

SESAMTM USER MANUAL

GENIE VOL. VI Menu Description



Concept design and analysis of offshore & maritime structures

DET NORSKE VERITAS

This page is blank for printing

SesamTM User Manual GeniE Vol. VI

Menu Description

Concept design and analysis of offshore structures

11 April 2011

Valid from program version 5.3

Developed and Marketed by DET NORSKE VERITAS

DNV Software Report No.: 00-000 / Revision 0, 11 April 2011

Copyright © 2011 Det Norske Veritas Software

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 Software 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.

11 April 2011

GeniE User Manual Menu Description Version 5.3

Table of Contents

1.	INTRODUCTION	9
2.	PULLDOWN MENUS	
21	THE FILE DUILL DOWN MENTL	14
2.1	2.1.1 New worksnace	14
2	2.1.1 Onen workspace	14 14
2	2.1.2 Save workspace	
2	2.1.2 Suve workspace	
	2.1.4 Set Default workspace folder	
2	2.1.4 Set Dejauti workspace jotaer	
4	2.1.5 Explore Current workspace	
4	2.1.0 Save graphics as	
4	2.1.7 Print graphics	
2	2.1.8 Save report	
	2.1.8.1 Structure	
	2.1.8.2 Properties	
	2.1.8.5 Masses	
	2.1.8.4 LOads	
	2.1.8.5 FEM Results	
	2.1.8.0 Explanation of the rEW Results settings	
	2.1.8.7 Plate code check	
2	2.1.0.0 Import	23
4	2191 XMI Concept Model	23
	2.1.9.1 FFM file	23
	2.1.9.3 SACS file	23
	2.1.9.4 STRUCAD 3D file	24
	2.1.9.5 ACIS SAT file	
	2.1.9.6 Intergraph PDS (SDNF file)	
	2.1.9.7 CadCentre PDMS (SDNF file)	
	2.1.9.8 Section Library	
	2.1.9.9 Rule Loads XML file	
	2.1.9.10 External Results SIN file	
	2.1.9.11 GENSOD file	
2	2.1.10 Export	
	2.1.10.1 FEM file	
	2.1.10.2 Results SIN file	
	2.1.10.3 XML Concept file	
	2.1.10.4 Intergraph PDS (SDNF file)	
	2.1.10.5 CadCentre PDMS (SDNF file)	
	2.1.10.6 GeniE journal file (JS)	
	2.1.10.7 Rule Loads XML file	
	2.1.10.8 ACIS SAT file	
2	2.1.11 Read command File	
2	2.1.12 Recent command file	
2	2.1.13 Recent workspaces	
2	2.1.14 Exit	
2.2	THE EDIT PULLDOWN MENU	
2	2.2.1 Undo/Redo	
2	2.2.2 Undo/Redo Dialog	

223 Set 1	Indo Mark	32
2.2.5 Ser Con	,	
$2.2.4 Cop_1$	4.0	
2.2.5 Dete		
2.2.0 Prop	iernes	
2.2.6.1	Section	
2.2.0.2	Material	
2.2.0.5	Fifactive flance	
2.2.0.4	Correction addition	
2.2.0.5	Plate type	
2.2.0.0	Mesh Property	
2.2.6.8	Load interface	38
2.2.6.9	Hinge	
2.2.6.10	Reinforcement	
2.2.6.11	Hydro property	
2.2.6.11	.1 Flooding	
2.2.6.11	.2 Hydrodynamic diameter	
2.2.6.11	.3 Conductor Shielding	40
2.2.6.11	.4 Element refinement	40
2.2.6.11	.5 Morison	40
2.2.6.11	.6 Marine growth	42
2.2.6.11	.7 Air drag	43
2.2.6.12	Beam Type	44
2.2.6.12	.1 Nonstructural	
2.2.6.12	2 Truss	
2.2.6.12	.3 Shim	
2.2.6.12	.4 Inner beam	
2.2.6.13	Pile Unaracteristics	
2.2.0.14	Content	40
2.2.0.13	Mesh Ontion	40
2.2.0.10	Permeable	47 47
2.2.0.17	Structure type	
2.2.0.10 2.2.7 Rule	s	48
2.2.7 1	Joint creation	48
2.2.7.2	Joint design	48
2.2.7.3	Tolerances	
2.2.7.4	Connected move	
2.2.7.5	Geometry	50
2.2.7.6	Units	50
2.2.7.7	Meshing	50
2.2.7.7.	General	51
2.2.7.7.2	2 Max/min angle	53
2.2.7.7.3	3 Jacobi	54
2.2.7.7.4	Eliminate edge	54
2.2.7.7.	5 Chord height	
2.2.7.8	Sets	
2.2.7.9	Compability	
2.2.8 Lice	nses/features	
2.2.8.1	CurvedGeometry	56
2.2.8.2	FrameCodecheck	
2.2.8.3		
2.2.8.4	Enforce use of GeniE.life license	
2.2.8.3 2.3 The Very err	Do not snow uns dialog again	
2.3 THE VIEW PUL 2.21 V^2		/ ۲۵/
2.3.1 VIEW	Tak	/ 3 ج
2.3.2 View		
2.3.3 View	y – Status bar	57
2.3.4 View	v – Refresh graphics	57
2.3.5 View	v – Toolbars	57
2.3.5.1	Main	57
2.3.5.2	View manipulation	57
2.3.5.3	Label	57
2.3.5.4	Selection	
2.3.5.5	Object Types	58

V	ersion	5.	3
		-	-

2356	Create Methods	58
2.3.5.7	Default Properties	
2.3.5.8	Loadcase	58
2.3.6 View	v – Options	58
2.4 THE INSERT P	JLLDOWN MENU	59
2.4.1 Beau	<i>n</i>	59
2.4.1.1	Straight Beam Dialog	59
2.4.1.2	Straight Beam	60
2.4.1.3	Straight Overlapping Beam	60
2.4.1.4	Pile	60
2.4.1.5	Curved Beam	60
2.4.2 Plat	Elat Diata Dialag	01
2.4.2.1	Flat Plate Dialog	01
2.4.2.2	Flat Plate	01 62
2.4.2.4	Skin/Loft Curves	62
2.4.2.5	Flat region	62
2.4.2.6	Curve-Net Interpolation	63
2.4.2.7	Sweep Curve	63
2.4.3 Supp	port	64
2.4.3.1	Support Point Dialog	64
2.4.3.2	Rigid Link Dialog	65
2.4.3.3	Support Point	65
2.4.3.4	Support Curve	66
2.4.4 Join	t	66
2.4.4.1	Joint Dialog	66
2.4.4.2	Joint.	66
2.4.4.3		07 67
2.4.5 Mus	Uniform Doint Mass	07
2.4.5.1	Generic Point Mass	07
2.4.6 Con	orartment Manager	07 68
247 Fear	pur men manager ure Edoe	68
2.1.7 I eu	ar Slicer	00 68
2.4.0 Eme	ling Geometry	00 60
2.4.9.1	Guide Plane Dialog	69
2.4.9.2	Poly Curve Dialog	
2.4.9.3	Guide Line Dialog	70
2.4.9.4	Fillet Curves Dialog	71
2.4.9.5	Guide Point Dialog	71
2.4.9.6	Guide Plane	71
2.4.9.7	Guide Point	72
2.4.9.8	Guide Line	72
2.4.9.9	Guide Spline	12
2.4.9.10	Guide Arc Elliptic	73
2.4.9.12	Guide Circle	73
2.4.9.13	Model Curve	74
2.4.9.14	Fillet Curves	74
2.4.10 P	rofile	75
2.4.11 E	quipment	75
2.4.11.1	COG offset	76
2.4.11.2	Specify footprint	76
2.4.11.3	Linear varying loads	77
2.4.11.4	Position	77
2.4.12 E	xplicit Load	78
2.4.12.1	Point Load	78
2.4.12.2	Life Load	/8/ 70
2.4.12.3 2 4 12 3	1 Pressure	۲۶ ۸۵
2.4.12.3	.2 Traction	
2.4.12.3	.3 Component Load	81
2.4.12.3	.4 Dummy Hydro Pressure	81
2.4.12.4	Prescribed Displacements	82

11 April 2011

Version 5.3

241	2.5 Line Temperature	82
2.4.13	Load Case	
2.4.14	Load Combination	
2.4.15	Insert – Environment	85
2.4.1	5.1 Insert Location	85
2.4.1	5.2 Insert Deterministic Time Condition	86
2.5 The To	OLS PULLDOWN MENU	87
2.5.1	Tools – Analysis	
2.5.1	.1 Create Mesh	
2.5.1	2 Activity Monitor	88 00
2	5.1.2.1 Linear structural analysis - static	
2	5.1.2.3 Wave Load Activity	
2.:	5.1.2.4 Pile Soil Analysis	
2.5.1	3 Export FEM	
2.5.1	.4 Import External Results SIN file	108
2.5.1	5 Frame Code Check	
2.5.1	.6 Advanced Results (Xtract)	109
2.5.1	8 Presentation	110 110
2.5.1	9 Beam Force/Stress Diagram	
2.5.1	10 Show Analysis and Results	
2.5.2	Equipment	112
2.5.2	.1 Import Weight List	112
2.5.3	Properties	113
2.5.3	.1 Create Scaled Materials	113
2.5.4	Structure	114
2.5.4	.1 Geometry	
2.:	5.4.1.1 Simplify Topology	
2.:	5.4.1.2 Heal Structure	114 115
2.5.4	2 Punch	115
2.5.4	3 Curve Punch	
2.5.4	4 Split	116
2.5.4	.5 Verify	116
2.5.4	.6 Stand-Alone Beams	117
2.5.5	Dimension	118
2.5.5	.1 Create Dimension	
2.3.3.	.2 Angle Between	118
2.3.0	Lusionise	110
2.5.0	2 Default names	110 119
2.6 THE HE	LP PULLDOWN MENU	
2.6.1	Help Topics	120
2.6.2	Status Lists on the Web	120
2.6.3	DNV Software on the Web	120
2.6.4	Memory Monitor	121
2.6.5	About GeniE	121
Tool bu	ttons	122
2.7 THE MA	IN TOOLBAR	
2.8 THE VIE	W MANIPULATION TOOLBAR	
2.9 THE LO.	ADCASE TOOLBAR	124
2.10 THE LA	adely 100ldak Rifeyt Types tooi rar	124 125
2.12 THE C	REATE METHODS TOOLBAR	
2.13 THE SE	ELECTION TOOLBAR	
3. THE	BROWSER MENU	128
3.1 ANALYS	SIS	
3.2 CAPACI	ΓΥ	
3.2.1	Create Members	130
3.2.2	Create Panels	131
3.2.3	Create Joints	131

3.2.4 Add R	un	131
3.2.4.1	Loadcases	131
3.2.4.2	General	132
3.2.4.2.1	AISC ASD 2005	132
3.2.4.2.2	AISC LRFD 2005	132
3.2.4.2.3	API LKFD 2003	.133
3.2.4.2.4	API WSD 2002	.134
3.2.4.2.5		.134
3.2.4.2.0	CSR Duik	135
32428	Danish Standard DS 412 / DS 449	136
3.2.4.2.9	Eurocode3 EN 1993-1-1 2005	136
3.2.4.2.10	ISO 19902 2007	138
3.2.4.2.11	NORSOK N-004 2004	139
3.2.4.3	Member and panel	139
3.2.4.3.1	AISC ASD 2005	140
3.2.4.3.2	AISC LRFD 2005	141
3.2.4.3.3	API LRFD 2003	142
3.2.4.3.4	API WSD 2002	143
3.2.4.3.5	API WSD 2005	144
3.2.4.3.6	CSR Bulk	145
3.2.4.3.7	USK Tank	146
3.2.4.3.8	Danish Standard DS 412 / DS 449	147
3.2.4.3.9	EUROCODES EN 1995-1-1 2005	14/
3.2.4.3.10	NORSOK N-004 2007	140 170
3244	loint	150
32441	APLI RFD 2003	150
3.2.4.4.2	API WSD 2002	
3.2.4.4.3	API WSD 2005	
3.2.4.4.4	ISO 19902 2007	152
3.2.4.4.5	NORSOK N-004 2004	152
3.2.5 Run A	<i>ll</i>	153
3.2.6 Code	Check Status	153
3.2.7 Gener	ate Code Check Loads	153
3.2.8 Execu	te Code Checks	154
3.2.9 Unda	te Members from Structure	154
3.2.10 Un	date Structure from Member	154
3 2 11 Set	active	154
3 2 12 Edi	it Description	155
3 2 13 De	loto	155
3.2.15 Del	1918	155
3.2.14 Ref $3.2.15$ P_{ref}	ume	155
3 3 ENVIRONMENT	<i>pc1ues</i>	156
3.3 Environment		150
3311 AU	Pronerties	150
3.3.1.2	New Wind Profile	
3.3.2 Direc	tions	160
3.3.2.1	Properties	
3.3.2.2	New Direction	160
3.3.2.3	New Direction set	160
3.3.3 Soil		161
3.3.3.1	Properties	161
3.3.3.2	New Sand	161
3.3.3.3	New Clay	162
3.3.3.4	New Scour	162
3.3.3.5	New Soil Data	163
3.3.3.6	New Soil Curves	163
3.3.4 Water		164
3.3.4.1	Properties	164
3.3.4.2	New Current Profile	164
3.3.4.3	New frequency set	165
3.3.4.4	New Phase Set	165
3.3.4.5	new wave height	166

DET NORSKE VERITAS SOFTWARE

Version 5.3

GeniE

334	5.1 New Wave Height Function	166
334	5.2 New Wave Height Surface	100
3.3.4	5.3 New Wave Height Set	
3.3.4.6	New Regular Wave Set	
3.4 EQUIPMENT		
3.5 PROPERTIE	5	
3.6 STRUCTURE	2	171
3.7 UTILITIES		171
3.7.1 M	esh Priority Sets	172
4. CONT	EXT SENSITIVE MENU	173
4.1 OBJECT TY	е Веам	
4.1.1 Ed	lit Beam	174
4.1.1.1	Local system	
4.1.1.2	Offset vector	175
4.1.1.3	Hinges	
4.1.1.4	Split points	
4.1.1.5	Move End	176
4.1.1.6	Translate	
4.1.1.7	Buckling factors	
4.1.2 Ce	entre of Gravity	177
4.1.3 Ce	ppy/Move Beam	177
4.1.3.1	Translate	
4.1.3.2	Kotate	l//
4.1.3.3	MIIIOI	1/0
4.1.3.4	3 Point Position	170
4136	General transformation	179
414 G	enerate Ioints	180
415 Io	in	180
416 D	wide	181
4.1.0 D	war Curvas	181
4.1.7 CC	slata	181
4.1.0 D	nene	101
4.1.9 Ke	nume Dronartias	102
4.1.10	Porm Posult Diagrams	102
4.1.11	Leholo	102
4.1.12	Calaur Calina	103
4.1.15		184
4.1.14	Mesh Locking	185
4.1.15	Namea Set	185
4.1.16	View Options	186
4.1.17	Visible Model	186
4.2 OBJECT TYL	YE PLATE	
4.5 OBJECT TY	e Iquipmen I	
4.4 OBJECT TY	e support Point	
4.6 ORIECT TVI	DESTRORT CHRVE	188
4.7 OBJECT TYL	PERFLORT LOAD	
4.8 OBJECT TYL	e Guide Plane	
4.9 OBJECT TYL	e Guide Line	
4.10 OBJECT T	PE WATER SURFACE	
4.11 OBJECT T	/pe Water Layer	
4.12 OBJECT T	/PE AIR LAYER	
4.13 OBJECT T	/pe Seabed	
4.14 OBJECT T	PE SOIL LAYER	
4.15 OBJECT T	PE SOIL BORDER	
4.16 UBJECT T	(PE MESH	
4.17 OBJECT T	Dedesien	
4.17.1	Computer Manch on Deven and an	191
4.17.2	Capacity Member Properties	192
4.17.3	Capacity Member Labels	192
4.17.3.1	Buckling Factor	
4.17.3.2	Description	

4.1	17.3.3	LoadCase	
4.1	17.3.4	Name	
4.1	17.3.5	Results	
4.1	17.3.6	Clear Labels	
4.17.	4	Capacity Member Color Code	
4.18 Овј	ECT TY	PE PLATE CAPACITY PANEL	
4.18.	1	Capacity Panel Properties	
4.18.	2	Capacity Panel Labels	
4.1	18.2.1	Capacity Model Local Coordinate System	
4.1	18.2.2	Description	
4.1	18.2.3	Idealization Method	
4.1	18.2.4	LoadCase	
4.1	18.2.5	Name	
4.1	18.2.6	Results	
4.1	18.2.7	Clear Labels	
4.18.	3	Capacity Panel ColorCode	
4.18.	4	Capacity Panel – CSR Tank Specific	
4.1	18.4.1	Start PULS Advanced Viewer – Current loadcase	
4.1	18.4.2	Start PULS Advanced Viewer – All loadcases	
4.1	18.4.3	Start PULS Excel Spreadsheet – Current loadcase	
4.1	18.4.4	Start PULS Excel Spreadsheet – All loadcases	
5. SI	HORT	COMMANDS AND WINDOWS COMPLIANCE	
6. TI	HE CO	OMMAND LINE INTERFACE SYSTEM	
6.1 The	GeniE .	Sript Command Reference	
6.2 Usef	UL SCR	PT COMMANDS	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	9	11 April 2011

1. INTRODUCTION

The purpose of this document is to list the available commands in GeniE. There are four alternative ways of accessing a command:

- From the pulldown menu
- From the tool button menu
- From the context sensitive menu when selecting an object(s) in the graphical window or from the browser
- From the command line interface window using the script language

The commands are listed in the following. For practical usage of the commands, reference is made to User Manuals Volume 1 through 5 as well as the tutorials found under GeniE's help site. You should have access to all the program extensions of GeniE to see all commands.

The script language is documented on GeniE's help site. For some of the commands, the relevant script command is given.

The program defaults for relevant menus are also shown.

Example on a pulldown menu:



11 April 2011

Example on a tool button menu



Example on a context sensitive menu (select object from graphic window or browser):



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	11	11 April 2011

Example on command line interface window using the script language:

😤 C:\DNV\Workspaces\GeniE\Deck\deck.gni - Genie	
<u>File Edit View Insert Tools H</u> elp	
│ □ ☞ ■ Q × ? 💁 ै ସ ≍ ४ ! </ ♦ ₽ ₽ ₽ ₽ .</td <td>🖉 🔹 Modelling - Structur 💌 🛛 LC_mass 💌 🗖 🕬 Name</td>	🖉 🔹 Modelling - Structur 💌 🛛 LC_mass 💌 🗖 🕬 Name
<mark>/ - □ - Ⅲ / - 条 - K</mark> 四 - 沈 認 % - ♀ 示 酸 ┣ 谷	- 12 - 12 - 12 12 2 42 12 12 12 12 12 12 12 12 12 12 12 12 12
deck Analysis Analysis Analysis Capacity Capa	
<pre>x Bm500_UserManual = Bm1.copyTranslate(Vector3d(2) >Bm1 Bm1 Bm100 Bm100 Bm102 </pre>	5m,0m,0m));
Messages Command Line Visual Clipboard Defaults	<u>}</u>
	NUM //

The user interface has a browser, a tab area, status bars in addition to the graphical window.



2. PULLDOWN MENUS

Pulldown menus are those menus available from the graphical window as shown to the right.

C:\Dnv\WorkSpaces\GeniE\Desc
File Edit View Insert Tools Help

There are 6 categories of commands available from the pulldown menu. These are explained below each with a detailed description of their subcommands in the following Sections.

Command category	Subcommand	Explanation
	New workspace	Make a new project
	Open Workspace	Open an existing project
	Save Workspace	Save the active project session
	Close Workspace	Close the active project session
	Set Default Workspace Folder	Specify default storage location
	Explore Current Workspace	Explore your current workspace's folder
spu	Save Graphics As	Make a picture
ımaı	Print Graphics	Print a picture
Соп	Old Save Report	Make a report using the previous template
File	Save Report	Make a report
	Import	Import data from various file formats
	Export	Export data to various file formats
	Read Command File	Read and execute a journal file
	Recent Command Files	Read and execute a journal file from 10 last sessions
	Recent Workspaces	Open a project from 10 last sessions
	Exit	Exit the program
	Undo	Go one step backwards
	Redo	Go one step forward
sl	Undo/Redo Dialog	Specify the step forward or backward from a list
nand	Set UndoMark	Specify a user defined undo mark
omn	Сору	Copy an object
lit C	Delete	Delete an object
Ed	Properties	Define or edit properties
	Rules	Specify rules for a number of automatic program actions
	Licenses/features	Manage which licenses to be used by GeniE

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	13	11 April 2011

Command category	Subcommand	Explanation
	Refresh Graphics	Clean up the graphics and remove all labels
	Browser	View browser or not
ands	Tabs	View the tabs or not
View Comma	Status Bar	View the status bar or not
	Toolbars	Decide which toolbars to see
	Restore Window Default Settings	Places the Genie window in the middle of the scre default size and setup of the interface.

ands	Tabs	View the tabs or not	
mm ⁶	Status Bar	View the status bar or not	
√ Co	Toolbars	Decide which toolbars to see	
View	Restore Window Default Settings	Places the Genie window in the middle of the screen with default size and setup of the interface.	
	Options	Create and modify view settings	
	Beam	Define a beam	
	Plate	Define a plate	
	Support	Define a support point or line	
	Joint	Define a joint concept	
	Mass	Define a mass object	
×	Compartment	Automatically create compartments	
and	Feature Edge	Insert a mesh control line	
Comm	Linear Slicer	Define a tool used for computing and adjusting target forces and moments, e.g. in length direction of a floating vessel.	
sert	Guiding Geometry	Insert guiding geometry	
ų	Profile	Define a 2D profile used in punch operations	
	Equipment	Define an equipment	
	Explicit Load	Define explicit load like e.g. point/line/pressure load	
	Load Case	Define a load case	
	Load Combination	Create load combinations	
	Environment	Define the environment for wind/wave/pile-soil analysis	
	Analysis	Control analysis activities and look at results	
ands	Equipment	Import weight lists	
mm.	Properties	Scale mass materials to target value	
C	Structure	Perform punching and validation of geometry	
lool	Dimension	Find length and angles	
	Customize	Set defaults for printing and naming conventions	
<u>x</u>	Help Topics	Access to user documentation, tutorials and script language	
lp nand	Status Lists on the Web	Access to on-line status list	
He	DNV Software on the Web	Access to DNV Software web site	
Ŭ	About GeniE	Lists sub-contracted software used in GeniE	

2.1 The File pulldown menu

The File pulldown menu is where you can create, save and work from existing projects.

You can also import and export data from your computer or from the internet, i.e. GeniE's help pages.

See also Section 3.1 of User Manual Volume 1 for more details.

2.1.1 New workspace

Purpose: Make a new project

This command is not scripted. If non-default database units are selected these will be logged. Typically when selecting "mm" as database units: 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;

GenieRules.Compatibility.version = "V4.0-14";

2.1.1 Open workspace

Purpose: Open up existing project *This command is not scripted.*

User Manual Vol. VI

11 April 2011

<u>F</u> ile	<u>E</u> dit	⊻iew	Insert	<u>T</u> ools	<u>H</u> elp	
D	<u>N</u> ew W	orkspa	ce			Ctrl+N
Ê,	Open V	Vorkspa	ace			Ctrl+O
H	<u>S</u> ave W	/orkspa	ice			Ctrl+S
!	⊆lose V	Vorkspa	асе			
:	Set Def	fault W	orkspace	Folder.		
I	Explore	e Currei	nt Works	pace		
:	Set Ext	ernal A	pplicatio	ns		
	Save <u>G</u>	raphics	As			
ļ	Print Graphics					
	Old Save <u>r</u> eport					
	Save <u>r</u> eport					
	Import					•
ļ	<u>E</u> xport					•
	Read Command <u>Fi</u> le					
I	Recent	Comm	and Files			•
	Recent Workspaces					
	E <u>×</u> it					

New Workspace	K
Workspace name:	
l .	
Location:	
C:\DNV\Workspaces\GeniE	
🗌 Set Database Units 🛛 🔽 Enable tolerant modelling	
Length 🗾	
Force N	
Temperature delC	
Command Input File	
C:\DNV\Workspaces\GeniE	
OK Cancel	

Open Workspace

Look in: 🗀 GeniE	▼ ← 🗈
C 26meshcontrol 45172A 45172B aaaa argo_Hold_Model_complete Aries	Chedid Chedid2 COPYFRAME Description Commands dnv_test_1 fggf
<	
File name: GeniE	Open
Files of type: GeniE workspace	ce (*.gni) 💌 Cancel

11 April 2011

2.1.2 Save workspace

Purpose: Save model to database *This command is not scripted.*

2.1.3 Close workspace

Purpose: Close a workspace

This command is not scripted.

2.1.4 Set Default workspace folder

Purpose: Choose were you will save your projects

This command is not scripted. The working directory is stored in the Registry for the current GeniE version (in the user profile for the current user)



2.1.5 Explore Current Workspace

Purpose: Opens a new file explorer window showing the content of the folder where your current workspace is located.

This command is not scripted.



Save graphics as 2.1.6

Purpose: Save a graphics file of the current display

This command is not scripted. To include picture generation in a journal file, see the last Chapter of this User Manual. Typically:

Graphics.saveImage(name,width,height);

🚞 45172A 🚞 chedid2 🚞 45172B COPYFRAME 🚞 aaaa Description Commands argo_Hold_Model_complete Conv_test_1 🚞 Aries 🚞 fggf < File name: models Save as type: GIF (*.gif)

Save Graphics As

Save in: 🗀 GeniE

🚞 26meshcontrol

16

2.1.7 **Print graphics**

Purpose: Print a graphics file directly on printer

This command is not scripted.

Print		? 🛛
Printer Name: Status: Type: Where:	Microsoft XPS Document Writer Ready Microsoft XPS Document Writer XPSPort:	▼ Properties
Comment:		F Print to file
Print range		Copies
C Pages C Select	from: 1 to: 1	1 2 3
		OK Cancel

2.1.8 Save report

Purpose: Save a print of your model to various formats for viewing in a notepad editor, an internet explorer, MS Excel or MS Word. The formats are text, HTML and XML.

This command is scripted and the scripting is dependent on your choice for which parts to include in the report. Typically parts of it may be:

jacket_cc1 = Report("jacket_cc");

jacket_cc1.add(ChapterLoads());

jacket_cc1.element(1).analysis = PSI;

jacket_cc1.element(1).add(TableLoadCase());

jacket_cc1.element(1).add(TableLoadSummary());



💌 듣 🗈 🖝 📼

🚞 Chedid

•

11 April 2011

? ×

>

Save Cancel

2.1.8.1 Structure

Purpose: To document the structural parts as shown to the right. The tick off marks indicates program default.

This command is scripted and the scripting is dependent on your choice for which parts to include in the report. Typically parts of it may be:

a1 = *Report("a")*;

a1.add(ChapterStructure());

a1.element(1).add(TableBeamCoordinate());

a1.element(1).add(TableBeamProperty());

a1.element(1).add(TableBeamHydroProperty());

2.1.8.2 Properties

Purpose: To document the properties as shown to the right. The tick off marks indicates program default.

This command is scripted. Typical parts of it when saving it as a Word document:

a1 = *Report("a")*;

a1.add(ChapterProperties());

a1.element(1).add(TablePropertySection());

a1.element(1).add(TablePropertyHydro());

a1.element(1).add(TableSectionProperty());

a1.saveAs("a1.doc", mrWordXML);

2.1.8.3 Masses

Purpose: To document the mass elements as shown to the right. The tick off marks indicates program default.

This command is scripted. Typical parts of it: a1 = Report("a"); a1.add(ChapterMasses()); a1.element(1).add(TableTotalMassAndCOG()); a1.element(1).add(TableSetMassAndCOG()); a1.element(1).add(TableSetContents());a1.element(1).add(TableSetBoundingBox());







2.1.8.4 Loads

Purpose: To document the load parts as shown to the right. The tick off marks indicates program default.

This command is scripted. Typically when documenting the "Load Summary", the "FEM Node Reaction" and the "FEM Node Displacement" for Analysis1:

a1 = Report("a");

a1.add(ChapterLoads());

a1.element(1).analysis = Analysis1;

a1.element(1).add(TableLoadSummary());

a1.saveAs("a1.doc", mrWordXML);

2.1.8.5 FEM Results

Purpose: To document the FEM Results as shown to the right. The tick off marks indicates program default.

Notice that the reaction forces will be printed for all finite nodes with a boundary condition with reference to the finite element number. In case you specify a joint for a boundary condition, the joint name is also used as a reference. Finite element node displacements are listed for specified joints only

```
This command is scripted. Typically when documenting the "Beam Force
Envelope", the "FEM Beam Stress Envelope", the
"FEMNodeReaction", the "FEM Node Displacement",
the "FEM Beam Force" and the "FEM Beam Stress" for
Analysis1:
demo_report = Report("demo_report");
demo_report.add(ChapterFEMResults());
demo_report.element(1).analysis = Analysis1;
demo_report.element(1).setNoLimit();
demo_report.element(1).add(TableFEMBeamForceEnvelope());
demo report.element(1).add(TableFEMBeamStressEnvelope());
demo_report.element(1).add(TableFEMNodeReaction());
demo_report.element(1).add(TableFEMNodeDisplacement());
demo_report.element(1).add(TableFEMBeamForce());
demo_report.element(1).add(TableFEMBeamStress());
demo_report.saveAs("demo_report.xml", mrWordXML);
```

Report				
🔄 🗢 Loads				
🔤 📰 🛛 Load Case				
🔤 📰 🛛 Load Summary				

	_	
Ø		Load Summary
		Load Summary Conceptual
		Load Summary Applied
		Load Summary FEM Load
		Load Summary FEM Reaction
		Load Summary FEM Difference
Ø		Load Sum
Ø		Load Combination
ً		Load Details - Conceptual
ً		Load Details - Applied
Ø		Point Load
Ø		Line Load
Ø		Surface Load
Ø		Temperature Load
Ø		Support Displacement
Ø		Equipment Load
Ø		Compartment Load



GeniE 18 2.1.8.6

11 April 2011

Explanation of the FEM Results settings

	3	
🚨 Report		
Define Report Name x528k_tr1	Image: Second Structure Selection Image: Second Structure Selection Image: Second Structure Selection Image: Second Structure Selection Image: Structure Selection Image: Second Structure Selections Image: Structure Selection Image: Second Structure Selections Image: Second Structure Selections Image: Second Structure Selections Image: Structure Selections Image: Second Structure Selections Image: Second Structure Selections	el
	Shape (for Beam Deflections only) C Linear C Cubic Cubic Cubic with local loads Number of interpolation points per element: 11	
Save Report		
Report format: Html		
rile name: x528k_tr1	Save View	

Loadcase selection

Analysis:

Select the analysis you want to report. The Beam Force Envelope table and Beam Stress Envelope table "scan" through all the loadcases in the analysis and show the worst results. For the other tables in this chapter, you only get the results individually from each loadcase of the selected analysis.

All loadcases:

Shows the results for all loadcases in the selected analysis.

Only:

Shows the results for the selected loadcase only.

Structure selection

All structure

Shows the results for all the structure in your model. Note that generating the report can be quite time consuming for a large model if you select all structure.

Only

Shows the results for the selected named set only.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	20	11 April 2011

Positions (for Beam Force, Stress and Deflections)

These settings are only influencing the Beam Force and Beam Stress table.

Worst position

Shows the results for the position on the beams having the largest moments for the Beam Force table, or the highest stresses as specified under Hotspot for the Beam Stress table.

All position

Shows the results in both the specified fixed positions and in the worst positions.

Fixed only

Shows the results only in the specified fixed positions.

Number of fixed positions per beam

Here you specify the number of positions along a beam that you want to check. The minimum is 2, meaning only the two end points, the maximum is 100. Default is 5. Note that if you are specifying a high number of fixed positions on a large model, it will be time consuming to generate the report.

Hotspot

Worst hotspot, Abs Max

Shows the absolute max value of the stress component given in the combo box.

Worst hotspot, Max/Min

Shows the maximum and minimum values of the stress component given in the combo box.

All hotspots

Shows the stresses in all hotspots.

Shape (for Beam Deflections only)

Linear

Implies that each finite element is approximated by a straight line, meaning that the two end points of each element are calculated.

Cubic

Implies that each finite element is approximated by a cubic spline function that matches the beam end rotations. Specified number of interpolation points per element is calculated along the spline.

Cubic with local loads

Implies that an analytical solution of deflections due to local element loads is superimposed on the cubic spline function. Specified number of interpolation points per element is calculated along the spline.

Number of interpolation points per element

Number of interpolation points must be between 5 and 25 and is only relevant for the cubic options.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	21	11 April 2011

2.1.8.7 Frame code check

Purpose: To document the results from a frame code check as shown to the right. The tick off marks indicates program default.

This command is scripted. Typically when documenting the "Summary Results", "Member Options Full" and "Member Result Brief" for all utilisation factors above 0.75 for all available code check runs:

a1 = Report("a");

a1.add(FrameCodeCheckChapter());

a1.element(1).run = Cc1.allRuns;

a1.element(1).worstLoadCase = true;

a1.element(1).limit = LimitLower("UfTot", 0.75);

a1.element(1).add(TableSummaryResult());

a1.element(1).add(TableMemberOptionsFull());

a1.element(1).add(TableMemberResultBrief());

a1.saveAs("a1.doc", mrWordXML);



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	22	11 April 2011
In this case the filter options are set from as shown to the right:	the dialogue	Capacity Manager Cc1

2.1.8.8 Plate code check

Purpose: To document the results from a plate code check as shown to the right. The tick off marks indicates program default.

➡ I III
 ➡ I III
 ➡ I III
 ➡ Panel Options Full
 ➡ Panel Result Brief
 ➡ Panel Result Full

This command is scripted. Typically when saving as an Excel document using the filter options as shown below:

a1 = Report("a");

a1.add(PlateCodeCheckChapter());

a1.element(1).run = Cc1.allRuns;

a1.element(1).worstPosition = true;

a1.element(1).worstLoadCase = true;

a1.element(1).add(TablePanelResultBrief());

a1.element(1).element(2).setPrimarySortColumn("UfTot", false);

a1.element(1).element(2).setInheritParentLimits(true);

a1.element(1).add(TablePanelResultFull());

a1.saveAs("a1.xls", mrExcelXML);

Capacity Manager	Cc1	•
 All runs Only 		-
 All loadcases Only 		-
 All Panels Selected Panel 	əls	
Panel Results	ase 🔽 Worst Check	
UfTot		
Upper		

2.1.9 Import

To import data from the following sources:

 XML Concept Model...

 FEM file...

 SACS file...

 STRUCAD3D file...

 ACIS SAT file...

 Intergraph PDS (SDNF file)...

 CadCentre PDMS (SDNF file)...

 Section library...

 Rule Loads XML file...

 External Results SIN file...

 GENSOD file...

2.1.9.1 XML Concept Model

Purpose: Import a structural concept model.

This command is scripted and a typical example is given below.

XmlImporter = ImportConceptXml();

XmlImporter.importLoads = true;

XmlImporter.DoImport("C:/Location/yourname.xml");

r Import Concepts from XM	L File	? 🛛
Look in: 🗀 GeniE) 💣 🎟 -
Cémeshcontrol 45172A 45172B aaaa argo_Hold_Model_complete Aries Chedid Chedid	COPYFRAME Description Commands drug_test_1 fggf ggg Jacket Mesh mesh	Module segft semi semi2 T104 test Tut1 Tut1
<		>
File name:	• •	Open Cancel
✓ Ignore XML Validation (liberal Transformation: Name prefix: ✓ Import loads	import)	

2.1.9.2 FEM file

Purpose: Import geometry and loads from FEM file and create concept model.

This command is scripted and a typical example is given below.

FemImporter = ImportMeshFem();

FemImporter.DoImport("C:/Location/T1.F EM");

Import FEM File	e						? 🗙
Look jn:	C fem_import			•	+ 🗈 💣	•	
My Recent Documents Desktop	Analysis1	T21.FEM					
My Documents							
My Computer: OSLLP7622							
My Network Places	File <u>n</u> ame:	my_fem_mo	delT21.FEM		•] [<u>O</u> pen
	Files of type:	FEM Files (*	.fem)		•		Cancel
✓ Properties	1:1 🗌 Element	::Plate 1:1	Import n	nesh into a	analysis		
Name Prefix:			Analysis:	Analy	sis2		83
Transformation:		-					

11 April 2011

GeniE 23

2.1.9.3 SACS file

Purpose: Import geometry and loads from a Sacs input file and create concept model. The items that are converted are documented on the Reference Document section on GeniE's help pages.

This command is scripted and a typical example is given below.

SacsImporter = ImportMeshSacs(); SacsImporter.setOutputUnits("m", "N");

SacsImporter.DoImport("C:/Location/yourname.inp");

2.1.9.4 STRUCAD 3D file

Purpose: Import geometry and loads from a Strucad input file and create concept model. The items that are converted are documented on the Reference Document section on GeniE's help pages.

This command is scripted and a typical example is given below.

Strucad3dImporter = ImportMeshStrucad3d();

Strucad3dImporter.setOutputUnits("m","N");

Strucad3dImporter.DoImport("C:/Location/ yourname.s3d");

2.1.9.5 ACIS SAT file

Purpose: Import geometry and create concept model

This command is scripted and a typical example is given below.

SatImporter = ImportGeometrySat(); SatImporter.DoImport("C:/Location/yourname.sat");

Look in: 🗀 GeniE 💽 🗕 🖻 🖝 💽 -🚞 26meshcontrol 🚞 Chedid 🚞 gggg 🚞 45172A 🚞 chedid2 🛅 Jacket 🚞 45172B COPYFRAME 🚞 Mesh 🚞 aaaa Commands Description Commands 🚞 model_1 🛅 argo_Hold Model complete anv_test_1 C Module 🚞 seaft 🗎 Aries 🛅 faaf < File name: Open Files of type: SACS Files (*.inp) • Cancel Lenath Unit: m • Force Unit: N -Element:Plate 1:1 Properties 1:1 Name Prefix: Transform...

Import SACS File

Import Strucad3d File	? 🛛
Look in: 🔁 GeniE 💌	+ 🗈 📸 🎟 -
26meshcontrol Chedid 45172A Chedid2 45172B COPYFRAME aaaa Description Comman argo_Hold_Model_complete drv_test_1 Aries Fggf	ds model_1 ds model_1 ds segft
File name: Files of type: Strucad3d Files (*.s3d) Length Unit: m Force Unit: N	Open Cancel
Properties 1:1 Element:Plate Name Prefix: Transform	1:1

Import ACIS	SAT file			? 🔀
Look in: 隘	GeniE		• ÷ €) 💣 🎟 -
Comparison of the second secon	ntrol _Model_complete ME n Commands 1	gggg Jacket Mesh Model_1 Module segft semi semi2 tota tota tota tota tota tota tota tot		Tut8_plate
<				>
File name:				Open
Files of type:	ACIS SAT files ('	'.sat)	-	Cancel
	🔲 Open as read	l-only		

11 April 2011

? 🗙

2.1.9.6 Intergraph PDS (SDNF file)

Purpose: Import geometry and create concept model

This command is not scripted.

Import from	Intergraph PI	OS (SDNF File)	ļ	? 🛛
Look in: 🗀	GeniE		- + 1	💣 🎟 •
Construction Construction Construction Construction Chedid Chedid Chedid Chedid Chedid CopyFRAM CopyFRAM CopyFRAM CopyFram	trol Model_complete IE Commands	gggg Jacket Mesh Model_1 Module segft semi semi semi tild4		Tut8_plate
<				>
File name:				Open
Files of type:	SDNF Files (*.sd	n)	•	Cancel
	🔲 Open as read	only		

2.1.9.7 CadCentre PDMS (SDNF file)

Purpose: Import geometry and create concept model

This command is not scripted.

Import from Cad	Centre PDMS (SDNF File	e) 🔹 🥐	×
Look in: 🗀 GeniE		▼ ← 🗈 💣 III-	
26meshcontrol 45172A 45172B asaa Arojes Arojes Chedid COPYFRAME Description Comm drv_test_1 fggf	anands	🔁 Tut8_plate	
<			
File name: Files of type: SDN	IF Files (*.sdn)	 ↓ Cancel]
, _ 0	pen as read-only		

2.1.9.8 Section Library

Purpose: Import AISC, Euronorm/Norwegian Standard Libraries or common bulb, angle or tee cross sections.

This command is scripted. The scripting depends on how many profiles that are selected. A typical example is given below where one pipe section OD101_6X10 has been selected.

OD101_6X10 = *PipeSection(101.6mm,10mm)*;

OD101_6X10.description = "NVS lib : 101,6x10 NS-EN 10210/10219";

reate/run cruss	section					
Pipe Bar	Box	Sec	tion	L Sect	ion	Channel
Unsymmetrical I	General Sec	tion	Cor	ne	Sect	ion Library
Library:	-	×	Brows	:e		
Subset:	-	1				
Filter						
	🗌 Channel 🔲 Pipe	8				
Section type	Box II					
		_				
Name match						
🗖 Regu	lar expression 8?					
Max height/diam	[m	1				
Min height/diam	[m	1				
	,					
Mumber of continue						
Number of sections	to be imported:					
		_ Im	port			
				\odot All	⊖ Sele	ected

Purpose: To import loads and boundary conditions as defined by Nauticus Hull using the CSR rules for bulk ships.

This command is scripted and a typical example is given below.

RuleLoadsImporter = ImportRuleLoadsXml(); RuleLoadsImporter.DoImport("C:/Location/ yourname.xml");

Import from	Rule Loads XML File 🛛 🛛 💽
Look in: 🔎	Description Commands 💽 👉 🗈 📸 🎫
File name:	Description Commands_RuleLoads Open
Files of type:	Rule Loads XML Files (*.xml)
Outer Shell:	▼ X 8?
Deck:	▼ X 8°5
Slicer:	▼ × 8?
Import XML	file as is-commands 8?

2.1.9.10 External Results SIN file

Purpose: To import a results file on SIN format. There are strict limitations on this functionality. See User Manual Volume 4 for documentation.

This command is scripted and a typical example is given below.

SinImporter = ImportResultsSin(); SinImporter.DoImport("C:/Location/T1.SIN");

2.1.9.11 GENSOD file

The stand-alone application GENSOD generates the soil data needed by SPLICE, storing them in GENSOD.inp.

Purpose : To import a GENSOD.inp file.

GeniE restores the information from the GENSOD.inp file by reproducing the jscommands, thus supplying the folders "Environment/Location" and "Environment/Soil" of the GeniE browser with the relevant data.



🤔 Open					×
Look in:	userdoc_vol6		-	🗢 🗈 💣 🎫	,
Ca.	Name	*		Date modified	Туре
Recent Places		No items	match your s	earch.	
Desktop					
Libraries					
Computer					
Network	•				4
	File name:			•	Open
	Files of type:	GENSOD input files.	(*.inp)	-	Cancel
		Open as read-only	/		

GeniE

Version 5.3

2.1.10 Export

To export data to the following sources:

<u>F</u> EM File
<u>R</u> esults SIN File
XML Concept Model
Intergraph PDS (SDNF file)
CadCentre PDMS (SDNF file)
<u>G</u> enie journal file (JS file)
<u>R</u> ule Loads XML file
ACIS SAT file

2.1.10.1 FEM file

Purpose: To export a FEM file with a user given name.

This command is not scripted unless you tick off for such option. To include FEM export in a journal file, see the last Chapter of this User Manual. Typically:

FemExporter = ExportMeshFem();

FemExporter.DoExport("C:/Location/T1.FEM");

Export to SESAM FEM File			? 🗙
Save in: 🗀 GeniE		- 🗢 🗈	-111 *
26meshcontrol 45172A 45172B aaaa argo_Hold_Model_complete Aries Chedid Chedid2 COPYFRAME Description Commands	Call drv_test_1 Garage Garage Garage Garage Garage Mesh Call drawe Mesh Call drawe Mesh Mes		T104 test Tut1 Tut8 Tut8_plate
<			>
File name: Description Com	mandsT1		Save
Save as type: SESAM FEM Fil	es (*.FEM)	•	Cancel
Journal export operation			

2.1.10.2 Results SIN file

Purpose: To export a SIN results file with a user given name.

