]	Toggle or set transparency of graph.
stippled [on off]	Toggle or set drawing of stippled lines.
grid [on off]	Toggle or set drawing of grid lines.
minmax [on off]	Toggle or set display of minimum and maximum values on the axes.
color <i>red green blue</i>	Set background colour of graph, <i>red</i> , <i>green</i> and <i>blue</i> are in range 0-255.
show {upperright entire none}	Draw graph in upper right corner, entire viewport or not at all.
selectnode [exclusive expand] <i>model node</i>	Add a node identified by model name and node number to the current selection. The exclusive option involves replacing the current selection. The expand option involves adding to the current selection all nodes on a line (a straight as possible path) between the last selected node (in a previous command) and the selected node.
selectbeam [exclusive expand] <i>model beam</i>	Add a beam element identified by model name and beam (element) number to the current selection. The exclusive option involves replacing the current selection. The expand option involves adding to the current selection all beams on a line (a straight as possible path) between the last selected beam (in a previous command) and the selected beam.
clearselection	Clear selected items.
clearitem <i>item-number</i>	Remove a specific item.
itemlabels [on off]	Toggle or set node/element numbers along curves
export <i>filename</i>	Export the curves to a text file (specify file extension, e.g. txt).
openfile <i>filename</i>	Open VTF-file with 2D series data. This file is inextricably tied to a VTF-file opened with File > Open.
filecurve [add] <i>series-ID curve-ID</i>	Draw a graph of selected data taken from a VTF-file with 2D series data, see the openfile command above.

Explanation of parameters:

<i>occurr</i>	A list of all occurrences. Each occurrence is identified by run number, result case and occurrence as follows: <i>run1 case1 occ1 [run2 case2 occ2 [...]]</i>
---------------	--

5.7 Animation Commands

5.7.1 Animation

An animation sequence is set up and played for the selected result case(s) and position/attribute/component. While the animation is played the model with results may be rotated and zoomed and settings such as defor-

mation scaling and result settings may be modified. However, the animation is only available until another result case or position/attribute/component is selected.

Syntax:

animation	<i>subcommand</i>
-----------	-------------------

Alternatives for *subcommand*:

frequency <i>start-angle stop-angle skip-by</i>	Setup frequency domain (complex) animation.
cases all [except <i>list-of-case-indices</i>] [<i>skip number</i>]	Setup animation over all occurrences (for time domain results).
modeshape <i>num-steps</i>	Setup mode shape animation, can also be used for static results.
maxfps <i>number</i>	Set maximum number of frames per second.
start	Start current animation.
stop	Stop current animation.
backward	Play current animation backward.
step	Step forward.
stepbackward	Step backward.
pause	Pause animation.
end	End animation and return to static display.
saveavi <i>filename</i>	Save animation to AVI-file using current settings.
savegif <i>filename</i>	Save animation to GIF-file using current settings.
savesize <i>x-size y-size</i>	Set image size for animation to save (give –1 to use current view size).
savecycle [on off]	Toggle or set cycling (back and forth) for animation to save.
saveloops [forever <i>loops</i>]	Set number of loops for saved animated GIF sequence.
saveframerepeat <i>repeats</i> [<i>firstframerepeat</i>] [<i>lastframerepeat</i>]	Set frame repeat (number of times each frame is stored) for animation to save. Optionally also number of repeats for first and last frames.
saveframerate <i>max-frames-per-second</i>	Set target frame rate for animation to save (give –1 to use current maxfps).
savetransparent [on off]	Toggle or set transparent background for saved animated GIF sequence.

APPENDIX A TUTORIAL EXAMPLES

A 1 Plate Connection with Bracket

This is the example used in **Section 3.1 Getting Started**. The GeniE, Prefem and Patran-Pre inputs required to create the model (in three slightly different alternatives) are provided.

A 1.1 GeniE Input

The model displayed by GeniE is shown in Figure A.1. The loads are shown by Prefem in Figure A.3.

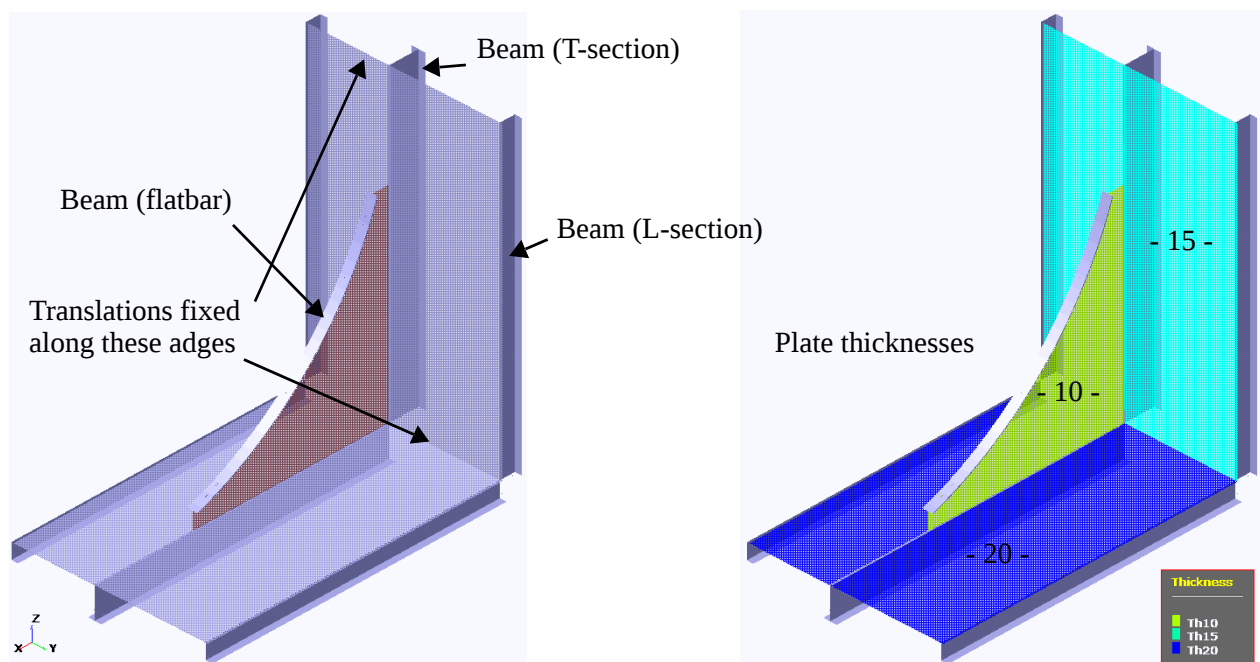


Figure A.1 The concept model displayed by GeniE

The GeniE input:**A.7.2 // GeniE input for creating a model of a plate connection with bracket**

```
GenieRules.Units.setDatabaseUnits("mm", "N", "delC");
GenieRules.Units.setInputUnit(Length, "mm");
GenieRules.Units.setInputUnit(Force, "N");
GenieRules.Units.setInputUnit(TempDiff, "delC");
GenieRules.Tolerances.useTolerantModelling = true;
// Define sections, material and plate thicknesses
Flatbar = BarSection(6,60);
Girder = UnsymISection(180,10,11,5.5,10,80,40,20);
Stiff = LSection(90,40,8,20);
Steel = Material(0,7.85e-9,2.1e5,0.3,0,0);
Th10 = Thickness(10);
Th15 = Thickness(15);
Th20 = Thickness(20);
// Set defaults for subsequent modelling
Stiff.setDefault();
Steel.setDefault();
Th20.setDefault();
// Create horizontal plate
Bm1 = Beam(Point(0,0,0),Point(1800,0,0));
Bm2 = Bm1.copyTranslate(Vector3d(0,480,0));
Bm3 = Bm1.copyTranslate(Vector3d(0,1150,0));
Bm2.section = Girder;
Pl1 = Plate(Point(0,0,0),Point(1800,0,0),Point(1800,1150,0),Point(0,1150,0));
Bm1.setBeamOffset(Vector3d(0,0,-61.47058824));
Bm2.setBeamOffset(Vector3d(0,0,-124.6261682));
Bm3.setBeamOffset(Vector3d(0,0,-61.47058824));
// Create vertical plate by copying and moving
Bm4 = Bm1.copyRotate(Point(0,0,0),Vector3d(0,1150,0),90);
Bm5 = Bm2.copyRotate(Point(0,0,0),Vector3d(0,1150,0),90);
Bm6 = Bm3.copyRotate(Point(0,0,0),Vector3d(0,1150,0),90);
Pl2 = Pl1.copyRotate(Point(0,0,0),Vector3d(0,1150,0),90);
Pl2.thickness = Th15;
autoMSet = Set();
autoMSet.clear();
autoMSet.add(Pl2);
autoMSet.add(Bm4);
autoMSet.add(Bm5);
autoMSet.add(Bm6);
autoMSet.moveTranslate(Vector3d(0,0,1800),geUNCONNECTED);
Delete(autoMSet);
// Further modelling
FEdge1a = FeatureEdge(Point(600,0,0),Point(600,480,0));
FEdge1b = FeatureEdge(Point(600,480,0),Point(600,1150,0));
FEdge2a = FeatureEdge(Point(1000,0,0),Point(1000,480,0));
FEdge2b = FeatureEdge(Point(1000,480,0),Point(1000,1150,0));
FEdge3a = FeatureEdge(Point(1200,0,0),Point(1200,480,0));
FEdge3b = FeatureEdge(Point(1200,480,0),Point(1200,1150,0));
FEdge4a = FeatureEdge(Point(1800,0,0),Point(1800,480,0));
```

```
FEdge4b = FeatureEdge(Point(1800,480,0),Point(1800,1150,0));
FEdge5a = FeatureEdge(Point(0,0,0),Point(0,480,0));
FEdge5b = FeatureEdge(Point(0,480,0),Point(0,1150,0));
FEdge6a = FeatureEdge(Point(0,0,600),Point(0,480,600));
FEdge6b = FeatureEdge(Point(0,480,600),Point(0,1150,600));
FEdge7a = FeatureEdge(Point(0,0,1000),Point(0,480,1000));
FEdge7b = FeatureEdge(Point(0,480,1000),Point(0,1150,1000));
FEdge8a = FeatureEdge(Point(0,0,1200),Point(0,480,1200));
FEdge8b = FeatureEdge(Point(0,480,1200),Point(0,1150,1200));
FEdge9a = FeatureEdge(Point(0,0,1800),Point(0,480,1800));
FEdge9b = FeatureEdge(Point(0,480,1800),Point(0,1150,1800));
Pl3 = Plate(Point(0,480,0),Point(1200,480,0),Point(1200,480,1200),
            Point(0,480,1200));
Pl3.thickness = Th10;
Point1 = Point(1800,480,1800);
Curve1 = GuideCircle(Point(1800,480,1800), Point(0,480,1800),Point(1800,480,0));
Pl3.explode(IndexedNameMask(4));
Pl3.explode(IndexedNameMask(4));
FEdge10 = FeatureEdge(Curve1);
Pl3.explode(IndexedNameMask(4));
Delete(Pl5);
Flatbar.setDefault();
Bm7 = CurvedBeam(ModelCurve(Point(1200,480,102.9437252),
                             Point(527.2077939,480,527.2077939),
                             Point(102.9437252,480,1200)));
FEdge11 = FeatureEdge(Point(1200,480,0),Point(1200,480,102.9437252));
FEdge12 = FeatureEdge(Point(0,480,1200),Point(102.9437252,480,1200));
FEdge13 = FeatureEdge(Point(0,480,0),Point(1800,480,1800));
Bm3.explode(IndexedNameMask(8));
Bm1.explode(IndexedNameMask(12));
Bm2.explode(IndexedNameMask(16));
Bm4.explode(IndexedNameMask(20));
Bm5.explode(IndexedNameMask(24));
Bm6.explode(IndexedNameMask(28));
Bm7.explode(IndexedNameMask(32));
// Set mesh densities
Me1 = NumberOfElements(1);
Me3 = NumberOfElements(3);
Me4 = NumberOfElements(4);
Mex = NumberOfElements(11);
FEdge1a.numberOfElements = Me3;
FEdge1b.numberOfElements = Me3;
FEdge2a.numberOfElements = Me3;
FEdge2b.numberOfElements = Me3;
FEdge3a.numberOfElements = Me3;
FEdge3b.numberOfElements = Me3;
FEdge4a.numberOfElements = Me3;
FEdge4b.numberOfElements = Me3;
FEdge5a.numberOfElements = Me3;
FEdge5b.numberOfElements = Me3;
FEdge6a.numberOfElements = Me3;
FEdge6b.numberOfElements = Me3;
```

```
FEdge7a.numberOfElements = Me3;
FEdge7b.numberOfElements = Me3;
FEdge8a.numberOfElements = Me3;
FEdge8b.numberOfElements = Me3;
FEdge9a.numberOfElements = Me3;
FEdge9b.numberOfElements = Me3;
FEdge11.numberOfElements = Me1;
FEdge12.numberOfElements = Me1;
FEdge13.numberOfElements = Me4;
Bm8.numberOfElements = Me4;
Bm9.numberOfElements = Me4;
Bm10.numberOfElements = Me4;
Bm11.numberOfElements = Me4;
Bm12.numberOfElements = Me4;
Bm13.numberOfElements = Me4;
Bm14.numberOfElements = Me4;
Bm15.numberOfElements = Me4;
Bm16.numberOfElements = Me4;
Bm17.numberOfElements = Me4;
Bm18.numberOfElements = Me4;
Bm19.numberOfElements = Me4;
Bm20.numberOfElements = Me4;
Bm21.numberOfElements = Me4;
Bm22.numberOfElements = Me4;
Bm23.numberOfElements = Me4;
Bm24.numberOfElements = Me4;
Bm25.numberOfElements = Me4;
Bm26.numberOfElements = Me4;
Bm27.numberOfElements = Me4;
Bm28.numberOfElements = Me4;
Bm29.numberOfElements = Me4;
Bm30.numberOfElements = Me4;
Bm31.numberOfElements = Me4;
Bm32.numberOfElements = Mex;
Bm33.numberOfElements = Mex;
// Make sets
Deck = Set();
Deck.add(Pl1);
Bulkhead = Set();
Bulkhead.add(Pl2);
Bracket = Set();
Bracket.add(Pl4);
// Boundary conditions
Sc1 = SupportCurve(ModelCurve(Point(0,0,0),Point(0,575,0),Point(0,1150,0)));
Sc2 = SupportCurve(ModelCurve(Point(0,0,1800),Point(0,575,1800),
                             Point(0,1150,1800)));
Sc1.boundary = BoundaryCondition(Fixed, Fixed, Fixed, Free, Free, Free);
Sc2.boundary = BoundaryCondition(Fixed, Fixed, Free, Free, Free, Free);
// Loads
LC1 = LoadCase();
LLoad1 = LineLoad(LC1, Point(1800,0,0), Point(1800,1150,0), 200, 0, 0, 200, 0, 0);
LC2 = LoadCase();
```

```

LLoad2 = LineLoad(LC2, Point(1800,0,0), Point(1800,1150,0), 0, 0, 10, 0, 0, 10);
LC3 = LoadCase();
PLoad1 = PointLoad(LC3, Point(1800,0,0), 0, 0, 0, 0, 720000, 0);
PLoad2 = PointLoad(LC3, Point(1800,480,0), 0, 0, 0, 0, 1725000, 0);
PLoad3 = PointLoad(LC3, Point(1800,1150,0), 0, 0, 0, 0, 1005000, 0);
LC4 = LoadCase();
SLoad1 = FlatSurfaceLoad(LC4, 0, 0, -0.015, Point(0,0,0), Point(1800,0,0),
                          Point(1800,1150,0), Point(0,1150,0));
SLoad2 = FlatSurfaceLoad(LC4, -0.01275, 0, 0, Point(0,0,600), Point(0,0,0),
                          Point(0,1150,0), Point(0,1150,600));
SLoad3 = FlatSurfaceLoad(LC4, -0.009, 0, 0, Point(0,0,1000), Point(0,0,600),
                          Point(0,1150,600), Point(0,1150,1000));
SLoad4 = FlatSurfaceLoad(LC4, -0.00675, 0, 0, Point(0,0,1200), Point(0,0,1000),
                          Point(0,1150,1000), Point(0,1150,1200));
SLoad5 = FlatSurfaceLoad(LC4, -0.00375, 0, 0, Point(0,0,1800), Point(0,0,1200),
                          Point(0,1150,1200), Point(0,1150,1800));
Analysis1 = Analysis(true);
Analysis1.add(MeshActivity());
Analysis1.add(LinearAnalysis());
Analysis1.add(LoadResultsActivity());

```

A 1.2 Prefem Input

The FE model as displayed by Prefem is shown in Figure A.2 while the loads are shown in Figure A.3.