This command is not scripted unless you tick off for such option. To include SIN export in a journal file, see the last Chapter of this User Manual. Typically:

SinExporter = ExportResultsSin();

SinExporter.DoExport("C:/Location/R1.SIN");

Export to SE	SAM results SIN File					? 🛛
Save in: 🗀	Mesh	•	¢	£	ď	
T1.SIN						
File name:	Description CommandsR1					Save
Save as type:	SESAM Results SIN Files (*.SIN)	_		•		Cancel
Journal export operation						

27

11 April 2011

Version 5.3

2.1.10.3 XML Concept file

Purpose: Export concept model to XML format. The parts of the concept model that are exported to xml format are documented in User Manual Volume 3.

This command is not scripted unless you tick off for such option. To include XML export in a journal file, see the last Chapter of this User Manual. Notice that export of loads is a separate tick off option. Typically:

XmlExporter = ExportConceptXml();

XmlExporter.exportLoads = true;

XmlExporter.DoExport("C:/Location/yourname.xml");

Export Conc	epts to XML File			
Save in: ն	Description Commands 📃 🔶 🖻 📸			
File name:	Save			
Save as type:	XML Files (*.xml)			
🔲 Journal exp	ort operation			
Export model topology (required to be a valid Genie model)				
Export model topology as a single blob				
Export compressed model topology (Uncheck if exporting to V3.4-xx)				
Expert moder topology separately for each concept				
Export subset				
Export a structure for graphic visualization				
Export loads				

2.1.10.4 Intergraph PDS (SDNF file)

Purpose: Export geometry to PDS format

This command is not scripted.

Export to Intergraph PDS (SDNF File)				
Save in: 🗀 GeniE		- 🗢 🔁	💣 🎟 •	
 26meshcontrol 45172A 45172B aaaa argo_Hold_Model_complete Aries Chedid chedid2 COPYFRAME Description Commands dnv_test_1 fggf gggg Jacket Mesh model_1 Module 	in segft semi semi TI04 tota Tut1 Tut8 Tut8_plate			
File name:			Save	
Save as type: SDNF Files (*.sd	n)	•	Cancel	

11 April 2011

Version 5.3

2.1.10.5 CadCentre PDMS (SDNF file)

Purpose: Export geometry to PDMS format

This command is not scripted.

Export to CadCentre PDMS (SDNF File)				
Save in: 🛅 GeniE		- 🗢 🗈	-111 *	
Comparison of the second seco	Carlos Segft Carlos Semi Carlos Semi Carlos			
File name:			Save	
Save as type: SDNF Files (*.sd	h)	•	Cancel	

2.1.10.6 GeniE journal file (JS)

Purpose: Create and export a clean journal file for frame and plate structures. The parts of the concept model that are exported to js format are documented in User Manual Volume 3.

This command is not scripted unless you tick off for such option. To include js export in a journal file, see the last Chapter of this User Manual. Notice that export of loads is a separate tick off option. Typically:

JsExporter = *ExportModelJS();*

JsExporter.DoExport("C:/Location/yourname.js");

Export Model to JS File
Save in: 🗀 Description Commands 💽 🔶 💼 🖝 🎫
Bescription Commands.js Description Commands_20080723_082843.js
File name: Save
Save as type: JS Files (*,js) Cancel
Journal export operation Try to fix topological inaccuracies in the model (slow)

2.1.10.7 Rule Loads XML file

Purpose: Export compartment and some structure information for use by Nauticus Hull to create CSR rule loads.

This command is not scripted unless you tick off for such option. Typically:

RuleLoadsExporter = ExportRuleLoadsXml();

RuleLoadsExporter.DoExport("C:/Location/ yourname.xml");

2.1.10.8 ACIS SAT file

Purpose: Export geometry to Acis Sat file format.

This command is not scripted.

Export to Rule Loads XML	File	? 🗙
Save in: 🗀 GeniE	1	I 💣 🎟 -
Cémeshcontrol 45172A 45172B aaaa Gargo_Hold_Model_complete Aries Chedid	chedid2 COPYFRAME Description Commands dnv_test_1 Ggg Ggg Gagg Jacket	Mesh model_1 Segft segft semi semi2 T104
File name: Description Com Save as type: Rule Loads XML	mands_RuleLoads Files (*.xml)	Save Cancel

Export to AC	IS SAT File		? 🗙
Save in: ն	Description Commands	← 🗈	-111 *
Description) Commands.sat		
File name:	Description CommandsT1		Save
Save as type:	ACIS SAT File (*.SAT)	-	Cancel
🔲 Journal exp	ort operation		

2.1.11 Read command File

Purpose: Read in and execute a journal file (.js file).

This command is not scripted directly, but the content of the .js file is scripted. You may edit the journal file to include the operation. Typically:

ReadCommandFile("C:/Location/yourname.js");

Open Command File **?**× 💽 🗕 🖻 🖝 🖛 Look in: 🗁 Description Commands Bescription Commands.js Bescription Commands_20080723_082843.js GeniE File name Open Cancel Files of type: GeniE Command Files (*.js) -Read command file made using other Genie version: Compatibility: Autodetect 8? -

2.1.12 Recent command file

Purpose: Read in and execute one of the 10 last used journal files

This command is not scripted directly, but the content of the .js file is scripted.

2.1.13 Recent workspaces

Purpose: Read in and open one of the 10 last used workspaces

This command is not scripted.

Recent Workspaces
1 Description Commands.gni
2 C:\Dnv\WorkSpaces\GeniE\Tut8_plate\Tut8_plate.gni
3 C:\Dnv\WorkSpaces\GeniE\fggf\fggf.gni
4 C:\Dnv\WorkSpaces\GeniE\Tut8\Tut8.gni
5 C:\Dnv\WorkSpaces\GeniE\segft\segft.gni
6 C:\Dnv\WorkSpaces\GeniE\Tut1\Tut1.gni

1 C:\Program Files\...\tubular_joint_tut\Tubular_joint_plate_in.js

 $\label{eq:constraint} 2 \ C:\Program Files\...\tubular_joint_tubular_joint_frame_in.js$ 3 C:\Program Files\...\Examples\FullTutorial1\Genie_tutorial_v_1.js

5 C:\Program Files\...\GeniE_PSIanalysis_ccAPIWSD_in.js

4 C:\Program Files\DNVS\GeniE\Examples\FullTutorial1\SectAndMat_v_1.js

7 C:\Dnv\WorkSpaces\GeniE\aaaa\aaaa.gni

Genie	X
♪	Save changes to C:\Dnv\WorkSpaces\GeniE\Description Commands\Description Commands.gni?
	Yes No Cancel

2.1.14 Exit

Purpose: Exit program and save workspace

This command is not scripted.

User Manual Vol. VI

11 April 2011

GeniE 31

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	32	11 April 2011

2.2 The Edit pulldown menu

The Edit pulldown menu is where you can copy, delete, undo or redo, and change the rules and properties for your working material like i.e. materials, thicknesses, meshing and joints.

2.2.1 Undo/Redo

Purpose: You can go back and redo recent work you are not satisfied with or things you have done wrong. The history log is deleted when you save the work.

This command is not scripted. This command will modify the script. When doing undo, you will remove recently added lines from the journal. When doing redo, you will add these lines back to the journal.

Ent View Insert Tools	Help						
い Undo Create Bm1	Ctrl+Z						
🖼 Can't <u>R</u> edo							
🛜 <u>U</u> ndo/Redo Dialog							
r⁄⊡ Set Undo <u>M</u> ark1	Ctrl+U						
🕼 <u>С</u> ору	Ctrl+T						
× Delete	Del						
<u>P</u> roperties							
<u>R</u> ules	+						
Licenses / features							

2.2.2 Undo/Redo Dialog

Purpose: Specify the step forward or backward from a list. You can go back or forward to undo or redo recent work you are not satisfied with or things you have done wrong. The history log is deleted when you save the work.

This command is not scripted.

U	Undo/Redo								
	✓ Enable interactive undo marks History list: ☐ Show only interactive marks ☐ Show only user defined marks								
		Mark Name	Userdefined	Undo <-	Redo ->	Current			
	1	Create Bm1	Interactive	Undo <-	Redo ->	<			
				0	K (Cancel Apply			

2.2.3 Set Undo Mark

Purpose: To include a user defined undo mark in the undo/redo dialogue.

This command is not scripted. You may include undo marks in the journal file so that it is easy for you to find locations for undo or redo operations. Typically:

setUndoMarker("yourname");

Undo/Redo								
Enable interactive undo marks History list Show only interactive marks Show only user defined marks								
	Mark Name	Userdefined	Undo <-	Redo ->	Current			
1	Command-line entry	User	Undo <-	Redo ->				
2	Command-line entry	User	Undo <-	Redo ->				
3	Command-line entry	User	Undo <-	Redo ->				
4	Start_Copying	User	Undo <-	Redo ->				
5	Copy 2 objects	Interactive	Undo <-	Redo ->				
6	Copy 2 objects	Interactive	Undo <-	Redo ->				
7	Command-line entry	User	Undo <-	Redo ->				
8	Command-line entry	User	Undo <-	Redo ->				
9	Command-line entry	User	Undo <-	Redo ->				
10	Start_Load_Application	User	Undo <-	Redo ->	<=======			
				OK	Cancel Apply			
11 April 2011

Version 5.3

2.2.4 Copy

Purpose: Copy a selected object using translate, rotate, mirror, 3 point position or a general transformation.

This command is scripted. The scripting depends on which operation that is carried out. Typically for a copy translate operation of beam Bm1 (10 meters in global zdirection):

Bm2 = Bm1.copyTranslate(Vector3d(0 m,0 m,10 m));

💽 Сору	X
Translate Rotate Mirror Scale	3 Point Position General transformation
Translation Vector	
Vector3d(0 m,0 m,10 m)	I
Copy 1 time(s)	
Preview	Cancel Apply

2.2.5 Delete

Purpose: Mark a selected object and delete it.

This command is scripted. The scripting depends on which object is deleted. Typically for a delete operation of plate Pl1:

Delete(Pl1);

2.2.6 Properties

Purpose: Edit and modify a property library

The commands for each of the available properties are shown in the following. They are all accessed from the following menu:



More details may be found on GeniE's help pages under Reference Documents -> Beam Section Parameters.

2.2.6.1 Section

Purpose: To define or edit a section property. The picture to the right shows the dialogue for definition of a pipe section. It is also possible to define section properties of type bar, box, I, L, Channel, Unsymmetrical I (also used to define T and bulb), general section and cones.

This command is scripted. The scripting depends on which property is created. Typically for a pipe section MyPipe where shear factors and fabrication method differ from default values:

MyPipe = PipeSection(1000mm, 25mm); MyPipe.shearFactorY = 0.8; MyPipe.shearFactorZ = 0.9; MyPipe.fabricationMethod = fmRolled;

It is also possible to import section profiles from section libraries. The available section libraries are accessed when you push the *Browse* button.

This command is scripted. The scripting depends on which section property that is imported. Typically for a HE200A section from the NSF_EN library:

HE200A = ISection(190mm,200mm,6.5mm,10mm); HE200A.description = "NVS lib : HE 200 A NS-EN 10034";



GeniE

34

Create/Edit Cross Section
Unsymmetrical I General Section Cone Section Library Pipe Bar Box I Section L Section Channel
New section Thickness Thickness

Create/Edit Cross Section	X
Pipe Bar Box Unsymmetrical I General Section Library: ▼ Subset: ▼ Filter ▼ Filter Box ✓ Box Image: Section type Box Bar L Name match Regular expression \$? Max height/diam [m] Min height/diam [m]	I Section L Section Channel n Cone Section Library X Browse
	Import C All C Selected

Open Section Library on : c:\program fi	iles\dnvs\genie\Li ? 🔀
Look in: 🗀 Libraries	· ← € 🗳 💷 ·
aisc_v3.kzy anglebar.xml dulb.xml flatbar.xml Material_library.xml New Text Document.xml	
File name: AISC_V3.KZY	<u>O</u> pen
Files of type: Section library (*.KZY, *.XML)	Cancel



11 April 2011

DET NORSKE VERITAS SOFTWARE

Version 5.3

11 April 2011

2.2.6.2 Material

Purpose: To create and edit a material property

This command is scripted. Typically for a linear isotropic material named MyMat:

MyMat = *MaterialLinear*(275000, 7.85, 210000000, 0.3, 1.2e-5, 3.0e-5, 430000);

It is also possible to define an isotropic shear material.

This command is scripted. Typically for an isotropic shear material named MyShearMat with axial reduction of 75:

MyShearMat = *MaterialShear*(75, 7.85, 210000000, 0.3, 1.2e-5, 3.0e-5);

	▼ X	Allow edit	
New material	Yield		[Pa] 🗙
σ_{\uparrow}	Density		 [Kg/m^3] ,
yield	Young		[Pa] 🗙
	Poisson		×
ν → _E	Thermal	, 	[delC^-1]
	Damping	, 	 [N*s/m] 🗙
	🔲 Tensile		[Pa]
	Damping		[N*s/m] 7 [Pa]

👪 Create/Edit Material Properties 🛛 👔			
Linear Isotropic Material Isotropic Shear Material			
	- × 🗸	Allow edit	
New material	A vial and unline On		
τ=shear	Axial-reduction 9?		*
\uparrow /	Density		[Kg/m^3] 🗙
	Young		[Pa] 🗙
0-axial	Poisson		×
E E	Thermal		[delC^-1] 🗙
	Damping		[N*s/m] 🛪
		,	
J		IK Cancel	

Notice that it is also possible to import a material library from *C:\ProgramFiles\DNVS\GeniE\Libraries\ Material_library.xml*.

The path name assumes you have installed the program using the default installation set-up. The library is imported by using the command *File/Import/XML Concept Model*.

The library contains 71 linear isotropic materials; these are documented in User Manual Volume 3 Appendix B.

DET NORSKE VERITAS SOFTWARE

11 April 2011

Version 5.3

2.2.6.3 Thickness

Purpose: To create and edit a thickness property

This command is scripted. Typically for a thickness 20mm:

My_thickness = *Thickness*(20mm);

Create/Edit Thickness			×
Thickness			
	•	Allow edit	
New thickness			
	[hickness		[m]
Thickness			
1			
		OK Cancel	Apply

2.2.6.4 Effective flange

Purpose: To create and edit an effective flange property

This command is scripted. Typically for an effective flange of 0,8 m:

my_eff_flange = EffectiveFlange(0.8);

Create/Edit Effective Flange	Width	0.8	[m]
	ОК	Cancel	Apply

2.2.6.5 Corrosion addition

Purpose: To create and edit a corrosion addition used when creating a finite element model using the CSR rules for bulk ships.

This command is scripted. Typically for a corrosion addition of 8 mm:

MyCorrosion = *CorrosionAddition*(8*mm*);

Create/Edit corrosion addition	X
Corrosion Addition	
Allow edit	
Additional thickness: [m] 🛪	
OK Cancel App	y

11 April 2011

Version 5.3

2.2.6.6 Plate type

Purpose: To create a non-structural or membrane plate type. Notice that the non-structural plate is used solely to close a volume like a compartment and that the non-structural plate does not carry any loads or contributes to the structural stiffness. The membrane plate will ensure that a plate will be represented as a membrane finite element when creating a FE model.

Create/Edit plate type	×
Non Structural Membrane	
×	
OK Cancel Apply	

This command is scripted. Typically:

MyNonStructPlate = PlateTypeNonstruct(); MyMembranePlate = PlateTypeMembrane();

2.2.6.7 Mesh Property

Purpose: To create a mesh density that can be applied to a plate, a beam or a feature edge.

This command is scripted. Typically:

MyMeshDensity = *MeshDensity*(2.5*m*);

It is also possible to define number of elements that can be applied to a beam or a feature edge.

This command is scripted. Typically:

MyMeshNoElements = *NumberOfElements*(8);

Create/Edit Mesh property 🛛 🔀
Mesh Density Number of elements
Allow edit
New meshDensity
Element Length [m] 🛪
Element Length
OK Cancel Apply
Create/Edit Mesh property
Mesh Density Number of elements
Allow edit
Number of Elements
OK Cancel Apply

2.2.6.8 Load interface

Purpose: To create a property to decide which beams to receive loads from equipments. *This command is scripted. Typically:*

MyLoadInterface = *LoadInterface();*

Property Sheet	
Load Interface	
Static	L Allow edit
State	
	OK Cancel Apply

2.2.6.9 Hinge

Purpose: To create hinges to specify which degrees of freedom for a beam end that is connected to another beam.

This command is scripted. Typically for a hinge where all the rotational degrees of freedom are not connected:

MyHinge = *Hinge*(1,1,1,0,0,0);

It is also possible to specify a flexible hinge whereby
stiffness for each degree of freedom is specified.

This command is scripted. Typically for a flexible hinge where 4 different spring stiffnesses are used (fixed in x and y):

MyFlexHinge = FlexibleHinge(Fixed, Fixed, Stiffness(500), Stiffness(12500), Stiffness(25000), Stiffness(75000));

Property Sheet		
Hinge Flexible Hinge		
×		
Hinge Fixation (in local coordinate system) Not binged Hinged		
dx		
dy _		
dz _		
хі		
ry _		
rz		
OK	Cancel	Apply

Property Sheet		X
Hinge Flexible Hinge		
	×	1
	^	
Hinge stiffness		
Fixed Free Spring	Spring	
	surrness	[N/m]
× -	0 N/m	[N/m]
7	0 NVm	[N/m]
- 1	Carina	[record
Fixed Free Spring	stiffness	
тх	0 N*m	[N*m]
ry	0 N*m	[N*m]
rz	0 N*m	[N*m]
	OK Car	ncel <u>Apply</u>

User Manual Vol. VI

DET NORSKE VERITAS SOFTWARE GeniE			User Manual Vol. VI
Version 5.3	39		11 April 2011
2.2.6.10 Reinforcemen	t		
Purpose: To create a reinforcement to be can or stub.	applied to a	Create/Edit Reinforcement	
			E Allem a B

This command is scripted. Typically:

MyReinforcement = *Reinforcement*(0.5,0.45*m*);

Reinforcement	Allow edit
New reinforcement	
Can diameter	Feinforcement length length=max(fac*diameter.min); fac min [m]
	OK Cancel Apply

2.2.6.11 Hydro property

Purpose: To define hydro properties to be applied to beams for use in hydrodynamic analysis based on the Morison theory (Wajac). For more details, please see User Manual Volume 2 and the Wajac User Manual.

2.2.6.11.1 Flooding

Purpose: To define a flooding ratio

This command is scripted. Typically:

MyFlooding = *Flooding*(1);

🎎 Create/Edit Hydro Pr	operty		
Flooding Hydrodynamic Diar	meter Conductor Shielding Ele	ment Refinement Morison Ma	arine Growth Air Drag
New C Edit Existing	Flooding1	Allow edit	
Flooding ratio	ଡ଼? Flooding: ସାହାର Wajac only accep	× its values 0 or 1.	
			Cancel Apply

2.2.6.11.2 Hydrodynamic diameter

Purpose: To define a hydrodynamic diameter

This command is scripted. Typically:

MyHydroDiamater = HydroDynamicDiameter(0.75m);

🎎 Create/Edit Hydro Pi	operty			
Flooding Hydrodynamic Dia	neter Conductor Shielding	Element Refinement	Morison Marine Grow	th Air Drag
• New C Edit Existing	HydroDynamicDiameter1	Allow edi	t	
♀? Hydrodynamic diameter:	[m]	×		
,			Cancel	Apply

DET NORSKE VERITAS SOFTWARE

GeniE 40

11 April 2011

Version 5.3

2.2.6.11.3 Conductor Shielding

Purpose: To define a conductor shielding

This command is scripted. Typically:

MyShielding = ConductorShielding(1m,2m);

🎎 Create/Edit Hydro Pr	operty		X
Flooding Hydrodynamic Diar	neter Conductor Shielding Eler	ment Refinement Morisor	n Marine Growth Air Drag
New C Edit Existing	ConductorShielding1	Allow edit	
♥? x distance:	[m] 🗙 y distance:	[m]	×
-			Cancel Apply

2.2.6.11.4 Element refinement

Purpose: To define an element refinement

This command is scripted. Typically:

MyRefinement = *ElementRefinement*(5);

4 Create/Edit Hydro Property	\mathbf{X}
Flooding Hydrodynamic Diameter Conductor Shielding Element Refinement Morison Marine Growth Air Drag	
New C Edit Existing ElementRefinement1 I Allow edit	
Refinement coefficient: 1 0? Detect Importation Maximum: 0 m [m] 9? Minimum: 0 m [m] 9?	
Cancel Apply	

2.2.6.11.5 Morison

Purpose: To define constant Morison coefficients

This command is scripted. Typically:

MyMorison = MorisonCoefficients(0,0.7,0.7,0,2,2);

iå (🚨 Create/Edit Hydro Property 🛛 🔀					
Floo	oding Hydrodynamic Diame	ter Conduc	tor Shielding Element Refinement Morison Marine Growth Air Drag			
Co	nstant Function of Diamete	r Function	of Roughness/Reynoldsnumber Function of Roughness/KC number By Rule			
	• New C Edit Existing	MorisonC	onstant1 🔽 Allow edit			
	×ر ۲	🔗 Cdx	0			
	Z X	😵 Cdy	0.7			
	Т	😵 Cdz	0.7			
	Cd = drag	ଡ? Cmx	0			
	Cm = inertia	😪 Cmy	2			
		⊗ ? Cmz	2			
-						
			Careat / Apple /			
			Cancei Apply			

GeniE 41

11 April 2011

Purpose: To define Morison coefficients as a function of diameter

This command is scripted. Typically (for 2 different diameters):

MyMorisonDiameter = MorisonDiameterFunction(Array(2,3), Array(1,0.9),Array(2,2.1));



This command is scripted.

Purpose: To define Morison coefficients as a function of Roughness and Keulegan Carpenter (KC) number

This command is scripted.

👬 Create/Edit Hydro Droperty			
Elonding Hudrodunamic Diameter Conductor Shi	alding Element Balinament	Morison Marine Growth Ai	r Drag
Constant Function of Diameter Europian of Pour		unction of Bouchness /KC sursh	
	griness/rreynoldshamber r (and torr of Hough iness into hame	ber by ficie
New C Edit Existing MorisonDiameter	1 🔽 Allow	edit	
Morison Coefficient Function Table: 💡		💥 🖿 🖬 🗸	Cdz 🔽 Cmz
Member diam Drag coeffici Inert	a coefficient GeniE	V4.0-14 Date: 21 Aug 2008 09:	51:26
2	_ =		
4			
6	merti		
8			
9 10	Drag		
11		$t_{1}, \dots, t_{n}, \dots, t_{n}, \dots, t_{n}$	0.8 1
	Hydro	odynamic drag <u>Member</u>	Diameter D
		1	(
		Cancel	Apply
🖞 Create/Edit Hydro Property			Þ
Flooding Hydrodynamic Diameter Conductor Shie	Iding Element Refinement	Morison Marine Growth Air	Drag
Constant Function of Diameter Function of Roug	hness/Reynoldsnumber Fu	inction of Roughness/KC numb	er By Rule
New C Edit Existing MorisonReynold	Allow e	edit	
Fill tool	Marison Co	efficient Function Table: 👓	
Roughness Number Table: 🔗	Roughness n Re	encient runction rable. gry	ia coeffici 🔼
Roughness n Repetitions			
	3		
4	5		
6	6 7		
7	8		
Fill table	10		
	12		
🙀 📴 🖬 🔽 Cdz 🔽 Cmz	C	Cleartable Remove	Insert
		Cancel	Apply
🎎 Create/Edit Hydro Property			1
Flooding Hydrodynamic Diameter Conductor Sh	ielding Element Refinement	Morison Marine Growth A	ir Drag
Constant Function of Diameter Function of Ro	ighness/Reynoldsnumber F	unction of Roughness/KC num	ber By Rule
New C Edit Existing MorisonKC1	Allow	<i>i</i> edit	
Fill tool	Marison C	oefficient Eurotion Table: 02	
Roughness Number Table: 🔗	Roughness n H	KC number Drag coeffici Ine	rtia coeffici 🔼
Roughness n Repetitions	1		
2	3		
4	5		
6	7		
8	9		
Fill table	10		
	12		
₩ I Han I Han I IV Ldz IV Lmz		Liear table Remove	Insert
		Cancel	Apply

11 April 2011

Version 5.3

Purpose: To define Morison coefficients according to the API recommendations. See Wajac User Manual for a further definition

This command is scripted. Typically:

MyAPIMorison = *MorisonByRule();*



2.2.6.11.6	Marine
growth	

Purpose: To define a constant value for marine growth

This command is scripted. Typically:

MyMarineGrowth = MarineGrowthConstant(20cm,1cm,2); MyMarineGrowth.use InForceCalculations(false);



Purpose: To define a linearly varying value for marine growth

This command is scripted. Typically:

MyMarineVarGrowth = MarineGrowthZLevelFunction(); MyMarineVarGrowth.add (0m,0.2m,0.03m,2); MyMarineVarGrowth.add (-50m,0.1m,0.02m,1); MyMarineVarGrowth.use InForceCalculations(false);



2.2.6.11.7 Air drag Purpose: To define a constant value for air drag

This command is scripted. Typically:

MyAirDrag = *AirDragConstant*(1,0.65);



Purpose: To define a value for air drag as function of the Reynold number.

This command is scripted.

Revnold num Y-transversal Z-transversal drag coeff 1 1 1 1 1 2 1 1 1 1 1 3 1	👪 Create/Edit Hydro Property	
Constant Function of Reynold number • New • Edit Existing AirDragReynold1 ✓ Allow edit Air Drag Function Table: • ? • Mexicon Table: • ? • Mexicon Table: • ? • Mexicon Table: • ? • Mexicon Table: • ? • Mexicon Table: • P • P • Cdy •	Flooding Hydrodynamic Diameter Conductor Shielding Elemen	nt Refinement Morison Marine Growth Air Drag
Air Dreg Function Table: Reynold num Y-transversal Z-transversal drag coeff 1 - - - 2 - - - 3 - - - 4 - - - 5 - - - 6 - - - 7 - - - 8 - - - 9 - - - 10 - - - 11 - - - 12 - - - Air drag along local z-axis U.S. 0.8 1	Constant Function of Reynold number	I Allow edit
Reynold num Y-transversal Z-transversal drag coeff 1	Air Drag Function Table: 💡	💥 陷 🖬 🔽 🗸 dy 🗸 Cdz
Air drag along local y-axis ^{ie} emoliti number R	Reynold num Y-transversal Z-transversal drag coef 1 2 3 4 3 4 4 5 5 6 5 6 6 7 7 8 9 10 11 11 11 12 12 12 12 14<	GeniE V4.0-14 Date: 21 Aug 2008 09:54.03
		Air drag along local y-axis ^{legmol} p number R

44

11 April 2011

2.2.6.12 Beam Type

Purpose: To define various beam classifications to enable pre-defined rules.

2.2.6.12.1 Nonstructural

Purpose: To classify a beam as non-structural in a Morison analysis. The beam will contribute to the wave load analysis but not in a structural analysis.

This command is scripted. Typically:

MyNonStruct = *BeamTypeNonstruct();*

Create/Edit Beam Type	×
Nonstructural Truss Shim Inner Beam	
Allow edit	
New Nonstruct	
OK Cancel	apply

2.2.6.12.2 Truss

Purpose: To classify a beam as truss member. In this case the bending stiffness of the member is neglected.

This command is scripted. Typically:

MyTruss = *BeamTypeTruss();*

Create/Edit Beam Type Nonstructural Truss Shim Image: Allow edit New truss

2.2.6.12.3 Shim

Purpose: To classify a beam as shim member. The shim elements are normally special connections between conductors and conductor frames whereby the conductor is free to move in vertical direction.

This command is scripted. Typically:

MyShim = BeamTypeShim(1250);

Create/Edit Beam Type		
Nonstructural Truss Shi	m Inner Beam	
	Allow edit	
, New shim	Stiffness: [N/m]	×
	OK Cancel	Apply

DET NORSKE VERITAS SOFTWARE

11 April 2011

Version 5.3

2.2.6.12.4 Inner beam

Purpose: To specify the connections between a beam and an inner beam.

This command is scripted. Typically:

MyFullCoupling = *InnerBeamSolid();*

New InnerBeam Fully coupled Linear dependency Spring	Allow edit	

It is also possible to specify a spring connection between the beams.

This command is scripted. Typically:

MySpringConnection = InnerBeamSpring(250,350,350,2500,0 kN*m,0 kN*m);

Create/Edit Beam Type				×
Nonstructural Truss Shi	m Inner Beam			
	▼ × I Allow	edit		
New InnerBeam	Total spring constant			
 Linear dependency Spring 	y Y-Z	symmetry		
	Axial stiffness, X		[N/m] 🗙	
	Shear stiffness, Y		[N/m] 🗙	
	Shear stiffness, Z		[N/m]	
	Torsional stiffness, X		[N*m] 🛪	
	Rotational stiffness, Y	0 N*m	[N*m]	
	Rotational stiffness, Z	0 N*m	[N*m]	
		OK Cancel	Apply	

2.2.6.13 Pile Characteristics

Purpose: To define specific pile properties to be used in a non-linear pile – soil analysis.

This command is scripted. Typically:

MyPileChar = *PileCharacteristics*(2.5, *tcFree*);

🎎 Create/Edit Pile Characte	ristics			
Pile Characteristics				
• New C Edit existing PileT	ype1	Allow edit		
Density of soil/fluid inside pile		[Kg/m^3] :	×	
Pile boundary condition Pile tip is free Pile tip is free Pile tip is free The pile is assumed to be infinitely long beneath the tip Calculate axial stiffness based on relevant q-z data The pile is assumed to be infinitely long beneath Z = 0 m [m]				
Avial stiffness replacements (ignored	0 N	[N]		
Bending stiffness replacement	0 N*m^2	[N*m^2]		
Shear stiffness replacement	0 N	[N]		
Torsional stiffness replacement	0 N*m^2/rad	[N*m^2/rad]		
		Cancel	Apply	

11 April 2011

Version 5.3

2.2.6.14 Wet Surface

Purpose: To define a wet surface to be used by a panel model (outside wetted surface and compartment definitions) and for load application to plates and shells.

This command is scripted. Typically:

MyWetSurface = *WetSurface();*

2.2.6.15 Content

Purpose: To define a liquid or solid content for use when filling a compartment.

This command is scripted. Typically:

MyLiquidContent =
LiquidContent(ctFuelOil, 900 tonne/m^3);

It is also possible to define a solid content.

This command is scripted. Typically:

MySolidContent = SolidContent(ctBulk, 2500, ContentCSRBulk(0.5, 22.5 deg));

👪 Create/Edit Wet Surface Properties	
Wet Surface	
New C Edit existing WS1	
Represents the wet part of the hull, subject to external hydro pressure	
OK Cancel	Apply



👪 Create/Edi	t Content 🛛 🛛 🛛 🛛
Liquid Content	Solid Content
⊙ New C	Edit existing Content1
Solid Type Density	Bulk [Kg/m^3] ×
Shape fund	Stion Compartment is full Width fraction, b/B 0.5 Angle of repose 22.5 deg
<u> </u>	Cancel Apply

DET NORSKE VERITAS SOFTWARE

GeniE 47

Version 5.3

2.2.6.16 Mesh Option

Purpose: To define a property that over-rules the default mesh settings.

This command is scripted. Typically for an edge:

MyMeshEdge = MeshOptionEdge(); MyMeshEdge.eliminateInternalVertices = true; MyMeshEdge.meshStrategy = LinearDistributionEdge;

Create/Edit mesh option	×
Mesh Options for Edge Mesh Options for Face	
Allow edit	
Remove internal vertices Override mesh strategy Uniform distribution Linear distribution	
OK Cancel Apply	

Create/Edit mesh option	×
Mesh Options for Edge Mesh Options for Face	
Allow edit	
✓ Force quad elements Ø? ✓ Prefer regular mesh (m x n) Ø? ✓ Remove internal vertices Ø? ✓ Remove internal edges Ø?	
Override mesh strategy Advancing front quad mesher Advancing front triangle mesher Sesam quad mesher	
OK Cancel Apply	

It is also possible to specify mesh options for a face.

This command is scripted. Typically for a face:

MyMeshFace = MeshOptionFace(); MyMeshFace.forceQuadElements = true; MyMeshFace.eliminateInternalEdges = true; MyMeshFace.eliminateInternalVertices = true; MyMeshFace.meshStrategy = AdvancingFrontQuadMesher;

2.2.6.17 Permeable

Purpose: To define a plate or shell as permeable even though the plate or shell is continuous (i.e. no holes). This is typically used when defining compartments. Loads are applied to the plate or shell.

This command is scripted. Typically:

MyPermeable = *Permeable(true);*

Create/Edit permeability	×
Permeable	
💌 🛪 🕅 Alla	w edit
✓ Permeable	
OK Cancel	Apply

2.2.6.18 Structure type

Purpose: To define or edit a structure type. Typically the structure type is applied to a part of the structure. When running a codecheck it is sometimes necessary to apply different structure types to different parts of the structure.

This command is scripted. Typically: My_inner_bottom = StructureType(stInnerBottom);

👪 Create/Edit Stru	icture Type	×
Structure Type		
My_inner_bottom	-	
Static	_	
Structure Type:	Inner Bottom Forecastle Deck Hatch Cover Shetered Deck Hatch Corer Weather Deck Hatch Side Coaming Hatch Side Coaming Hoper Tank Top Inner Bottom Inner Bottom Inner Side Longtudinal Bulkhead Longtudinal Girder Weh Frame	Pply

11 April 2011

2.2.7 Rules

Version 5.3

Purpose: Change or edit the rules for creation of joints, automatic design of tubular joints, tolerant modelling, connected move of beams, definition of groups (not sets), input units, meshing, scripting of sets and compatibility checking of log files between various program versions. Each option is described in the following:

2.2.7.1 Joint creation

Purpose: Rules for automatic creation of joints. The joints will be defined and named according to the rules specified as well as the naming convention.

This command is scripted. Typically, when activating the automatic joint generation and the exclusion criteria as shown:

GenieRules.JointCreation.exclude(geFreeBeamEnds,true); GenieRules.JointCreation.exclude(ge2BeamAligned,true); GenieRules.JointCreation.exclude(geThroughBeamPure,true); GenieRules.JointCreation.autoGenerate = true;

Joint Creation Rule	×	
Automatic joint generation	ОК	
Selection-aware classification	Cancel	
Exclusion Criteria Exclude pure through-beam intersections		
Exclude 2-beam aligned intersections		
Exclude free beam ends		
Exclude intersections with beam ends		
Exclude intersections with through-beams		

2.2.7.2 Joint design

Purpose: Rules for calculating cans, stubs, cones and planewise gap in a tubular joint.

Tubular joint rule	
Cans Property: <u>KAutomatics</u> Can length: max(d*canFac,canMin) canFac: 0.25 canMin: 0.3 m [m] Chord alignment tolerance: 5 deg [deg] Gap Minimum gap: 0.0508 m [m] Gap tolerance: 0.001 m [m] Plane tolerance: 1 deg [deg] Brace angle 10 deg [deg] Brace angle 10 deg [deg] Iterations: 2 Flush braces to surface of chord	Stubs OK Property: <automatic> Stub length: max(d*stubFac,stubMin) stubFac: 1 stubHin: 0.6 m Min: 0.6 m Image: Attempth to copy reinforcements when copying joint Cone Image: length L=[r2·r1]/ratio: ratio=tan(angle) angle: 9.462322208 dc [deg] Conical Section: <a>(Automatic>)</automatic>

Joint greation ... Joint design ... Tolerances ...



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI

49

11 April 2011

Command	Script
canFac "0.3" and canMin "0.4m"	GenieRules.JointDesign.setDefaultCanRule(0.3,0.4 m);
Chord alignment tolerance "3 deg"	GenieRules.JointDesign.chordAlignmentTolerance = 3 deg;
stubFac "0.9" and stubMin "0.5m"	GenieRules.JointDesign.setDefaultStubRule(0.9,0.5 m);
Minimum gap "0.075 m"	GenieRules.JointDesign.minimumGap = 0.075 m;
Gap tolerance "0.002 m"	GenieRules.JointDesign.gapTolerance = 0.002 m;
Plane tolerance "2 deg"	GenieRules.JointDesign.planeTolerance = 2 deg;
Brace angle move limit "11 deg"	GenieRules.JointDesign.braceAngleMoveLimit = 11 deg;
Iterations "3"	GenieRules.JointDesign.iterations = 3;
Flush braces to surface of chord	GenieRules.JointDesign.flushBraces = true; or false
Cone ratio 1/"8"	GenieRules.JointDesign.coneAngle = 7.125016349 deg;
Cone angle "10 deg"	GenieRules.JointDesign.coneAngle = 10 deg;

2.2.7.3 Tolerances

Purpose: Set tolerances when working with tolerant modelling.

This command is scripted. Typically, when modifying the point tolerance to 0.05 m and angular tolerance to 3 degrees:

GenieRules.Tolerances.pointTolerance = 0.05; GenieRules.Tolerances.angleTolerance = 3 deg;

2.2.7.4 Connected move

Purpose: Rules for connected move of beams.

This command is scripted. Typically, when setting connected move as default and including all structural points in calculation:

GenieRules.ConnectedMove.defaultConnected = true; GenieRules.ConnectedMove.useStructuralPoints = true;

Tolerances Rule 🛛 🔀		
✓ Use tolerant modelling		
Point Tolerance: 0.01 m	[m]	
Angular Tolerance: 0 deg	[deg]	
ОК	Cancel	



Input Units

Database Units Length

Force

2.2.7.5 Geometry

Purpose: To specify whether GeniE shall make groups when e.g. copying or moving structural members. This can lead to a better performance when reading a command file or during modelling. The command must be used with care.

This command is scripted. Typically, when enabling grouping when reading a command file:

GenieRules.Geometry.creationGrouping = cgGroupingJournal;

2.2.7.6 Units

Purpose: Specify default input units and how labels etc, will appear on the desktop.

This command is scripted. Typically, when changing input unit to mm with fixed format display:

GenieRules.Units.setInputUnit (Length, "mm"); GenieRules.Units.setDisplay Format(Length, 6, mcFixed);

nenii (abi	e j	Der	ived units will not be updated until you press apply	
Unit Name	Unit	Display Format	Display Precision	~
Angle	deg	General 🗾	6	
Force	N	General 💌	6	
Length	m	General 💌	6	
TempDiff	delC	General 💌	6	
Time	5	General 💌	6	
				~
Reset to d	atabas	e units		

Mass

Angle

2.2.7.7 Meshing

Purpose: Set global rules for creation of mesh. These may be override by local settings.

There are five categories of global mesh settings. For guidance in how these are used, please see Chapter 6 of User Manual Volume 3.

This Section lists the commands and the associated script command.

Rules Meshing	
eneral Max/Min Angle Jacobi Eliminate edge Ch	nord Height
General FEM options Use second order elements Superelement type: Model topology (Performed before meshing) ✓ Always simplify topology Split periodic geometry if needed Bement preferences Prefer regular mesh (m x n) ✓ Allow triangular elements ✓ Prefer point mass as node mass Use edilling elements ?? Use eccentric hinges ??	Other preferences Include unused properties ✓ Automatic load combination FEM numbering Round off Mesh Density ✓ Adjust number of elements Ø? ✓ Write loads separate Ø? ✓ Write load Case names Use long LoadCase names Use long Set names Use long Property names Face mesher C Advancing front quad mesher C Seam quad mesher © Seam quad mesher
Scantling idealizations Thickness: msGross Ignore beam eccentricities ? Use co-centric beams ?	Edge mesher Image: Construction Image: Construction Idealisations Image: Construction Image: Construction



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	51	11 April 2011

Command	Script
Use second order elements	GenieRules.Meshing.elementType = mp2ndOrder; or mp1stOrder;
Specify superelement type to "2"	GenieRules.Meshing.superElementType = 2;
Always simplify topology	GenieRules.Meshing.autoSimplifyTopology = true; or false
Split periodic geometry if needed	GenieRules.Meshing.autoSplitPeriodicGeometry = true; or false
Prefer rectangular mesh (m x n)	GenieRules.Meshing.preference(mpPreferRectangularMesh, true); or false
Allow triangular mesh	GenieRules.Meshing.preference(mpAllowTriangularElements, true); or false
Prefer point mass as node mass	GenieRules.Meshing.preference(mpPreferPointMassAsNodeMass, true); or false
Use drilling elements	GenieRules.Meshing.preference(mpUseDrillingElements, true); or false
Use eccentric hinges	
Specify scantling idealization according to the choices in the combo box: msGross msNetCSR_Bulk msNetCSR_Tank msNetDNV_1A1 msNetGeneric	Examples GenieRules.Meshing.scantlings = msGross; GenieRules.Meshing.scantlings = msNetCSR_Bulk;
Ignore eccentricities	GenieRules.Meshing.ignoreEccentricities = true;); or false
Use co-centric beams	GenieRules.Meshing.useCocentricBeams = true;
Include unused properties	GenieRules.Meshing.preference(mpIncludeUnusedProperties, true); or false
Automatic load combination FEM numbering	GenieRules.Meshing.FemLoadcaseNumbering = mmCombinationsAutomatic;); or Manual
Round off mesh density	GenieRules.Meshing.preference(mpMeshDensityRounded, true); or false
Adjust number of elements	GenieRules.Meshing.preference(mpAdjustNumberOfElements, true); or false
Write loads separate	GenieRules.Meshing.preference(mpWriteLoadsSeparate, true); or false
Use long LoadCase names	GenieRules.Meshing.preference(mpUseLongLoadcaseNames, true); or false
Use long Set names	GenieRules.Meshing.preference(mpUseLongSetNames, true); or false
Use long Property names	GenieRules.Meshing.preference(mpUseLongPropertyNames, true); or false

2.2.7.7.1 General

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	52	11 April 2011

Advancing front quad mesher	GenieRules.Meshing.faceMeshStrategy = AdvancingFrontQuadMesher;
Advancing front triangle mesher	GenieRules.Meshing.faceMeshStrategy = AdvancingFrontTriangleMesher;
Sesam quad mesher	GenieRules.Meshing.faceMeshStrategy = SesamQuadMesher;
Uniform distribution	GenieRules.Meshing.edgeMeshStrategy = UniformDistributionEdge;
Linear distribution	GenieRules.Meshing.edgeMeshStrategy = LinearDistributionEdge;
Remove internal vertices	GenieRules.Meshing.eliminateInternalVertices = true; or false
Remove internal edges	GenieRules.Meshing.eliminateInternalEdges = true; or false

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	53	11 April 2011

Command	Script
Fail meshing if $\alpha \ge$ "145"	GenieRules.Meshing.activate(mpMinAngle,mpFail,true);
	GenieRules.Meshing.setLimit(mpMaxAngle,mpFail,145);
Split element in two if $\alpha \ge$ "130"	GenieRules.Meshing.activate(mpMaxAngle,mpSplit,true);
	GenieRules.Meshing.setLimit(mpMaxAngle,mpSplit,130);
Fail meshing if $\beta \leq $ "15"	GenieRules.Meshing.activate(mpMinAngle,mpFail,true);
	GenieRules.Meshing.setLimit(mpMinAngle,mpFail,15);
Split element in two if $\beta \leq "30"$	GenieRules.Meshing.activate(mpMinAngle,mpSplit,true);
	GenieRules.Meshing.setLimit(mpMinAngle,mpSplit,30);

2.2.7.7.2 Max/min angle



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	54	11 April 2011

Command	Script
Fail meshing if Jrel \geq "8"	GenieRules.Meshing.activate(mpMaxRelativeJacobi,mpFail,true);
	GenieRules.Meshing.setLimit(mpMaxRelativeJacobi,mpFail,8);
Split element in two if $Jrel \ge$	GenieRules.Meshing.activate(mpMaxRelativeJacobi,mpSplit,true);
"6"	GenieRules.Meshing.setLimit(mpMaxRelativeJacobi,mpSplit,6);
Fail meshing if Jrel \leq "0.2"	GenieRules.Meshing.activate(mpMinNormalizedJacobi,mpFail,true);
	GenieRules.Meshing.setLimit(mpMinNormalizedJacobi,mpFail,0.2);
Split element in two if Jrel \leq	GenieRules.Meshing.activate(mpMinNormalizedJacobi,mpSplit,true);
"0.4"	GenieRules.Meshing.setLimit(mpMinNormalizedJacobi,mpSplit,0.4);

2.2.7.7.3 Jacobi



2.2.7.7.4 Eliminate edge

Command	Script
Relative length to mesh density	GenieRules.Meshing.activate(mpMinEdge,true);
"0.2"	GenieRules.Meshing.setLimit(mpMinEdge,0.2);
Absolute length "0.05 m"	GenieRules.Meshing.activate(mpMinEdgeByLength,true);
	GenieRules.Meshing.setLimit(mpMinEdgeByLength,0.05m);

🕯 Rules Meshing	X
General Max/Min Angle Jacobi Eliminate	e edge Chord Height
 Eliminate edges shorter than Relative length to mesh density: C Absolute length: 	8? 0.1 8? 0 m
	Cancel Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	55	11 April 2011

Command	Script
Increase mesh density when	GenieRules.Meshing.activate(mpMaxChord,true);
relative chord height $>$ "0.25"	GenieRules.Meshing.setLimit(mpMaxChord,0.25);

General Max/Min Angle Jacobi Eliminate edge Chord Height Maximum free relative chord height Maximum free relative chord height Maximum free relative chord height Image: the state of the state o

2.2.7.8 Sets

Purpose: Set rules for compact or verbose scripting of sets.