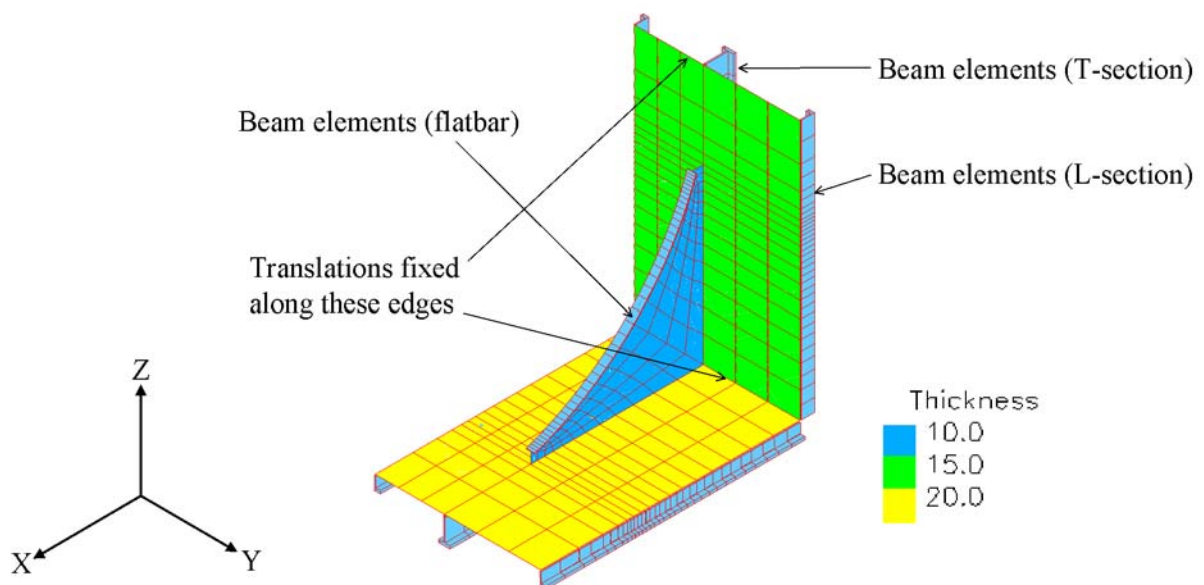


Figure A.2 The FE model displayed by Prefem

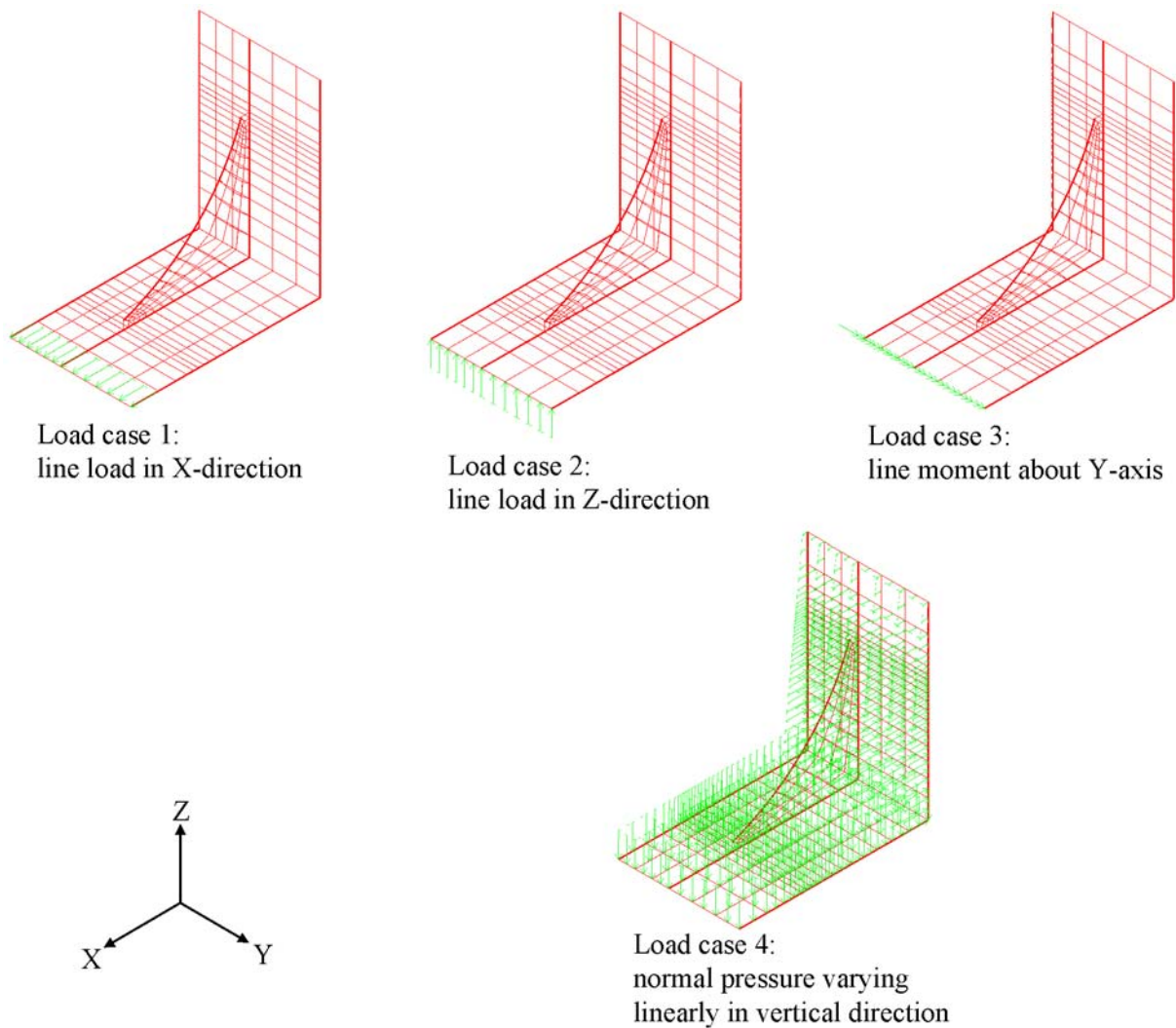


Figure A.3 The loads displayed by Prefem

The Prefem input:

```
% Prefem input for creating a model of a plate connection with bracket
%
GENERATE SURFACE A 1 3 1 12 1 3 1 3 1 3 1 12
      CARTESIAN 0 0 0
      1200 0 0      600 0 0      END
      0 480 0      0 670 0      END
      0 0 1200      0 0 600      END
%
DEFINE SET DECK UNION-WITH SURFACES AU&&1 END END
      BULKHEAD UNION-WITH SURFACES AS1&& END END
. .
%
```

```

DELETE GEOMETRY ( ALL-SURFACES-INCLUDED EXCLUDE DECK BULKHEAD )
..
%
CUT AI122 PREDEFINED-PLANE YZ-PLANE 80
CUT AK221 PREDEFINED-PLANE XY-PLANE 80
%
DEFINE ARC ARC1 P00 P01 AP323 22 END
      SURFACE SU0 AI121 AK221 ARC1 AI122 AK121
..
DEFINE LINE CUTLINE AP323 AP121 1
..
CUT SU0 CUTLINE
DEFINE SET BRACKET UNION-WITH SURFACES ( SU1 SU2 ) END END
..
%
DELETE GEOMETRY ( ALL-LINES-INCLUDED ALL-POINTS-INCLUDED EXCLUDE AP323 )
..
%
PROPERTY THICKNESS DECK 20
                  BULKHEAD 15
                  BRACKET 10
..
%
PROPERTY SECTION FLATBAR BAR 6 60 60 1 1
                  GIRDER  I 180 11 10 10 80 20 1 1
                  STIFF   L 90 8 40 20 1 1 NEGATIVE
..
CONNECT SECTION FLATBAR ( ARC1 LI2 )
          SECTION GIRDER ( AI&21 AK12& )
          SECTION STIFF  ( AI&11 AI&31 AK11& AK13& )
..
%
PROPERTY LOCAL-COORDINATE-BEAM ZX-PLANE +X-GLOBAL-INFINITY AK1*
..
%
PROPERTY ECCENTRICITY-BEAM ( AI&21 AK12& AI&11 AI&31 AK11& AK13& )
          CALCULATED-NEGATIVE-Z-OFFSET
..
%
PROPERTY MATERIAL STEEL ELASTIC 2.1E5 0.3 7.85E-9 0.0 0
..
CONNECT MATERIAL STEEL ( ALL-SURFACES-INCLUDED ALL-LINES-INCLUDED )
..
%
SET ELEMENT-TYPE
  SURFACE ALL-SURFACES-INCLUDED SHELL-8NODES
  LINE ( ARC1 LI2 AI&21 AK12& AI&11 AI&31 AK11& AK13& ) BEAM-3NODES
..
%
SET NUMBEROF-ELEMENTS AI2&1 4
                      AK1&2 4
..

```

```

%
SET ELEMENT-LENGTH-RATIO AI1&1 ARITHMETRIC-SEQUENCE 4 1.0
                        LI2 ARITHMETRIC-SEQUENCE P02 4 1
                        AI2&1 ARITHMETRIC-SEQUENCE 1 4
                        AK1&1 ARITHMETRIC-SEQUENCE 4 1
                        ARC1 ARITHMETRIC-SEQUENCE P02 4 1
                        AK1&2 ARITHMETRIC-SEQUENCE 1 4
..
%
PROPERTY BOUNDARY-CONDITION AJ1&1
        FIX FIX FIX FREE FREE FREE
        GLOBAL
..
PROPERTY BOUNDARY-CONDITION AJ1&3
        FIX FIX FREE FREE FREE FREE
        GLOBAL
..
%
PROPERTY LOAD 1 LINE-LOAD AJ3&1 GLOBAL 200 0 0 END
        MIDDLE-SURFACE-SHELL-ELEMENT
..
PROPERTY LOAD 2 LINE-LOAD AJ3&1 GLOBAL 0 0 10 END
        MIDDLE-SURFACE-SHELL-ELEMENT
..
PROPERTY LOAD 3 LINE-MOMENT AJ3&1 GLOBAL 0 3000 0 END
        MIDDLE-SURFACE-SHELL-ELEMENT
..
PROPERTY LOAD 4 NORMAL-PRESSURE ( AS* AU* )
        LINEAR-2POINTS-VARYING AP111 0.015 AP113 0.003 END MIDDLE-SURFACE
..
%
MESH ALL

```

A 1.3 Patran-Pre Input

The FE model created by the following Patran-Pre input is more or less the same as the one displayed by Prefem in Figure A.2 and Figure A.3. Note that after reading the input into Patran-Pre you should interactively write the Input Interface File (T#.FEM) by the following operation in Patran-Pre:

Application: Analysis: Analyze - Selected Model - Interface file

The Patran-Pre input:

```

$ Patran-Pre input for creating a model of a plate connection with a bracket
$
db_set_pref( 303, 3, 0, FALSE, 0.600000002, "" )
ga_view_aa_set( -67., 0., -34. )
point_size( 9 )
$
$ Define geometry
$
STRING asm_create_line_xyz_created_ids[VIRTUAL]

```

```
asm_const_line_xyz( "1", "<0 480 0>", "[0 0 0]", "Coord 0", @
asm_create_line_xyz_created_ids )
$
asm_const_line_xyz( "2", "<0 670 0>", "Point 2", "Coord 0", @
asm_create_line_xyz_created_ids )
$
STRING sgm_sweep_surface_e_created_ids[VIRTUAL]
sgm_const_surface_extrude( "1", "<1200 0 0>", 1., 0., "[0 0 0]", "Coord 0", @
"Curve 1 2", sgm_sweep_surface_e_created_ids )
$
sgm_const_surface_extrude( "3", "<600 0 0>", 1., 0., "[0 0 0]", "Coord 0", @
"Surface 1.2 2.2", sgm_sweep_surface_e_created_ids )
$
sgm_const_surface_extrude( "5", "<0 0 1200>", 1., 0., "[0 0 0]", "Coord 0", @
"Curve 1 2", sgm_sweep_surface_e_created_ids )
$
sgm_const_surface_extrude( "7", "<0 0 600>", 1., 0., "[0 0 0]", "Coord 0", @
"Surface 5.2 6.2", sgm_sweep_surface_e_created_ids )
$
STRING sgm_transform_point_created_ids[VIRTUAL]
asm_transform_grid_translate( "16", "< 0 0 80>", "Coord 0", 1, FALSE, FALSE, @
"Point 5", sgm_transform_point_created_ids )
$
asm_transform_grid_translate( "17", "< 80 0 0>", "Coord 0", 1, FALSE, FALSE, @
"Point 11", sgm_transform_point_created_ids )
$
asm_transform_grid_translate( "18", "< 1800 0 0>", "Coord 0", 1, FALSE, @
FALSE, "Point 14", sgm_transform_point_created_ids )
$
STRING sgm_create_curve_2d_created_ids[VIRTUAL]
sgm_const_curve_2d_arc2point_v2( "3", 1, 0., FALSE, FALSE, 1, "Coord 0.2", @
"Point 18", "Point 16", "Point 17", FALSE, sgm_create_curve_2d_created_ids )
$
STRING sgm_curve_break_par_created_ids[VIRTUAL]
sgm_edit_curve_break_parametric( "4", 0.5, "Curve 3", TRUE, @
sgm_curve_break_par_created_ids )
$# Question from application SGM
$# Do you wish to delete the original curves?
$? YES 38000217
$
STRING sgm_surface_2curve_created_ids[VIRTUAL]
sgm_const_surface_2curve( "9", "Surface 1.3", "Curve 4", @
sgm_surface_2curve_created_ids )
$
sgm_const_surface_2curve( "10", "Surface 5.3", "Curve 5", @
sgm_surface_2curve_created_ids )
$
$ Geomodel finished
$
$ Define mesh seed
$
ui_exec_function( "mesh_seed_display_mgr", "init" )
```

```

mesh_seed_create( "Curve 1 2", 1, 3, 0., 0., 0. )
mesh_seed_create( "Surface 1.1 5.1", 3, 12, 0.25, 0., 0. )
mesh_seed_create( "Surface 3.1 7.1", 3, 4, 4., 0., 0. )
mesh_seed_create( "Curve 4", 3, 11, 4., 0., 0. )
mesh_seed_create( "Curve 5", 3, 11, 0.25, 0., 0. )
mesh_seed_create( "Surface 9.3", 1, 1, 0., 0., 0. )
mesh_seed_create( "Surface 10.3", 1, 1, 0., 0., 0. )
mesh_seed_create( "Surface 9.1", 1, 4, 0., 0., 0. )
$
$ Define mesh
$
INTEGER fem_create_mesh_surfa_num_nodes
INTEGER fem_create_mesh_surfa_num_elems
STRING fem_create_mesh_s_nodes_created[VIRTUAL]
STRING fem_create_mesh_s_elems_created[VIRTUAL]
fem_create_mesh_surf_4( "IsoMesh", 49152, "Surface 1:10", 1, ["100.0"], @
"Quad8", "#", "#", "Coord 0", "Coord 0", fem_create_mesh_surfa_num_nodes, @
fem_create_mesh_surfa_num_elems, fem_create_mesh_s_nodes_created, @
fem_create_mesh_s_elems_created )
$
mesh_seed_display_mgr.erase( )
INTEGER fem_create_mesh_curve_num_nodes
INTEGER fem_create_mesh_curve_num_elems
STRING fem_create_mesh_c_nodes_created[VIRTUAL]
STRING fem_create_mesh_c_elems_created[VIRTUAL]
fem_create_mesh_curv_1( "Surface 1.1 3.1 1.3 3.3 2.3 4.3 5.1 7.1 5.3 7.3 6" // @
".3 8.3 Curve 4 5", 16384, 100., "Bar3", "#", "#", "Coord 0", "Coord 0", @
fem_create_mesh_curve_num_nodes, fem_create_mesh_curve_num_elems, @
fem_create_mesh_c_nodes_created, fem_create_mesh_c_elems_created )
$
REAL fem_equiv_all_x_equivtol
INTEGER fem_equiv_all_x_segment
fem_equiv_all_group3( [" "], 0, "", 1, 0.600000002, FALSE, @
fem_equiv_all_x_equivtol, fem_equiv_all_x_segment )
$
$ Material and properties
$
material.create( "Analysis code ID", 32, "Analysis type ID", 1, "steel", 0, @
"Date: 09-Aug-04 Time: 14:08:06", "Isotropic", 1, "Directionality", @
1, "Linearity", 1, "Homogeneous", 0, "Linear Elastic", 1, @
"Model Options & IDs", ["None", "", "", "", ""], [30, 0, 0, 0, 0], @
"Active Flag", 1, "Create", 10, "External Flag", FALSE, "Property IDs", [ @
"Elastic Modulus", "Poisson Ratio", "Density", "Structural Damping Coeff", @
"Thermal Expansion Coeff"], [2, 5, 16, 30, 24, 0], "Property Values", ["2.1E5" @
, "0.3", "7.85E-9", "0", "0", ""] )
$
elementprops_create( "bracket10", 51, 25, 35, 1, 1, 20, [13, 1080, 1071, 20], @
[5, 1, 3, 9], ["m:steel", "10", "", ""], "Surface 9 10" )
$
elementprops_create( "plate15", 51, 25, 35, 1, 1, 20, [13, 1080, 1071, 20], [ @
5, 1, 3, 9], ["m:steel", "15", "", ""], "Surface 5:8" )
$

```

```

elementprops_create( "plate20", 51, 25, 35, 1, 1, 20, [13, 1080, 1071, 20], [ @
5, 1, 3, 9], ["m:steel", "20", "", ""], "Surface 1:4" )
$
$ Define beam cross sections
$
beam_section_create( "bar6_60", "BAR", ["60", "6"] )
$
elementprops_create( "flatbar", 13, 30, 42, 1, 1, 20, [13, 39, 6, 2097, 2098, @
2081, 7613, 2082, 7614], [5, 11, 2, 1, 1, 2, 2, 2, 4], ["m:steel", "bar6_60", @
"<1 0 0>", "", "", "", "", "", "", ""], "Curve 4 5" )
$
beam_section_create( "T_180_80", "T", ["80", "180", "10", "10"] )
$
elementprops_create( "girder", 13, 20, 42, 1, 1, 20, [13, 39, 6, 7602, 7603, @
2097, 2098, 7608, 7609, 2081, 7613, 2082, 7614], [5, 11, 2, 1, 1, 1, 1, 1, 1, @
2, 2, 2, 4], ["m:steel", "T_180_80", "<-1 0 -1>", " ", "", " ", "", "", "", "", @
, "", "", ""], "Surface 1:7:2.3" )
$
beam_section_create( "L_90_40", "L", ["40", "90", "20", "8"] )
$
elementprops_create( "stiffener", 13, 23, 42, 1, 1, 20, [13, 39, 6, 2097, 2098, @
7601, 7607, 2081, 7613, 2082, 7614], [5, 11, 2, 1, 1, 4, 1, 2, 2, 2, 4], [ @
"m:steel", "L_90_40", "<1 0 1>", " ", " ", " Negative", " ", "", "", "", "", " ", @
"Surface 1:7:2.1 2:8:2.3" )
$
$ Flush beams with plates: Tools - Sesam - AutoCalculate Beam Offsets
$
sesam_offset_rules.set( ["Flush Top", "Flush Bottom", "Flush Left", @
"Flush Right", "Centroid"], ["Surface 1:7:2.1 2:8:2.3", "Surface 1:7:2.3", "", @
"", ""] )
sesam_beam_offset.calculate( 2, ["girder", "stiffener"] )
$
uil_toolbar.wireframe( )
$
$ Loads/boundary conditions
$
loadcase_create2( "lc1", "Static", "", 1., [""], [0], [0.], "", 0., FALSE )
loadcase_create2( "lc2", "Static", "", 1., [""], [0], [0.], "", 0., FALSE )
loadcase_create2( "lc3", "Static", "", 1., [""], [0], [0.], "", 0., FALSE )
loadcase_create2( "lc4", "Static", "", 1., [""], [0], [0.], "", 0., FALSE )
$
loadcase_current_set( "lc1" )
loadsbc_create2( "fix1", "Displacement", "Nodal", "", "Static", ["Curve 1 2"] @
, "Geometry", "Coord 0", "1.", ["<0 0 0", ">", "<", ">"], [ "", "" ] )
$
loadsbc_create2( "fix2", "Displacement", "Nodal", "", "Static", [ @
"Surface 7:8.2"], "Geometry", "Coord 0", "1.", ["<0 0", ">", "<", ">"], [ "" @
, "" ] )
$
loadsbc_create2( "lineload_x", "Element Force[2 node Line]", "Element Uniform", @
"2D", "Static", ["Surface 3:4.2"], "Geometry", "Coord 0", "1.", ["< 200", ">" @
, "<", ">", " " ], [ "", "", "" ] )

```

```

$
loadcase_current_set( "lc2" )
loadsbcsc_create2( "lineload_z", "Element Force[2 node Line]", @
"Element Uniform", "2D", "Static", ["Surface 3:4.2"], "Geometry", "Coord 0", @
"1.", ["< 0 0 10 >", "< >", " "], ["", "", ""])
$
loadcase_current_set( "lc3" )
loadsbcsc_create2( "linemoment", "Element Force[2 node Line]", @
"Element Uniform", "2D", "Static", ["Surface 3:4.2"], "Geometry", "Coord 0", @
"1.", ["< >", "<0 3000 0 >", ""], ["", "", ""])
$
$ Field for variation of pressure load
$
fields_create( "varz", "Spatial", 1, "Scalar", "Real", "Coord 0", "", "Table", @
1, "", "", "Z", "", "", "", FALSE, [0., 1800.], [0.], [0.], [[[0.015]]] @
0.003]] )
$
loadcase_current_set( "lc4" )
loadsbcsc_create2( "pressure", "Pressure", "Element Uniform", "2D", "Static", [ @
"Surface 1:8"], "Geometry", "", "1.", [" ", "f:varz", " "], ["", "", ""])
$
$ Define groups (sets)
$
sys_poll_option( 2 )
ga_group_create( "deck" )
ga_group_entity_add( "deck", "Point 1:9 Curve 1 2 Surface 1:4" )
ga_viewport_group_post( "", "deck" )
$
sys_poll_option( 0 )
sys_poll_option( 2 )
ga_group_create( "bulkhead" )
ga_group_entity_add( "bulkhead", "Point 1:3 10:15 Curve 1 2 Surface 5:8" )
ga_viewport_group_post( "", "bulkhead" )
$
sys_poll_option( 0 )
sys_poll_option( 2 )
ga_group_create( "bracket" )
ga_group_entity_add( "bracket", "Point 16 17 19 Curve 4 5 Surface 9 10" )
ga_viewport_group_post( "", "bracket" )
$
sys_poll_option( 0 )
$
$ Group reorganize
$
bv_group_reorganize( 4, ["default_group", "deck", "bulkhead", "bracket"], [0, @
0, 0, 0, 0, 0], [1, 1, 1, 1] )
$
$ FEM finished
$
$ Write T#.fem file
$ Must be executed manually
$ Application: Analysis: Analyze - Selected Model - Interface file

```

APPENDIX B STRESSES AND OTHER RESULTS

This appendix provides information on and definition of the various stress components. Also, an overview of all result attributes and components is given.

B 1 Element Local Coordinate Systems

B 1.1 Membrane and Shell Elements

The local z-axis for membrane and shell elements is normal to the element and is determined by the pre-processor. The local x-axis is for a linear analysis using Sestra determined as the projection of the X-axis of the superelement onto the membrane/shell surface (the LOCS command of Sestra may, however, specify otherwise). The local y-axis completes a right-handed cartesian coordinate system. However, if the superelement's X-axis has no projection (the element is parallel with the YZ-plane) then the local x-axis will be parallel with the superelement's Y-axis instead. Figure B.1 illustrates this.

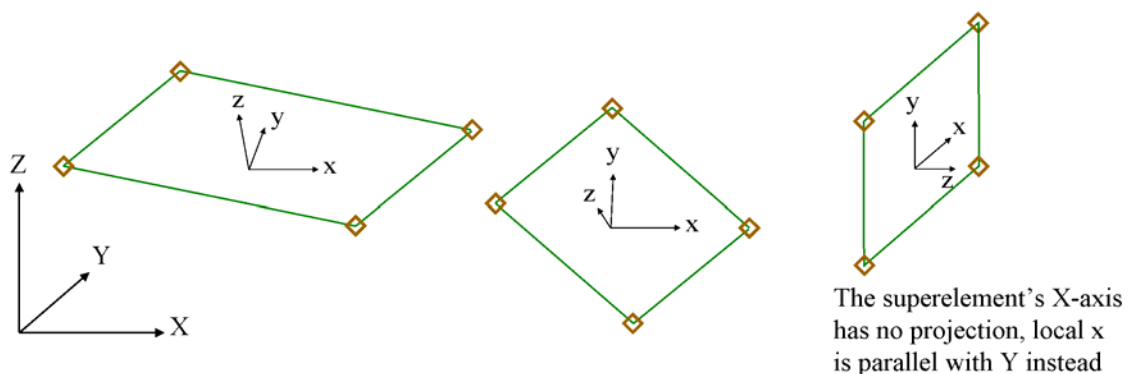


Figure B.1 Local coordinate system for membrane and shell elements

B 1.2 Solid Elements

The local coordinate system of solid elements is parallel with the coordinate system of the superelement.

B 2 Result Attributes and Components

As explained in Section 3.2.1 there are five result positions: Nodes, Elements, Element average, Result-points and Surface resultpoints. Each of these contain a selection of result attributes which in turn contain result components. This section explains the attributes and their components.

B 2.1 Result Attribute: DISPLACEMENT

The displacements are available in the form of the three translational components X, Y, Z and the three rotational components RX, RY, RZ. (The unit of rotations is always in radians.) In addition, the component ALL being the vector sum of the three translational components is available.

B 2.2 Result Attribute: VELOCITY

This attribute is only available for quasi-static and dynamic analysis.

In the case of frequency domain analysis (dynamic or quasi-static) it is computed by Xtract based on the DISPLACEMENT attribute as follows:

- Real velocity component = $-\omega$ * Imaginary displacement component
- Imaginary velocity component = ω * Real displacement component

Note: For a quasi-static analysis Xtract cannot compute the velocities unless the frequencies are available on the results file. This in turn requires the frequencies to be input to Sestra (on FRLO or FRQL, normally as part of the S-file).

In the case of time domain analysis velocities must have been requested to be stored on the results file in Sestra (RESVA option on FRSP command). The components of the attribute correspond to the components of the attribute DISPLACEMENT.

B 2.3 Result Attribute: ACCELERATION

This attribute is only available for quasi-static and dynamic analysis.

In the case of frequency domain analysis (dynamic or quasi-static) it is computed by Xtract based on the DISPLACEMENT attribute as follows:

- Real acceleration component = $-\omega^2$ * Real displacement component
- Imaginary acceleration component = $-\omega^2$ * Imaginary displacement component

Note: For a quasi-static analysis Xtract cannot compute the accelerations unless the frequencies are available on the results file. This in turn requires the frequencies to be input to Sestra (on FRLO or FRQL, normally as part of the S-file).

In the case of time domain analysis velocities must have been requested to be stored on the results file in Sestra (RESVA option on FRSP command). The components of the attribute correspond to the components of the attribute DISPLACEMENT.

B 2.4 Result Attribute: REACTION-FORCE

The reaction forces are three reaction forces, X-FORCE, Y-FORCE and Z-FORCE, and three reaction moments, RX-MOMENT, RY-MOMENT and RZ-MOMENT.

B 2.5 Result Attribute: G-STRESS

This attribute contains the general stress components:

- SIGXX (σ_x) – stress in the direction of the local x-axis – membrane, shell and solid elements
- SIGYY (σ_y) – stress in the direction of the local y-axis – membrane, shell and solid elements
- SIGZZ (σ_z) – stress in the direction of the local z-axis – solid elements
- TAUXY (τ_{xy}) – shear stress in the direction of the local x/y-axes – membrane, shell and solid elements
- TAUXZ (τ_{xz}) – shear stress in the direction of the local x/z-axes – 6 and 8 node shell and solid elements
- TAUYZ (τ_{yz}) – shear stress in the direction of the local y/z-axes – 6 and 8 node shell and solid elements
- VONMISES – von Mises stress – membrane, shell and solid elements

See Section B 1 for information on the local coordinate system of the various elements.

B 2.6 Result Attribute: P-STRESS

This attribute contains the principal stress components:

- P1 – the highest principal stress – membrane, shell and solid elements
- P2 – the second highest (and lowest in case of membrane and shell elements) principal stress – membrane, shell and solid elements
- P3 – the lowest principal stress – solid elements

See Section B 3.3 for information on the calculation of principal stresses.

B 2.7 Result Attribute: PM-STRESS

This attribute contains the principal membrane stress components, i.e. principal stresses based on the membrane stress components only:

- P1 – the highest principal stress – membrane and shell elements
- P2 – the lowest principal stress – membrane and shell elements

Note: For membranes PM-STRESS components are equal to P-STRESS components.