This command is scripted. Typically, when disabling compact scripting of sets:

GenieRules.Sets.scriptCompact = false;

2.2.7.9 Compability

Purpose: To specify criteria when reading in command files created in previous program releases.

👪 Compatibility	
✓ Use Plate Snapping	ОК
✓ Plate sorting, COG first ¥? ✓ Use Curve Snapping \$?	Cancel

Command	Script
Use Plate Snapping	GenieRules.Compatibility.enable(PlateSnapping, true); or false
Plate sorting, COG first	GenieRules.Compatibility.enable(PlateSortingCOGFirst, true); or false
Use Curve Snapping	GenieRules.Compatibility.enable(CurveSnapping, true); or false



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	56	11 April 2011

2.2.8 Licenses/features

Here you can select which licenses should be used when GeniE is running on your computer. It can be useful if you are several persons sharing one license for one of the features listed.

2.2.8.1 CurvedGeometry

This should be checked if you are using curved geometry.

2.2.8.2 FrameCodecheck

This should be checked if you are using frame code checks.

2.2.8.3 PlateCodecheck

This should be checked if you are using plate code checks.

2.2.8.4 Enforce use of GeniE.lite license

This should be checked if you want to enforce using a GeniE.lite license, even if a full GeniE license is available.

2.2.8.5 Do not show this dialog again

This should be checked to stop this dialog appearing each time you start GeniE.

👪 Select GeniE features 🛛 🛛 🔀		
۲		
	License	Use
1	CurvedGeom	<u> </u>
2	FrameCodech	v
3	PlateCodeche	v
r Enf I Do	orce use of Ge not show this c	niE.lite license dialog again
		OK

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	57	11 April 2011

2.3 The View pulldown menu

Here you can modify the settings of what you want to see, and you can define the graphics of what you want to see.

None of the commands are scripted. The actions are persistent, i.e. the changes are permanent for the ongoing and new jobs until you modify the view settings again.

<u>B</u> rowser T <u>a</u> bs Status Bar	
Refresh Graphics Toolbars	•
Options	Alt+0

2.3.1 View – Browser

Tick off for browser if you want to see it or if you want to remove it from your view

2.3.2 View – Tab

Tick off for viewing command lines interface windows or if you want to remove it

2.3.3 View – Status bar

Show or hide the status bar.

2.3.4 View – Refresh graphics

Redraws the current graphic views and remove any label.

2.3.5 View – Toolbars

Tick off option for customizing how many toolbars you want on the desktop.

When an option is ticked off it is part of the desktop.

For a description of each toolbar menu, see following Chapters in this User Manual.







5 🕄 🛛	ame	-
-------	-----	---

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	58	11 April 2011
2.3.5.4 Selection		
2.3.5.5 Object Types	5	
/ • ■ • ⊞ /² • ≗ • K ⊨	l •	
2.3.5.6 Create Metho	ods	
沈 認 城 • 今 - 法 観		
2.3.5.7 Default Prop	erties	
<pre>_<no section=""></no></pre> <pre>_</pre>	Ino thickness>	
2.3.5.8 Loadcase		
<no loadcase=""></no>		

2.3.6 View – Options

Access to a number of options controlling the view (Display settings, Category settings, Cursor feedback, Load and result, Presentation). Consult User Manual Volume 1 for further details.

Most of the data specified under View|Options is stored in the Registry for the current GeniE version (in the user profile for the current user). The exception is the creation of colour coding maps which is logged to the journal file and stored in the program database.

View Options			X
General Settings Mouse Color Co	ding Annotation / Diagrams	Browser	
 Model Capacity Models Color code legend Environment E valuators FEM Guiding geometry Hydro Coads and Equipment Structure Utility Utility, Selection Working Set, Active Working Set, Inactive 	Property	Value	Defined where
Default display Save As	Delete	R	estore defaults
		OK Cancel	Apply

2.4 The Insert pulldown menu

From the Insert pulldown menu you can define guiding geometry, feature edges, structural parts and loads (including equipments). In addition you can define compartments environments.

Each of these commands is listed in the following with a typical script command.

Insert	Tools Help
<u>B</u> ea	am 🕨
Pla	ie 🕨 🕨
Sup	port 🕨
<u>j</u> oir	nt 🕨
<u>M</u> a:	55
<u></u> or	npartment •
Eea	ature Edge
Line	ar Slicer
<u>G</u> ui	ding Geometry 🔹 🕨
P <u>r</u> o	file
Equ	ipment •
Exp	licit Load 🔹 🕨
Loa	id <u>⊂</u> ase
Loa	d Co <u>m</u> bination
Env	vironment •

2.4.1 Beam

Here you can choose between different types of beams and specify the end points manually or snapping to points. See Section 3.4 of User Manual Volume 1 for details.

2.4.1.1 **Straight Beam Dialog**

Purpose: Insert a beam and specify end points manually. You can also specify section and material properties manually if these differ from default settings.

This command is scripted. Typically, when inserting Bm2 between (0, 0, 0) and (0, 10, 0) and using section type Pipe_1000mm and material type St 355:

Bm2 = Beam(Point(0 m, 0 m, 0 m), Point(0 m, 10 m, 0 m));Bm2.Material = St 355;*Bm2.Section* = *Pipe_1000mm*;

See Chapter 3 for a definition of reference point modelling.





59

11 April 2011

DET NORSKE VERITAS SOFTWARE

Version 5.3

GeniE 60

11 April 2011

2.4.1.2 Straight Beam

Purpose: Insert a beam graphically by snapping to 2 points.

This command is scripted. Typically, when inserting Bm3 (automatically name) between Point1 and Point 2 when using reference point modelling:

Bm3 = *Beam*(*Point1*, *Point2*);

2.4.1.3 Straight Overlapping Beam

Purpose: To insert an additional beam between two points where there is a straight beam(s) from before.

This command is scripted. Typically, when inserting Bm4 on top of another beam (automatically name) between Point1 and Point 2 when using reference point modelling:

Bm4 = *Beam*(*Point1*, *Point2*, *geAllowOverlap*);

2.4.1.4 Pile

Purpose: To insert a pile (a special variant of a beam concept) whereby the top coordinate is defined by a point, the bottom is defined by an elevation and the direction of a guiding beam. The pile may be a combination of a straight beam and an overlapping beam (i.e. the pile is inside a pile guide or a leg).

This command is scripted. Typically, when inserting Pile1 from Point1 and down to elevation z=-50m using the direction of Bm3 (reference point modelling has been used):

Pile1 = Pile(Point1,GuideLine(Point1, Point1 + Bm3.localSystem.xVector, 0).intersect(ZPlane3d(-50m)));

2.4.1.5 Curved Beam

Bm6 = *CurvedBeam*(*Point1*,*Point2*,*Point3*);

Purpose: Insert a curved beam graphically by snapping to 3 points or more

This command is scripted. Typically, when inserting Bm6 by referring to Point1, Point2 and Point3 (shown with both reference point modelling as well as regular coordinate modelling):

Bm6 = *CurvedBeam*(*Point*(10 m, 10 m, 10 m), *Point*(10 m, 20 m, 20 m), *Point*(10 m, 10 m, 30 m));

Straight Beam <u>D</u> ialog
🖊 Straight Beam
✓ Straight Overlapping Beam
Pile
🔁 <u>C</u> urved Beam

Straight Beam <u>D</u> ialog
🖊 Straight <u>B</u> eam
Straight Overlapping Beam
Pile
🖊 Curved Beam

Straight Beam <u>D</u> ialog
/ Straight Beam
Straight Overlapping Beam Pile
Zurved Beam

Straight Beam <u>D</u> ialog	

Straight	<u>p</u> eam	
Straight	Overlapping	Beam

Pile

Curved Beam

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	61	11 April 2011

2.4.2 Plate

From the insert plate menu you can insert plates and shells using different modelling techniques. For more details see Section 3.5 of User Manual Volume 1 and Section 3.3 of User Manual Volume 3.

2.4.2.1 Flat Plate Dialog

Purpose: Insert a plate and specify corner points manually.

This command is scripted. Typically, when inserting Pl1 using the coordinate values as shown to the right:

Flat Plate <u>D</u> ialog	
🚰 Sweep Curves Dialog	
🔲 Flat <u>P</u> late	
🔷 Skin/Loft <u>C</u> urves	
🚞 Sweep Curve	

Pl1 = *Plate*(*Point*(0 m, 0 m, 0 m), *Point*(0 m, 10 m, 0 m), *Point*(10 m, 10 m, 0 m), *Point*(10 m, 0 m, 0 m));

Pl1.Material = *St_355;*

Pl1.Thickness = *Th_5mm*;

Insert Plate	X
Name : Pl1	
Define (at least 3) corner points :	
1	
n <	2 3
Thickness : Th_5mm 🗨	
Material : St_355 💽	
	OK Cancel Apply

2.4.2.2 Sweep Curves Dialog

Purpose: To sweep (or extrude) a curve(s) along another curve or vector to form a shell.

This command is scripted. Typically, when inserting Pl2 by sweeping Curve2 50 m in x-direction:

Pl2 = *SweepCurve*(*Curve2*, *Vector3d*(50m,0m,0m));



👪 Sweep Curves	×
Curves to sweep (from selection):	
Curve2	_
Sweep curves along	
C Curve:	
 Vector: 50m 0m 0m 	
OK Cancel Apply	

GeniE 62

Flat Plate Dialog ...

🔚 Sweep Curves Dialog ...

Skin/Loft Curves

Sweep Curve

Flat Plate

11 April 2011

Version 5.3

2.4.2.3 Flat Plate

Purpose: Insert a plate graphically by snapping to 4 points or more. A triangular plate is made when 1st and 4th snap point is the same.

This command is scripted. Typically, when inserting Pl3 that has 5 corners (notice that 1^{st} and 6^{th} snap point must be the same – remember to use the tool bar Snap Point Loop):

Pl3 = *Plate*(*Point*(10 m, 0 m, 50 m), *Point*(10 m, 8.75 m, 50 m), *Point*(8.75 m, 10 m, 50 m), *Point*(3.75 m, 7.5 m, 50 m), *Point*(3.75 m, 0 m, 50 m));

2.4.2.4 Skin/Loft Curves

Purpose: To use a skin operation in between two curves or more. It is also possible to include existing shells or plates to guide the shell surface tangent (this is called lofting).

This command is scripted. Typically, when defining Pl4 from a skin operation between Curve1, Curve2 and Curve3:

Pl4 = *SkinCurves*(*Array*(*Curve2*, *Curve3*, *Curve4*));

Similarly, when using plate Pl2 as guidance to control the initial surface of Pl4 (this is a lofting operation):

Pl4 = *LoftCurves*(*Pl2*, *Array*(*Curve2*, *Curve3*, *Curve4*));

2.4.2.5 Flat region

Purpose: To create a flat region consisting of one or several plates. The region is created based on a plane and existing structure forming one or several enclosed areas.

This command is scripted. Typically, when defining Pl5 from a plane at z=4 m, using the area enclosed by the plates Pl1, Pl2, Pl3 and Pl5:

Pl5 = *Plate*(*ZPlane3d*(4 *m*), *Array*(*Pl1*, *Pl2*, *Pl3*, *Pl4*), *Point*(1.5 *m*, 0.6 *m*, 4 *m*));



Flat Plate <u>D</u> ialog Sweep Curves Dialog
Flat Plate
Skin/Loft <u>C</u> urves
Sweep Curve

2.4.2.6 Curve-Net Interpolation

Purpose: To construct a surface that interpolates two arrays of N and M curves, which form a set of (N-1)x(M-1) rectangular patches. Each patch is formed by any two consecutive curves of both arrays.

This command is scripted. Typically, when defining P15 from such an operation between the curve arrays Array(Curve1, Curve2, Curve3) and Array(Curve4, Curve5):

Pl5 = *InterpolateCurveNet(Array(Curve1, Curve2, Curve3), Array(Curve4, Curve5));*

Note that one of the curves may be a single point, provided that the same intersection restrictions hold for this point as well. For instance the command

Pl5 = *InterpolateCurveNet(Array(Point1,Curve1, Curve2, Curve3), Array(Curve4, Curve5));*

will give a valid result, provided that Point1 is the intersection point Curve4 and Curve5, thus forming a triangular patch (which is a degenerate rectangular patch).

2.4.2.7 Sweep Curve

Purpose: To insert a plate or shell by graphically referring to two curves. The first curve defines the width of the object while the second curve specifies the direction.

This command is scripted. Typically, when defining Pl5 by referring to Curve2 and Curve 5:

Pl5 = *SweepCurve*(*Curve2*, *Curve5*);





11 April 2011

2.4.3 Support

Version 5.3

Here you can insert support and specify the support conditions graphically or manually

See Section 3.12 of User manual Volume 1 and Chapter 5 of User Manual Volume 3 for more details on how to define support conditions.

2.4.3.1 Support Point Dialog

Purpose: Insert a support and specify location manually. You may choose between a boundary condition fixed, free, prescribed (used by a load case), dependent (used together with rigid link support), super (used when defining a superelement) and a spring. It is also possible to define a boundary stiffness matrix.

This command is scripted. Typically, when defining a boundary condition Sp1 with fixed translations and free rotations:

Sp1 = *SupportPoint*(*Point*(*15*,*0*,*0*));

Sp1.boundary = BoundaryCondition(Fixed, Fixed, Fixed, Free, Free);

Alternatively when using the boundary stiffness matrix option to define Sp2. In this case values along the diagonal have been inserted:

Sp2.boundary = BoundaryStiffnessMatrix(Stiffness(100 N/m), Stiffness(200 N/m), Stiffness(300 N/m), Stiffness(4000 N*m), Stiffness(5000 N*m), Stiffness(6000 N*m));

Support	\mathbf{X}
Name : Sp1	Cancel
Position :	Apply
Boundary Condition C Boundary Stiffness Matrix Boundary conditions	
I✓ Let x change y and z Spring Fixed Free Prescribed Dependent Spring stiffness	
x UN/m [N/m]	
z [N/m]	
Image: Fixed Free Prescribed Dependent Super Spring stiffness	

Support Point Dialog ...

🛥 Support <u>R</u>igid Link Dialog

A Support Point

🊃 Support Curve

Support 🔀								
١	lame :	Sp2						Cancel
F	Position :				2			Apply
	O Bou	Indary Conditio	on 🖲 Bour	idary Stiffness I	Matrix			
Γ	Stiffne	ss Matrix						
		1	2	3	4	5	6	Boundary Type
	1	0 N/m	0 N/m	0 N/m	0 N	0 N	0 N	Spring
	2	0 N/m	0 N/m	0 N/m	0 N	0 N	0 N	Spring
	3	0 N/m	0 N/m	0 N/m	0 N	0 N	0 N	Spring
	4	0 N	0 N	0 N	0 N*m	0 N*m	0 N*m	Spring
	5	0 N	0 N	0 N	0 N*m	0 N*m	0 N*m	Spring
	6	0 N	0 N	0 N	0 N*m	0 N*m	0 N*m	Spring 💌
	,							Fixed
								Free
								Prescribed
								Dependent
								Super
	Spring							

Rigid Link Dialog 2.4.3.2

Purpose: To insert boundary conditions so that they are dependent upon a point.

This command is scripted. Typically, when defining a rigid link Sp1 where the input parameters are: Independent point is Point7

- Box centre is Point8
- Box extent is $\delta x = 0.2m$, $\delta y = 0.6m$ and $\delta z = 1m$
- Include all edges in region

- Boundary condition: all degrees of freedom are dependent

Sp6 = *SupportRigidLink(Point7, FootprintBox(Point8, Vector3d*(0.2,0.6,1), *LocalSystem*(*Vector3d*(1 m,0 m,0 *m*), *Vector3d*(0 *m*,0 *m*,1 *m*))));

Sp6.includeAllEdges = true;

Sp6.boundary = *BoundaryCondition*(*Dependent*, Dependent, Dependent, Dependent, Dependent, *Dependent*);

👪 Insert Support Rigid Link 🛛 🔀
Name: Sp1
Independent point: X 8?
Boundary condition of independent point
✓ Let x change y and z Spring Fixed Free Prescribed x
z [N/m]
Let rx change ry and rz Fixed Free Prescribed Dependent Super Spring stiffness
Region of dependent points
Include all edges in region
C Include only support points and curves in region
Box center: X
Box extent X: [m] 🛪
Box extent Y: [m] 🛪
Box extent Z: [m] 🛪 💡
Local coordinate system orientation LocalSystem(Vector3d(1 m,0 m,0 m), Vector.
OK Cancel Apply

2.4.3.3 **Support Point**

Purpose: Insert a support point graphically by snapping to 1 point. Per default the support point will be fixed in all six degrees of freedom. To modify these you need to select the support point(s) and edit the conditions.

This command is scripted. Typically, when defining a support point at (0m, 0m, 0m) and modify the rotational degrees of freedom

Sp1 = *SupportPoint(Point(0 m,0 m,0 m))*;

Sp1.boundary = *BoundaryCondition*(*Fixed*, *Fixed*, *Fixed*, *Free*, Free, Free);



65

2.4.3.4 **Support Curve**

Purpose: Insert support conditions along a line; curved or straight. Per default all degrees of freedom are fixed. When making a finite element model all finite element nodes belonging to the support curve will receive same boundary condition.

This command is scripted. Typically, when defining a support curve SC5 and modify some of the degrees of freedom:

Sc5 = *SupportCurve(ModelCurve(Point(24 m,5 m,0 m), Point*(24.5 m, 5 m, 0 m), *Point*(25 m, 5.5 m, 0 m)));

Sc5.localSystemRule = ConstantLocalSystem(LocalSystem(Vector3d(1 m,0 m,0 m), *Vector3d*(0 m, 0 m, 1 m)));

Sc5.boundary = *BoundaryCondition*(*Fixed*, *Free*, *Free*, *Free*, *Fixed*, *Fixed*);

2.4.4 Joint

Insert single joints manually or graphically. Alternatively by using the joint creation rules as defined from *Edit/Rules/Joint Creation*. For more details on how to do it, see Section 3.6 of User Manual Volume 1.

2.4.4.1 **Joint Dialog**

Purpose: Insert a joint by specifying its coordinates manually. Notice that a joint can only be created at a beam intersection.

This command is scripted. Typically, when defining a joint Jt1 at a beam intersection at (4m, 11.5m, 10m):

Jt1 = Joint(Point(4 m, 11.5 m, 10 m));

2.4.4.2 Joint

Purpose: Insert a joint by snapping to 1 point. Same conditions as above apply.

This command is scripted. Typically, when defining a joint Jt1 at a beam intersection at (4m, 11.5m, 10m):

Jt1 = Joint(Point(4 m, 11.5 m, 10 m));

User Manual Vol. VI

Support Point Dialog ... 💶 Support Rigid Link Dialog <u> Support Point</u> 🚾 Support Curve

> Joint <u>D</u>ialog K Joint Generate Joints

Insert Joint Jt1 ΟK Name : Define joint coordinate Cancel Coordinate : 000 Apply

Joint <u>D</u> ialog	
K Joint	
<u>G</u> enerate Joints	



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	67	11 April 2011

2.4.4.3 Generate Joint

Purpose: Insert joints according to the joint creating rules by selecting a beam or beams. Number of joints to be defined depends on the filter settings defined in *Edit/Rules/Joint Creation*.

This command is scripted. Typically, when referring to a beam with 6 intersections a total of 6 joints are created. In this case the rules include that joints shall be created at each beam intersection:

Jt2 = *Joint*(*Point*(0 *m*,28.5 *m*,10 *m*));

Jt3 = Joint(Point(4 m, 28.5 m, 10 m));

Jt4 = *Joint*(*Point*(*12 m*,*28.5 m*,*10 m*));

Jt5 = Joint(Point(18 m, 28.5 m, 10 m));

Jt6 = Joint(Point(24 m, 28.5 m, 10 m));

Jt7 = *Joint*(*Point*(27 *m*,28.5 *m*,10 *m*));

2.4.5 Mass

There are two alternatives for adding a specific point mass. They are described in the following. See Section 4.6.2 for further references.

2.4.5.1 Uniform Point Mass

Purpose: To define a same mass contribution in global x, y and z directions.

This command is scripted. Typically, when adding 1000 tonnes at a specific position:

🎎 Create Point Mass			×
Uniform Point Mass Generic	e Point Mass		
p1	Name : Mass1 Position : Mass:	X [Kg] X	_
	Ca	incel Apply	

Mass1 = PointMass(Point(10,10,10), 1000 tonne);

2.4.5.2 Generic Point Mass

Purpose: To define a mass with different contribution in global translation and rotation x, y and z directions.

This command is scripted. Typically, when adding different mass elements using directional and a diagonal mass matrix:

Mass2 = PointMassMatrix(Point(25,25,25), 5000 Kg, 4000 Kg, 3000 Kg, 10000 Kg*m^2, 8000 Kg*m^2, 6000 Kg*m^2);

hiform	Point Mass	ieneric Point N	lass			
Vame Positic	on :	1	×	Uniform C Direction	Mass nal Mass	 Diagonal Mass Matrix Symmetric Mass Matrix
Lo	ocal System:	2	3	4	5	6
1	0 Kg	0 Kg	0 Kg	0 Kg*m	0 Kg*m	0 Kg*m
	0 Ka	0 Kg	0 Kg	0 Kg*m	0 Kg*m	0 Kg*m
2	0113		0 Ka	0 Ka*m	0 Kg*m	0 Kg*m
2 3	0 Kg	0 Kg	ong	- · · · · · · · · · · · · · · · · · · ·		
2 3 4	0 Kg 0 Kg*m	0 Kg 0 Kg*m	0 Kg*m	0 Kg*m^2	0 Kg*m^2	0 Kg*m^2
2 3 4 5	0 Kg 0 Kg*m 0 Kg*m	0 Kg 0 Kg*m 0 Kg*m	0 Kg*m 0 Kg*m	0 Kg*m^2 0 Kg*m^2	0 Kg*m^2 0 Kg*m^2	0 Kg*m^2 0 Kg*m^2
2 3 4 5 6	0 Kg*m 0 Kg*m 0 Kg*m 0 Kg*m	0 Kg 0 Kg*m 0 Kg*m 0 Kg*m	0 Kg*m 0 Kg*m 0 Kg*m 0 Kg*m	0 Kg*m^2 0 Kg*m^2 0 Kg*m^2	0 Kg*m^2 0 Kg*m^2 0 Kg*m^2	0 Kg*m^2 0 Kg*m^2 0 Kg*m^2

	Joint <u>D</u> ialog
K	Joint
	<u>G</u> enerate Joints

2.4.6 **Compartment Manager**

Purpose: To automatically create compartments for use when creating loads from content or to define a tank subjected for filling in HydroD. The compartment information is also used by Nauticus Hull to automatically generate rules based loading conditions according to the CSR Bulk Rules. All closed voids will form a compartment. For more details, see Section 4.4.1 of User Manual Volume 1.

This command is scripted. Typically: *My_Comp_Manager* = *CompartmentManager()*;

2.4.7 Feature Edge

Purpose: To insert an edge (or line) for controlling the quality of your mesh. Consult Section 3.12.2 of User Manual Volume 1 or Chapter 6 of User Manual Volume 3 to see how this feature can be used.

This command is scripted. Typically:

FEdge1 = FeatureEdge(Point(24 m, 5 m, 1 m), *Point*(24 m, 5. m, 7 m));

2.4.8 Linear Slicer

Purpose: A linear slicer is used to calculate correction moment and shear forces to achieve target values according to CSR rules for bulk ships. The correction moments are automatically applied to the model. To do this it is necessary to first define the linear slicer and then select the slicer to see the moment and shear forces before and after corrections have been applied. This feature is further documented in the training material for Nauticus Hull.

The linear slicer can also be used to compute forces and moments along a defined direction for a given load case.

This command is scripted. Typically:

Slicer1 = LinearSlicer(Point(18 m, -1 m, 10 m), Point(12 m, 30 m, 10 m), Vector3d(0,0,1), 50, 30);

Slicer1.segments = 10;



Create Feature Edge		
Name: FEdge1	_	ОК
End 1 :	×	Cancel
End 2:	x	Apply

🌃 Create L	inear Slicer	X
Name :	Slicer1	
End1:		×
End 2 :		×
C Slicer profile	\$	
Segments:	4	
Width:	[m] ×	
Height:	[m] ×	
Local	Vector3d(0,0,1)	
L		
	Apply(Close


2.4.9 Guiding Geometry

Guiding geometry is often used as a starting position when creating structural parts like beams, plates, stiffeners and shells. There are a number of different guiding geometries that can be used. They are listed in the following. For a detailed explanation on how to create and use guiding geometries, reference is made to Section 3.3 of User Manual Volume 1 and Section 3.2 of User Manual Volume 3.

2.4.9.1 Guide Plane Dialog

Purpose: Insert a guide plane and specify corner points manually, elevations (x, y, or z), length of spacing (constant or varying) in two directions.

This command is scripted. Typically when using different relevant values for u(1, 2, 2, 1) and v(1, 2, 3, 4) and creating 3 guide planes simultaneously by using a step length of 5 meters 2 times:

GuidePlane1 = GuidePlane(Point(0 m,0 m,0 m),Point(10 m,0 m,0 m),Point(10 m,10 m,0 m),Point(0 m,10 m,0 m),4,4,1,2,2,1,1,2,3,4);

GuidePlane1.snapmode = true;

GuidePlane2 = GuidePlane(Point(0 m,0 m,5 m),Point(10 m,0 m,5 m),Point(10 m,10 m,5 m),Point(0 m,10 m,5 m),4,4,1,2,2,1,1,2,3,4);

GuidePlane2.snapmode = true;

GuidePlane3 = GuidePlane(Point(0 m,0 m,10 m),Point(10 m,0 m,10 m),Point(10 m,10 m,10 m),Point(0 m,10 m,10 m),4,4,1,2,2,1,1,2,3,4);

GuidePlane3.snapmode = true;

11 April 2011

	Guide Plane <u>D</u> ialog
	Poly <u>⊂</u> urve Dialog
	Guide Line Dialog
_)	Fillet Curves Dialog
	Guide <u>P</u> oint Dialog
	<u>G</u> uide Plane
\mathbf{r}_{i}	Guide <u>P</u> oint
\sim	Guide Line
r-2	Guide <u>S</u> pline
U	Poly Curve
6	Guide <u>A</u> rc Elliptic
۲	Guide ⊆ircle
м	Model Curve
•	<u>Fillet</u> Curves

Guide Plane			
Name: GuidePlane1			
Type: XY plane At: 0 m [m] Cancel			
Step: 0 m [m] No: 0 Apply			
P4: [Point(0 m,10 m,0 m) P3: [Point(10 m,10 m,0 m) P3			
- (w) y ↑ x(w) P1 P2 P2 P2 P2 P2 P2 P2 P2			
P1: Point(0 m,0 m,0 m) P2: Point(10 m,0 m,0 m)			
u spacings: 4 v spacings: 4			
1.0 1.0 1.0 1.0 1.0			
4 4 G Relative C Absolute			

2.4.9.2 Poly Curve Dialog

Purpose: Insert a guide curve and specify coordinates manually. The options "Enforce tangent vectors" is used to control the vectors of the poly curve at the start and end position.

This command is scripted. Typically when defining 6 reference points to construct the poly curve using the spline option only and controlling the line tangent at start (direction 1,1,0) and end (direction 0,1,0) positions:

Curve1 = *PolyCurve()*;

Curve1.clear();

Curve1.addPoint(Point(1, 1, 1), ggSpline);

Curve1.addPoint(Point(2, 2, 2), ggSpline);

Curve1.addPoint(Point(3, 3, 3), ggSpline);

Curve1.addPoint(Point(4, 4, 4), ggSpline);

Curve1.addPoint(Point(2, 4, 2), ggSpline);

Curve1.addPoint(Point(1, 5, 7), ggSpline);

Curve1.startDeriv(Vector3d(1,1,0));

Curve1.endDeriv(Vector3d(0,1,0));

Curve1.rebuild();

2.4.9.3 Guide Line Dialog

Purpose: Insert a straight guide line and specify end points manually

This command is scripted. Typically between the two points:

Curve1 = *GuideLine*(*Point*(1,1,1), *Point*(2,2,2), 4);

Notice that the number "4" in the expression above is used to divide the guide line into 5 equal parts so that there are 4 internal snap points in addition to the 2 end snap points. This means that if you use "6" the guide line will have 6 internal snap points. To do this you can write in the command input window (remember to click the Enter button):

Curve1 = GuideLine(Point(1,1,1), Point(2,2,2), 6);

👪 Guide Line		×
Name: Curve1		OK Cancel
Define end points :		
End 1 :	×	
End 2 :	×	

11 April 2011

👪 Poly Curve						
Nam	ne: Curve1		Fit Curve To	View OK		
	Curve de	efinition INVALID	Auto Curve T	ype Cancel		
Label curve points Enforce tangent vectors End 1 End 2						
	X [m]	Y [m]	Z [m]	Curve Type		
1				Spline 💌		
2				Spline 🗾		
3				Spline 🗾		
4				Spline 🗾		
5				Spline 🗾		
6				Spline 🗾		
7				Spline 🗾		
8				Spline 💌		
9				Spline 💌		
10				Spline 🗾		
11				Spline 💌		
12				Spline 💌		
13				Spline 💌		
14				Spline 💌		
10				Coline V		

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	71	11 April 2011

2.4.9.4 Fillet Curves Dialog

Purpose: Fillet curves are used to define the curvature between two straight lines. When fillet curves are used, the two straight lines in question and the fillet curves are automatically joined to a composite curve.

This command is scripted. Typically when selecting two curves and specifying a radius of 2 m. The coordinate values are automatically proposed by GeniE):

Curve4 = filletCurves(Curve2, Point(3 m,0 m,0 m), Curve3, Point(5 m,1 m,0 m), 2);

2.4.9.5 Guide Point Dialog

Purpose: Insert a guide point and specify coordinates manually.

This command is scripted. Typically:

Point1 = *Point(7.5 m,5 m,0 m);*

🎎 Fillet Curves				
First Curve:	×	at:		×
Second Curve:	×	at:		×
Radius:	[m]	×		
			Apply	Close

Create Po	int	×
Name:	Point1	
Position:		×
ОК	Cancel Apply	

2.4.9.6 Guide Plane

Purpose: Insert a guide plane graphically by snapping to 4 points. Per default the guide plane will receive 4 equal spacing lengths in u and v directions.

This command is scripted. Typically:

GuidePlane3 = GuidePlane(Point(10 m,0 m,0 m),Point(10 m,10 m,0 m),Point(10 m,10 m,10 m),Point(10 m,0 m,10 m),4,4,1,1,1,1,1,1,1);



GeniE 72

Version 5.3

2.4.9.7 Guide Point

Purpose: Insert a guide point graphically by snapping to 1 point.

This command is scripted. Typically:

Point3 = Point(2.5 m, 0 m, 10 m);

2.4.9.8 Guide Line

Purpose: Insert a straight guide line graphically by snapping to 2 points.

This command is scripted. Typically (notice that the default value for internal snap points is "3"):

Curve5 = *GuideLine(Point(0 m, 0 m, 0 m), Point(2.5 m, 2.5 m, 0 m), 3);*

If you want to change number of internal snap points to 4 you can type the following in the command line window:

Curve5 = *GuideLine(Point(0 m, 0 m, 0 m), Point(2.5 m, 2.5 m, 0 m), 4);*

2.4.9.9 Guide Spline

Purpose: Insert a guide spline graphically by snapping to 3 points or more.

This command is scripted. Typically when using 5 snap points (remember to double click the final snap point):

Curve12 = GuideSpline(Array(Point(5 m,0 m,0 m),Point(7.5 m,0 m,0 m),Point(10 m,2.5 m,0 m),Point(10 m,5 m,2.5 m),Point(10 m,5 m,5 m)),3);

6	Guide Plane <u>D</u> ialog
F	Poly <u>⊂</u> urve Dialog
6	Guide Line Dialog
F 🗗	Fillet Curves Dialog
6	Guide <u>P</u> oint Dialog
	<u>a</u> uide Plane
• •	Guide <u>P</u> oint
N 6	Guide <u>L</u> ine
P 9	Guide <u>S</u> pline
U, E	oly Curve
60	Guide <u>A</u> rc Elliptic
•	Guide <u>⊂</u> ircle
M 1	<u>M</u> odel Curve
Ð	jillet Curves

	Guide Plane <u>D</u> ialog
	Poly <u>C</u> urve Dialog
	Guide Line Dialog
T,	Fillet Curves Dialog
	Guide <u>P</u> oint Dialog
	<u>G</u> uide Plane
•	Guide <u>P</u> oint
\mathbb{N}	Guide <u>L</u> ine
r"	Guide Spline
U	<u>P</u> oly Curve
G	Guide <u>A</u> rc Elliptic
۲	Guide <u>C</u> ircle
м	<u>M</u> odel Curve
•	<u>Fillet</u> Curves



DET NORSKE VERITAS SOFTWARE

User Manual Vol. VI

11 April 2011

Version 5.3

2.4.9.10 Poly Curve

Purpose: Insert a poly curve graphically by snapping to 3 points or more.

This command is scripted. Typically when using 6 snap points (remember to double click the final snap point):

Curve14 = PolyCurve(Array(Point(5 m, 10 m, 10 m), Point(7.5 m, 10 m, 10 m), Point(10 m, 7.5 m, 10 m), Point(10 m, 5 m, 10 m), Point(7.5 m, 2.5 m, 10 m)), Point(5 m, 2.5 m, 10 m)));

Curve14.modifyCurveType(2, ggStraight);

Curve14.modifyCurveType(4, ggStraight);

Notice that in this case there are two parts that are straight as a result of an automatic curve fitting process. This can be changed from editing the poly curve.

2.4.9.11 Guide Arc Elliptic

Purpose: Insert a guide arc elliptic graphically by snapping to 3 points (origin, start and end of ellipse)

This command is scripted. Typically:

Curve20 = *GuideArcElliptic(Point(10 m,2.5 m,7.5 m), Point(10 m,5 m,5 m), Point(10 m,2.5 m,10 m), true);*

Guide Plane <u>D</u>ialog ... Poly <u>C</u>urve Dialog ... Guide Line Dialog ... Fillet Curves Dialog ... Guide Point Dialog ... Guide Plane Guide Plane Guide Plane Guide Line Guide Spline Dialog Line Guide Spline Coly Curve Guide Arc Elliptic Guide Circle Model Curve Fillet Curves



2.4.9.12 Guide Circle

Purpose: Insert a guide circle graphically by snapping to 3 points (origin, radius, plane). The plane is a result of the three points.

This command is scripted. Typically:

Curve24 = *GuideCircle(Point(10 m,2.5 m,2.5 m), Point(10 m,5 m,2.5 m), Point(10 m,0 m,5 m));*

2.4.9.13 Model Curve

Purpose: Insert a model curve by snapping between two points. The model curve will follow the surface curvature (if any) between the two points. Per default there are 3 internal snap points- these can not be altered.

This command is scripted. Typically:

*Curve*25 = *ModelCurve*(*Array*(*Point*(5 m, 2.5 m, 10 m), *Point*(6.327271284 m, 2.174330704 m, 14.9859724 m), *Point*(7.5 m, 2.5 m, 20 m)), 3);

2.4.9.14 Fillet Curves

Purpose: To create a fillet curve by clicking on two adjacent lines. GeniE will propose a minimum fillet radius in between; this can be altered by typing a new radius followed by Enter.

This command is scripted. Typically when using a radius of 1 m:

Curve29 = filletCurves(Curve10, Point(5.688202143 m,5.688202143 m,0 m), Curve4, Point(5 m,4.148269176 m,0 m), 1);

Guide Plane <u>D</u> ialog
Poly <u>C</u> urve Dialog
Guide Line Dialog
📄 Fillet Curves Dialog
Guide Point Dialog
📆 Guide Plane
Guide Point
📉 Guide Line
💤 Guide Spline
🛄 <u>P</u> oly Curve
🚰 Guide <u>A</u> rc Elliptic
🔟 Guide <u>C</u> ircle
Model Curve
Fillet Curves

Guide Plane <u>D</u> ialog
Poly <u>⊂</u> urve Dialog
Guide Line Dialog
🗃 Fillet Curves Dialog
Guide Point Dialog
🔢 <u>G</u> uide Plane
Guide Point
📉 Guide Line
📂 Guide Spline
🛄 <u>P</u> oly Curve
🚰 Guide <u>A</u> rc Elliptic
回 Guide <u>C</u> ircle
Model Curve
Fillet Curves
Guide Plane <u>D</u> ialog



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI

75

Version 5.3

2.4.10 Profile

Purpose: Insert a profile to be used for punching or cut operation (from *Tools/Structure/Punch*). See Section 3.6.7 of User Manual Volume 1 for details on how to use this feature.

This command is scripted. Typically:

Manhole = *ProfileRR*(2*m*, 1*m*, 0.25*m*);

Guide Profile Properties	
Rounded Rectangle	
× •	
	Height [m] 🛪
	Width [m] ×
	Radius [m] 🗙
OK.	Cancel Apply

2.4.11 Equipment

Purpose: Define prismatic equipment for use in load cases to generate masses or loads. For further details see Section 3.9 of User Manual Volume 1.

This command is scripted. Typically with height 2m, length 5m, width 3m and mass 1000 tonne:

Equipment1 = PrismEquipment(5m,3m,2m,1000
tonne);



2.4.11.1 COG offset

Purpose: To define the COG manually relative to the footprint centre. The COG may also be outside the equipment box. Define the equipment, select the equipment from the browser, RMB and choose Properties to edit the equipment.

This command is scripted. Typically when defining the COG position at (1m, 1m, 3m):

Equipment1.centreOfGravity
(Vector3d(1m,1m,3m));



Object Properties Section Ma	aterial Equipment Load Interface Local Syst	em
Name : Equipment1		
	H : 2 m	[m]
	L: 5 m	[m]
	W : 3 m	[m]
	Mass : 1000000 Kg	[Kg]
Specify Footprint	COG offset from footprint center:	_
Linear varying loads		
Load case : <n a=""></n>		_
Position :		
]		
	OK Cancel	Apply

2.4.11.2 Specify footprint

Purpose: To define the footprint of the equipment. There are 4 templates that can be edited. The footprint may be outside the equipment border. Define the equipment, select the equipment from the browser, RMB and choose Properties to edit the equipment. When you click on the tab Specify Footprint there are 4 templates available.

There may be more than 4 footprints.

This command is scripted and dependent on which template is chosen and the size of each footprint. Typically:

Equipment1.clearFootprint();

Equipment1.addToFootprint(1.25 m, 2.5 m, 0.75 m, 1.5 m); Equipment1.addToFootprint(1.25 m, 2.5 m, -1.5 m, -0.75 m); Equipment1.addToFootprint(-2.5 m, -1.25 m, 0.75 m, 1.5 m); Equipment1.addToFootprint(-2.5 m, -1.25 m, -1.5 m, -0.75 m);

Properties		X
Object Properties Section Ma	aterial Equipment Load Interface Local System	n
	H: 2m L: 5m W: 3m Mass: 1000000 Kg	[m] [m] [Kg]
Specify Footprint Specify Footprint Linear varying loads Load case : <n a=""> Position :</n>	COG offset from footprint center: Vector3d(1 m,1 m,3 m)	
]	OK Cancel	Apply



11 April 2011

GeniE 76

2.4.11.3 Linear varying loads

Purpose: To specify whether generated loads shall be constant or linear (linear is the default option). It is necessary to place the equipment in a load case before this can be modified from linear to constant.

This command is scripted. Typically when altering to constant loads or linear for the equipment Helideck in load case LC_heli:

LC_heli.setCurrent(); LC_heli.constantLoad(Helideck); LC_heli.varyingLoad(Helideck);

User Manual Vol. VI

11 April 2011

Name : Equipment 1	ateriar Equipmont Ebad interrace Ebdar Sys	em
	H: 2m L: 5m W: 3m Mass: 1000000 Kg	[m] [m] [m] [Kg]
Specify Footprint Linear varying loads Load case : <n a=""> Position :</n>	COG offset from footprint center: Vector3d(1 m,1 m,3 m)	_

2.4.11.4 Position

Purpose: To specify the exact location of the equipment. The position denotes the location of the centre of the equipment bottom plane. It is necessary to place the equipment in a load case before the position can be modified.

This command is scripted. Typically when modifying the location to (15m, 20m, 12m):

LC_heli.placeAtPoint(Helideck, Point(15m,20m,12m), LocalSystem(Vector3d(1m,0 m,0m), Vector3d(0m,0m,1m)));

Properties		
Object Properties Section Ma	terial Equipment Load Interface Local Syst	em
Name : Helideck		
	H: 10 m	[m]
	L: 15 m	[m]
↓ L = ZJ	W : 13 m	[m]
	Mass : 50 tonne	[tonne]
Specify Footprint	COG offset from footprint center:	_
Linear varving loads	Vector3d(0 m,0 m,5 m)	
Load case : LC_heli		
Position : Point(11 m,11.	5 m, 10 m)	
	Cancel	Apply

2.4.12 Explicit Load

Purpose: To Insert and define an explicit load like point load, line load, surface load, prescribed displacement and line temperature. A load case must be defined and set to current before any loads can be applied. For more details on how to apply loads, reference is made to Section 3.11 of User Manual Volume 1 and Chapter 4 of User Manual Volume 3.

2.4.12.1 Point Load

Purpose: To define a point load. This load can only be applied along a beam to give effect. The load is independent of the structure meaning that it does not follow when the relevant beam is moved or copied.

The point load may be defined according to the global coordinate system (default) or a local coordinate system.

This command is scripted. Typically:

PLoad1 = PointLoad(LC1, Point(4 m,5 m,10 m), 100 kN, 200 kN, -300 kN, -1000 kN*m, -2000 kN*m, 3000 kN*m);

2.4.12.2 Line Load

Purpose: Define a constant or linearly varying line load between two points along a beam. This load can only be applied along a beam to give effect. The load may be independent or dependent of the structure. To make the line load independent of the beam the load intensities relate to two end points. In this case the line load does not follow when moving or copying the beam.

This command is scripted. Typically for a linear variation in z-direction:

 $\begin{array}{l} LLoad1 = LineLoad(LC1, \ FootprintLine(Point(4\ m, 0\ m, 10\ m), \ Point(12\ m, 0\ m, 10\ m)), \\ Component1dLinear(Vector3d(0\ kN/m, 0\ kN/m, -300\ kN/m)), \\ Vector3d(0\ kN/m, 0\ kN/m, -100\ kN/m))); \end{array}$

11 April 2011

Point Load ... Line Load ... Surface Load ... Prescribed Displacement ... Line Temperature ...

Insert Point Load in: LC	1		
F _{xy2} p p p			OK Cancel Apply
Name: PLoad1	×		
Forces and Moments :	n		1
Fx: ON [7	v]	Mx: 0 N*m	[N*m]
Fy: ON [*	N]	My: 0 N*m	[N*m]
Fz: 0 N [1	٧]	Mz: 0 N*m	[N*m]



GeniE 78

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	79	11 April 2011

To make the line load dependent of the beam the load intensities relate to a beam(s) end positions. In this case the line load follows when moving, but not copying the beam. To switch from independent to dependent line load choose from footprint "line" or "beam".

You can select multiple beams graphically and apply the same line load to all by using the radio button "Apply to selected beams".