See Section B 3.3 for information on the calculation of principal stresses.

B 2.8 Result Attribute: D-STRESS

This attribute contains the decomposed stress components, i.e. general stresses decomposed in membrane and bending parts. This attribute is relevant for shell elements only.

- SIGMX (σ_{Mx}) – membrane part of stress in the direction of the local x-axis
- SIGMY (σ_{My}) – membrane part of stress in the direction of the local y-axis
- SIGBX (σ_{Bx}) – bending part of stress in the direction of the local x-axis
- SIGBY (σ_{By}) – bending part of stress in the direction of the local y-axis
- TAUMXY (τ_{Mxy}) – membrane part of shear stress in the direction of the local x/y-axes
- TAUBXY (τ_{Bxy}) – twisting part of shear stress in the direction of the local x/y-axes
- TAUBXZ (τ_{Bxz}) – bending related shear stress in the direction of the local x/z-axes
- TAUBYZ (τ_{Byz}) – bending related shear stress in the direction of the local y/z-axes
- MVONMISES – von Mises stress based on membrane stresses

See Section B 3.1 for information on the calculation of decomposed stresses.

B 2.9 Result Attribute: R-STRESS

This attribute contains the stresses integrated through the thickness, often referred to as stress resultants in literature. This attribute is relevant for membrane and shell elements only.

- NXX – normal force per unit length in the direction of the local x-axis
- NXY – shear force per unit length in the membrane/shell surface ($NYX = NXY$)
- NXZ – shear force per unit length normal to the membrane/shell surface in a section normal to the local x-axis

- NYY – normal force per unit length in the direction of the local y-axis
- NYZ – shear force per unit length normal to the membrane/shell surface in a section normal to the local y-axis
- MXX – bending moment per unit length in a section normal to the local x-axis
- MXY – twisting moment per unit length in a section normal to the local x-axis ($MYX = MXY$)
- MYY – bending moment per unit length in a section normal to the local y-axis

See Section B 3.2 for information on the calculation of the stress resultants.

B 2.10 Result Attribute: G-FORCE

This attribute contains the beam forces and moments. This attribute is relevant for beam elements only.

- NXX – axial force
- NXY – shear force in the direction of the local y-axis
- NXZ – shear force in the direction of the local z-axis
- MXX – torsional moment
- MXY – bending moment about the local y-axis
- MXZ – bending moment about the local z-axis

B 2.11 Result Attribute: B-STRESS

This attribute contains beam stresses computed by Xtract based on forces and moments (G-FORCE). This attribute is relevant for beam elements only.

- SIGXX – axial (normal) stress due to axial force alone
- SIGBYX – axial (normal) stress in the most extreme fibre due to moment about y-axis
- SIGBYX2 – axial (normal) stress in the extreme fibre on the “opposite side” of SIGBYX due to moment about y-axis
- SIGBZX – axial (normal) stress in the most extreme fibre due to moment about z-axis
- SIGBZX2 – axial (normal) stress in the extreme fibre on the “opposite side” of SIGBZX due to moment about z-axis
- TAUTX – shear stress due to torsional moment
- TAUXY – shear stress due to shear force in y-direction
- TAUXZ – shear stress due to shear force in z-direction

See Section B 4 for information on the calculation of beam stresses.

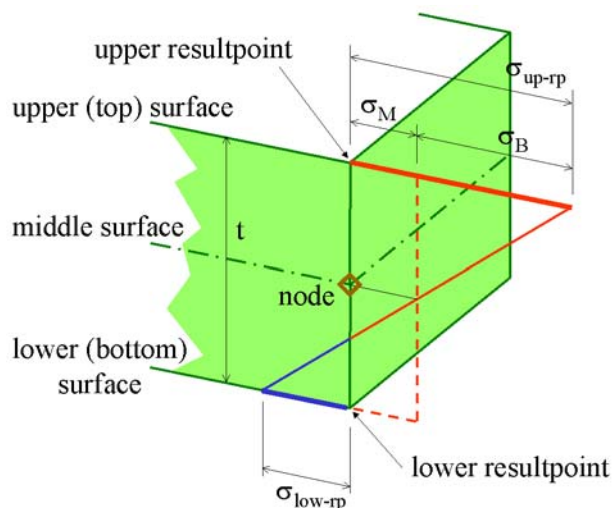
B 3 Derived Stresses for Membrane, Shell and Solid Elements

Several derived stress components are computed for the membrane, shell and solid elements based on the general stress components (G-STRESS). These are (see also Section B 2):

- Decomposed stresses (D-STRESS), i.e. general stresses decomposed into membrane and bending parts
- Principal stresses (P-STRESS)
- Principal membrane stresses (PM-STRESS)
- Stress resultants (R-STRESS), i.e. general stresses integrated through the membrane/shell thickness giving forces and moments per length unit
- von Mises stress
- Maximum and minimum stresses for a complex result case
- Maximum and minimum stresses for a complex result case combined with static (non-complex) result cases

B 3.1 Decomposed Stresses

The D-STRESS components are computed based on the G-STRESS as illustrated in Figure B.2 through Figure B.6.



σ represents σ_x and σ_y

G-STRESS:

σ_{up-rp} is stress in upper resultpoint

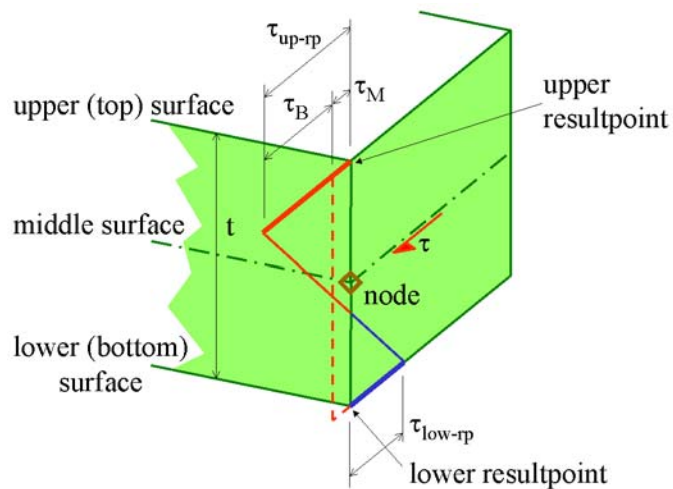
σ_{low-rp} is stress in lower resultpoint

D-STRESS:

$\sigma_M = (\sigma_{up-rp} + \sigma_{low-rp})/2$ is membrane stress

$\sigma_B = (\sigma_{up-rp} - \sigma_{low-rp})/2$ is bending stress

Figure B.2 D-STRESS – normal stress – for 4 node shell elements



τ represents τ_{xy}

G-STRESS:

τ_{up-rp} is shear in upper resultpoint

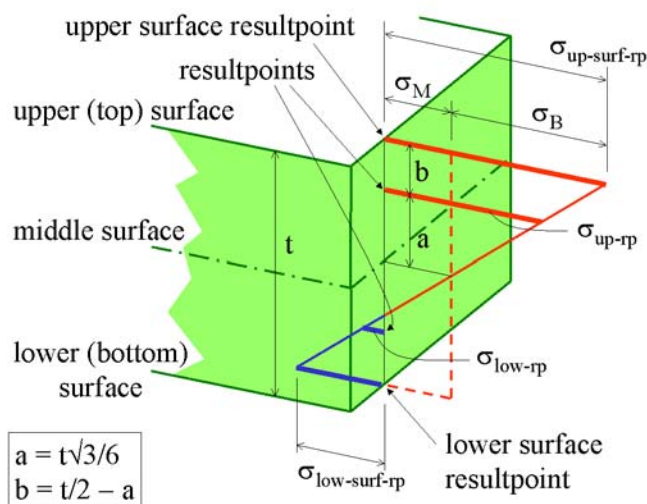
τ_{low-rp} is shear in lower resultpoint

D-STRESS:

$\tau_M = (\tau_{up-rp} + \tau_{low-rp})/2$ is membrane shear

$\tau_B = (\tau_{up-rp} - \tau_{low-rp})/2$ is twisting shear

Figure B.3 D-STRESS – membrane shear – for 4 node shell elements



σ represents σ_x and σ_y

G-STRESS:

σ_{up-rp} is stress in upper resultpoint

σ_{low-rp} is stress in lower resultpoint

$\sigma_{up-surf-rp}$ is stress in upper surface resultpoint

$\sigma_{low-surf-rp}$ is stress in lower surface resultpoint

D-STRESS:

$\sigma_M = (\sigma_{up-surf-rp} + \sigma_{low-surf-rp})/2$ is membr. stress

$\sigma_B = (\sigma_{up-surf-rp} - \sigma_{low-surf-rp})/2$ is bending stress

Figure B.4 D-STRESS – normal stress – for 8 node shell elements

Figure B.4 through Figure B.6 show a section through the result points of an 8 node shell element.

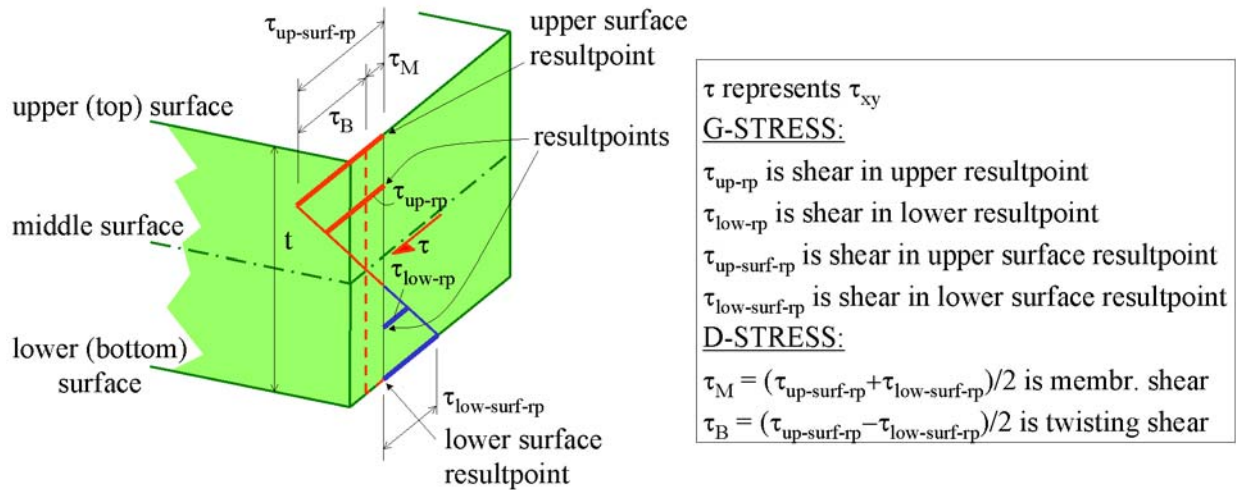


Figure B.5 D-STRESS – membrane shear – for 8 node shell elements

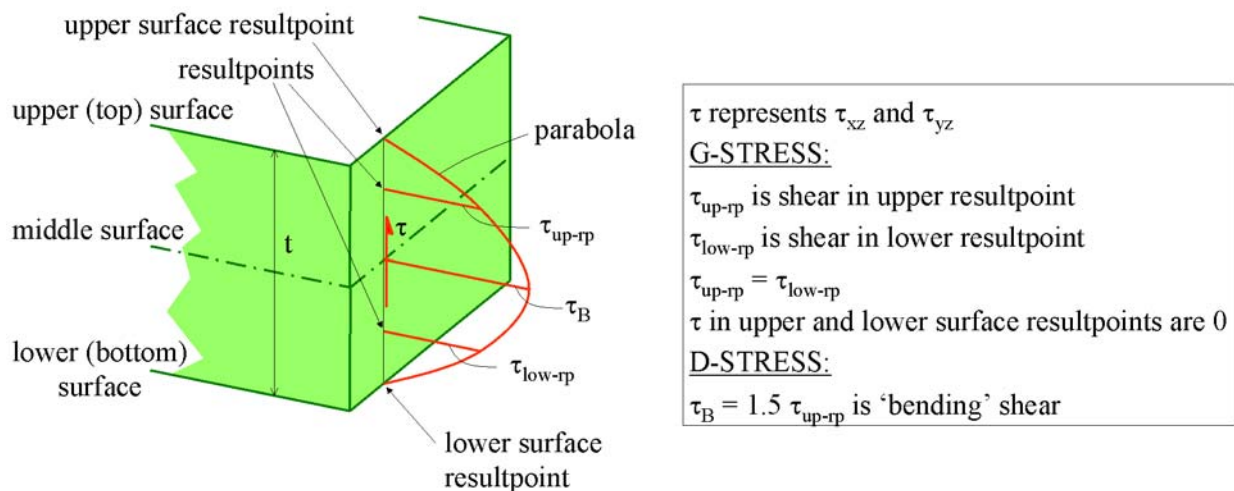
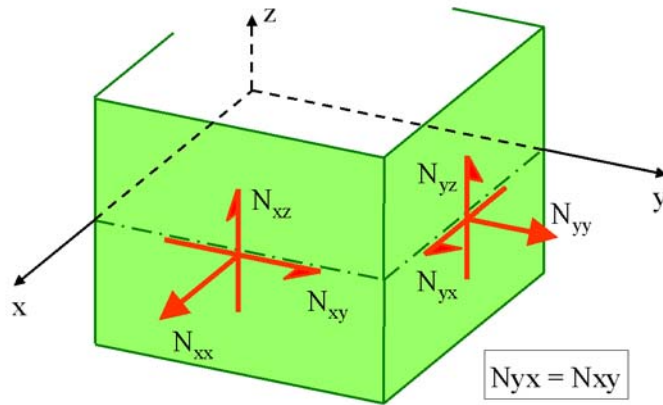


Figure B.6 D-STRESS – ‘bending’ shear – for 8 node shell elements

B 3.2 Stress Resultants

The R-STRESS components are computed based on the D-STRESS as illustrated in Figure B.7 and Figure B.8.

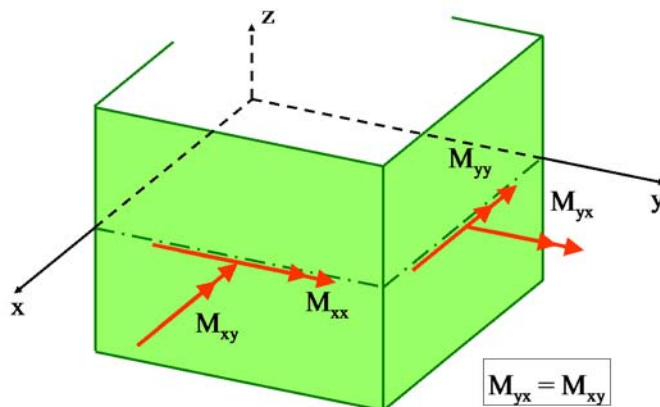
R-STRESS forces:

$$N_{\text{normal}} = \sigma_M t \quad (N_{xx} \text{ and } N_{yy})$$

$$N_{\text{memb-shear}} = \tau_M t \quad (N_{xy})$$

$$N_{\text{bend-shear}} = \tau_{\text{up-rp}} t \quad (N_{xz} \text{ and } N_{yz})$$

Figure B.7 R-STRESS – forces – for 4 and 8 node shell elements (N_{xz} and N_{yz} do not exist for 4 node shell)

R-STRESS moments:

$$M_{\text{bend}} = \sigma_B t^2/6 \quad (M_{xx} \text{ and } M_{yy})$$

$$M_{\text{twist}} = \tau_B t^2/6 \quad (M_{xy})$$

Figure B.8 R-STRESS – moments – for 4 and 8 node shell elements

B 3.3 Principal Stresses

B 3.3.1 Principal Stresses for Thin Shell (2D Problem)

For triangular 3 and quadrilateral 4 node shell elements for which only σ_x , σ_y and τ_{xy} exist the principal stresses P_1 and P_2 are found from (the plus applies to P_1 and the minus to P_2), also see Figure B.9:

$$P_{1,2} = \frac{\sigma_x + \sigma_y}{2} \pm \frac{1}{2} \sqrt{(\sigma_x - \sigma_y)^2 + 4\tau_{xy}^2} \quad (\text{B.1})$$

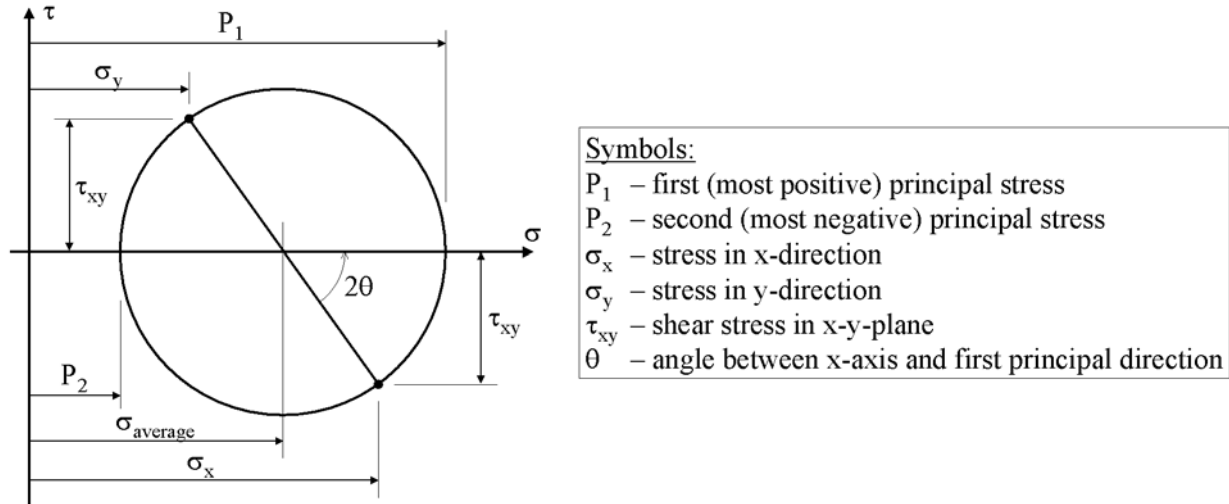


Figure B.9 Mohr's circle for 2D

B 3.3.2 Principal Stresses for Thick Shell

For triangular 6 and quadrilateral 8 node shell elements having an incomplete three by three stress tensor (σ_x , σ_y , τ_{xy} , τ_{xz} and τ_{yz} exist but σ_z does not exist) the computations are based on the full stress tensor as for the solid elements (Section B 3.3.3) except that the smallest of the eigenvalues found is returned as P_2 (the middle eigenvalue is irrelevant).

B 3.3.3 Principal Stresses for Solids (3D Problem)

Solid elements have a complete three by three stress tensor. According to general stress theory the principal stresses of the 3D problem are the eigenvalues P_i of

$$\left(\begin{bmatrix} P & 0 & 0 \\ 0 & P & 0 \\ 0 & 0 & P \end{bmatrix} - \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix} \right) \mathbf{v} = \mathbf{0} \quad (\text{B.2})$$

where $\mathbf{v}$ is a three by three matrix containing the direction cosines of the three principal stress directions (the three eigenvectors). The three eigenvalues of Equation (B.2) are the roots of

$$\det \left(\begin{bmatrix} P & 0 & 0 \\ 0 & P & 0 \\ 0 & 0 & P \end{bmatrix} - \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix} \right) = 0 \quad (\text{B.3})$$

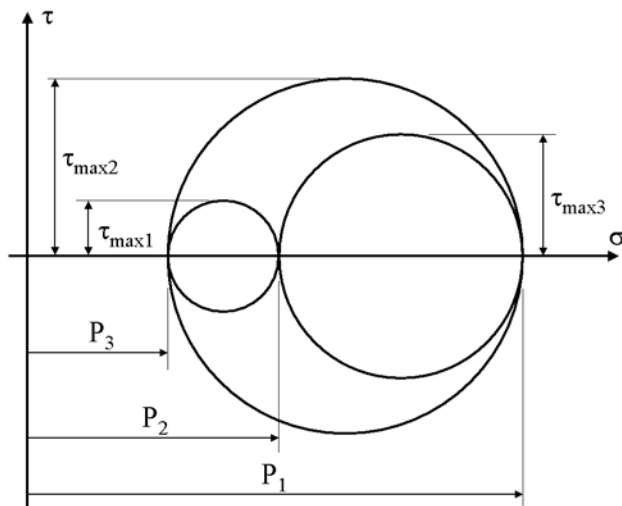
This is the same as solving the cubic equation

$$-P^3 + I_1 P^2 - I_2 P + I_3 = 0 \quad (\text{B.4})$$

where I_i (invariants of the stress tensor) are

$$\begin{aligned} I_1 &= \sigma_x + \sigma_y + \sigma_z \\ I_2 &= \sigma_x \sigma_y + \sigma_y \sigma_z + \sigma_x \sigma_z - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{xz}^2 \\ I_3 &= \sigma_x \sigma_y \sigma_z + 2\tau_{xy} \tau_{yz} \tau_{xz} - \sigma_x \tau_{yz}^2 - \sigma_y \tau_{xz}^2 - \sigma_z \tau_{xy}^2 \end{aligned} \quad (\text{B.5})$$

Equation (B.4) is solved and the eigenvalues are sorted so that $P_1 \geq P_2 \geq P_3$. These are the three principal stresses. Also see Figure B.10.



Symbols:

- P_1 – first (most positive) principal stress
- P_2 – second principal stress
- P_3 – third (most negative) principal stress
- $\tau_{\max 1}$ – max shear stress when rotating about first principal axis
- $\tau_{\max 2}$ – max shear stress when rotating about second principal axis
- $\tau_{\max 3}$ – max shear stress when rotating about third principal axis

Figure B.10 Mohr's circle for 3D

B 3.4 von Mises Stress

The von Mises stress is computed according to the following formulae depending on the type of element.

For thin shell and membrane elements (2D problem):

$$\sigma_{\text{vonMises}} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3\tau_{xy}^2} \quad (\text{B.6})$$

For thick shell element:

$$\sigma_{\text{vonMises}} = \sqrt{\sigma_x^2 + \sigma_y^2 - \sigma_x \sigma_y + 3(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)} \quad (\text{B.7})$$

For solid element (3D problem):

$$\sigma_{\text{vonMises}} = \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x\sigma_y - \sigma_x\sigma_z - \sigma_y\sigma_z + 3(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2)} \quad (\text{B.8})$$

Von Mises stress based on membrane stresses only, i.e. the MVONMISES component of the D-STRESS attribute for thin and thick shell elements:

$$\sigma_{\text{MvonMises}} = \sqrt{\sigma_{Mx}^2 + \sigma_{My}^2 - \sigma_{Mx}\sigma_{My} + 3\tau_{Mxy}^2} \quad (\text{B.9})$$

B 3.5 Sequence of Computation of Stresses

The stresses in the Resultpoints positions, i.e. the stresses read from the results file, are the basis for all stresses presented by Xtract. From the Resultpoints stresses the extrapolated stresses within the elements, the nodal averaged stresses, the decomposed stresses, the derived stresses (von Mises and principal stresses) and combined stresses are found. The sequence in which these computations are made matters. For example, computing von Mises stresses followed by averaging these stresses in the nodes gives a different result than first averaging basic stress components in the nodes followed by computing the von Mises stress. (The latter sequence is the correct procedure followed by Xtract.) When the results are complex the number of calculations both increases and gets more complex which makes the sequence of the computations even more important.