This command is scripted. Typically for a constant line load in z-direction between relative positions 0.2 and 0.7 on beam Bm14:

LLoad2 = LineLoad(LC1, FootprintBeam(Bm14, 0.2, 0.7), Component1dLinear(Vector3d(0 kN/m, 0 kN/m, - 300 kN/m), Vector3d(0 kN/m, 0 kN/m, -100 kN/m)));

2.4.12.3 Surface Load

Purpose: To define a surface load on a plate or a shell. This load can only be applied on a plate or shell to give effect. The load may be independent or dependent of the structure. To make the line load independent of the plate the load intensities relate to three or more points. In this case the line load does not follow when moving or copying the beam.

There are a number of different intensities that can be used. Slide the "Intensities" bar to see the options.

Pressure Component Load Dummy Hydro Pressure Pressure Traction

Na	ame: LLoad2				
^c ootprints	Beam: Bm14			Apply to selec	ted beams
	Parameter positions Start: 0.2 End: 0.7	<mark>8</mark> 3			
ī	🗌 Linear varying load				
sities	fx1: 0 kN/m	[kN/m]	fx2:	0 kN/m	[kN/m]
Inten	fy1: 0 kN/m	[kN/m]	fy2:	0 kN/m	[kN/m]
	fz1: -300 kN/m	[kN/m]	fz2:	-100 kN/m	 [kN/m]
	Local coordinate system	LocalSyste	:m(Vec	stor3d(1 m,0 m	
				Chara	Alu

Jå C	reat	e Surface Load		\mathbf{X}
Na	me: - Def	SLoad1 ine (at least 3) corner	points	
Footprints	1	000 0100 10100 1000		Surface Normal:
	- Cor	stant Pressure		
Intensities	(Constant intensity: 0	Pa	[Pa]
			E	Close Apply

For plates the surface load may be independent or dependent. For shells the surface load must always be dependent by using a wet surface. In the following, all examples are based on a wet surface WS1.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	80	11 April 2011

🎎 Create Surface Load

1 000

0100

10100

1000

Linear Pressure Function

Create a pressure function described by the equation k1*x+k2*y+k3*z+k4

0 KPa/m

0 KPa/m

0 KPa/m

0 KPa

Liquid pressure (positive values only)

Define (at least 3) corner points

2

Surface Normal: Vector3d(0 m,0 m,-1 m)

[KPa/m]

[KPa/m]

[KPa/m]

Apply

[KPa]

Close

Name: SLoad1

Footprints

Intensities

k1

k2 k3

k4

-

2

Constant

Javascript

é

3 Point Varying

Linear Function

2.4.12.3.1 Pressure

Purpose: To define a pressure on a plate or a shell. There are four options: Constant, 3 Point Varying, Linear Function (as shown below) and Javascript. The option Liquid pressure (positive values only) is among others used to compute hydrostatic pressure as only the positive pressure component is computed. Notice that this is the default option.



SLoad1 = SurfaceLoad(LC1, FootprintWetSurface(WS1),

2.4.12.3.2 Traction



SLoad2 = SurfaceLoad(LC1, FootprintWetSurface(WS1), Traction2dConstant(100 KPa, Vector3d(0 m,0 m,-1 m)));

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	81	11 April 2011
2.4.12.3.3 C	omponent Load	
Purpose: To define a component le shell. There are two options: Consideration below) and Javascript. This command is scripted. Typical and Fz = 200 KPa: SLoad3 = SurfaceLoad(LC1, FootprintWetSurface(WS1), Comp KPa, 0 KPa, 200 KPa));	bad on a plate or a tant (as shown ly for Fx = 100 KPa onent2dConstant(100 $\boxed{Component Load} \\ \hline{Constant} \\ \hline{S Point Varying} \\ \hline{Linear Function} \\ \hline{Javascript} \\ \hline{S}$	Vertex SLoad3 Wet Surface Select a wet surface to apply loads on WS1 Image: Standard Component 2d) Fx: 0 KPa Fy: 0 KPa KPa [KPa] Fy: 0 KPa KPa [KPa]

TA Create St

2.4.12.3.4 Dummy Hydro Pressure

Dummy Hydro Pressure 💌

 \mathbb{R}

C Constant C 3 Point Varying C Linear Function C Javascript

Purpose: To define a wet surface for use in HydroD. The wetted surface is used to decide the surfaces subjected to the sea (always finite element load case number 1) or surfaces belonging to a compartment (one load case per compartment). There are no load intensities to add since all the load application will be done in HydroD (Wadam).

Footprints	me: SLoad4 Wet Surface Select a wet surface to apply loads on WS1	
Intensities I	Dummy Hydro Pressure Dummy hydro pressure is used to transfer external sea pressure and internal tank pressure to WADAM. All dummy hydro pressure loads created in the LoadCase with FEM number 1 will represent external sea pressure. All dummy hydro pressure loads created in other LoadCases will represent tanks (That is, one tank per LoadCase)	
	Close Apply	

This command is scripted. Typically:



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	82	11 April 2011

2.4.12.4 Prescribed Displacements

Purpose: To specify a prescribed displacement at a support point. It must be used in connection with a prescribed boundary condition. See Section 3.11.5 of User Manual Volume 1 and Section 4.2.4 of User Manual Volume 3 for more details.

This command is scripted. Typically for a support point that has prescribed conditions in x and y direction with prescribed displacements of 2mm and 5mm respectively:

Sp3.boundary = BoundaryCondition(Prescribed, Prescribed, Fixed, Fixed, Fixed);

PDisp1 = PrescribedDisplacement(LC1, Sp3, 2mm, 5mm, 0 m, 0 deg, 0 deg, 0 deg);

2.4.12.5 Line Temperature

Purpose: To define constant or linearly varying temperature intensity between two points along a beam. Notice that the temperature is constant over the beams cross section. It is possible to specify the temperature by referring to two explicit points; in this case the load becomes independent of the beam and will not follow when the beam is moved or copied.

There are two options: Linear (as shown below or Javascript). Slide the "intensities" bar to change between the two options.

Temperature C Linear Javascript

6

For more details, see Section 3.11.4 of User Manual Volume 1 or Section 4.2.3 of User Manual Volume 3.

This command is scripted. Typically for a constant temperature of 100 delC between two points:

LLoad5 = LineLoad(LC1, FootprintLine(Point(2 m,0 m,1 m), Point(2 m,5 m,1 m)), Temperature1dLinear(100 delC, 100 delC));

insert Prescrineu Disp	lacei	mem			
da	-			(JK.
dy				Ca	ncel
r dx				Aj	oply
K					
Name : PDisp1					
Support :	-	×			
Translations and Rotation	ns				
dx: 0 m	[m]	IX:	U deg		[deg]
dy: 0 m	[m]	ry:	0 deg		[deg]
dz: 0 m	[m]	rz:	0 deg		[deg]
		_			

👪 Create Line Load	X
Name: LLoad1	
E Cotpuist P1: p2: I I I I I I I I I I I I I I I I I I I	X X
t1 tensities t2	
1: 0 delC 12: 0 delC	
	Close Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	83	11 April 2011

Temperature C Linear Javascript

 \mathbb{R}

It is also possible to specify the temperature by referring to relative positions of a beam(s); in this case the load becomes dependent of the beam and will follow when the beam is moved but not when it is copied.

There are two options: Linear (as shown below or Javascript). Slide the "intensities" bar to change between the two options.

	🎎 Create Line Load	
ise	Name: LLoad5	
	Beam: Parameter positions Start: 0 End: 1 9? 1 1 1 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 1 2 1 2 1 2 1 2 1 1 2 1 1 2 1 1 1 1 1 1 1 1 1 1 1 1 1	☑ Apply to selected beams
		Close Apply

This command is scripted. Typically for a varying temperature load between the two end positions on beam Bm20:

LLoad6 = LineLoad(LC1, FootprintBeam(Bm20), Temperature1dLinear(50 delC, 100 delC));

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	84	11 April 2011

2.4.13 Load Case

Purpose: To define a load case where equipments, weight lists, and explicit loads are applied. For more details see Section 3.8 of User Manual Volume 1 and Section 4.1 of User Manual Volume 3.

This command is scripted. Typically:

LC1 = LoadCase();

2.4.14 Load Combination

Purpose: To define load combinations built up of other load cases. It is possible to use a load combination in another load combination. For more details see Section 3.8 of User Manual Volume 1 and Section 4.1 of User Manual Volume 3.

This command is scripted. Typically:

LC_Comb = LoadCombination(); LC_Comb.addCase(LC1, 1); LC_Comb.addCase(LC3, 1);

Insert Load Co	ombinat	ion		×
Name: LC_Comb	5	D	D	
	Factor 1	Phas 0	LoadCase	
M www rc3	1	0	LoadCase	
		OK		Cancel

👪 Insert Load Case			
Name: LC1			
Dummy Hydro Pressure			
ОК	Cancel	Apply	

GeniE 85

11 April 2011

Version 5.3

2.4.15 Insert – Environment

Purpose: To define global properties for air, water and the soil. Furthermore, the data for running a deterministic time condition (air and/or wave) is set up herein. Please see Volume 2 of the User Manual for more details.

2.4.15.1 Insert Location

Purpose: To define properties for air, water and the soil. All data for the air, water and the soil will be scripted upon clicking the "OK" or "Apply" buttons. The program default values are shown on the pictures. For the soil you may define soil type, soil curves and soil data from this input dialogue or select from the browser *Environment -> Soil*. See next Chapter for details.

This command is scripted. Typically:

Location1 = Location(0m, -1 m); Location1.gravity = 9.80665 m/s^2; Location1.air.density = 1.226 Kg/m^3;

Location1.air.kinematicViscosity = 1.462e-005 m^2/s;

Location1.water.density = 1025 Kg/m^3;

Location1.water.kinematicViscosity = 1.19e-006 m²/s;

Location1.seabed.normaldirection = Vector3d(0 m, 0 m, 1 m);

Location1.relativeSoilLayers = false;

å Create/I	dit Location					X
📚 Name	Location1	-				
Gravitur 9	00665 m/s^2 [m/s^2]					
Air Jazza						
o≡ jwa Densitu:	1.226 Ka/m^3				[Ka/m	^31
Kinematic vis	cosity: 1.462e-005 m^2	2/s			[m^2/	s]
				OK	Cancel	Apply
👪 Create/I	dit Location					X
📚 Name	_ocation1	•				
Gravity: 9.1	30665 m/s^2 [m/s^2]					
Air Wat	er Soil					
Density:	1025 Kg/m^3				[Kg/m	^3]
Kinematic vis	cosity: 1.19e-006 m^2/	\$			[m^2/s	2]
Waterline Z	Om				[m]	
				OK	Cancel	Apply
👪 Create/I	dit Location					
📚 Name	_ocation1	•				
Gravity: 9.	80665 m/s^2 [m/s^2]					
Air Wat	er Soil					
Seabed Z (mu	dline): -1 m					[m]
Seabed delta:	Q2				_	×
Soil layers	relative to seabed Z 8?	•				
Soil layers: 🖇	ç.					
Z Bott	om Soil Type	Soil Curves	Soil Data		Sublayers	^
2						=
3 4						
5						~
			Г	ок	Cancel	Apply

🙇 Location

💹 Deterministic Time Condition

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	86	11 April 2011

2.4.15.2 Insert Deterministic Time Condition

Purpose: To define how the environmental data will be used in a deterministic time domain analysis. The picture below shows that there is no default values set up for the wave load condition. As a minimum, the values for a regular wave set and the wave model must be specified to create a wave load condition. The relevant data for Current, Wind, Direction Set, Phase set, Wave height set, Wave height function may be defined from this dialogue or from the browser *Environment -> Directions* or *Environment ->Water*. See next Chapter for details. The Wave model may be of type *Airy, CalmSea, Cnoidal, Stoke5 or StreamFunction*. For StreamFunction it is also necessary to fill in the order as an integer between 1 and 24.

A New Wave Load Condition						
Deterministic Time	85					
Same Condition1	_					
Wave components		Assign war	ve component prope	rties		
Regular wave set:	- x [Current	profile:	~		
,	r	Wind p	rofile:	~		
Direction set:	_	Waver	nodel:	~		
Frequency set:	~	Order:				
Phase set:	~		Fill all	\ ?		
Wave height set:	-		Fill selected	83		
C Wave height function:	~	Fill e	equal components	\ ?		
Specifu value: ♀						
Period Height Phase Direction	Current profile		Wind pr	ofile	Wave model	Order
1						
3						
5						
6						
8						
9 10						
Location: Location1	-				Cancel	Apply

This command is scripted. The example below shows the script commands where the wave set "WaveSet1" (period 15 sec, wave height 5m, direction 0 deg and phase angle 0 deg) and the current profile "CurrentProfile1" (with varying current from 1 m/s^2 to 0.1 m/s^2 between elevation 10m above sea level down to sea bottom at -100m) are used. In addition the wave model is set to Stokes 5th. Typically:

WaveSet1 = RegularWaveSet();

WaveSet1.heightType(rwsHeight);

WaveSet1.add(RegularWave(0,5,WavePeriod(15),0));

CurrentProfile1_Elevations = Array(10,0,-50,-100);

CurrentProfile1_Directions = Array(0,0,0,0);

// 1 m/s² at 10m and 0m, 0.5 m/s² at -50m, 0.0 m/s² at -100m

CurrentProfile1_Velocities = *Array*(1,1,0.5,0);

CurrentProfile1 = *CurrentProfileRelDir(CurrentProfile1_Elevations,CurrentProfile1_Directions,CurrentProfile1_Velocities,dtRelativeHeading);*

Condition1 = DeterministicTime(Location1);

Condition1.waterSurface.regularWaveSet = WaveSet1;

Condition1.populate();

Condition1.component(1).water.current(CurrentProfile1);

Condition1.component(1).waterSurface.waveModel(Stokes5());

2.5 The Tools pulldown menu

The Tools pulldown menu contains commands to create a mesh, execute analysis, look at results, import equipment lists, create new material based on mass scaling, clean-up and divide of geometry, find length and angles and customize names and picture generation.

Each of these commands is listed in the following with a typical script command.

2.5.1 Tools – Analysis

Run your analysis, view results, export or import Sesam neutral files, perform code checking using previous code checking standards as part of Framework, locate finite elements and results presentations.

Create <u>M</u> esh	Alt+M
Activity Monitor	Alt+D
Export <u>F</u> EM File	
Import External Results SIN file	
Frame Code <u>C</u> heck (Framework)	
Advanced <u>R</u> esults (Xtract)	
Locate FE	
Presentation	Alt+P
Beam Result Diagrams	Alt+G
Show Analysis and Results	

Dimension

Customize ...

2.5.1.1 Create Mesh

Purpose: Create a finite element mesh and produce a FEM file.

This command is not scripted. You may edit the journal file and include CreateMesh();

Create <u>M</u> esh	Alt+M
<u>A</u> ctivity Monitor	Alt+D
Export <u>F</u> EM File	
Import External Results SIN file	
Frame Code <u>C</u> heck	
Locate FE	
Presentation	Alt+P
Beam Force/Stress Diagram	
Show Analysis and Results	

Tools Help Analysis • Equipment • Properties • Structure •

۲

Alt+C

11 April 2011

Version 5.3

2.5.1.2 Activity Monitor

Purpose: To select one of the predefined processes for executing a linear structural analysis with or without wave loads, pile and soil. The analysis programs Sestra, Wajac and Splice are used by GeniE to solve the analyses; these programs require a separate license key to run.

Each of the processes is pre-defined, but it is possible to edit and keep the settings for later use within the same workspace.

This command is scripted. Typically: Analysis1 = Analysis(true); Analysis1.add(MeshActivity()); Analysis1.add(LinearAnalysis()); Analysis1.add(LoadResultsActivity());

2.5.1.2.1 Linear structural analysis - static

Purpose: To execute and control input data for a static linear structural analysis. For more details see Section 3.13 of User Manual Volume 1 and Chapter 7 of User Manual Volume 3.

Provided you tick off for "Journal activity executions" this command is scripted. Typically:

Analysis1.execute();

When the analysis has been performed each activity will contain a status flag *Success*, *Warning* or *Error*.

To investigate you right click on one of the activities to find more information. In the example to the right the listing file from Sestra is chosen.

This command is not scripted.

🎎 Create Linear Static Analy	sis 🔀
Name: Analysis1 Automatically import global load Available activities Meshing Wave Load Activity Linear Structural Analysis Pile Soil Analysis Load Results	adcases
ОК	Cancel

👪 Activity Monitor			X
۲		Start	Cancel
Journal activity executions Activity	Duration	Status	Generate Input
🗹 💁 1 - Analysis1 - Analysis	0s	Not Started	
🗹 🌮 1.1 - Meshing (Always Regen	0s	Not Started	
1.1.1 - Delete loads	0s	Not Started	
1.1.2 - Generate loads	0s	Not Started	
1.1.3 - Delete mesh	0s	Not Started	
1.1.4 - Generate mesh	0s	Not Started	
$\mathbf{M} \stackrel{\mathbf{Kr}}{= \mathbf{R}}$ 1.2 - Linear Structural Analys	0s	Not Started	Yes
R 1.3 - Load Results	0s	Not Started	



DET NORSKE VERITAS SOFTWARE

Version 5.3

It is also possible to set up criteria for the meshing. Right click the activity *Meshing* to access the relevant commands.

The *Meshing Rules* will open the command *Edit*/*Rules*/*Meshing*.

The *Export beams as members* will ensure that the finite element model contains member data in addition to pure finite element data -a

member may be built up from several finite elements. You should always use this option unless you

know you will not need it.

The *Smart load combinations* will reduce the analysis time since the load combinations are not part of a structural analysis. The results in GeniE include load combinations since all results are combined after the analysis. You are advised to use this option unless there are particular reasons for not doing so. Typical examples may be if you want all load combinations to be part of Sestra or you want to use load combinations for other analysis

purposes like in Usfos.

From the Override Global Superelement Data you can set the superelement type. The default is 1 and you can modify it from here or from

the Meshing Rules (*Edit/Rules/Meshing*).

When ticking off for the *Set Mesh Priority* the meshing will be performed accordingly, i.e. prioritized plates or beams will be meshed first. Please see the Chapter "The Browser Menu" to learn how to create a mesh priority.

The Mesh Subset option is used to mesh parts of the concept model only. A boundary condition may also be part of a set – this means that a complete finite element model may be generated for sets (for analysis or for creation of super elements) as intersecting loads are automatically included.

This command is scripted, typically: Analysis1.step(1).subset = bottom_deck; Analysis1.step(1).meshPriority = MeshPriority1;

🞎 Activity Monitor Loading results Start Cancel Journal activity executions Activity Duration Status Generate Input 🗹 🎭 1 - Analysis1 - Analysis 19s Success 🗹 🤁 🛛 1.1 - Meshing (Always R 🔁 Edit acti \checkmark 1.1.1 - Delete loads 0s \checkmark 1.1.2 - Generate loads 0s Success V 1.1.3 - Delete mesh 0s Success

7s

12s

Ωs.

Success

Success

Success

🎎 Mesh activity		×
Meshing Rules	Regenerate mesh option Always Regenerate Mesh	% ? ▼
Smart load combination	ns .	
Top Superelement Type Superelement Type	1 1 1	
🔲 Set Mesh Priority		
🔲 Mesh Subset	_	
Pile boundary condition	Pile Soil Interaction	
0K	Cancel Ap	ply

å Mesh activity	
Meshing Rules Export beams as members Smart load combinations Override Global Superelement	Regenerate mesh option 🔗? Always Regenerate Mesh 🔍 Data
Top Superelement Type	
Superelement Type 2 ✓ Set Mesh Priority Mesh Subset MeshPiel Pile boundary condition Pile Soi	iority1
OK Ca	ncel Apply

11 April 2011

Yes

 \checkmark

1.1.4 - Generate mesh

■ Kr 1.2 - Linear Structural Analys...

🗹 R 1.3 - Load Results

To modify the analysis activity you right click the activity *Linear Structural Analysis* and select *Edit activity*.

The *Datacheck Only* will set up the input file to Sestra so a data check is performed but no execution of finite element analysis.

This command is scripted, typically: Analysis1.step(2).dataCheckOnly = true;

If you want to edit and keep the input file to Sestra you should deselect the *Automatic generation of input files* and click on the file sestra.inp to edit. If you do not deselect this option the modified input file to Sestra is not kept for subsequent analysis.

This command is scripted, typically: Analysis1.step(2).generateInput = false;

The option for Warp Correction will use Sestra's option for improving the stiffness for warped 4 noded shell elements (i.e. elements with "nodes out of plane").

This command is scripted, typically: Analysis1.step(2).warpCorrection = false;

🚨 Activity Monitor ۲ Start Cancel Journal activity executions Activity Duration Status Generate Input ✓ ⁹ 1 - Analysis1 - Analysis ✓ Ø 1.1 - Meshing (Always Regen... Not Started 0s Not Started 0s V 1.1.1 - Delete loads Not Started 0s ً 1.1.2 - Generate loads 0s Not Started Image: Constraint of the second system is a second system of the second system is a second system of the second system is a second system of the 1.1.3 - Delete mesh 0s Not Started 1.1.4 - Generate mesh 0s Not Started 1.2 - Linear Structura 0s Er activity 0s

🚨 Linear Analysis		×
 □ Datacheck Only □ Analysis type ○ Static Analysis ○ Eigenvalue Analysis 	tomatic generation of input files Eigenvalues Solver egLanczos Number of Modes 10 Shift 0	
Advanced	e elements)	ply

sestra.inp sestra.lis sestra.mnt

GeniE 90

DET NORSKE VERITAS SOFTWARE	GeniE	Use
Version 5.3	91	

2.5.1.2.2 Linear structural analysis - eigenvalue

Purpose: To execute and control input data for an analysis.

Provided you tick off for "Journal activity executi command is scripted. Typically:

Analysis1 = Analysis(true);

Analysis1.add(MeshActivity());

Analysis1.add(LinearAnalysis());

Analysis1.step(2).setEigenvalueAnalysis(egLancz

Analysis1.add(LoadResultsActivity());

Analysis1.execute();

The default option is to use Lanczos solver with 10 eigen-modes. To modify you right click the activity Linear Structural Analysis and select Edit activity.

The following eigenvalue algorithms are available for free vibration analysis:

- An implicitly restarted Lanczos' method combined with a multifrontal solver
- Householder's method
- Subspace Iteration
- Lanczos' method

This command is scripted, typically:

Analysis1.step(2).setEigenvalueAnalysis (egMultifrontLanczos, 15, 2);

8			
eigenvalue	👪 Create Line	ar Static Analy	/sis 🔀
ions" this os, 10);	Name: Analys Automatic Available activi Meshing Wave Loa Vinear Str Pile Soil A Load Res	sis1 ally import global lo ties ad Activity uctural Analysis nalysis ults	adcases C Static C Eigenvalue
	OK		Cancel
👪 Activity Monitor			×
۲		Start	Cancel
Activity Image: Source of Control of Co	Durat kalysis Os Always Regen Os al ods Os ate loads Os ate loads Os ate mesh Os uctural Analys Os	ion Status Not Started Not Started Not Started Not Started Not Started Not Started Ef activity sestra.inp sestra.lis sestra.mnt	Generate Input
1 Inear Analysis			
Datacheck Only Analysis type Static Analysis Eigenvalue Analysis Advanced Warp Correctio	Automatic gene Eigenvalu Solver Number o Shift n (4 node elements)	eration of input files les egLanczos f Modes 10	
Eigenvalues Solver egSubspa Number of Manager			

egMultifrontLanczos

egSubspace

Shift

r Manual Vol. VI

11 April 2011

DET NORSKE VERITAS SOFTWARE

2.5.1.2.3 Wave Load Activity

Purpose: To execute and control input data for a wave load activity with or without a subsequent linear structural analysis automatically accounting for the wave, wind or current loads. For more details, see e.g. Section 3.2.3 of User Manual Volume 2.

It is required to define a wave load condition that can be referred to when defining the wave load activity (Insert/Environment/Deterministic Time Condition).

Provided you tick off for "Journal activity executions" th command is scripted. Typically:

Analysis1 = Analysis(true);

Analysis1.add(MeshActivity());

Analysis1.add(WaveLoadActivity (Condition1));

Analysis1.add(LinearAnalysis());

Analysis1.add(LoadResultsActivity());

Analysis1.execute();

In the example above the wave load condition Condition1 has been chosen.

To run the analysis it is required to define execution directives for Wajac. This is done by editing the activity Wave Load Analysis.

	Available activities Meshing Vave Load Activity Linear Structural Analysis Pile Soil Analysis Load Results	• •	Static Eigenvalue
his	Wave Load Condition	×	
	ОК		Cancel

Analysis2

Name:

Ĵå	Activity Monitor			
3		Γ	Start	Cancel
		L		
	Journal activity executions			
	Activity	Duration	Status	Generate Input
	Activity	Duration Os	Status Not Started	Generate Input
	Activity 9 🍨 1 - Analysis1 - Analysis 9 🕅 1.1 - Meshing (Always Regen	Duration Os Os	Status Not Started Not Started	Generate Input
	Activity 9 1 - Analysis1 - Analysis 9 1.1 - Meshing (Always Regen 1.1.1 - Delete loads	Duration Os Os Os	Status Not Started Not Started Not Started	Generate Input
	Activity	Duration Os Os Os Os	Status Not Started Not Started Not Started Not Started	Generate Input
	Activity 1 - Analysis1 - Analysis 1 - Meshing (Always Regen 1.1.1 - Delete loads 1.1.2 - Generate loads 1.1.3 - Delete mesh	Duration Os Os Os Os Os	Status Not Started Not Started Not Started Not Started Not Started	Generate Input
	Activity 1 - Analysis1 - Analysis 1.1 - Meshing (Always Regen 1.1.1 - Delete loads 1.1.2 - Generate loads 1.1.3 - Delete mesh 1.1.4 - Generate mesh	Duration Os Os Os Os Os Os	Status Not Started Not Started Not Started Not Started Not Started Not Started	Generate Input
	Activity 1 - Analysis1 - Analysis 1.1 - Meshing (Always Regen 1.1.1 - Delete loads 1.1.2 - Generate loads 1.1.3 - Delete mesh 1.1.4 - Generate mesh 1.2 - Wave Load Analysis, C	Duration Os Os Os Os Os Os Os	Status Not Started Not Started Not Started Not Started Not Started Not Started	Generate Input

0s

Not Started

🗹 🥂 1.4 - Load Results

🕯 Activity Monitor			X
۲		Start	Cancel
🔲 Journal activity execution	ns		
Activity	Duration	Status	Generate Input
🗹 💁 1 - Analysis1 - Analysis	; Os	Not Started	
🗹 🕅 1.1 - Meshing (Alway	s Regen Os	Not Started	
1.1.1 - Delete load	s Os	Not Started	
1.1.2 - Generate lo	oads Os	Not Started	
1.1.3 - Delete mesi	h Os	Not Started	
1.1.4 - Generate m	iesh Os	Not Started	
🗹 💁 1.2 - Wave Load Ana	alveie C Ne	Not Started	Yes
☑ 📲 1.3 - Linear Structur	🚉 Nit activity	lot Started	Yes
R 1.4 - Load Results	wajac.inp	lot Started	
	wajac.lis		

🎎 Create Wave Load + Linear Static Ana...

Automatically import global loadcases

11 April 2011

GeniE

92

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	93	11 April 2011

2.5.1.2.3.1 Load calculation

A wave load calculation performed in GeniE by use of Wajac must be a deterministic time domain analysis. The type of load calculation is determined by the hydro condition dataset selected.

The following input parameters should be considered depending on your task (the dialogue below is filled out with data from a typical wave load condition *Condition1*):

ĴÅ Ed	👪 Edit Wave Load Run. 🛛 🕅																
۲	Bur: Analyzis4.step[2] Vave load condition: Condition1																
۰L	C Load calculation C Added mass and damping only C Data check only V Automatic generation of input files 9?																
Deter	ministic sea	istates	Added	mass an	d dampin	ig Rules	Buoyar	ncy Morison (Jutput	Special options							
⊙ F Th	Phase step C Time step C Single step □ Prepare for gust wind induced fatigue P? The condition was inconsistent! You must review and press DK to confirm.																
Se	astates tabl	le para	meters														
	Design lo	ad calo	culation		SI	ten length:		10 deg		g? [deg]							
	🗌 Maxim	num ba	ise shear					00									
	🔲 Maxim	num ov	erturning	moment	IN	umper or step	08:	36									
	_				Ci	urrent blocka	ge fact	or: 1									
	Whee	eler stre	etching	85	W	/ave kinemat	ics fact	or: 1									
	🔽 Buoya	ancy ca	alculation					1									
			Fill	all		Fill sele	cted										
Spec	ify value: 🤤	?															
	Seastate	Period	Direction	Height	Phase	Wave mod.	Order	Current	Wind	Stretching	Step length [deg]	Num.steps	Buoyancy	Design load	Current b.fac.	Wave k.fac.	1.LC num.
1	1	14 s	270 deg	26 m	-60 deg	Stokes5		CurrentProfile1									
2	2	12 s	180 deg	15 m	-60 deg	Stokes5		CurrentProfile1									
3	3	14 S	90 deg	28 m	-60 deg	Stokes5		CurrentProfile1									
5	4	15 \$	0 deg	27 m	-ou deg	CalmSea		CurrentProfile1					Off				
6	-					oamood											
7																	
																ОК	Cancel Apply

Deterministic seastates:

Phase step, *Time step* or *Single step* can be selected. Please notice that if you want to create analysis results for use in a gust wind analysis in Framework only the option *Single step* can be used.

In case you check for *Maximum base shear* and Maximum overturning moment there are only two load cases part of the load transfer file to the structural analysis.

Appropriate *Current blockage factor* may be specified by the user. The total current profile will be multiplied by this factor after calculation of apparent wave period and before combination with wave kinematics.

Wave kinematics factor: Appropriate spreading factor may be specified by the user. Horizontal velocities and accelerations from the selected wave theory and apparent wave period will be multiplied by this factor.

When ticking off for the *Wheeler stretching*, both waves and current is stretched for linear wave theory (Airy waves). For non-linear wave theory only the current is stretched.

You may also decide not to include Buoyancy calculation in the hydrodynamic load calculation

When you use the *Fill All* button the relevant data is filled in to the complete table (the white cells as shown above). If you want to fill only parts of the table, you should select the relevant rows (one row at a time or continuous rows) at the bottom and click *Fill Selected*.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	94	11 April 2011

This command is scripted. By using the default options as shown above, typically:

Analysis1.step(2).deterministicSeastates().populate();

Analysis1.step(2).deterministicSeastates().seastate(1).dataPhase(NoStretching,10 deg,36,true,NoDesignLoads,1,1);

Analysis1.step(2).deterministicSeastates().seastate(2).dataPhase(NoStretching,10 deg,36,true,NoDesignLoads,1,1);

Analysis1.step(2).deterministicSeastates().seastate(3).dataPhase(NoStretching,10 deg,36,true,NoDesignLoads,1,1);

Analysis1.step(2).deterministicSeastates().seastate(4).dataPhase(NoStretching,10 deg,36,true,NoDesignLoads,1,1);

Analysis1.step(2).deterministicSeastates().seastate(5).dataPhase(NoStretching,0,1,true, NoDesignLoads,1,1);

Added mass and damping only:

Added mass and damping are calculated and written to the Loads Interface File independently of the choice of load calculation type. The default value is not to calculate added mass and damping for transfer to subsequent dynamic structural analysis.

Include internal water option involves that added mass is written to the Loads Interface File including internal water in flooded members.

The Use $C_m=2.0$ option involves that $C_m = 2.0$ (added mass coefficient = 1.0) is used for all elements irrespective of the Cm values used in the force calculation.

Calculate damping option means that damping is written to the Loads Interface File.

 Idit Wave Load Run

 Image: Second Se

This command is scripted. By selecting the above options above, typically: Analysis1.step(2).addedMassAndDamping().calculateAddedMass(true); Analysis1.step(2).addedMassAndDamping().calculateDamping(true); Analysis1.step(2).addedMassAndDamping().includeInternalWater(true); Analysis1.step(2).addedMassAndDamping().useCm(true);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	95	11 April 2011

Rules:

It is possible to compute the wave loads in accordance with the API rules. The default is user definition of the wave load computations (i.e. no API rules).

The *Include Doppler effect* involves that the Doppler effect may be accounted for by using apparent wave (API 2.3.1b1).

Include effect of wake encounter involves that the effect of wake encounter may be included. There are two options:

Vertical: The dependency of wake encounter will be applied to all members within 15 degrees of the vertical (API 2.3.1b7) as specified by API.

Orbital plane: The dependency of wake encounter will be applied to all members within 15 degrees of the orbital plane of the wave kinematics.

🎎 Edit Wave Load Run	
Pun: Analysis4.step(2)	Wave load condition: Condition1
 Load calculation C Added mass and data 	Jamping only 🥅 Data check only 🔽 Automatic generation of input files 💡
Deterministic seastates Added mass and da	Jamping Rules Buoyancy Morison Output Special options
🔲 Include Doppler effect	83
Include effect of wake encounter	85
Vertical	
C Orbital plane	
Maximum angle: 0 deg	[deg]
Adjust for current 💡	

- If a maximum angle is given this value will substitute 15 degrees.

- The effect of wake encounter may be *adjusted for current* (API C2.3.1b7).

This command is scripted. By selecting the above options above (and maximum angle 25 degrees), typically: Analysis1.step(2).rules().setRuleType(wrAPI); Analysis1.step(2).rules().includeDoppler(true); Analysis1.step(2).rules().applyWakeType(wrToVertical); Analysis1.step(2).rules().adjustForCurrent(true); Analysis1.step(2).rules().maxAngle = 25 deg;

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	96	11 April 2011

Buoyancy:

It is possible to override the default options for how to calculate the buoyancy – the default options are shown below.

Assuming horizontal free surface:

- Buoyancy is calculated assuming a horizontal surface above each member.

- The largest of the surface elevations above the member ends is used at both ends.

- This option is default and is only available in time domain.

Using actual free surface:

- Buoyancy forces are calculated using the actual, i.e. a non-horizontal water-plane that gives small buoyancy components in the horizontal plane.

ی	Run: And	alysis4.step(2)	•	Wave load cond	ition: Condition1	•	
Load ca	lculation (C	Added mass an	d damping onl	y 🥅 Data check	only 🔽 Automa	tic generation of inp	ut files 💡
Deterministic	seastates	Added mass ar	nd damping F	lules Buoyancy	Morison Outp	out Special option:	3
- Buouanci	u forces with	non-horizontal i	water-plane				
	C A		, all plane				
A.5	 Assumir 	ng horizontal free	e sufface				
83	○ Using a	ctual free surfac	e				

By default the *Include Buoyancy due to steel area* option is on meaning that buoyancy of the steel is included.

This command is scripted. By selecting the above options above, typically:

Analysis1.step(2).buoyancy().horisontalFreeSurface(true); Analysis1.step(2).buoyancy().actualFreeSurface(false); Analysis1.step(2).buoyancy().steelAreaBuoyancy(true);

Morison:

You may define if velocity decomposition shall be used when calculating the normal components of the drag forces, i.e. how to handle the v^2 term in the Morison equation.

Normal to elements involves that the absolute value of the normal velocity is multiplied with the velocity component normal to the member in the Morison equation (i.e. abs(vn) is used). This is the default option.

Total involves that the absolute value of the normal velocity is replaced by the absolute value of the total velocity in the Morison equation (i.e. abs(v) is used.)



This command is scripted. By selecting the above options above, typically: Analysis1.step(2).morison().normalToElement(true);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	97	11 April 2011

Output:

Under the Output tab you can specify the overturning moment reference point, how much to add to the printed file from Wajac and details for the loads interface file.

The *Moment reference point* about which the overturning moments are calculated are by default at the mud line directly below or above the origin of the global coordinate system. An alternative point may be specified by you.

👪 Edit Wave Load Run
Run: Analysis4.step(2) Vave load condition: Condition1
ⓒ Load calculation ○ Added mass and damping only □ Data check only ▼ Automatic generation of input files ♀?
Deterministic seastates Added mass and damping Hules Buoyancy Morison Output Special options
Global results Print file Loads interface file
Moment reference point:
Point(0 m,0 m,0 m)

This command is scripted. By specifying the z-value to 50 meters, typically: Analysis1.step(2).output().globalResults().momentRefPoint = Point(0 m, 0 m, 50 m);

The *Print file* option allows you to add additional details to the Wajac listing file from the hydrodynamic analysis.

Under *Structure Set* you can select parts of structure that should have force intensities printed. The selected sets will be included in the *Add operations* for *Deterministic* and *Fluid kinematics*.

18 mm									
44 Edit Wave Load Run									
Bun: Analysis4.step(2) Vave load condition: Condition1									
C Load calculation C Added mass and damping only									
Deterministic seastates Added mass and damping Rules Buoyancy Morison Dutput Special options									
Global results Print file Loads interface file									
Element loads									
Structure set: 📃 👻 🛠 😵									
Deterministic									
Seastate: V Step: V Direction: V Period: V									
Add 💡									
2									
- Fluid kinematics									
Point 2									
Add Ø?									
Points									

This command is scripted. By selecting the named set "Jacket", sea state number "2" and step "10", typically:

Analysis1.step(2).output().printFile().addDeterministicData(Jacket,2,10);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	98	11 April 2011

The *Loads interface file* option controls type of file format and a manual control of the first load case number.

The default option is to use an unformatted file and that the first wave load case number is a continuation of the basic load case numbers.

👪 Edit Wave Load Run
Run: Analysis4.step(2) Wave load condition: Condition1
⑦ Load calculation ○ Added mass and damping only □ Data check only ☑ Automatic generation of input files ♀?
Deterministic seastates Added mass and damping Rules Buoyancy Morison Output Special options
Global results Print file Loads interface file
Generate loads interface file C Formatted files (ASCII) Unformatted files (binary) See Specify first loadcase number:

This command is scripted. By selecting unformatted files and start of first wave load case number to "45", typically:

Analysis1.step(2).output().loadsInterfaceFile().formatted(false);

Analysis1.step(2).output().loadsInterfaceFile().loadCaseNumbering(true);

Analysis1.step(2).output().loadsInterfaceFile().firstLoadCaseNumber = 45;

Special options:

Under the Special options tab you can eliminate structural parts from the wave load analysis. You can also add additional still water levels.

An *Eliminated structure* set may be selected from named sets and this part of the structure is eliminated from hydrodynamic load calculations.

Exclude eccentricities in calculations will disregard any eccentricities of the beams. If you choose this option, the beam length will in most cases be increased (node to node). As such this is not the default option.

The Additional still water level you can add additional still water levels. The water levels are relative to the still water level specified under the environment data.

👪 Edit Wave Load Run
Bun: Analysis4.step(2) Vave load condition: Condition1
Coad calculation ○ Added mass and damping only □ Data check only ▼ Automatic generation of input files Preterministic seastates Added mass and damping Bules Buguancul Morison Dubrut. Special options
Use MacCamy-Fuchs' formula
Eliminated structure:

This command is scripted. By excluding eccentricities, eliminating the named set "Topside" and adding two new still water levels at +2m and -3m relative to the base still water level, typically:

Analysis1.step(2).specialOptions().excludeEccentricities(true);

Analysis1.step(2).specialOptions().useEliminatedStructure(true);

Analysis1.step(2).specialOptions().eliminatedStructure = Topside;

Analysis1.step(2).specialOptions().removeAllLevels();

Analysis1.step(2).specialOptions().addLevel(2m);

Analysis1.step(2).specialOptions().addLevel(-3m);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	99	11 April 2011

2.5.1.2.3.2 Added mass and damping only

This option implies that no load cases will be calculated and written to the loads interface file – only added mass and damping are considered.

There are two tabs that can be accessed from this option; the *Added mass and damping* and the *Special options*.

Notice that at least one of the *Calculate added mass* and *Calculate damping* must be selected.

For a definition of input parameters, please see previous section on *Load calculations*.

👪 Edit Wave Load Run
Run: Analysis4.step(2) Vave load condition: Condition1
← Load calculation 🕫 Added mass and damping only 🧮 Data check only 🔽 Automatic generation of input files 💡
Deterministic seastates Added mass and damping Rules Buoyancy Morison Output Special options
✓ Calculate added mass
☑ Include internal water 😵
☑ Use C_m=2.0
✓ Calculate damping
Henry 1 10
J& Edit Wave Load Run
Run: Analysis4.step(2) Vave load condition: Condition1
🔿 Load calculation 🏹 Added mass and damping only 🥅 Data check only 🔽 Automatic generation of input files 💡
Deterministic seastates Added mass and damping Rules Buoyancy Morison Output Special options
🔲 Use MacCamy:Fuchs' formula 🛛 😵
In phase
O Phase lag
Con structure:
Eliminated structure: Topside
Exclude eccentricities in calculations
Additional still water level: 💡?
Still water level [m]
1 2
3 4
5

This command is scripted. By selecting all the options under "Added mass and damping", typically:

Analysis1.step(2).addedMassAndDampingOnly(true);

Analysis1.step(2).addedMassAndDamping().calculateDamping(true);

Analysis1.step(2).addedMassAndDamping().useCm(true);

Analysis1.step(2).addedMassAndDamping().includeInternalWater(true);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	100	11 April 2011

2.5.1.2.3.3 Data check only

The data check option prevents Wajac from a full execution of the analysis; a data check will be performed of the input data to determine whether they are consistent.

The result from a data check can be inspected on the Wajac listing file.

👪 Edit Wave Load Ru	ın			
📚 🛐 Run: Anal	vsis4.step(2) 💌	Wave load condition:	Condition1	
	dded mass and damping o	nly 🔽 Data check only	Automatic generation of input files (<mark>8</mark> 3
Deterministic seastates 🛛	Added mass and damping	Rules Buoyancy M	orison Output Special options	

This command is scripted. By activating the option "Data check only", typically:

Analysis1.step(2).dataCheckOnly(true);

2.5.1.2.3.4 <u>Automatic generation of input files</u>

The default option ("True") implies that the input file for Wajac is automatically generated. In other words, if you modify one of the parameters this file is automatically updated prior to the next analysis run. If you want to have a manual control of the input file you should deselect this option.

In this case a *Generate input files* button appears. When you click on this button you create an input file based on the parameters defined. This file may be edited and changed. See also the previous section for linear static analysis on how to control and edit the input file.

When deactivating the automatic input generation, it should be noted that changes in the environment, structure or properties should not be performed, as then the correspondence between the GeniE concept model and the Wajac input file may become out of sync and invalid.