The sequence of these computations is as follows:

- 1 **Resultpoints stresses** are the ones found on the results file.
- 2 **Surface resultpoint stresses** are computed for higher order (6 and 8 node) shell elements in upper and lower surfaces by linear extrapolation.
- 3 **Elements stresses** are computed by bi-linear extrapolation to the nodes, or rather to the upper and lower surfaces at the nodes, from the Resultpoint stresses (from Surface resultpoint stresses in the case of higher order shell elements). In the case of lower order 4 node plate elements no extrapolation is required as the Resultpoint stresses are positioned in the upper and lower surfaces at the nodes.
- 4 **Element average stresses**, positioned in the upper and lower surfaces in the middle of the elements, are found by averaging the element corner values.
- 5 **Nodes stresses** are found in the nodes, or rather in the upper and lower surfaces at the nodes, by averaging element stresses of adjoining coplanar shell/plate elements.
- 6 **D-STRESS stress components** (membrane and bending) are computed for all positions (Surface resultpoints, Elements, Element average and Nodes) as explained in Section B 3.1.
- 7 **Combinations** are created based on stress components (SIGXX, SIGYY, TAUXY, etc.) of the relevant attribute – for complex result cases the combinations are made for all phase steps through the 360 degree cycle, 10 degree intervals by default.
- 8 **Principal and von Mises stresses** are computed as the final step as explained in Section B 3.3 and Section B 3.4, respectively. No stress computations are performed based on these stresses. For complex result cases the principal and von Mises stresses are computed for all phase steps and the phase is determined based on the selected [abs] max/min criterion.

- 9 **Scan cases** are based on (scaled) result cases that individually have been taken through the procedure above before determining result value and result case according to the selected [abs] max/min criterion.

B 4 Beam Stresses Derived from Beam Forces and Moments

The B-STRESS components are found by dividing the forces and moments by areas and minimum sectional moduli. Notice the components SIGBYX2 and SIGBZX2 and how they are computed. The motivation for these two components is explained in Figure B.11.

- $SIGXX = NXX / AREA$
- $SIGBYX = MXY / WYMIN$
- $SIGBYX2 = MXY / WYMIN2$
- $SIGBZX = MXZ / WZMIN$
- $SIGBZX2 = MXZ / WZMIN2$
- $TAUTX = MXX / WXMIN$
- $TAUXY = NXY / SHARY$
- $TAUXZ = NXZ / SHARZ$

The parameters above are:

- $AREA$ – area of beam cross section
- $WYMIN = I_y / z_{max}$ – minimum section modulus about y-axis
- $WYMIN2 = I_y / z_{max2}$ – minimum section modulus about y-axis on the “opposite side” of z_{max} ; see Figure B.11
- $WZMIN = I_z / y_{max}$ – minimum section modulus about z-axis
- $WZMIN2 = I_z / y_{max2}$ – minimum section modulus about z-axis on the “opposite side” of y_{max} ; see Figure B.11
- $WXMIN$ – minimum torsional section modulus about shear centre ($= I_x / r_{max}$ for a pipe)
- $SHARY$ – shear area in the direction of the y-axis. The shear area depends on the type of cross section and is computed by the preprocessor.
- $SHARZ$ – shear area in the direction of the z-axis. The shear area depends on the type of cross section and is computed by the preprocessor.

Note: Except for SIGXX (which is constant over the section) the B-STRESS components will in general occur at different positions and can therefore neither be added nor be used to compute von Mises stress.

Note: Both components SIGBYX and SIGBYX2 will have the same sign as the moment MXY while the two components in reality have opposite signs. This is because it is not taken into account which of the components occur on the positive side of the y-axis and which occur on the negative side (WYMIN and WYMIN2 are always positive). Correspondingly, both components SIGBZX and SIGBZX2 will have the same sign as the moment MXZ.

Note: For sections symmetric about the y-axis the SIGBYX and SIGBYX2 components will be equal. Correspondingly, the SIGBZX and SIGBZX2 components will be equal for sections symmetric about the z-axis.

Note: For box and double-symmetric I sections the SIGBYX and SIGBZX stresses added to the SIGXX gives the highest axial (normal) stress over the section. The signs of SIGBYX and SIGBZX must, however, be set equal to the sign of SIGXX before adding for reasons explained above. Such adding of stresses must be done outside Xtract, for instance in a spreadsheet after saving a table of stresses to file in Xtract.

Note: For pipe sections the SIGBYX and SIGBZX stresses multiplied by $\sqrt{2}$ and added to the SIGXX gives the axial (normal) stress in the 45°/135°/225°/315° points of the section. The signs of SIGBYX and SIGBZX must, however, be set equal to the sign of SIGXX before adding for reasons explained above. Such adding of stresses must be done outside Xtract, for instance in a spreadsheet after saving a table of stresses to file in Xtract.

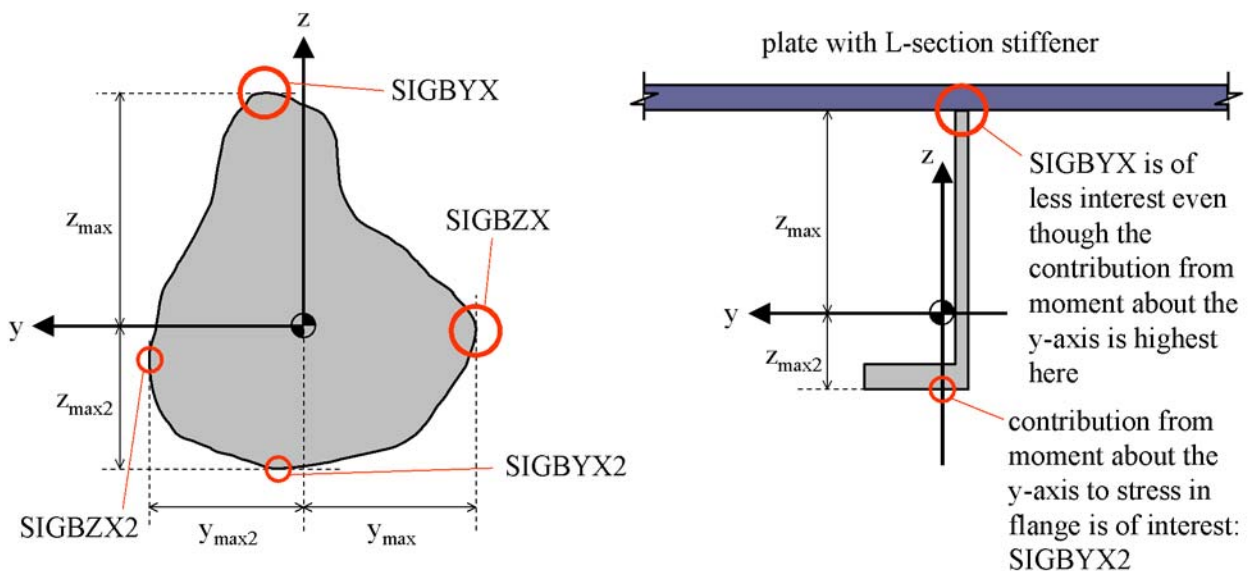


Figure B.11 B-STRESS components due to moments

B 5 Maximum and Minimum Stress Involving Complex Result Cases

This section describes some basic concepts and properties of complex results. Thereafter, relevant manipulations such as combining, scaling and scanning are discussed. Finally, the manipulations of complex results available in Xtract are described.

B 5.1 Basic Concepts and Properties of Complex Results

B 5.1.1 Extracting Physical Response from Complex Results

A complex result case in Sesam is comprised of a real and an imaginary part (as opposed to a non-complex – for example a static – result case that only contains a real part). The real and imaginary parts describe the harmonic (sinusoidal) response and may graphically be presented by Figure B.12.

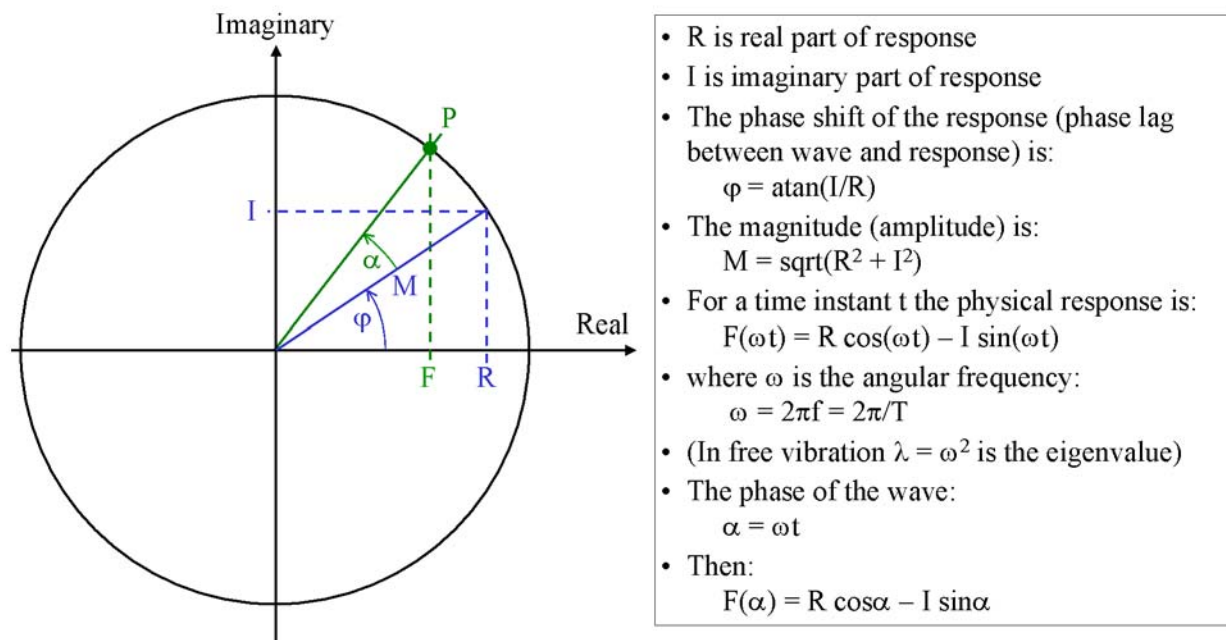


Figure B.12 The complex circle

Note the terminology:

- The term 'static' is used for a non-complex load case.
- $\varphi = \text{atan}(I/R)$ is termed the 'phase shift of the response' or only 'phase shift'.
- $\alpha = \omega t$ is termed the 'phase of the wave' even though the harmonic loading is not necessarily due to a wave in water.

Note that the maximum value of the response $F(a)$ occurs at $\alpha = -\varphi$ (the point P lies on the Real axis on the positive side). A positive φ involves that the response is ahead in time of the wave. In Sesam, $\alpha = 0$ corresponds to the wave crest being positioned above the origin.