۵	B	un: Ar	nalysis5.ste	ep(2)	•	Wave I	oad co	ndition: Conditi	ion1		•		
⊙ L Dete	_oad calcula	ation (C astates	Added m	nass and nass and	d dampin d dampin	g only 🔲 D g Rules	ata che Buoyar	eck only 🥅 Aut	omatic ; Output	generation of Special opti	input files	§ 8 ?	
€ Tł	Phase step ne condition	C 1 was in	' Time step consistent	⊂ S !You m	ingle stej iust revie	o 🥅 Prepar w and press	e for gu OK to c	ust wind induced	l fatigue	<u>8</u> ?			
Se	eastates tab Design lo	ile para ad calc num ba	meters culation se shear		St	ep length: umber of step	05:	10 deg 36		 [deg]			
	🔲 Maxin	num ov	erturning r	noment						-			
	✓ Maxin✓ Whee✓ Buoya	num ov eler stre ancy ca	erturning r tching alculation	noment	c. 	urrent blocka 'ave kinemat	ge facti ics fact	or: 1 or: 1					
Spec	I Maxin Whee I Buoya	num ov eler stre ancy ca	erturning r tching alculation Fill a	noment &?		urrent blocka 'ave kinemat Fill seler	ge fact ics fact cted	or: 1		-			
Spec	I Maxin I Whee I Buoya cify value: €	num ov eler stre ancy ca ?? Period	erturning r tching alculation Fill a	noment &? all Height	Cu W Phase	urrent blocka 'ave kinemat Fill selev Wave mod.	ge fact ics fact cted Order	or: 1 or: 1 	Wind	Stretcl	hing	Step length	Idea
Spec	Maxin Whee Whee Whee Grade Buoya Cify value: Seastate 1	num ov eler stre ancy ca ?? Period 14 s	erturning r tching alculation Fill a Direction 270 deg	Height 26 m	Phase -60 deg	irrent blocka 'ave kinemat Fill seler Wave mod. Stokes5	ge fact ics fact cted Order	or: 1 or: 1 Current Current CurrentPro file1	Wind	Stretc	hing	Step length	[deg
Spec	Maxin Maxin Monore Mon	num ov eler stre ancy ca Period 14 s 12 s	erturning r tching alculation Fill a Direction 270 deg 180 deg	Height 26 m 15 m	Phase -60 deg -60 deg	irrent blocka 'ave kinemat Fill seler Wave mod. Stokes5 Stokes5	ge fact ics fact cted Order	or: 1 or: 1 Current CurrentProfile1 CurrentProfile3	Wind	Stretcl	hing	Step length	[deg
Spec	✓ Maxin ✓ Whee ✓ Buoya ✓ Seastate 1 2 3	eler stre ancy ca Period 14 s 12 s 14 s	erturning r tching alculation Fill a Direction 270 deg 180 deg 90 deg	Height 26 m 15 m 28 m	Phase -60 deg -60 deg	verent blocka ave kinemat Fill selev Wave mod. Stokes5 Stokes5 Stokes5	ge fact ics fact cted Order	or: 1 or: 1 Current CurrentProfile1 CurrentProfile1 CurrentProfile1	Wind	Stretc	hing	Step length	[deg
Spec	☐ Maxin ☐ Whee ☑ Buoya Cify value: Seastate 1 2 3 4	Period 14 s 12 s 14 s 15 s	erturning r tching alculation Fill a Direction 270 deg 180 deg 90 deg 0 deg	Height 26 m 15 m 27 m	Phase -60 deg -60 deg -60 deg	irrent blocka ave kinemat Fill seler Wave mod. Stokes5 Stokes5 Stokes5 Stokes5	ge fact ics fact cted Order	Current CurrentProfile1 CurrentProfile1 CurrentProfile1 CurrentProfile1	Wind	Stretc	hing	Step length	[deg
Spec	☐ Maxin ☐ Whee ☐ Buoya ☐ Seastate 1 2 3 4 5	Period 14 s 12 s 15 s	erturning r tching alculation Fill a Direction 270 deg 180 deg 90 deg 0 deg	Height 26 m 15 m 27 m	Phase -60 deg -60 deg -60 deg	rrent blocka ave kinemat Fill seler Wave mod. StokesS StokesS StokesS StokesS StokesS StokesS StokesS	ge fact	Current CurrentProfile1 CurrentProfile1 CurrentProfile1	Wind	Stretc	hing	Step length	[deg
Spec	✓ Maxin ✓ Maxin ✓ Where ✓ Buoya Cify value: § Seastate 1 2 3 4 5 5	Period 14 s 12 s 14 s 15 s	erturning r tching alculation Fill a Direction 270 deg 90 deg 0 deg	Height 26 m 15 m 27 m	Phase -60 deg -60 deg	rrent blocka ave kinemat Fill seler Wave mod. Stokes5 Stokes5 Stokes5 Stokes5 CalmSea	ge fact	or: 1 Current CurrentPro file1 CurrentPro file1 CurrentPro file1	Wind	Stretc	hing	Step length	[deg
Spec	Maxin Whee Whee Whee Seastate 1 2 3 4 5	Period 14 s 15 s	erturning r tching alculation Fill a Direction 270 deg 180 deg 90 deg 0 deg	Height 26 m 15 m 27 m	Phase -60 deg -60 deg -60 deg	rrent blocka ave kinemat Fill seler Wave mod. Stokes5 Stokes5 Stokes5 Stokes5 Stokes5 CalmSea	ge fact	Current CurrentProfile1 CurrentProfile1 CurrentProfile1	Wind	Stretc	hing	Step length	[deg

This command is scripted. By activating the option "Automatic generation of input files", typically: Analysis4.step(2).generateInput = false; DET NORSKE VERITAS SOFTWARE

User Manual Vol. VI

11 April 2011

Version 5.3

2.5.1.2.4 Pile Soil Analysis

Purpose: To execute and control input data for a pile soil analysis. For more details, see e.g. Section 3.6 of User Manual Volume 2.

It is required to define a wave load condition that can be referred to when defining the wave load activity (I - i) = (I - i)

(Insert/Environment/Deterministic Time Condition).

Provided you tick off for "Journal activity executions" this command is scripted. Typically:

Analysis1 = Analysis(true);

Analysis1.add(MeshActivity());

Analysis1.add(WaveLoadActivity (Condition1));

Analysis2.add(PileSoilAnalysis (Condition1));

Analysis1.add(LinearAnalysis());

Analysis1.add(LoadResultsActivity());

Analysis1.execute();

In the example above the wave load condition Condition1 has been chosen.

To run the analysis it is required to define execution directives for Wajac (see previous Section) and Splice (including Gensod). This is done by editing the activity *Pile Soil Analysis*.

å Create Wave Load + Pile Soil	Analysis 🛛 🔀
Name: Analysis2	_
Automatically import global loade	ases
Available activities Meshing Wave Load Activity Linear Structural Analysis Pite Soil Analysis Load Results	
Wave Load Condition	
OK	Cancel

۶		Start	Cancel
Dournal activity executions	Duration	Status	Generate Inpul
Methoday	Duration	Not Started	Generate input
Participation (Always Regen)	05	Not Started	
1.1.1 - Delete loads	05	Not Started	
✓ 1.1.2 - Generate loads	0s	Not Started	
✓ 1.1.3 - Delete mesh	0s	Not Started	
✓ 1.1.4 - Generate mesh	0s	Not Started	
M. Sa 1.2 - Wave Load Analysis, C	0s	Not Started	Yes
Soil Analysis, Condit	0s	Not Started	Yes
1.3.1 - Soil (Gensod)	0s	Not Started	
1.3.2 - Sestra, Reduction	0s	Not Started	
✓ 1.3.3 - Splice	0s	Not Started	
1.3.4 - Sestra, Retracking	0s	Not Started	
D 1.4 Lond Decults	0e	Not Started	

🎎 Activity Monitor			
Sournal activity executions	[Start	Cancel
Activity	Duration	Status	Generate Input
 I - Analysis1 - Analysis I - Analysis1 - Analysis I - Meshing (Always Regen I.1 Delete loads I.1.2 - Generate loads I.1.3 - Delete mesh I.1.4 - Generate mesh I.2 - Wave Load Analysis, C I.3 - Pile Sol Analysis, Condit 	Os Os Os Os Os Os Os	Not Started Not Started Not Started Not Started Not Started Not Started	Yes
✓ 1.3.1 - Soil (Gensod) ✓ 1.3.2 - Sestra, Reduction ✓ 1.3.3 - Splice ✓ 1.3.4 - Sestra, Retracking ✓ 1.3.4 - Sestra, Retracking	0s 24 0s 0s 0s 0s	Edit activity Not Started Not Started Not Started Not Started	

2.5.1.2.4.1 Nonlinear calculation

To run a pile soil analysis in GeniE by use of Splice (including Gensod) requires that special geotechnical data is given for the soil material, soil curves, group effects and how to handle loads at the soil surface. In addition number of iterations and convergence criteria must be specified for the non-linear pile soil analysis performed in Splice.

In addition there are other properties for the piles and the soil that need to be defined, see pile characteristics as defined in e.g. Sections 2.2.6.1 and 2.4.1.4 as well as soil definitions from the browser menu *Environment*.

C Linear calculation	 Nonlinear calculation Data check only Automatic generation of input files
Soil Splice	
material coerrs Curve gene	ration Group effects Loads at surface
Material coefficients	
on Tan (phi):	<u>8</u> ? 1
on Shear Strength	©? 1
on Skin Friction	<u>8</u> ? 1
on Pile Tin Besistance	₽? 1
	.,

Material coefficients

Material coefficients can be seen as safety factors used to modify the soil properties without changing the actual values. The actual values are divided by the coefficients meaning that a coefficient greater then one (1) reduces the capacity.

There are four coefficients related to the friction angle (Tan (phi)), un-drained shear strength, skin friction and pile tip resistance. The default values for the soil material coefficients are 1; i.e. the actual value is the same as the capacity.

This command is scripted. Typically: Analysis1.step(3).soil.materialCoeffs.tanPhiCoeff = 1.1; Analysis1.step(3).soil.materialCoeffs.shearStrengthCoeff = 1.2; Analysis1.step(3).soil.materialCoeffs.skinFrictionCoeff = 1.3; Analysis1.step(3).soil.materialCoeffs.pileTipResistanceCoeff = 1.4;

Curve generation

Lowest level of cyclic PY generation: - Cyclic PY data will be generated for all layers below the specified limit.

Lowest undrained shear stiffness with stiff clay procedures:

- The API code distinguishes between "soft clay" and "stiff clay". Procedures for calculating PY data are, however, only given for "soft" clay. Clay layers will be considered "stiff" when the undrained strength value given is higher than the limit specified. In such case the Reese et al (1975) are used instead of the API "soft" clay procedures.

Zone of influence ratio to pile radius: - Outside the zone of influence the shear displacements due to pile axial loading are neglected. Typical values may be 10 to 30.

Curve shape factor TZ:

- Dimensionless factor that governs the shape of the TZ curve up to peak resistance. Zero gives a linear curve. Typical values may be 0.8 to 0.99.

👪 Edit Pile Soil Analysis		
📚 🖹 Run: Analysis1.st	ep(3) 💌 Wave load	condition: Condition1
C Linear calculation C I	Nonlinear calculation 🖵 Data c 🔽 Autom	wheck only 📚 atic generation of input files
Material coeffs Curve generation	m Group effects Loads at surfa	ace
Curve Generation Lowest level with cyclic PY curve generation	% ? 0 m	[m]
Lowest undrained shear stiff with siff clay procedures		[KPa]
Zone of influence ratio to pile radius	Q ? 10	
Curve shape factor TZ	8 ? 0.9	
L		
	OK	Cancel Apply

The two latter values are needed for the Kraft et al (1981) procedures to generate tz-data based on tz-code = 200.

This command is scripted. Typically: *Analysis1.step(3).soil.curveGeneration.lowestLevelWithCyclicPY = 1 m;* Analysis1.step(3).soil.curveGeneration.lowestShearStiff = 120 KPa; *Analysis1.step(3).soil.curveGeneration.zoneOfInfluenceTZ = 20; Analysis1.step(3).soil.curveGeneration.curveShapeFactorTZ* = 0.85;

Group effects

Group interaction effects (pile-soil-pile interaction) are calculated from the elastic halfspace theory assuming an E-modulus that increases linearly with depth.

This command is scripted. Typically:

Analysis1.step(3).soil.groupEffects. averagePoissonRatio = 0.6;

Analysis1.step(3).soil.groupEffects. modulusOfElasticity(10000 KPa,0 m, 20000 KPa,-150 m);

👪 Edit Pile Soil Analysis 🛛 🔀
Run: Analysis1.step(3) Vave load condition: Condition V
C Linear calculation
Soil Splice Material coeffs Durve generation Group effects Loads at surface
Modulus Of Elasticity
10000 KPa [KPa] 0 m [m]
10000 KPa [KPa] -100 m [m]
Average Poisson Ratio 🔗 0.5
OK Cancel Apply

11 April 2011

104

Version 5.3

Loads at surface

Different types of vertical loading may exist at the soil surface in the vicinity of the pile. Such loading will influence the vertical stress down along the pile and hence the lateral resistance.

📚 👔 Run: Analysis1	.step(3)	•	Wave	load c	ondition:	Condition	1
C Linear calculation (Soil Splice	Nonlin	hear calculation	D A	ata ch utomat	eck only ic genera	tion of inpu	t files
Material coeffs Curve generation	ation G	roup effects	Loads at	t surfac	e		
Modulus Of Elasticity	83	Z relative to	seabed				
10000 KPa	[KPa]	0 m		[m]			
10000 KPa	[KPa]	-100 m		[m]			
Average Poisson Ratio	83	0.5					

Splice (Gensod) approximately accounts for:

- A uniform Vertical stress at soil surface
- An embankment type loading
- A circular loaded area with the pile at its centre

The positions of the different types of surface loads are therefore given relative to a hypothetical average pile. Vertical loads are assumed to act at a Z level corresponding to the general scoured soil surface. The required inputs for each of the 3 options are:

- Vertical stress at soil surface:
 - Uniform vertical stress acting over a large area
- Embankment loads are defined by:
 - Vertical stress under the flat part of the embankment loading
 - Width of the sloping part of the embankment loading
 - Horizontal distance from the pile axis to the toe of the embankment. Positive if the pile is located outside the embankment, negative if the pile is inside.
- Circular loaded area is defined by:
 - Vertical stress under the circular loaded area
 - Radius of the circular loaded area

This command is scripted. Typically:

Analysis1.step(3).soil.loadsAtSurface.verticalStressAtSurface = 200 KPa;

Analysis1.step(3).soil.loadsAtSurface.verticalStressUnderEmbankment = 100 KPa;

Analysis1.step(3).soil.loadsAtSurface.widthOfEmbankmentSlopingPart = 1 m;

Analysis1.step(3).soil.loadsAtSurface.distancePileToEmbankmentToe = 0.5 m;

Analysis1.step(3).soil.loadsAtSurface.verticalStressUnderCircularLoadedArea = 50 KPa;
11 April 2011

Splice

The solver settings that are used to control the Splice analysis are:

- Max iterations. The iterations are stopped when the convergence criterion is met or when the maximum number of iterations is reached. Maximum value is 50.
- Convergence criterion. The convergence criterion is met when the last iteration change in displacements along the pile is less than the specified length value.

👪 Edit Pile Soil Analysis 🛛 🔀
Run: Analysis1.step(3) Vave load condition: Condition1
C Linear calculation
Soil Splice
Solver
Solver settings
Max Iterations 82 20
Convergence Criterion Q ? 0.001 m [m]
,
OK Cancel Apply

This command is scripted. Typically:

Analysis1.step(3).splice.solver.maxIterations = 25; Analysis1.step(3).splice.solver.convergenceCriterion = 0.0015 m;

2.5.1.2.4.2 Linear calculation

It is possible to neglect the pile and soil analysis by inserting fixed boundary conditions along the piles and disregard the pile-soil analysis. This is the same as executing a wave and structural analysis where all piles are fixed at the finite element positions.

This command is scripted. Typically:

Analysis1 = Analysis(true);

Analysis1.add(MeshActivity());

Analysis1.add(WaveLoadActivity (Condition1));

Analysis1.add(PileSoilAnalysis (Condition1));

Analysis1.add(LoadResultsActivity());

Analysis1.step(3).nonlinearAnalysis(false);

ta Edit Pile Soil Analysis	×
Bun: Analysis1.step(3) Vave load condition: Condition1	4
C Nonlinear calculation C Nonlinear calculation C Automatic generation of input files Solid Solid	
Material coeffs Curve generation Group effects Loads at surface	
Material coefficients	
on Tan (phi): 0?	
on Shear Strength 8?	
on Skin Friction 8?	
on Pile Tip Resistance 8?	
OK Cancel Apply	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual V	ol. VI
Version 5.3	106	11 April	1 2011
It is required to specify the pile boundary "Fixed" from the mesh activity.	y conditions to	Wesh activity	×
		Meshing Hules	
This command is scripted. Typically:		Smart load combinations	
Analysis1.step(1).pileBoundaryCondition	n = pmFixed;	Override Global Superelement Data Top Superelement Type	-

2.5.1.2.4.3 Data check only

The data check only option will run the Gensod activity, but only check the input data for the Splice analysis.

This command is scripted. Typically: Analysis1.step(3).dataCheckOnly(true);

👪 Edit Pile Soil Analysis 🛛 🔀
Bun: Analysis1.step(3) Vave load condition: Condition1
C Linear calculation Nonlinear calculation Linear calculation C Linear calculation
Material coeffs Curve generation Group effects Loads at surface
Material coefficients
on Tan (phi): 💡 1
on Shear Strength 8? 1
on Skin Friction 😵 1
on Pile Tip Resistance 🔗 1
OK Cancel Apply

Superelement Type

🔲 Set Mesh Priority

Pile boundary condition

🔲 Mesh Subset

ΟK

1

Fixed Pile Soil Interaction Ŧ

•

2

Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	107	11 April 2011

2.5.1.2.4.4 Automatic generation of input files

The default option ("True") implies that the input files for Gensod and Splice are automatically generated. In other words, if you modify one of the parameters this file is automatically updated prior to the next analysis run. If you want to have a manual control of the input file you should deselect this option.

In this case a *Generate input files* button appears. When you click on this button you create an input file based on the parameters defined. This file may be edited and changed. See also the previous section for linear static analysis on how to control and edit the input file.

When deactivating the automatic input generation, it should be noted that changes in the environment, structure or properties should not be performed, as then the correspondence between the GeniE concept model and the Gensod and Splice input files may become out of sync and invalid.

👪 Edit Pile Soil Analysis 🛛 🔀
Run: Analysis1.step(3) Vave load condition: Condition1
C Linear calculation Nonlinear calculation Linear calculation O Linear calculation
Suil Splice Material Coeffs Curve generation Group effects Loads at surface
Material coefficients
on Tan (phi): 8? 1
on Shear Strength 8? 1
on Skin Friction 🔗? 1
on Pile Tip Resistance 🛛 🖓 1
Generate input files

This command is scripted. By activating the option "Automatic generation of input files", typically: Analysis1.step(3).generateInput = false; 108

Version 5.3

2.5.1.3 Export FEM

Purpose: To export an existing finite element model (FEM) to a designated area with a user given name.

This command is not scripted. You can include the export capability by editing the journal file to typically include

FemExporter = ExportMeshFem();

FemExporter.DoExport("C:/Location/T1.FEM");

2.5.1.4 Import External Results SIN file

Purpose: To import a results file on SIN format. There are strict limitations on this functionality. See User Manual Volume 4 for documentation.

This command is scripted and a typical example is given below.

SinImporter = ImportResultsSin();

SinImporter.DoImport("C:/Location/T1.SIN");

	//
file	
Import Exte	rnal Results SIN file 🛛 ? 🗙
Look in: 障	Module 💌 🗢 🖻 📸 🖬 🗸
Analysis1	
11.5IN	
File name:	
Files of twee	
Files of type:	SIN files (".sin)
	Open as read-only

Export from 20080916_114729_T1.FEM to new FEM File

Save in: 🛅 Workspaces

T1.FEM

Save as type: SESAM Interface File (*.FEM)

🚞 a 🚞 abc

Шb

🚞 um

File name:

Frame Code Check

2.5.1.5 Frame Code Check

Purpose: Starting the code check program Framework and data transfer of FEM and concept data. See Appendix C of User Manual Volume 4 for more details.

This command is not scripted.

Perform frame code checks using SESAM Framework based on the current FEA analysis results: Analysis1\20080725_105001_R1.sin Options when creating new Framework database							
Use automatic FEM-based member generation in Framework							
Import beam concepts from re-	sults file						
Framework may have to split beams into members. Use the options below to control beam-to-member interpretation. Structure criteria Split at all structural joints Split at can-reinforcements Do not split beams							
Create new Framework database Bun Framew Use existing database Cancel							

11 April 2011

-111 🗢 🗈 🗣 💽

Ŧ

?

Save

Cancel

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	109	11 April 2011

2.5.1.6 Advanced Results (Xtract)

Purpose: Starting the general post processor Xtract.



This command is not scripted.

Locate FE 2.5.1.7

Purpose: Feature allowing you to locate finite element and node numbers based on manual input of numbers or based on criterions for corner angle, relative Jacobi determinant, edges, aspect ratio, triangle element type or warping. See Section 6.1.5 of User Manual Volume 3 for more details.

This command is not scripted.

Locate Finite Element(s) and Node(s) Elements Nodes Locate Clear Clear Close Show element number 🔽 Show node number Highlight element Highlight node • - | -Auto Detect Corner angle larger than: 150deg [deg] Corner angle smaller than: [deg] 20dea 🔲 Relative Jacobi larger than: Min Jacobi smaller than: 0.2 Shortest edge smaller than: 0.01m [m] Absolute length: 5 Maximum aspect ratio: 🔲 Is element a triangle Warping Node rel distance: 8? 0.0001

2.5.1.8 Presentation

Purpose: A graphic driven menu on presenting various types of results, setting attributes, and changing load cases. For more details, see Section 3.14 of User Manual Volume 1 or Chapter 8 of User Manual Volume 3.

This command is not scripted.



11 April 2011

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	111	11 April 2011

2.5.1.9 Beam Force/Stress Diagram

Purpose: To present and export beam force & stress data to Excel or other tools supporting the xml standard. See Chapter 4 of User Manual Volume 4 for more details.

This command is not scripted.

👪 Beam Deflection, Force and Stresses Display										
Result Selection Graphs Max/Min Tables Beam Line Tables S?										
• 9 • 9	Image: Sort selected beams/segments into continuous beam lines Select Deflection Image: Sort selected beams/segments into individual beams/segments Select Deflection									
	Automatically update beam lines grid when selection changes									
<u> </u>			Update grid from	seli	ection					
	Beam Line		Load Case/Envelope		Component		Hotspot			<u>^</u>
	Beam3	•	Comb_1	-	tauNxz	-	TAbsMax 🗾			
	Beam3	•	Comb_1	•	sigMxy	-	AbsMax 🗾			
	Beam3	•	Comb_1	-	Nxz	-	<u> ∏</u> AbsMa×			
	Beam3	•	Comb_1	-	Мху	-	AbsMax			
	Beam3	•	Comb_1	-	Dz	_	T AbsMax			
	<u> </u>	•	Comb_1	-	tauNxz	-	AbsMax 💌			E
		•	Comb_1	-	tauNxz	-	AbsMax 💌			
		•	Comb_1	-	tauNxz	-	AbsMax 💌			
	<u></u>	•	Comb_1	•	tauNxz	•	AbsMax 💌			
		•	Comb_1	•	tauNxz	•	AbsMax 💌			
		•	Comb_1	•	tauNxz	-	AbsMax 💌			
		•	Comb_1	•	tauNxz	•	AbsMax 💌			
		•	Comb_1	•	tauNxz	•	AbsMax 💌			
		•	Comb_1	•	tauNxz	-	AbsMax 💌			
		•	Comb_1	•	tauNxz	-	AbsMax 💌			
		•	Comb_1	•	tauNxz	-	AbsMax 💌			
		•	Comb_1	•	tauNxz	•	AbsMax 💌			-

2.5.1.10 Show Analysis and Results

Purpose: Feature to import existing analysis results in case you choose not to load the results when opening a workspace. The picture to the right shows the pop-up menu when you open an existing workspace. In case you answer "no" you can load the finite element mesh and the analysis results by the use of the command *Tools/Analysis/Show Analysis and Results*.



This command is not scripted.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	112	11 April 2011

2.5.2 Equipment

From the *Tools/Equipment* menu you can import weight lists.

2.5.2.1 Import Weight List

Purpose: Read in an existing weight list from an external source in XML (EXtendable Markup Language) or CSV (Comma Separated Values) format. See Section 3.10 of User Manual Volume 1 for more information.

The data must at least contain the following information; the order on the data file does not make any difference as long as the headers are the same.

- weight_item.name : a unique name of the item (mandatory)
- weight_item description : additional description (optional)
- weight_item.weight.dry : the mass of the item (mandatory)
- weight_item.position.x/y/z : position of the item in global axis system (mandatory)
- weight_item.dimension.dx/dy/dz : dimension of the item (optional, but recommended)

Note that the weight list is dimension less and the data will be imported according to the current *input unit settings*.

This command is scripted, typically

WeightList("C:/Location/Your_weight_list.xml");

Or

WeightList("C:Location/Your_weight_list.csv");

Note that the above will import data, but not place the equipment items in individual load cases, see Section 3.10 of User Manual Volume 1 for instructions on how to do this.

Import	Weight List		
Import Weig	;ht List File		? 🛛
Look in: 🔎	Description Commands	▼ ← 🛍	☆
🚞 Analysis1			
File name:			Open
Files of type:	XML Weight Lists (*.xml)	•	Cancel
	🔲 Open as read-only		

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	113	11 April 2011

2.5.3 Properties

It is possible to scale the masses to match a target mass from the context sensitive menu in the Browser Utilities Sets. This option will use mass density factors to scale the masses. I.e. no new material properties will be made.

This command is scripted, typically when scaling the mass of the set "bottom_deck" with a factor of 2.6):

MassFactor1 = MassDensityFactor(2.6);

bottom_deck.massDensityFactor = MassFactor1;

2.5.3.1 Create Scaled Materials

Purpose: Make a new material and connect to beam or plate following a mass scaling operation. See Section 3.16.5 of User Manual Volume 1 for more details.

This command is scripted, typically when making new materials affected of the mass scaling factor 2.6 (in this case the material St48 is affected – a new material St48_2_6 is made and applied to the structural parts of the set):

St48_2_6 = St48.copy(); St48_2_6.density = St48.density * 2.6; Bm1.material = St48_2_6; Bm2.material = St48_2_6; Bm3.material = St48_2_6;



Create Scaled Materials



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	114	11 April 2011

2.5.4 Structure

The *Tools/Structure* pulldown menu contains features for cleaning up the geometry, punch operations, split structural parts and a verification of the structure.

2.5.4.1 Geometry

This option gives access to features for simplify topology, heal structure and split periodic geometry.

2.5.4.1.1 Simplify Topology

Purpose: Remove un-necessary topology lines or points introduced by previous plate/plate or plate/beam or plate/feature edge intersections. See also Section 3.3.6 of User Manual Volume 3.

This command is scripted, typically:

SimplifyTopology();

2.5.4.1.2 Heal Structure

Purpose: Feature for reconnection of bad beam models. Model healing analyses each beam in the model and searches for candidates that should intersect the beam in question. To do this it is necessary to categorize the beams. The healing operation assumes that a beam which beam ends touch the interior of another beam is categorized as less important than the beam it touches. For more details, see Section 3.6.9 of User Manual Volume 1.

This command is scripted, typically based on a selection of 10 beams:

HealStructureSet = Set();

HealStructureSet.add(Bm1);

HealStructureSet.add(Bm2);

HealStructureSet.add(Bm3);

HealStructureSet.add(Bm4);

.....

HealStructureSet.add(Beam10); HealStructure(HealStructureSet); Delete(HealStructureSet); Heal Structure Split Periodic Geometry

Simplify Topology

Simplify Topology

Heal Structure

Split <u>P</u>eriodic Geometry



2.5.4.1.3 Split Periodic Geometry

Purpose: Insert an edge along the surface in the longitudinal direction of a periodic geometry like for example a tube. Tubular surfaces may be made from a sweeping operation where a 360 degree guideline is used as reference. In some cases GeniE is not able to make a finite element mesh on such surfaces. In these rare events, you can use this feature to insert an edge along the surface in the longitudinal direction. For more details, see Section 6.2.1.2 of User Manual Volume 3.

This command is scripted, typically:

SplitPeriodicGeometry();

2.5.4.2 Punch

Purpose: Create holes or cut lines in the structure using a userdefined profile. The profile can be defined from the dialogue box or from *Insert*/*Profile*. By using the "Punch" option plate and stiffener material will be removed from the model. The "Cut" option will create cut lines where the profile intersects the model. Please see Section 3.3.1.11 of User Manuals for details on how to do both operations.

This command is scripted, typically when using profile "Man-hole" at start position (1m, 1m, 1m) in negative x-direction. Furthermore the profile coordinate system is aligned with the global y-direction and the length of the cut profile is 5 meters:

Man_hole.punch(*Point*(1 m, 1 m, 1 m), *Vector3d*(-1,0,0), *Vector3d*(0,1,0),5);

2.5.4.3 Curve Punch

Purpose: Remove all structure within a volume as defined by a selection of curves that form a closed loop and a given vector (or a curve). For more details, see Section 3.3.1.10 of User Manual Volume 3.

This command is scripted, typically when using curves Curve1 -> Curve 4 (making a closed loop) and a vector of 10 meters in negative z-direction:

PunchWithCurves(*Array*(*Curve1*,*Curve2*,*Curve3*,*Curve4*), *Vector3d*(0,0,-10)); Simplify Topology Heal Structure Split Periodic Geometry



Profile Punch / Cut	×		
▼ New Profile			
Profile start position			
× © Punch			
Sweep direction Sweep length C Cut			
Profile local z Execute	4		
Close			



Ĵå P	Punch with Curves	
۲	Closed polygon from curve(s) in selection:	
	Sweep curves along	
	Curve:	×
	C Vector:	
	OK Cancel Apply	

11 April 2011

GeniE

115

2.5.4.4 Split

Purpose: Split selected beams and/or plates to minor parts based on a) intersection with other beams, plates or feature edges or b) intersection with a guide curve.

GeniE

116

This command is scripted, typically when dividing plate Pl1 into 3 minor plates:

Validate(*Pl1.primitivePartCount* == 8);

Pl1.explode(CloneNameMask());

Validate(Pl1_1);

Validate(Pl1_2);

Validate(Pl1_3);

2.5.4.5 Verify

Purpose: To verify the concept model. This feature is described in Section 3.3.7 of User Manual Volume 3.

This command is not scripted.

🎎 Model Verification	
 Disjoint model parts Beams intersecting plates at a single point Partially split plates Edges shorter than: Detect Sliver Faces Inm [m] Tolerant Entities Geometrical consistency check (ACIS) Fast Thorough Free beam ends Free beam ends Free face edges Misplaced reinforcements Mismatching eccentricities Essential structural properties 	
 Interactive model checking Automatic geometrical consistency check model on save/restore (time consuming for large models) 	
Verify Close	

<u>G</u> eometry	•
Punch	

◆ Curve Punch ... Split ... Verify ...

Geometry Punch ... S Curve Punch ...

> <u>S</u>plit ... <u>V</u>erify ...



۲



11 April 2011

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	117	11 April 2011

2.5.4.6 Stand-Alone Beams

Purpose: To identify beams that are not fully attached to any shell. This feature is described in Section 3.3.8 of User Manual Volume 3.

This command is not scripted.

	Geometry
	<u>P</u> unch
00	C <u>u</u> rve Punch
	Split
	<u>V</u> erify
	Stand-Alone Beams

👪 Stand Alone Bea 🔉	×
Number of selected elements: 0	
Show Selection	
C Show Complement	
C Create Named Set	
Cancel	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	118	11 April 2011

2.5.5 Dimension

Find length between points or angle between beams

2.5.5.1 Create Dimension

Purpose: Find distance between two points and present the value graphically.

This command is not scripted.

2.5.5.2 Angle Between

Purpose: Find angle between two beams and present value graphically.

This command is not scripted.

2.5.6 Customise

The customize pulldown menu allows you to specify options for how to print pictures and define settings for default names.

2.5.6.1 Print

Purpose: To decide whether to include a border around graphics and/or reverse black and white background when printing a picture. Notice that these settings do not apply for picture generations.

This command is not scripted.

Customise 🛛 🗙
Print Default names
Print-
Include border around graphics
Reverse black and white
,
OK Cancel Apply

📕 Create Dimension	
😕 Angle Between	



2.5.6.2 Default names

Purpose: To specify a default naming schema. See Section 3.1.9 of User Manual Volume 1 for more details.

The program default names are shown to the right.

Print Default n					
	lames				
Default Short N	Names D	efault Lor	ng Names	Use copy clor	ne name rule
P	refix	Counter	Suffix	Example:	
Plate:	PI	1		PI1	
Beam:	Bm	1		Bm1	
Joint: J	lt	1		Jt1	
Equipment:	Equipment	1		Equipment1	
Load case /	LC	1		LC1	
Support:	Sp	1		Sp1	
Guide Plane:	GuidePlane	1		GuidePlane1	
Point:	Point	1		Point1	
Guide Curve:	Curve	1		Curve1	
Material:	Mat	1		Mat1	
Section:	Sct	1		Sct1	
Thickness:	Tck	1		Tck1	
			OK	Cancel	Applu

This command is scripted. By modifying the prefix, counter value and suffix as shown to the right, the script commands are: DefaultName(typeFlatPlate, "MyPL",10, "UM"); DefaultName(typeStraightBeam, "MyBm",10, "UM"); DefaultName(typeFrameJoint, "MyJt",10, "UM"); DefaultName(typeEquipment, "MyEquipment",10, "UM"); DefaultName(typeBasicState, "MyLC",10, "UM"); DefaultName(typeBasicState, "MyLC",10, "UM"); DefaultName(typePlane, "MyGuidePlane",10, "UM"); DefaultName(typePlane, "MyGuidePlane",10, "UM"); DefaultName(typePlane, "MyGuidePlane",10, "UM"); DefaultName(typePoint, "MyPoint",10, "UM"); DefaultName(typeGuideCurve, "MyCurve",10, "UM"); DefaultName(typeGuideCurve, "MyCurve",10, "UM"); DefaultName(typeMaterial, "MyMat",10, "UM");

DefaultName(typeThickness, "MyTck", 10, "UM");

Print Default	t names			
Default Short Names Default Long Names T Use copy clone name rule				
	Prefix	Counter	Suffix	Example:
Plate:	MyPL	10	UM	MyPL10UM
Beam:	MyBm	10	UM	MyBm10UM
Joint:	MyJt	10	UM	MyJt10UM
Equipment:	MyEquipme	10	UM	MyEquipment10UM
Load case / combination:	MyLC	10	UM	MyLC10UM
Support:	MySp	10	UM	MySp10UM
Guide Plane:	MyGuidePla	10	ИМ	MyGuidePlane10UM
Point:	MyPoint	10	ИМ	MyPoint10UM
Guide Curve:	MyCurve	10	ИМ	MyCurve10UM
Material:	MyMat	10	UM	MyMat10UM
Section:	MySct	10	UM	MySct10UM
Thickness:	MyTck	10	UM	MyTck10UM

11 April 2011

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	120	11 April 2011

2.6 The Help pulldown menu

Get help online from the GeniE help pages, open DNV on web or list copyrights and 3rd party software used

2.6.1 Help Topics

Purpose: Get help online from GeniE's help pages in user manuals, videos or tutorials.

This command is not scripted.

<u>H</u> elp Topics		F1
Status Lists on the We	ю Ист.	
Memory Monitor	WCD	
8 About GeniE		

2.6.2 Status Lists on the Web

Purpose: Starts the Status List program and looks up GeniE items. *This command is not scripted.*

Help Topics	F1
Status Lists on the Web DNV Software on the Web	
Memory Monitor 8 <u>A</u> bout GeniE	

2.6.3 DNV Software on the Web

Purpose: Start-up of DNV Software address on web. *This command is not scripted.*

Help Topics	F1
Status Lists on the Web	
DNV Software on the Web	
Memory Monitor % % About GeniE	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	121	11 April 2011
2.6.4 Memory Monitor Purpose: Shows the memory consumption of the divided into the different categories that GeniE	e current workspace, is using.	Help Topics F1 Status Lists on the Web DNV Software on the Web Memory Monitor Image: Comparison of the Web Image: Provide the Web Image: Comparison of the Web
If you are working on the limits of your system's available memory, or if you are close to the maximum memory GeniE can utilize, it is a good idea to keep an eye on the memory usage listed in the memory monitor. <i>This command is not scripted.</i>	Overall memory usage Virtual memory 790. Physical memory 155. Hoops memory Allocated 2.51 Used 1.64 Objectstore memory Used 0.29 Acis memory Used N/A	MB MB MB MB MB MB MB MB MB MB

2.6.5 About GeniE

Purpose: Lists 3rd party software used in GeniE. *This command is not scripted.*

Help Topics	F1
Status Lists on the Web	
<u>D</u> NV Software on the Web	
Memory Monitor	
😵 About GeniE	



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	122	11 April 2011

Tool buttons

There are a number of tool buttons that give access to the most commonly used commands. They may be dynamic, meaning that the active tool button may give access to different commands. A dynamic tool button is recognised by having a pulldown arrow embedded on the GUI. One example is the tool button for Insert Plate. It can give access to several commands, like inserting a flat plate or inserting a curved plate using a skin/loft curve operation.



The other available choices behaves in the same way when it comes to selecting and setting them active.

The tool buttons are grouped in eight Toolbars described below. All tool buttons have tool tips, these are listed in **bold**.

2.7 The Main toolbar



Tool button	Tool tip	Description
D	New Workspace	Create a new workspace

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	123	11 April 2011

₫ 1	Open Workspace	Open existing workspace		
IJ	Save Workspace	Save current workspace		
R.	Copy with transform	Copy selection using transformation		
×	Delete	Delete selection		
ę	About	Explains GeniE program version and subcontractors		

2.8 The View Manipulation toolbar

🖢 🐣 🖑 🗟 🛒 🖉 ! 🝼 🔍 ଦଳ ମଧ୍ୟ ଛ • 🖌	▼ Default display
---------------------------------	-------------------

Tool button	Tool tip	Description		
1	Rotate	Rotation in all 3 degrees of freedom (use right mouse button RMB)		
40	Zoom	Zoom in or zoom out (RMB). Place pointer to decide where to zoom. Moving mouse upwards or to the right mean zoom-in, while down or to the left means zoom-out		
Ś	Pan	Move model to desired position on display (RMB)		
Đ	Zoom rubberband	Create a rubber band to zoom in (RMB)		
	Fit	Automatic scale view so that whole model is shown on display		
ø	Spin	Remembers the last rotation and speed of it and makes this a continuous spin		
!	Refresh graphics	Cleans up graphics and remove all dimensions		
1	Color code	Toggle on/off your selection of labelling		
Ŷ	Iso view	View from isometric point		
6 +	View from X	View in negative X-direction		
ъ	View from Y	View in positive Y-direction		
đ	View from Z	View in negative Z-direction		
\$	Save Model View	Save and retrieve a model view		
4	Outline view	Display beams in outline view (3D view, but no thickness)		
	Wireframe view	Display beams in wireframe view		
4	Solid view	Display beams in solid view (3D view including thickness)		
Default display 🔽	Display configuration	Select the current display configuration. There are 10 system configurations. Additional display configurations can be made from <i>View/Options</i> .		

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	124	11 April 2011

2.9 The Loadcase toolbar



Tool button	Tool tip	Description
<no loadcase=""></no>	Default loadcase selection	Set loadcase to current and display current loadcase

2.10 The Labels toolbar

Tool button	Tool tip	Description
Name	Label	Labels selected object (Coordinates, Diagram value, Material, Name, Section, Thickness)

2.11 The Object Types toolbar

✓ ✓ ▼ □ ▼ | Щ ┍╯ ▼ | <u>א</u> ▼ K | Щ ▼

Tool button	Tool tip	Description		
/ -	Straight beam	Insert a straight beam between two snap points		
-	Straight overlapping beam	Insert overlapping beams between two snap points		
•	Pile	Insert a pile between an elevation and snap point along a beam axis		
1 *	Curved Beam	Insert a curved beam between three or more snap points		
•	Flat plate	Insert a plan plate between four or more snap points		
* •	Skin/loft curves	Skinning: Insert a curved shell between two or more curved lines Lofting is a variant of skinning involving both shells and curves to control the shape of the new surface to be created.		
	Curve-net Interpolation	Insert a curved shell interpolating two or more arrays of N and M curved lines, which form a set of $(N-1)x(M-1)$ rectangular patches.		
-	Sweep Curves	Insert a straight or curved shell by sweeping a curve along another curve or vector		
Ħ	Guide plane	Insert a guide plane between four snap points. The guideplane will be created having 5 equal relative spacings		
• •	Guide point	Insert a guide point by clicking one snap point		
\sim	Guide line	Insert a guide line between two snap points		
t^{2} \star	Guide spline	Insert a guide curve (spline) between three or more snap points		
U -	Poly Curve	Insert a poly curve with three or more snap points		
	Guide elliptic arc	Insert a guide ellipse with three snap points (origin, start, end)		
• •	Guide circle	Insert a guide circle with three snap points(origin radius, plane)		
•	Model curve	Insert a model curve along a topological edge with start and stop points		
·) •	Fillet curves	Insert a part of a circle with tangents to two straight lines		
<u><u>A</u> •</u>	Support point	Insert a support point		
·····	Support curve	Insert a support curve along beam, model curve, or guide lines		
K	Joint	Insert a single joint		
¥ ►	Dimension	Find and display the length between two points $(1^{st} \text{ and } 2^{nd} \text{ click on points}, 3^{rd} \text{ click to position the length on the display})$		
₩ ▼	Angle between	Find and display angle between two beams (1 st click on 1 st beam, 2 nd click on 2 nd beam)		

2.12 The Create Methods Toolbar

						×
75	82	₩.	•	P	=×.	R

Tool button	Tool tip	Description		
1 23	Snap point	Positioning beam, plate, guideplane, support points etc. one by one		
盟	Snap point loop	Positioning beam, plate, guideplane, support points etc. sequentially, e.g. end of beam is start of next beam		
™ # . ►	Snap perpendicular	Insert a beam perpendicular to another, 1^{st} click is from start point, 2^{nd} click on perpendicular beam		
► Bi- ¹ F	Snap tangential	Insert a guideline tangential to another curve, 1 st click is from start point, 2 nd click on curve		
	Snap plane	Temporary snap points are defined at the intersection between beams and a snap plane		
434 ★	Snap eccentric	Connect a beam to another beam end using its eccentric position.		
S	Undo snap point	Undo selection of previous snap point		
R.	Clear snap points	Undo selection of all snap points		
R	Reference point modelling	Specifies journalling of reference point modelling on the journal file		

2.13 The Selection Toolbar

Tool button	Tool tip	Description		
×	Selection	Method for selecting one by one or rubberband (LMB). Together with Shift more advanced selections can be made		
14 14	Polygon select	Method for advanced selection where you can make an arbitrary select area (LMB). Make sure that you make a closed envelope.		
	Enclosed by rubberband	Objects need to be fully enclosed by rubberband to be part of selection		
	Touched by rubberband	Objects need to be touched by rubberband to be part of selection		
-	Select visible	Visible objects only selected		
**	Filter beam	Toggle on/off for selection of beams		
*	Filter segment	Toggle on/off for selection of segmented members		
E	Plate selection on/off	Toggle on/off for selection of plates		
•	Filter side	Toggle on/off for selection of one side of a plate. Used when applying wet surface for panel modelling		
	Guide selection on/off	Toggle on/off for selection of guide planes		
4	Guide curve selection on/off	Toggle on/off for selection of guide curves		
A	Support selection on/off	Toggle on/off for selection of support (points and curves)		
K ⋧	Joint selection on/off	Toggle on/off for selection of joints		
¢	Equipment selection on/off	Toggle on/off for selection of equipment		
₽ [™]	Diagram selection on/off	Toggle on/off for selection of load diagrams		
4	Environment selection on/off	Toggle on/off for selection of environment		
	Feature edge selection on/off	Toggle on/off for selection of feature edge		
47.3	Load selection on/off	Toggle on/off for selection of loads		
	Compartment selection on/off	Toggle on/off for selection of compartments		

11 April 2011

3. THE BROWSER MENU

This Chapter describes how the browser menu works with particular focus on commands only available from the browser; the capacity manager and the commands necessary to define the environment. For all of these commands there are examples on the scripting as well. The other commands are available from the *Tools Pulldown Menu* and the scripting is documented in the previous Chapter.



By modifying the "Fields" to also include "Section" and "Material" the browser content is changed:



Name	Description	X[m]	Y [m]	Z [m]	Mass [tonne]	Section	Material
🧼 Flare	Prism Equipment					<none></none>	<none></none>
🥪 Helideck	Prism Equipment					<none></none>	<none></none>
🚞 Weight Lists	Folder						

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	129	11 April 2011

3.1 Analysis

1 st level browser	2 nd level browser	3 rd level browser	Available commands
Activities			New Analysis
	<activity name=""></activity>		Activity Monitor
			Edit Analysis
			New Loadcase
			New Load Combination
			Set Active
			Paste
			Delete
			Rename
		<activity name="">.step(n)</activity>	Edit Activity(n)
Load Cases			New Loadcase
			New Load Combination
			Set Active
			Paste
	<loadcase name=""></loadcase>		Set Current
			Generate Applied Loads
			Recompute Load Sums
			New Load Combination
			Properties
			Сору
			Paste
			Delete
			Rename

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	130	11 April 2011

3.2 Capacity

These commands are only available from the *Browser Menu*. Examples of the script commands are thus also listed in this Section. For more details on member and joint capacity checking, see Chapter 3 of User Manual Volume 4. For plate checking according to CSR Bulk rules, see Chapter 2 of User Manual Volume 5.

The content of the Capacity Folder is created when you right click the folder and define a new capacity manager.

This command is scripted. Typically when using "Analysis1" as basis for the capacity manager:

Cc1 = *CapacityManager(Analysis1)*;

🚞 UM	Create Members	
🗄 🧰 Analysis	Create Panels	all the runs in a snapshot
E E Capacity	Create Joints	
All Runs	Add Run	
	Run All	
	Code Check Status	
	Generate Code Check Loads	
	Execute Code Checks	
	Update Members From Structure	
	Update Structure From Members	
	Set Active	
	Edit Description	
	Delete	_
	Rename	
	Properties	
	Fields	
	Save HTML Report	

3.2.1 Create Members

Purpose: To define members for use in a member check. It is possible to make members of parts of the structure by referring to "Subsets" (or "Named Sets"). A continuous structural beam may be split at joints, at incoming beams or at beam ends to form the capacity members.

This command is scripted. Typically when using the options "Split at incoming beam" and referring to the subset "Row_1":

MemberCreationOpts = MemberCreationOption(); MemberCreationOpts.splitAtJoint = false; MemberCreationOpts.splitAtIncomingBeam = true;

MemberCreationOpts.splitAtBeamEnd = false;

Cc1.createMembers(Row_1, MemberCreationOpts);



👪 Capacity A	Manager	×
📚 Name:	Cc1	
Analysis:	•	×
01	K Cancel	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	131	11 April 2011

3.2.2 Create Panels

Purpose: To define panels for use in a plate check. It is possible to make panels of parts of the structure by referring to "Subsets" (or "Named Sets"). A panel may be defined using the option "cmMinBox" or "cmMaxAreaMoment".

This command is scripted. Typically when using the options "cmMinBox":

Cc1.createPanels(cmMinBox);

3.2.3 Create Joints

Purpose: To define structural joints that is part of a tubular joint check. It is required that the joints are defined as joints (*Insert/Joint*) prior to the analysis.

This command is scripted. Typically:

Cc1.createJoints();

👪 Create Panels	×
Capacity Manager: Cc1	
□ Subset	v
Method: cmMinBox 💌	
ОК	ancel



3.2.4 Add Run

Purpose: To define which code check standard to use and which elements to include in a code check. The available code check standards are shown to the right.

This command is scripted. Typically when using API WSD 2005:

Cc1.AddRun(ApiWsdRun2005());

Cc1.run(1).includeMembers = true;

Cc1.run(1).includeJoints = true;

3.2.4.1 Loadcases

Purpose: To define the load cases to be part of a code check run (the load cases as shown to the right is an example).

This command is scripted. Typically when using the load case "LC_Total" only:

Cc1.run(1).addLoadCase(LC_total);

When removing a load case from the run the typical script command is: Cc1.run(1).removeLoadCase(LC_total);



Loadcases G	Loadcases General Member Joint				
Available Loa	Available LoadCases Included LoadCases				
Name	DesignConditio	Add ->	Name DesignCondition		
RKFLC_flare RKFLC_heli RKFLC_mass	Operating Operating Operating Operating	<-Remove			
	opoloang	Add All			
		Remove All			

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	132	11 April 2011

3.2.4.2 General

Purpose: To define global and general code checking factors.

Some general factors are common for most frame code checks.

This command is scripted, typically:

Cc1.run(1).generalOptions.computeLoadsAsNeeded = true;

Cc1.run(1).generalOptions.purgePositionResults = true;

Most factors are different for the various code checking standards. They are listed in the following paragraphs.

3.2.4.2.1 AISC ASD 2005

Purpose: To define global and general code checking factors according to AISC ASD 2005.

This command is scripted, typically:

Cc1.AddRun(AiscAsdRun());

Cc1.run(1).generalOptions.tensionFactor = 1.67;

Cc1.run(1).generalOptions.torsionFactor = 1.67;

Cc1.run(1).generalOptions.shearFactor = 1.67;

Cc1.run(1).generalOptions.bendingFactor = 1.67;

Cc1.run(1).generalOptions.compressionFactor = 1.67;

3.2.4.2.2 AISC LRFD 2005

Purpose: To define global and general code checking factors according to AISC LRFD 2005.

This command is scripted, typically:

- Cc1.AddRun(AiscLrfdRun());
- *Cc1.run(1).generalOptions.torsionFactor* = 0.9;

Cc1.run(1).generalOptions.shearFactor = 0.9;

Cc1.run(1).generalOptions.bendingFactor = 0.9;

Cc1.run(1).generalOptions.tensionFactor = 0.9;

Cc1.run(1).generalOptions.compressionFactor = 0.9;

C	Common frame check options		
	Performance/Memory Compute loads when needed V Purge position results, keep only worst	85 85	

🚨 Create Code Check Run 🛛 🔊 🔊			
Capacitu Manager, [[24		ОК
	op coord		Consel
Code Check: AISC A	ISD 2005		Lancel
Include: 🔽 Mer	mbers		
Loadcases Gener	al Member		
AISC ASD			
- Safety factors		-	
Axial Tension	1.67		
Axial Compression	1.67		
Bending	1.67		
Shear	1.67		
Torsion	1.67		
Use F12.1 for cro	oss sections not covered in F	2 through F11	
Exclude Torsion	Effects according to chapter	G 💡?	
Common frame	check options		
- Performance/Merr	noru		
Compute loads	when needed	83	
Purge position	results, keep only worst	83	
L			

å (Create Code Che	sk Run		
Ca Co Inc	pacity Manager: Cc de Check: AISCLAR clude: V Memb	1 FD 2005		OK Cancel
AI	oadcases General SC LRFD	Member		
	- Resistance factors Axial Tension	0.9		
	Axial Compression Bending	0.9		
	Shear Torsion	0.9		
	Use F12.1 for cross Exclude Torsion Ef	sections not covered in F2	through F11	
Ċ	ommon frame cl	neck options		
Γ	Performance/Memory Compute loads w	y hen needed	<mark>8</mark> 3	
	Purge position re:	sults, keep only worst	85	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	133	11 April 2011
3.2.4.2.3 API LRFD 200)3	
Purpose: To define global and general factors according to API LRFD 2003. prevail for members and joints. <i>This command is scripted, typically:</i> <i>Cc1.AddRun(ApiLrfdRun());</i> <i>Cc1.run(1).generalOptions.RFPipeXP</i> <i>Cc1.run(1).generalOptions.RFPipeBet</i> <i>Cc1.run(1).generalOptions.RFPipeBet</i> <i>Cc1.run(1).generalOptions.RFPipeTet</i> <i>Cc1.run(1).generalOptions.RFPipeCo</i> <i>Cc1.run(1).generalOptions.aisc.torsio</i> <i>Cc1.run(1).generalOptions.aisc.torsio</i> <i>Cc1.run(1).generalOptions.aisc.shear.</i> <i>Cc1.run(1).generalOptions.aisc.shear.</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.tensio</i> <i>Cc1.run(1).generalOptions.aisc.comptio</i> <i>0.9;</i>	code checking The factors Press = 0.8 ; ear = 0.95 ; md = 0.95 ; mp = 0.85 ; mFactor = 0.9 ; Factor = 0.9 ; mgFactor = 0.9 ; mFactor = 0.9 ; mFactor = 0.9 ; mFactor = 0.9 ;	Create Code Check Run Image: Cc1 Im

Cc1.run(1).generalOptions.RFJointKTens = 0.95; Cc1.run(1).generalOptions.RFJointKComp = 0.95; Cc1.run(1).generalOptions.RFJointKipb = 0.95; Cc1.run(1).generalOptions.RFJointYTens = 0.9; Cc1.run(1).generalOptions.RFJointYComp = 0.95; Cc1.run(1).generalOptions.RFJointYComp = 0.95; Cc1.run(1).generalOptions.RFJointYipb = 0.95; Cc1.run(1).generalOptions.RFJointYopb = 0.95; Cc1.run(1).generalOptions.RFJointXTens = 0.9; Cc1.run(1).generalOptions.RFJointXTens = 0.9; Cc1.run(1).generalOptions.RFJointXComp = 0.95; Cc1.run(1).generalOptions.RFJointXipb = 0.95; Cc1.run(1).generalOptions.RFJointYield = 0.95; Cc1.run(1).generalOptions.RFJointYield = 0.95; Cc1.run(1).generalOptions.RFJointWeld = 0.54;

Loadcases General Member Joint API LRFD Cap-end forces included 8? MEMBER JOINT Joint check resistance factors TΥ Х Κ 0.95 0.9 0.9 Tension 0.95 0.95 0.95 Compression In plane bending 0.95 0.95 0.95 Out of plane bnd. 0.95 0.95 0.95 0.95 Yield stress Weld 0.54

8?

8?

Exclude Torsion Effects according to chapter G 🛛 😵

Common frame check options

Compute loads when needed

✓ Purge position results, keep only worst

DET NORSKE VERITAS SOFTWARE

11 April 2011

Version 5.3

3.2.4.2.4 API WSD 2002

Purpose: To define global and general code checking factors according to API WSD 2002.

This command is scripted, typically:

Cc1.AddRun(ApiWsdRun());

Cc1.run(1).generalOptions.aisc.torsionFactor = 1.67;

Cc1.run(1).generalOptions.aisc.shearFactor = 1.67;

Cc1.run(1).generalOptions.aisc.bendingFactor = 1.67;

Cc1.run(1).generalOptions.aisc.tensionFactor = 1.67;

Cc1.run(1).generalOptions.aisc.compressionFactor = 1.67;

2	2	1 2	5 /	DI	wsn	2005	
5	.4.	4.2	.J P	AP1	WSD	2005	

Purpose: To define global and general code checking factors according to API WSD 2005.

This command is scripted, typically:

Cc1.AddRun(ApiWsdRun2005());

Cc1.run(1).generalOptions.aisc.torsionFactor = 1.67;

Cc1.run(1).generalOptions.aisc.shearFactor = 1.67;

Cc1.run(1).generalOptions.aisc.bendingFactor = 1.67;

Cc1.run(1).generalOptions.aisc.tensionFactor = 1.67;

Cc1.run(1).generalOptions.aisc.compressionFactor = 1.67;

🕯 Create Code Check Run 🛛 🔀
Capacity Manager: Cc1 OK Code Check: API WSD 2002 Cancel Include: Members Joints Loadcases General Member Joint
Cap-end forces included
Axial Tension 1.67 Axial Compression 1.67 Bending 1.67 Shear 1.67
Torsion 1.67 Use F12.1 for cross sections not covered in F2 through F11 Exclude Torsion Effects according to chapter G Q?
Performance/Memory ✓ Compute loads when needed ♀? ✓ Purge position results, keep only worst ♀?

👪 Create Code Check Run	
Capacity Manager: Cc1 Code Check: API WSD 2005	OK Cancel
Include: 🔽 Members 🔽 Joints	
Loadcases General Member Joint API WSD	
Cap-end forces included	
AISC ASD	
Safety factors Axial Tension 1.67	
Axial Compression 1.67	
Bending 1.67	
Shear 1.67	
Torsion 1.67	
□ Use F12.1 for cross sections not covered in F2 through F11 □ Exclude Torsion Effects according to chapter G ♀? Common frame check options Performance/Memory ♀ ▼ Compute loads when needed ♀? ▼ Purge position results, keep only worst ♀?	

DET NORSKE VERITAS SOFTWARE

Version 5.3

3.2.4.2.6 CSR Bulk

Purpose: To define global and general code checking factors according to CSR Bulk. The checks can be combined yield and buckling or separate buckling and yield.

GeniE

135

The options for Transverse stress option C1 and Poisson Correction option are shown below.

Direct Loads Extreme Bending General Bending



This command is scripted, typically:

Cc1.AddRun(CSRBulkRun());

Cc1.run(1).generalOptions.checkBuckling = true;

Cc1.run(1).generalOptions.checkYield = true;

Cc1.run(1).generalOptions.safetyFactorBuckling = 1;

Cc1.run(1).generalOptions.safetyFactorYield = 1;

Cc1.run(1).generalOptions.transverseStressOption = tsGeneralBending;

Cc1.run(1).generalOptions.poissonCorrectionOption = psElementStress;

🗯 Create Code Check Run ΟK Capacity Manager: Cc1 Code Check: CSR Bulk Cancel -Include: ✓ Panels Loadcases General Panel CSR Bulk Checks Yield and Buckling C Buckling Only O Yield Only **Global Options** 1 Safety Factor Buckling Safety Factor Yield Г Transverse stress option C1 General Bending -Element Stress Poisson correction option -Purge subpanel results (buckling only)

3.2.4.2.7 CSR Tank

Purpose: To define global and general code checking factors according to CSR Tank.

Check "Parallel" to utilize all available processors when running PULS. This makes the codecheck run faster, but as it takes all available processor capacity it will not be possible to do other work on your computer while PULS is running.

This command is scripted, typically: Cc1.AddRun(CSRTankRun()); Cc1.run(1).generalOptions.parallelMode = true;

🎎 Create Code Check Run	×
Capacity Manager: Cc1 Code Check: CSR Tank - July 2008 Include: Loadcases General Panel	OK Cancel
Allowable Usage Factors Seagoing Harbour 1 0.8	
PULS calculation mode © Non-linear © Automatic © Linear only	
PULS performance settings ✓ Parallel ✓ Caching Size 1000	

11 April 2011

DET NORSKE VERITAS SOFTWARE

11 April 2011

3.2.4.2.8 Danish Standard DS 412 / DS 449

Purpose: To define global and general code checking factors according to Danish Standard DS 412 and DS 449.

This command is scripted, typically:

Cc1.AddRun(DSRun());

Cc1.run(1).generalOptions.isDS449CodeCheck = true;

Cc1.run(1).generalOptions.gammaFy = 1.15;

Cc1.run(1).generalOptions.gammaFu = 1.41;

🚻 Create Code Check Run	×
	ОК
Capacity Manager: CC1	
Code Check: Danish Standard DS4 💌	Cancel
Include: 🔽 Members 🔽 Joints	
Loadcases General Member Joint Danish Standard	
Danish Standard Section	
C DS 412 O DS 449	
Tabular values	
Safety Class	
© Normal C High	
Control Material	
C Normal C Strict	
Partial coefficients (gamma)	
Fy (yield stress): 1.15	
Fu (tensile strength): 1.41	
E (modulus of elasticity): 1.34	
Tg (punching strength): 1.28	
Common frame check options	
- Performance/Memory-	
Compute loads when needed	82
Purge position results, keep only worst	82

3.2.4.2.9 Eurocode3 EN 1993-1-1 2005

Purpose: To define global and general code checking factors according to Eurocode3 EN 1993-1-1 2005. There are different national annexes; they have different safety factors.

This command is scripted, typically for annex "Standard":

Cc1.AddRun(EN199311Run());

Cc1.run(1).generalOptions.nationalAnnex =naStandard;

Cc1.run(1).generalOptions.partialFactorM0 = 1;

Cc1.run(1).generalOptions.partialFactorM1 = 1;

Cc1.run(1).generalOptions.method1 = true;

🎎 Create Code Check Run	
Capacity Manager: Cc1 Code Check: <mark>Eurocode3 EN 1993-1-1 2005</mark> ▼ Include: ▼ Members	OK Cancel
EN 1993-1-1 National Annex Standard Safety factors Partial factor M0 1 Partial factor M1 1	
Interaction factors Method 1 Common frame check options Performance/Memory Compute loads when needed Purge position results, keep only worst ?	

This command is scripted, typically for annex "Danish Normal":

Cc1.AddRun(EN199311Run());

Cc1.run(1).generalOptions.nationalAnnex = DanishNormal;

Cc1.run(1).generalOptions.partialFactorM0 = 1.1;

Cc1.run(1).generalOptions.partialFactorM1 = 1.2;

Cc1.run(1).generalOptions.method1 = true;

This command is scripted, typically for annex "Danish Stricter":

Cc1.AddRun(EN199311Run());

Cc1.run(1).generalOptions.nationalAnnex = naDanishStricter;

Cc1.run(1).generalOptions.partialFactorM0 = 1.045;

Cc1.run(1).generalOptions.partialFactorM1 = 1.14;

Cc1.run(1).generalOptions.method1 = true;

This command is scripted, typically for annex "Norwegian grouse":

Cc1.AddRun(EN199311Run());

Cc1.run(1).generalOptions.nationalAnnex = naNorwegian;

Cc1.run(1).generalOptions.partialFactorM0 = 1.05;

Cc1.run(1).generalOptions.partialFactorM1 = 1.05;

Cc1.run(1).generalOptions.method1 = true;

137

11 April 2011

Properties	×
Object Properties Edit Code Check Run	
Capacity Manager: Cc1 Code Check: Eurocode3 EN 1993-1-1 2005 Include: Vembers	
Loadcases General Member	
EN 1993-1-1	
National Annex Danish Normal 🗨	
Safety factors Partial factor M0 1.1	
Partial factor M1 1.2	
Interaction factors Method 1	

Properties
Object Properties Edit Code Check Run
Capacity Manager: Cc1 Code Check: Eurocode3 EN 1993-1-1 2005 Include: V Members Loadcases General Member
EN 1993-1-1 National Annex Danish Stricter Safety factors Partial factor M0 1.045 Partial factor M1 1.14
Interaction factors Method 1

👪 Create Code Check Run
Capacity Manager: Cc1 Code Check: Eurocode3 EN 1993-1-1 2C Cancel Include: Members
Loadcases General Member EN 1993-1-1
National Annex Norwegian Safety factors Partial factor M0 1.05
Interaction factors Method 1

DET NORSKE VERITAS SOFTWARE

11 April 2011

Version 5.3

3.2.4.2.10 ISO 19902 2007

Purpose: To define global and general code checking factors according to ISO 19902 2007. The factors prevail for members and joints.

This command is scripted, typically for members:

Cc1.AddRun(ISO19902Run());

Cc1.run(1).generalOptions.RFPipeXPress = 1.25;

Cc1.run(1).generalOptions.RFPipeShear = 1.05;

Cc1.run(1).generalOptions.RFPipeBend = 1.05;

Cc1.run(1).generalOptions.RFPipeTens = 1.05;

Cc1.run(1).generalOptions.RFPipeComp = 1.15;

This command is scripted, typically for joints: Cc1.run(1).generalOptions. $C1_Y_ax = 25$; Cc1.run(1).generalOptions. $C1_X_ax = 20$; Cc1.run(1).generalOptions. $C1_K_ax = 14$; Cc1.run(1).generalOptions.C1_mom = 25; Cc1.run(1).generalOptions. $C2_Y_ax = 11$; $Cc1.run(1).generalOptions.C2_X_ax = 22;$ Cc1.run(1).generalOptions. $C2_K_ax = 43$; *Cc1.run(1).generalOptions.C2_mom = 43; Cc1.run(1).generalOptions.RFJointYield = 1.05;* Cc1.run(1).generalOptions.RFJoint = 1.05; Cc1.run(1).generalOptions.RFJointZj = 1.17;

🏜 Create Code Check Run 🛛 📉 🔀
Capacity Manager: Cc1 Code Check: S0199022007 ▼ Include: ✓ Members ✓ Joints Loadcases General Member Joint ISO19902 ✓ Cap-end forces included ✓ Use Comm. A.13 Axial Compression
MEMBER JOINT Partial resistance factors 1.05 Axial compressive strength 1.18 Bending strength 1.05 Shear/Torsion strength 1.05 Hoop buckling strength 1.25
EN 1993-1-1 National Annex Standard Safety factors Partial factor M0 Partial factor M1 I Interaction factors Method 1 Common frame check options Performance/Memory Compute loads when needed Partial Partia Partial Partial Partial Partial Partial Partial Partial Partia P
✓ Purge position results, keep only worst

	C1	C2	-
Axial Y	25	11	
Axial X	20	22	
Axial K	14	43	
Moment	25	43	-

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	139	11 April 2011

3.2.4.2.11 NORSOK N-004 2004

Purpose: To define global and general code checking factors according to NORSOK N-004 2004.

This command is scripted, typically:

Cc1.AddRun(NorsokRun());

Cc1.run(10).generalOptions.materialFactor = 1.15;

🜡 Create Code Ch	eck Run		X
Capacity Manager: [Code Check: NORS Include: ✓ Mer Loadcases Gener NORSOK N-004	Cc1 DK N-004 2004 nbers IV Joints al Member Joint]	OK Cancel
 ✓ Cap-end forces i ✓ Use Comm. 6.3.: Material factor EN 1993-1-1 	ncluded 9 ? 3 Axial Compression 1.15	_	
National Annex Safety factors Partial factor M0 Partial factor M1	Standard	•	
Interaction factors Common frame Performance/Merr Compute loads Purge position	Method 1 check options ory when needed results, keep only worst	▼ <u> </u>	

3.2.4.3 Member and panel

Purpose: To define global and member or panel specific code checking factors. The factors are different for the various code checking standards. They are listed in the following.

Properties	X
Object Properties	Edit Code Check Run
Capacity Manage Code Check: A	er: Cc1 PI WSD 2005
Include: 🔽	Members 🔽 Joints
Loadcases G	eneral Member Joint
API WSD A	ISC

DET NORSKE VERITAS SOFTWARE

11 April 2011

Version 5.3

3.2.4.3.1 AISC ASD 2005

Purpose: To define global and member specific code checking factors according to AISC ASD 2005.

This command is scripted, typically when using lateral torsional buckling modification 2:

Cc1.run(1).memberOptions. lTorsionalBucklingFactor = 2;

Create Code Check Run		×
Capacity Manager: Cc1		IK I
Code Check: AISC ASD 2005	✓ Car	ncel
Include: 🔽 Members		
Loadcases General Member		
about u-axis	02	
Buckling length Member Len	ngth 👻 [m]	
Effective length factor 1	•	
Moment amplification	•	
about z-axis	netry	
Buckling length Member Len	ngth 🚽 [m]	
Effective length factor	~	
Moment amplification	~	
Bending Moment Option	ending Moment Bending Moment	
Stiffener Spacing 🔗? None	[m]	
Lateral torsional buckling modification	•	
Effective length factor 1 for torsional buckling		
Length between lateral supports		
Length between lateral supports Top flange None	▼ [m]	
Length between lateral supports Top flange None Bottom Flange None	• [m] • [m]	
Length between lateral supports Top flange None Bottom Flange None L section specific	▼ [m] ▼ [m]	
Length between lateral supports Top flange None Bottom Flange None L section specific Chapter E5: C Method a •	▼ [m] ▼ [m] Ø? Method b	
Length between lateral supports Top flange None Bottom Flange None L section specific Chapter E5: Method a	▼ [m] ▼ [m] Method b Short leg	
11 April 2011

Version 5.3

3.2.4.3.2 AISC LRFD 2005

Purpose: To define global and member specific code checking factors according to AISC LRFD 2005.

This command is scripted, typically when using length between lateral supports equal to 1m and 2m for top and bottom flange respectively:

Cc1.run(1).memberOptions. topFlangeLSupportSpacing = 1;

Cc1.run(1).memberOptions. bottomFlangeLSupportSpacing = 2;

Create Code Check F	tun	
Capacity Manager: Cc1 Code Check: AISC LRFD 2 nclude: V Members		OK Cancel
Loadcases General Me	mber	
about y-axis Buckling length Effective length factor Moment amplification about z-axis Buckling length Effective length factor	Member Length [m] 1 [m] 1 [m] V y-z symmetry [m] 1 [m]	
Axial compression and b Bending Moment Option	ending Max Bending Moment C Local Bending Moment None	
Lateral torsional buckling modification Effective length factor for torsional buckling		
Length between lateral :	supports	
Top flange Bottom Flange	None (m) None (m)	
L section specific Chapter E5: C Me	thod a 💿 Method b	

GeniE 142

11 April 2011

Version 5.3

3.2.4.3.3 API LRFD 2003

Purpose: To define global and member specific code checking factors according to API LRFD 2003. The factors prevail for tubular and non-tubular members.

This command is scripted, typically when using a moment amplification of 1.2:

Cc1.run(1).memberOptions.momentAmplificationY = *1.2;*

Cc1.run(1).memberOptions.momentAmplificationZ = 1.2;

This command is scripted, typically when using "Chapter E5 Method a" connected to "Short leg" for L profiles:

Cc1.run(1).memberOptions.aisc.E5Method = moE5a;

Cc1.run(1).memberOptions.aisc .connectionToLongLeg = false;

M A Br S M C

🕯 Create Code Check F	Run	×
Capacity Manager: Cc1 Code Check: API LRFD 20		OK Cancel
Include: 🔽 Members	✓ Joints	
Loadcases General Me API LRFD AISC	ember Joint	
about y-axis Buckling length Effective length factor Moment amplification	Member Length ▼ [m] 1 ▼ 1 ▼	
about z-axis Buckling length Effective length factor Moment amplification	y-z symmetry Member Length v [m]	
Axial compression and b	ending ?? Max Bending Moment C Local Bending Moment	
Stiffener Spacing 💡?	None 💌 [m]	
Lateral torsional buckling modification	1 💌	
Effective length factor for torsional buckling	1	
Length between lateral	supports	
Top flange	None 🗨 [m]	
Bottom Flange	None 💌 [m]	
L section specific Chapter E5: C Me Connected to: C Lo	ହି? ethod a ି Method b ng leg ି Short leg	



11 April 2011

Version 5.3

3.2.4.3.4 API WSD 2002

Purpose: To define global and member specific code checking factors according to API WSD 2002. The factors prevail for tubular and non-tubular members.

This command is scripted, typically when specifying flooding to "Not flooded":

Cc1.run(1).memberOptions.flooding = cfNotFlooded;

This command is scripted, typically when specifying the buckling length to 5 m:

Cc1.run(1).memberOptions.aisc.bucklingY = BucklingLength(5 m, 1);

Cc1.run15).memberOptions.aisc.bucklingZ = BucklingLength(5 m, 1);

Properties	
Object Properties Edit Code	Check Run
Carachi Masaran Cal	
Capacity Manager: [CC1	
Code Check: API WSD 20	U2
Loadcases General Me	
API WSD AISC	
- about u-axis	
Buckling length	Member Length [m]
Effective length factor	1
Moment amplification	1 💌
about z-axis	V-z symmetry
Buckling length	Member Length [m]
Effective length factor	1 💌
Moment amplification	1
Stiffener Spacing	8;
Member	None [m]
Cone	None [m]
Flooding	From Structure
	OK Cancel Apply
Properties	
Object Properties Edit Code	Check Run
Capacity Manager: Cc1	
Code Check: API WSD 200	JZ
Loadcases General Me	
API WSD AISC	Contra 1
	O
Buckling length	Member Length [m]
Effective length factor	1
Moment amplification	1
- about z-axis	✓ y-z symmetry
Buckling length	Member Length 🚽 [m]
Effective length factor	1
Moment amplification	1
Cliffonor Copping 92	None [m]
Suiteriel Spacing #:	
Lateral torsional	
Lateral torsional buckling modification	· · ·
Lateral torsional buckling modification Effective length factor for torsional buckling	
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral	1 supports
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flance	1 supports
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Elange	I supports None V
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Flange	I supports None Im None
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Flange L section specific	I supports None ▼ [m] None ▼ [m]
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Flange L section specific Chapter E5: C Me	1 supports None ▼ [m] None ▼ [m]
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Flange L section specific Chapter E5: C Me Connected to: C Lo	I supports None ▼ [m] None ▼ [m] ethod a Method b ng leg Short leg
Lateral torsional buckling modification Effective length factor for torsional buckling Length between lateral Top flange Bottom Flange L section specific Chapter E5: O Me Connected to: O Lo	I supports None ▼ [m] None ▼ [m] Supports None ▼ [m] None ▼ [m] Supports Supports (m]

Version 5.3

3.2.4.3.5 API WSD 2005

Purpose: To define global and member specific code checking factors according to API WSD 2005. The factors prevail for tubular and non-tubular members.

This command is scripted, typically when assigning a different buckling length factor (0.5) about the z-axis:

Cc1.run(1).memberOptions.bucklingZ = BucklingLength(moMemberLength, 0.5);

This command is scripted, typically when assigning a different buckling length 600 c m about the z-axis:

Cc1.run(1).memberOptions.aisc.bucklingZ = BucklingLength(600 cm, 1);

🖞 Create Code Check F	Run		
Capacity Manager: Cc1 Code Check: API WSD 200 Include: V Members Loadcases General Me API WSD AISC about y-axis Buckling length Effective length factor	Is ▼ ✓ Joints mber Joint Member Length ▼ 1 ▼	<mark>@</mark> ?	OK Cancel
Moment amplification	1		
about z-axis Buckling length Effective length factor Moment amplification	Member Length	[m]	
Axial compression and b Bending Moment Option	ending Max Bending Mome CLocal Bending Mom	<mark>ଡ</mark> ? nt ent	
Stiffener Spacing		83	
Member	None 💌	[m]	
Cone	None	[m]	
Flooding	From Structure		

🌡 Create Code Check I	Run	×
Constitution Cont		ок
Lapacity Manager: LCT		Connect
Code Check: API WSD 20	105 <u>-</u>	Lancel
Include: 🔽 Members	✓ Joints	
Loadcases General M	ember Joint	
API WSD AISC		
about y-axis	<u> </u>	1
Buckling length	Member Length 🗾 [m]	
Effective length factor	1 •	
Moment amplification	1 💌	
about z-axis	y-z symmetry	
Buckling length	Member Length 🚽 [m]	
Effective length factor	1 👻	
Moment amplification	1	
Autol a second data and d	handing On	
Axial compression and	Max Bending Moment	
Bending Moment Option	n C Legal Banding Moment	
	Cocal Benuing Moment	
Stiffener Spacing 💡?	None 💌 [m]	
Lateral torsional		
buckling modification		
Effective length factor for torsional buckling	1	
Length between lateral	supports	,
Top flange	None [m]	
Top flange Bottom Flange	None [m] None [m]	
Top flange Bottom Flange L section specific	None ▼ [m] None ▼ [m]	
Top flange Bottom Flange L section specific Chapter E5: C M	None (m) None (m) ethod a (Method b	
Top flange Bottom Flange L section specific Chapter E5: C M Connected to: C Lo	None ▼ [m] None ▼ [m] ethod a	

3

GeniE 145

11 April 2011

Version 5.3

3.2.4.3.6 CSR Bulk

Purpose: To define global and panel specific code checking factors according to CSR Bulk.

This command is scripted, typically when modifying the panel net thickness to "Average Idealised Panel" and the top rotation boundary for panel buckling to "Clamped":

Cc1.run(1).panelOptions.panelThickness = ptAverageIdealisedPanel;

Cc1.run(1).panelOptions.rotationBoundaryTop = rbClamped;

Properties			
Object Properties Edit Code Ch	neck Run		
Capacity Manager: Cc1 Code Check: CSR Bulk Include: Loadcases General Pane CSR Bulk Check Buckling for © Whole and Subpanel © Whole Only © Subpanels Only	s		
Panel Options			
Correction Factor F1	Sniped	•	
Panel Net Thickness	Minimum Idealised Panel	💌 [m]	
Panel Length (a)	From Idealised Panel	▼ [m]	
Panel Width (b)	From Idealised Panel	💌 [m]	
Subpanel Length (a')	Two B	▼ [m]	
Rotation Boundary for Par	nel Buckling		
Тор	Simply Supported	•	
Left	Simply Supported	•	
Right	Simply Supported	•	
Bottom	Simply Supported	•	
	-		
	ОК	Cancel	Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	146	11 April 2011

3.2.4.3.7 CSR Tank

Purpose: To define global and panel specific code checking factors according to CSR Tank.

Note that CSR Tank categorizes the panels into two groups, stiffened or unstiffened. You can also force panels to be treated as stiffened or unstiffened.

This command is scripted, typically: Cc1.AddRun(CSRTankRun()); Cc1.run(1).panelOptions.unstiffened.idealizedLength = 1;

Create Code Check Run	×	Create Code Check Run
Capacity Manager: Cc1 Code Check: CSR Tank - July 2008 V Include: V Panels	OK Cancel	Capacity Manager: Cc1 Code Check: CSR Tank - July 2008 ▼ Include:
Loadcases General Panel CSR Tank Panel Type From Structure • ?? UNSTIFFENED STIFFENED		Loadcases General Panel CSR Tank Panel Type From Structure ?? UNSTIFFENED STIFFENED Geometry
Length From Idealized Shape [m] Spacing From Idealized Shape [m] Net Thickness Average From Structure [m]	<u>8</u> ;	Length From Idealized Shape [m] ?? Width From Idealized Shape [m] Net Thickness Average From Structure [m]
Settion 1 <from structure=""> Section 2 <from structure=""> End Support Continuous Number of Primary From Structure Number of Secondary 0 Boundary Conditions</from></from>	8.	Boundary Conditions In-plane Support Integrated Rotational Support Left Simply Supported N Upper Simply Supported Lower Simply Supported N N N N N N N N N
In-plane Support Integrated Loads Include Lateral Pressure Adjust Stress Stress Scaling Factor SigmaX: SigmaX: TauXY: TauXY: Ta	\$? \$?	Loads Include Lateral Pressure <pre> Adjust Stress P? Stress Scaling Factor SigmaX: Stress BigmaX: 0 SigmaY: 0 TauXY: 0</pre>
Results Buckling Assessment Method From Structure	δ 3	Buckling Assessment Method From Structure

11 April 2011

Version 5.3

3.2.4.3.8 Danish Standard DS 412 / DS 449

Purpose: To define global and member specific code checking factors according to Danish Standard DS 412 and DS 449

This command is scripted, typicallywhen modifying the buckling length:

Cc1.AddRun(DSRun());

Cc1.run(2).memberOptions.bucklingY = BucklingLength(1, 1);

Cc1.run(2).memberOptions.bucklingZ = BucklingLength(1, 1);

3.2.4.3.9 Eurocode3 EN 1993-1-1 2005

Purpose: To define global and member specific code checking factors according to Eurocode3 EN 1993-1-1 2005.

This command is scripted, typically when modifying the lateral torsional buckling Factor C1 to "3":

Cc1.run(1).memberOptions.LTBFactorC1 = 3;

Obje	ct Properties Edit Code Che	eck Run		
Ca	pacity Manager: Cc1			
Co	de Check: Danish Standard	DS412/DS4		
Inc	lude: 🔽 Members 🔽	Joints		
	oadcases General Memb	er Joint		
	Danish Standard	1		
	about y-axis	Member Length	- [m]	
	Buckling length	Member congor		
	Effective length	Disped Pinned		
	C1 factor			
	Column case for buckling	lao	_	
	about z-axis	y-z symmetry		
	Buckling length	Member Length	- [m]	
	Effective length	1	~	
	C1 factor	Pinned Pinned	-	
	Column case for buckling	a0	-	
		From Shrushure		
	Flooding	From Structure	•	
-				

🚨 Create Code Check I	Run		
Capacity Manager: Cc1			ОК
Code Check: Eurocode3 E	N 1993-1-1 2008 -		Cancel
Include: Vembers			
	amber]		
EN 1993.1.1			
1			
- about y-axis	Member Length	- [m]	
Effective length factor	1	- I	
Moment factor	1		
Buckling curve	Automatic		
about z-axis	v-z summetru		
Buckling length	Member Length	- [m]	
Effective length factor	1	-	
Moment factor	1	-	
Buckling curve	Automatic	-	
Stiffener Spacing 💡?	None	▼ [m]	
Lateral torsional bucklin	, Ig		
Factor C1	1	•	
Factor kc	1	•	
Curve C General	λιτο 0.4	_	
Rolled or equiv welde	d β 0.75	_	
- Length between lateral	supports		
Top flange	None	- [m]	
Detters Flaves	None	• [m]	
Bottom Flange	Inone	• [m]	
Section classification	Automatic	•	
Interaction factor method 2	Automatic	•	
Hollow and welded box	specific	_	
Hot finished	Thick welds		
Evoludo "concerneti	in" unago factor from f	iormula (C. 2)	
Exclude "conservativ	ve usage ractor from f	ormula (6.2)	

GeniE 148

11 April 2011

Version 5.3

3.2.4.3.10 ISO 19902 2007

Purpose: To define global and member specific code checking factors according to ISO 19902 2007. The factors prevail for tubular and non-tubular members.

This command is scripted, typically when modifying moment reduction factors about y and z-axis to "Case1" and "Case3":

Cc1.run(1).memberOptions.momentReductionY = *moCase1;*

Cc1.run(1).memberOptions.momentReductionZ = moCase3;

This command	is scripted, t	ypically w	vhen disablin	g the
"Hot finished"	and "Thick	welds" fo	r hollow and	welded
box profiles:				

Cc1.run(9).memberOptions. EN1993_1_1.hotFinish = false;

Cc1.run(9).memberOptions. EN1993_1_1.thickWelds = false;

🕯 Create Code Check F	tun	×
Capacity Manager: Cc1		OK
Code Check: ISO19902 20	07 🗾	Cancer
Include: 🔽 Members	Joints	
Loadcases General Me ISO19902 EN 1993-1-1	ember Joint	
about y-axis		
Buckling length	Member Length 🗾 [m]	
Effective length factor	1 🔹	
Moment reduction	1 💌	
about z-axis	y-z symmetry	
Buckling length	Member Length 📃 [m]	
Effective length factor	1 👻	
Moment reduction	1 👻	
Axial compression and b	pending	
Moment Bending Option	 Max Bending Moment Local Bending Moment 	
Stiffener Spacing	8?	
Member	None 💌 [m]	
Cone	None 💌 [m]	

From Structure

•

Flooding

🖞 Create Code Check I	Run	×			
		ок			
Capacity Manager: Uc1					
Code Check: ISO19902 20	07 💌	Cancel			
Include: Members	Joints				
Loadcases General Me	ember Joint				
IS019902 EN 1993-1-1					
about y-axis	<u>8</u> ?	1			
Buckling length	Member Length 📃 [m]				
Effective length factor	1 🔹				
Moment factor	1 •				
Buckling curve	Automatic 🗨				
about z-axis	y-z symmetry				
Buckling length	Member Length 📃 [m]				
Effective length factor	1 🗸				
Moment factor	1 🔽				
Buckling curve	Automatic 📃				
Stiffener Spacing 💡?	None 💌 [m]				
Lateral torsional bucklin	g	1			
Factor C1	1 🔹				
Factor kc	1 🔹				
Curve C General XIII 0.4					
Rolled or B 0.75					
Length between lateral	supports				
Top flange	None 👻 [m]				
Pottern Flamme	None [m]				
bottom Hange					
Section classification	Automatic 🗨				
Interaction factor method 2	Automatic 🗨				
Hollow and welded box	specific				
V Hot finished	Thick welds				
Exclude "conservativ	e" usage factor from formula (6.2)				

3.2.4.3.11 NORSOK N-004 2004

Purpose: To define global and member specific code checking factors according to NORSOK N-004 2004. The factors prevail for tubular and non-tubular members.

This command is scripted, typically when specifying flooding to status "flooded".

Cc1.run(1).memberOptions.flooding = cfFlooded;

de check. Indrisolen	D04 2004 <u>+</u>	
clude: 🔽 Members	Joints	
.oadcases General M	ember Joint	
NORSOK N-004 EN 19	93-1-1	
- about usavia		0-
Buckling length	Member Lenath	·
Effective length factor		
Moment factor		
Buckling curve	Automatic 🔹	
about zavis		
Buckling length	Member Length	
Effective length factor		
Moment factor		
Buckling curve	Automatic	
Stiffener Spacing V?		
Eactor C1		
Factorika		
Curve O General	λ _{LT,0} 0.4	
equiv. welde	ed β 0.75	
Length between lateral	supports	
Top flange	None 💌 [m]	
Bottom Flange	None 🗨 [m]	
Section classification	Automatic 🗨	
Interaction factor method 2	Automatic 🗨	
Hollow and welded box	specific Thick welds	
Exclude "conservativ	/e'' usage factor from formula (6.2)

🖞 Create Code Check Run 🛛 🛛 🔀
ПК
Capacity Manager: Cc1
Code Check: NORSOK N-004 2004 Cancel
Include: 🔽 Members 🔽 Joints
Loadcases General Member Joint NORSOK N-004 EN 1993-1-1
about y-axis
Buckling length Member Length 🗨 [m]
Effective length factor 1
Moment reduction 1
about z-axis V-z symmetry
Buckling length Member Length 🚽 [m]
Effective length factor
Moment reduction
Axial compression and bending
Bending Moment Option C Local Bending Moment
Stiffener Spacing
Member None (m)
Cone None (m)
Flooding From Structure

This command is scripted, typically when specifying the factor kc for the lateral torsionl buckling to "1.2":

Cc1.run(1).memberOptions. EN1993_1_1.LTBFactorKc = 1.2;

3.2.4.4 Joint

Purpose: To define global and tubular joint specific code checking factors. The factors are different for the various code checking standards. They are listed in the following.

Properties
Object Properties Edit Code Check Run
Capacity Manager: Cc1 Code Check: API WSD 2005 Include: V Members V Joints
Loadcases General Member Joint API WSD

3.2.4.4.1 API LRFD 2003

Purpose: To define global and tubular joint specific code checking factors according to API LRFD 2003.

Properties					
Object Properties Edit Code Che	ck Run				
Capacity Manager: Cc1 Code Check: API LRFD 2003 Include: I Members I Loadcases General Member API LRFD Joint Braces	Joints ₉₇ Joint				
Brace	Brace Type	Gap [m]	Load Transfer	Through Brace	Weld Thickness [m]
Cc1.run(4)	Loadpath 💌	From Structure 🗾 💌			None
	YT X K KTT KTK	From Structure None Manual			None 🛛 🔍 Manual
	Geometry Loadpath Interpolate		(0)		Cancel Apply

This command is scripted, typically when modifying brace type to "KTT", a gap value of 2in and a weld thickness of 15mm:

Cc1.run(4).braceOptions.braceType = braceTypeKTT;

Cc1.run(4).braceOptions.gap = 2 in;

Cc1.run(4).braceOptions.weldThickness = 15 mm;

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	151	11 April 2011

3.2.4.4.2 API WSD 2002

Purpose: To define global and tubular joint specific code checking factors according to API WSD 2002.

Properties					
Object Properties Edit Code Ch	eck Run				
Capacity Manager: Cc1 Code Check: API WSD 2002 Include: V Members V Loadcases General Mem API WSD Joint Braces	Joints per Joint				
Brace	Brace Type	Gap [m]	Load Transfer	Through Brace	Weld Thickness [m]
Cc1.run(5)	Loadpath 💌	From Structure 🗾			None
	YT X K KTT KTK Geometry Loadnath	From Structure None Manual		IK]	None Manual
	Interpolate				

This command is scripted, typically when modifying brace type to "X", Load Transfer active and Through Brace active:

Cc1.run(5).braceOptions.braceType = braceTypeX; Cc1.run(5).braceOptions.loadTransfer = true;

Cc1.run(5).*braceOptions.throughBrace* = *true*;

3.2.4.4.3 API WSD 2005

Purpose: To define global and tubular joint specific code checking factors according to API WSD 2005.

Properties				×
Object Properties Edit Code Che	eck Run			
Capacity Manager: Cc1 Code Check: API WSD 2005 Include: V Members V Loadcases General Memb API WSD Joint Braces	Joints er Joint			
Brace	Brace Type	Gap [m]	Through Brace	
Cc1.run(6)	Loadpath 💌	From Structure 💌		
	YT X K KT KTK Geometry Loadpath	From Structure None Manual	Cancel Apply	

This command is scripted, typically when modifying brace type to Interpolate: 20% YT, 30% X, 30% K, 10% KTT and 10% KTK:

Cc1.run(6).braceOptions.braceType = braceTypeInterpolate(0.2, 0.3, 0.3, 0.1, 0.1);

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	152	11 April 2011

3.2.4.4.4ISO 19902 2007

Purpose: To define global and tubular joint specific code checking factors according to ISO 19902 2007.

Properties				
Object Properties Edit Code Che	ck Run			
Capacity Manager: Cc1 Code Check: ISO19902 2007 Include: V Members V Loadcases General Member ISO19902	Joints er Joint			
Brace	Brace Type	Gap [m]	Through Critical Brace Joint	Brace Utilization
Cc1.run(9)	Loadpath 💌	From Structure		1 💌 💟
	ΥΤ X КП КПК КПК	From Structure None Manual		Ub From Run Manual
	Geometry			
	Interpolate			

This command is scripted, typically when modifying the brace utilisation to 90% (or 0.90):

Cc1.run(9).braceOptions.braceUtilization = 0.9;

3.2.4.4.5 NORSOK N-004 2004

Purpose: To define global and tubular joint specific code checking factors according to NORSOK N-004 2004.

Properties					X
Object Properties Edit Code Che	ck Run				
Capacity Manager: Cc1 Code Check: NORSOK N-004 Include: V Members V Loadcases General Member NORSOK N-004	2004 Joints er Joint				
Brace	Brace Type	Gap [m]	Through Brace	2	
Cc1.run(10)	Loadpath	From Structure			
·			(OK)	Cancel Apply	

This command is scripted, typically when neglecting any gap values part of the structural model: Cc1.run(10).braceOptions.gap = gtNone;

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI

3.2.5 Run All

This option is intended to be used when doing redesign.

It re-runs the analysis, generates the code check loads and executes the code check in one single operation.

3.2.6 Code Check Status

Checks if the structural connection is ok.