B 5.1.2 Stress Variation through a Cycle (360 Degrees)

The basic stress components (σ_x , σ_y , τ_{xy} , etc.) as well as other responses (displacements, forces, etc.) are harmonic. The derived von Mises stress and the principal stresses, however, are non-linear combinations of the basic stress components and therefore non-harmonic. The graph in Figure B.13 illustrates this (the stress components are the ones belonging to a four node shell element). As can be seen the von Mises stress is a non-harmonic variation (always > 0) repeating itself every 180 degree. (For a combination of a static result case and a complex result case the von Mises will be non-harmonic and repeat itself every 360 degree, see Figure B.14.) The first principal stress (P1) will tend to follow the most positive of the stress components σ_x and σ_y .

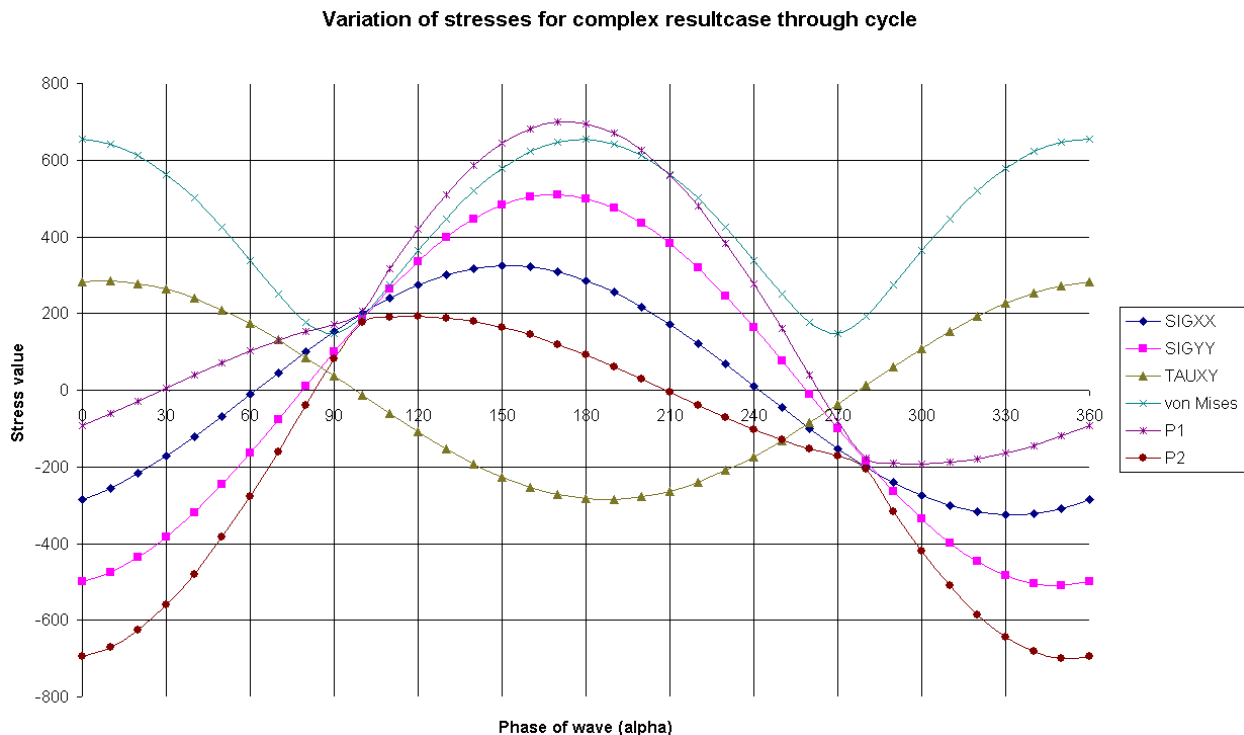


Figure B.13 Typical variation of stresses through a cycle for a complex result case

From the above it is apparent that finding the maximum von Mises and principal stresses through a cycle of a complex (plus a static) result case requires (unless a more advanced method is employed) searching for the maximum value by stepping through the cycle, e.g. by a ten degree interval (default value of Xtract).

B 5.2 Manipulations of Complex Results

B 5.2.1 Presenting a Single Complex Result Case

For a given harmonic attribute (e.g. σ_x) of a complex result case the following values are relevant for presentation:

- The real part, R, corresponding to phase = 0
- The imaginary part, I, corresponding to phase = 270 (or -90)
- The magnitude (amplitude), $M = \text{SQRT}(R^2 + I^2)$, which is the maximum value of the component through the cycle
- The phase shift of the response, $\phi = \text{ATAN}(I/R)$, which is the phase lag between the wave and the component (entering the negative phase shift for phase yields a value of the component equal to the magnitude)
- The response F for a given value of the phase (α) of the incoming wave

For the derived non-harmonic von Mises and principal stresses the following values are relevant for presentation:

- The stress at a given value of the phase (α) of the incoming wave
- The maximum stress found by stepping through the whole cycle (360 degrees)
- The value of the phase of the incoming wave giving the maximum stress above

All these values may be presented by Xtract as explained in Section 3.4.1.

B 5.2.2 Scaling of a Complex Result Case

Scaling a complex result case may be done while maintaining its harmonic nature. This is relevant for scaling results corresponding to a unit wave amplitude to results for a real harmonic wave.

This is available in Xtract through the Results > Scale complex result case command, see Section 3.4.4.

B 5.2.3 Adding Complex Result Cases of Equal Frequencies

Adding (with scaling if desired) the real and imaginary parts, respectively, of two or more complex result cases is meaningful *provided that the complex result cases have the same frequency* (their phase shifts may differ though). The resulting case will be harmonic.

This feature is presently available in Prepost but not in Xtract. There is no control in Prepost as to whether the frequencies of the complex cases to be combined are in fact equal. The user is therefore not restrained from creating meaningless combinations of complex cases of different frequencies. The combination is termed 'quasi-static' and is assigned zero frequency.

B 5.2.4 Combining Several Static and Several Complex Result Cases

Generally, in order to combine a complex result case with other result cases it must first be evaluated for a phase of the incoming wave (α). This because adding a constant value (static result case) and a harmonic function does not yield a harmonic function and neither will two harmonic functions with different frequencies.

Combining a static result case (or any number of such), denoted by subscript s , and several complex result cases, denoted by subscripts 1, 2, 3, ... can be done by specifying a phase of the wave for each complex case thereby evaluating each complex case prior to adding the response:

$$F(\alpha) = f_s R_s + f_1(R_1 \cos \alpha_1 + I_1 \sin \alpha_1) + f_2(R_2 \cos \alpha_2 + I_2 \sin \alpha_2) + f_3(R_3 \cos \alpha_3 + I_3 \sin \alpha_3) + \dots$$

Normally, the complex result cases are not correlated, i.e. they (the incoming waves) may occur with any possible combination of phase of the wave. Therefore, the user normally wants to find the combination of phases ($\alpha_1, \alpha_2, \alpha_3, \dots$) giving the maximum von Mises or principal stress. As demonstrated above, the maximum von Mises for a single complex result case is found by stepping through the cycle. Therefore – unless some assumptions or simplification are made – finding the maximum for a combination of several complex result cases involves evaluating all possible combinations of steps through the cycle of all complex cases, a task that increases exponentially in size when number of complex cases increases. Therefore, combining several complex result cases is normally of little practical interest.

Xtract will not allow the user to put more than a single complex result case into a combination.

B 5.2.5 Combining Several Static Result Cases with a Single Complex Result Case

Combining one or more static (non-complex) result cases with a single complex result case is a highly relevant feature for all postprocessors dealing with complex results. The task is to determine a phase for the complex result case, evaluate the complex case for this phase and finally add (with factors) this evaluation with the static case(s). The phase may be determined either by simply specifying it or as the phase giving maximum or minimum of a certain stress component. Xtract provides this feature through the Results > Combine command, see Section 3.4.2.

For illustration purposes a graph of the stress variation through the cycle for a combination of a static result case and a complex result case is shown in Figure B.14. (The stress components are the ones belonging to a four node shell element.) The von Mises stress is non-harmonic and repeats itself every 360 degree. The principal stresses, P1 and P2, are also non-harmonic.

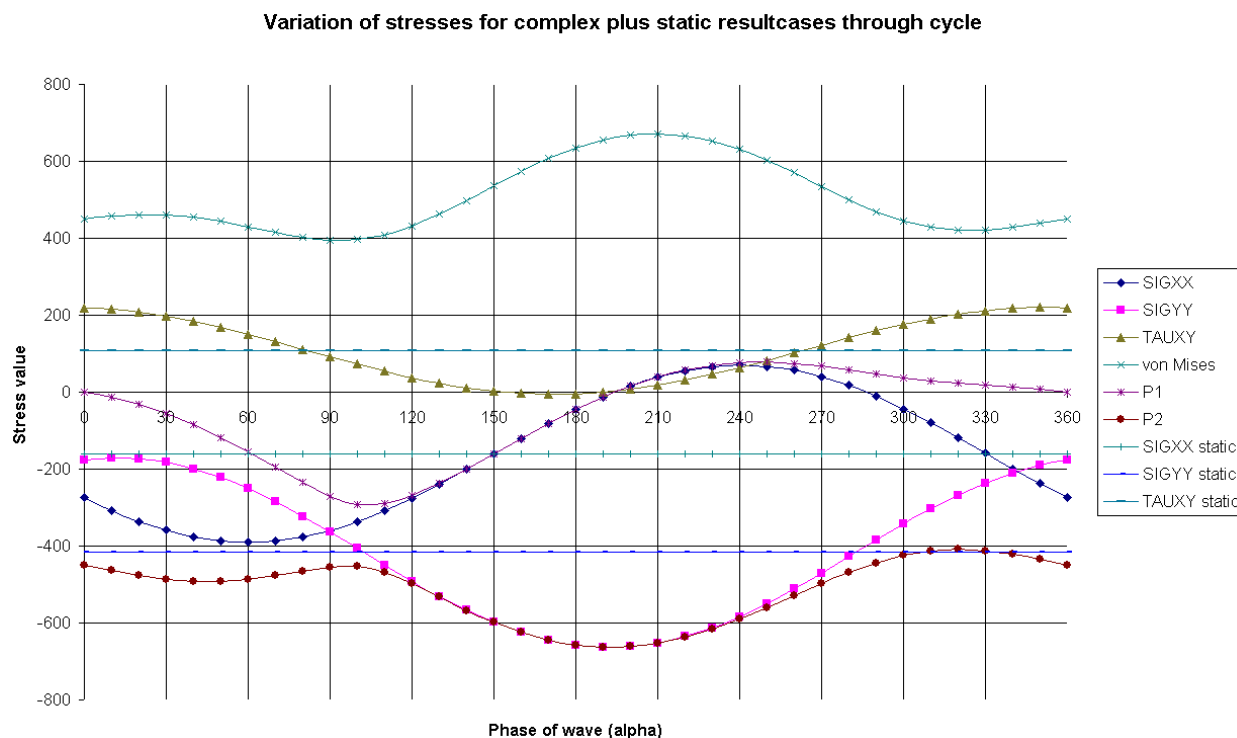


Figure B.14 Typical variation of stresses through a cycle for a complex plus a static result case

B 5.2.6 Scanning Complex Result Cases

Scanning several complex result cases for the maximum or minimum value of a certain stress component is also a required postprocessing feature in connection with complex results. The task is to determine a phase for each complex result case, evaluate the complex cases for these phases and finally select the maximum/minimum value of the stress component. The phase may be determined either by simply specifying it or, more relevantly, as the phase giving maximum/minimum of the stress component in question. Xtract provides this feature through the Results > Scan command, see Section 3.4.3.