This command is not scripted.

3.2.7 Generate Code Check Loads

Purpose: To compute forces and moments at system defined locations for use in the subsequent code check. See User Manual Volume 4 and 5 for details.

This command is scripted, typically:

Cc1.run(1).generateCodeCheckLoads();



Create Members
Create Panels
Create Joints
Add Run
Run All N
Code Check Status.
Generate Code Check Loads
Execute Code Checks
Update Members From Structure
Update Structure From Members
Set Active
Edit Description
Delete
Rename
Properties
Fields
Save HTML Report



11 April 2011

11 April 2011

Create Members... Create Panels...

Create Joints... Add Run...

Execute Code Che

Set Active Edit Description... Delete...

Rename...

Properties... Fields ... Save HTML Report ...

Generate Code Check Loads

3.2.8 Execute Code Checks

Purpose: To perform the code check according to the parameters as defined in the Add Run and the code check forces and moments computed by the generate Code Check Loads.

This command is scripted, typically:

Cc1.executeCodeChecks();

3.2.9 Update Members from Structure

This is used when you change something in the concept model, like removing a beam or altering segmentation, and you want this change to be reflected in the capacity model. Information is moved *from* the concept model *to* the capacity model.

This command is scripted, typically:

Cc1.updateMembersFromStructure();

3.2.10 Update Structure from Member

This is used to move information from the capacity model to the concept model.

This command is scripted, typically:

Cc1.updateStructureFromMembers();

Update Members From Structure Update Structure From Members 😽
Set Active

Update Members From Structure	
Update Structure From Members	
Set Active	

3.2.11 Set active

Purpose: To switch between various capacity managers and code checks and set one to active.

This command is scripted, typically when switching from capacity manager Cc2 to Cc1:

Cc1.setActive();



GeniE 154

3.2.12 Edit Description

Purpose: To add user defined text for a description of a capacity manager or a code check run.

This command is scripted, typically when adding the text UM Example: Cc1.description = "UM example";

🞎 Edit Desc	ription	
Object:	Cc1	
Description:	Capacity Manager	~
		Connect
	UK	Lancel

3.2.13 Delete

Purpose: To delete a capacity manager or a code check run. *This command is scripted, typically:* Delete(Cc1);



3.2.14 Rename

Purpose: To rename a capacity manager or a code check run. *This command is scripted, typically when renaming to UM_Cap_Manager: Rename(Cc1,"UM_Cap_Manager");*



3.2.15 Properties

Purpose: To modify a capacity manager or a code check run.

This command is scripted. The scripting depends on which modifications are performed to the capacity manager or a code check run.



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	156	11 April 2011

3.3 Environment

Purpose: To define the environmental data for air (wind), water (wave and current) and soil for use in hydrodynamic and pile-soil analyses. There may be several locations whereby it is possible to run a number of analyses using the different locations.

The locations are defined from the Environment Folder; RMB and select New Location.



A location refers to data defined in the Environment sub-folders for Air, Directions, Soil and Water.

For more details on how to define the environmental data, see User Manual Volume 2.

👪 Create/Edit Lo	cation	
📚 Name 🛛 Location	12 💌	
Gravity: 9.80665 m	/s^2 [m/s^2]	
Air Water Soi		
Density:	0.001226 tonne/m^3	[tonne/m^3]
Kinematic viscosity:	1.462e-005 m^2/s	[m^2/s]
		Cancel Apply

👪 Create/Edit Location 🛛 🔀					
📚 Name Location	Name Location2				
Gravity: 9.80665 m	/s^2 [m/s^2]				
Air Water Soi					
Density:	1.025 tonne/m^3	[tonne/m^3]			
Kinematic viscosity:	1.19e-006 m^2/s	[m^2/s]			
Waterline Z	Om	[m]			
	СК	ancel Apply			

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	157	11 April 2011

When filling in a table as shown to the right, data can be created from this view. Alternatively, the data as defined in the subfolders can be used.



👪 Create/Edit Locatio	n		×
Same Location2	_		
Gravity: 9.80665 m/s^2	[m/s^2]		
Air Water Soil			
Seabed Z (mudline): -75	m		[m]
. –			
Seabed delta: 8?			▼ ×
Soil layers relative to sea	abedZ 💡?		
Soil layers: 8?			
Z Bottom Soi	Type Soil Cu	rves Soil ()ata Sublayers 🔥
1			
2			
4			
5			
6			
17			

This command is scripted. The scripting depends on which data are used to define the location. Typically: Location1 = Location(124 m, 0 m); Location1.gravity = 9.80665 m/s^2; Location1.air.density = 0.001226 tonne/m^3; Location1.air.kinematicViscosity = 1.462e-005 m^2/s; Location1.water.density = 1.025 tonne/m^3;

Location1.water.kinematicViscosity = 1.19e-006 m²/s;

Location1.seabed.normaldirection = Vector3d(0 m, 0 m, 1 m);

Location1.seabed.seabedDelta = Scour1;

Location1.insertSoilBorder(-1.5 m);

••

Location1.insertSoilBorder(-80 m);

Location1.soil(1).soilCurves = SoilCurves1;

Location1.soil(1).soilData = SoilData1;

Location1.soil(1).soilType = Sand1;

Location1.soil(1).numberOfSublayers = 1;

```
•••
```

Location1.soil(7).soilCurves = SoilCurves1; Location1.soil(7).soilData = SoilData6; Location1.soil(7).soilType = Sand5; Location1.soil(7).numberOfSublayers = 2;

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	158	11 April 2011

When a location is defined, the associated wave load condition including current, wind and wave model is defined. The data can be entered in the view below or the data as defined in the subfolders can be used.

This command is scripted. The scripting depends on which data are used to define the condition. Typically:

Condition1 = *DeterministicTime(Location1)*;

Condition1.waterSurface.regularWaveSet = WaveSet1;

Condition1.populate();

Condition1.addCalmSea();

Condition1.component(1).water.current(CurrentProfile1);

Condition1.component(1).waterSurface.waveModel(Airy());

Condition1.component(2).water.current(CurrentProfile1);

Condition1.component(2).waterSurface.waveModel(Stokes5());

Condition1.component(3).water.current(CurrentProfile1);

Condition1.component(3).waterSurface.waveModel(Stokes5());

Condition1.component(4).water.current(CurrentProfile1);

Condition1.component(4).waterSurface.waveModel(Stokes5());

👪 New Wave Load Condition			
Deterministic Time	85		
🗞 Name Condition_1	_		
Wave components	Assign wave component	t properties	
<u>8</u> 5	83		
Regular wave set:	👻 🗶 Current profile:	V	
	Wind profile:	v	
Direction set:	Wave model:	~	
Frequency set:	Urder:		
F Phase set:	Fill all	₽ ?	
Wave height set:	Fill selected	 	
C Wave height function:			
		ents ¥?	
Specify value: 8?			
Period Height Phase Direction	Current profile V	Vind profile Wave m	odel Order
2			
3			
5			
6			
8			
10			

🖃 💼 UME	
🗄 🧰 Analysis	
🛁 Capacity	
🚊 🧰 Environment	
🧰 Air	
📄 Directions	
Location1	
- 💼 Soil	Edit location
📄 🔛 Water	Set Active location
🗄 🧰 Equipment	
🗄 🧰 Properties	New Wave Load Condition
🗄 🧰 Structure	Information
🗄 💼 Utilities	
	Сору
	Paste
	Delete
	Rename
	Labels •
	ColorCode •
	Named set
	View options
	Visible model
	Fields
	Save HTML Report
	ouver time report to

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI

3.3.1 Air

Purpose: To define and modify wind profile data. These data can be used later when defining a wave load condition(s).

🖃 💼 UME Name Description 🗄 🚞 Analysis 🦲 Capacity 🖻 🚞 Environment 🔁 Air Properties... Directions New Wind Profile ... Paste 🦲 Soil Fields ... 🦲 Water Save HTML Report ... 🗄 💼 Equipment 🗄 🧰 Properties 🗄 🧰 Structure 🗄 🛅 Utilities

11 April 2011

3.3.1.1 Properties

Purpose: To define a new wind profile or modify existing wind profiles.

This command is scripted. The scripting depends on which parameters are modified or entered to create a new wind profile, see below.

Properties				
Wind Profile Base				
Name	Use	Description		
<none></none>		no wind profile b		
Create/Edit Wind Profile Base				
<u>,</u>				
		OK	Cancel	Apply

3.3.1.2 New Wind Profile

Purpose: To define a new wind profile. Notice that there are three ways of describing the wind profile formula.

This command is scripted. By using a set of unrealistic wind profile parameters a typical script sequence may be: Reference height 50 m, exponent 1, wind velocity 5 m/s, direction 0 deg and gust factor 2.

WindProfileRelDir1 = WindProfileRelDir(5, 50, 1, dtRelativeX, wpGeneral, 0, 2);

👪 Create/Edit \	🏙 Create/Edit Wind Profile 🛛 🔀				
Wind profile					
• New C Edit	existing WindProfileReID)ir1			
C Direction relative	e to wave heading 📀 D	irection relative to	x-axis		
Wind profile form	nula: General	-	83		
	Extreme General		Dim.		
	RNormal	1	[m] 🛪		
	Wind profile exponent:		×		
	Average wind velocity:		[m/s]	×	
$V_0 \int_{\nabla}^{Z_0}$	Wind direction:		[deg]	×	
\vdash	Gust factor:		×		
V	Mean period ratio:				
		(Cancel	Apply	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	160	11 April 2011

3.3.2 Directions

Purpose: To define discrete directions or a set of directions. These data can be used later when defining a wave load condition(s).

3.3.2.1 Properties

Purpose: To define or modify a direction or direction set.

This command is scripted. The scripting depends on which parameters are modified or entered to create a new direction or direction set, see below

3.3.2.2 New Direction

Purpose: To define a new direction.

This command is scripted. Typically when defining three directions with angles 0 deg, 15 deg and 30 deg:

Direction1 = Direction(0 deg);

Direction2 = *Direction(15 deg);*

Direction3 = *Direction(30 deg);*

3.3.2.3 New Direction set

Purpose: To define a new direction set.

This command is scripted. Typically when using a step of 90 degrees between 0 deg and 360 deg:

DirectionSet1 = DirectionSet(Array(0 deg,90 deg,180 deg,270 deg));

👪 Create/Edit Direction 🛛 🔀				
Direction Direction Set				
Name DirectionSet1				
First direction: 0 [deg]				
Last direction: 360 [deg]				
Step: [deg]				
Fill table 🛛 😵 ? Row operations: 😵 ?				
Specify direction: Q? Insert Remove Clear table				
Direction [deg]				
1				
2				
4				
5				
8				
9				
10				
" Cancel Apply				

Properties				\mathbf{X}
Direction Base				
Name	Use	Description		
<none></none>		no direction base		
Create/Edit D	irection Bas	e		
		ОК	Cancel	Apply



Create/Edit Direction				
Direction Direction	tion Set			
📚 🔍 New	C Edit existing Direction1			
Angle:	[deg] 🗙			
ОК	Cancel Apply			

GeniE

11 April 2011

Version 5.3

3.3.3 Soil

Purpose: To define soil properties that can be used when defining a location. The soil properties may also be defined from the input dialogue for a location – for both alternatives the soil properties are found under the subfolder Soil.

🖃 💼 UME Name Description 🗄 🚞 Analysis 🫅 Capacity Properties... 😑 🧰 Environment New Sand ... 🚞 Air New Clay ... Directions New Scour ... New Soil Data ... 🔁 Soil New Soil Curves ... 🚞 Water Equipment Paste Fields ... 🗄 🚞 Structure Save HTML Report ... 🗄 🛅 Utilities

3.3.3.1 Properties

Purpose: To define or modify soil properties.

This command is scripted. The scripting depends on which parameters are modified or entered to create a new soil property, see below.

Properties					3
Soil Soil Properti	es Seab	ed Delta			
Name	Use	Description			
<none></none>		no soil			
Create/Edit Soil					
		OK	Cancel	Apply	

3.3.3.2 New Sand

Purpose: To define sand properties.

This command is scripted. Typically angle 40 deg, mass 1.99 tonne/m³, over-consolidation ratio 1, friction ratio 1 and t-z curve z displacement 1m:

Sand1.ocRatio = 1;

Sand1.frictionRatio = 1;

Sand1.roTotal = 1.99 tonne/m^3;

Sand1.phi = 40 deg;

Sand1.zDisplacement = 0.99 m;

👪 Create/Edit Soil			X
Sand Clay			
New C Edit existing Soil4			
The angle of internal friction		[deg] 🗙	
mass density that corresponds to the total unit weight		[tonne/m^3]	×
Gensod			
Code for open gap		_	
Over-consolidation Ratio		×	
Friction			
residual/peak skin friction ratio	1		
T-Z curve z displacement from peak to residual skin friction		[m]	
		Cancel	Apply

GeniE	User Manual Vol. VI
162	11 April 2011
r tå Create/Edit Soil	
relevant Sand Clay	ng Soil3
Undrained Shear Stre	ngth
	Values given relative to seabed
suz2	[KPa] Z2 -100 m [m]
to the total unit weight	sponds [tonne/m^3] 🗙
strain at half of max stre	888 X
,100 J factor for API code Gensod Code for open gap Over-consolidation Ba	0.25
	GeniE 162 relevant Sand Clay New C Edit existing Undrained Shear Stree suz1 suz2 mass density that come to the total unit weight API strain at half of max stree J factor for API code Gensod Code for open gay Over-consolidation Ra

Friction

residual/peak skin friction ratio T-Z curve z displacement from peak to residual skin friction

3.3.3.4 New Scour

Purpose: To define scour properties.

This command is scripted. Typically with relevant data as shown below:

Scour1.localSlope = 20 deg;

Scour1.localScour = 1.0 m;

Scour1.generalScour = 0.5 m;

👪 Create/	Edit Seabed delta		×
Scour			
• New 0	Edit existing Scour2		
	General scour (z) Local Scour around piles (zL)		[m] × [m] ×
	Side slope of local scour holes (a)		[deg] ×
		Lancel	Apply

1

Γ

[m]

Cancel Apply

GeniE 163 User Manual Vol. VI

11 April 2011

Version 5.3

3.3.3.5 New Soil Data

Purpose: To define soil data properties.

This command is scripted. Typically:

SoilData1.peakSkinFrictionTension = 3 KPa;

SoilData1.peakSkinFrictionCompression = 5 KPa;

SoilData1.soilShearModulus = -1 KPa;

SoilData1.poissonRatio = 0.5;

SoilData1.peakTipStress = 0 KPa;

SoilData1.peakTipStressDisplDiamRatio = 0.05;

SoilData1.peakSkinFrictionDisplDiamRatio = 0.01;

3.3.3.6 New Soil Curves

Purpose: To define soil curves properties. There are three sets of curves to choose between:

pyAPI1984	tzAPI1993	qzAPI1993
pvAPI1987	tzAuto Bilinear	gzAutoBilinear
pyAutoBilinear	tzAutoLinear	qzAutoLinear
pyAutoLinear	tzAuto Trilinear	gzAuto Trilinear
pvAuto Trilinear	tzKraft	gzElastic Theory
pyDNV1980	tzManual	gzManual
pyISO2004		
pyManual		

Notice that the manual option has been disabled.

This command is scripted. Typically with relevant data as shown below:

SoilCurves2 = SoilCurves(pyAPI1984,tzAPI1993,qzAPI1993);

🚺 Create/Edit Soil Propertie:	5								— X
Soil Data Soil curves									
• New C Edit existing	oilCurves2								
P-Y curve type pyAPI T-Z curve type tzAPI Q-Z curve type qzAPI Manual P-Y Manual T-Z M	1984 • • • • • • • • • • • • • • • • • • •) Adia	uet 0-7	,1					
Diameters: Remove current o Remove all curr	▼ [m] ♀? urve ♀? res ♀?			·				Г	Show current curve
P [Pa]	Y [m]	1	Ger	niE V5.	3-09 Date:	01 Apr 20	011 12:59:26	3	<u></u>
1 2 3 4 5 6 7 8 9 10				0 0.5 1	0	0.2	0.4	0.6	0.8 1
					OK		Can	cel	Apply

oli Data Soli curves					
New C Edit existing SoilData1					
Initial value of soil shear modulus				[Pa]	<mark>%</mark> ??
Soil poisson ratio				×	<mark>8</mark> ?
Peak skin friction — Constant — In compression In tension			Z given relative to se	eabed	83
[Pa] ×	[Pa]	×	Z1	[m]	×
[Pa] ×	[Pa]	×	Z2	[m]	×
Ratio between displacement to reach peak skin friction and pile diameter				×	83
Tip resistance					
Peak tip stress			0 Pa	[Pa]	83
Ratio between displacement to reach peak tip stress and pile diameter				×	83

164

Version 5.3

3.3.4 Water

Purpose: To define water properties that can be used when defining a condition. The water properties may also be defined from the input dialogue for a condition – the water properties will be found under the subfolder Water.

🖃 💼 UME Name Description Angle [deg] Analysis Environment Properties... Air Directions New Current Profile ... New Frequency Set ... New Phase Set ... 🦲 Soil Wave Height ۲ 🔄 Water New Regular Wave Set ... 🗄 💼 Equipment 🗄 🧰 Properties Paste Fields ... 🗄 🧰 Structure 🗄 🛅 Utilities Save HTML Report ...

3.3.4.1 Properties

Purpose: To define or modify water properties.

This command is scripted. The scripting depends on which parameters are modified or entered to create a new water property, see below.

Properties	
Current Profile Base	Phase Set Frequency Set Wave Height Regular Wave Set
Name	Use Description
<none></none>	no current profile
Create/Edit C	urrent Profile Base
	OK Cancel Apply

3.3.4.2 New Current Profile

Purpose: To define a current property.

This command is scripted. Typically:

CurrentProfile2_Elevations = Array(30 m,0 m,-42 m,-124 m);

CurrentProfile2_Directions = Array(0 deg,0 deg,0 deg,0 deg,0 deg);

CurrentProfile2_Velocities = Array(1.1 m/s,1.1 m/s,0.5 m/s,0 m/s);

CurrentProfile2 = CurrentProfileRelDir(CurrentProfile2_Elevations, CurrentProfile2_Directions, CurrentProfile2_Velocities,dtAlongHeading);

🎎 Create/Edit Cur	ent Profile			×
Current profile				
• New C Edit exis	ng Current Profile	2		
O Direction relative to	vave heading 🛛 🔿	Direction relativ	e to x-axis	
Specify profile: 💡?	۲	Direction along	wave heading	
Z=Hwave Z	z [m] [Direction [deg	Velocity [m/s]	
	_			-
3				
	_			-
3				
Z=-Depth				~
<u>,</u>		Ca	ncel Apply	

3.3.4.3 New frequency set

Purpose: To define a frequency set. The input may be in period (s), wavelength (m) or frequency (rad/s).

This command is scripted. Typically for period input: First 2 s, last 18 s, step 2 s:

FrequencySet1 = *FrequencySet(WavePeriod(),Array(2 s,4 s,6 s,8 s,10 s,12 s,14 s,16 s,18 s));*

11 April 2011

👪 Frequency Set	X
S Name FrequencyS	etl 🗸
C Period C Wavelengt	h 🕫 Frequency
First value:	[rad/s] 🛪
Lastvalue:	[rad/s] 🛪
Step value:	[rad/s] 🛪
Fill table	Row operations: 💡
Specify value: 8?	Insert Remove
Val	ue [rad/s]
1	
3	
4	
5	
7	
8	
10	
Clear table OK	Cancel Apply

3.3.4.4 New Phase Set

Purpose: To define a phase set. The default values are shown to the right.

This command is scripted. Typically for first angle 0 deg, last angle 90 deg and angle step 15 deg:

PhaseSet1 = PhaseSet(Array(0 deg, 15 deg, 30 deg, 45 deg, 60 deg, 75 deg, 90 deg));

👪 Phase Angles S	et	X
🕭 Name 🖻	haseSet1	•
First angle: Last angle: Angle step:	0 345 15	[deg] [deg] [deg]
Specify phase:	83 83	Row operations: 97
1 2 3 - 4 - 5 - 6 - 7 - 8 - 9 - 10 -	Pi	hase [deg]
Clear table	OK	Cancel Apply

GeniE

11 April 2011

Version 5.3

3.3.4.5 New Wave Height

Purpose: To define the wave height through a function, surface or set.



3.3.4.5.1 New Wave Height Function

Purpose: To define the wave height with corresponding wave periods and heights.

This command is scripted. Typically for three corresponding sets

WaveHeights1 =
WaveHeightFunction(Array(5 s,7 s,9 s),
Array(10 m,12 m,11 m));

🚺 Create/Edit Wa	ave Height		×
Wave Height Funct	ion Wave Height Surface	Wave Height Set	
🗞 Name Wave	eHeights1	•	
Periods and wave	heights: 🔗?		👯 陷 월
Wave Pe	riod Wave Height [m]	GeniE Wave Hei	ght Function [®]
1 2 3 4 5 6 7 8 9 10		Mono United States of the stat	0.4 0.6 0.8 1 od T
	ОК	Cancel	Apply

3.3.4.5.2 New Wave Height Surface

Purpose: To specify the wave heights as a function of wave heading and period.

👗 Crea	te/Edit Wave H	leight												x
Wave H	leight Function	Wave Height	Surface Wa	ve Height Set										
🏷 N	ame WaveHei	ghtSurface1											•	
First wa	ve period:	2s	[s]											
Wave	period step:	1s	[s]											
Numbe	r of wave periods	s: 20												
First wa	ive heading:	-180	[deg]											
Wave	neading step:	30	[deg]											
Numbe	r of wave headin	gs: 12												
Wave	heights: 💡?													
	Period\Headi	-180 deg	-150 deg	-120 deg	-90 deg	-60 deg	-30 deg	0 deg	30 deg	60 deg	90 deg	120 deg	150 deg	<u>^</u>
1	2 s													
2	35													=
4	5.8													
5	6 s													
6	7 s													
7	8 s													
8	9 s													
											ок	Cancel	Apply	,

166

DET NORSKE VERITAS SOFTWARE
Varsian 5.3

167

This command is scripted. Typically when using the input parameters as shown to the right:

WaveHeightSurface1 = *WaveHeightSurface*(2s, 1s, 2, 0, 30, 3);

WaveHeightSurface1.set(2 s,0 deg,5 m);

WaveHeightSurface1.set(2 s,30 deg,6 m);

WaveHeightSurface1.set(2 s,60 deg,7 m);

WaveHeightSurface1.set(3 s,0 deg,10 m);

WaveHeightSurface1.set(3 s,30 deg,11 m);

WaveHeightSurface1.set(3 s,60 deg,12 m);

🎎 Create/Ec	dit Wave I	Height		×					
Wave Height Function Wave Height Surface Wave Height Set									
Name WaveHeightSurface1									
First wave pe	First wave period: 2s [s]								
Wave period	step:	1s	[s]						
Number of wa	ave period	s: 2							
First wave he	ading:	0	[deg]						
Wave headir	ng step:	30	[deg]						
Number of wa	ave headir	ngs: 3							
Wave heigh	ts: 💡?								
Per	iod\Headi	0 deg	30 deg	60 deg					
1 2 s		5 m	6 m	7 m					
2 3 s 10 m			11 m	12 m					
OK Cancel Apply									

3.3.4.5.3 New Wave Height Set

Purpose: To define a set of wave heights.

This command is scripted. Typically when using first height 2m, last height 20m and step height 2m:

WaveHeightSet1 = WaveHeightSet(Array(2 m, 4 m, 6 m, 8 m, 10 m, 12 m, 14 m, 16 m, 18 m, 20 m));

Create/Edit Wave Height
Wave Height Function Wave Height Surface Wave Height Set
Name WaveHeightSet1
First height: [m] X
Step height: [m] ×
Fill table 🔗? Row operations: 🔗?
Specify value: 9? Insert Remove Clear table
Value [m]
1
2
4
5
6
8
9
10
OK Cancel Apply

3.3.4.6

Purpose: To define a regular wave set for use when defining a wave condition.

New Regular Wave Set	
Regular Wave Set	

Name	WaveSet1	1		-			
Wave Set T Period	ype C Wave	length C f	Frequency	<u>8</u> ?		-Height/Amplitude	olitude <mark>8</mark> ?
Fill tools	C Heig	ght C P	hase	C Direc	tion	<u>8</u> 5	
Sequence					_ Singl	le value	
First value	ə:		[s] 🛪	\$	Value	e:	[s] 🛪
Lastvalue	ə:		[s] 🛪	¢		Fill all	83
Step valu	e:		[s] 🛪	c l			
	1					Fill selected	<u>8</u> .5
Fill tab		combine all wit	ih all	<u>8</u> ;		Fill equal componen	ts 💡?
Specify value	. <u>0</u> 2						
Period [s]	Height [m]	Phase [deg]	Direction [deg]			
				,			

This command is scripted.	
Typically for input parameters as shown	
to the right:	

	Period [s]	Height [m]	Phase [deg]	Direction [deg]
1	14 s	26 m	-60 deg	0 deg
2	12 s	15 m	-60 deg	90 deg
3	14 s	28 m	-60 deg	180 deg
4	15 s	27 m	-60 deg	270 deg
5				

WaveSet2 = RegularWaveSet();

WaveSet2.heightType(rwsHeight);

WaveSet2.add(RegularWave(0 deg,26 m,WavePeriod(14 s),-60 deg));

WaveSet2.add(RegularWave(90 deg,15 m,WavePeriod(12 s),-60 deg));

WaveSet2.add(RegularWave(180 deg,28 m,WavePeriod(14 s),-60 deg));

WaveSet2.add(RegularWave(270 deg,27 m, WavePeriod(15 s),-60 deg));

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	169	11 April 2011

3.4 Equipment

1 st level browser	2 nd level browser	3 rd level browser	Available commands
Equipment			New Equipment
	Weight list		Import weight list

3.5 Properties

1 st level browser	Available commands	Available sub-commands
Common for all	Fields	
	Save HTML Report	
Beam Types	Beam Types	
	New Beam Type	
	Colour code all visible properties	
Compartment Content	Contents	
	New Content	
	Colour code all visible properties	
Corrosion Addition	Corrosion Additions	
	New Corrosion Addition	
	Colour code all visible properties	
Effective Flange	Effective Flange	
	New Effective Flange	
	Colour code all visible properties	
Hinges	Hinges	
	New Hinge	
Hydro	Properties	
	New Flooding	
	Morison	New Morison Constant
		New Morison Diameter
		New Morison Reynold
		New Morison KC
		New Morison By Rule
	Marine Growth	New Marine Growth Constant
		New Marine Growth Z-level
	Air Drag	New Air Drag Constant
		New Air Drag Reynold
	New Hydrodynamic Diameter	New Hydrodynamic Diameter

11 April 2011

Version 5.3

	New Conductor Shielding	New Conductor Shielding
	New Element Refinement	New Element Refinement
Load Interfaces	Load Interfaces	
	New Load Interface	
Mass Density Factors	New Property	
Materials	Materials	
	New Material	
	Colour code all visible properties	
Mesh	Mesh Properties	
	New Mesh Properties	
	Colour code all visible properties	
Mesh Options	Mesh Options	
	New Mesh Option	
	Colour code all visible properties	
Permeability	Permeability	
	New Permeability	
	Colour code all visible properties	
Pile Characteristics	Pile Characteristics	
	New Pile Characteristics	
	Colour code all visible properties	
Plate Types	Plate Types	
	New Plate Type	
	Colour code all visible properties	
Reinforcements	Reinforcements	
	New Reinforcement	
	Colour code all visible properties	
Sections	Sections	
	New Section	
	Colour code all visible properties	
Structure Type	New Structure Type	
	Structure Type	
	Colour code all visible properties	
Thicknesses	Thicknesses	
	New Thickness	
	Colour code all visible properties	

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	171	11 April 2011

1 st level browser	Available commands	Available sub-commands
Wet Surface	Wet Surfaces	
	New Wet Surface	
	Colour code all visible properties	

3.6 Structure

1 st level browser	2 nd level browser	3 rd level browser	Available commands
Structure			New Beam
			New Plate
			New Support
			New Joint
	Features		New Feature Edge
	Point Masses		New Point Mass
	Supports		New Support
			New Support Rigid Link

3.7 Utilities

1 st level browser	2 nd level browser	3 rd level browser	Available commands
Utilities			
	Evaluators		New Linear Slicer
	Guiding Geometry		New Guide Plane
			New Guide Point
		Curves	New Poly Curve
			New Guide Line
		Points	New Guide Point
		Profiles	New Profile
		Transformations	New Transform
	Mesh Priorities		New Mesh Priority
	Model Views		
	Reports		
	Sets		New Set

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	172	11 April 2011

3.7.1 Mesh Priority Sets

Purpose: To define mesh priority sets that can be used during meshing. A mesh priority set will denote the meshing sequence.

A mesh priority set is defined from the command *New Mesh Priority* by right clicking the sub-folder Mesh Priorities

There can be several mesh priority sets, but only one can be used for a meshing activity.

When pushing the button "Add Mesh Priority" a new priority is created. There can be many mesh priorities.

Each priority level contains structural parts and it is possible to move these up and down. The "Add selection" requires that the structural parts are selected in advance. Remember to click the button "Add Selection" at the far right to include the selection in the priority. The "Select in graphics" allows you to select during the priority definitions; the "Add Selection" button must also be used in this case.

This command is scripted. Typically for a mesh priority set with three priorities containing Bm1, Bm10 and Bm100 respectively: MeshPriority1 = MeshPriority(); MeshPriority1.addMeshPriority(); MeshPriority1.meshPriority(1).add(Bm1); MeshPriority1.addMeshPriority(); MeshPriority1.meshPriority(2).add(Bm10); MeshPriority1.addMeshPriority(); MeshPriority1.meshPriority(3).add(Bm100);

👪 Create Mesh Priority Set	×
Name: MeshPriority1	
	Add Mesh Priority
	Move Up
	Move Down
	Add Selection
[<u> </u>	Cancel

🎎 Create Mesh	Priority Set	×
Name: Mesh	Priority1	
Priority 1		Add Mesh Priority
Priority 2		Move Up
At nony .	Add selection	Move Down
	Remove priority levels	Add Selection
	Move Up	
1	Move <u>D</u> own	
	Select in graphics	Cancel

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	173	11 April 2011

4. CONTEXT SENSITIVE MENU

This Chapter describes how the context sensitive menu works as well as lists those commands only available from this menu. The context sensitive menu is activated by selecting an object(s) from the browser or from the graphics.

Example of context sensitive menu for a beam selected from the browser. Select the beam and right click.



Example of context sensitive menu for a beam selected from the graphics. Select the beam and right click.

The context sensitive menu differs from object to object. But they have several equal commands and the object type "beam" is used to exemplify the context sensitive menu.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	174	11 April 2011
4.1 Object type Beam Note that Edit Beam, Centre of Gravity, I	Move, Join Beams, Join Se	Edit Beam Centre of Gravity egments, Cover from pulldown Move
or from toolbar menus.		Generate Joints Join Divide Cover Curves Delete Rename
		Properties Beam Result Diagrams
		Labels > ColorCode >

Mesh Locking
Named set...
View options...
Visible model

4.1.1 Edit Beam

Purpose: To modify beam specific parameters like local system, offset vector, hinges, split points, move ends, translate and buckling factors.

These commands are scripted. The content depends on which operations are carried out. See below for examples.

4.1.1.1 Local system

Purpose: To modify the local system of a beam.

This command is scripted. Typically when rotating the beam Bm21 90 degrees around the beam local x-axis: Bm21.rotateLocalX(90);

Edit Beams
Local System Offset Vector Hinges Split Points Move End Translate
Local system interpretation Guide local system @? Surface normal @? Explicit local system @? Relative to plate @?
Rotation
C Rotate around local X-axis [deg]
Flip local X (preserving the Y-vector) Flip local X (preserving the Z-vector)
Specify local coordinate system
C Local X vector Vector3d(0 m,1 m,0 m)
C Local Y vector Vector3d(-1 m,0 m,0 m)
C Local Z vector Vector3d(0 m,0 m,1 m)
Local System Local System (Vector3d(0 m,1 m,0 m), Vector3d(
Cancel Apply

Version 5.3

4.1.1.2 Offset vector

Purpose: To add eccentricities to the beam ends.

This command is scripted. Typically when specifying an eccentricity of 5cm in negative global z-direction for beam Bm21:

Bm21.setBeamOffset(Vector3d(0 m, 0 m, -5cm));

4.1.1.3 Hinges

Purpose: To add a hinge to a beam end. The hinge properties are defined from the *Edit/Property/Hinge* or from the browser folder *Properties/Hinges*.

This command is scripted. Typically when adding the hinge "Hinge1" to end1 of Bm1:

Bm21.setEndHinge(1,Hinge1);

4.1.1.4 Split points

Purpose: To add additional vertices (or snap points) to a beam.

This command is scripted. Typically when distributing 4 points evenly of Bm31:

Bm31.splitAt(0.2);

Bm31.splitAt(0.4);

Bm31.splitAt(0.6);

Bm31.splitAt(0.8);

Edit Beams	× ×
Local System Offset Vector Hi	Hinges Split Points Move End Translate Buc 💶 🕨
- losert new solit points	
Distribute:	split points evenly
Single split at parameter [0]	[0,1] 0.5
Show All Points	
<u> </u>	
	OK Cancel Apply



[m]

Cancel

Apply



Local System Offset Vector Hinges Split Points Move End Translate Buc

Whole beam

Flush top
 No offset (centric)
 Flush Bottom
 Add constant value

ΟK

11 April 2011

Specify offset vector:

GeniE

175

🔲 Edit Beams

Align section:

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	176	11 April 2011

4.1.1.5 **Move End**

Purpose: To move one beam end. The end point can be moved to another point, by a vector definition or extended/reduced along its own local x-axis. In the latter case a positive value means extension of the beam while a negative value means a reduction of the length.

This command is scripted. Typically when reducing the length with 150 cm relative to the end point 2 of *Bm31*:

Bm31.extendEnd(2, -150 cm);

Similarly, when moving the end1 a vector of (0,5m,0):

Bm31.moveEnd(1, *Vector3d*(0 m, 5 m, 0 m));

Edit Beams Local System | Offset Vector | Hinges | Split Points | Move End | Translate | Buckli End Point Displacement as distance along beam ΟK Cancel Apply

4.1.1.6 Translate

Purpose: To move a beam with a translation vector.

This command is scripted. Typically when moving *Bm21 2000mm in positive global z-direction):*

Bm31.extendEnd(2, -150 cm);

Bm21.moveTranslate(Vector3d(0m,0m,2000mm));

4.1.1.7 **Buckling factors**

Purpose: To define local bucking factors for individual beams. These factors can be referred to when defining a capacity model. These factors are also transferred to Framework if you want to do code checking based on older code checking standards. See User Manual Volume 4 for more details.

This command is scripted. Typically when specifying buckling factor of 0.9 and 0.8 for buckling around local y and z-axis.

Bm21.buckling = BucklingFactor(0.9, 0.8);

Edit Beams
Local System Offset Vector Hinges Split Points Move End Translate Buckling
Translation vector
×
OK Cancel Apply

Edit Beams	
Offset Vector Hinges Split Points Move End Translate	Buckling Factors
Buckling factors in beam local system	
ky 🛐	
Kemove buckling factor from selection	
ОК	Cancel Apply

11 April 2011
DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	177	11 April 2011

4.1.2 Centre of Gravity

Purpose: To compute the mass and centre of gravity.

This command is not scripted.

Compute Mass/Centre Of Gravity			
Visual Feedback De-select any selected objects that do not contribute Highlight computed centre of gravity			
Number of significant digits: Values for selected objects, relat Mass: Press Apply to compute COG: Press Apply to compute Ixx: n/a Iyy: n/a Izz: n/a	6 tive to Centre Of Gravity (COG) lxy: n/a lxz: n/a lyz: n/a		
	Close Apply		

4.1.3 Copy/Move Beam

Purpose: To copy or move a beam using translation, rotation, mirroring, 3 point position or a general transformation.

These commands are scripted. The content depends on which operations are carried out. See below for examples.

4.1.3.1 Translate

Purpose: To copy a beam using a translation vector.

This command is scripted. Typically copying Bm1 with the shown vector:

Bm2 = Bm1.copyTranslate(Vector3d(0 m, 0 m, 10 m));

4.1.3.2 Rotate

Purpose: To copy a beam by defining a rotation point, a vector and a rotation angle.

This command is scripted. Typically copying Bm1 around point (0,0,0) with vector (0,0,1) and angle 90 degrees:

Bm103 = *Bm21.copyRotate*(*Point*(0 m, 0 m, 0 m), *Vector3d*(0,0,1),90 *deg*);

🗖 Сору 🛛 🔀
Translate Rotate Mirror 3 Point Position General transformation
Translation vector Vector3d(0 m,0 m,10 m)
Copy 1 time(s)
Preview Cancel Apply



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	178	11 April 2011

4.1.3.3 Mirror

Purpose: To copy a beam by defining a mirror plane built up from a point and a normal vector.

This command is scripted. Typically copying Bm1 around point (0,5,0) with vector (0,1,0):

Bm2 = *Bm1.copyMirror(Point(0 m,5 m,0 m), Vector3d(0,1,0));*

A typical move operation would be

autoMSet = Set();

autoMSet.clear();

autoMSet.add(Bm1);

autoMSet.moveMirror(Point(0 m,5 m,0 m), Vector3d(0,1,0),geUNCONNECTED);

Delete(autoMSet);

4.1.3.4 Scale

Purpose: To copy a beam by defining a scaling factor and a point to be used as the scaling centre.

This command is scripted

A typical scale operation would be

Beam1_1 = Beam1.copyScale(5,Point(0,0,0));

Beam2_1 = Beam2.copyScale(5,Point(0,0,0));

💽 Сору		X	
Translate Rotate Mirror Scale 3 Point Position General transfo			
Scaling factor:		×	
Scaling center:		×	
Choose the bounding-box center			
<u> </u>			
Prev	riew Cancel Appl	у	



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	179	11 April 2011

4.1.3.5 **3** Point Position

Purpose: To copy beams by defining a source plane and a destination plane. For both planes three points must be defined.

This command is scripted. Typically copying Bm1and Bm2 using the following input parameters





Bm11 = *Bm1.copy3Point*(*Point*(27 m,0 m,10 m),*Point*(27 m,0 m,4 m),*Point*(27 m,28.5 m,10 m), *Point*(37 m,0 m,10 m),*Point*(37 m,0 m,4 m),*Point*(37 m,28.5 m,10 m));

Bm12 = *Bm2.copy3Point(Point(27 m,0 m,10 m),Point(27 m,0 m,4 m),Point(27 m,28.5 m,10 m), Point(37 m,0 m,10 m),Point(37 m,0 m,4 m),Point(37 m,28.5 m,10 m));*

4.1.3.6 General transformation

Purpose: To copy a beam by defining a general transformation. For examples, see Section 3.3.12 of User Manual Volume 3.

This command is scripted depending on which operation is carried out.

🗖 Сору		
Translate Rotate Mi	irror 3 Point Position General transformation	
Transformation:	▼ X	
	Create Translation	
Translate	Translation vector:	- x
C Mirror		
C Scale C 3 Point Position		
C Move Point C Combine		
Create		
<u> </u>		
,	V Preview Cancel	Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	180	11 April 2011

4.1.4 Generate Joints

Purpose: To generate joints based on the rules as defined in the Edit|Rules|Joint creation.

This command is scripted. Typically when selecting a beam that is subject to definition of four joints based on the rule definition:

Jt1 = Joint(Point(4 m,0 m,10 m)); Jt2 = Joint(Point(12 m,0 m,10 m)); Jt3 = Joint(Point(18 m,0 m,10 m));

Jt4 = *Joint*(*Point*(24 m,0 m,10 m));

Joint Creation Rule		
Automatic joint generation	ОК	
Selection-aware classification	Cancel	
Exclusion Criteria Exclude pure through-beam intersectio	ns	
Exclude 2-beam aligned intersections		
🔽 Exclude free beam ends		
Exclude intersections with beam ends		
Exclude intersections with through-bea	ims	

4.1.5 Join

Purpose: To join beams and segments.

This command is scripted. Typically when joining Bm144 and Bm145:

Bm144.joinBeams(Bm145);

Bm144.joinSegments(1,2);

tå Join	
 Join Beams and Segments Join Plates Join Guide Curves 	Apply Close
I Join only if concepts have same set membership ∫ Join Beams and Segments	
 ✓ Join Aligned Beams ✓ Join Beam Segments ✓ Join all segments 	
 Join only segments with compatiple section 	on/material

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	181	11 April 2011

4.1.6 Divide

Purpose: To divide beams and segments using a number of division criteria.

This command is scripted. Typically when dividing Bm1 with an X-plane at x = 24 m (there are now two beams Bm1 and Bm2):

Bm2 = Bm1.divide(XPlane3d(24 m));

Jå Divide	
Divide Beams and Segments Divide Support Curves Divide Plates Divide Guide Curves Explode all structure in selection into simpler parts Divide Reams	Apply Close
 Explode all beams in selection into simpler beams Divide beams at position Create segment border at position Divide beams with plane Divide beams with structure 	

4.1.7 Cover Curves

Purpose: To create a plate by covering the area enclosed by the selected beams.

This command are scripted. Typically: Pl1 = CoverCurves(Beam1,Beam2,Beam3,Beam4);

4.1.8 Delete

Purpose: To delete the selection.

This command is scripted. Typically: Delete(Bm1); Delete(Bm2);

Delete Confirmati	on	x
? Are y	ou sure you want to delete these 2 items?	
	Yes No	

4.1.9 Rename

Purpose: To rename the selected item.

This command is scripted. Typically: Rename(Beam3, "Beam3_renamed");

Rename Obje	ct	X
Descripti	on: Straight Beam	
Name: Beam3		
	Cancel	ОК
·		

4.1.10 Properties

Purpose: To add or modify properties to the selected beam.

These commands are scripted. Typically when adding profile type HE600A to Bm1:

Bm1.section = *HE600A*;

Properties					X
Object Properties	Section	Material	Thickness	Effective Flange	Corrosior 1
Name	Use	Descri	ption	Diamet	er 🔺
<none></none>		no sec	tion		
Bar100	0	Bar Se	ection		
I 1200	50	I Secti	ion, syste		E
1 400	0	I Secti	on		
I 1600	0	I Secti	on		
1 1800	0	I Secti	ion		
1 11000	0	I Secti	ion		
11200	0	I Secti	ion		
1 11400	0	I Secti	ion		
T ISEC100	22	I Secti	ion		*
					4
Create/Edit Section Set Default					
OK Cancel Apply					

4.1.11 Beam Result Diagrams

Purpose: To open the "Beam Deflection, Force and Stresses Display" dialog.

This command is not scripted.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	183	11 April 2011

4.1.12 Labels

It is possible to label the following parameters. To remove the labels from the graphic view, use the "Clear labels" option or click the refresh button:

- Edit Beam... **Buckling Factor** _ Centre of Gravity ... Coordinates Copy... _ Move... Description _ Generate Joints Eccentricities Join... _ Divide... **Eccentricity Symbols** _ Cover Curves Delete... Ends _ Rename... Ends, X _ Properties... Beam Result Diagrams... Ends, Y _ **Buckling Factor** Labels ₽ Ends, Z _ ColorCode Coordinates ₽ Mesh Locking Description Hinge names ۶ _ Eccentricities Named set... Hinges _ Eccentricity Symbols View options... Ends Length Visible model ₽ _ Ends, X Local Coordinate System Ends, Y _ Ends, Z Name _ Hinge names Hinges Properties Length Beam Type Local Coordinate System Mesh Lock Coordinates **Corrosion Addition** Name Properties Þ Beam Type Hydro Air Drag Propertyvalues **Corrosion Addition** ۲ Hydro Diameter Clear Labels Effective Flange Hydro Air Drag Hydro Element Refinement Hydro Diameter Hydro Element Refinement Hydro Flooding Hydro Flooding Hydro Marine Growth Hydro Marine Growth Hydro Morison Hydro Morison Hydro Shielding Hydro Shielding Material Number Of Elements Material Reinforcement Section Number Of Elements .
 - Reinforcements
 - Section

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	184	11 April 2011
 Property Values Mesh Density Number Of Elements Thickness Width Clear Labels 	B C D E E E E H H L L L N N P P C C	uckling Factor ioordinates lescription ccentricities ccentricity Symbols nds nds, X nds, Y nds, Z linge names linges ength ocal Coordinate System Mesh Lock Coordinates lame roperties roperty values idea Labels Mesh Density Number of Elements Thickness Width

4.1.13 Colour Coding

It is possible to do colour coding of the following parameters. To view the structure without the colour coding you can click on the paint brush symbol to toggle on/off colour coding.

- Properties
 - Beam Type
 - Corrosion Addition
 - Effective Flange
 - Hydro Air Drag
 - Hydro Diameter
 - Hydro Element Refinement
 - Hydro Flooding
 - Hydro Marine Growth
 - Hydro Morison
 - Hydro Shielding
 - Material
 - Number Of Elements
 - Property
 - Reinforcements
 - Section
 - Thickness



Version 5.3

11 April 2011

- Property Values
 - Thickness
 - Width



4.1.14 Mesh Locking

It is possible to lock or unlock the mesh.

- Lock
- Unlock

These commands are scripted.



4.1.15 Named Set

Purpose: To include or remove a selection in a set.

This command is scripted. Typically when including two beams in the set "MySet"

MySet = *Set();*

MySet.add(Bm1);

MySet.add(Bm2);



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	186	11 April 2011

4.1.16 View Options

Purpose: Access to the same view settings as defined under the *View/Option* pulldown menu.

These commands are not scripted.

🗖 View Options 🛛 🔀			
General Settings Mouse Color Coding Annotation / Diagrams Browser			
 Model Capacity Models Color code legend Environment Evaluators FEM Hydro Loads and Equipment Structure Utility Utility, Selection Working Set, Active Working Set, Inactive 	Property	Value	Defined where
Default display Save As Delete Restore defaults			
OK Cancel Apply			

4.1.17 Visible Model

Purpose: To decide which parts of a model to show in the graphic view. It is possible to:

- Show selection only (Alt + S)
- Add selection (Alt + Plus)
- Remove selection (Alt + Minus)
- Show All (Alt + A)
- Show Complement (Alt + Q)

Edit Beam		
Centre of Gravity		
Сору		
Move		
Generate Joints		
Join		
Divide		
Cover Curves		
Delete		
Rename		
Properties		
Labels 🕨		
ColorCode 🔹 🕨		
Mesh Locking 🔹 🕨		
Named set		
View options		
Visible model 🔹 🕨	Show selection only	Alt+S
	Add selection	Alt+Plus
	Remove selection	Alt+Minus
	Show all	ALT+A
	Show complement	ALT+Q

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	187	11 April 2011

4.2 Object type Plate

Note that Centre of Gravity, Move, Flip Normal, Labels, Named Set, and Visible Model are not available from pulldown or from toolbar menus.

4.3 Object type Equipment

Note that Place, Place a Copy, Named Set, Visible Model are not available from pulldown or from toolbar menus.

4.4 Object type Joint

Note that most of these are not available from pulldown or from toolbar.

Сору
Move
Add Can/Stub
Add Gap
Flush braces
Split braces
Add LJF
Remove joint eccentricities
Select Cans
Select Stubs
Select connected beams
Delete
Rename
Properties
Labels •
ColorCode •
Named set
View options
Visible model





DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	188	11 April 2011

4.5 Object type Support Point

Note that Move, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus.

4.6 Object type Support Curve

Note that Move, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus.

4.7 Object type Explicit Load

Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus

4.8 Object type Guide Plane

Note that Move, Named Set, Visible Model are not available from pulldown or toolbar menus.







Сору ...



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	189	11 April 2011

4.9 Object type Guide Line

Note that Move, Create beam, Create Feature Edge, Create Support Curve, Join Curves, Cover Curves, Labels, Named Set, Visible Model are not available from pulldown or toolbar menus

Copy
Move
Create Beam
Create Support Curve
Create Feature Edge
Edit GuideLine
Join
Divide
Cover Curves
Delete
Rename
Properties
Labels •
ColorCode •
Named set
View options
Visible model

4.10 Object type Water Surface

Edit WaterSurfac	e
Labels	•
ColorCode	►
Named set	
View options	
Visible model	•

4.11 Object type Water Layer

Edit Water Layer	
Labels	•
ColorCode	•
Named set	
View options	
Visible model	•

4.12 Object type Air Layer



4.13 Object type Seabed

Edit Seabed	
Properties	
Labels	•
ColorCode	۲
Named set	
View options	
Visible model	۲

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	190	11 April 2011

4.14 Object type Soil Layer

Edit Soil Layer Properties	
Labels ColorCode) }
Named set View options Visible model	•

4.15 Object type Soil Border



4.16 Object type Mesh

Labels	Description
ColorCode	Element numbers
Mesh Locking	Mesh Lock Coordinates
Named set	Name
View options	Node numbers
Visible model	Node symbols
Visible model 🕨	Relative Jacobi Clear Labels





DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	191	11 April 2011

4.17 Object Type Beam Capacity Member

Examples on the context sensitive menu for capacity members are shown in the following:





Properti	25											x
Object Prop	erties Redesig	n										
Run:	Run: Cc1.allRuns Loadase:											
Recal	Recalculation history I Automatic Recalculate I Colorcode UPtot No recalculation done/selected. Image: Second control of the second											
Membe	Position Range	Position	Section	Material	Buckling Length,Factor	Stiffener Spacing [m]	Status	UfTot	Formula	GeomCheck	SubCheck	L A
Beam24	-0.00 - 1.00	1.00	I200 💌	St48 💌	KL(5 m, 1) 💌	5	ОК	0.62	ufH1	Geom OK	AISC member	U
												•
<u>,</u>								OK		Cancel	Apply	

11 April 2011

Version 5.3

Redesign... Properties...

ColorCode

Named set... View options... Visible model

Labels

4.17.2

60

۶

۶

۲

Capacity Member Properties

Object Properties Edit Mem	ber Data	
		0-
Buckling length	Member Length	\ ?
Effective length factor	0.95	
Moment amplification	1	
about z-axis	Member Length	1
Effective length factor	1	
Moment amplification		
Stiffener Spacing 💡?	None 💌 [m]	1
Lateral torsional buckling modification	1 💌	
Effective length factor for torsional buckling	1	
- Length between lateral	supports	
Top flange	None 👻 [m]]
Bottom Flange	None [m]	I
L section specific		<u>8</u> ?
Chapter E5: C M	ethod a 💿 Method b	
Connected to: 💽 Lo	ong leg 🗢 Short leg	

4.17.3 Capacity Member Labels



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	193	11 April 2011



4.17.3.1 Buckling Factor







DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	194	11 April 2011



4.17.3.5 Results

As an example we will take CapModel to show



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	195	11 April 2011

4.17.3.6 Clear Labels

This clears all labels from the graphical display. It gives the same result as clicking the exclamation mark button.

4.17.4 Capacity Member Color Code



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	196	11 April 2011

4.18 Object Type Plate Capacity Panel

Examples on the context sensitive menu for capacity members are shown in the following:



4.18.1 Capacity Panel Properties

Properties	Properties 🛛
Labels > ColorCode >	Object Properties Edit Panel Data
Named set View options Visible model	Check Buckling for Whole and Subpanels Whole Only Subpanels Only
	Panel Options Correction Factor F1 Sniped Panel Net Thickness Minimum Idealised Panel Panel Length (a) From Idealised Panel Panel Width (b) From Idealised Panel Subpanel Length (a') Two B Rotation Boundary for Panel Buckling Top Simply Supported Left Simply Supported Right Simply Supported Bottom Simply Supported
	OK Cancel Apply

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	197	11 April 2011

4.18.2 Capacity Panel Labels



Note: Buckling Factor only applies to beam members.

4.18.2.1 Capacity Model Local Coordinate System







F	Properties			
L	abels I		Buckling Factor	
0	ColorCode	•	Capacity Model Local Coordinate System	
1	Named set		Description	
	liew ontions		Idealization Method	
	/isible model		LoadCase	
	visible model ,	-	Name	
			Results	•
			Clear Labels	

4.18.2.3 Idealization Method



4.18.2.4 LoadCase







DET NORSKE VERITAS SOFTWARE

Version 5.3

199

11 April 2011

4.18.2.6 Results

As an example CapModel is used to show how details can be labelled.

Capacity Model Local Coordinate System Description LoadCase	
Name	
Results 🕨 🕨	CapModel
Clear Labels	F1
	Formula
	GeomCheck
	LengthA
	NetThick
	SigmaX1
	SigmaX2
	SigmaY1
	SigmaY2
	Status
	SubCheck
	TauXY
	UfBuckComb
	UfBuckComb_withLoads
	UfBuckComb_withSubpanel
	UfBuckSigmaX
	UfBuckSigmaY
	UfBuckTau
	UfTot
	UfTot_LC
	UfYield
	WidthB
	YieldStrength



DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	200	11 April 2011

4.18.2.7 Clear Labels

4.18.3 Capacity Panel ColorCode

The result for NetThick is used as an example to show how colour coding works.





Version 5.3

11 April 2011

4.18.4 Capacity Panel – CSR Tank Specific

Start PULS Advanced Viewer - Current loadcase	
Start PULS Advanced Viewer - All loadcases	
Start PULS Excel Spreadsheet - Current loadcase	
Start PULS Excel Spreadsheet - All loadcases	
Properties	
Labels	•
ColorCode	×
Named set	
View options	
Visible model	►

4.18.4.1 Start PULS Advanced Viewer – Current loadcase

_		_
	Visible model)
	View options	
	Named set	
	ColorCode)
	Labels)
	Properties	
	Start PULS Excel Spreadsheet - All loadcases	
	Start PULS Excel Spreadsheet - Current loadcase	
	Start PULS Advanced Viewer - All loadcases	
	Start PULS Advanced Viewer - Current loadcase	



4.18.4.2 Start PULS Advanced Viewer - All loadcases

Start PULS Advanced Viewer - Current loadcase	
Start PULS Advanced Viewer - All loadcases	
Start PULS Excel Spreadsheet - Current loadcase	
Start PULS Excel Spreadsheet - All loadcases	
Properties	
Labels	•
ColorCode	۲
Named set	
View options	
Visible model	•



4.18.4.3	Start PULS Excel Spreadsheet – Current loadcase
----------	---



	© - (° - -	$\sim \times \times$					PulsExcel	_Cc1.run2.xls	[Compatil	oility Mode] - M	licrosoft Exc	cel		-	-		-		X
File	Home	Insert Pa	age Layout	Formulas	Data	Review	View											∾ 🕜 ⊏	. e X
Paste	X Cut La Copy ▼	Arial	Π - E	• 10 • A		= <mark>-</mark> »	• भा • ख	🐨 Wrap Text	enter -	General	÷.0 .00	Conditional Fo	rmat Cell	← ■ Insert	Delete F	ormat	Σ AutoSum Fill ▼	Sort & Fir	nd &
Ť	Format Pair	nter -	Eant				Alignment			Number	.00 9.0 F	Formatting * as T	able * Styles		Calls	* 0	Clear *	Filter * Sel	lect *
	Δ21	- (m	fr (Cc1 run(2) n	anel(PI905	11) 1 C3	Angriment		14	Number	· 4	Style			Cells			uning	~
	A	C	D	E	F	G	Н		J	К	L	М	N	0	Р	Q	R	S	TE
1		-	_	_	-	-					_			-					^
2 3 4 5 6 7 8	Ĵå dnv	PULS 2	2.0 - U I	nstiffen	ed pan	el (U3)	input										ф		=
9	Clear	input sheet		Import unsti	ffened panel	s from									Cal	culate pa	anel(s)		
10	Define co	murated pa	nel	Calculate ar	ad export pa	nels to				Allowable us	sage facto	r:		Analysis	options:				
12 13 14		inguted pa								Ultimate cap Buckling str	pacity: ηυσ rength: η _{BS}	C S	1 1	Row t Comb	oy row inations of	input	 Delete Save 	e old results old results	
16 ld	ease see comment entification	Geometry	narked with r	red triangles) to	Material	ination of feat	ures/input. A p	Applied loa	ads	the description sr	neet.			Bounda	ry condi	tions			
17	Identification	Length of	Width of	Plate	Modulus of	Poisson's	Yield st.	Axial	Axial	Transversal	Transvers	al Shear	Pressure	In plane	Rotation	al support			
18	of panel	plate L1	L2	thickness tp	elasticity E	ratio ນ	geo geo	G14	Stress	Stress 0.4	Gas	Stress	(fixed)	support	Left	Right	Upper	Lower	
20		mm	mm	mm	MPa		MPa	MPa	MPa	MPa	MPa	MPa	MPa		MNm/m	MNm/m	MNm/m	MNm/m	
21 C 22 23 24	c1.run(2).panel	2400	750	14	206000	0.3	235	0		0 0		0 21.2	() Int	SS	SS	SS	SS	
25 26 27																			
28 29																			
14 4 1	► ► Descriptio	on Stiffe	ned nanel (S3) - Input	Stiffene	d panels (S3) - Output	Unstiffer	ned plate	(U3) - Input /				-				l	▶ []
Ready	beschper	Juli 2 Julie	neu parier (ooy apuc	<u></u>	a panala (55	y output	A onsumer	ica place	(obj input/							100% —		+ ;;
																		-	

Version 5.3

4.18.4.4 **Start PULS Excel Spreadsheet – All loadcases**

_			_
	Visible model		•
	View options		
	Named set		
	ColorCode		•
	Labels		Þ
	Properties	20	
	Start PULS Excel Spreadsheet - All loadcases	2	
	Start PULS Excel Spreadsheet - Current loadc	ase	
	Start PULS Advanced Viewer - All loadcases		
	Start PULS Advanced Viewer - Current loadca	ase	

	9 · © · -	C AX	1 1			* / *	PulsExcel	_Cc1.run2.xls	[Compatibili	ty Mode] - M	icrosoft Exe	cel		m -	-		•		
File	Home	Insert Pa	age Layout	Formulas	Data	Review	View											۵ ()	- 6 X
Ē	∦ Cut	Arial		• 10 • A	. ∧ = =	= 😑 🕺	- Pfl -	🚽 Wrap Text	Ge	neral	٣	≦\$		÷			Σ AutoSum	` 打	A
Paste	Format Pain	nter B Z	<u>u</u> .	- 🌺 -	<u>A</u> · ■ ■			Merge & Ce	enter - 🗐	- % ,	00. 0. 0. 00.	Conditional F Formatting ▼ as	ormat Cell Table * Styles *	Insert [Delete F	ormat	2 Clear *	Sort & Filter *	Find & Select ▼
	Clipboard	Gi .	Font	1	Es.		Alignment		Es.	Number	Es.	Sty	les		Cells			Editing	
	A21	▼ (°	f _x (Cc1.run(2).p	anel(PI905,	11).LC1													
4	A	С	D	E	F	G	H	- I	J	K	L	M	N	0	Р	Q	R	S	T
3 4 5 6 7 8 9	Clear	input sheet		Import unsti	ffened panels	s from	mput								Cal	culate p	anel(s)		[
10																			1
	D 6			o 1 1 1						Allowable us	age tacto	r.		Analysis (options.				
2	Define co	orrugated par	nel	Calculate ar	nd export par	nels to				Allowable us	age facto	ir:		Analysis Row b	options: yrow		Delet	e old result	-
2 3 4	Define co	orrugated par	nel	Calculate ar	nd export par	nels to				Allowable us Ultimate cap Buckling str	age facto acity: ηυσ ength: η _{BS}	r: c s	1 1	Row b Combin	y row nations of	finput	DeletSave	e old results old results	
2 3 4 5 Ple	ase see comment	t boxes (cells n	nel	Calculate ar	nd export par	nels to	tures/input. A p	anel illustration	is found in the	Allowable us Ultimate cap Buckling str e description sh	age facto eacity: ηυσ ength: η _{BS} eet.	r: c s	1 1	Row by Combin	y row	finput	Delet Save	e old results	
2 3 4 5 Ple 6 <mark>Id</mark>	Define co	t boxes (cells n	nel	Calculate ar	or further expla	nels to	tures/input. A p	Applied loa	is found in the	Allowable us Ultimate cap Buckling str e description sh	age facto pacity: ηυα ength: η _{BS} eet.	C S Shear	1 1 Pressure	Analysis Row by Combin Boundary	y row nations of y condinations	finput tions	Delet Save	e old results	
2 3 4 5 Ple 6 Ide 7 8	Define co ase see comment entification Identification of panel	t boxes (cells n Geometry Length of plate	nel narked with r Width of plate	Calculate an red triangles) fi Plate thickness	or further expla Material Modulus of elasticity	nels to nation of fea Poisson's ratio	tures/input. A p Yield st. plate	anel illustration Applied Ioa Axial stress	is found in the ds Axial stress	Allowable us Ultimate cap Buckling str e description sh Transversal stress	age facto pacity: ηυα ength: η _{BS} eet. Transvers stress	c s sal Shear stress	1 1 Pressure (fixed)	Row by Combin Boundary In plane support	y row nations of <mark>y conding</mark> Rotation	finput <mark>tions</mark> al support	Delet Save	e old results	
2 3 4 5 Ple 6 Id 7 8 9	Define co ase see comment entification Identification of panel	t boxes (cells n Geometry Length of plate L1	with of plate	Calculate an red triangles) fo Plate thickness tp	nd export par or further expla Material Modulus of elasticity E	nels to nation of fea Poisson's ratio v	tures/input. A p Yield st. plate G _{Pp}	anel ∎ustration Applied Ioa Axial stress σ _{1,1}	is found in the ds Axial stress σ _{1,2}	Allowable us Ultimate cap Buckling str e description sh Transversal stress G _{2,1}	age facto pacity: ηυ ength: η _{BS} eet. Transvers stress σ ₂₂	r: C S Sal Shear stress τ ₁₂	1 1 Pressure (fixed) p	Analysis (Row b) Combin Boundar In plane support	y row nations of y condi Rotation	finput tions al support Right	Delet Save Upper	e old results old results Lower	
2 3 4 5 Ple 6 Id 7 8 9	Define co	t boxes (cells n Geometry Length of plate L1 mm	width of plate L2 mm	Calculate an red triangles) for Plate thickness tp mm	or further expla Material Modulus of elasticity E MPa	nels to Ination of fea Poisson's ratio U	tures/input. A p Yield st. plate σ _{Fp} MPa	anel illustration Applied Ioa Axial stress σ _{1,1} MPa	is found in the ds Axial stress σ _{1,2} MPa	Allowable us Ultimate cap Buckling str e description sh Transversal stress G _{2,1} MPa	age facto pacity: ηυς ength: η _{BS} eet. Transvers stress σ ₁₂ MPa	sal Shear stress tr ₁₂ MPa	1 1 Pressure (fixed) p MPa	Analysis of Row b Combin Boundar In plane support	y row nations of y condi Rotation Left	finput tions al support Right MN/m	Delet Save	e old results old results Lower MNm/m	
2 3 4 5 Ple 6 Id 7 8 9 9 0 1 Cc	Define co ase see comment entification of panel	t boxes (cells n Geometry Length of plate L1 mm 2400	width of plate L2 mm 750	Calculate an red triangles) fr Plate thickness tp mm 14	or further expla Material Modulus of elasticity E MPa 206000	nels to nation of fea Poisson's ratio 0	tures/input. A p Yield st. plate σ _{Fp} MPa 235	anel illustration Applied Ioa Axial stress σ _{1.1} MPa 31.3	is found in the ds Axial stress σ _{1,2} MPa 31.3	Allowable us Ultimate cap Buckling str e description sh Transversal stress $\sigma_{2,1}$ MPa 5.5	age facto pacity: ηυα ength: η _{BS} eet. Transvers stress σ ₁₃ MPa 5	c sal Shear stress T12 MPa 5.5 - 2.8	1 1 Pressure (fixed) p MPa 0	Analysis of Row bi Combin Boundar In plane support Int	y row nations of y condi Rotation Left <u>MNm/m</u>	finput tions al support Right MNm/m SS	Delet Save	e old results old results Lower MNm/m SS	
2 3 4 5 Ple 5 Ide 7 3 1 Cc	Define co ase see comment intentification of panel 1.run(2).panel 1.run(2).panel	t boxes (cells n Geometry Length of plate L1 mm 2400 2400	width of plate L2 mm 750 750	Calculate ar red triangles) fr Plate thickness tp mm 14 14	nd export par or further expla Modulus of elasticity E MPa 206000 206000	Poisson's ratio 0.3 0.3	tures/input. A p Yield st. plate G _{Pp} MPa 235 235	panel illustration Applied loa Axial stress σ _{1,1} MPa 31.3 0 0	is found in the ds Axial stress σ _{1,2} MPa 31.3 0	Allowable us Ultimate cap Buckling str e description sh Transversal stress $\sigma_{2,1}$ MPa 5.5 0	age facto pacity: ηυα ength: η _{BS} eet. Transvers stress σ ₅₂ MPa 5	r: c s s Shear stress tris MPa 0 8 8	1 1 Pressure (fixed) p MPa 0 0	Analysis of Row by Combined Co	y row nations of y condi Rotation Left <u>MNm/m</u> SS SS	finput tions al support Right MNm/m SS SS	Delet Save Upper MNm/m SS SS	e old results old results Lower MNm/m SS SS	
2 3 4 5 Ple 6 Id 7 8 9 0 1 Cc 3 Cc 3 Cc	Define co ase see comment entification of panel 1.run(2).panel 1.run(2).panel 1.run(2).panel	t boxes (cells n Geometry Length of plate L1 mm 2400 2400 2400	width of plate L2 mm 750 750 750	Calculate an red triangles) fr Plate thickness tp mm 14 14 14	nd export par or further expla Material Modulus of elasticity E MPa 206000 206000 206000	Poisson's ratio 0.3 0.3 0.3	Vield st. plate σ _{Fp} MPa 235 235 235	Applied loa Axial Axial stress σ _{1.1} MPa 31.3 0 0 0	is found in the ds Axial stress σ _{1,2} MPa 31.3 0 0 0	Allowable us Ultimate cap Buckling str e description sh Transversal stress $\sigma_{2,1}$ MPa 5.5 0 0	age facto vacity: ηυι ength: η _{BS} eet. Transvers stress σ ₂₂ MPa 5	r: sal Shear stress τ ₁₅ MPa 0 8 0 21.2	1 1 Pressure (fixed) p MPa 0 0 0	Analysis of Row by Combined Co	y row nations of rotations Rotation Left MNm/m SS SS SS	finput tions al support Right MNm/m SS SS SS	Delet Save Upper MNm/m SS S	e old results old results Lower MNm/m SS SS SS SS	
2 3 4 5 Ple 6 Ide 7 8 9 0 1 Cc 3 Cc 4 5 5 6 7	Define co ase see comment intification of panel 1.run(2).panel 1.run(2).panel	t boxes (cells n Geometry Length of plate L1 2400 2400 2400	hel harked with r yildth of plate L2 mm 750 750 750 750	Calculate ar red triangles) fr Plate thickness tp mm 14 14 14	nd export par or further expla Material Modulus of elasticity E MPa 206000 206000 206000	Poisson's ratio 0.3 0.3 0.3	tures/input. A p Yield st. plate σ _{Fp} MPa 235 235 235	Applied Ioa Axial stress G _{1,1} MPa 31.3 0 0	is found in thi ds Axial stress or ₁₂ MPa 31.3 0 0 0	Allowable us Ultimate cap Buckling str e description st Transversal stress o _{g,1} <u>MPa</u> 5.5 0 0	age facto vacity: ηυι ength: η _{BS} eet. Transvers stress σ ₂₂ MPa 5	r: c sal Shear stress tres MPa 6.5 -2.8 0 8 0 21.2	1 1 Pressure (fixed) P MPa 0 0 0 0	Analysis (Row b Combin In plane support Int Int Int	y row nations of rotation Rotation Left MNm/m SS SS SS	finput tions al support Right MNm/m SS SS SS	Delet Save Upper MNm/m SS SS SS	e old results old results Lower MNm/m SS SS SS SS	
2 3 4 5 6 6 6 1 0 7 7 8 9 9 0 0 1 1 Cc 3 3 Cc 3 3 Cc 3 7 7 8 8 9 9 7 7 8 8 9 9 7 7 8 8 9 9 7 7 8 8 9 9 7 7 8 8 9 9 7 7 8 8 8 9 9 7 8 7 8	Define co ase see comment entification of panel 1.run(2).panel 1.run(2).panel 1.run(2).panel	t boxes (cells no Geometry Length of plate 2400 2400 2400	width of plate L2 mm 750 750 750	Calculate ar red triangles) fo Plate thickness tp mm 14 14 14	or further explanation of the export parameter of the explanation of t	Poisson's ratio v 0.3 0.3 0.3	tures/input. A γ Yield st. plate σ _p MPa MPa 235 235 235	panel ilustration Applied Ioa Axial stress σ _{1.1} MPa 31.3 0 0	is found in the ds Axial stress $\sigma_{1,2}$ MPa 31.3 0 0 0	Allowable us Ultimate cap Buckling str e description st description st descriptio	age facto pacity: ηυκ ength: η _{BS} eet. Transvers stress σ _{2,2} MPa 5	r: sal Shear stress T ₁₂ MPa 0 21.2	1 1 Pressure (fixed) p MPa 0 0 0	Analysis (Row b' Combin Boundary In plane support Int Int Int	y row nations of Rotation Left MNn/m SS SS SS	finput al support Right MNm/m SS SS SS	Delet Save	e old results old results Lower MNm/m SS SS SS SS	
2 3 4 5 6 6 1 6 7 8 9 0 1 2 2 Cc 3 3 Cc 4 5 6 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 7 8 9 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Define co ase see comment entification of panel 1.run(2).panel 1.run(2).panel 1.run(2).panel	t boxes (cells m Geometry Length of plate L1 2400 2400 2400	width of plate L2 mm 750 750 750	Calculate ar red triangles) for Plate thickness tp mm 14 14 14	nd export pan material Maderial Modulus of elasticity E MPa 206000 206000 206000	Poisson's ratio v 0.3 0.3 0.3	tures/input. A Yield st. plate G _{F9} MP 235 235 235	Applied loa Axial Axial Stress G _{1,1} MPa 31.3 0 0 0	is found in the ds Axial stress σ _{1,2} MPa 31.3 0 0 0	Allowable us Ultimate cap Buckling str a description st Transversal stress $\sigma_{2,1}$ MPa 5.5 0 0 0	age facto pacity: ηυι ength: η _{BS} eet. Transvers stress σ ₁₃ MPa 5	r: c s sal Shear stress τ ₁₃ MPa 0 8 0 21.2	1 1 (ficed) p MPa 0 0 0	Analysis (Row b' Combin Boundary In plane support Int Int Int	y row nations of Rotation: Left <u>MNm/m</u> SS SS SS SS	Finput tions al support Right MNm/m SS SS SS SS	Delet Save Upper MNnv/m SS SS SS	e old results old results Lower MNm/m SS SS SS SS	
2 3 4 5 6 6 1 2 2 7 8 9 0 0 2 2 Cc 3 3 Cc 4 5 5 6 6 7 8 9 9 0 0 2 2 Cc 3 3 Cc 4 5 9 0 0 0 2 2 5 7 8 9 0 0 1 1 1 5 7 8 9 0 1 1 5 7 8 9 0 1 1 1 5 7 8 9 0 1 1 5 7 8 9 0 1 1 1 5 7 8 9 0 1 1 1 5 7 8 8 9 0 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	Define co ase see comment entification of panel 1.run(2) panel 1.run(2) panel 1.run(2) panel 1.run(2) panel	tboxes (cells n Geometry Length of plate L1 2400 2400 2400 2400	width of plate L2 mm 750 750 750	Calculate ar red triangles) fr Plates thickness tp mm 14 14 14 14 53) - Inputt	or further explanation of the second	Poisson's ratio 0.3 0.3 0.3 0.3 0.3	tures/input. A Yield st. plate G _{P0} MPa 235 235 235 3) - Output	Applied loa Axial stress ort,1 MPa 31.3 0 0 0 0	is found in the ds Axial stress σ _{1,2} MPa 31.3 0 0 0 0 0 0 0 0 0 0 0 0 0	Allowable us Ultimate cap Buckling str e description st stress c _{2,1} MPa 5,5 5,0 0 0 0 3) - Input 2	age facto pacity: ηυι ength: η _{BS} eet. Transvers stress σ ₁₃ MPa 5	r: sal Shear stress ^{Tr} : 0 8 0 21.2	1 1 (fixed) P MPa 0 0 0	Analysis (Row b' Combin Boundary In plane support Int Int Int Int Int	y row nations of Rotation Left MNm/m SS SS SS	Finput tions al support Right MNm/m SS SS SS SS	Delet Save Upper MNnv/m SS SS SS	e old results old results Lower MNm/m SS SS SS SS	

11 April 2011

5. SHORT COMMANDS AND WINDOWS COMPLIANCE

GeniE comes with a number of short commands utilising e.g. ALT, CTRL, and keys F1-F10 like in other Windows applications. Below is a table listing the available ones:

GeniE

205

SHORT COMMAND	2nd LEVEL PULLDOWN	3rd LEVEL PULLDOWN
CTRL+N	File New Workspace	
CTRL+O	File Open Workspace	
CTRL+S	File Save Workspace	
CTRL+T	Edit Copy	
ALT+O	View Options	
ALT+C	Tools Customize	
ALT+M		Tools Analysis Create mesh
ALT+D		Tools Analysis Activity Monitor
ALT+P		Tools Analysis Presentation
ALT+G		Tools Analysis Beam Result Diagrams
ALT+S		Visible Model Show selection only
ALT+Plus		Visible Model Add selection
ALT+Minus		Visible Model Remove selection
ALT+A		Visible Model Show all
ALT+Q		Visible Model Show complement
ALT+B		View Options General:Cur. Background
Del	Edit Delete	

КЕҮ	TOGGLE COMMAND
F1	Help
F2	Rotate - rotation in all 3 degrees of freedom
F3	Zoom - zoom in or zoom out
F4	Pan - move model to desired position on display
F5	View Iso - view from isometric point
F6	View from X - view in negative X-direction
F7	View from Y - view in negative Y-direction
F8	View from Z - view in negative Z-direction
F9	Fit screen
F10	Spin - remembers the last rotation and speed of it and makes this a continuous spin
F11	Toggles the snap perpendicular, tangential, plane mode

11 April 2011

6. THE COMMAND LINE INTERFACE SYSTEM

GeniE is primarily intended to be operated by the graphical user interface. All program features may, however, also be accessed by GeniE commands.

The GeniE commands are basically used to create journals during interactive sessions. The journal file can now be used to re-create the model (you may also edit and change the journal file). The GeniE commands may also be used to write a model input file directly or to invoke features that have no graphical interface.

There are 3 ways of entering commands into GeniE,

- 1. by typing or pasting commands into the Command Line tab in GeniE
- 2. by the "Read command file" option in the File menu of GeniE
- 3. by starting GeniE with an input command file from the command prompt (DOS window)
- E.g. "C:\Program Files\DNVS\GeniE\Program\GeniEr" MyProject /NEW /COM=MY JOURNAL.JS /EXIT

Please note that if you are using another editor than e.g. MS Notepad, you need to specify that the output format is for PC format and not Unix format.

GeniE supports two kinds of commands

- Specific GeniE commands
- General JScript commands

Specific GeniE commands are typically for creating and editing GeniE model entities such as plates and beams. A simple session of GeniE commands may be:

```
// Create 2 points
Point1 = Point(0, 0, 0);
Point2 = Point(0, 0, 10);
// A double slash indicates a comment line
// Copy the two points 10 m in x-direction
Point3 = Point1.copyTranslate(Vector3d(10 m, 0 m, 0 m));
Point4 = Point2.copyTranslate(Vector3d(10 m,0 m,0 m));
// Note that all command must end with ();
// Create a beam between point 3 and 4
BeamA1 = Beam(Point3, Point4);
BeamA1.material = Material1;
BeamA1.section = Section1;
11
// Create a support at Point3, rotate the local Z-axis 30 deg. and define the
boundary conditions
Sp1 = SupportPoint(Point(10 m, 0 m, 0 m));
Sp1.rotateLocalZ(30);
Spl.boundary = BoundaryCondition(Free, Fixed, Fixed, Free, Fixed, Free);
```

Note that the objects in the GeniE model have properties and functions that are addressed by a dot like in "BeamA1.material" or "Point1.copyTranslate".

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	207	11 April 2011

When using the GeniE Command Line tab you will get a list of relevant properties and functions by pressing the Tab key while typing the command;

>	Beaml.	
	StraightBeam	
	beamType	
	copy3Point	-

The command window will also provide a tool tip to assist you with the command syntax;

GeniE also supports the general programming language JScript. By combining GeniE commands with JScript you may use programming features in your model input file.

Typically you may start by defining all basic model data as variables in the beginning of the file and then refer to these variables when creating the model.

```
// Coordinate arrays
var X1 = new Array();
var Y1 = new Array();
var Z1 = new Array();
// X1-values
X1[0] = -23;
X1[1] = 12;
X1[2] = 34;
// Y1-values
Y1[0] = -45;
Y1[1] = -20;
Y1[2] = -5;
// Z1 elevations
Z1[0] = -128.987;
Z1[1] = -124.987;
11
Point A = Point(X1[0], Y1[0], Z1[0]);
```

Further you may use arithmetic expressions within the GeniE commands or to do separate calculations;

TopElevation = Z1[1] + 2.45; Point B = Point(2.54, 0.0, (TopElevation + 10));

Please note that most mathematical functions are addressed through the Math object in JScript.

```
Radius = 5.0;
MyArea = Math.PI()*Math.pow(Radius,2);
Print(MyArea);
78.539816
```

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	208	11 April 2011

For more sophisticated modelling you may use For loops and If testes to program the creation of your model. You may also invoke other applications that support Automation (e.g. Excel) to exchange data with your GeniE model. Automation is a technology that allows software packages to expose their features to scripting tools and other applications.

Example of creating beams in a loop:

```
//Adding beams to a leg
for (k = 0; k < 5; k++)
{
 var Bms1 = new Array();
Bms1[k] = Beam(LegPoint[k], LegPoint[k+1]);
Bms1[k].name = "LegBeam" + k;
Bms1[k].section = LegSection;
Bms1[k].material = LegMaterial;
}</pre>
```

6.1 The GeniE JSript Command Reference

The GeniE commands are described in the JScript Command Reference found in the GeniE help menu. The most feasible approach to writing a command file is rather to create a sample model interactively and then use the generated journal as a template. When you have found the kind of object you are working with you may use the Command Reference to get a full list of relevant features and the command syntax for accessing these. The following pictures show how you find information about the script command for generating a basic beam and which other commands are associated this operation.



GeniE

11 April 2011

Version 5.3

Structure	
<u>Beam</u>	Construct a new straight or curved beam from the given guide curve
<u>Beam</u>	Construct a new straight or curved beam from the given points
<u>Beam</u>	Create a straight Beam
ConstantLocalSystem	Define a constant local system for the curve
CriterionInPlane	
<u>CurvedBeam</u>	Curved Beam
FeatureEdge	Feature Edge
GuideLocalSystem	Let the curves local X follow the direction of the curve
Joint	Represent a tubular joint
NormalToCurvePlane	If a curve lies in a plane, use the normal as the curve local Z
Pile	Pile
Plate	Flat Plate
PointMass	PointMass
RelativeToPlate	The local Z for the curve is locked to the normal of a shell
Shell	Curved Shell
<u>SimplifyTopology</u>	Simplifies the structure by removing unnecessay topology.
SplitAtPoint	Split structure at the given point
SplitStructure	Explode selected structure at all structural connection points
<u>StraightBeam</u>	Straight Beam
SupportCurve	Represent a Support Curve
SupportPoint	Represent a Support Point

Looking up straight beam found under the class Structure.

Function Detail

Beam

BasicBeam Beam(Point p1, Point p2, OverlapPolicy overlapPolicy)

Create a straight Beam

Parameters:

p1 - End 1 of the Beam p2 - End 2 of the Beam

p2 - End 2 of the Beam overlapPolicy - Specifies how this beam is to be inserted into the model

Example:

//Create a beam that is allowed to overlap existing beams: b1 = Beam(Point(0,0,0), Point(1,0,0), geAllowOverlap); //create a beam that will recover portions of existing beams in order to make room for itself: b2 = Beam(b1.end1), Point(2,2,0), geEnforceThis);

Overview **Class Hierarchy** Genie D3.0-10 25-Oct-2004 **BasicBeam** Class hierarchy: ModelObject +--<u>NamedObject</u> | +--<u>Transformable</u> +--<u>BasicConcept</u> | +--BasicBeam Direct Known Subclasses: CurvedBeam, Pile, StraightBeam **Function Summary** BasicBeam divideAt (double parameter) Divide beam at parameter value and return the second half of the beam divideSeqmentAt(int iseg, double parameter) Divide beam segment at parameter value explode(NameMask nameMask) Split structure into smaller parts
 extendEnd(long iend, Length extension)

 Extend or shorten beam at end 1 or 2 along direction of beam
 Material getSegmentMaterial (int) Section getSequentSection(int) Point intersect (Plane3d plane)

The *Function Detail* now documents the command itself and practical examples.

You may look at further details belonging to the BasicBeam. When scrolling down the Function Summary the additional commands (ore features) are listed.

209

GeniE 210

Version 5.3

Functio	on Summary
ModelObject	<pre>copy3Point(Point sp1, Point sp2, Point sp3, Point dp1, Point dp2, Point dp3) Copy the object without scaling from one location to another.</pre>
ModelObject	copyMirror (Point pl, LengthVector3d v1) Mirror a copy of the object around pl and v1
ModelObject	copyRotate (Point pl, LengthVector3d p2, Angle angle) Rotate a copy of the object around p1 and v1 the angle angle
Array	<pre>copyRotate(Point pl, LengthVector3d p2, Angle angle, int n) Make n copies of the object, incrementing the angle for each copy</pre>
	copyRotate (Point pl, LengthVector3d p2, Angle angle, int n, NameMask nameMask) Make n copies of the object, incrementing the angle for each copy

Scroll down to *copyRotate* to find out how this command works

11 April 2011

By clicking the link "copyRotate" you will get a detailed description of this function as follows:

ModelObject copyRotate(Point p1, LengthVector3d p2, Angle angle)

Rotate a copy of the object around p1 and v1 the angle angle

Decription:

Make one copy of the object. The transformation is defined as counterclockwise rotation around the given axis vector at the anchor point.

Parameters:

 ${\tt pl}$ - Point on rotation axis

p2 - Rotation axis vector

angle - Rotation angle

Returns:

the copied object

Example:

```
//Rotate Bm1 45 degrees around Bm1.end1 and the axis (0,0,1):
Bm2=Bm1.copyRotate(Bm1.end1,Vector3d(0,0,1),45deg);
```

The description shows the syntax of the command with the type and name of each parameter. By clicking the link for the type like <u>LengthVector3d</u> you will get all valid forms for entering a vector in GeniE.

More about general JScript commands may be found e.g. at http://msdn.microsoft.com/scripting.

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	211	11 April 2011

6.2 Useful script commands

GeniE creates a journal file including all the operations you perform except a few – typically related to making a finite element model (when not part of analysis activity), export the FE model or graphical interactions. These commands may be executed directly from the Command Line Interface or when importing a journal file containing the commands. You may find some of the following script commands useful.

Command	Description
CreateMesh();	Force the creation of a mesh (same as Alt+D or Tools/Analysis/Create Mesh)
<pre>FemExporter = ExportMeshFem(); FemExporter.DoExport(name);</pre>	Export a finite element model to the default working directory (same as <i>File/Export/FEM File</i>), e.g. FemExporter.DoExport("T1.FEM");
GenieRules.Meshing.preference(mpUseDrillingE lements,true);	Activation of 3 and 4 noded drilling elements (FTAS and FQAS)
Graphics.move(vector);	Move the view of the model along the given vector in the global system, e.g. Graphics.move(Vector3D(1,1,0));
Graphics.pan(x,y);	Pan a model. Same as the graphical operation 'Pan (F4)', e.g. Graphics.pan(1,10); will move the model 1 pixel to the right and 10 pixels up. Note that x and y are measured in pixels.
Graphics.rotate(rotationAxis,angle);	Rotate the view of the model a given angle around the centre of the model with a rotation axis given in the global system, e.g. Graphics.rotate(Vector3d(0,0,1),45); The model is now rotated 45degrees around global z- axis.
Graphics.rotate(rotationCenter,rotationAxis,angle);	Rotate the view of the model a given angle around rotationCenter with a rotation axis given in the global system, e.g. Graphics.rotate(Point(2,5,3),Vector3d(0,0,1),45); The model is now rotated 45degrees around global z- axis around the point with coordinate values (2,5,3)
Graphics.rotationCenter;	Return the models centre of rotation, i.e. when you rotate the model graphically (using the default rotation scheme, the model is rotated around the centre of the model. Graphics.rotationCenter);: will return this point.
Graphics.saveImage(name);	Save the image in a given format (given by the filename), e.g. Graphics.saveImage("Picture.jpg");

DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
Version 5.3	212	11 April 2011

Command	Description	
Graphics.saveImage(name,width,height);	Save the image in a given format, scaled to the size in pixels given by width and height, e.g. Graphics.saveImage("Picture.jpg",2000,2000);	
Graphics.zoomArea(left,bottom,right,top);	Rubberband zoom. Same as the graphical operation 'Zoom Rubberband'. The input (left,bottom,right,top) are given in pixels. The bottom left pixelcoordinate of the 3d-view is 0,0. The top right pixelcoordinate of the 3d-view is dependent on your screen size. (You can find this value on your computer by File Save Graphics As <save> Under Pixels Width Pixels Height Pixels Width is the rightmost pixel coordinate of the screen. Pixel Height is the top coordinate of the screen. If the width and height of the screen are 600,600, the size of our 3d viewport is (0,0,600,600). If you want to zoom in on the middle portion of the screen, you may use <i>Graphics.zoom</i> <i>Area(150,150,450,450);</i>. If you want to zoom out, you may use <i>Graphics.zoomArea(-150,- 150,750,750);</i>.</save>	
Graphics.setEye(eyePos);	The command gives a view of the model focused on its origin,e.g. Graphics.setEyePosition(Point (-30,30,5),Vector3d(1,-1,0));. To be used to recreate a specific view.	
Graphics.viewISO();	Same as the graphical operation 'View ISO (F5)' Note, you need to refresh graphics to yield immediate screen update	
Graphics.viewFromX();	Same as the graphical operation 'View from X (F6)' Note, you need to refresh graphics to yield immediate screen update	
Graphics.viewFromY();	Same as the graphical operation 'View from Y (F7)' Note, you need to refresh graphics to yield immediate screen update	
Graphics.viewFromZ();	Same as the graphical operation 'View from Z (F8)' Note, you need to refresh graphics to yield immediate screen update	
Graphics.fitModel();	Same as the graphical operation 'Fit (F9)' Note, you need to refresh graphics to yield immediate screen update	
JsExporter = ExportModelJS(); JsExporter.DoExport(name)	Export a js file to the default working directory (same as <i>File/Export GeniE Journal File</i>), e.g. JsExporter.DoExport("C:/Location/yourname.js")	
DET NORSKE VERITAS SOFTWARE	GeniE	User Manual Vol. VI
-----------------------------	-------	---------------------
Version 5.3	213	11 April 2011

Command	Description
<platename_a>.join(<platename_b>);</platename_b></platename_a>	Command for joining plate B into plate A. The new plate keeps name plate A. Example:
	Joining Pl124 with Pl121 has the command Pl121.join(Pl124);
Math.timer(long t0);	Useful feature that will return elapsed seconds from t0. You need to specify start time (t0), when to measure elapsed time and print to the journal file window. Example returning elapsed time at t1 and t2 (to be edited in the journal file):
	t0=Math.timer(0); t1=Math.timer(t0); print(t1); -> returns time since t0 in seconds t2=Math.timer(t0); print(t2); -> returns time since t0 in seconds
SinExporter = ExportResultsSin(); SinExporter.DoExport(name);	Export a results file to the default working directory (same as <i>File/Export SIN File</i>), e.g. SinExporter.DoExport("R1.SIN");
XmlExporter = ExportConceptXml(); XmlExporter.exportLoads = true/false; XmlExporter.DoExport(name);	Export a xml file to the default working directory (same as <i>File/Export XML File</i>), e.g. XmlExporter.DoExport("Semi.xml");

By using these commands together with commands automatically created during GeniE sessions it is possible to run GeniE from batch mode to do among others

- Create structure
- Apply loads
- Model environment (wave, current, air, soil)
- Perform analysis structural, wave and pile-soil (or export FEM model for later usage in e.g. a superelement analysis)
- Specify view settings, create and save graphics images

----- 0 -----